

Carlson Software 2024

Carlson Software Inc.

User's manual

March 26, 2024

Contents

Chapter 1. Introduction	1
Carlson Directory Structure	2
Installation Guide	3
Carlson Migration	9
Starting AutoCAD with Carlson	11
License Models	12
Carlson Registration	14
Network License Setup	15
Network License Monitoring	17
Tablet Template	18
Troubleshooting and Customizing Setup	19
Loading Carlson Menus	21
Obtaining Technical Support	22
Command Entry	23
Transparent Commands	23
What is New	25
Setting Up a Project	31
New/Startup Wizard	31
Layer and Style Defaults	34
Carlson File Types	34
File Selector	37
Standard Report Viewer	41
Annotative Text	42
Report Formatter Dialog	43
Instruction Manual and Program Conventions	54
Language Localization	55
Patch Management	59
License Agreement	61
Chapter 2. General Commands	66
File Menu	67
New	67
Project Explorer	69
Get Project from Data Depot	78
Layout Manager	81

Output Layouts to PDF	84
Merge PDF Files	85
Carlson Community	86
Carlson_PhotoCapture	89
Import 3D Viewer File	93
Import LandXML File	93
Import RoadXML File	95
Import 12D File	96
Import Google Earth File	97
Civil 3D All	98
Civil 3D Labels	98
Civil 3D Linework	98
Civil 3D Pipes	98
Import MxRoad GENIO	98
Import InRoads Settings	99
Import ePlan File	100
Export LandXML File	100
Export RoadXML File	103
Export Polyline File	105
Export Civil 3D Drawing	106
Export Civil 3D Alignments	107
Export Civil 3D Points	107
Export Civil 3D Surface	107
Export 12D File	108
Export Google Earth File	108
Export ePlan File	111
Export Drawing to AutoCAD 14	111
Set Google Tags	112
Microstation DGN	113
Write Polyline File	114
Drawing Cleanup	115
Save Drawing Named by Current Time	119
Drawing Explorer	119
Geolocate DWG Files	122
Open Drawing	122
Draw Polyline File	127
Translate Layers	128
3DSolid and Polymesh Converter	129

Remove XData	129
Remove Civil 3D Custom Objects	129
Remove Reactors	130
Remove Groups	130
Unlock Attributes	131
Record Script, End Script, Run Script	131
Drawing Save Log	131
Edit Menu	132
Erase by Layer	132
Erase by Closed Polyline	133
Erase Outside	134
Temporary Erase	134
Copy To Layer	135
Copy Polyline Section	135
Offset To Layer	136
Offset to Area	136
Multiple Offsets	137
Median Offset	137
Variable Offset	138
Drift_Offset	138
Block Explode	139
Extend to Intersection	140
Extend Arc	140
Extend by Distance	141
Break by Crossing Polyline	143
Break Polyline at Specified Distances	143
Break at Intersection	144
Break at Selected Point	144
Chamfer By Chord Length	144
Change Elevations	145
Change Attribute Style	145
Change Style	146
Change Colors	147
Colors From Layer	147
Colors By Elevation	148
Change Lineweight	148
Change Block/Inserts Rotate	149
Change Block/Inserts Substitute	149

Change Block/Inserts Resize	150
Pivot Point Rotate by Bearing	150
Entity Insertion Point Rotate	151
Select by Length	151
Select by Block	152
Select by Area	152
Select by Filter	153
Select Similar	153
Text Enlarge/Reduce	154
Rotate Text	154
Move Text	154
Move Text with Leader	155
Move Attributes with Leader	156
2D Scale	157
Change Text Font	158
Change Text Size	159
Change Text Width	159
Change Text Oblique Angle	160
Convert Text To MText	160
Fix Text Overlaps	161
Flip Text	162
Flip Text By Twist Screen	162
Split Text into Two Lines	163
Text Explode To Polylines	163
Text Capitalization	163
Text Math	164
Add Prefix/Suffix To Text	164
Remove Spaces From Text	165
Line Up Text	165
Join Text Entities	165
Replace Text	166
Move Attributes with Leader	166
Consolidate Table	167
Select by Elevation	167
2D Align	168
Entities to Polylines	169
Reverse Polyline	169
Reduce Polyline Vertices	169

Densify Polyline Vertices	170
Join Two Polylines	171
Merge Closed Polylines	171
Draw Polyline Blips	172
Draw Polyline Start/End	173
Set Polyline Origin	174
Add Intersection Points	174
Add Polyline Vertex	175
Add Polyline Arcs	176
Edit Polyline Vertex	176
Edit Polyline Section	177
Remove Duplicate Polylines	178
Remove Polyline Arcs	178
Remove Polyline Segment	179
Remove Polyline Vertex	180
Remove Polyline Blips	180
Create Polyline ID Labels	181
Join Polyline By Trace	182
Smooth Polyline	182
Make Arcs Tangential	183
Check Elevation Range	183
Highlight Non-Perpendicular Intersections	184
Highlight Non-Tangent Polylines	184
Highlight Crossing Plines	185
Highlight Unclosed Polylines	186
Close/Open Polylines	186
Buffer Offset	187
Change Polyline Width	187
Offset 3D Polyline	187
Fillet 3D Polyline	189
Join 3D Polyline	190
Trim 3D Polyline	190
Add Point by Two Slopes	190
Add Points At Elevation	190
Join Nearest	191
Solid Fill Polyline	192
3D Entity to 2D	193
View Menu	194

Zoom Points	194
Zoom Selection	194
Zoom Most	195
Zoom Extents on All Layouts	195
Align Paper With Model Space	195
3D Viewer Window	196
3D Viewer 3DX Model File	206
Surface Object View Properties	206
Draw Model View	206
Set UCS to World	206
Set Level	207
Level Manager	207
Change Layer	207
Freeze Layer By Selection	208
Restore Due North	208
Restore Layer	209
Set Layer	209
Turn Off By Selection	209
Freeze Layer	209
Surface 3D Viewer	210
Change Layer Color	210
Add/Replace Layer Prefix	211
Freeze Layer By Pick	211
Thaw Layer	211
Isolate Layer	212
Twist Screen: Standard	212
Twist Screen: Line Pline or Text	213
Twist Screen: Surveyor	213
Twist To 3D View	213
Restore World View	214
Display Order by Layer	214
Thaw/On All Layers	216
Freeze Layer By Name	216
Thaw Layer By Name	216
Isolate Layer By Name	216
Turn Off By Pick	216
Turn Off By Selection	217
Turn On Layers	217

Lock Layers	217
Show/Hide By Selection	217
Save/Restore Layer State	218
Draw Menu	218
2D Line	219
3D Line	220
2D Polyline	221
3D Polyline	225
3 Point	230
PC, PT, Radius Point	230
PC, PT, Radius Length	230
PC, PT, Tangent	231
PC, Radius, Chord	231
PC, Radius, Arc Length	231
2 Tangents, Radius	232
2 Tangents, Arc Length	232
2 Tangents, Chord Length	232
2 Tangents, Mid-Ordinate	233
2 Tangents, External	233
2 Tangents, Tangent Length	234
2 Tangents, Degree of Curve	234
2 Tangents, Through Point	234
Tangent, PC, Radius, Arc Length	235
Tangent, PC, Radius, Tangent Length	235
Tang, PC, Radius, Chord Length	236
Tang, PC, Radius, Delta Angle	236
Arc From Last Point	236
Compound or Reverse	237
3D Arc	237
2-Radius Curve Series	238
3-Radius Curve Series	238
Best Fit Curve	239
Curve Calc	240
Spiral Curve	241
Text Wizard	242
Hatch Wizard	243
Multileader with Text	244
Field Text	245

Text on Line	246
Text on Polyline	247
Text Import	248
Text Export	248
Text Mask	248
Text UnMask	249
Draw Box Around Text	249
White Solid Behind Text	250
Trim Linework Through Text	250
Insert Symbols	250
Insert Multi-Point Symbols	253
Draw 3D Model File	256
Draw Table	256
Arrowhead With Line	257
Arrowhead	257
Curve - Arrow	257
Leader With Text	258
Special Leader	260
Callout Leader	260
Bold Curve Leader	261
Flow Leader	261
Draw Standard Item	262
Draw Label For Companion Linework	264
Draw Standard Item by Quick Key	265
Item Standards Manager	265
Draw By Example	267
Sequential Numbers	267
Boundary Polyline	269
Shrink-Wrap Entities	270
Centroid Point	272
Section Line	273
SmarTrace	274
Polyline by Slope Ratio	274
Polyline by Nearest Found	275
Inquiry Menu	275
Point ID	276
Layer ID	276
Layer Report	277

Layer Inspector	277
Drawing Inspector	277
Bearing & 3D Distance	279
Find Point	280
Calculator	280
Curve Info	282
Polyline Info	283
Block Data Report	283
Layer Report	285
Angle Info	285
Display-Edit File	286
Text File To Report Formatter	286
Display Last Report	286
Load Last Report Formatter	286
Load Saved Report	287
Settings Menu	287
Drawing Setup	287
Set Project/Data Folders	294
Store Project Archive	304
Extract Project Archive	304
Configure Carlson	305
Data Depot	326
Carlson Settings Explorer	347
Settings File Manager	349
Toolbar Manager	350
Edit Symbol Library	351
Layer Library	352
2D Linework Library	355
3D Viewer/Model Library	356
Quick Keys	358
Title Block	360
Mortgage Block	362
Rescale Drawing	362
Set/Reset X-Hairs	363
Save/Load Tablet Calibration	363
Create AutoCAD Icon	364
Point Object Snap	364
Point Label Settings	364

System Variable Editor	365
Select CAD Default Settings	368
Points Menu	368
Point Defaults	370
Draw-Locate Points	372
List Points	376
Import Text/ASCII File	378
Export Text/ASCII File	381
Set Coordinate File	383
CooRDinate File Utilities	384
Point Group Manager	394
Edit Points	401
Erase Points	402
Freeze Points	402
Thaw Points	403
Lock Points	403
Unlock Points	404
Report Locked Points	404
Translate Points	404
Rotate Points	406
Align Points	408
Scale Points	409
Move Points	411
Coordinate Transformation	412
Edit Point Attributes	415
Edit Multiple Pt Attributes	418
Dynamic Point Attributes	422
Move Point Attributes Single	422
Move Point Attributes with Leader	423
Scale Point Attributes	424
Erase Point Attributes	424
Twist Point Attributes	425
Resize Point Attributes	425
Mask Point Attributes	426
Trim by Point Symbol	426
Freeze Point Attributes	427
Thaw Point Attributes	427
Fix Point Attribute Overlaps	427

Coordinate Cloud	430
Change Point LayerColor	433
Vertical Datum Utilities	434
Link Linework With Points	434
Search Point Files	435
Point Attribute Layout Manager	435
Geoid Utilities	436
Renumber Points	437
Explode Carlson Points	437
Convert Surveyor1 to CRD	438
Convert CRD to TDS CR5/Convert TDS CR5 to CRD	438
Convert CRD to Land Desktop MDB	438
Convert Land Desktop MDB to Carlson Points	439
Convert Civil 3D to Carlson Points	439
Convert Carlson Points to Land Desktop	440
Convert Land Desktop to Carlson Points	440
Convert Softdesk to Carlson Points	440
Convert Carlson Points to C&G	441
Convert C&G to Carlson Points	441
Convert Carlson Points to Simplicity	441
Convert Simplicity to Carlson Points	442
Convert Leica to Carlson Points	442
Convert Geodimeter to Carlson Points	443
Convert Carlson Points to Ashtech GIS	443
Convert Carlson Points to Softdesk	443
Convert CAICE KCM to Carlson CRD	443
Convert PacSoft CRD to Carlson CRD	443
Convert Carlson Points to Eagle Point	444
Convert Eagle Point to Carlson Points	444
Search Published Control	444
Help Menu	447
Carlson On-line Manual with Movies	447
Carlson Academy	447
Command Search	448
Serial Number Report	449
About Carlson	450

Chapter 3. Survey Module	454
Survey Menu	455
Carlson Cloud	455
Data Collectors	461
Drone Mission Planning	485
Edit-Process Raw Data File	490
Edit-Process Level Data	542
Edit Process SDMS File	549
SurvNET	549
Introduction	550
Starting Survnet	552
Menu System Overview	555
File Menu	558
Settings Menu	558
Process Menu	582
Tools Menu	594
View Menu	602
SurvNET Window Menu	606
Toolbars	606
Raw Traverse Data	607
SurvNET Editor	618
Data Collector Transfer	618
Example Projects	618
Network Processing Reports	620
2D-1D Local Coordinate System	623
2D-1D State Plane Coordinate System	630
GPS Network	635
GPS Vectors and Total Station	639
Vertical Adjustment	650
Draw Field to Finish	656
Field to Finish Inspector	729
Edit Field to Finish Codes	731
Enter Deed Description	731
Draw Field to Finish	734
Deed Reader	807
Import Angle/Distance File	809
Process Deed File	810
Deed File By Interior Text	813

Deed Linework ID	814
Deed Correlation	814
Legal Description Writer	817
Closure by Point Numbers	826
Map Check by Pnts	828
Mapcheck by Screen Entities	830
Cut Sheet	832
Set Point Elevations by 3D Polylines	846
Set Point Elevations by Surface Model	847
Polyline Report	847
Polyline to Deed File	849
Polyline to RW5 File	849
Crandall Polyline Adjustment	849
Compass Polyline Adjustment	850
Grant Boundary Adjustment	852
4 Sided Building	852
CAPD Command Reference	853
COGO Menu	855
Inverse	855
Occupy Point	857
Traverse	858
Side Shots	860
Enter-Assign Point	860
Raw File On/Off	861
Line On/Off	861
Visual COGO	862
Locate by Line Bearing	865
Locate by Turned Angle	866
Locate by Azimuth	866
Locate by Bearing	867
Locate by Delta	867
Pick Intersection Points	867
Linework Intersection Points	868
Bearing-Bearing Intersect	869
Distance-Distance Intersect	870
Bearing-Distance Intersect	871
Perpendicular Intersect	872
Tangent Intersect	873

2 Point - 2 Point Intersect	874
Resection	874
Benchmark	876
Numeric Pad COGO	877
Line by Angle-Distance	878
Tangent Line from Circles	878
Geodetic Traverse	879
Geodetic Double Proportion Line Division	879
Irregular Boundary Adjustment	880
Geodetic Single Proportion Line Division	882
Geodetic Double Break	882
Geodetic Middle Break	882
GLO Corner Proportioning	883
Section Subdivision	883
One Way Control	884
Two Way Control	885
Three Way Control	886
Four Way Control	888
Solar Observations	889
Triangle Solutions	892
Best Fit Point	893
Best Fit Circle	894
Best Fit Centerline	896
Best Fit Line by Average	898
Best Fit Line by Least Squares	899
Point on Arc	900
Divide Between Points	901
Divide Along Entity	902
Elevation Along Entity	902
Interval Between Points	903
Interval Along Entity	903
Create Points from Entities	904
Offset Points By Interval	907
Building Offset Extensions	910
Radial Stakeout	911
Area/Layout Menu	913
Area Defaults	914
Inverse with Area	918

Area by Lines & Arcs	920
Area by Interior Point	921
Area by Closed Polylines	921
Digitize Areas	923
Label Last Area	923
Area Table Defaults	924
New Area Table	927
Set Active Area Table	927
Edit Area Table Properties	927
Remove Area Table Rows	930
Consolidate Area Table	931
Tag Area Descriptions	931
Identify Area Descriptions	932
Untag Area Descriptions	932
Area Descriptions By Interior Text	932
Hinged Area	933
Sliding Side Area	933
Area Radial from Curve	934
Bearing Area Cutoff	936
Lot Layout	938
Cleanup Lot Linework	939
Set Linework Angles To Nearest Second	941
Set Linework Intersections To Perpendicular	941
Offsets & Intersections	941
Cul-de-Sacs	943
Footprint Creator	943
Elevate 2D Polylines	953
Parking	954
Set Back Measure-Move	956
Draw Lot Setback	957
Fit Structure	958
Design Parking Inside Perimeter	970
Edit Parking Inside Perimeter	973
Parking Perimeter Tools	973
Parking Buffer Zone	983
Parking Design	985
Parking Island	996
Handicap Marker	997

Parking Network	998
Lot Network Settings	1002
Lot Network Boundary	1007
Tag Sub-Area	1007
Untag Sub-Area	1007
Identify Sub-Area	1008
Report Sub-Area	1008
Hatch Sub-Areas	1008
Erase Sub-Areas Hatch	1009
Label Sub-Areas	1009
ROW Offsets Editor	1009
Lot Network Road Network	1010
Lot Network Linework	1014
ID Linework	1015
Untag Linework	1015
Point Check	1016
Set Lot Edge Angles To Nearest Second	1016
Lot Network Subdivide Area	1017
Size Lot by Frontage	1019
Lot Network Sliding Side Area	1019
Lot Network Hinged Area	1019
Lot Network Labels	1020
Lot Network Report	1020
Lot Number Report	1021
Lot Network Inspector	1022
Check Lot Network Parameters	1022
Find Lot Name	1023
Lot Network Renumber Lots	1023
Connect Driveways	1023
Tag ROW Segments to Skip Frontage	1024
Tag No-Building Area	1025
Untag No-Building Area	1025
Lot Network - Assign Lot Type	1025
Assign Building Type	1026
Lot Network Output To Lot File	1026
Set Lot File	1027
Design Lot	1027
Polyline to Lot File	1028

Lot File by Pick Interior	1029
Lot File by Interior Text	1030
Lot File by Closed Linework	1030
Lot Manager	1031
Lot Inspector	1035
Right-of-Way Crossing Table	1036
Define Lot Attributes	1039
Import Lot File From MDB Database	1041
Export Lot File to MDB Database	1041
Export Lot File To Old SurvCADD	1041
Set CRD File for Lot Files	1042
Lot File to Centerline	1042
Annotate Menu	1043
Angle/Distance	1043
Annotation Defaults	1045
Auto Annotate	1053
Custom Label Formatter AD	1062
Draw End Point Leaders	1064
Dynamic Annotation Note	1064
Fix Label Overlaps	1065
Switch Bearing/Azimuth Quadrant	1067
Mirror Selected Labels	1067
Mirror and Flip Selected Labels	1068
Flip Last Label	1068
Flip ON/OFF	1068
Move Label with Leader	1069
Bearing with Leader	1071
Distance with Leader	1071
Bearing-Distance with Leader	1072
Distance-Bearing with Leader	1073
Azimuth-Distance with Leader	1073
Flip Selected Labels	1074
Global Reannotate	1074
Survey Text Defaults	1075
Offset Dimensions	1076
Building Dimensions	1077
Adjoiner Text	1078
Draw Grid	1078

Draw Legend	1080
Draw North Arrow	1083
Draw Barscale	1085
Distance Table	1085
Create Point Table	1086
Update Point Table	1088
Point To Point Table	1088
Table Defaults	1089
Table Header	1093
Set Table Position	1094
Curve Table	1094
Line Table	1094
Swing Table	1095
Railroad Curve Table	1096
Edit Table Properties	1096
Edit Table Values	1101
Split Table	1102
Merge Tables	1103
Order Table	1105
Delete Table Elements	1105
Label Arc	1105
Stack Label Arc Auto Place	1107
Stack Label Arc Pick Side	1107
Custom Label Formatter	1109
Draw Text On Arc	1110
Draw Text on Tangent	1112
Edit Text on Arc or Tangent	1113
Fit Text Inside Arc	1113
Fit Text Outside Arc	1114
Arc Dimensions	1114
Change Polyline Linetype	1115
Polyline to Special Line	1117
Polyline to Tree Line	1117
Add Zig to Polyline	1118
Add Culvert to Polyline	1118
Sketch Tree Line	1119
Special Line/Entity	1120
Guard Rail	1120

Label Angle	1120
Label Coordinates/Elevation	1121
Label Point Comparison	1123
Label LatLong	1124
Label Curb Flow Elevations	1125
Replot Descriptions	1126
Textbox	1127
Label Offset Distances	1128

Chapter 4. CGSurvey Module 1129

CGFile	1130
Current Information	1130
Coordinate Files	1131
New	1131
Open	1132
Close	1133
Save As	1133
Export Coordinates to ASCII	1134
Import ASCII File into Coordinates	1145
Close Raw File	1146
Close Map Check File	1146
CGDos Drawings	1146
Open Dos Drawing	1147
Setup DOS Dwg	1148
Convert Old CG Dos Level File to New Format	1149
Convert Old CG Dos Raw File to New format	1149
Convert Old CG Dos Cross Section File to New Format	1149
Convert Old CG Dos Template File to New Format	1150
Empty Print File	1150
Print View Print File	1150
CGTrav	1151
Quick Traverse	1151
Edit Raw File	1154
Data Collector Transfer	1155
Reduce Traverse	1208
Edit Map Check File	1223
Reduce Map Check File	1223
Visual Map Check	1225
Create StarNet File	1227

CGCogo	1230
Inverse	1231
Intersects	1233
Station Offset	1240
Coords From Station Offset	1241
Create Point Group From Station Offset	1243
Display Centerline Stations	1243
Station Offset From Coords	1244
Points on Line	1245
Curves	1248
Calculate Horizontal	1248
Curve Between Tangents	1249
Middle Ordinate Solution	1250
Points on Arc	1251
Spiral Curve Design	1253
Spiral Curve Stakeout	1254
Stakeout Horizontal	1255
Tangent Between Curves	1256
Vertical Curve Design	1257
Area Summary	1258
Roadways	1259
Right-of-Way Easements	1259
Intersections/Cul-de-sacs	1259
T Intersections	1259
X Intersections	1260
Y Intersections	1261
Bubble Cul-de-Sac	1262
Standard Cul-de-Sac	1262
Stake-Out	1265
Angles Right	1265
Radial Stake Out	1265
Best Fit	1266
Triangulation	1270
NAD83	1271
CGDraw	1273
Drawing Settings	1273
Set Line Type	1273
Global Edit	1274

Border	1276
Coordinate Grid	1277
Text on Arc	1278
Create	1278
Move	1279
Edit	1279
Delete	1280
Draw Mapcheck	1280
Multi-Draw	1282
Plot Points and Symbols	1286
Plot Points on Screen	1286
Remove Points from Screen	1287
Graphic Scale	1287
Lines and Polylines	1288
Lines by Point Number	1288
Lines by Description	1289
Lines by Codes	1290
Polylines by Point	1290
Fit Polylines	1291
Calls	1293
Place Calls	1293
Move Calls	1295
Reverse Calls	1295
Tables	1295
Coordinates	1295
Call Table	1297
Curve	1299
Auto Map	1300
Draw	1300
Erase	1306
Leaders	1306
Text	1306
Coordinate Leader	1307
Point Label	1308
Station-Offset	1308
Query	1309
Drop C&G Attributes	1310
re-Associate Coord. file	1310

Refresh Screen	1312
CGMngmt	1313
Point Manager	1313
Edit Coordinates (CGEditor)	1320
Auto Create Points	1321
Manual Storage	1323
Delete	1324
Fix Coords	1325
List	1325
Renumber Points	1326
Transformations	1327
Combined Transformations	1327
Best Fit Transformation	1331
Copy Coordinates	1333
Description Tables	1334
Create	1335
Edit	1335
Set Current	1339
Close Current	1339
Set Default	1339
Convert to SurvCE FCL file	1339
Point Groups	1340
Create	1341
Edit	1343
CGTopo	1344
Topographic Settings	1344
Erase Surface from DWG	1344
Tin	1344
Main Contours	1345
Intermediate Contours	1345
All topo items	1345
Label Contours	1345
Place Labels	1345
Delete Labels	1346
CGTools	1347
CG Options	1347
Copy Entity to Layer	1365
Layer Control	1366

Pick Layers to Freeze	1366
Pick Layers to Thaw	1366
Freeze ALL Layers	1366
Thaw ALL Layers	1366
Pick Layers to turn Off	1367
Pick Layers to turn On	1367
Turn OFF ALL Layers	1367
Turn ON ALL Layers	1367
Pick Current Layer	1367
Elevations	1368
Descriptions	1368
Point Code	1368
Auto Point Number	1368
Auto Point Plot	1369
Auto Lines	1369
CG Snap	1369
Off	1370
Points	1370
Lines	1370
Points and Lines	1370
Zoom to Point ID	1370
Windows Calculator	1371
CGEditor	1371

Chapter 5. Civil Module	1417
SiteNet Menu	1418
3D Data Menu	1418
Change Elevations	1419
Set Polyline To Elevation	1420
Polyline Elevation Editor	1420
Wall Polyline Profile Editor	1426
2D to 3D Polyline by Surface Model	1427
2D to 3D Polyline by Screen Entities	1428
2D to 3D Polyline by Points	1429
2D to 3D Polyline by Slope From Reference	1430
2D to 3D Polyline by Text	1431
2D to 3D Polyline by Text With Leader	1432
2D to 3D Polyline by Start/End Elevations	1434
2D to 3D Polyline by Profile	1434

LWPolyline to 3D Polyline	1435
Assign Contour Elevations: Multiple in Series	1435
Assign Contour Elevations: From Contour Labels	1436
Assign Contour Elevations: Single Elevation Group	1437
3D Entity By Surface Model	1437
Pad Polyline By Interior Text	1438
Draw Building Envelope Polyline	1439
Define Lot Edge Grade Rules	1440
Elevate Lot Edges by Grade Rules	1442
Elevate Pads by Grade Rules	1444
Tag Pad Reference Point	1447
Tag Inner Pads	1448
Grade Rules Editor	1448
Convert Spot Elev To Points	1450
Draw Spot Elevations	1452
Move Spot Elevations	1455
Spot Elevation Links Manager	1455
Adjust Elevation Labels	1456
Elevate Text	1457
Interpolate Points	1457
Elevation Between Points	1457
3 D Points - Interpolate Entity	1458
Points by Slope Ratio	1459
Create Ridge Polylines From Contours	1460
Create Breaklines From Triangulation	1461
Offset 3D Polyline	1462
Fillet 3D Polyline	1463
Join 3D Polyline	1464
Merge Crossing 3D Polylines	1464
3D Polyline by Slope on Surface	1465
Elevate Intersection Curb Return	1466
Min/Max Slopes 3D Polyline	1469
Follow TIN Edges	1469
Extend To Elevation	1470
Break 3D Polyline by Surface	1471
Check 3D Polylines Match Vertices	1472
Add Points At Elevation	1473
Draw Template Polylines	1473

Draw Curb Ramp	1476
Label Pad Elevation	1477
Update Pad Elevation Labels	1478
Label Elevations Along Polyline	1479
Move Elevation Labels	1481
Label Polyline High/Low Points	1482
Label Polyline Segment	1483
Highlight Segments by Slope	1485
Highlight Crossing Breaklines	1486
Report 3D Polyline Station/Elevation	1487
Story Stake from Surface Entities	1488
Story Stake By Points/Polyline	1489
Tag Non-Surface Points	1491
Untag Non-Surface Points	1492
Report Non-Surface Points	1493
Non-Surface Entities	1494
Tag Hard Breakline Polylines	1495
Highlight Hard Breakline Polylines	1495
Identify Hard Breakline Polylines	1495
Untag Hard Breakline Polylines	1496
Extend 3D Polyline to Surface	1496
Surface Menu	1496
Tag Predefined Boundaries	1497
Identify Predefined Boundaries	1498
Untag Predefined Boundaries	1498
Triangulation Data Check	1498
Triangulate & Contour	1499
Triangulation File Utilities	1514
Surface Manager	1522
Rebuild Inside Perimeter	1531
Contour from Triangular Mesh	1532
Contours from Grid File	1533
Contour from TIN File	1540
Contour From Section File	1541
Smooth Contours	1542
Reduce Contour Vertices	1544
Edit Contours	1545
Contour ID	1546

Color Contours by Elevation	1547
Color Contours by Interval	1548
Highlight Index Contours	1549
Highlight Depression Contours	1549
Draw Contour Gradient Marks	1550
Change Contour-Plines Width	1551
Trim Contour-Plines by Pline	1551
Contour Elevation Label	1552
Local Elevation Label	1554
Draw Contour Label Marker Line	1554
Move Label Along Contour	1555
Flip Contour Labels-Text	1555
Tablet Calibrate	1555
Digitize Contours (Polyline)	1557
Display Last TIN Error Log	1558
Make 3D Grid File	1559
Grid File Utilities	1563
Edit 3D Grid	1584
Merge Grid Files	1584
One Triangulation Surface Volumes	1585
Two Triangulation Surface Volumes	1587
Three Triangulation Surface Volumes	1589
One Grid Surface Volumes	1591
Two Grid Surface Volumes	1593
Volumes By Layer	1597
Cut/Fill Report	1599
Cut/Fill Labels	1600
Cut/Fill Color Map	1602
Cut/Fill Slope Color Map	1604
Cut/Fill Grid Map	1605
Cut/Fill Contours	1607
Cut/Fill Slope Lines	1609
Cut/Fill Movement	1612
Cut/Fill Centroids	1616
Draw 3DPoly Perimeter	1617
Draw 3DPoly Base Breakline	1618
Calculate Stockpile Volume	1619
Calculate Pond/Pit Volume	1621

Design Pad Template	1623
Edit Pad Template	1632
Design Bench Pond	1634
Edit Bench Pond	1640
Design Valley Pond	1642
Draw Triangular Mesh	1645
Draw 3D Grid File	1646
Quick Draw Surface	1648
Quick Contours	1649
Draw Surface Boundary	1649
Draw Surface Intersection	1649
Surface Inspector	1650
Surface 3D FlyOver	1652
Surface 3D Viewer	1662
Motion 3D Viewer	1663
Linear 3D Object Polylines	1665
Render, Shade, and Hide Commands	1666
Line of Sight Viewshed	1666
Elevation Zone Analysis	1667
Slope Report	1670
Slope At Points	1672
Slope Zone Analysis	1674
Slope Direction Analysis	1678
Convert LDD Contours	1680
Convert LDT/Civil3D Surface Drawing	1681
Import NASA World Elevation Dataset	1681
Import National Elevation Dataset	1682
Import Google Earth Surface	1682
Import/Export Trimble TTM File	1683
Export DEM TIF File	1684
Export Topcon TIN File	1685
Centerline Menu	1685
Design Centerline	1685
Centerline Editor	1688
Polyline to Centerline File	1698
Edit Centerline On-Screen	1698
Draw Centerline File	1699
Centerline Report	1700

Centerline ID	1701
Station Polyline/Centerline	1702
Label Station-Offset	1711
Offset Point Entry	1715
Calculate Offsets	1718
Distance Between Two Entities	1719
Centerline Conversions	1720
Enter Right of Way	1721
Polyline to Right of Way	1722
Label/Draw Right of Way	1722
Horizontal Speed Table	1723
Profile Menu	1725
Quick Profile	1726
Profile from Surface Entities	1728
Profile from Grid or Triangulation Surface	1729
Profile from 3D Polyline	1730
Profile from 3D Points	1731
Profile from Section File	1732
Profile from Points on Centerline	1733
Profile from Polyline on Profile Grid	1734
Profile from Layers	1734
Profile from Pipe Polylines	1735
Profile from 3D Solids	1736
Enter Profile On-Screen	1736
Road Profile Editor	1738
Design Road Profile	1743
Create Curb Profiles	1745
Design Sewer/Pipe Profile	1749
Sewer/Pipe Profile Editor	1756
Profile Editor	1759
Draw Profile	1764
Draw Profile Grid	1812
Add Grid Ticks and Dots	1813
Add Grid Lines	1814
Adjust Profile Grid	1814
Adjust Draw Profile Settings	1815
Adjust Plan/Profile Sheet	1816
Move Sewer Profile Labels	1817

Draw Plan View Sheets	1818
Horizontal Axis Elevations	1821
Horizontal Axis Crossings	1822
Profile to 3D Polyline	1824
Profile To Points	1825
Profile Report	1827
Polyline Slope Report	1829
Station-Elevation-Slope Report	1830
Sag & Crest Report	1833
Pipe Depth Summary	1833
Label Profile On Centerline	1835
Profile ID	1839
Review Profile Links	1840
Trench Template Editor	1840
Draw Typical Trench Template	1841
Point Placement on Profile	1842
Restricted Grade Design	1844
Draw Single Manhole	1844
Best Fit Profile	1847
Merge Profiles	1848
Average Profiles	1849
Draw Pipe 3D Polyline	1849
Assign Pipe Width to Polyline	1850
Identify Pipe Polylines	1850
Label Pipe Polylines	1851
Profile Offset Text	1851
Label Profile Differentials	1853
Label Sewer Laterals	1854
International Roughness Index	1855
Restricted Grade Design	1857
Calculate Intersection Point	1857
Slope/Length/Elevation Calculator	1858
Sight Distance Design	1858
Plot VC from Tangents	1859
Enter Roadside Ditch	1859
Draw Roadside Ditch	1860
Profile Conversions	1861
Section Menu	1863

Section Alignment Editor	1864
Draw Section Alignment	1866
Sections from Surface Entities	1867
Sections from Grid or Triangulation Surface	1869
Sections from Polylines	1869
Sections from Points	1871
Sections from 3D Polylines	1872
Sections from Coordinate File	1873
Sections from Profile	1874
Sections from Layers	1874
Sections Points from Surface Entities	1875
Sections Points from Right of Way	1876
Sections Points from Pipes	1876
Quick Section	1876
Tablet Calibrate	1878
Digitize Sections Plan	1879
Digitize Sections XSec	1880
Digitize End Areas	1882
Section Conversions	1883
Create Overshoot Section File	1884
Draw SectionTemplate DWG	1884
Point Placement on Section	1885
Point Offset Report/Plot	1887
Section File Editor	1888
Draw Section File	1896
Section Report	1918
Offset & Elevation Report/Plot	1920
Elevation Zone Section Volumes	1922
Draw Mass Diagram	1923
Mass Diagram Report	1924
Mass Haul Report	1924
Mass Haul Analysis	1925
Cut/Fill Width Analysis	1929
Cut Sheet	1930
Design Regrade	1931
Calculate Haul Factors	1934
Sections to 3D Polylines	1935
Sections to Points	1936

Design Section Staging	1937
Draw Pipe 3D Polyline	1939
Assign Pipe Width to Polyline	1939
Slope Stake Report	1939
Update Sections from Slope Stake	1942
Extend Sections to Offset Limits	1942
Extend Progress Sections	1942
Section Area Report	1944
Slope Zone Section Analysis	1944
Color 3D Lines By Slope	1945
Regrade Fill Slope	1946
Overlay Section File	1946
Average Section Files	1947
Merge Sections	1948
Compare Section Files	1948
Move Section Leader Labels	1949
Update Sections from Polylines	1949
Review Section Links	1949
Section ID	1950
Set Descriptions By Offset	1950
Set Descriptions By Polylines	1950
Calculate Section Volumes	1951
Calculate End Area	1954
Edit Process End Area File	1955
Roads Menu	1956
Design Template	1956
Draw Typical Template	1972
Template Transition	1973
Template Grade Table	1977
Super Elevation Editor	1979
Draw Super Elevation Color Centerline	1982
Draw Super Elevation Diagram	1983
Template Series Editor	1989
Topsoil Removal/Replacement	1991
Assign Template Point Profile	1992
Assign Template Point Centerline	1994
Template Grade By 3D Polyline	1998
Edit Curb Ramps	1998

Tag Crub Ramps	2000
Road Rehabilitation Profile	2001
Define Median Adjustments	2014
Define Road Design Parameters	2016
Vehicle Path Tracking Profile	2017
Vehicle Path Tracking	2022
Template ID Library	2029
Draw Road Stripes	2029
Road Stripe Library	2030
Process Road Design	2031
Road Network	2061
Process Design Sections	2131
Road Design Inspector	2132
Locate Template Points	2133
Shrink/Swell Table	2136

Chapter 6. Hydrology Module 2138

Surface Menu	2139
Universal Soil Loss	2140
Watershed Menu	2143
Define Watershed Layers	2143
Watershed Analysis	2152
Merge Watersheds	2159
Run Off Tracking	2159
Watershed Above Points	2160
Watershed Area Grid	2162
Run Off Tracking - Grid Surface	2163
3D Polyline Flow Values	2164
Rainfall Frequency and Amount	2164
Rainfall Frequency and Amount - NOAA Atlas 14	2165
Rainfall Frequency and Amount - Lookup Table	2166
USDA Soil Area Download	2166
Sub-Watersheds By Land Use	2166
Curve Numbers & Runoff	2167
Calculate C-Factor	2170
Time of Concentration - SCS Method	2172
Time of Concentration - TR-55 Method	2173
Time of Concentration - Rational Method	2176
Time of Concentration - Kinematic Wave Method	2177

Time of Concentration - Regression Equation, LA County	2178
Time of Concentration - Maricopa County, AZ	2182
Time of Concentration - Kirpich Method	2184
Time of Concentration - Kerby-Kirpich Method	2185
Peak Flow - Graphical Method	2186
Peak Flow - Rational Method (General)	2187
Peak Flow - Rational Method(Riverside S. California)	2188
Peak Flow - Rational Method (KYDOT)	2189
Peak Flow - Tabular Hydrograph Method	2190
Peak Flow - Modified Rational Method (LA County)	2192
Watershed Settings (Save and Load)	2194
Pipe Routing Hydrograph	2194
Design Detention Pond	2196
Hydrograph Editor	2198
Runoff Hydrograph - SCS Method	2199
Runoff Hydrograph - TR-55 Tabular Method	2203
Runoff Hydrograph - Rational Method	2204
Pipe Routing Hydrograph	2207
Reservoir Routing Hydrograph	2209
Channel Routing Hydrograph - Convex Method	2210
Channel Routing Hydrograph - Modified Att-Kin Method	2212
Draw Flow Polylines TR20	2215
Locate Structures TR20	2215
Locate Reach	2215
Edit Layout Element	2215
Hydrograph Development	2215
Single Runoff Hydrograph	2215
Add Hydrographs	2215
Subtract Hydrographs	2216
Report Hydrograph	2216
Draw Hydrograph	2217
SEDCAD	2217
SEDCAD Draw Flow Polylines	2217
SEDCAD Locate Structures	2217
SEDCAD Label Structure Layout	2218
HEC-RAS_Manager	2219
Prepare HEC-RAS Input File	2230
Draw Hec-Ras Watermark	2234

Import Flow Velocity Points	2236
Import Flow Depth Points	2238
Channel Routing Hydrograph - Modified Att-Kin Method	2240
Structure Menu	2241
Design Detention Pond	2242
Detention Pond Sizing	2252
Detention Pond Sizing - Linear Storage Estimate Method	2253
Rectangular Pond Design	2254
Design Spillway	2257
Drop Pipe Spillway Design	2258
Rectangular Weir Design	2261
Advanced Weir Design	2262
Orifice Design	2265
Pond Exfiltration Design	2267
Multiple Outlet Design	2269
Stage-Storage Editor	2272
Calculate Stage-Storage	2276
Draw Stage-Storage Curve	2277
Stage-Discharge Editor	2283
Draw Stage-Discharge Graph	2284
Report Stage-Discharge	2285
Merge Stage-Discharge Files	2286
Channel Lining Analysis	2287
Channel Design - NonErodible Mannings Equation	2291
Channel Design - Erodible Mannings Equation	2293
Grass Channel Design	2294
Pipe Culvert Design	2304
Sewer Pipe Design: Individual	2318
Sewer Pipe Design: Sewer Network Segment	2319
Sewer Pipe Design: Read Profile	2319
Lift Station Design	2320
Network Menu	2322
Sewer Network Model	2323
Sewer Network Settings	2323
Set Sewer File	2333
Set Surface File	2333
Plan View Label Settings	2333
Save Sewer Network File	2339

Import Haestad Network	2339
Rational Rainfall Library	2340
SCS Rainfall Library	2348
Inlet Library	2355
Sewer Structure Library	2361
Pipe Size Library	2366
Curve Number Library	2369
Drainage Runoff Library	2370
Overland Sheet Flow Surface Mannings N Library	2371
Pavement Mannings N Library	2372
Pipe Manning's n Library	2373
Input Sewer Line	2374
Edit Sewer Structure	2375
Edit/Create Sanitary/Utility Structure	2389
Edit/Create Sewer Structure - Plan View	2395
Edit/Create Sewer Structure - Profile View	2395
Remove Sewer Structure	2397
Lateral Design Overview	2398
Create/Edit Lateral Structure	2404
Import Laterals From Text File	2410
Remove Lateral Structure	2410
Check Sewer Network Parameters	2411
Check Reference Centerlines and Surface	2411
Check/Label Clearance	2411
Collision Conflicts Check	2413
Find Sewer Structure	2417
Report Sewer Network	2418
Sewer Network Quantities Report	2422
Sewer Network Inspector	2423
Sewer Network Hydrographs	2424
Erase Plan View	2425
Transform Elevations	2426
Identify Sewer Network Entity	2426
Set Parameter For Entire Network	2427
Spreadsheet Sewer Editor	2427
Draw Sewer Network Plan View	2432
Draw Sewer Network Data Table	2432
Draw Sewer Network Centerlines	2435

Draw Sewer Network Profile	2435
Draw Sewer Network-3DFaces	2437
Move Sewer Label	2438
Pipe Elevation Label	2440
Draw IDF Curve	2441
Find And Replace Data Values	2443
Renumber Structure Names	2444
Set Rim Elevation to Match Surface	2445
Transform Coordinates	2445
Check/Label Clearance	2446
Compare Sewer Networks	2448
Move Sewer Profile Pipe Labels	2449
Pipe Network Drawing Links	2450
Pipe Network ID	2451
Edit/Create Structure With Inverts	2451
Review Sewer Network Links	2452
Review Sewer Profile Links	2452
Import Carlson Sewer Network	2453
Import Network From 3D Polylines	2454
Import Sewer Network From 2D Polylines	2455
Import Network from Centerline/Profile	2457
Import Network From Text File	2458
Import ADAC	2458
Import Network From Bentley	2458
Import StormCAD Network	2460
Import Sewer Network From Esri Shapefiles	2460
Export Carlson Sewer Network	2461
Export ADAC Network	2461
Export To Points	2462
Export To Profiles	2462
Export To Esri shapefiles	2463
Export Pre-Carlson 2020 Sewer Network	2463
EPA_SWMM	2464
Utility Network	2469
Pressure Pipe Network	2475
HydroCAD	2502
HydroNet Explorer	2502

Chapter 7. GIS Module	2515
GIS Data Menu	2516
Carlson GIS and Esri	2516
GIS Database Settings	2517
Define Template Database	2519
GIS Data Editor	2521
GIS Inspector	2522
GIS Query/Report	2523
Hatch GIS Polylines	2525
Mark GIS Polylines	2527
Data Capture From CAD Perperties	2528
Data Capture Text By Sample	2529
Data Capture Enclosed Text	2531
Data Capture Block Attributes	2533
Data Capture Add Point Data to Linework	2535
Label GIS Point Data	2537
Label GIS Polyline: Closed Polyline Image	2539
Label GIS Polyline: Closed Polyline Data	2541
Label GIS Polyline: Label Along Polylines	2542
Links Manager	2543
Create Links	2544
Erase Links	2545
Audit Links	2545
Create GIS Polylines by Interior Text	2546
Point GIS Editor	2547
Web Feature Service (WFS)	2558
ESRI ArcGIS Services	2560
Import SQLite	2561
Import ESRI SHP File	2562
Import ESRI GeoDatabase	2564
Import OpenGIS GML	2565
Import GIS Data from SurvCE	2566
Import ODBC data source	2566
ArcGIS Feature/Image/Map/Services	2568
Export SQLite File	2569
Export SHP File	2571
Export DWG File with Esri MSC	2572
Export ESRI Projection File	2573

Export ESRI File GeoDatabase	2573
Export GIS Data to SurvCE	2574
Export OpenGIS GML	2575
Convert GIS Links to AcadMap	2575
Convert AcadMap to GIS Links	2576
Database File Utilities	2577
Define Block Database Links	2578
Update Block Data	2579
Define Note File Prompts	2580
Insert Block with GIS Data	2582
Update Drawing Coordinates	2583
How to Setup ODBC Data Sources	2583
GIS Tools Menu	2585
Process Layerized Text	2586
Create Unique Polylines	2587
Label Object Data Areas	2588
Polygon Processor	2590
Polygon Inspector	2590
Polygon Query	2590
Polygon Export to Map	2591
Topology Area Check	2591
Topology Point Check	2593
Define Area Layers	2593
Report Areas By Layers	2594
Properties Converter	2594
Isolate Layers	2596
Mark Insertion Points	2596
Write Block by Polyline	2596
Insert with Join/Align	2597
Snap Linework To Points	2597
Set Text Layer By Text String	2597
GIS Image Menu	2598
Create World File from Image Alignment	2598
Create World File by Image in Drawing	2599
Create Image World File	2599
Edit World File	2599
Place Image by World File	2600
Place Image by World File	2600

Place Google Earth Image	2601
Place Web Map Service (WMS) Image	2603
Convert CAL Images	2605
Import MrSID Images	2605
Image 3D Viewer	2605
Drape Image on Surface	2606
Color TIN By Image	2607
Create Image from Drawing	2608
Fit Image To Aspect Ratio	2608
Image Inspector	2609
Place Camera Symbol/Image	2611
Attach Image to Entity	2611
Detach Image From Entities	2612
Audit Image Links	2613
Draw Entity Images	2613
Image Set Manager	2614
Load Image Set	2621
Draw Image Boundaries	2621
Place Image By Point	2621
Place Image By Circle	2622
Image Zoom Window	2622
Image Zoom Previous	2622
Image Zoom Extents	2622
Image Pan	2622
Image Redraw	2622
Clear Image Set	2623
Chapter 8. Construction Module	2624
Surface	2625
Triangulate & Contour	2625
Triangulation File Utilities	2634
Elevate Menu	2637
Change Elevations	2637
Set Polyline to Elevation	2638
Polyline Elevation Editor	2638
2D to 3D By Surface Model	2644
2D to 3D Polyline by Points	2645
2D to 3D Polyline-By Text	2646
2D to 3D By Text With Leader	2647

2D to 3D Polyline by Start-End Elevations	2649
Draw Building Envelope Polyline	2649
Pad Polyline By Interior Text	2650
Convert Spot Elev To Points	2652
Assign Contour Elevation - Multiple in Series	2655
Assign Contour Elevation - From Contour Labels	2656
Assign Contour Elevation - Single Elevation Group	2657
Drape 3D Polyline On Surface	2657
Edit Polyline Vertex	2658
Edit Contours	2659
Snap Contours to 3D Polylines	2660
Takeoff Menu	2661
Project_Checklist	2661
Define Layer Target/Material/Subgrade	2661
Edit Selected Layer	2665
Set Layer For Existing	2665
Set Layer For Design	2666
Set Layer For Other	2666
Update Colors For Set Elevations	2666
Boundary Polyline	2666
Areas Of Interest	2667
Hatch Subgrade Areas	2669
Erase Subgrade Hatches	2671
Draw Subgrade Hatch Legend	2671
Report Subgrade Areas	2671
Label Subgrade Areas	2672
Erase Subgrade Labels	2672
Topsoil Removal and Replacement	2672
Special Fill Areas	2675
Perimeter Polylines Properties	2676
Make Existing Ground Surface	2676
Make Design Surface	2678
View Overexcavate Surface	2679
Make Overexcate Surface From Existing/Design Surfaces	2680
Make Overexcavate Surface From Surface File	2680
Make Overexcavate Surface From Strata	2680
Make Overexcavate Surface From Screen Entities	2680
Adjust Overexcavate Surface	2681

Draw Overexcavate Surface 3D Faces	2681
Erase Overexcavate Surface 3D Faces	2681
Draw Overexcavate Cut Color Map	2681
Erase Overexcavate Cut Color Map	2681
Clear Overexcavate Surface	2681
Make Top Surface	2682
View Top Surface	2683
Draw Top Surface 3D Faces	2684
Removal Settings	2684
Removal Manager	2685
Import Removal Text ASCII File	2685
Draw Removal Field to Finish	2686
Draw Removal Breakline	2686
Removal Entities	2687
Set Removal Boundary	2688
Make Removal Surface	2688
Draw Removal Surface	2688
Draw Removal Contours	2689
Make Combined Removal Surface	2689
Calculate Removals Volumes	2689
3D Drive Simulation	2690
FlyOver Along 3D Polyline	2691
Surface Manager	2692
Surface Inspector	2693
Surface Report	2694
Graphic Reports	2695
Quick Profile	2700
Cut/Fill Centroids	2701
Cut/Fill Map Legend	2703
Color Surface By Layer	2703
Draw Surface As Grid	2704
Import/Export Carlson Triangulation Files	2705
Make User Defined Surface	2705
Volumes By Triangulation	2706
Calculate Stockpile Volume	2706
Calculate Pond/Pit Volume	2708
Set Active Surfaces	2709
Design Surface Vertical Offset	2710

Existing Surface Vertical Offset	2710
Merge Existing With Design	2710
Calculate Total Volumes	2710
Calculate Volumes Inside Perimeter	2714
Draw 3DPoly Perimeter	2714
Draw 3DPoly Base Breakline	2715
Design Surface 3D Viewer	2716
Existing Surface 3D Viewer	2717
Import MicroStation DGN	2718
Material Quantities	2718
Linework Length Report	2725
Tag Segments To Skip Linear	2725
Cost Summary Report	2726
Make Layout	2727
Place Image On Layout	2728
Save Layout Layers	2729
Restore Layout Layers	2730
Convert Dashes To Polylines	2730
Roads Menu	2731
Sections From Polylines On Section Grids	2732
Offset 3D Polyline By Grade Table	2733
Display Menu	2734
Existing Drawing	2735
Existing Contours	2735
Existing Surface	2736
Design Drawing	2737
Design Contours	2737
Design Surface	2738
Cut/Fill Contours	2738
Cut/Fill Labels	2739
Cut/Fill Color Map	2740
Other Drawing	2740
Display Options	2741
Chapter 9. GeoTech Module	2748
Drillhole Menu	2749
Drillhole Strata Settings	2749
Drillhole Import	2751
Place Drillhole	2753

Edit Drillhole	2754
Label Drillhole	2756
Strata Polylines	2757
Drillhole Reports	2758
Make Strata Surface	2760
Geologic Column Profile	2760
Clear Strata Surface	2766
Draw Strata Cut Depth Contours	2766
Erase Strata Cut Depth Contours	2767
Draw Strata Cut Color Map	2767
Erase Strata Cut Color Map	2767
Draw Strata Surface	2767
Erase Strata Surface	2768
Map Menu	2768
Plan View To PDF	2768

Chapter 10. Trench Module 2770

Trench Menu	2771
Input Trench Line	2771
Create Trench Network Structure	2773
Edit Trench Network Structure	2775
Trench Spreadsheet Editor	2775
Remove Trench Network Structure	2776
Find Trench Network Structure	2777
Import Trench Network Data - Utility Network	2777
Import Trench Network Data - Utility Network	2777
Export Trench Network Data	2777
Trench Network File Backup	2778
Plain View Label Settings	2778
Draw Trench Network - Plan	2779
Draw Trench Network Centerline	2779
Draw Trench Network - Profile	2780
Define Pipe Groups	2783
Trench Template Editor	2784
Draw Typical Trench Template	2785
Trench Subgrade Areas	2786
Trench Network Quantities	2787
Trench Quantities By Profiles	2790
Report Trench Network	2790

Digitize Menu	2793
Tablet On	2793
Tablet Off	2793
Tablet Calibrate	2793
Digitizer Setup	2795
Save Tablet Calibration	2796
Load Tablet Calibration	2796
Digitize	2796
Digitizer Settings	2816
Digitize Existing	2818
Digitize Design	2818
Digitize Other	2818
Digitize Point	2819
Digitize Spot Elevation	2819
Digitize 2D Polyline	2820
Digitize 3D Polyline	2822
Digitize Rectangle	2823
Digitize Perimeter	2824
Digitize Areas	2824
Digitize Contour Polyline	2825
Digitize Profiles	2826
Digitize Sections	2828
Digitize End Areas	2830
Raster Menu	2831
Draw Raster Image	2832
Set Raster Image	2834
Save As Raster Image	2835
Raster Edit Options	2835
Trace Circle	2837
Trace Rectangle	2838
Trace Line	2839
Trace Polyline	2839
Raster Nearest Snap	2843
Raster EndPoint Snap	2843
Merge Raster Files	2843
Cut Image	2845
Crop Image	2845

Remove Speckles	2846
Mirror Image	2846
Decompress Image	2847
Deskew Image	2847
Shrink Resolution	2847
Black/White By Threshold	2848
Replace Color	2848
Brighten/Darken Image	2849
Negative Image	2849
Rubber Sheet	2849
Undo Raster Edit	2851
Import Raster To Vector	2852
Raster To Text	2852
Vector To Text	2854
Set Text Color	2855
Set Background Color	2855
BIM Menu	2856
Import SketchUp File to Drawing	2856
Import Architectural IFC File to Drawing	2857
Import Architectural IFC File to Model	2857
Pipe Connector	2858
Model 3D Viewer	2860
Export Surface Architectural IFC File	2860
Export Solid Model IFC File	2861
PDF Menu	2861
Import PDF File	2862
Convert Polylines To Text	2864
Convert Dashes To Polylines	2865
Drawing to 3D PDF	2867
3D Viewer to 3D PDF	2867
Convert LDD Contours	2868
Chapter 12. Field Module	2869
COGO Menu	2870
Tape Baseline	2870
Cutsheet Spreadsheet Editor	2871
Field Menu	2872
Configure Field	2872
Equipment Setup	2885

Align GPS To Local Coordinates	2911
Point Store	2915
StakeOut	2923
Auto Points at Interval	2926
Track Position	2928
Satellite SkyPlot	2929
Monitor GPS Position	2930
Benchmark	2930
Resection	2931
Building Face Surface	2934
Pattern Point Survey	2934
Point Check By Robotics	2935
Pipe Network	2935
Carlson Field Icon Menu	2937
Typical Alignment Scenarios	2938
Surface Menu	2939
Elevation Difference	2939
Prepare Story Stake	2941
Story Stake By Points	2942
Story Stake Along Polyline	2943
Roads Menu	2944
Centerline Position	2945
Offset Stakeout	2947
Slope Staking	2951
Slope Inspector	2955
GIS Menu	2956
Equipment Menu	2956
Apache Lightbar	2956
CSI GBX Pro	2957
Depth Sounder	2958
Geodimeter	2959
InnerSpace Tech depth sounder	2960
Laser Atlanta	2960
Leica Disto	2961
Leica GPS System 500	2961
Leica TC Series	2962
Manual Total Station	2964
Mikrofyn Lightbar	2964

Navcom Configuration Guide	2964
Navcom GPS Setup	2966
Nikon Total Stations	2969
OmniStar Otto	2970
Simulation GPS	2970
Sokkia	2971
Topcon Total Stations	2973
Trimble	2975

Chapter 13. Point Clouds Module 2979

Point Cloud Recommended Specifications	2980
Point Clouds Getting Started	2980
Point Cloud Setup	2981
Point Clouds Step-by-Step Tutorial	2981
Point Clouds Project Manager	2992
Scene Tab	2993
Camera Tab	3002
Data Tab	3004
Project Tab	3005
Project Settings	3007
PointCloud Viewer	3012
Scenes	3017
Data Objects	3020
Adjust Color	3026
Change Projection	3026
Control Point Editor	3027
Coordinate Points	3031
Target Points	3037
Create Solid	3040
Cloud Objects	3041
Create Cloud	3042
Clean Cloud	3043
Import Cloud	3045
Export Cloud	3047
Merge Clouds	3049
Register Cloud	3050
Resample Cloud	3052
Smooth Cloud	3054
Subtract Cloud	3054

Transform Cloud	3055
Scale Cloud	3056
Instrument Data Project Items	3057
Processed Data Project Items	3059
Scans	3061
TINs	3067
Action Tab	3073
Bare Earth	3076
Create Point	3078
Create Grid	3086
Create Polyline	3087
Create Text	3096
Extract Breaklines	3098
Extract Color Lines	3101
Extract Contours	3107
Extract Curb	3109
Extract Intersection	3110
Extract Plane	3111
Extract Power Lines	3113
Extract Profile	3114
Extract Polyline	3116
Extract Sections	3118
Extract Slices	3120
Extract Slope Edge	3121
Extract Volume	3123
Image Drape	3124
The Transformation Panel	3125
Common Utilities	3129
Layer Properties Manager	3129
Polyline Editor	3131
Point Editor	3134
Item Properties	3140
Chapter 14. Natural Regrade Module	3142
Highlight Tractive Force Zones	3143
Problems Addressed by Natural Regrade with GeoFluv	3145
The Fluvial Geomorphic Solution	3146
Description of Software	3148
Links with Other Software	3157

Software Compatibility	3158
Data Entry	3158
DWG Tab	3158
Summary	3159
Documentation References	3160
Natural Regrade Menu	3160
Design GeoFluv Regrade	3160
Natural Regrade File	3161
Natural Regrade Global Settings	3162
Setup Tab	3165
Select GeoFluv Boundary	3166
Select Main Channel	3166
Data for Main Channel	3167
Pre-disturbed Surface	3169
Channels Tab	3169
Channel Add	3170
Channel Delete	3171
Channel Name	3171
Channel Transition	3171
Current Channel	3171
Current Channel Settings	3172
Data for Current Channel	3174
Profile	3175
Report	3175
Output Tab	3176
Preview	3177
Data for GeoFluv Work Area	3178
Draw Design Surface	3178
Save Design Surface	3180
Update Cut/Fill	3180
Summary Report	3182
DWG Tab	3183
Draw GeoFluv Contours	3183
3D GeoFluv Contour Viewer	3185
3D GeoFluv Surface Viewer	3187
Calculate GeoFluv Volume	3188
Cut/Fill Centroids	3188
GeoFluv Channel Cross-Section Report	3191

GeoFluv Channel Inspector	3192
View Longitudinal Profile	3193
Edit Longitudinal Profile	3194
Auto Longitudinal Profile	3195
RIVERMorph tab	3197
Settings	3199
Reread Valley Bottoms	3202
Create Vegetation Scene	3202

Chapter 15. Geology Module 3205

Drillhole Menu	3206
Mining Project Settings	3206
Define Drillhole	3207
Define Strata/Bed	3209
Define Lookup Database	3214
Define Geologic Order	3216
Define Attributes	3216
Define Equations	3217
Define Ferm Codes	3218
Define Horizon Codes	3219
Import Drillhole	3220
Import Qualities	3231
Import Bed Names	3232
Import Dip/Azimuth	3233
Coal Section to Drillhole	3234
Reassign Database File	3235
Audit Database File	3235
Convert Drillholes to External Database/Convert Drillholes to Drawing Data	3235
Export Drillholes	3235
Drillholes to Points	3236
Place Drillholes	3236
Spot Drillholes	3238
Edit Drillhole	3238
Drillhole Data Sheet	3243
Pit/Channel Samples - Import from Text File	3244
Pit/Channel Samples - Import from CRD File	3245
Pit/Channel Samples - Import from Coal Section	3246
Export Pit/Channel Samples	3247
Draw Pit/Channel Sample Text	3247

Place Pit/Channel Sample	3248
Edit Pit/Channel Sample	3249
Standard Drillhole Text	3249
Drillhole Text Formatter	3252
Draw Geologic Column	3255
Quick Geologic Column	3272
Find Drillhole	3273
Drillhole Inspector	3274
Select Drillholes By Filter	3275
Drillhole Core Images	3276
Set Google Earth Drillhole Tags	3277
Draw eLogs	3278
Create Drillhole From eLog	3281
Inspect eLog	3285
Apply New Definition	3285
Change Drillhole Symbol Number	3285
Change Drillhole Symbol Size	3286
Change Process On/Off Status	3286
Change Drillhole Surface Z	3286
Assign Strata Correlation	3287
Composite Duplicate Strata	3288
Composite Strata By Beds	3289
Composite Strata By Interval	3289
Composite Strata by Elevation	3291
Set Strata Key Status	3294
Fill-in Missing Key as Zero	3294
Add Strata	3294
Delete Strata	3296
Rename Strata	3296
Reset Invalid Attribute	3296
Delete Attribute	3297
Rename Attribute	3297
Assign Bed Names	3297
Fill-in Bed Names	3298
Remove All Non-Key Bed Names	3298
Remove Bed Names After Last Key	3298
Split Bed	3299
Rename Bed	3299

Statistical Analysis	3300
Standard Drillhole Report	3302
Custom Drillhole Report	3303
Graphic Drillhole Report	3304
Change History Report	3312
Duplicate Drillhole Report	3312
Drillholes Without Key Strata	3313
Key Strata Missing Bed	3314
Compare Strata To Model	3314
Drillhole Top to Surface Model	3315
Bed Composite Report	3315
Invalid Strata Report	3317
Missing Strata Report	3318
Strata Order and Count Report	3318
Attribute Validation Report	3319
Define Parameters	3320
Compliance Drillhole Report	3323
Compliance Areas from Grids	3325
Compliance Areas from Drillholes	3326
Mark Compliance Drillholes	3328
Set Strata by Parameters	3329
Split Bed by Parameters	3330
Tag Drillholes for Processing	3331
StrataCalc Menu	3332
StrataCalc Data Sheet	3332
Draw Outcrops	3335
Draw Depth Contours	3338
Make Strata Grid Files	3339
Define Strata Grids AutoRun	3347
Run Strata Grids AutoRun	3352
Strata Isopach Maps	3352
Define Strata Isopach AutoRun	3354
Run Isopach Maps AutoRun	3356
Smooth Contours	3356
Reduce Contour Vertices	3358
Edit Contours	3359
Contour Elevation Label	3360
Move Label Along Contour	3362

Flip Contour Labels-Text	3363
Strata Quantities in Series	3363
Selected Strata Quantities	3365
AutoRun Strata Quantities	3367
Define Geologic Model	3368
Define Mining Model	3371
Geologic To Mining Model	3371
Draw Fault Line	3372
Import Fault Line	3373
Edit Fault Line	3374
Calculate Fault Shift	3376
Report Fault Lines	3376
Highlight Fault Lines	3377
Identify Fault Polylines	3378
Draw Fault Surface	3378
Draw Fault Labels	3379
Draw Fault Symbols	3379
Draw Heave Zones	3380
Tag Heave Zones	3381
Create Strata Polylines at Faults	3382
Trim Isopach Within Fault Zone	3383
Apply Faults to Grid	3383
Remove Faults From Grid	3386
Make Unfaulted Strata Grid	3386
Strike-Dip Symbol Editor	3386
Draw Strike-Dip Symbol	3388
Tag Strata Polylines	3389
Offset Strata Polylines	3389
Report Strata Polylines	3390
Highlight Strata Polylines	3390
Identify Strata Polylines	3391
Untag Strata Polylines	3391
Name Limit Polylines	3391
Report Limit Polylines	3393
Highlight Limit Polylines	3394
Identify Limit Polylines	3394
UnTag Limit Polylines	3395
Define Washing Curve	3396

Apply Wash	3398
Calculate Variogram	3403
Surface Mine Reserves	3412
Define Surface Mine Auto Run	3426
Underground Mine Reserves	3427
Blending Weighted Average	3430
Calculate Residuals	3431
AutoRun Residuals	3433
Fence Diagram	3435
Quick Fence	3444
Fence Polylines	3445
Block Diagram	3446
Voronoi Diagram	3448
Geologic Model Viewer	3450
Color Elev Grid by Strata	3450
Block Model Menu	3451
Make Block Model	3452
Define Block Model Auto-Run	3458
Block Model Editor	3466
Import Block Model	3467
Export Block Model	3468
Define Grade Parameters	3469
Grade Parameters By Attribute Range	3472
Draw Block Model	3472
Draw Grade Boundary	3474
Make Grade Solid Model	3474
Label Block Model	3476
Block Model Inspector	3477
Compare Drillholes to Block Model	3479
Block Model 3D Viewer	3479
Block Model Statistics	3483
Color Pits by Grade Parameters	3484
Color Elev Grid by Block Model	3485
Crop Block Model	3487
Attribute Grid from Block Model	3487
Prepare Value Block Model	3488
Optimized Pit Design	3496
Grids Menu	3502

Make 3D Grid File	3502
Make Top Of Key Grid	3506
Make Nearest Data Point Grid	3507
Surface Inspector	3509
Grid History Review	3510
Draw 3D Grid File	3512
Draw Surface Boundary	3514
Draw Surface Intersection	3515
Contour From Grid File	3515
One Surface Volumes	3522
Two Surface Volumes	3525
Grid File Utilities	3529
Merge Grid Files	3551
Merge Elevation for Zero Thickness	3552
Cleanup Grid Area	3552
Resource-Reserve Classification	3552
Convert As Determined Qualities	3556
Composite Qualities Analysis	3557
Case Studies	3558
Case Study #1: Techniques of Geological Compositing	3558
Case Study #2: Outcrop and Subcrop Modeling	3570
Case Study #3: Techniques Of Gridding	3579
Case Study #4: Limestone Block Modeling	3589
Case Study #5: Block Modeling by Quality Attributes	3611
Chapter 16. Underground Mining Module	3621
Notes Menu	3622
Insert Mining Symbols	3622
Edit Mining Symbols Library	3624
Locate by Bearing	3625
Locate by Azimuth	3625
Mine Note Defaults	3625
Mine Note Left/Right/Face	3627
Mine Note Auto Left/Right	3628
Mine Note From CoORD File	3628
Mine Note on Centerline	3629
Offsets from ASCII File	3629
Elevation-Thickness Points	3630
Mine Map Insert Commands	3631

Dynamic Blocks and Tool Palettes	3633
Works Menu	3641
Basic Projections	3642
Advanced Projections	3644
Projections & Ventilation	3653
Rooms	3657
Label Projection Distances	3658
Panel & Room Label Block	3658
Draw Outline	3659
Draw Perimeter	3659
Draw Pillars	3660
Highlight Unclosed Plines	3660
Chamfer Pillars	3660
AutoMine Connections	3661
Auto-Connect Pillars	3664
Pillar Cut	3666
Boundary Enclosure	3667
Draw 3D Mine Model	3669
Configure Section Info	3670
Place Coal Sections	3672
Edit Coal Sections	3673
Import Coal Sections From Text File	3673
Export Coal Sections To Text File	3673
Quantities by Avg Method	3674
Quantities by Grid Method	3677
Quantities by Centerlines	3680
Report Tons & Acres	3682
Property Menu	3684
Assign Property Names	3685
Remove Property Data	3685
Property Data Report	3686
Setup Property Link	3686
Property Names By Text	3689
Identify Property Lines	3689
Label Property Lines	3690
Find Property Lines	3691
Show All Property Lines	3691
Break Polyline by Property	3692

Check Property Polyline Overlaps	3692
Extract Centroid Data	3693
Underground Menu	3693
Mining Project Settings	3694
Timing Project Manager	3695
Set Attribute by Grid File	3709
Edit Panel Attributes	3710
Assign Panel Attributes	3711
Reset Panel Attributes	3712
Subdivide Panel	3712
Place Panel	3713
Pick & Place Panel	3715
Auto Panel Layout	3717
Auto Place Panel by Text	3719
Edit Panel	3720
Update Edited Panel	3724
Recalculate Extraction	3726
Panel Precedence By Polylines	3727
Find Panel	3729
Remove Panel	3729
Reverse Panel	3729
Connect Mine Plans	3730
Disconnect Mineplans	3733
Import Mine Plan	3733
Add Pin Point	3735
Remove Pin Point	3737
Set Current Position	3737
Panels Report	3739
Underground Timing	3739
Clear Timing Map	3760
Solid Menu	3762
Make Solid From Surfaces	3762
Make Solid From Sections	3770
Make Solid From Survey Points	3772
Make Solid From Points By Poisson	3773
Make Solid From Points By Sections	3776
Make Solid From Points By 3D Triangulation (Projection)	3778
Make Solid From Points By 3D Triangulation (Tunnel)	3780

Make Solid From Points By 3D Triangulation (Advancing Front)	3782
Make Solid From Polylines	3784
Make Solid From Design	3785
Make Solid From Entities	3786
Tunnel Network	3786
View Solid	3790
Draw Solid 3D Faces	3792
Draw 3D Solid	3792
Draw Solid Centerline	3793
Draw Solid Perimeter	3793
Section Solid	3793
Profile Solid	3795
Calculate Solid Volumes	3795
Assign Solid Data To Polyline	3796
Add Pinpoint To Solid	3797
Solid Timing	3797
View 3D Timing	3816
Clear Solid Timing	3820
Import File To Solid	3820
Export Solid	3821
Smooth Points	3821
Reduce Points	3821
Erase Isolated Points	3822
Create Gap Points	3822
Interior Points Filter	3823
Edit Solid	3824
Simplify Solid	3825
Heal Solid	3826
Remove Solid Dangles	3826
Smooth Solid	3826
Clean Solid	3827
Refine Solid	3827
Label Solid	3828
Report Offset Point To Solid	3828
Merge Solids	3829
Combine Solids	3830
Add Points To Solid	3831
Translate Solid	3831

Rotate Solid	3832
Scale Solid	3832
Flip Solid Order	3833
Intersect Solids	3833
Trim Solid	3833
Color Solid	3834
Solid Information	3834
ID Solid	3834
Triangulate & Contour	3835
Section Alignment Editor	3850
Draw Section Alignment	3852
Sections from Grid or Triangulation Surface	3853
Sections from Polylines	3853
Sections from Points	3855
Section File Editor	3856
Draw Section File	3864
Sections to 3D Polylines	3886
Design Tunnel Template	3887
Draw Typical Tunnel Template	3888
Tunnel Series Editor	3888
Tunnel Section Comparison Report	3889
Centerline Editor	3889
Polyline to Centerline File	3899
Draw Centerline File	3899
Profile Editor	3901
Profile from 3D Polyline	3905
Subsidence Menu	3906
DXFOUT Mine Plan to SDPS	3906
Prediction Point Output	3907
Post-Subsidence Contours	3908
Contour SDPS Output File	3909
Draw Subsidence Profile	3910
Suggested Standards	3911
Case Studies	3917
Case Study #6: Underground Mine Layout and Timing	3917
Case Study #8, Part 1: Underground Mine Mapping Procedure	3931
Case Study #8, Part 2: Computing Tonnage and Acreage	3942

Boundary Menu	3950
Name Pit Polylines	3950
Assign Pit Names By Layer	3951
Find Pit	3951
Label Pit/Site Names	3951
Pit Label Formatter	3951
Hatch Pits	3953
Identify Pit Polylines	3954
Remove Pit Names	3955
Check Pit Polyline Overlaps	3955
Pit by Interior Point	3955
Pit By Interior Text	3956
Pit Matrix Layout	3957
Pit Layout by Advance	3963
Pit Layout by Width	3965
Pit Layout by Rate	3966
Import Pit Points	3968
Merge Pits	3969
Assign Directions	3970
Display Directions	3975
Reverse Directions	3976
Clear Directions	3976
Assign Pit Precedence	3977
Pit Precedence by Polylines	3977
Clear Pit Precedence	3979
Clear Pit Bench Quantities	3979
Remove Empty Benches	3980
Merge Small Benches	3980
Assign Pit Attributes	3982
Reassign Pit Attr Grids Folder	3983
Assign Timing Grids	3983
Reassign Timing Grids Folder	3984
Import Pit Timing Data	3984
Pit Quantities Report	3986
Pit Points Report	3987
Edit Pit	3988
Pit Inspector	3994

Assign Property Names	3996
Remove Property Data	3996
Property Data Report	3996
Setup Property Link	3997
Property Names By Text	4000
Identify Property Lines	4000
Label Property Lines	4001
Find Property Lines	4002
Show All Property Lines	4002
Check Property Polyline Overlaps	4003
Break Polyline by Property	4003
Extract Centroid Data	4004
Reserves Timing Menu	4004
Mining Project Settings	4004
Surface Mine Reserves	4005
Define Surface Mine Auto-Run	4020
Timing Project Manager	4021
Set Attribute By Grid File	4037
Surface Production Timing	4038
Pit Scheduler	4045
Surface Equipment Timing	4050
View 3D Surface History	4081
View 3D Timing	4083
Clear Timing Map	4087
Haul Road Manager	4088
Haul Fleet Manager	4091
Haul Cycle Analysis	4093
Gantt Charts	4104
Surface Menu	4110
Define Dragline Equipment	4110
3D Dragline	4111
Range Diagram	4114
Dozer Push	4124
Draw Dragline Limits	4129
Cut and Place (Spoil Removal)	4130
Cut Only (Coal Removal)	4134
Flatten Spoil Top	4135
Cast Blast Profile	4136

Polyline to Centerline File	4138
Area To Section Report	4138
Dragline Section Report	4139
Edit Dragline Sequence	4140
Process Dragline Sequence	4141
Tag Slope Groups	4145
Design Dragline Pit	4145
Design Bench Pit	4148
Design Spoil Pile	4154
Design Fill Surface	4156
Design Ramp Profile	4160
Vertical Pit Quantities	4161
Update Grid File	4162
Dragline Pits	4163
Regrade Backfill	4164
Blast Pattern Layout	4166
Interval Along Entity	4168
Blast Point Report	4168
Cast Blast Report	4169
Output To Reame	4170
Draw Reame File	4172
Output to SB-Slope	4172
Draw SB-Slope File	4172
Carlson Grade Project	4173
3D Grade Simulation	4174
Spoil Menu	4175
Assign Spoil Names	4176
Spoil by Interior Point	4176
Spoil By Interior Text	4178
Spoil Layout by Width	4178
Find Spoil Name	4179
Label Spoil Names	4179
Remove Spoil Names	4180
Clear Spoil Volumes	4181
Merge Spoils	4181
Assign Directions	4182
Display Directions	4185
Reverse Directions	4186

Clear Directions	4187
Calculate Spoil Volumes	4187
Edit Spoil Volumes	4189
Edit Spoil Source	4190
Spoil Placement Timing	4191
Case Studies	4199
Case Study #7: Dragline Range Diagrams	4199
Case Study #9: Surface Timing With Benches	4216
Chapter 18. Tutorials	4235
CAD Basics	4236
Entering a Deed	4245
Making a Plat	4281
Field to Finish for Faster Drafting	4318
Intersections and Subdivisions	4341
SurvNET	4366
Contouring, DTM and Design	4396
Contouring, Break Lines and Stockpiles	4411
A Dozen Tools for Surface Design	4428
Calculate Volumes by Five Methods	4464
Basic Road Design with Volumes	4488
Road Rehabilitation	4514
Hydrology and Watershed Analysis	4530
Stormwater Network Design	4556
Data Extraction for HydroCAD	4584
ESRI to Office to Field and Back	4596
Takeoff/SiteNET CAD File From Start To Finish	4608
GeoTech Drillholes and Strata	4627
Takeoff Trench Network Quantities	4635
Digitizing	4648
PDF Section Import	4667
GeoTech Reports	4675
Cut Sheet	4690
Geodetic Reports	4705
Pressure Pipe Network Design	4712

Chapter 19. LDT Migration Guide	4729
Data File Types and Storage	4730
Settings	4731
Survey	4732
Points and Point Groups	4737
Surfaces and Contours	4737
Line and Curve Labeling	4739
Volumes	4740
Alignments	4740
Profiles	4741
Roadway Cross Sections	4742
Roadway Templates	4743
Design Control	4744
Roadway Intersections	4745
Cul-de-sacs	4746
Grading	4747
LandXML Data Transfer	4747
Pipes	4747
Lots	4747
Text Height	4748
North Rotation	4749
Drawing Cleanup	4749
 Chapter 20. Programming Interfaces	 4751
LSP Functions	4752
Coordinate API	4753
DTM API	4759
Road API	4764
3D Viewer API	4766
Lat/Lon and Grid Projections API	4767
 Index	 4767

Introduction

1

Carlson 2024 is application software for land development, civil engineering, surveying, construction, GIS and mining engineering which runs with AutoCAD and IntelliCAD. The CAD engine serves as the graphics engine and drawing editor for Carlson.

Carlson 2024 consists of the following programs: Survey, Civil, Hydrology, GIS, Field, GeoTech, Trenching, CAD Net, Construction, Natural Regrade, Point Clouds, Geology, Underground Mining and Surface Mining. Each of these programs can run independently or together with each other.

Carlson Directory Structure

The figure below shows the default directory structure for the Carlson program files. The EXEC, LSP, and Support sub-directories have files that are accessed by the program. You should never need to access these files directly yourself. The Carlson Projects sub-directories are the user's directories.

ROOT

\Program Files\Carlson2024

\EXEC

Executables

- Programs (.EXE, .DTA)

\Support - a set of fixed program support files that are not user-customized

Support

- Help files (.CHM, .PDF)

- Shared programs (.DLL)

\SupTemplate - a set of support files to be copied to individual user's profile

Support

- Drawing blocks (.DWG)

- Menus (.MNU, .CUI)

\LSP

Lisp & ARX

- Programs (.LSP, .DLL, .ARX, .DCL)

\UserTemplate - a set of files to be copied to individual user's profile

User setup files

- Settings files (.INI)

\Carlson Projects (customizable during installation)

User data files

- Survey files (.RW5, .CRD, etc)

Startup directory, user drawings

- Drawings (.DWG)

Each user on multi-user system gets his own copy of files from support and settings areas. Depending on your operating system the location will vary:

On Windows XP and earlier it would typically be: C:\Documents and Settings*Your User Name*\Application Data\Carlson Software\Carlson2024

On Windows Vista would typically be: C:\Users*Your User Name*\AppData\Local\Carlson Software\Carlson2024

On Windows 7 would typically be: C:\Users*Your User Name*\AppData\Roaming\Carlson Software\Carlson2024 followed by folder name specific to the version of AutoCAD or IntelliCAD you are running. Please note that by default Windows will be hiding the "Application Data" ("AppData" on Vista) location from browsing, but you can still type it in, or change viewing preferences.

Installation Guide

Workstation Installation - Step-by-step instructions. For network licensing server installation instructions, please see http://www.carlsonsw.com/support/knowledge-base/?action=display_topic&topic_id=55.

What's New - Complete list of improvements for Carlson 2024.

Registration

Documentation - For full documentation, including context sensitive help, press the F1 key at any time while running Carlson 2024.

Fixes - A complete list of fixes for Carlson 2024 can be found on at the Carlson Software Knowledge Base.

Support - List of all support resources.

System Requirements

Carlson's system requirements are no greater than that of the AutoCAD® version you are running. See your AutoCAD installation guide for the minimum system configuration. It is always recommended that you use the highest performance PC possible.

For detailed information on system requirements please see this page: <http://www.carlsonsw.com/support/system-requirement/>

64-bit Windows (for details on Windows support for specific AutoCAD version, please refer here):

Carlson 2024 will operate with the following versions of CAD:

IntelliCAD 10.0 built-in (supports Windows Vista through Windows 10)

AutoCAD 2017-2024

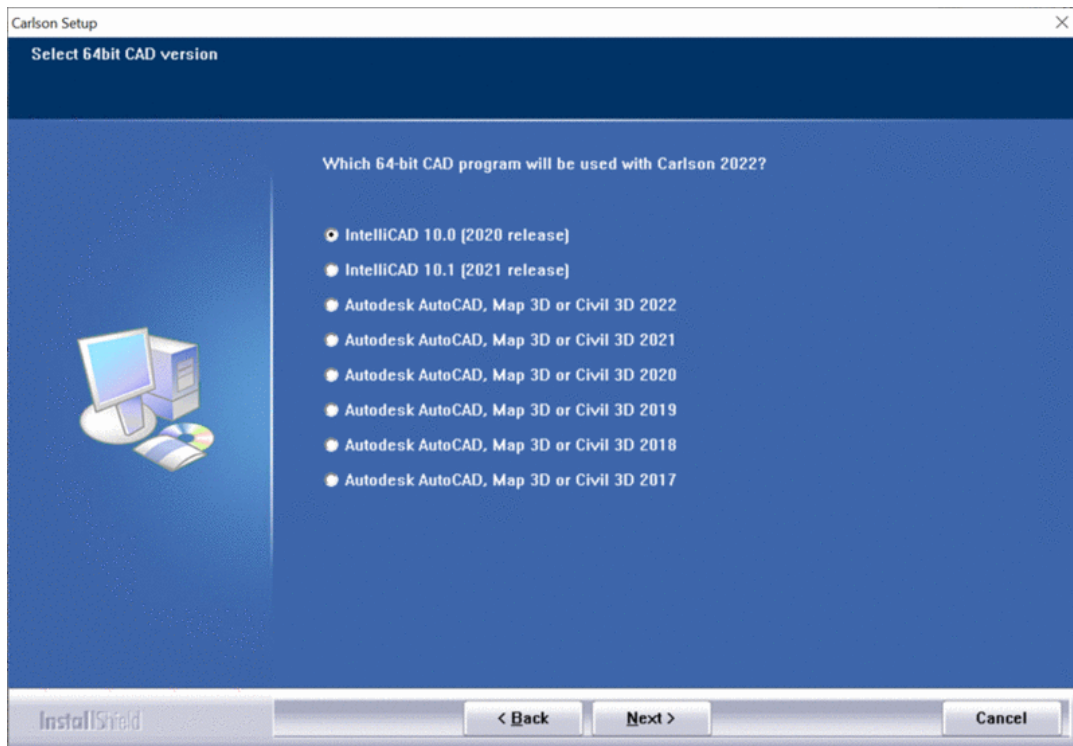
AutoCAD Map 2017-2024

Civil 3D 2017-2024

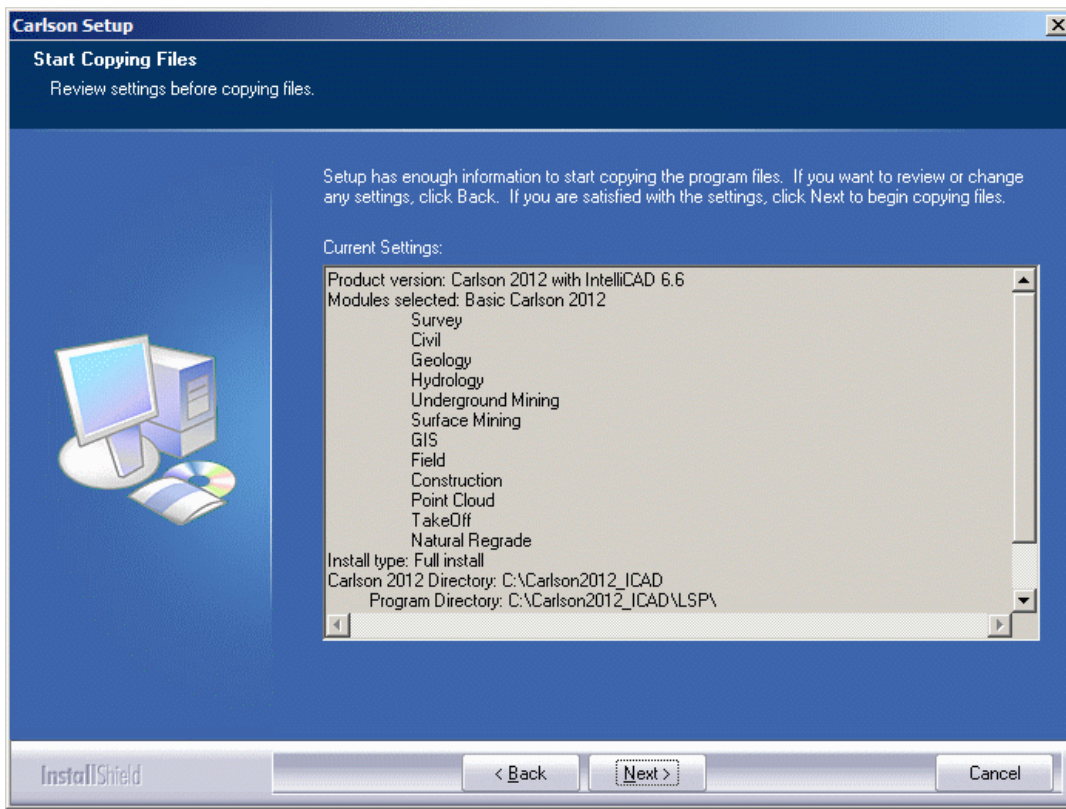
Note: Carlson requires a minimum screen resolution of 1024x768.

Workstation Installation

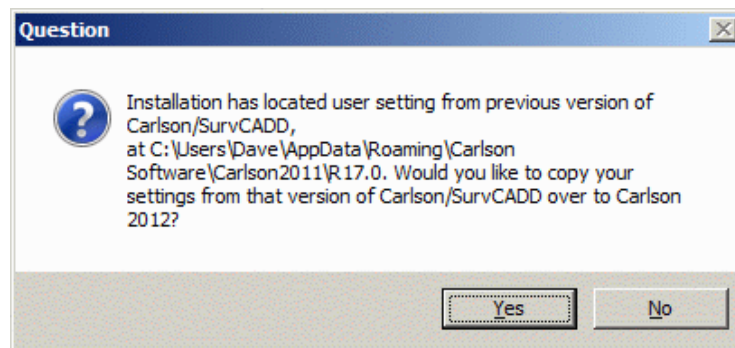
- Insert the CD and the program will initialize automatically. If the setup program does not automatically begin, click Start, select Run, type D:\Launch.exe and click OK.
- After the program begins, select Install Carlson 2024.



- Select which version of CAD you are installing Carlson 2024 on.
- During the installation process, you will be prompted for your serial number. This can be found either on the CD case or on your distribution sheet. **IMPORTANT:** Each legal copy of Carlson 2024 has its own serial number.
- Specify which folder you prefer to install Carlson 2024 in. In case of using multiple versions of AutoCAD, please select individual folders of Carlson 2024 for each version.
- Review the installation options, and click "Next" to continue:



- In the course of your installation, you may get a prompt to carry over your settings if you had SurvCADD installed on your system. Please state your preference:



- At this point, installation will start. After all the files have been copied, you have to select which copy of AutoCAD you will be using, as you may have multiple AutoCAD verticals installed on the same system.
- If you see installation take long time without any apparent activity, please make sure any other applications on computer are shutdown. This delay is caused by component registration waiting for other applications to exit.
- Installation will create a separate shortcut on your desktop for each major component of the program.

Performing Silent (Scripted) Installation

Whenever an administrator needs to install several identical computers, it may be beneficial to install once recording all the input and then deploy the rest of the computers using the input recorded when installing on the first one.

Carlson-specific Method

This method is based on creation of a plain-text configuration file and supplying this file as a command line parameter when launching the install.

If your configuration file is called setup.ini, the command line would look like this:

```
Carlson2024.64bit.exe ini=setup.ini
```

the path to the may be specified, but spaces in the path or file names are not currently supported

The contents of the setup.ini are as the following:

Entry	Description
EULAAccepted=1	Confirms that the EULA was reviewed and accepted, for example as a part of this manual. (Required)
Platform=Icad9_x64	Specifies target version of CAD. See below of acceptable values. (Required)
Serial=xxx-xxx-xxxx-xxxx	Serial number provided with purchase. (Required)
ImportProfile=Carlson2023	Name of the profile to be used to import settings (Optional)
DestDir=Path_To_Installation	Destination folder (Optional)
WorkDir=C:\Carlson Projects	Projects folder (Optional)
SettingsDir=C:\Carlson Projects\Settings	Settings Folder (Optional)
Language=en	Language code for the interface: en, win (Windows default), es, pt, cs, hr, pl, ru. (Optional)
Migration=Path_to_migration_file	Path to Carlson Settings Migrator generated zip file (Optional)

Acceptable values of the Platform parameter:

Platform Keyword	Description
Icad10_x64	IntelliCAD 10.x 64-bit
R24.3_X64	AutoCAD 2024 64-bit based products
R24.2_X64	AutoCAD 2023 64-bit based products
R24.1_X64	AutoCAD 2022 64-bit based products
R24.0_X64	AutoCAD 2021 64-bit based products

R23.1_X64	AutoCAD 2020 64-bit based products
R23.0_X64	AutoCAD 2019 64-bit based products
R22.0_X64	AutoCAD 2018 64-bit based products
R21.0_X64	AutoCAD 2017 64-bit based products

Native InstallShield Method

Carlson is using InstallShield InstallScript engine which supports this feature in the following manner:

- Start command prompt and change to the drive letter corresponding to CD-ROM drive where Carlson disc is located
- Run Carlson install with additional switches like this:

```
setup.exe /r /f1c:\some_folder\setup.iss /f2c:\some_folder\setup.log
```

where /r designates recording, /f1 response file and /f2 log file. Please note lack of space after /f switches. Apparently simple paths with no special (non-alpha) characters are required.

- Copy setup.iss to location accessible on the network
- On the target machine, run the setup as the following

```
setup.exe /s /f1network_folder\setup.iss /f2c:\some_folder\setup.log
```

where /s designates silent install, /f1 pre-recorded response file and /f2 log file

- The install runs very stealthy and does not generate any messages. The log file will contain return code of 0 on successful install.

Please note that this method will use identical Carlson serial number on all the silently installed machines. This is correct in case of network license mode, but is not proper for standalone licenses. In that case, prior to registration, please navigate to Help, About Carlson, Change registration, remove scripted number and enter proper unique serial number assigned to the workstation. After that, the new number is OK to register.

Network License Server Installation

For network licensing server installation instructions, click [here](#) or go to the Knowledge Base at www.carlsonsw.com.

Running for the first time

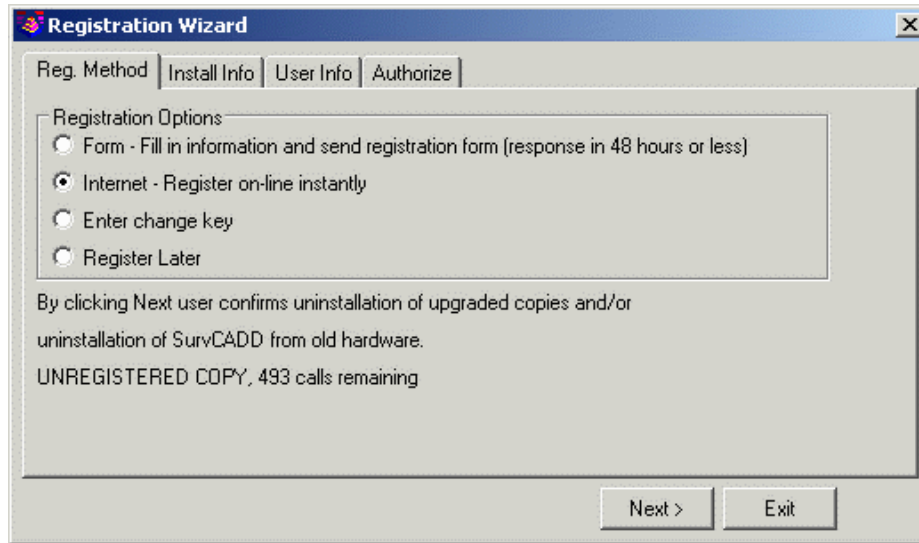
When you launch the program for the first time, it will try to connect to the updates server, to see if any program updates has been posted since the CD was created. Unless you indicate otherwise, the program will do this on a weekly basis, to make sure that your Carlson product is up to date on fixes to all known problems.

Registration

IMPORTANT ! ALL INSTALLATIONS OF THIS SOFTWARE REQUIRE A UNIQUE SERIAL NUMBER. If you are running this software on more computers than you have purchased licenses for, you must buy additional

copies.

Each Carlson program is licensed for use on **one** workstation which must be registered. The registration records your company name and AutoCAD serial number. To register your copy of Carlson, start Carlson and choose "Register Now". The following dialog will appear.



Note: Carlson Software will no longer issue change keys over the telephone. There are four registration options.

Fax: This method allows you to print out the required information on a form which you then fax to Carlson Software. The fax number is printed on the form. The change key will be faxed back to you within 72 hours.

Internet: Register automatically over the Internet. Your information is sent to a Carlson Software server, validated and returned in just a few seconds. If you are using a dial-up connection, please establish this connection before attempting to register.

Enter pre-authorized change key: If you originally chose the Fax method above, you will need to choose this method now to enter the change key that is faxed back to you.

Register Later: If you wish, you may defer registration up to 30 days. After this time, Carlson will enter demo mode which displays a message each time a Carlson command is run.

After you select the registration method, choose Next and select the type of installation you are performing. Choose Next again to review the copyright information and to fill out the required information. At this point, if you are using the Fax method, press the Print Fax Sheet button. If you are registering using the Internet method, press Next and the process will start.

If you have any problems with Internet registration, please repeat this process, and use the Fax method. There is more registration information on the Carlson Software website at <http://www.carlsonsw.com/registration.html>.

Tip: If Carlson is running, you may access the registration dialog by choosing *About Carlson* from the Help menu, then pick the Change Registration button. This also allows you to add additional serial numbers you purchased without reinstalling software. Each serial number requires separate registration.

via Discussion Groups

- Carlson Software operates user discussion groups located at <http://update3.carlsonsw.com/forum/>. You can participate in user-to-user discussion on tips, tricks and problems. Our staff monitors these groups to ensure that all the issues are addressed. Visit our website at <http://www.carlsonsw.com> for information on how to access these groups.

- You may also access the Carlson Software Knowledge Base. Visit it directly at <http://www.carlsonsw.com/support/knowledge-base/>.

via Electronic Mail

The Technical Support e-mail address is support@carlsonsw.com.

via Phone

Phone: (606) 564-5028

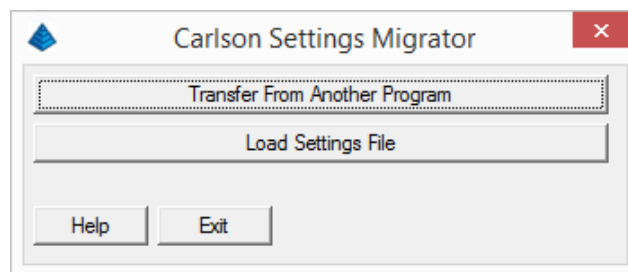
via Web Site

Check the Carlson Software Web Site at <http://www.carlsonsw.com> for:

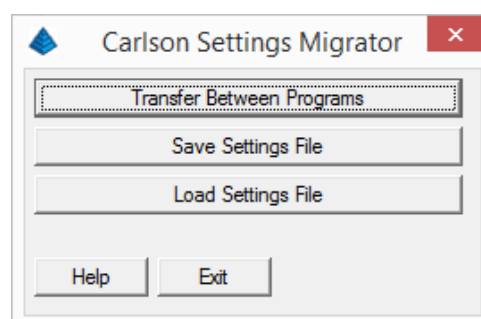
- Knowledge Base, discussion groups, technical support documents and newsletters
- Carlson Software manuals (PDF) and training movies
- Training and seminar schedules
- Step by step procedures on popular called-in topics
- Carlson Software and Autodesk downloads and updates (Feel free to register for automatic update notification of updates when you come to that area.)

Carlson Migration

When user runs the program for the first time, the Carlson Settings Migrator window will be displayed.

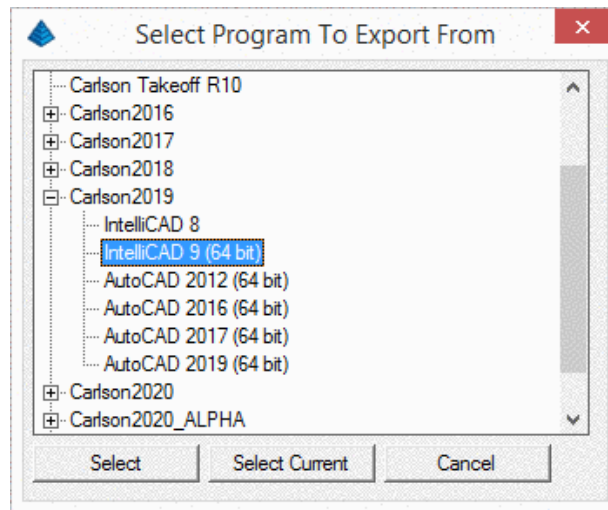


The user has an option to select another instance of Carlson product installed on the same computer to import the settings from or select a packaged import file saved on a different computer. The Settings Migrator can be copied to another computer to export the settings from there. When used internally, this interface is slightly different:

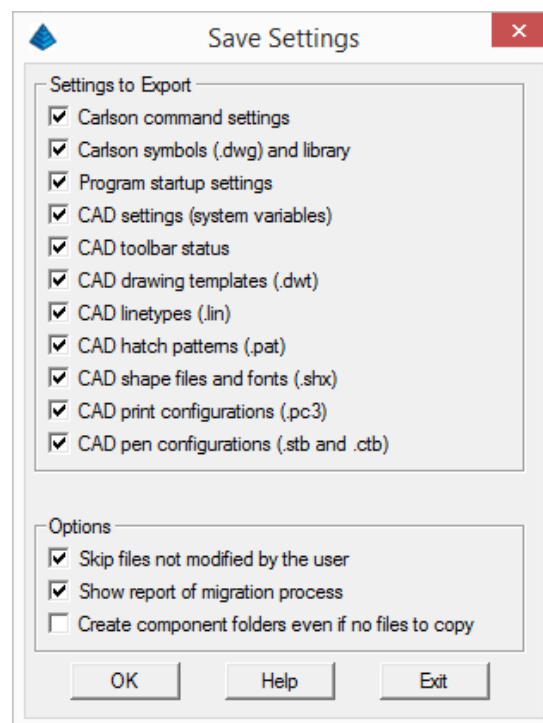


This can be used to move, export or import settings on another machine or even version of Carlson.

Select an instance to export from and click *Select*:



When an instance of the program is selected to export from, the following settings dialog is displayed:



Settings selections are fairly straightforward and most users probably will want to leave them on by default, but the options are as the following:

Skip files not modified by the user - only selected files not originally included with the program will be included. This cuts down on the migration data set size. Additionally, for some file types like fonts and hatch patterns, a care should be exercised when copying them around as they are could be protected by vendor copyrights.

Show report of migration process - displays report outlining settings and files processed

Create component folders even if no files to copy - migration data set is Zip file with sub-folders for individual file types. Most of the time, when "*Skip files not modified by the user*" toggle is used, there are no files for the particular

component to copy. This setting creates component folder anyways. This will allow an administrator to edit Zip file and included additional files in the corresponding folder. This way, by exporting from old version and adding more files, a custom initial data set can be created for the new user to hit ground running.

Additionally, the data migration file can be specified during silent installation for automatic deployment, as described in the Installation Guide.

Starting AutoCAD with Carlson

To start Carlson, use the icon which was created on your desktop and in Carlson submenu under Start > Programs (or All Programs). When you do so, CAD engine associated with Carlson is started with a customized profile, which contains all of the setting changes necessary to run Carlson. This profile prevents harmful conflicts between Carlson and other CAD based products. The changes in menus, toolbars and preferences will be saved in that profile for further use. The CAD engine associated with Carlson is selected during the installation of Carlson.

Custom Template Profile Name

The company with the internal CAD profile setup can specify the desired profile name to be used as a template for setting up Carlson 2024. The existing template will be copied and new profile for Carlson 2024 will be created, changing minimal number of settings required to run Carlson. As a result, most of the profile will match company setup and no further customization will be necessary.

Additionally, there is a way to specify the template as a property of the shortcut starting the program. Under the Target setting in the Windows Properties for the desktop shortcut, add `TEMPLATE=tempatename.dwt` to the list arguments.

Specifying which AutoCAD product is used to start the program

As already mentioned, in case when there are several AutoCAD vertical products installed for the same version of AutoCAD, user can configure which AutoCAD product is used with Carlson. However, in some cases, like having to switch between AutoCAD products based on availability of the network licenses, it may be more convenient to control this with the use of different shortcuts rather than with the program setting. Under the Target setting in the Windows Properties for the desktop shortcut, add `ENGINE=engine_name` to the list arguments. For example for AutoCAD 2019, engine name is `ACAD-2001:409` and for Civil 3D 2019 is `ACAD-2000:409`. These values change often with new releases of AutoCAD and it is best to look them up in the `scjstart.ini` located in the same folder as `scjstart.exe` pointed by the shortcut. This file will contain the value for AutoCAD product currently configured to start with Carlson.

Startup Module

The Carlson module that is loaded when the program is started is set by the desktop icon. The Carlson install program creates different desktop icons depending on how many modules are licensed in your serial number. Under the Target setting in the Windows Properties for the desktop icons, the `SCSTATE` setting controls the startup module as follows:

`SCSTATE=1` Survey

`SCSTATE=2` Civil

`SCSTATE=3` Geology

`SCSTATE=4` Hydrology

`SCSTATE=5` Underground Mining

`SCSTATE=6` Surface Mining

`SCSTATE=7` GIS

`SCSTATE=8` Field

SCSTATE=9 Construction
SCSTATE=10 Point Cloud
SCSTATE=11 GeoTech
SCSTATE=13 Trench
SCSTATE=14 Natural Regrade
SCSTATE=15 CADNet
SCSTATE=17 Civil Suite (any module Survey, Civil, GIS, Hydrology depending on last loaded)
SCSTATE=18 Mining (any module Geology, Surface or Underground depending on last loaded)

License Models

Carlson Software supports a multitude of the license models to better fit the specific needs of the customer. There are three retail license modes: regular standalone license, network license and hardware lock license. There are four additional special license modes: a demo license, not for resale license, educational standalone and educational network license. These license modes as well as their features, benefits and restrictions are described with further details below.

Standalone license model

This is the most typical license model. The user is issued a serial number containing a license to run one or more modules of Carlson application. This authorizes user to install and register software on one machine and one machine only. The exception is made for use of home or laptop computer of the same person. Both computers have to be registered online and home/laptop system is specifically designated during the registration.

Multiple serial numbers containing separate features can be combined within the same installation by entering them in Change Registration dialog, found under Help, About Carlson.

When the computer is no longer available and software needs to be moved onto the new system, the move can be performed by using hardware upgrade option during registration, which marks old system as decommissioned and allows registration on new system.

The upgrade serial numbers are tied to the original serial numbers from the older version and continued use of older version is only allowed if upgrade serial number is used on the same system as original serial number. Otherwise the use of the original serial number is terminated within 60 days of the upgrade registration on different system.

Network license model

This is the license models used by the larger companies needing flexibility of the software licensing when license is not tied to a particular machine. The company is purchasing a pool of licenses consisting of the appropriate count of individual modules to better fit the needs of its users. The pool can be expanded by adding more licenses at later date.

The licenses are loaded into the license server and software is installed locally on the workstations with a network serial number instructing software to look for the license on the network server. For details on the installation process, please refer to the Installation Guide found on CD. Procedure for requesting and installing the license on the server is outlined in the Knowledge base: http://update.carlsonsw.com/kbase_main.php?action=display_topic&topic_id=55

Even if purchased as "seat" or as a part of the Suite, the modules are treated by the license server as individual modules, allowing separate users to have simultaneous use of separate modules from the same "seat". Each

user is going to request and hold the license for the module he is currently uses. That license is held as long as the program is running. When user switches the modules, the held license is released and new module license is acquired.

The network license mode does not allow for the home/laptop license the standalone license mode has, but it allows users to use WAN or VPN software to get licenses from the remote servers over the network and it allows to commute the license from the server and install the license temporarily on the workstation. Commuting is discussed in further detail in the Administrator's guide included with the license server. The commuted license automatically returns to the server when it expires or it can be returned by user earlier when computer returns to the network. Licenses commuted from the server are not available to other users contacting the server for the license, so commuting essentially decreases the server license pool for the commute period.

Software configured to use any serial numbers with network license mode enabled can not use any serial numbers with other license models enabled - it will treat them all as network license serial numbers. Therefore it is not possible to share a module on network but have other modules in standalone mode.

Hardware lock licensing

In this model, the physical hardware key is issued. This key is an USB device which has to be plugged in to the computer in order for the computer to be able to use software. This allows a single user to setup software on multiple computers or even locations and carry his license with him in form of USB device. This is also a primary model of licensing in the countries where software-only protection is not considered sufficient protection due to rampant software piracy.

User is responsible for the cost of the device in addition to the cost of the software itself if this licensing model is employed.

Demo license model

Demo serial numbers allow full use of software within the demo period. The license is limited both by the duration and built-in use counter. Once the demo period expires, the user can get extensions from Carlson customer service in two week increments if there is a demonstrated need to continue with decision process.

Educational institutional license

Accredited institution can receive the network license for the institutional license server for the purpose of teaching course on Carlson software at a nominal cost of software maintenance instead of list price. The university will be responsible for student support and for commuting the licenses for student use in the course. For all practical purposes this is a regular network license, except for the different pricing structure.

Educational student license model

Students currently enrolled at college can purchase a significantly discounted standalone license which will run for 1 year from the date of purchase. A proof of enrollment is required and software running in this mode will be checking online weekly for license extension until 1 year runs out. After this, the additional purchase is required if enrollment remains current. Second and home computers are not allowed with this license model.

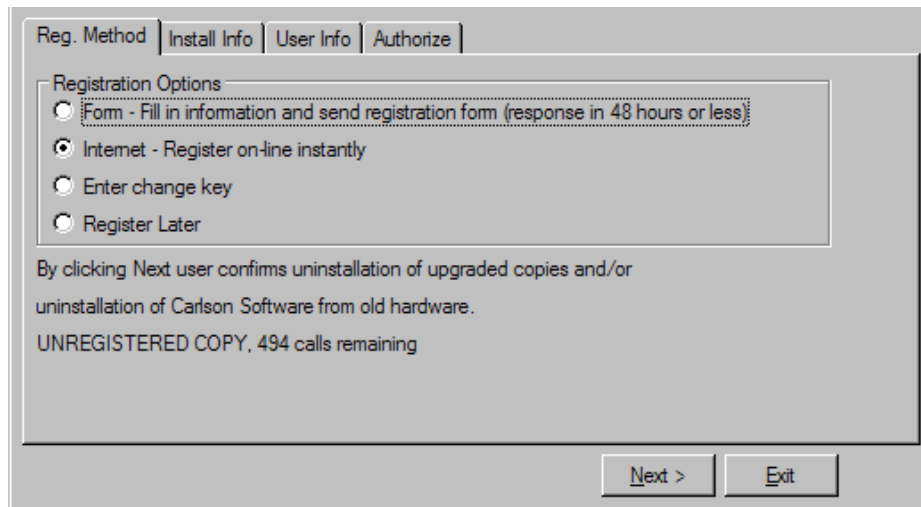
Not for resale license model

This is essentially a standalone license model governed by additional restrictions of license use. Serial numbers designated as "not for resale" are issued only to Carlson personnel and Carlson resellers. These licenses should never be installed at end-user systems or sold to the end-user. Such use constitutes a violation of Carlson agreements.

Carlson Registration

Each Carlson program is licensed for use on **one** workstation which must be registered. The registration records your company name, Carlson serial number and AutoCAD serial number. To register your copy of Carlson, start Carlson and choose "Register Now". The following dialog will appear.

Note: Carlson Software will no longer issue change keys over the telephone. There are four registration options.



Fax: This method allows you to print out the required information on a form which you then fax to Carlson Software. The fax number is printed on the form. The change key will be faxed back to you within 72 hours.

Internet: Register automatically over the Internet. Your information is sent to a Carlson Software server, validated and returned in just a few seconds. If you are using a dial-up connection, please establish this connection before attempting to register.

Enter pre-authorized change key: If you originally chose the Fax method above, you will need to choose this method now to enter the change key that is faxed back to you.

Register Later: If you wish, you may defer registration up to 30 days. After this time, Carlson will enter demo mode which displays a message each time a Carlson command is run.

After you select the registration method, choose Next and select the type of installation you are performing, choose Next again to review the copyright information and to fill out the required information. At this point, if you are using the Fax method, press the Print Fax Sheet button. If you are registering using the Internet method, press Next and the process will start.

If you have any problems with Internet registration, please repeat this process and use the Fax method. The registration form is available on the Carlson Software website at <http://www.carlsonsw.com/registration.html>.

Tip: If Carlson is running, you may access the registration dialog by choosing *About Carlson* from the Help menu, then pick the Change Registration button.

Network License Setup

The up to date version of this process is always available online at http://www.carlsonsw.com/support/knowledge-base/?action=display_topic&topic_id=55

Quick Summary of Network License Manager Installation Procedure

Install the Carlson Software product which you intend to network onto each of the workstations using the network serial number provided to you. Once you have completed that task, please do the following on the network server:

1. Download the Sentinel Server.exe from the link below and install it to the server: Sentinel Server ver. 8.2.3
2. Download the Carlson LMTools.zip from the link below to the server and extract the files in a location on your computer which is both easy to find and easy to remember. You will need these files in the future for license management purposes: Carlson LM Tools 8.2.3.zip
3. If you are existing user of Carlson and you are upgrading to a later version of Carlson Software (2010 and beyond) **You do not need to request new license files...** the network license files for older versions will work with your upgraded serial numbers and software.
4. Open the LMTools folder and run the **regclient.exe** to request license files for your new license manager (see more details below). You will receive a reply (to the e-mail address specified in your request) containing your license file.
5. Save the .txt license file (from your reply e-mail) into the LMTools folder.
6. Run the **WLMAAdmin.exe** file, also found in the LMTools folder, to point the license manager to the new license file (see more details below).
 - On the left-hand side of WLMAAdmin you will see Subnet Servers with a plus (+) beside it. Please click on the plus beside Subnet Servers to expand the list of servers.
 - Right-click on the subnet server which you intend to use for the license manager.
 - Select **ADD A FEATURE > FROM A FILE > TO A SERVER AND IT'S FILE.**
 - A Windows browser window will open within the LMTools folder to allow you to select the license file that you received in your e-mail.

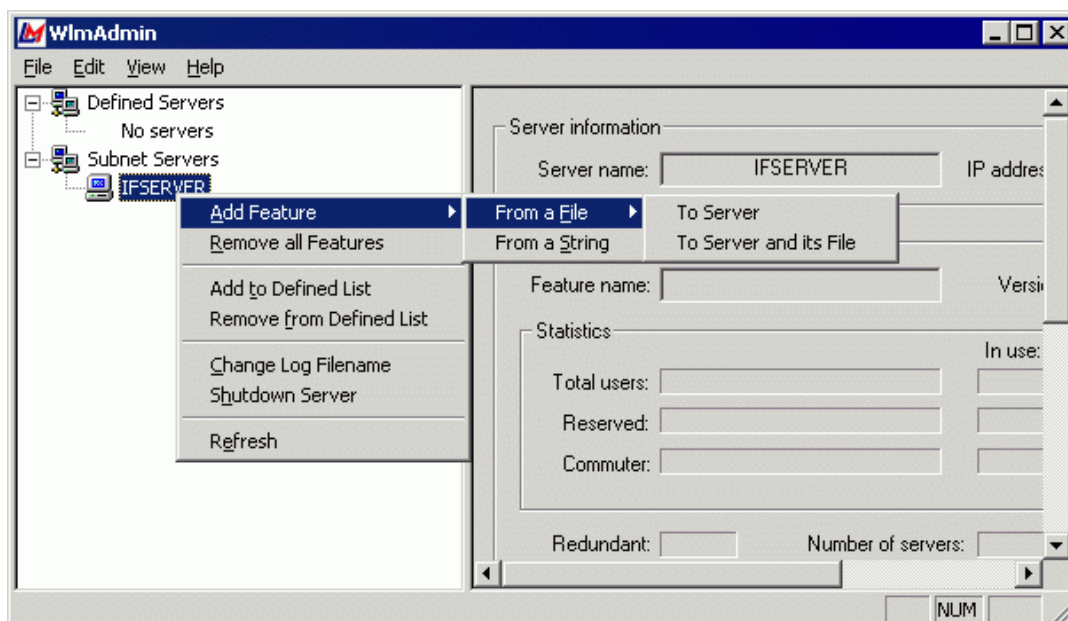
In order to **REQUEST A LICENSE FILE**, please follow these steps:

1. Download LMTools8_2_3.zip and extract it into a folder of your choice.
2. If you are planning on setting up redundant license server pool, please refer to Knowledge base Article 620.
3. One of the files found in the LmTools.zip is **regclient.exe**. Execute this program on the **SERVER** computer.

- Please fill out the entire form and provide real contact information as we will use it to contact you and provide you with your license.
- Make sure to use the **Carlson network serial number** provided to you for the product being licensed.
- Make sure to select an appropriate reason for the request.
- In the note field include sufficient information about what has lead to the problem. Provide any additional information you may have in this field (i.e. the number of seats (nodes) requested for each module).
- Click on **Submit** to transmit the request directly to the Carlson server when you are finished. Make sure that any local firewall (if present) allows the connection through. In case of transmission failure, you will be given alternative submission instructions.

Installing The License File

When you receive your license file, perform the following:



1. Run the **WlmAdmin.exe** found in the LMTools folder.
2. Click "+" before Subnet Servers, to expand the list.
3. Highlight your server name on the list.

NOTE: If the server is not on the list please verify the following:

- Sentinel Server is installed
 - SentinelLM service is running (Control Panel > Administrative Tools > Services)
 - The computer running the WlmAdmin.exe is on same computer or at least on same subnet as license server
 - The license server does not have a local software firewall or modify the firewall rules to let licensing traffic through (see Knowledgebase Article 269 for more info)
4. Right Click on the Server name and select **Add Feature > From a File > To Server AND It's File**
 5. Browse to select the license file which has been emailed to you.

NOTE: If the license has been rejected:

- Make sure original license request (regclient.exe) was originated from the SERVER computer.
 - This license was not yet added (multiple additions of the same file generate errors).
 - The Server name and IP address showing for the server (within WlmAdmin.exe) are correct
6. You can now verify licenses available by clicking "+" in front of the server name

Install The Software Being Licensed

Now, install the software being licensed on the workstations by using the serial number provided to you with the installation package. All workstations on the network will share the same serial number.

NOTE: Automatic server discovery only works if the workstations and server are located within the same subnet. In other instances an environment variable needs to be set on the workstation through the following procedure:

1. Go to Control Panel > System > Advanced Environment Variables
2. Under System Variables, click on "New"
3. Set the "Variable name" to **LSFORCEHOST**, and the "Variable value" to the name or IP address of the server (SafeNet's Support recommends using the license server IP address)
4. Click OK all the way out.

NOTE: If you intend to use commuter licenses, please refer to Knowledgebase Article 507 for additional steps to be performed on the workstation by administrator.

Verify that the licensing works on the client machine. Please note that license is not when the program starts, but after the first or second consecutive command.

Troubleshooting

For troubleshooting, please refer to Knowledgebase Article 211. The Complete Administrator's Guide has been included in the Carlson LmTools8_2_3.Zip file.

Network License Monitoring

To view the current available count of licenses on the server, you can run the WLMAdmin.exe from within the Carlson LMTools folder. These tools were used to install the license and are available for download.

The WLMAdmin.exe will not automatically refresh to show which licenses are in use. You will need to click on the View drop-down and select "Refresh allÂ".

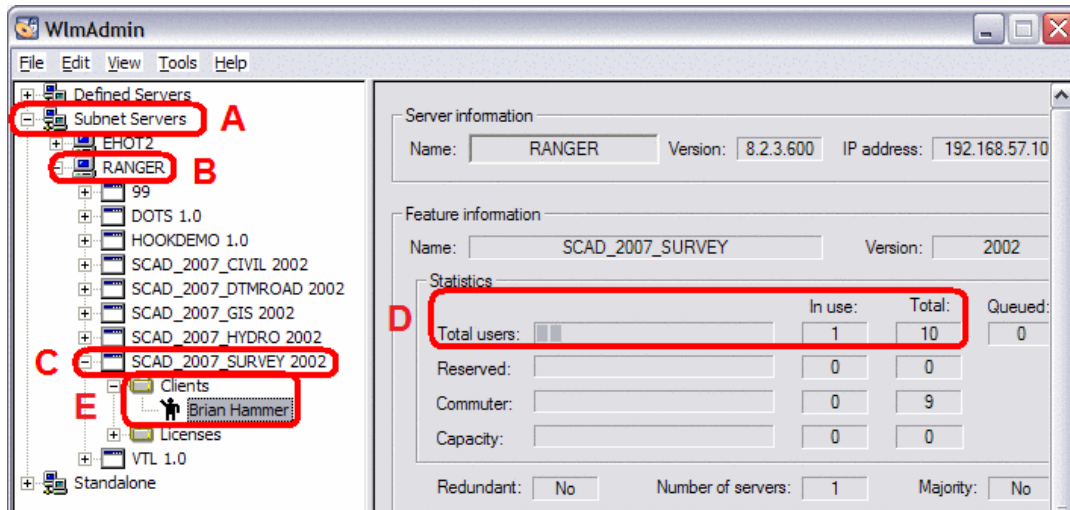
A Click on the + beside Subnet Servers to show all servers on the subnet.

B Click the + beside the Server Name to show the modules licensed to this server.

C Click on the module of your choice to highlight it.

D The total number of licenses, as well as those in use, can be found under Feature information.

E To see which user is has a particular license; click on the + beside the module name. Then, click on the + beside the Clients folder.

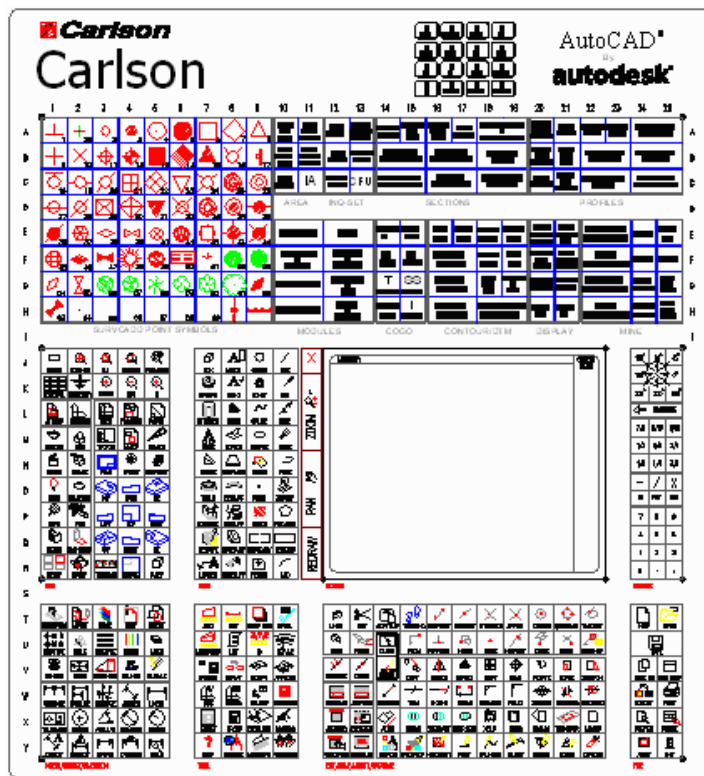


Carlson also has a set of integrated network manager functions under Help->About Carlson->Registration. Click **Network Licenses** to access the dialog which has the features which can list the features available on the server, count of the licenses and also allow the user to commit or return the license.

Tablet Template

If you have a digitizer you will probably want to plot a copy of the Carlson tablet menu. After installing Carlson, start Carlson and plot a copy of the drawing file TABLET.DWG located in the Carlson support directory (ie: \Carlson\SUP\TABLET.DWG), at a scale of 1=1. You may want to review the drawing in the drawing editor to determine layer names and colors to set up a color version of the template. You can also modify this drawing to add your own symbols and details to the tablet menu. After plotting, secure the Carlson template drawing to your digitizer. The proper scale is 1=1 for a 12" x 12" tablet. It can also be plotted at a smaller or larger scale to suit other tablet sizes.

Once you have a hard copy of the tablet, the tablet needs to be configured. Type TABLET at the command prompt. Refer to the CAD Reference Manual for details on the menu area points and number of columns and rows. The tablet template is in the standard template format which means if you are using any other templates you can easily switch between different control menus.



Troubleshooting and Customizing Setup

To install Carlson 2024 on Windows, you must have Administrator permissions to write to the system registry during the install.

Successful install of Carlson consists of four key items:

1. Windows registry settings
2. Carlson icon on desktop and in the Start menu
3. Carlson 2024 profile created in AutoCAD or IntelliCAD
4. carlson.ini configuration file in USER sub-directory of Carlson

The Carlson profile in AutoCAD/IntelliCAD is created and configured by the helper application called SCJSTART. The application is typically located in C:\ProgramData\Carlson Software\Carlson2024\ followed by subfolder specific to CAD version. Please note that ProgramData is folder hidden by default Windows Explorer settings. In this location, there is a file called scjstart.ini which is generated during install and which controls SCJSTART.EXE behavior.

If you have trouble starting the application, review the scjstart.ini contents to see if anything on the system has changed since the installation and needs to be corrected.

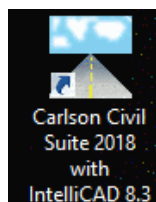
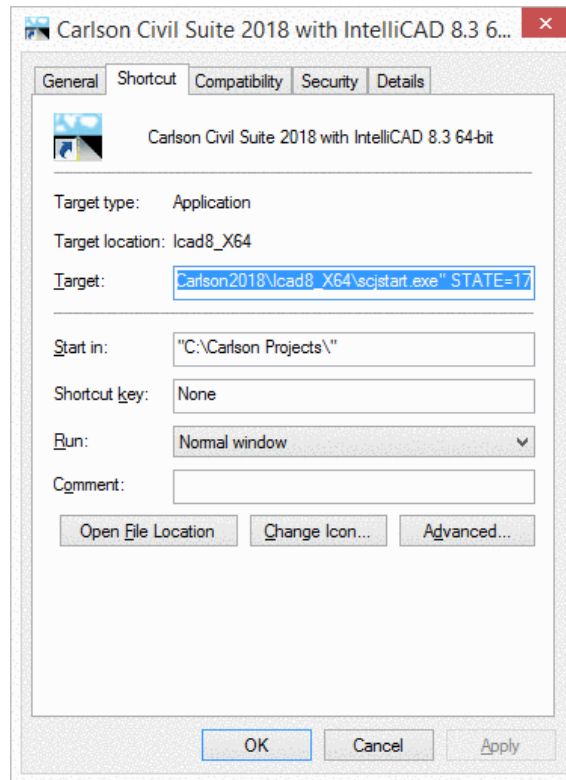
Desktop and Menu icons for Carlson 2024

A Carlson specific profile is used in order to configure Carlson correctly and start the Carlson menu when AutoCAD or IntelliCAD starts. Profiles are used by the CAD to separate different product environments. If profile is missing, the profile from old version of Carlson will be used as template to copy to new version or if that is not present, the first of the existing profiles is used instead. Once profile is copied, it is adjusted to start new version of Carlson.

Carlson shortcuts point to the scjstart.exe and command line parameter called STATE controls which module activated when the product is launched.

To verify that your Carlson icon is set up correctly, do the following:

- Right click on the Carlson icon and select Properties. Click on the Shortcut tab.
- Target field should contain the following executable **SCJSTART.EXE** (located in the Carlson\USER directory).
- Work directory defaults to the same directory where drawings are stored
- Make changes as needed and click OK.



User can control which template is used to create new drawings in the program by specifying additional command line switch to the Target field of the shortcut. Add **TEMPLATE=template_name.dwt** to the end of the input. You can create a copy of shortcut with different template name to be able to switch between templates. The template being specified should already be known to the program by being within one of the searched folders.

Checking Carlson.INI configuration file.

The last step in troubleshooting the installation is to verify the Carlson.INI file in the Carlson USER directory. Open that file using Notepad (Click Start, Programs, Accessories, Notepad) or any other editor. Then verify and/or modify the top few lines to match your setup. Save it and restart Carlson.

If the procedures outlined above did not help or you have trouble following them, please feel free to contact Carlson Software Tech Support.

Loading Carlson Menus

The Carlson programs are loaded when the Carlson menu is loaded. The Carlson menu is named cs###base.mnu (for AutoCAD 2005 and earlier) or cs###base.cui (for AutoCAD 2006 to AutoCAD 2009) or cs###base.cuix (for AutoCAD 2010 and later) where the ## is the version of AutoCAD or icadbase.mnu for IntelliCAD. For example, in AutoCAD 2014 the Carlson menu name is cs14base.cuix. The Carlson menu is located in the Carlson SUP folder. The Carlson installation and desktop icons should automatically setup AutoCAD to load the Carlson menu. To manually load the Carlson menu, first make sure that the Carlson SUP folder is listed in the Support File Search Path under AutoCAD or IntelliCAD Options. Then run the menuload or cuiload AutoCAD command or the menu IntelliCAD command and load the Carlson menu. The Carlson SUP folder is under the Windows user folder such as:

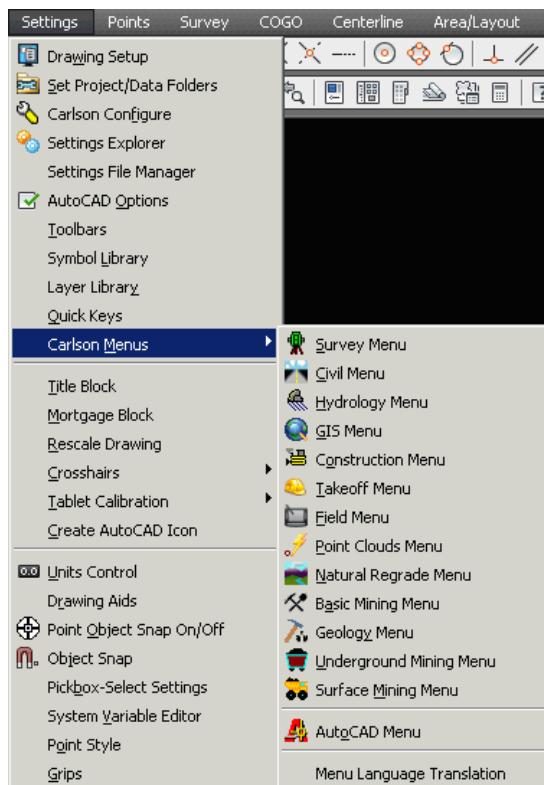
```
%appdata%\Carlson Software\Carlson_Version\CAD_Version\SUP
```

The Carlson menu has over 80 pull-down menus for the different Carlson programs. Each program has its own set of pull-down menus. The menu set for one program is active at a time. To switch current program menu, choose the menu from Settings>Carlson Menu or choose from the Modules toolbar.

By default, the Carlson menu is setup as the Main Customization File in AutoCAD. To setup the Carlson menu as the partial menu,

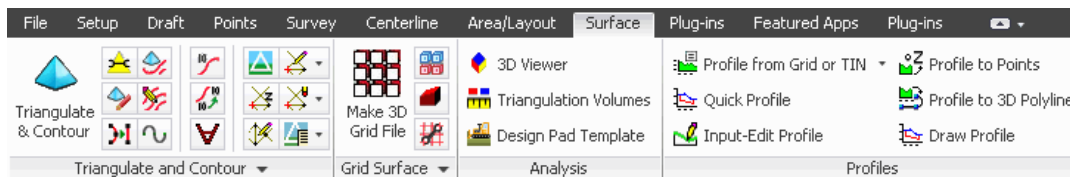
1. Start AutoCAD with Carlson by using the Carlson desktop icon
2. Run CUILOAD and unload all the menus. Then load your Main Customization File. Then load the Carlson menu.
3. In Carlson Configure->Startup Settings, turn on No Menu Reset.





The Carlson toolbars are defined in the Carlson menu. To turn on more toolbars, right-click in the toolbar space such as between the pull-down menus and the drawing window. This right-click menu has a list of all toolbars and you can pick on the toolbar name to activate. In AutoCAD, another way to activate toolbars is type CUI at the command prompt which brings up the Customize User Interface dialog where you can click on toolbars. In IntelliCAD, another way to turn on toolbars is type TOOLBAR at the command line which brings up a list of toolbars that you can toggle on.

For AutoCAD 2012 and higher, Carlson ribbons are available. Ribbons are like advanced toolbars. These ribbons are defined in the Carlson menu and can be accessed by setting the AutoCAD Workspace to one of the Carlson default workspaces such as Survey. Use the AutoCAD workspace command or toolbar to set this workspace. You can also use the CUI command to activate the Carlson ribbons and workspaces.



Obtaining Technical Support

via Discussion Groups

- Carlson Software operates user discussion groups located at [news://news.carlsonsw.com](http://news.carlsonsw.com). You can participate in user-to-user discussions on tips, tricks and problems. Our staff monitors these groups to ensure that all the issues are addressed. Visit our website at <http://www.carlsonsw.com> for information on how to access these groups.
- You may also access the Carlson Software Knowledge Base. Visit it directly at http://update.carlsonsw.com/kbase_main.php.

via Electronic Mail

- The Technical Support e-mail address is support@carlsonsw.com.

via Phone/Fax

- Phone: (606) 564-5028
- Fax: (606) 564-6422

via Web Site

Check the Carlson Software web site at <http://www.carlsonsw.com> for:

- Knowledge Base, discussion groups, technical support documents and newsletters
- Carlson Software manuals (PDF) and training movies
- Training and seminar schedules
- Step by step procedures on popular called-in topics
- Carlson Software and Autodesk downloads and updates (Feel free to register for automatic update notification of updates when you come to that area.)

via Training

- Basic, advanced and update training is available from Carlson College. Enroll on our webpage or call 606-564-5028 and ask for Carlson College.

Command Entry

Commands may be issued by selecting a pulldown menu, screen menu, digitizer tablet item, or by typing a command at the command prompt. Pulldown menus have a row of header names across the top of the screen. Selecting one of these header names displays the possible commands under that name. Screen menu items are shown in the screen menu (typically on the right side of the screen). The screen menu can be toggled off and on inside of the AutoCAD Options dialog. The Pulldown menus are the primary method for Carlson command selection. Each section of this manual shows the pulldown menu which contains the commands that are explained in that section. Pulldown menus are sometimes also referred to as dropdown menus.

Command availability depends on which menu is loaded. Carlson menus have a mixture of both Carlson and CAD commands. This allows you to execute the commonly used CAD commands from the menus while running Carlson.

Quick Keys are user-defined short cut names that can be typed in to start commands. To review the current set of Quick Keys, run the Quick Keys command in the Settings pulldown menu. Quick Keys are explained in more detail in the next section.

For command entry at the Command: prompt, pressing Enter repeats the last command. Also the prompt history records the sequence of previous commands, and you can run these previous commands without invoking the menu. To access the commands, use the keyboard up and down arrows. The up arrow moves backwards in the history and the down arrow moves forward. As you press the arrows, the previous command names appear at the command prompt. When you get to the command that you want to run again, press Enter.

Transparent Commands

Transparent commands can be issued at the Command: prompt while running other commands without exiting the current command. The transparent command will run and then return to the current command. To run a transparent command, you can either type in the command name or pick a button on the Transparent Command toolbar. There are nine Carlson transparent commands for point input and one for entity selection. The point commands can be used at any command prompt that is for a point location. The selection command can be used at any "Select objects" or "Select entities" prompt.



'2: Gets the 2D (x,y) coordinate by point number from the current coordinate file. You can enter a single point number or a range of points (ie 1-10).

'3: Gets the 3D (x,y,z) coordinate by point number from the current coordinate file.

'CL: Gets a point by station-offset from a centerline. The routine prompts for a centerline, station and offset left or right. The centerline can be defined by a centerline file, segment on a polyline or linework entity. The position along the centerline can be specified by entering the station, picking a point that gets projected onto the centerline, or by a distance factor. While picking the position along the centerline, the program shows the station and offset of the cursor in real-time. The 'CL command can be used as a Perpendicular From snap by picking a centerline and entering a station to come perpendicular from (the routine defaults to the station of the last point) and then entering an offset for the perpendicular distance.

'PIPT: Gets a point that is the PI position for an arc. The routine prompts to select an arc.

'CENTROID: Gets a point that is the centroid position for a closed polyline. The routine prompts for the polyline to process.

'TRAVPT: Gets a point by angle and distance from a starting point.

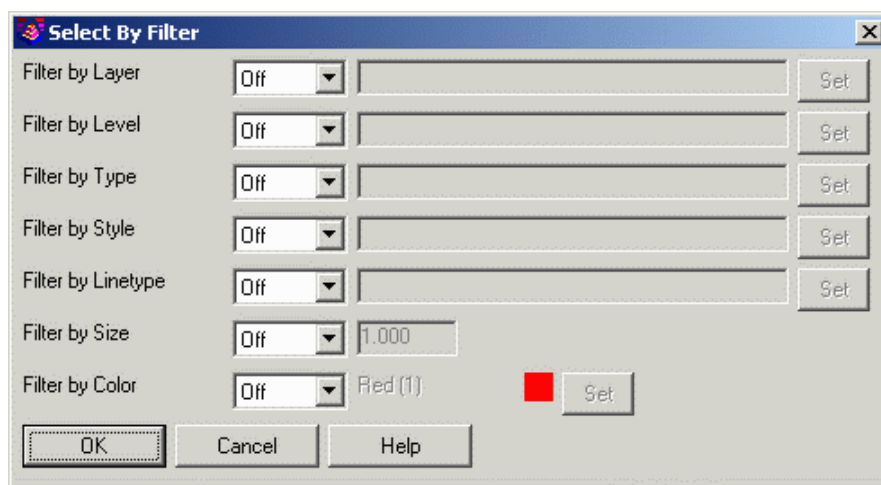
'TANGPT: Gets a point that is tangent to the selected centerline or polyline. The routine prompts for the centerline, station and distance. The station defaults to the position along the centerline of the last picked point.

'BBPT: Gets a point by Bearing-Bearing Intersection. The routine follows the same prompts as the Bearing-Bearing Intersect command in the COGO menu.

'BDPT: Gets a point by Bearing-Distance Intersection. The routine follows the same prompts as the Bearing-Distance Intersect command in the COGO menu.

'DDPT: Gets a point by Distance-Distance Intersection. The routine follows the same prompts as the Distance-Distance Intersect command in the COGO menu.

'FILTER: Makes a selection set of entities using an entity property filter. Available properties to filter include layer, level, type, style, linetype, size and color. For each property, you can choose whether to include or exclude from the selection using this property. When using multiple properties for the filter, they are combined using an AND operator so that all properties must match to pass the filter.



Prompts

For '2 to draw a polyline from points 100-102.

Command: *pl*

PLINE

Specify start point: '2

>>**Enter point numbers:** 100-102

Resuming PLINE command.

Specify start point: 10308.02109999,10213.95245576

Current line-width is 0.00

Specify next point or [Arc/Halfwidth/Length/Undo/Width]:

10307.46576250,10214.50318930

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:

10268.05722717,10245.06058831

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: *press Enter to end*

For 'CL to draw a polyline starting at station 200, right 50 from a centerline:

Command: *pl*

PLINE

Specify start point: 'cl

>>**CL File/Segment/<Select centerline polyline>:** *pick a polyline*

>>**Station range:** 0.00 to 256.36

>>**Factor/<Pick point on centerline or Enter Station <191.162>>>:** 200

>>**Centerline starting station <0.0>:** *press Enter to use zero*

>>**Offset (negative for left):** 50

Resuming PLINE command.

Specify start point: 5155.46856081,5271.30066060

Current line-width is 0.00

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: *pick a point*

What is New

General

- AutoCAD 2024 - Added support for this new version.
- IntelliCAD 11.1 - Added support for this new version.
- IntelliCAD 10.1 - Improved speed and new features like sheet sets with the Publish command.
- Toolbars - Added over 30 new toolbar icons and a new ribbon for GeoTech.
- Open Drawings Map - Added method for custom properties for drawing records and method to store a drawing database on a shared network location. From the Map view, added function to open Windows File Explorer for the drawing's project folder.
- Settings Server - Added method to copy data files into project folders.
- Project Data Sub-Folders - Added separate Program Settings folder for settings files.
- Data Depot - Added details to the history log and the ability to restore a previous version of a data file. Adding files to Data Depot supports working from project level with controls on which files to add.
- Data Depot - Ability to log in or out of the server from the Project Setup dialog box.
- Data Depot - Ability to specify destination folder location when getting a project from the Data Depot.
- Carlson Configure - Added setting to control font used for dialogs.
- Select CAD Default Settings - New command to choose classic or modern CAD user interface settings.

- Drawing Cleanup - Added functions to remove duplicate text and to remove unused registered apps.
- Import Polyline File - Added method for locating by station/offset values from the file.
- Drawing Inspector - For polylines, added reporting of current position distance. For centerline polylines, added reporting of spiral data.
- Draw MText - In many commands that create MText, added an option to set the Text Frame to draw a box around the text.
- Translate Layers - Added method to import layer names from the drawing.
- Line Up Text - Added method to pick two points to define angle, option to respace and option to use text for alignment reference.
- Arrowhead with Line - New command to draw a two point leader with an arrowhead.
- Boundary Polyline - Added option to create 3D polyline.
- Shrink-Wrap Entities - Added options to ignore zero elevation entities for input, to elevate zero elevation vertices, and to erase the input entities.
- Add Intersection Points - Added option to average elevations at intersection within a tolerance.
- Draw Label For Companion Linework - New command for Draw To Standard to create a label using properties by layer.
- Draw Standard Item by Quick Key - New command to create entities for Draw To Standard by typing a quick key instead of dialog list selection.
- Reports - For PDF reports, added option to make PDF report files read-only. For Report Formatter, new option to put text lines at the bottom of the report and added function to sort the list of available fields.
- 3D Viewer - Added zoom extents function.
- Angle Info - Added ability to handle 3D angles.
- Layer ID - Added option to label the layer name.
- Command Search - Added reporting of toolbars for the commands.

Survey Commands

- Drone Mission Planning - New command to create a drone mission plan.
- Carlson Photo Capture - Added method to download files by file type.
- Input/Entry Distance Scale - Added method to scale distances in many COGO commands such as Enter Deed Description.
- Import Text/ASCII File - Added ability to read points from a SurvCE/Layout job database (.jdb) file. Added functions to save and load settings to a .rpnt file.
- Import Distance/Angle File - Added field for deed name for importing multiple deeds.
- Edit Process Raw Data - Added function to update the descriptions in the RW5 from a coordinate file. Added function to apply Field-to-Finish substitution codes to the descriptions in the RW5 file. Added report option for the description on the same line as the data. Added an optional warning on processing for points with a blank description. Added option to use a larger font for the spreadsheet. For processing GPS records added storing the lat/lon and RMS values to the CRDB as attributes.
- Edit Process Level Data - Added method to adjust elevations based on the number of turns.
- SurvNET - Added option to merge output point text file with control points.
- Draw Points - Added option to label elevations in inches.
- Point Group Manager - Added methods to create a point group of points unassigned to a group and to create group from point number suffix.
- Field to Finish - Added ability to run Triangulate & Contour to draw contours or make a TIN of the points and polylines. Added new utility feature. For pipe features, added ability for separate pipe feature settings per code, option to label structure names along the pipe and option to draw both pipe 3D faces and polylines. For tree features, added offset by tree symbol size as factor of text offset, added option to draw labels as MText and added settings to control order of input parameters. Added setting for annotative points per code. For GIS attributes on a line, added support for multiple attributes. Added option to match first symbol scale when creating a second symbol. New special code of OO for Offset Only that draws only the offset polyline. For OH offset horizontal special code, added method to repeat the offset multiple times. For hedge linetype, added ability to control width using OH or OFB special codes. For RECT special code, added setting for using a

third point as a rectangle corner or an offset for the rectangle width. For text attributes prefix/suffix settings, as string of [DESC] gets replaced by the point description. Added method to draw only points with GPS status of Fixed. Added option to hide the drawing under the Mtext labels. Added support for using fixed parameters with multiple codes.

- Offset Point Entry - Added option for slope stations.
- Annotation - Added option to use a custom block for endpoint leaders. For Stack Label Arc and Annotate With Leader, added option to create as MLeaders. For bearing leaders, added support for annotative leaders.
- Line/Curve Table - Added option to draw the table as MText without the grid lines.
- Stack Label Arc Auto Place - New command to automatically place the arc labels at the mid-point of the arc.
- Stack Label Arc Pick Side - Added option whether to draw leader.
- Label Angle - Added method to create an angle table. Added ability to handle 3D angles.
- Distance Table - New command to create a distance table of pairs of points and distances.
- Offset Dimensions - Added option to use CAD's current dimension style.
- Linetypes - Added method for multiple dashes between the symbol/letter such as —C—C—.
- Legal Description - Added method to report exterior angles.
- Map Check - Added support for azimuth format angles.
- Deed Correlation - Added method to screen pick points.
- Process Deed File - Added option to report using the Report Formatter.
- Area Descriptions By Interior Text - New command to tag area descriptions by automatically finding text inside polylines.
- Best Fit Line - Added method to screen pick CAD points, method to add more points to the calculation and option to set output layer.
- Convert CRD to ASCII By Folder - New command to batch process converting coordinate files to text files.

Civil Commands

- Triangulation Surface Manager - Add Point inside TIN hole has new option to fill the hole.
- Triangulation File Utilities - Added a Remove Spikes tool and added function to draw drainage low points.
- Grid File Utilities - Added export for DEM files.
- Triangulate & Contour - Added option to use linetype for depression contours.
- TIN Rebuild Inside Perimeter - Added method to apply inverse distance to adjust elevations within a perimeter. Added method to raise low points within a perimeter to fill-in low points to prevent ponding.
- 2D to 3D Polyline By Profile - New command to elevate a polyline using a profile.
- 2D to 3D Polyline By Screen - Added method to extrapolate elevations.
- Curb Ramp - Added graphic preview to dialog. Added option to taper only the first ramp section. Added method to do curb-cut only with no ramp. Added method to pick the curb position separately from the polyline selection. Added a separate curb taper method.
- Wall Polyline Editor - Added layback slope, freeboard, bury depth, output to solid model (.mdl), color/texture for 3D viewing, and report including backfill quantities and wall volume. Added double-click on the wall polyline to run the wall polyline editor. Added support for using wall polylines in Takeoff design surface modeling.
- Elevate Pads By Grade Rules - Added option to add break points to lot lines and option to create swales.
- Cut/Fill Movement - Added option to label cells.
- Stockpile Volume - Added method to use lowest elevation around perimeter as flat base elevation.
- Surface 3D FlyOver - Added support for viewing solid models (.mdl). Added settings for camera based light and shading for front, back or both sides. Add Model to the scene also draws a symbol in the drawing when the Model Library has the model defined with a CAD symbol.
- Slope At Points - Added method to use a coordinate file and two points to define the slope.
- Elevation Zone Analysis - Added method to use TIN surface and set colors in the TIN file for the elevation zones.
- [doc] Vehicle Path Tracking - Added option to draw section view of vehicle with dimensions and added vehicle section view preview to dialog.
- Polyline To Centerline File - Added method to define station at a reference point.

- Label Station Offset - Added method to label CL crossings in plan view with option to label layer names.
- Design Parking Inside Perimeter - Added ability to create islands at ends of parking rows. Added option whether to create spaces around the perimeter and option to add remainder parking space to the corner.
- Design Parking Along Alignment - Added support for alignments with multiple segments, ability to use a polyline or centerline file for the alignment, and support for angled parking. Added functions to create auto treatments and to mirror treatments between left and right sides.
- Draw Template Polylines - Added option to densify the template polylines.
- Template Series File - Added interpolation option for none.
- Shrink/Swell Table - New command to define different shrink/swell factors by station range for cut/fill and subgrade materials.
- Draw Super Elevation Color Centerline - New command to draw the centerline polyline with colors for the super elevation transition zones.
- Process Road Design - Added method to Template Point Profile to apply change in slope to another grade for a divided highway. Added method to use clip lines at the ends of the road for skewed ends.
- Road Network - Added method to create a bottom of subgrade surface. Added method to hatch the template grade areas. Added method for road template gaps by station range.
- Lot Network > Road Network - Added option to chamfer ROW intersections instead of arcs.
- Station Polyline/Centerline - Added method to store deflection angle into the point description.
- Draw Profile - Added option for fixed grid top/bottom elevations. Added option to draw horizontal line at bottom of elevation axis labels. Added option for line under label base elevations. For utility profiles, added pipe thickness and option to limit max offset for station by reference CL. For hatch, added option to label areas. For sewer profile labels above the rim, added method to place by grid top location and option for vertical labels with a leader from the rim. Added option to label manhole names using MText.
- Draw Section File - Added method to label the total section distance. For tunnel sections, added support for end areas from a single closed section. Added option to create labels as MText. Added ability to set a description for each station that can be drawn in Draw Section File and reported in Section Report. For slope labels, added filter by user-specified slope amount.
- Section File Editor - Added ability in translate elevations to use a profile to adjust the sections.
- Section Report - Added method to only report user-specified offsets.
- Set Descriptions By Offset - New command to set cross section descriptions by offset.
- Draw Mass Diagram - Added method to graph as immediate cut/fill difference instead of accumulative.

Hydrology Commands

- Rain Lookup Table - New command to store custom rain events.
- Merge Watersheds - New command to combine the polyline perimeters of neighboring watersheds.
- Time of Concentration - Added Kerby-Kirpich method.
- Channel Routing - Added Muskingom-Cunge and Modified Puls methods.
- Pond Exfiltration - Added functions to save and load parameters to a .pdx file.
- Drop Pipe Spillway - Added lookup table for entrance loss coefficient. Added functions to save and load parameters to a .drop file. Added PDF report and function to draw diagram to the drawing.
- Pipe Culvert Design - Added report option for stage-discharge graph and option to draw HGL on pipe detail graph. Added support for Muskingom-Cunge method.
- Advanced Weir Design - Added functions to save and load parameters to a .weir file.
- Orifice Design - Added functions to save and load parameters to a .outlet file.
- Multiple Outlet Design - Added PDF report method with diagram and stage-discharge graph.
- Sewer Network - Added ability to automatically set inlet as on-grade or sag based on the surface. Added support for Inlet Control Devices to limit intercepted flow. Added support for structures with underdrains. Added new yard inlet structure. For SCS rainfall lookup, added methods to choose from lookup table, rain map or NOAA. Added method to round elevations by decimal precision. For draw plan view, added setting for direction of pipe polylines as upstream or downstream. For data table, added ability to control which structures to include in the table. For Review Sewer Network Links, added a method to reassign the data files. Added pipe thickness to editor and reports. Added setting for having raised rim heights. Added a new Line

- Name field for each structure.
- Collision Check - Added option to draw 3D lines between the crossing pipes along with 3D text for 3D viewing. Added coloring of pipe portions that have conflicts. Added report of minimum horizontal and vertical clearance. Added method to check for collision with a surface model. Added option to color pipe segments with a conflict.
- Compare Sewer Networks - New command to report differences between two sewer network models.
- Draw Sewer Profile - Added ability to draw profiles for normal and critical depth lines.
- HydroNet Explorer - Added support for circular pipe reach and culverts. Added support for the Clark and Snyder hydrograph method. For draw plan view, added options to create MText labels, draw connection lines, use default symbols and save/load settings. For reports, added method to control the order of elements.
- Utility Network - For pipe labels, added settings to control offset from line and layer for labels. For creating profile at depth, added method to create streamlined profile with min number of profile points and ability to set max depth and min segment length. Added report function to check pipe depths and min/max cover. Added a description field for each utility with option to label in Draw Profile and Draw Sections. For laterals, added location by offset and slope.
- Pressure Pipe Network - Upgraded calculation engine to EPANet 2.2. Added ability to set tank level to create specified pressure. Added ability to add multiple connections or laterals at a time with a specified station interval and demand. Upgraded interface for setting up demand patterns. Added ability to draw symbols for valves in plan view.
- Pipe Network ID - New command to pick a pipe network entity and report which pipe network model it belongs to.
- Review Network Drawing Links - New command to review pipe network entities in the drawing and report network files with ability to reassign network files.
- HEC RAS - Added method to output a cross section of the water level. Added support for version 6.0.

GIS Commands

- Upgraded User-Interface - New dialogs for many commands including GIS Data Editor and GIS Data Inspector.
- Define GIS Features - Added import for StartNET, ESRI SHP and Carlson GIS .db features.
- Point GIS Editor - Added import for StartNET attributes.
- Place Camera Symbol - Added method to use points.

Geology Commands

- StrataCalc - Added new configure setting to limit pinchout by strata name range.
- Draw Geologic Column - Added complete custom control over formatting and which values to label. Added option to label bed thickness.
- Drillhole Text - Added method to create MText.
- Google Earth Drillholes - New command to tag drillhole symbols with drillhole/strata labels for viewing in Google Earth.

Surface/Underground Mining Commands

- Surface Mine Reserves - Added custom strip ratio formula. Added non-key tonnage to stored pit attributes.
- Prepare Value Block Model - Added cut surface and value calculation method tracking and grid info. Added export to text file function.
- Assign Directions - Added support to define directions by lines or leaders.
- Surface Equipment Timing - For 3D Pick, added ability to turn benches on/off. Added delay amounts to the report. Added output of period centroids to the spoil timing output file. Added option to skip non-key for timing.

- View Surface History - Added method to color surfaces by timing periods.
- Label Pit Formatter - Added options for commas in numbers and for higher order rounding.
- Block Model Viewer - Improved loading speed by over 10 times. Added option to merge matching grade blocks, option to render the outer shell of grades, option to tag blocks for inspecting in the viewer and ability to set the transparency of block by grade.
- Block Model Statistics - Added method to report by grades.
- Make Solid From Design - Added support for vertical alignments. Added option to draw cross section and template 3D polylines.
- Make Solid By 3D Triangulation - New command to create a 3D solid for a tunnel from a point cloud.
- Create Gap Points - New command to create points on a plane to fill in gaps in the point cloud.
- View Solid - Added setting for separate color for back side of 3D Faces.
- Draw 3D Mine Model - Added method to output an MDL file.

CADnet Commands

- 3D Image Viewer - Added method to download aerial image from Google Earth.

Construction/Trench Commands

- Convert Dashes to Polylines - Added individual method to trace by picking starting segment.
- Topsoil Removal - Added method to assign topsoil removal as outside of subgrade areas.
- Input Trench Line - Added option to interpolate invert elevations in case of junction without elevation label.
- Trench Network Quantities - Added option to use a user-specified TIN as the reference surface. For the plan view map, added option to label stations.

Point Cloud Commands

- Import - Added function to import cloud data from CAD points.
- Export - Added support for E57 format and RCS format.
- Bare Earth - Added option to put the ground points into a region instead of creating a new cloud. Added method to create a cloud of the non-ground points.
- Create Polyline - Added method to create polylines with tangential arcs.
- Edit Polyline - Added function to zoom scene to current polyline in editor dialog.
- Extract Slices - New function to view cross sections of cloud data and create points and polylines.
- Curb Breaklines - New method to create 3D polylines for top and bottom of curb by using edge clusters.
- Create Breaklines By Slope - New function to create 3D polylines at the boundaries of slope zones.
- Color Lines - Added new method for creating polylines that follow color lines in the cloud such as roadway paint stripes and new method to follow color boundaries such as edge between pavement and grass.
- Slope Lines - New function to create breakline polylines by following a slope grade break.
- Feature Extraction - Added batch process method to extract multiple features at a time.
- Power Lines - New function to extract overhead wires as polylines.

Data Conversions

- Geoids - Added geoids for Germany-Hessen, Thailand and Lithuania.

Setting Up a Project

Over 200 program settings can be specified in the Configure command under the Settings menu. These values are used to initialize Carlson program options when opening a new or existing drawing. Among these settings is the coordinate point number format, file and printer output options and settings for each module.

To set the drawing defaults, edit the template drawing (.DWT file). This drawing is loaded when new drawings are created. In the template drawing you can set the layers and AutoCAD /IntelliCAD variables. For example you could create your standard layers and set variables as you like such as BLIPMODE off. For Carlson, the drawing template should be set to Carlson##.dwt where the ## is the AutoCAD version number. For Carlson running in AutoCAD 2007, the template name is Carlson07.dwt. The Carlson template is located in the Carlson support directory (i.e. C:\Carlson2008\SUP\Carlson07.dwt). To customize the template, run the *OPEN* command and choose the drawing template. In the Select File dialog, set the type of file to Drawing Template (DWT) instead of regular drawings (DWG). Then make your changes and *SAVE* the drawing as Carlson##.dwt.

When starting a new drawing, one of the first steps is to run Drawing Setup under the Settings menu. Drawing Setup sets the drawing scale, the unit mode as either English or metric, and the text, symbol and linetype size scalers. The initial values for these Drawing Setup variables are set in Configure. When a drawing is saved, the Drawing Setup variables are saved with the drawing.

In Carlson, the text style height should be set to zero. The Carlson routines will set the text height according to the drawing scale and text size scaler set in Drawing Setup. For example, if the horizontal scale is set to 50 and the text size scaler is 0.1, Carlson will draw the text with a height of 5 ($50 * 0.1$). Then when the drawing is plotted at 1"=50', the text will be 0.1 inches. Use the *STYLE* command to set the text style height to zero.

The Set Data Directory command in the Settings menu can be used to specify the directory for the project data files. By default the drawing is stored in the Carlson WORK directory and the data files are stored in the DATA directory. The drawing file is the (.DWG) file. The data files are the coordinate (.CRD) file, profile (.PRO) file, grid (.GRD) file and other Carlson data files. In Configure>Project/Data Folders, there is an option to store all data files in the directory of the drawing. With this option active all the files for the drawing C:\Carlson2008\Work\JOB500\JOB500.dwg would be stored in C:\SCAD2006\WORK\JOB500.

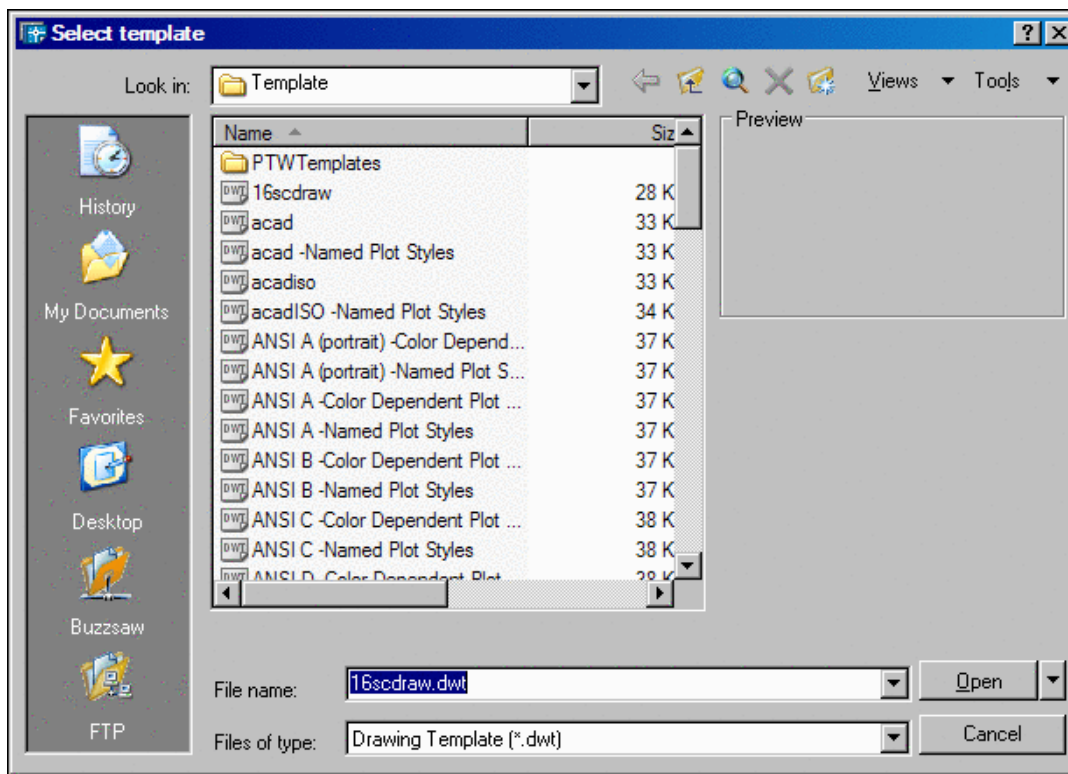
Another level of file management is the automatic project file recall. Every drawing remembers the data files that are being used for the drawing. When the drawing (.DWG) file is saved with the SAVE, SAVEAS, or QSAVE command, Carlson writes a settings file that contains all the active data file names. Then when the drawing is reopened, the data files default to their previous settings. For example, you won't have to choose which coordinate file to use unless you want to change it. The settings file is stored in the same directory as the drawing file and has the same name as the drawing with a .INI extension. For example, a drawing survey.dwg would have a settings file called survey.ini. You can turn off the INI files with the Save Drawing INI Files option in Configure under General Settings.

New/Startup Wizard

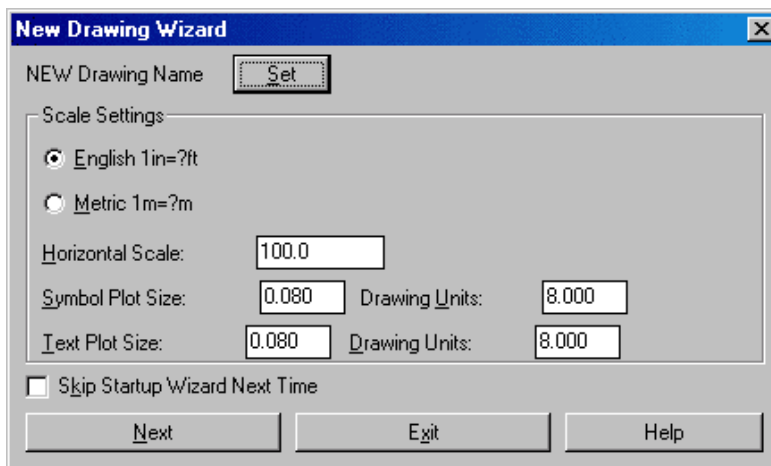
The New command is used for starting a new Carlson drawing. This page describes this New command and the Startup Wizard, along with the Carlson variables, associated with it.

Built into this routine is a Startup Wizard that can step you through and make the new Carlson drawing setup process easier. For creating a new drawing in Carlson, the Startup Wizard guides you through starting and setting up the drawing. This wizard is optional, and can be turned on or off in the Configure > General Settings command, which is part of the File pulldown. There is also a dialog box option, shown and mentioned below, that allows you to disable this feature. You can also exit out of the Startup Wizard at any time.

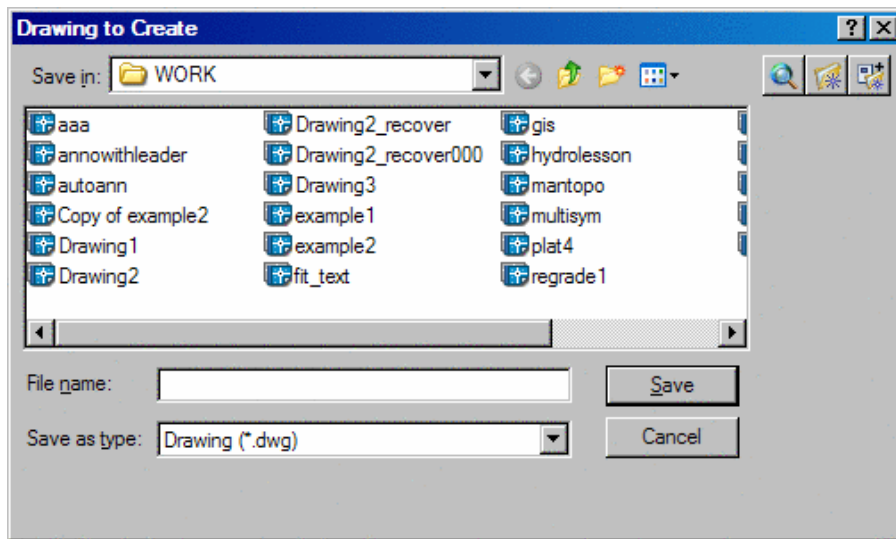
When the *New* drawing command is executed, you first get the standard Select template dialog box. While there are many templates to choose from, and there is an Open option, typically you want to go with choosing the Carlson drawing template (CARLSON17.DWT). The drawing template will set of some basic drawing parameters such as default layer names.



After selecting the template, the Carlson Startup Wizard begins by opening the *New Drawing Wizard* dialog box.

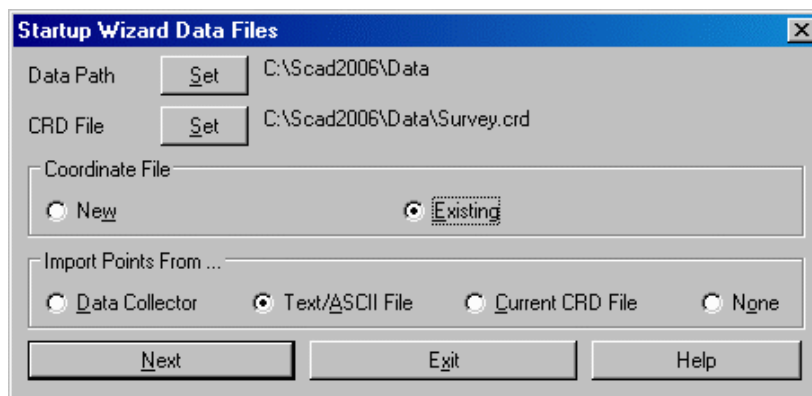


This dialog is used to set the drawing name and scale. The first step to do is set the drawing (.DWG) name by picking the *Set* button. This brings up the file selection dialog. Change to the directory/folder ("*Save in*" field) where you want to store the drawing. You can either select an existing folder or create a new folder. To select an existing folder, pull down the *Save in* field to select a folder or drive, click the Move Up icon next to the *Save in* field and/or the pick the folder name from the list. To create a new folder, pick the Create New Folder icon to the right of the *Save in* field. Then type in the drawing name in the *File name* field and click the *Save* button.



After setting the drawing name, you can set the drawing horizontal scale, symbol size, text size and unit mode (English or Metric). Notice that at the lower left corner of the *New Drawing Wizard* dialog there is an option to Skip Startup Wizard Next Time. Typically, you would leave this option unchecked, as the Wizard is a handy tool for new drawing setup. Now click the Next button.

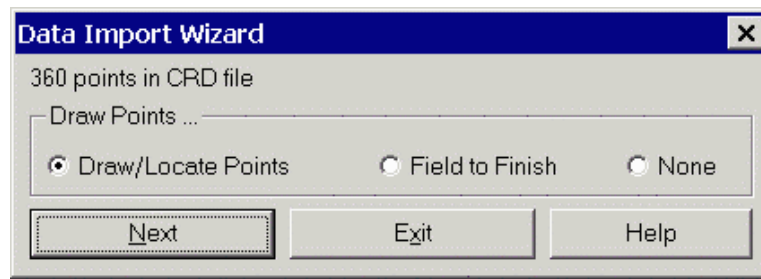
The next startup dialog sets the Data Path and CRD File. The Data Path is the folder where Carlson will store the data files such as raw (.RW5) files and profile (.PRO) files. The *Set* button for the Data Path allows you to select an existing folder or create a new folder. See the Set Data Directory command for more information. The coordinate (.CRD) File is the coordinate file for storing the point data. There is an option to create a new or existing coordinate (.CRD) file. The new option will erase any point data that is found in the specified CRD file. The existing option will retain any point data in the specified coordinate (.CRD) file. If the specified coordinate (.CRD) file does not exist, the wizard will create a new file.



The next wizard step depends on the Import Points option. The Data Collector option will start the data collection routines to download data from a collector. The Text/ASCII option will import point data from a text/ASCII file. See the Data Collection and Import Text/ASCII File commands for more information on running these routines. The Current CRD File option is a popular one to choose for bringing in coordinates. If the None option is set, then the Startup Wizard is finished.

Once point data has been imported from the data collector, text/ASCII file or CRD file, the wizard guides you through drawing the points. There are options to run Draw/Locate Points, Field To Finish or None. If None is selected, then the Startup Wizard is finished. Draw/Locate Points will import the points into the drawing using the same symbol and layer for all the points. From the Draw/Locate Points dialog, set the symbol, layer and point attributes to draw (description, elevation) and then pick the Draw All button. The Field To Finish command will import the points into the drawing using different layers and symbols depending on the point descriptions that refer to the code table defined in Field to Finish. Also Field to Finish can draw linework. See the Draw/Locate Point and

Field To Finish commands for more information on running these routines. After drawing the points, the wizard will zoom the display around the points. Then the wizard is finished.



Pulldown Menu Location: File

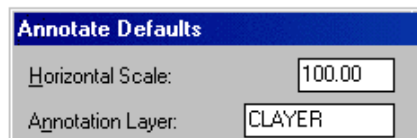
Keyboard Command: new

Prerequisite: None

Layer and Style Defaults

Many Carlson commands have default layers such as AREATXT for area labels and BRGTX for bearing and distance annotations. These layers can be specified in dialogs for the corresponding commands and several can be set in Configure. Sometimes you may want to use the current layer and it can be an extra step to have to open the dialog to set the layer. In this case, instead of using the default layer that set in the dialog, the default layer can be set as "CLAYER" which will use the current layer. For example, if the annotation layer is set to CLAYER then annotation will be drawn in the current layer instead of BRGTX or whatever the annotation layer used to be.

This same concept applies for text styles. Several commands have specific text styles and if you want to use the current style instead of the command style, use the name "CSTYLE" for the style name.



Carlson File Types

- .AAN Auto-Annotate Settings
- .ADF Annotation Default Settings
- .ARX AutoCAD Runtime Extension For Carlson Program
- .ATR Strata attribute definitions
- .AVG Mining Composite Quality Analysis
- .BLK Mining Block Model
- .CAL Mining equipment calendar
- .CAP Capacity file for hydrology (stage-storage)
- .CDF Geology Channel Sample File Format
- .CDS MDL Laser Raw Data
- .CDT Mining custom date table
- .CFG Configure Configuration Settings
- .CFZ Cut/Fill Color Map Zones
- .CGC C&G Coordinate File
- .CGR C&G Raw Data
- .CH Corehole definition
- .CL Centerline file
- .CLT Culvert Settings

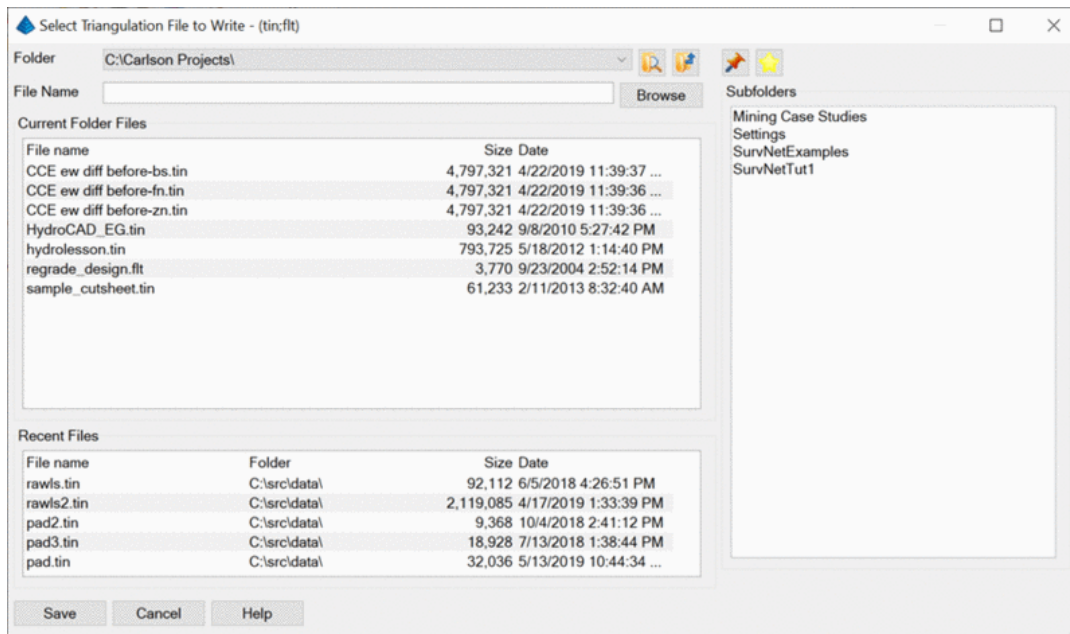
.CN Hydrology CN Factors
.COG Cadvantage Coordinate Data
.COT Multiple Outlet Design Data
.CQT Mining custom quantity table
.CRB Template Curb Definition
.CRD Coordinate file (point#, northing, easting, elevation, description) in binary form
.CTL SDMS Format Raw Data
.CTR Auto-Run Strata Isopach Maps
.CUI Customized User Interface AutoCAD Menu
.CUT SMI Format Cutsheet
.CVT SEDCAD Format Hydro Network
.DAT GPS Localization Definition
.DCF Deed Correlation File
.DCL Carlson dialogs
.DEM Digital Elevation Model
.DEQ Drillhole equations
.DHF Drillhole Text File
.DHT Dragline History
.DIL StrataCalc Convert As-Determined Qualities
.DLL Carlson programs files
.DTF Drillhole Data Format
.DTS Drillhole Text Settings
.DWG Drawings
.DXF Drawing Exchange Format
.DZR Dozer Push Settings
.EQO Mining equipment options
.EQU Mining equipment definitions
.ERD Erodible Channel Settings
.EW Earthwork Section End Areas
.EXE Carlson programs
.FCL Feature Code List for SurvCE
.FEN Fence Diagram Settings
.FLD Field to Finish code definitions
.FLT Triangulation mesh
.FMS Report Formatter Settings
.FRM Ferm Codes
.GCL Geologic Column Settings
.GEO Settings for Draw Geologic
.GFU Grid File Utilities macro command recorder file
.GIS GIS prompting definitions for note files
.GPF Grade Parameter File For Block Model
.GRD Grid file - a DTM surface model
.GRR Lot Edge Grading Rules
.GSQ Grid sequence history file for mining
.H1 Hydrograph
.H2I HEC-2 Water Surface Profile
.HYD Watershed settings
.HZN Horizon code definitions
.IMP Drillhole Import Format
.INI Program user preferences settings
.LAY Layer State
.LEV Level Raw Data
.LGD Annotate Draw Legend definition file
.LOT Lot file for Survey

.LPT Coordinate Transformation Point File
.LQE Least-Squares Measurement Error Settings
.LSP Carlson Lisp programs
.LSQ Least-squares input data
.LSS Localization Settings
.LTD Lot Attribute Definition
.LTN Lot Network Settings
.MAT Materials Library
.MDN Template Median Definition
.MIN Underground Mining Report
.MNC Compiled menus
.MNL Carlson Lisp programs for corresponding menu startup
.MNR Compiled menus
.MNS Compiled menus
.MNU Carlson source menus
.MNX Compiled menus
.MPD Mining Timing Project File
.MPJ Mining project definitions
.MXS Cross section alignment
.NOT Note file - additional descriptions for points in corresponding .CRD file
.OWN Mining Ownership Data
.PAN Mining Underground Panel Data
.PAR Parameters for mining quality compliance
.PAT Hatch Pattern Definitions
.PDD Enter Deed Description data file
.PFF Mining Pit Format File
.PFS Draw Profile Settings
.PHN Modem Settings For Carlson Field
.PIT Mining Pit Parameter File
.PLN Plan view polyline file for Dozer 2000
.PNA Mining panel attributes
.PPQ Polygon Processor Query
.PRE Geologic Model grids for strata model
.PRF Point Range File
.PRJ Project settings file
.PRO Profile (station, elevation, descriptions)
.PSZ Pipe Size Definitions
.PTA Mining pit attributes
.PVF System Variable Settings
.PVL Sewer Network Plan View Label Settings
.PVS Mining Projection and Ventilation Settings
.RCF Drainage Runoff Library
.RCL Runoff Coefficient Layer Definitions
.RDF Road Design File
.RDN Road Network File
.RDS Roadside Ditch File
.REC Seismic Record Output
.REP Hec-Ras Report File
.RES Mining Auto-Run Residuals Settings
.RME Reame Slope Stability Settings
.RMP RiverMorph Project
.RNF Road Network Stakeout for SurvCE
.ROW Right-of-Way offsets for Lot Network
.RPT Report Formatter Data

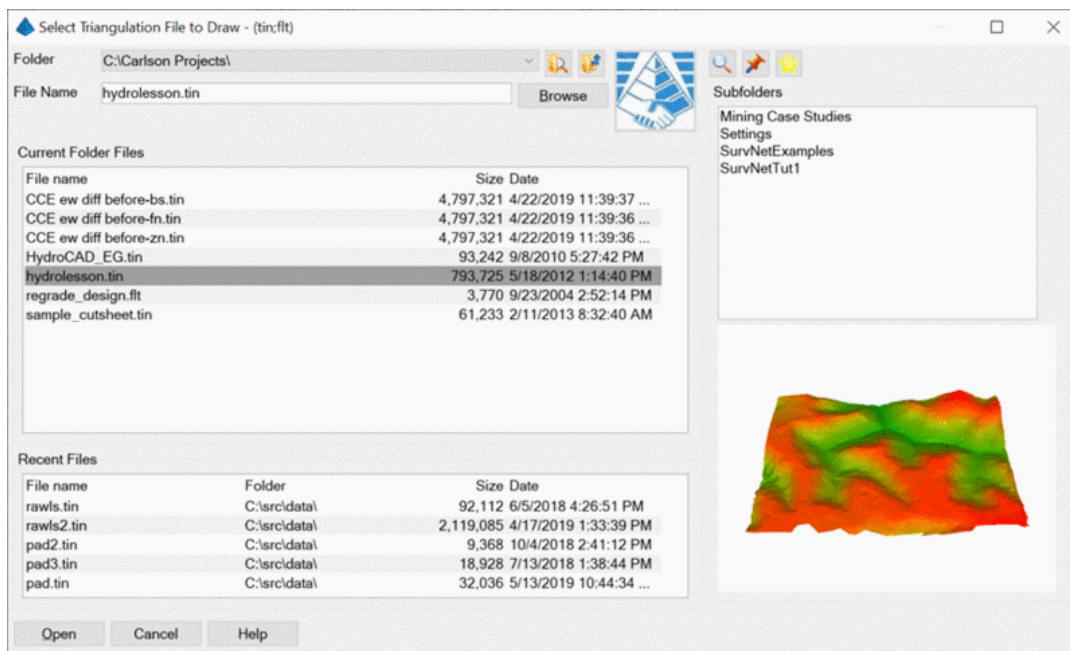
- .RSV Mining Reserve Description
- .RUN Auto-Run Strata Grids definition file
- .RW5 Raw file of traverse data
- .SC Coal Section Configuration
- .SC5 SurvCE Raw Data Archive
- .SCR COGO script file
- .SCT Cross section data (station, offset, elevation, descriptions)
- .SDF Strata Definitions
- .SDT SB-Slope Stability File
- .SEQ Dragline sequence file
- .SEW Sewer network file for hydrology
- .SGD Slope Group Definition for Design Pad Template
- .SIF Surface Inspector File
- .SLB Slide library
- .SLD Slide image
- .SLI Symbol Library
- .SST Draw Section Sheet Settings
- .STG Stage-discharge data for hydrology
- .STR Sewer Structure Library
- .SUP Superelevation file
- .TAB Hydrology Peak Flow Data
- .TCF Digitizer Tablet Configuration File
- .TCH Trench Structure Template
- .TIM Mining Timing Settings
- .TIN Triangulation Surface
- .TKD Takeoff Material Definitions
- .TOP Topsoil Definition for Road Design
- .TPC Template point centerline definition
- .TPL Template file for Section-Profile
- .TPP Template point profile definition
- .TPT Template transition file for Civil Design
- .TRG Takeoff Layer Target Settings
- .TRI Triangulate & Contour Settings
- .TSF Road Template Series File
- .UAO Report Formatter Attribute Options
- .UAT Report Formatter User Attributes
- .UDP User-Defined Projection For Grid Coordinates
- .VOL Auto-Run Strata Quantities definition file
- .WST Watershed Structure File
- .XML LandXML Data Exchange File
- .ZIP Compressed Data Files
- .ZON Earthwork Zone Definitions

File Selector

Throughout Carlson Software, various commands create, utilize or interact with data in files that support the project you are working with. The *File Selector* dialog box provides a common interface for identifying new and/or utilizing existing data files.



File Selector - New File tab



File Selector - Existing Files tab

Folder: The initial default folder location for a given file type is determined through the Data Folder Setup routine. When working within a project, the program tracks file selection for each file type and this folder will default to the folder of the last selected file. Use the drop-down control to select the folder from the list of past file selections and the default folder. You can also use the **Select Folder** button to browse for another folder.

File Name: Enter the desired file name. If a file extension is not provided with the file name, the first file extension listed in the dialog box header will be utilized.

Browse: Selects the file from the standard Windows file dialog.

Control



Action

Select Folder: Browse for a desired folder location if the folder location is not displayed in the *Folder* drop-list.

Moves up a folder level from the current folder.

Carlson Community: This button launches a dialog to select a file from the Carlson Community which is a file warehouse on an on-line server. Carlson Software customers can post files to share such as a Field-to-Finish code table for a certain application.

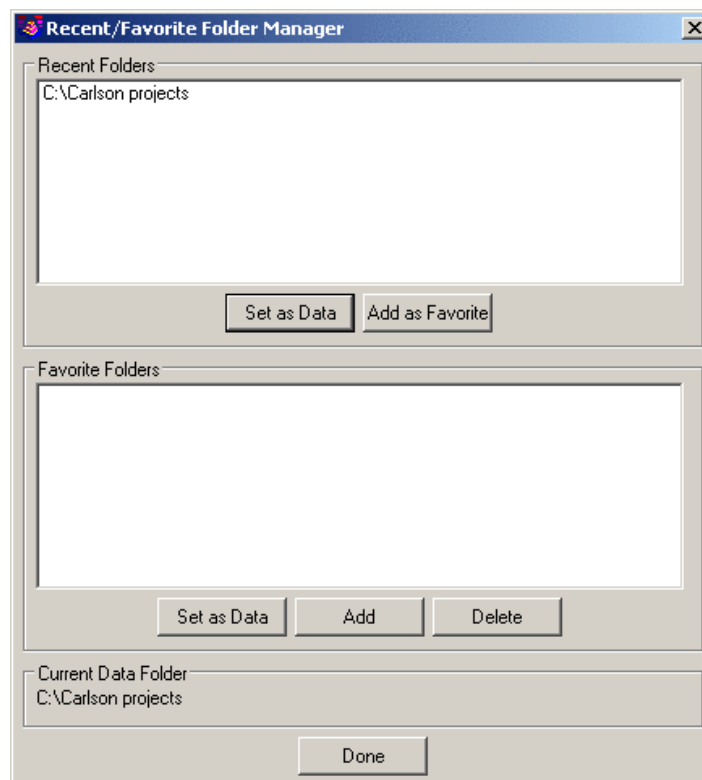
Existing File tab only. Quickly search for file(s) that meet specific file properties. See the Find File section for additional information.

Re-assigns the currently selected folder in the *Recent Folders* drop-list as the new **Data Folder**. Use the Set Project/Data Folder > *Re-assign Data Folders* option as needed to restore the Data folder.

Removes the selected subfolder from your computer.

Renames the selected subfolder on your computer.

Allows the management of Recent and Favorite folders as described below:



Recent Folders: Displays the list of current project and recent folders.

Set as Data: Performs the same action as the Set Data Button.

Add as Favorite: Adds the currently selected folder as a *favorite* folder.

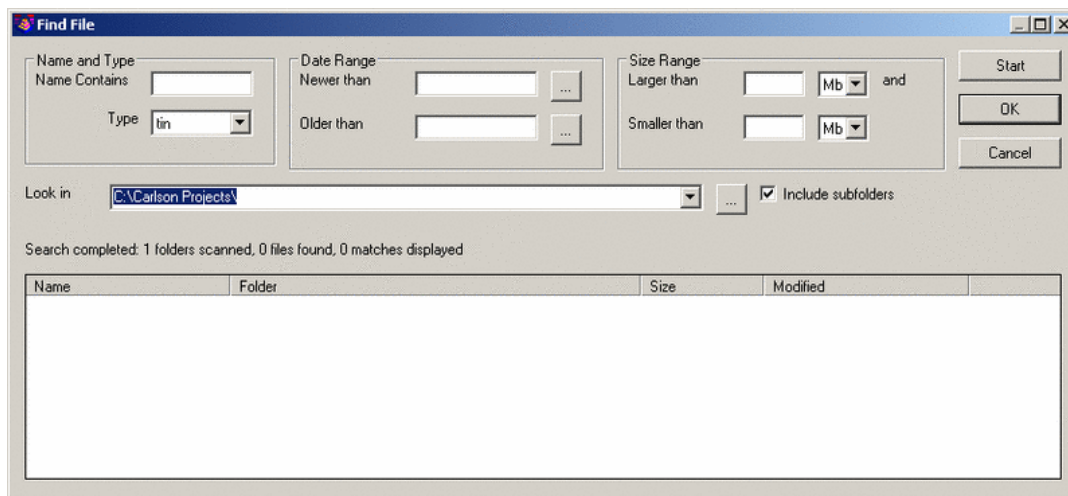
Favorite Folders: Displays the list of favorite folders (folders that are typically used or accessed repeatedly across projects).

Set as Data: Performs the same action as the Set Data Button.

Add: Allows any folder to be located and added as a favorite.

Delete: Removes the selected folder from the *Favorite Folders* list but does not delete any content found within the folder.

As the data files and folders for project(s) start to increase, the Find File option provides a mechanism to search for a particular file which meet certain file characteristics (all user-specified characteristics must met for the file(s) to be located):



Name Contains: Identify the string of characters located in the file name. For example, *Base* would return "BaseMap", "basement", "Subbase", etc.

Type: Indicate the specific file extension to be searched (the drop-list selection adjusts according to what is expected by the underlying command).

Newer than: Locate the file(s) that have been modified after the specified date.

Older than: Locate the file(s) that have been modified prior to the specified date.

Larger than: Locate the file(s) whose file size is larger than the value specified.

Smaller than: Locate the file(s) whose file size is smaller than the value specified.

Look in: Locate the file(s) in just the folder that is specified.

Include Subfolders: When enabled, the *Look in* folder is searched along with any sub-folders.

Start: Manually initiate the Find File routine.

As file(s) are located, select/highlight the desired file and click the OK button to return to the File Selector dialog box.

Note:

- In most cases, when a file name is typed into the Existing tab of the File Selector dialog box that does not exist in the *Recent Folders*, the routine will alert you the file does not exist and prompts if it should be created. If you instruct the program to create the file, it carries out the action of the New File tab.

Standard Report Viewer

Many Carlson routines display output in the Standard Report Viewer as shown below. A project name and job number can be added to the report header by filling out values for them in the Settings->Drawing Setup command. The format for the date in the upper right of the report is controlled by the Date Format setting in Settings->Configure->General Settings. The report can be edited directly in the report viewer. Report Viewer commands are described below.

Open: This allows you to open an ASCII file and display the contents in the report viewer.

Save: Save the contents of the report viewer to a text file.

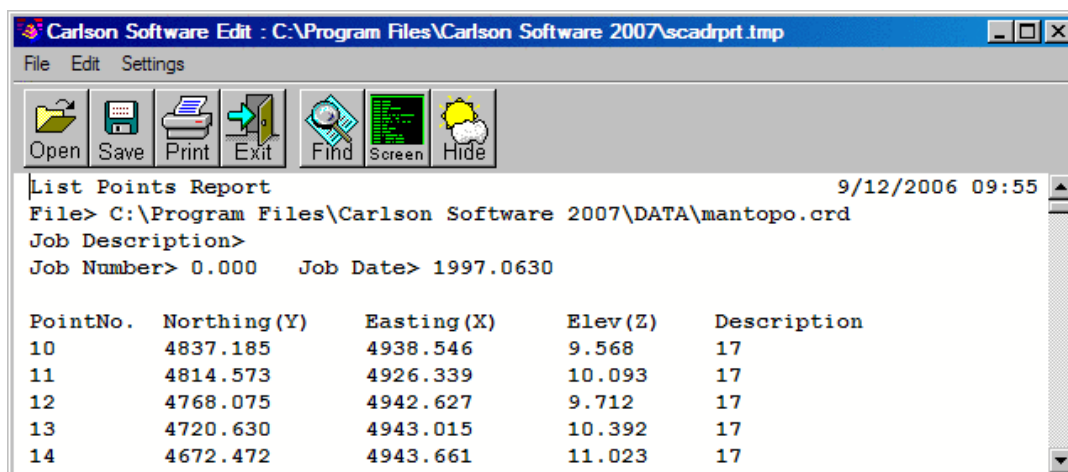
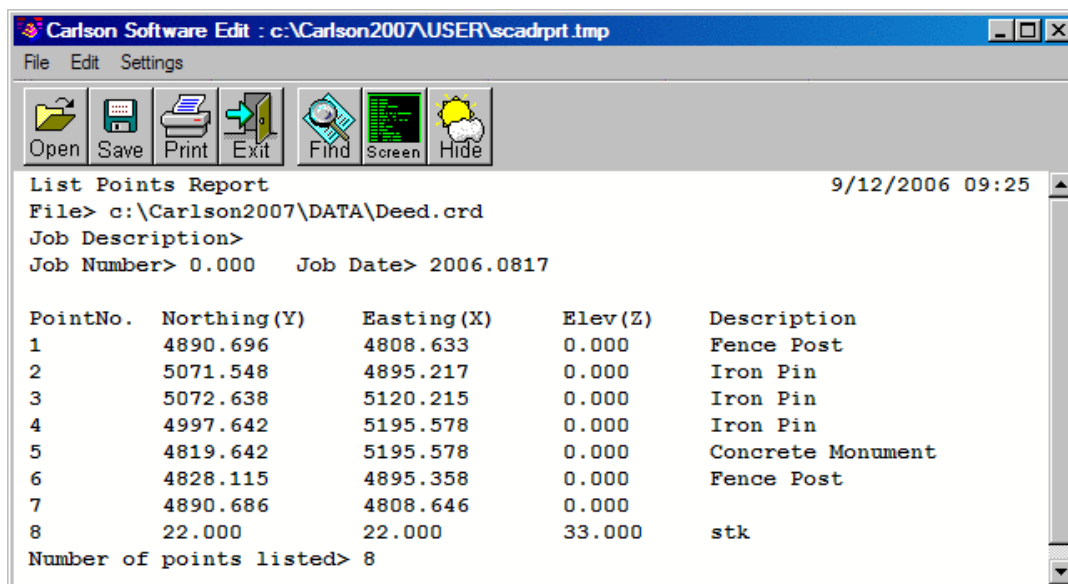
SaveAs: This allows you to save the contents of the report viewer to a file.

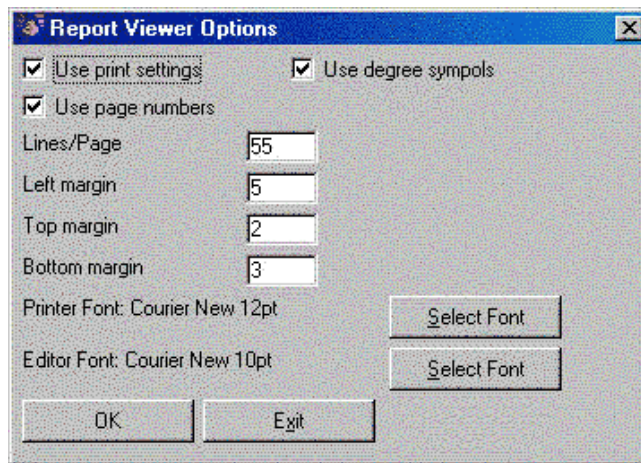
Append To: This allows you to append the contents of the report viewer to another file.

Print: Print the contents of the report viewer. This will open the standard windows Print dialog where you can choose the printer and modify any of the printer settings before you actually print.

Screen: Draws the report in the current drawing. The program will prompt you for a starting point, text height, rotation, layer and whether you want it inserted as Mtext or Text.

Undo: Reverses the effect of your last action. If you mistakenly deleted some text, stop and choose the Undo command to restore it. The key combination Ctrl+Z also performs this action.





Select All: Selects all the text in the report viewer.

Cut: Deletes the selected text and places it on the Windows® clipboard.

Copy: Copies the selected text to the Windows® clipboard.

Paste: Inserts ASCII text from the Windows® Clipboard into the report viewer at the cursor.

Search: Opens the Find Text dialog. Allows you to search for text in the report viewer.

Replace: Opens the Find and Replace Text dialog. Allows you to search for text and replace it.

Options: Opens the Report Viewer Options dialog. In this dialog, you can specify print settings, such as lines per page and margins. You can also specify the font used in the report viewer. This font is used for both the display and for printing.

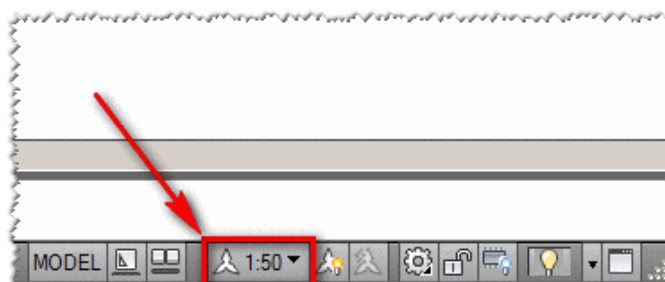
Hide: This button allows you to minimize the report viewer window and give focus back to the Carlson CAD screen. This allows you to return to working on the Carlson CAD screen without closing the report. You can re-activate the report by picking on the minimized report viewer icon.

Annotative Text

For CAD systems that support the creation and use of "Annotative Objects" (e.g. AutoCAD 2008 and later), various Carlson entities can be placed into a drawing that can make use of available "Annotative Styles," including:

- Attributes of Carlson Points via Point Defaults
- Text-based Annotation Labels via Carlson Configure>General Settings

When a plot scale is needed that is different than the Horizontal Scale established under Drawing Setup, change the *Annotative Scale* as shown below.

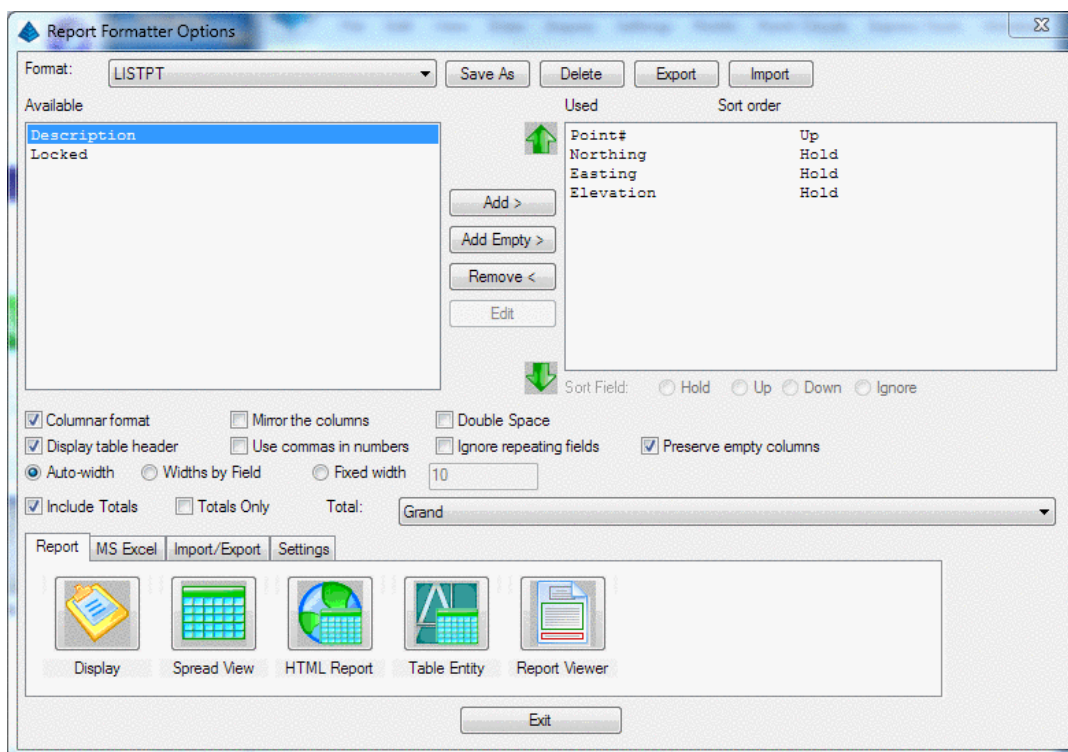


Note:

- Items placed as "annotative entities" will only make use of the annotation scales present in the drawing at the time of the entity placement.

Report Formatter Dialog

The Carlson Report Formatter routine is a highly customizable and flexible reporting engine that can be used to create a variety of output document types. A number of Carlson routines provide an option to use Report Formatter Options and allows you to specify how and which results of calculations should be presented in the report. In addition to the Standard Report Viewer, reports can be generated in web-friendly HTML format along with data formats compatible with Microsoft Excel or Microsoft Access.



Format: Select an available report format from the list of pre-established report configurations or key-in a new report format name.

Save As: Saves the current configuration of the active report format. The next time that you come to the Report Formatter from the same Carlson routine, it will recall this last format. To pick another format, select from list of formats in the left top corner and pick which format to use.

Delete: Removes the current report configuration from the listing of available report formats.

Export: Sends ALL available report formats to an XML-based "formatter style" (*.FMS) file.

Import: Imports the contents of a previously exported *.FMS file.

Control



Action

Moves the selected report option in the **Used** listing above the preceding entry until it reaches the top of the list.

Moves the currently selected item from the **Available** listing to the **Used** listing.

Removes the currently selected item from the **Used** listing and makes it selectable once again to the **Available** listing.

Moves the selected report option in the **Used** listing below the following entry until it reaches the bottom of the list.

Report Content Controls

Sort Field: For the selected "field" of data, indicate its sort method:

- **Hold:** The given field is not sorted and prohibits the sorting of subsequent columns.

- **Up:** The given field is sorted in ascending order.
- **Down:** The given field is sorted in descending order.
- **Ignore:** The given field is not sorted and permits sorting for the next column(s).

Columnar Format: When enabled, this toggle groups a given field of data into a column in the report. When disabled, each field of data is placed onto its own row in the report (the report data is output in a single column).

Mirror the Columns: (Suggested for short reports only). When enabled, this toggle transposes columns from the report into rows and vice versa.

Double Space: Adds a blank row between data rows.

Underscore Line: Adds a solid line between data rows.

Display Table Header: When enabled and exporting the report to an *HTML Report* format, the field "keys" as defined in the "Attribute Options" control are included in the report header.

Use Commas in Numbers: When enabled, this option will insert commas into numeric fields for every three digits.

Ignore Repeating Fields: When enabled, only the first occurrence of a repeating field is display in the report. Subsequent occurrences of the repeated field (*e.g.* the point description) are suppressed until a different value in the repeating field is encountered.

Preserve Empty Columns: When enabled, custom named columns can be added to the report which have no data in the rows.

Auto-width: When enabled, the width of each column is automatically set to be the wider of the column heading or the data contained within the column.

Widths by Field: When enabled, the width of each column established in the *Attrib Options* control (found in the *Settings Tab*) is used per field.

Fixed-width: When enabled, specify the width of each column.

Include Totals: When enabled, the total of each field is reported.

Totals Only: When enabled, only the total of each field is reported.

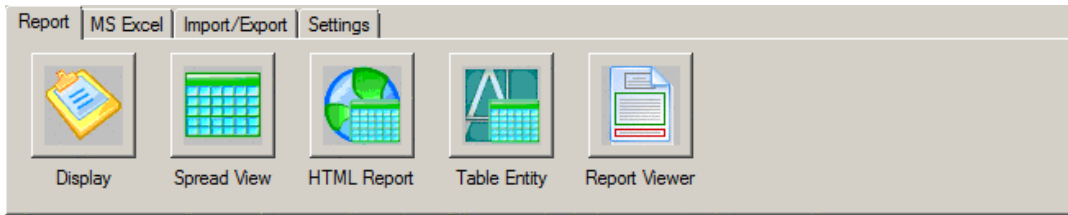
Total: Select the desired total for the *Used* field(s) of data. The summation level is defined by this "Total" pop-up list. By default, only grand total will be displayed at the bottom of the list. By picking the next item in that box, you will get subtotals added each time the value in first column is changed. It makes most sense to use this kind to summation if the corresponding column is sorted.

For example if the first column is "Area Name" and it is sorted, and "Total" is set to "Grand, Area Name" the report will have a sub-total for each distinct Area Name.

Once the desired fields of data and reporting options have been specified, the output can be generated and manipulated using one of four tabs:

1. Report
2. MS Excel
3. Import/Export
4. Settings

Choose one of the output options:



Control



Action

Sends the current report to the Standard Report Viewer command. Upon exiting the Viewer, you come back into the Report Formatter for further data manipulation as needed.

Sends the current report to a "spreadsheet" interface where it can be further exported to a variety of popular file formats. Additional information is provided in the Spreadsheet discussion.

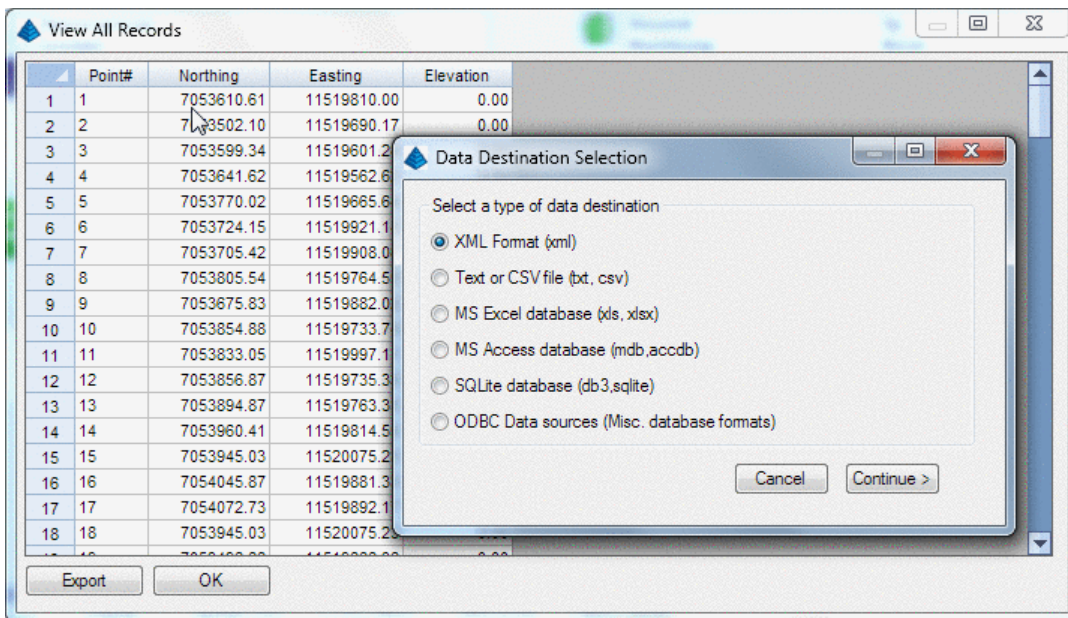
Sends the current report to an Internet/web-ready HTML file format and displays the report using the HTML viewer that is configured on your computer.

Places the current report as a table-type of entity into the current drawing. Additional information is provided in the Table Entity discussion.

Places the current report into a special-formatted report. Additional information is provided in the Report Viewer discussion.

Report Tab Options

Spreadsheet



Export: This button has the same output options as the Export function under the Import/Export Tab. Use this option to create a variety of popular file formats, including:

- XML Format (xml)
 - Text or CSV File (txt, csv)
 - MS Excel "database" (xls, xlsx)
 - MS Access database (mdb, accdb)
 - SQLite database (db3, sqlite)
-
- ODBC Data Sources (Misc. database formats)

For commands that process reports using perimeter polylines, the Report Formatter has an option to create GIS links between the polylines and the database records when the Export to MS Access function is used. Some commands that can utilize this functionality are Surface Mine Reserves with the pit polylines and Underground Timing with the panel polylines.

When the polyline data is available for the GIS Links, there will be a report field called **Handle**. This Handle field is the entity name for the polyline and serves as the hook for the GIS link. The Handle field does not need to be put into the report **Used** list in order to create the links. When the Export function is called with the MS Access method, there is a pop-up window prompt for whether to create the GIS links. When these links are created, you can then use the GIS Data commands from the GIS module to manage and report the data.

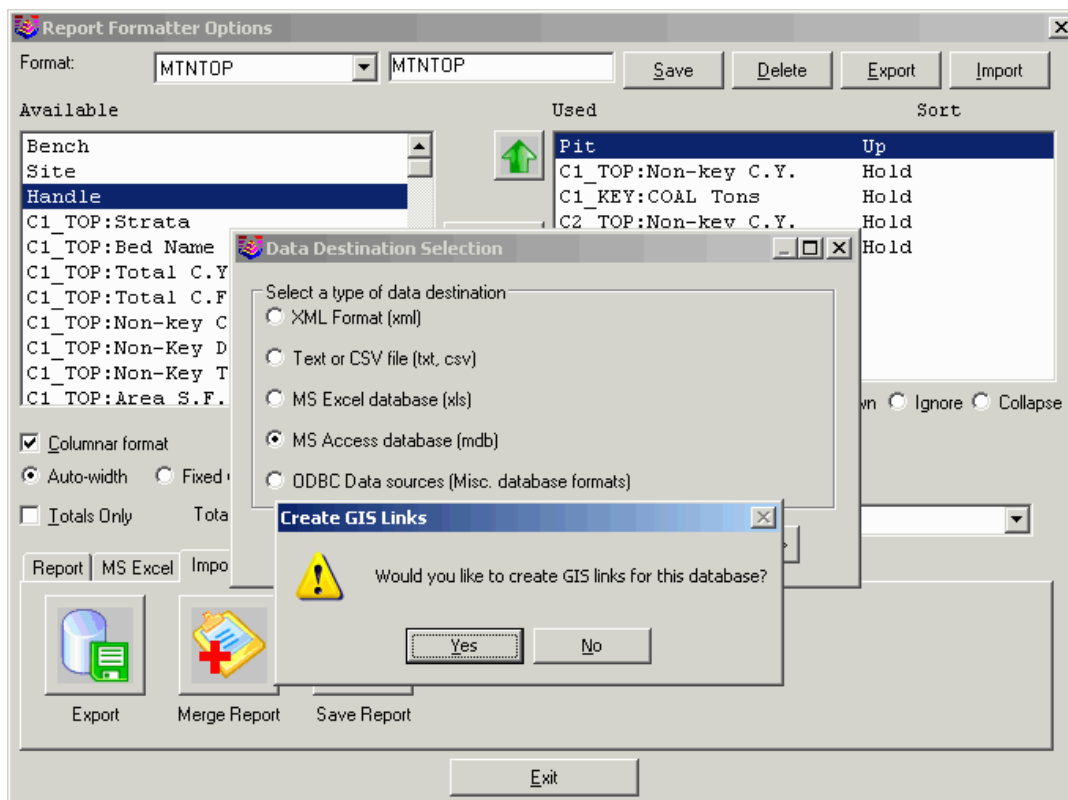
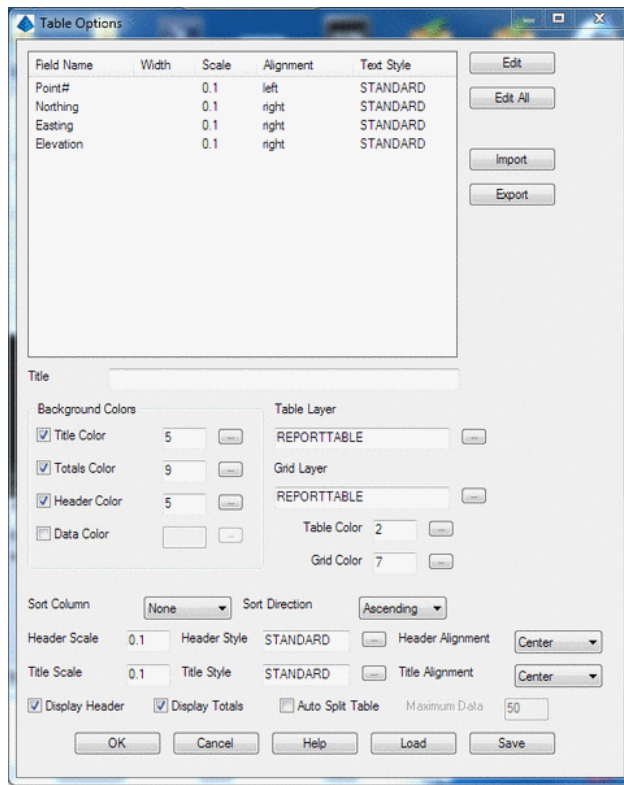


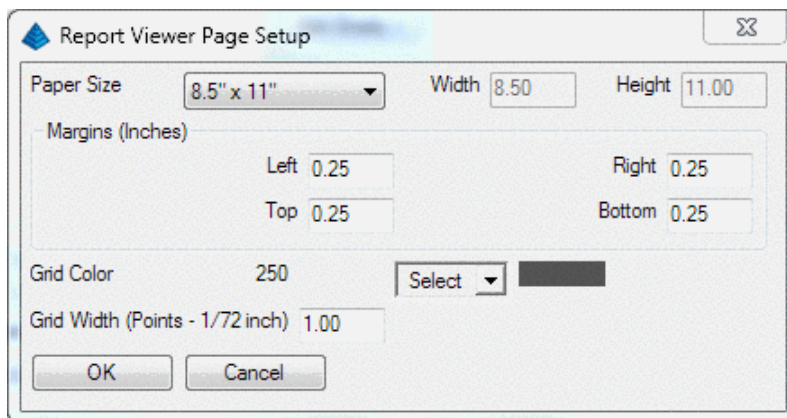
Table Entity



The data for the Table Entity is put into a queue and the table is not drawn until the Report Formatter is closed. Then the program prompts for a location to draw the table and provides options shown above to control items such as the header names, sizes, alignments, styles, colors and layers. You can also set whether to draw the table header and totals.

Report Viewer

The Report Viewer option provides the ability to produce more professional looking reports that contain horizontal and vertical dividing lines and can also be exported to a variety of common report formats.



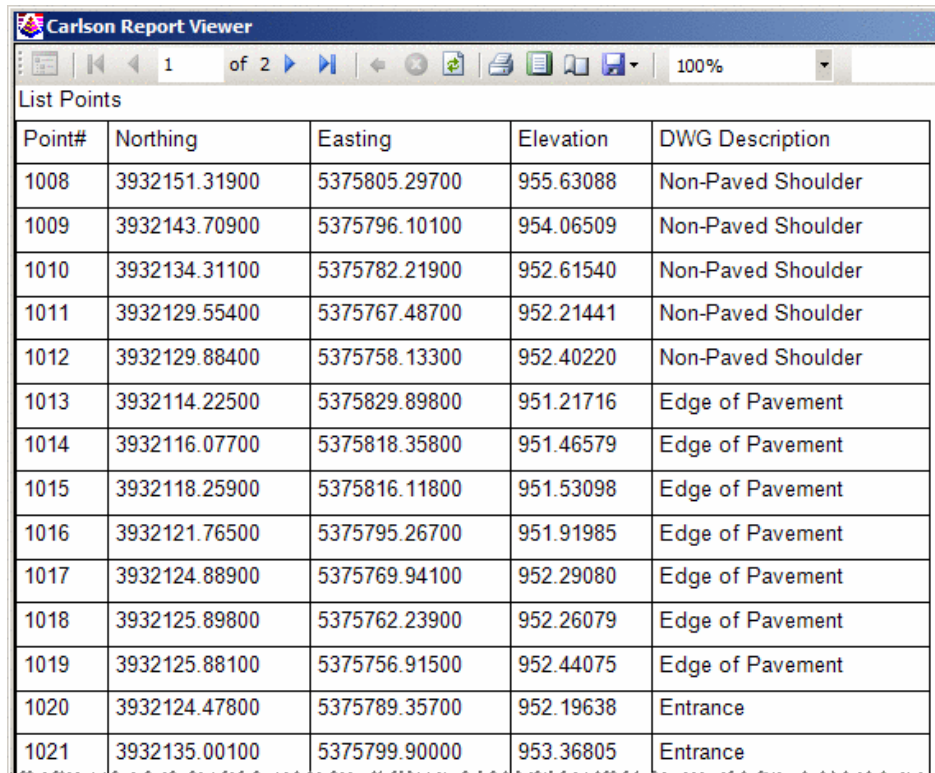
Indicate the paper size you will be printing to along with desired values for:

- Left Margin
- Right Margin
- Top Margin
- Bottom Margin

Grid Color: Used to choose the color for report grid lines.

Grid Width: Used to specify the grid line width in points (1/72 of an inch).

Upon specifying the desired values and clicking **OK**, the Report Viewer dialog box will display:

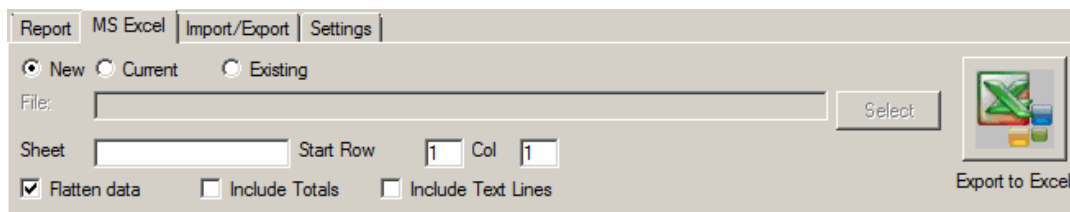


The screenshot shows the Carlson Report Viewer window. The title bar reads "Carlson Report Viewer". Below the title bar is a toolbar with navigation and utility icons. The main area displays a table titled "List Points".

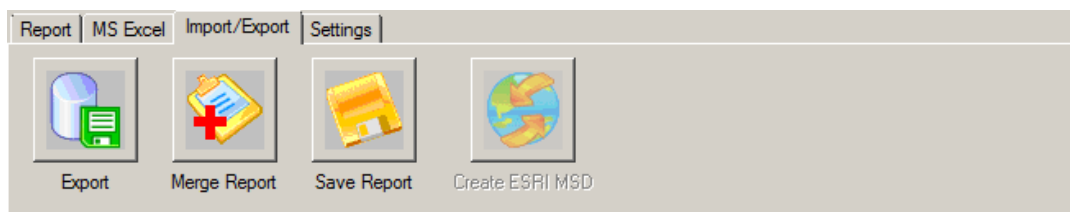
Point#	Northing	Easting	Elevation	DWG Description
1008	3932151.31900	5375805.29700	955.63088	Non-Paved Shoulder
1009	3932143.70900	5375796.10100	954.06509	Non-Paved Shoulder
1010	3932134.31100	5375782.21900	952.61540	Non-Paved Shoulder
1011	3932129.55400	5375767.48700	952.21441	Non-Paved Shoulder
1012	3932129.88400	5375758.13300	952.40220	Non-Paved Shoulder
1013	3932114.22500	5375829.89800	951.21716	Edge of Pavement
1014	3932116.07700	5375818.35800	951.46579	Edge of Pavement
1015	3932118.25900	5375816.11800	951.53098	Edge of Pavement
1016	3932121.76500	5375795.26700	951.91985	Edge of Pavement
1017	3932124.88900	5375769.94100	952.29080	Edge of Pavement
1018	3932125.89800	5375762.23900	952.26079	Edge of Pavement
1019	3932125.88100	5375756.91500	952.44075	Edge of Pavement
1020	3932124.47800	5375789.35700	952.19638	Entrance
1021	3932135.00100	5375799.90000	953.36805	Entrance

Controls within the report viewer allow you to:

- navigate through the page(s) of information
- refresh the report
- send the report to a printer
- switch between print layout and print preview mode
- re-configure the page setup
- export the report to other document applications
- specify the "zoom" level while displaying the report
- searching the report for a given search criteria



Several Microsoft Excel export options are provided. You may specify a spreadsheet file to load before the export, as well as a left upper cell to start with and sheet name to use. Text lines which are reported when using built-in viewer may be skipped when using Microsoft Excel export.



Control



Action

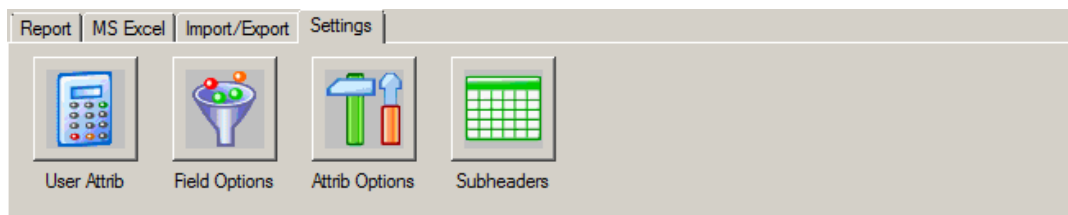
This button contains the same export options as described in the Spreadsheet discussion.

This button allows the current report to be combined with a previous report.

This button saves all the report data values as well as all the report format settings into a single *.RPT file that can be shared with others, merged with other reports or loaded at a later time.

For commands and data that conform to the ESRI Mapping Specification for DWG/DXF (MSD), this button creates an ESRI MSD-compatible report.

Import/Export Tab Options



Control



Action

This option allows you to define new fields as equations based on existing fields. Additional information is provided in the User-defined Attributes discussion.

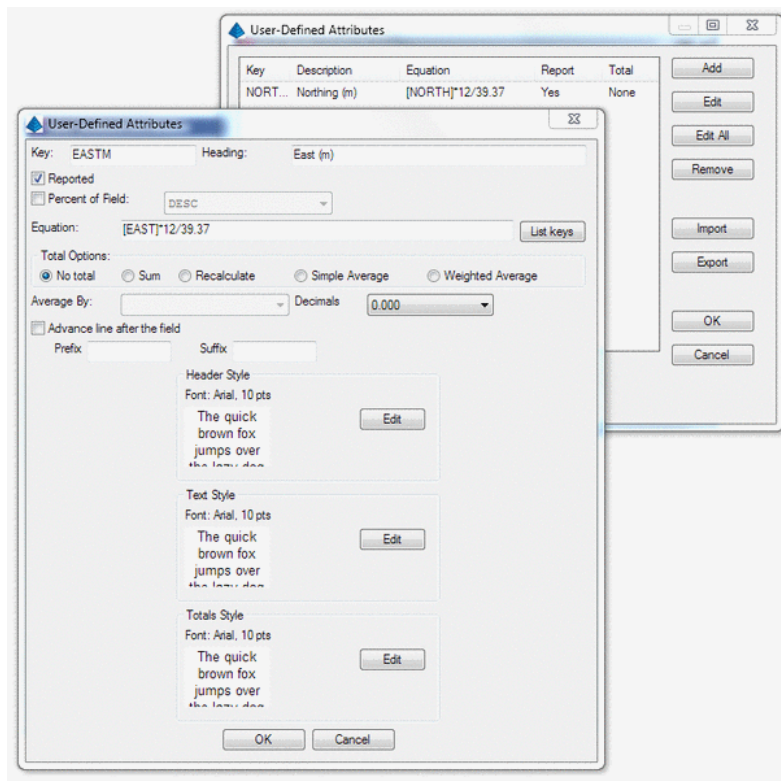
This option allows you to further customize additional content (*e.g.* Date/Time, Report Name, *etc*) into the report header, body and/or report footer. Additional information is provided in the Field Options discussion.

This option allows you to control several parameters of each field including title names, number of decimal places, *etc.* Additional information is provided in the Attribute Options discussion.

This option allows you to create more condensed reports yet still maintain easily decipherable column headings and/or permits a 2-line column heading. Additional information is provided in the Subheaders discussion.

Settings Tab Options

User-defined Attributes



You can create highly customizable fields of data using parametric equations from other fields of program-generated data... all without the use of an external spreadsheet! These User-Defined Attributes are mathematical manipulations to existing attributes. For example, if the report includes a LENGTH attribute reported in feet, you can create a User-Defined Attribute to also report this value in inches. When a User-Defined Attribute is added or edited, the above dialog will appear.

Key: This is a unique named used to identify the attribute. The Key can be used to reference the attribute in other User-Defined Attribute equations.

Heading: This is the label for the value that will show in the report. It is often useful to use a very descriptive name for the Heading.

Reported: This toggle controls whether to list this attribute as available for reporting or to only use internally as an attribute for the equation of other attributes.

Percent Of Field: This toggle defines the attribute to be a percentage of the total of another field such as the lot area percent of the total area. When this option is enabled, you will not be able to define an equation for the attribute.

Equation: This is where the attribute value is defined. To use the value of another attribute in the equation, the attribute name is entered in brackets as shown in the dialog above. The below table lists the mathematical operators that may be used in the equation.

Operation	Symbol
Add	+
Subtract	-
Divide	/
Multiply	*
Exponent	^
Grouping	()

Compare - Greater Than	>
Compare - Less Than	<
Compare - Equal To	=
Logic - AND	&
Logic - OR	—
Logic - NOT	!
Absolute Value	ABS()

Total Options: This controls how to total the attribute.

Sum: The total is a sum total of all report entries of the attribute.

Simple Average: The total is an average of all reported entries of the attribute.

Weighted Average: The total is a weighted average of the reported entries of the attribute. The attribute will be weighted based on the attribute selected in the **Average By** dropdown list.

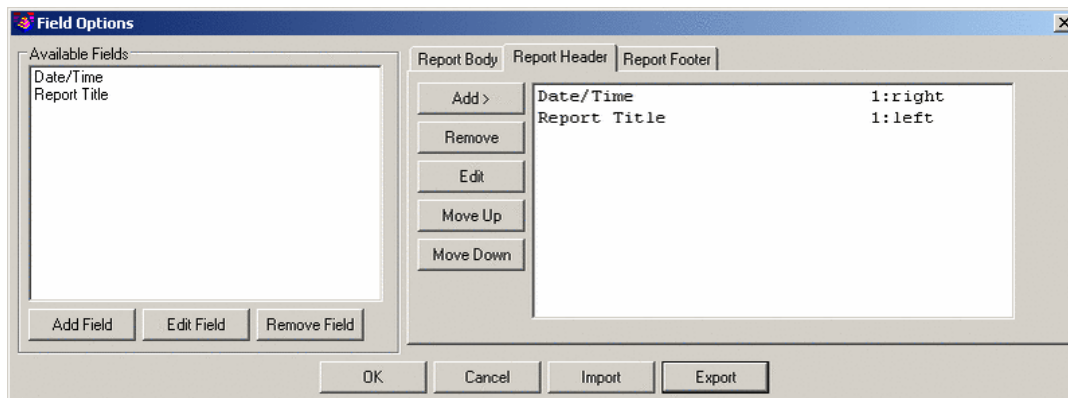
Recalculate: This total is not based on the individual entries of this attribute, but is instead recalculated based on the totals of referenced attributes. For example, consider the below report in which values A and B are naturally reported and C is a User-Defined Attribute equal to A+B. Each attribute uses a different Totaling option. In this case, the total for C is recalculated based on the totals for A and B rather than on the previous reported entries for C.

	A	B	C
	1	4	5
	3	8	11
	5	9	14
Total	3	21	24
Total by	Simple Average	Sum	Recalculate

Decimals: This dropdown lists controls the reported precision of the attribute.

Advance line after the field: This toggle adds a blank row to the report after the attribute is reported.

Field Options

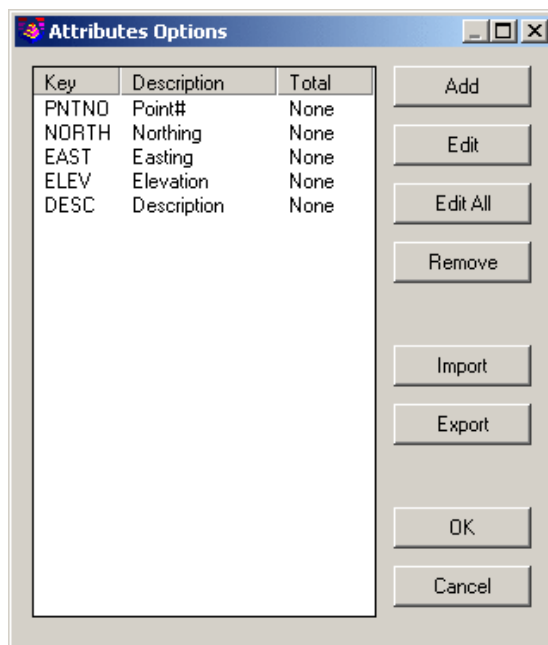


"Field Options" can be placed into one or more locations within the report:

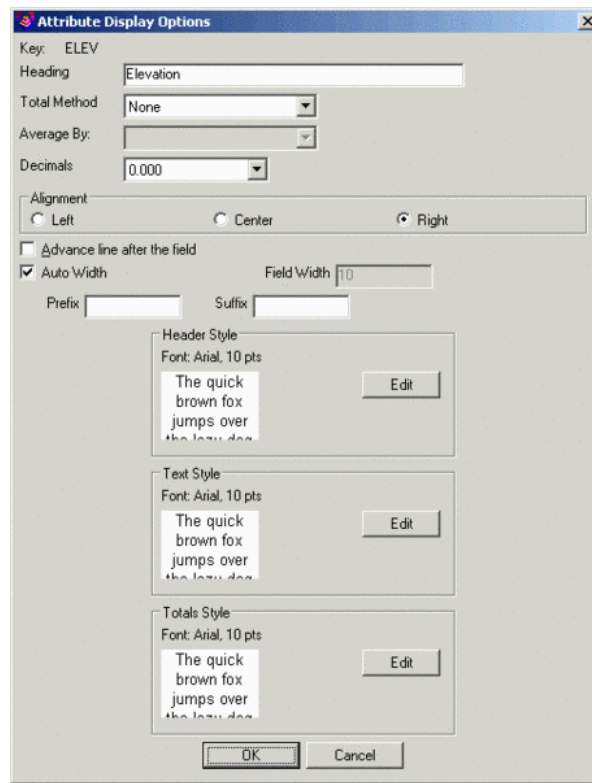
- Report Header
- Report Body
- Report Footer

Select the desired location for the field and then click on the various controls to add or remove the item from the desired location or change its formatting information.

Attribute Options



The Attribute Options dialog shows a list of all the attributes in the report. Use the Edit button to set the attribute properties.

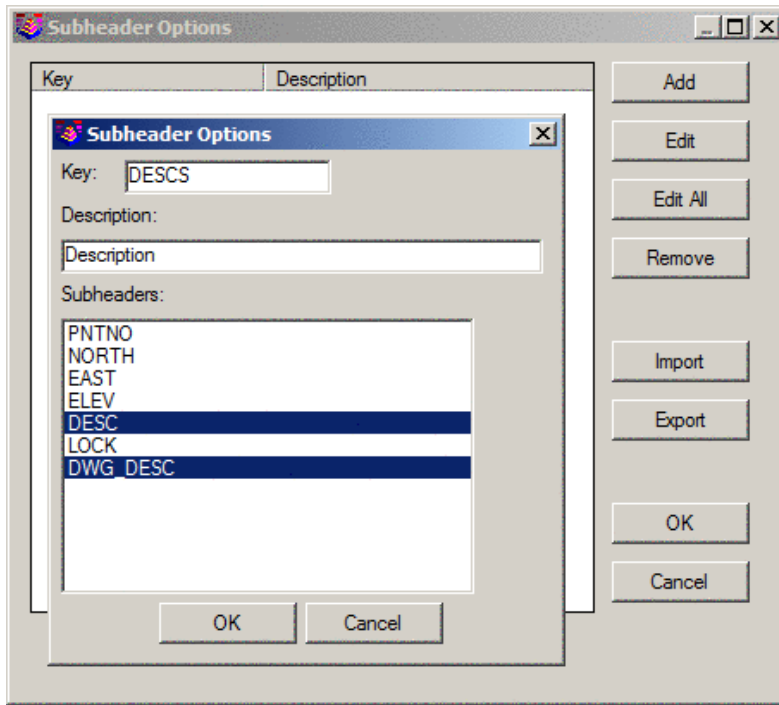


The Heading is the label to use for this attribute. The Total Method chooses between:

- Sum
- Simple Average
- Weighted Average
- No total

For Weighted Average, you can select another attribute to weight the average by such as tons for averaging a strata quality. When controlling the number of **Decimal** places, the values can also be rounded up to the nearest integer level such as when earthwork volumes need to be reported to the nearest 1000. The **Cell Format** controls the Excel cell format for this data field. The Default setting uses the cell format that the Carlson program output to Report Formatter. The **Alignment** controls the justification of the attribute. By default, numbers are right justified and strings are left justified. There is an option to **Advance Line After the Field** of the report which inserts a blank row and shifts the next attributes down to a new line in the report. The **Auto Width** option sets the attribute width to fit the attribute heading and all the values. Otherwise you can set a fixed **Field Width**. The **Prefix** and **Suffix** allow you to add to the attribute value such as adding the "ft" suffix to indicate feet units. The **Style** settings control the font to use for the header, body and total for the attribute. Use the Report Viewer output method to use the different style settings.

Subheaders



Subheaders may be created for situations where many columns of data may make the report too wide. You can use the subheaders to stack the column names. For example, a point list report with descriptions from the coordinate file also listed with descriptions from the drawing file could be setup with "Description" as a subheader (use Ctrl+Click functionality to select the desired fields that are to get a subheader). Consider (table borders added for visual clarity):

	Description	
CRD		DWG
TDC / Oak		Deciduous Tree Oak
Using Subheaders		

Contrast with:

CRD Description		DWG Description
TDC / Oak		Deciduous Tree Oak
Without Subheaders		

Instruction Manual and Program Conventions

Westwood

Italic text represent responses by the user that should be typed in and followed by the Enter key.

Number/<Pick point>:

Bold text represents prompts or questions that the computer program will ask the user.

<90.0000>

Values enclosed in corner brackets represent default values obtained by pressing Enter with a blank response.

[end on]

Lower case text enclosed in brackets in Command prompts denotes an *OSNAP* mode that is turned on by the command.

Language Localization

To run the program in another language, go to Settings->Carlson Configure->Localization Options and choose Windows Settings for the Language option.

Then set the language in Windows.

In Windows Vista, XP and earlier:

Run Start->Control Panel->Regional and Language Options. In Windows Vista, choose the language under the Formats tab. In earlier Windows, choose the language under the Regional Options tab.

In Windows 7:

Run Start->Control Panel->Clock, Language and Region. Then pick Region and Language and choose the language under the Formats tab.

There is a second Windows setup step for languages that don't use character set 1252 for English and Western languages. This second step is to run Windows Control Panel and go to the Advanced tab of Regional and Language Options and select the language. Then reboot Windows. For example, with Russian, you need to choose Russian under this Advanced tab.

After the language is set in Windows, restart Carlson to use the new language settings.

After restarting Carlson with the language setup, go to Settings->Carlson Menus->Menu Language Translation to translate the pull-down menus into the current language. This function will prompt for the source menu file to translate. For running on AutoCAD, there is only one menu file to translate which is called csXXbase.cui or .cuix where XX is the AutoCAD version. For example, for AutoCAD 2012, the menu to translate is called cs12base.cuix. For running on IntelliCAD, there is a base menu to translate and then separate menus for each module. The base menu is called icad6base.mnu for IntelliCAD 6 and icadbase.mnu for IntelliCAD 7. The menus for the modules are named after the module. For example, the survey.mnu is for the Survey module. For the IntelliCAD menus, you need to run this function multiple times to translate the multiple menus. After translating the menus, restart Carlson once more.

The language translation tables are stored in the Carlson EXEC\LANG\XX\LC.Messages folder. The XX is for the language abbreviation code. Each language has its own sub-folder with a 2 character name. There are 2 translation tables for each language called carlson-office.mo for the program prompts and carlson-office-mnu.mo for the pull-down menus.

Carlson provides tools to allow you to improve the translations by editing the source translation tables which are stored in PO files: carlson-office.po and carlson-office-menu-po. There are two methods for modifying the translations. One method is an on-line editor called POOTLE that is on the Carlson webpage, www.carlsonsw.com, under Support. Or you can use this link: <http://translate.carlsonsw.com/>. This system requires that you register with your name and email. Then you have access to edit the translation tables. After editing the translation table, you can update the translations within Carlson by rebuilding the MO files. To rebuild the MO files in POOTLE, choose the language then Carlson Office Products then Show Editing Functions. POOTLE will then list the translation tables carlson-office.po and carlson-office-mnu.po. Pick the MO function next to the table and save the MO file to the Carlson\EXEC\LANG\XX\LC_MESSAGES folder where XX is the language code such as FR for French.

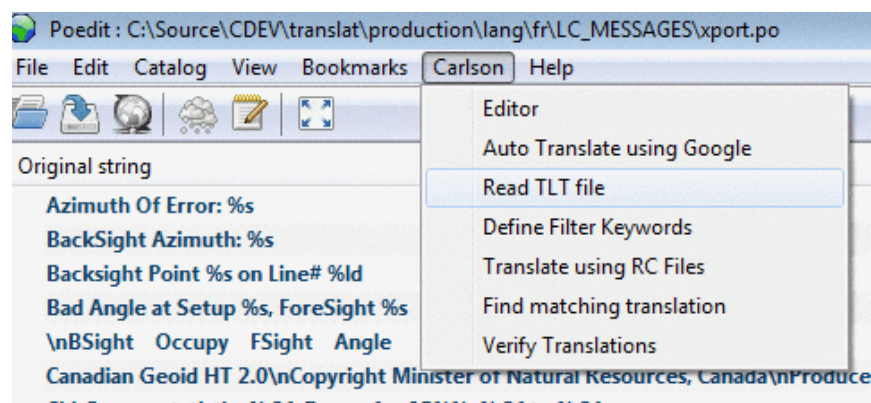
The other translation method is a desktop editor called POEdit. This desktop editor has more features than the on-line editor which could be useful if you are doing a lot of editing. This desktop editor distributed on request. When saving a PO file in POEdit, the program automatically creates the companion MO file which goes to the Carlson\Exec\LANG\XX\LC_MESSAGES folder for running. For more instructions the POEdit desktop editor, go to the Carlson webpage under Support.

After updating the MO files, the program will pick up the new translations for command prompts. The pull-

down menus are updated separately by running the Settings->Carlson Menus->Translate Menu command. You need to select the English base menu and then the translated menu is generated using the same name as the base menu with the language name added as a suffix. For the AutoCAD platform, there is a single CUI menu to translate called CSxxBASE.CUI where xx is the AutoCAD version. For example, for AutoCAD 2010 this menu is called CS10BASE.CUI. The French translated version of this menu will be called CS10BASE-FR.CUI. For the IntelliCAD platform, there is a base menu called ICADBASE.MNU as well as menus for each of the modules such as SURVEY.MNU and CIVIL.MNU. The French translated versions of these menus will be called ICADBASE-FR.MNU, SURVEY-FR.MNU and CIVIL-FR.MNU. After generating the translated menu, load it using CUILOAD command in AutoCAD and the MENU in IntelliCAD.

POEdit with Carlson Menu

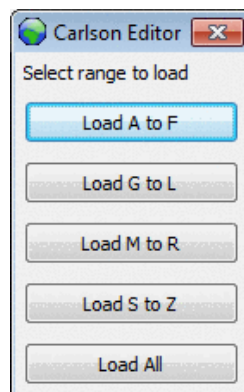
Carlson has customized and added a new Menu "Carlson" to the POEdit. If the Carlson customized EXE is installed user will be able to use this menu.



The Carlson menu consists of following commands.

1. Editor

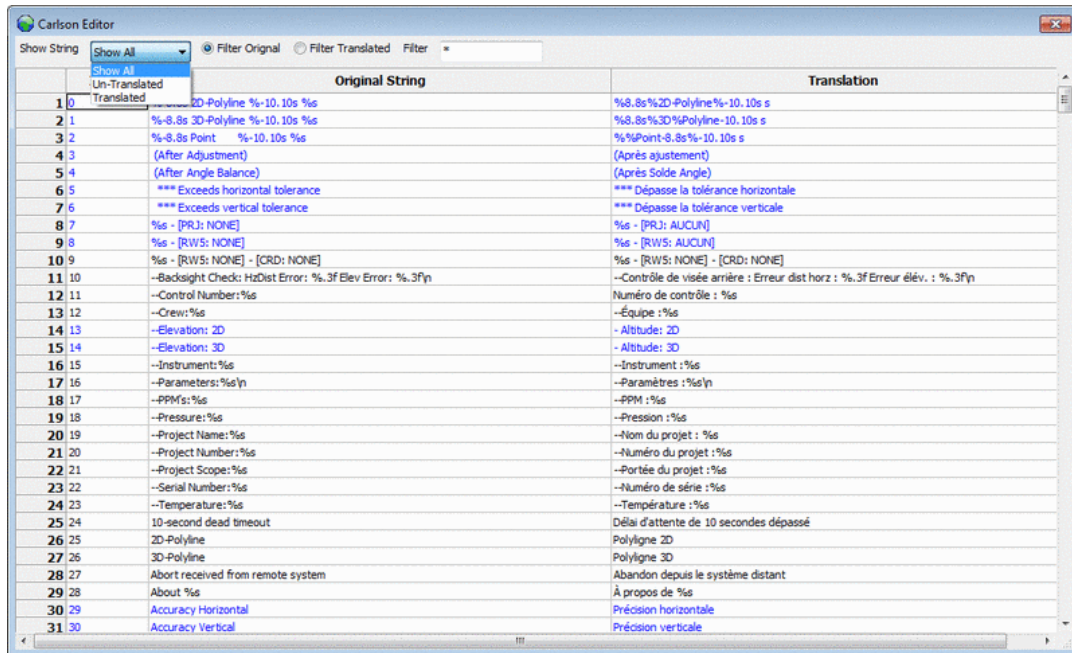
This command let user to do the translation in a spread sheet styled dialog. The PO File is divided into separate groups or ranges for the purpose reducing the amount of string user sees when doing the translation. The Carlson Editor Range dialog allows user to select the alphabetical range of strings.



The selected range is then loaded in a user friendly spread like dialog for better editing. This also provides some filtering tools to the user to be able to focus on the strings of interest. User can select to see the all, un-translated or

translated string. Also filters the original or translated strings based on the filter key words.

An un-translated string can be auto-translated using Google translation, right click the string to be translated and select Auto-Translation to fetch the translation from Google. This will work if the CURL binary files are present to the BIN\CURL folder in POEdit installation directory.



2. Auto-Translate Using Google

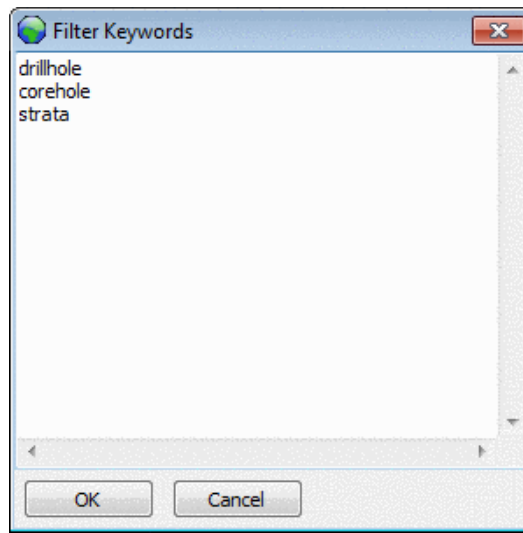
Un-translated strings can be auto-translated using Google translation using this command. This will work if the CURL binary files are present to the BIN\CURL folder in POEdit installation directory. This command does not translate strings that contain special characters used in C format i.e. string with '%' or '\\' characters.

3. Read TLT File

This command can be used to get the translations from TLT Files that already have translations, user is prompted to select a TLT file and the matching string will auto-translated if a translation was found.

4. Define Filter Keywords

This command is used for defining the filter keywords for the strings that user does not want to translate. For example translating only the Carlson survey strings user can choose to remove all the strings that contains mining words like drillhole, corehole, strata etc.

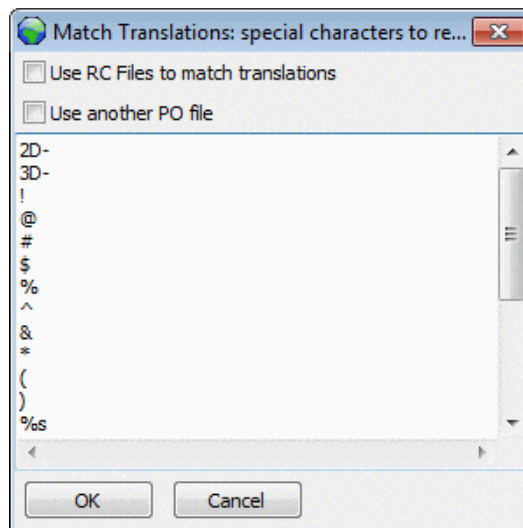


5. Translate using resource (RC) files

This command allows user to select SurvCE English and translated resource files and translates the matching strings between PO and English RC file if a translation was found in Translated RC file.

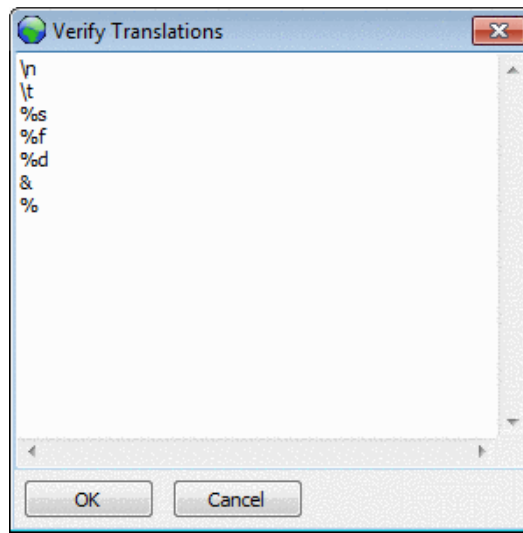
6. Find Matching Translations

This command can be used to search for matching translation in resource (RC) files and another PO File or currently loaded PO file by removing the defined special characters. For example if a string "OK" is already been translated it can look for all the strings like "&OK" etc and set the translation.



7. Verify Translation

This command can be used to verify if the equal number of special characters is present in the translated strings as there is in original string, to avoid user error in C format strings. For example if "OK\tCancel" is translated to "Acceptar\Cancele" this will prompt for user to correct the translation.



Patch Management

After the initial release of a major version, there are typically a few software service patches to fix errors. These service patches are delivered as update installs that contain only the updated program files. These update installs are not the full install and they designed to be installed onto an existing installation.

Check Patch Status

To see the software build that you are currently running, go to the Help->About Carlson command and look at the build number which is the date of the build. This number has six digits where the first two are the year, the next two are the month and the last two are the day. For example, a build on July 4, 2021 would have a build number of 210704.

Patch Notification

There are two ways to be notified of new patches. One way is that the program is setup by default to check the Carlson server once a week for new patches. This check is done when the Carlson program is started. If there is no update, the check routine displays a dialog reporting that the check didn't find a new patch. If there is a new patch, then the routine prompts for whether to install the update. If you choose to install the patch, the routine downloads the patch and steps through the install. You have the option to turn off the checking in the check status dialog by checking the "Never check again" toggle. If the checking is turned off, it can be turned on again by running Windows->All Programs->Carlson Software->Check For Updates.

The other method to be notified of new patches is to go to the Carlson website, www.carlsonsw.com, and go to Support->Downloads. Then select the product name from the product list. Then on the download page, there is a field for entering your email address for getting an email notification when updates are posted. This way you get notified immediately when an update is ready.

Manually Installing a Patch

There are two ways to manually install a patch. One is to run the Check For Updates function under the Windows->Programs->Carlson Software menu. This routine checks the Carlson server for the latest build version and

compares to your build version. If the server has a more recent build, the program downloads the patch and steps through the install.

The second method is to go to the Carlson website, www.carlsonsw.com, and go to Support->Downloads and select the software from the list of products. Then there is a list of the full install and the latest patch. The you can download and run the patch.exe.

To install the patch, you need Windows write permission to the program folder.

Managing Patches Centrally

Depending on your company administration policies, three different approaches to the central patch management can be considered:

1. Portion of the application is installed centrally, on the server, so that the files can be updated in the single central location instead of doing it on every workstation
2. Local patch downloads are disabled and instead a patch is downloaded to central location and launched on every workstation through the script
3. Local patch downloads are enforced, even if user chose to disable them manually.

Method 1: Partial central install on the server

This method involves running usual installation of Carlson product on the server as if it is a regular workstation. The installation path should be a folder which is accessible through the network share.

Next the product is installed on the workstations using the "Remote install" option:

User will then get a prompt to select the location where the program is installed on the server:

This will result in a smaller local installation with vital updatable files located centrally.

The update process is performed on the server by either installing patch manually or running "Check for updates" from Programs menu.

Methods 2 & 3: Using Windows mechanisms to control local installation of the patches by the end-users

The setting controlling whether the local install on the user workstation is automatically checking for updates is located at the following location in registry (Carlson 2024 on IntelliCAD 11.1) :

```
HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\Carlson Software\SurvCADD\2024\ICAD111
```

The value name is NONET and value "1" (string) disables automatic update. Value of "0" or no value allows automatic updates.

Domain administrator can configure these settings to be changed using group policies on the domain server:

- Run "Group Policy Management" tool located in Control Panel, Administrative tools
- Navigate to your domain, Group Policy Objects
- Create new policy by right-clicking and selecting "New"
- Give policy a fitting name
- Right-click on policy and select Edit
- Right-click on Startup, select properties
- Click on Show File and copy your script (see below) into the folder which opens
- Click "Add" and select the script you copied

Other methods for running scripts on startup or login will work as well, as long as they are configured to run with administrative rights.

Save the following snippets into batch file (.bat or .cmd) and place them where your launching setup is looking for it.

Patches are delivered using InstallShield PackageForWeb tool. These packages can be installed interactively by user as long as they are launched by a script with Administrative rights. User interaction with patching process can be controlled in the following manner.

To minimize interaction, but still provide user with information on patching progress, please use the following switches:

```
Patch_file.exe -s -a minimal
```

To avoid any interaction, resulting in unconditional delivery of the patch as needed, please use the following switches:

```
Patch_file.exe -s -a stealth
```

License Agreement

Copyright 1992-2023 Carlson Software All Rights Reserved

CAUTION! READ THIS NOTICE BEFORE USING SOFTWARE

Please read the following Software License Agreement before using the SOFTWARE. Using this SOFTWARE indicates that you have accepted its terms and conditions.

Carlson 2024

END-USER LICENSE AGREEMENT FOR CARLSON SOFTWARE

IMPORTANT-READ CAREFULLY: This Carlson Software End-User License Agreement ("EULA") is a legal agreement between you (either an individual or a single entity) and Carlson Software, Inc for the software accompanying this EULA, which includes computer software and may include associated media, printed materials, and "online" or electronic documentation ("SOFTWARE PRODUCT" or "SOFTWARE"). By exercising your rights to use the SOFTWARE, you agree to be bound by the terms of this EULA. If you do not agree to the terms and conditions of this EULA, you may not use the SOFTWARE. IF YOU DO NOT AGREE TO THE TERMS AND CONDITIONS OF THIS EULA, DO NOT INSTALL OR USE ANY PART OF THE SOFTWARE.

Carlson Software, Inc., referred to as "LICENSOR", develops and/or licenses proprietary computer programs and sells use licenses for such proprietary computer programs together with or apart from accompanying copyrighted material and documentation and;

End User desires to obtain the benefits thereof and, in return for which, is willing to abide by the obligations and fee agreements applicable to LICENSOR's use licenses in LICENSOR's proprietary computer programs.

For good and valuable consideration, including but not limited to license grant in accordance with this Agreement by LICENSOR to End User's covenant regarding LICENSOR's proprietary rights, LICENSOR agrees to permit End User to utilize materials representing LICENSOR's product or products subject to the following terms and conditions:

1. License Grant: Subject to the terms, conditions and limitations of this EULA, LICENSOR hereby grants End User a personal, limited, non-exclusive, non-transferable, license to utilize the Software Product you have purchased. The license granted in this EULA creates no license, express or implied, to any other intellectual property of Licensor, except for the specific Software Product which they have lawfully purchased from LICENSOR.

This EULA grants you the following rights: You may install and use one copy of the SOFTWARE PRODUCT, or any prior version for the same operating system, on a single computer. The primary user of the computer on which the SOFTWARE PRODUCT is installed may make a second copy for his or her exclusive use on a portable computer.

Storage/Network Use. You may also store or install a copy of the SOFTWARE PRODUCT on a storage device, such as a network server, used only to install or run the SOFTWARE PRODUCT on your other computers over an internal network; however, you must acquire and dedicate a license for each separate computer on which the SOFTWARE PRODUCT is installed or run from the storage device. A license for the SOFTWARE PRODUCT may not be shared or used concurrently on different computers.

2. Exclusive Source. End User shall obtain all LICENSOR authorized product materials through LICENSOR or LICENSOR'S authorized representative and no other source. LICENSOR authorized product materials include, but are not limited to, manuals, license agreements and media upon which LICENSOR's proprietary computer programs are recorded. End User shall make no copies of any kind of any of the materials furnished by LICENSOR or LICENSOR's authorized representative, except as specifically authorized to do so in this EULA. End User is not entitled to make archival copies of those portions of LICENSOR's product(s) that are provided on a machine readable media.

3. Proprietary Rights of Licensor. End User agrees that LICENSOR retains exclusive ownership of the trademarks and service marks represented by its company name and logo and all of the documentation and computer recorded data related thereto. End User also agrees that all techniques, algorithms, and processes contained in LICENSOR's computer program products or any modification or extraction thereof constitute TRADE SECRETS OF LICENSOR and will be safeguarded by End User, but in no event shall End User exercise less than due diligence and care in accordance with the laws of the country of purchase and International Law, whichever operates to best protect the interests of LICENSOR. End User shall not copy, reproduce, re-manufacture or in any way duplicate all or any part of LICENSOR products WHETHER MODIFIED OR TRANSLATED INTO ANOTHER LANGUAGE OR NOT, or in any documentation, or in any other material provided by LICENSOR in association with LICENSOR's computer program products regardless of what manner of storage and retrieval the product exists, except as specified in this Agreement and in accordance with the terms and conditions of this Agreement which remain in force. End User agrees that in the event End User breaches this EULA, End User will be liable for damages as may be determined by a court of competent jurisdiction.

4. Restrictions. End User's rights and obligations under this EULA are nonexclusive and personal in nature, and the intellectual property Licensor grants to End User is subject to applicable law other than bankruptcy law. End User may not transfer or assign the SOFTWARE, rights under this EULA or accompanying user documentation, or any updates of the SOFTWARE which may be provided under this EULA, to a third party unless End User receives written consent from Licensor at least 30 days prior to the completion of transfer. Licensor reserves the right to deny transfer or assignment if, in its sole discretion, Licensor determines the transfer not to be a necessity. Whether or not a transfer or assignment is allowed shall be determined in Licensor's sole discretion after taking into consideration certain factors to find the existence of a necessity including, but not limited to, merger or acquisition of an entity, complete asset acquisition, change of control, severe economic hardship, severe loss of human resources or significant loss in business divisions, or winding down of entity affairs.

If Carlson consents to a transfer, such transfer shall be allowed only as a one-time permanent transfer of this EULA and Software to another end user, provided the initial End User retains no copies or previous versions of the Software. The transfer must include all of the Software, including all component parts, any media and printed materials, any upgrades, this EULA, and any associated license key. The transfer may not be an indirect transfer, such as a consignment, rental or lease. No corresponding Maintenance Agreement rights shall transfer with the SOFTWARE transfer to the subsequent end user. Prior to the transfer, the subsequent end user receiving the Software from the initial End User must agree to all terms of this EULA, with the added condition that no further

transfers to third parties are permitted for any reason whatsoever, and shall agree to the terms and conditions of a new Maintenance Agreement with Licensor.

You may not reverse engineer, decompile, or disassemble the SOFTWARE or alter the images utilized in the SOFTWARE and user documentation. The SOFTWARE PRODUCT is licensed as a single product. Its component parts may not be separated for use on more than one computer. You shall communicate to any individual user in your facility that they are bound by the restrictions of this license agreement may not copy or alter the SOFTWARE for use outside End User's facilities.

Upgrades. If you purchase an upgrade of a SOFTWARE PRODUCT and you use it on different machine from one where upgraded SOFTWARE PRODUCT was used, use of original SOFTWARE PRODUCT must be discontinued and confirmed within 30 days. If such use is not discontinued, it is a material breach of this EULA and LICENSOR shall be entitled to all remedies available to it under this EULA, and under the laws of Kentucky, USA.

5. Security Mechanisms. Licensor and its affiliated companies take all legal steps to eliminate piracy of their software products. In this context, the Software Product may include a security mechanism that can detect the installation or use of illegal copies of the Software Product, and collect and transmit data about those illegal copies. Data collected will not include any customer data created with the Software. By using the Software Product, you consent to such detection and collection of data, as well as its transmission and use if an illegal copy is detected. Licensor also reserves the right to use a hardware lock device, license administration software, and/or a license authorization key to control access to the Software. You may not take any steps to avoid or defeat the purpose of any such measures. Use of any Software without any required lock device or authorization key provided by Licensor is prohibited.

6. Audit Rights. End User agrees that LICENSOR has the right to require an audit (electronic or otherwise) of the LICENSOR Materials and the Installation thereof and access thereto. As part of any such audit, LICENSOR or its authorized representative will have the right, on fifteen (15) days' prior notice to End User, to inspect End User's records, systems and facilities, including machine IDs, serial numbers and related information, to verify that the use of any and all LICENSOR Materials is in conformance with this Agreement. End User will provide full cooperation to enable any such audit. If LICENSOR determines that End User's use is not in conformity with this EULA, End User will obtain immediately and pay for a valid license to bring End User's use into compliance with this EULA and other applicable terms and pay the reasonable costs of the audit. In addition to such payment rights, LICENSOR reserves the right to seek any other remedies available at law or in equity, whether under this Agreement or otherwise.

7. Warranty. THE PRODUCT IS PROVIDED "AS IS" WITH ALL FAULTS. TO THE MAXIMUM EXTENT PERMITTED BY LAW, LICENSOR HEREBY DISCLAIMS ALL WARRANTIES, WHETHER EXPRESS OR IMPLIED, INCLUDING WITHOUT LIMITATION IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, AND WARRANTIES THAT THE PRODUCT IS FREE OF DEFECTS AND NON-INFRINGEMENT, WITH REGARD TO THE SOFTWARE, AND THE ACCOMPANYING WRITTEN MATERIALS. YOU BEAR ENTIRE RISK AS TO SELECTING THE PRODUCT FOR YOUR PURPOSES AND AS TO THE QUALITY AND PERFORMANCE OF THE PRODUCT. THIS LIMITATION WILL APPLY NOTWITHSTANDING THE FAILURE OF ESSENTIAL PURPOSE OF ANY REMEDY. In any event, LICENSOR will not honor any warranty shown to exist for which inaccurate or incorrect identifying data has been provided to LICENSOR. The product(s) provided are intended for commercial use only and should not be utilized as the sole data source in clinical decisions as to levels of care.

8. LIMITATION OF LIABILITY. EXCEPT AS REQUIRED BY LAW, LICENSOR AND ITS DISTRIBUTORS, DIRECTORS, LICENSORS, CONTRIBUTORS AND AGENTS (COLLECTIVELY, THE "LICENSOR GROUP") WILL NOT BE LIABLE FOR ANY INDIRECT, SPECIAL, INCIDENTAL, CONSEQUENTIAL OR EXEMPLARY DAMAGES ARISING OUT OF OR IN ANY WAY RELATING TO THIS EULA OR THE USE OF OR INABILITY TO USE THE PRODUCT, INCLUDING WITHOUT LIMITATION DAMAGES FOR LOSS OF GOODWILL, WORK STOPPAGE, LOST PROFITS, LOSS OF DATA, AND COMPUTER FAILURE OR MALFUNCTION, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGES AND REGARDLESS

OF THE THEORY (CONTRACT, TORT OR OTHERWISE) UPON WHICH SUCH CLAIM IS BASED. THE LICENSOR GROUP'S COLLECTIVE LIABILITY UNDER THIS AGREEMENT WILL NOT EXCEED THE GREATER OF \$500 (FIVE HUNDRED DOLLARS) AND THE FEES PAID BY YOU UNDER THIS LICENSE (IF ANY).

9. Update Policy. LICENSOR may, from time to time, revise the performance of its product(s) and in doing so, incur NO obligation to furnish such revisions to any End User nor shall it warrant or guarantee that any revision to the SOFTWARE will perform as expected by the End User on End User's equipment. At LICENSOR's option, LICENSOR may provide such revisions to the End User.

10. Customer Service. Although it is the LICENSOR's customary practice to provide reasonable assistance and support in the use of its products to its customers, LICENSOR shall not be obligated to any End User to provide technical assistance or support through this Agreement and may at LICENSOR's sole election charge a fee for customer support.

11. Termination of End User License. If any one or more of the provisions of this Agreement is breached, the license granted by this Agreement is hereby terminated. In the event of such termination, all rights of the LICENSOR shall remain in force and effect. Any protected health information data of End User maintained on LICENSOR'S data base shall upon reasonable notice to End User and at the discretion of LICENSOR may be destroyed.

12. Copyright. The SOFTWARE (including, but not limited to, any images, photographs, animations, video, audio, music and or text incorporated into the SOFTWARE), and all intellectual property rights associated with it, whether exists in a tangible media or in an electronic image media is owned by LICENSOR and is protected by United States copyright laws and international treaty provisions and all other commonwealth or national laws. LICENSOR reserves all intellectual property rights in the Products, except for the rights expressly granted in this Agreement. You may not remove or alter any trademark, logo, copyright or other proprietary notice in or on the Product. This license does not grant you any right to use the trademarks, service marks or logos of LICENSOR or its licensors. You may not copy any user documentation accompanying the SOFTWARE.

13. Injunctive Relief. It is understood and agreed that, notwithstanding any other provision of this Agreement, LICENSOR has the unequivocal right to obtain timely injunctive relief to protect the proprietary rights of LICENSOR.

14. Entire Agreement. This EULA constitutes the entire agreement between the parties and supersedes any prior agreements. This EULA may only be changed by mutual written consent.

15. End User Agreement Acknowledgment. The End User hereby accepts all the terms and conditions of this Agreement without exception, deletion, alteration. End User acknowledges they are authorized to enter this agreement on behalf of any organization for which the license is sought. Any unauthorized use of LICENSOR products will be considered a breach of this Agreement, subject to liquidated damages and otherwise unlawful and willful infringement of LICENSOR's trade secrets and/or proprietary products.

16. Payment and Refund Policy. The use of the SOFTWARE herein is deemed a commercial use and under the terms of this license agreement End User shall not be entitled to any refund of purchase price. End User agrees to pay all user fees promptly. LICENSOR is authorized by End User to suspend any further access to SOFTWARE in the event fees are not fully paid. End user entity shall promptly pay any and all access and use charges incurred regardless of the end user. End user is responsible for protecting any pass word and user identity supplied to End User.

17. Loss/Theft/Misuse. End user shall promptly report to LICENSOR the theft or other loss of any password and/or user identity required to access SOFTWARE. LICENSOR shall not be responsible for maintaining the integrity of End User data in the event that end user's data base is accessed and/or altered by an unauthorized end user due to the failure of licensed End User to protect its password or user identity. End User shall be responsible for any costs incurred by LICENSOR due to the negligence or reckless disregard of End User's failure to protect its

password or user identity.

18. Civil/Criminal Investigation. End user shall fully cooperate with LICENSOR and or any person authorized by LICENSOR (including local, state, or federal law enforcement officials) to investigate any alleged theft, misuse or unauthorized use of SOFTWARE or data related thereto.

19. U.S. Government Restricted Rights. The SOFTWARE PRODUCT and documentation are provided with RESTRICTED RIGHTS. Use, duplication, or disclosure by the Government is subject to restrictions as set forth in subparagraph (b)(1)(ii) and (c) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 or subparagraphs (c)(1) and (2) of the Commercial Computer Software-Restricted Rights at 48 CFR 52.227-19, as applicable.

20. Governing Law. This EULA shall be governed and construed in accordance with the laws of the Commonwealth of Kentucky, USA.

General Commands

2

File Menu

All the Carlson module menus share seven of the same pull-down menus which are:

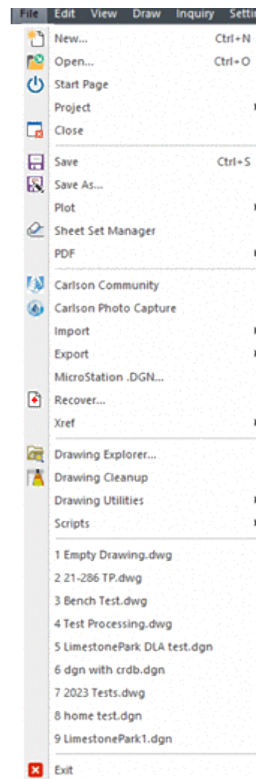
1. File
2. Edit
3. View
4. Draw
5. Inquiry
6. Settings
7. Points

These menus (along with Window and Help) form a "core" group of commands which can be accessed regardless of the Carlson module that is being used. The common pull-down menus contain general commands that are applicable within all the modules. Many of these commands are AutoCAD or IntelliCAD commands which are described in your CAD Reference manual. The Carlson commands located in the common pull-down menus are explained in the next sections.

Note:

- Within a module, other pull-down menus (typically three or four) are often specific to the module but certain commands (*e.g.* Triangulate & Contour) may be found in several modules.

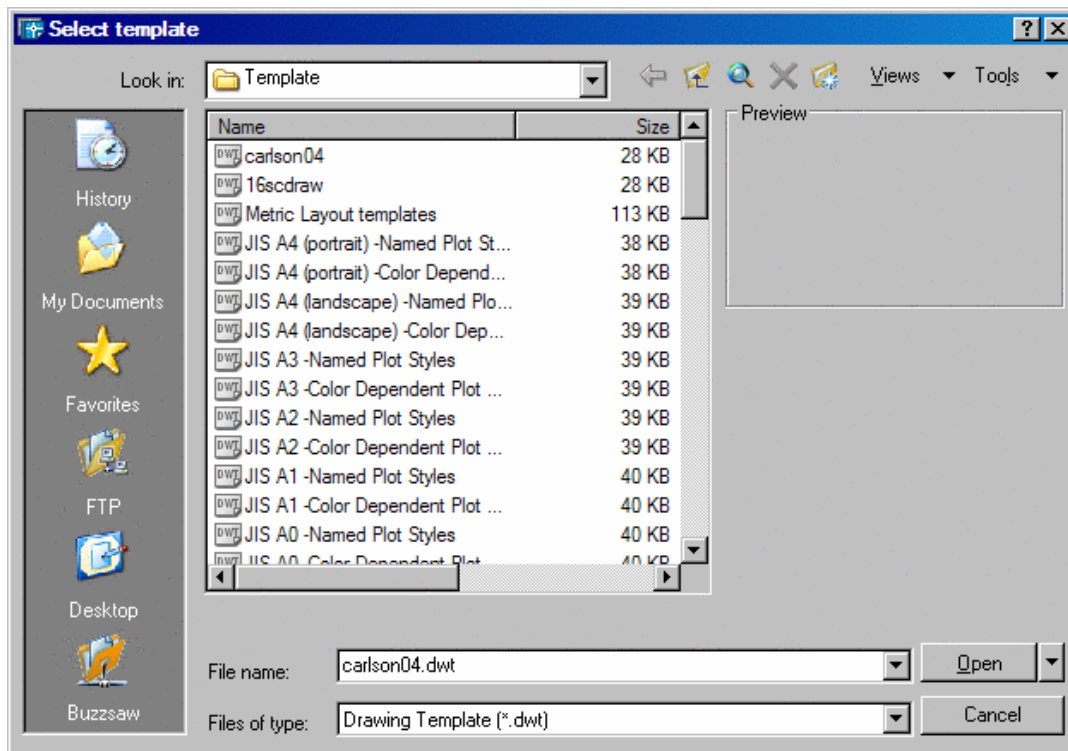
The set of commands varies slightly between the CAD platforms and versions.



New

This command allows you to create a new drawing file. This routine defines the settings for a new drawing. You can start a new drawing file by selecting New, and then picking a template file. The first dialog for the New command, called Select Template, lists all template files that currently exist in the drawing template file location. Choose a file to use as a starting point for your new drawing. A preview image of the selected file is displayed to the right.

The template file (.DWT) that you use will depend on the version of AutoCAD or IntelliCAD that you are running. After choosing the template, click the Open button at the lower-right. Next, you will either see the New Drawing Wizard dialog box, or you will be taken to a blank screen. Should you use the wizard, a new drawing name will need to be chosen in order to get to the next step.



If the wizard is in use, the following options will be available to you in the New Drawing Wizard dialog. The New command starts a new drawing using default settings defined in one of the Carlson .DWT template files, depending on the measurement system you've chosen. You cannot modify the surv.dwt or surviso.dwt templates. To start a new drawing based on a customized template, see Use a Template.



English: This option starts a new drawing based on the Imperial measurement system. The drawing is based on the surv.dwt template, and the default drawing boundary (the drawing limits) is 12 Ã- 9 inches.

Metric: This option starts a new drawing based on the metric measurement system. The drawing is based on the surviso.dwt template, and the default drawing boundary (the drawing limits) is 429 Ã- 297 millimeters.

The New command creates a new drawing, using the settings defined in a template drawing you select. Template drawings store all the settings for a drawing and may also include predefined layers, dimension styles, and views.

Template drawings are distinguished from other drawing files by the .DWT file extension. They are normally kept in the template directory. Several template drawings are included with Carlson. You can make additional template drawings by changing the extensions of drawing file names to .DWT.

Remember that there are two methods that you can use to work on a Carlson drawing. One is the New command, and the other is the more generic Open command. If you need to open an existing drawing, use Open, also found in the File menu, and then choose an existing file name.

Pulldown Menu Location: File

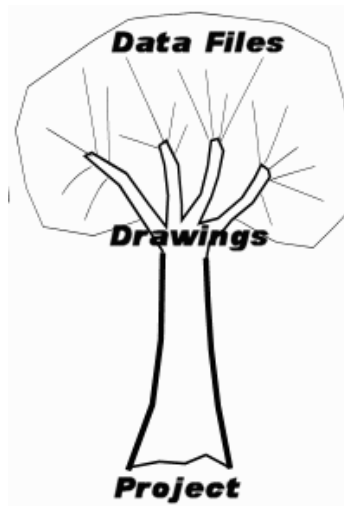
Keyboard Command: new

Prerequisite: None

Project Explorer

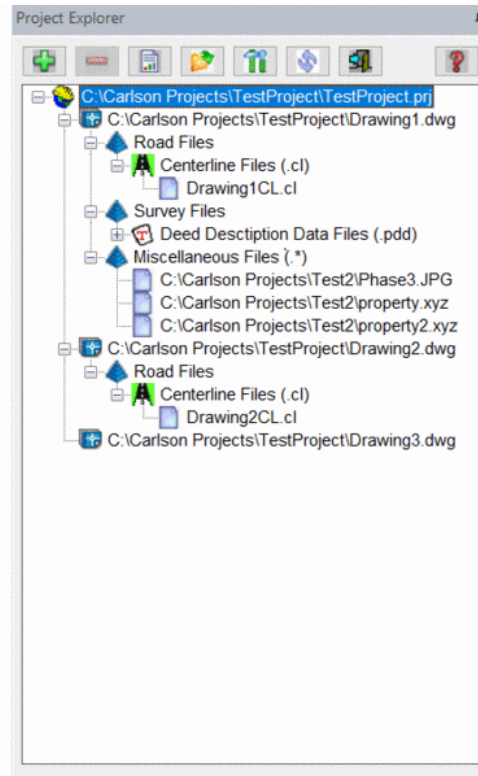
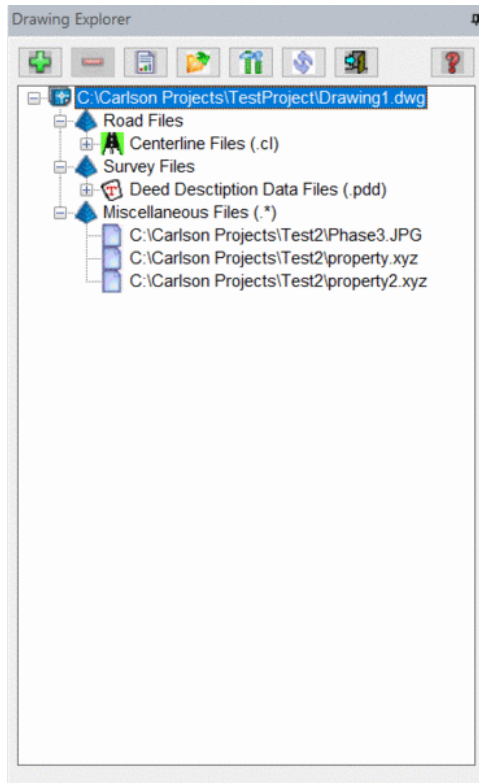
This tool is used for management of a complete project. A project can contain multiple drawings, and each drawing within that project can contain multiple associated data files.

Think of the Project Explorer as the trunk of the hierarchical tree structure that develops into a project as illustrated below:



As you work in a drawing, Carlson keeps track of the files that you create (such as TIN and coordinates files). These are related to the drawing and you can use the Drawing Explorer to manage them. The Project Explorer is used to manage multiple drawings. In the following illustration, two views of the same Project are displayed:

- *By Drawing* - shown on the left.
- *By Project* - shown on the right.



Control



Action

Adds a drawing and its data file(s) into the project.

Removes the selected file(s) along with any "child" (sub-ordinate) files from the project. Note that removing a data category or file at the "project" level also removes the category or file at the "drawing" level.

Creates a report through the Report Formatter Options dialog box. The Report Formatter can be used to move to the *Available* entries on the left to the *Used* area on the right. When coupled with the Up/Down options to control the order, highly customized reports can be generated and saved for subsequent use. Click the Display to obtain the report.

The Change Directory option allows you to instruct Carlson Software to re-direct the location for files from an old folder location to a new folder location. This option is helpful if project data files are manually moved to a new folder location.

The Settings button allows you to create Categories of file types and assign data files to a particular Category and assign how project data files are presented in the Project Explorer.

The Refresh button re-reads the current Project file along with the various Drawing file settings and updates the Project listing appropriately.

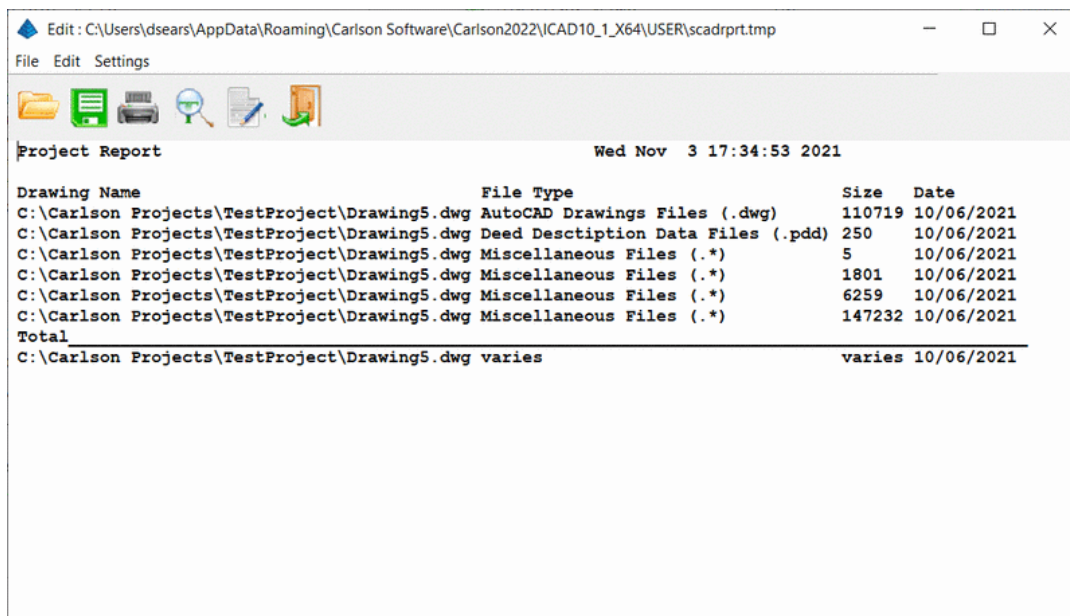
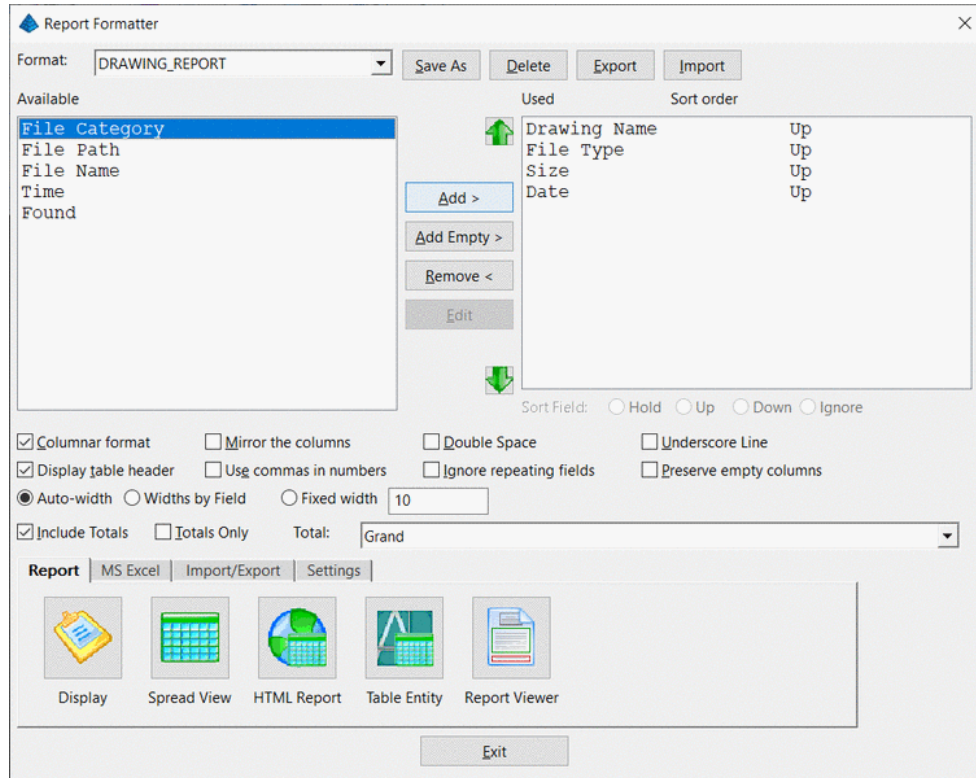
Exits from the Project Explorer command.

Displays the on-line help.

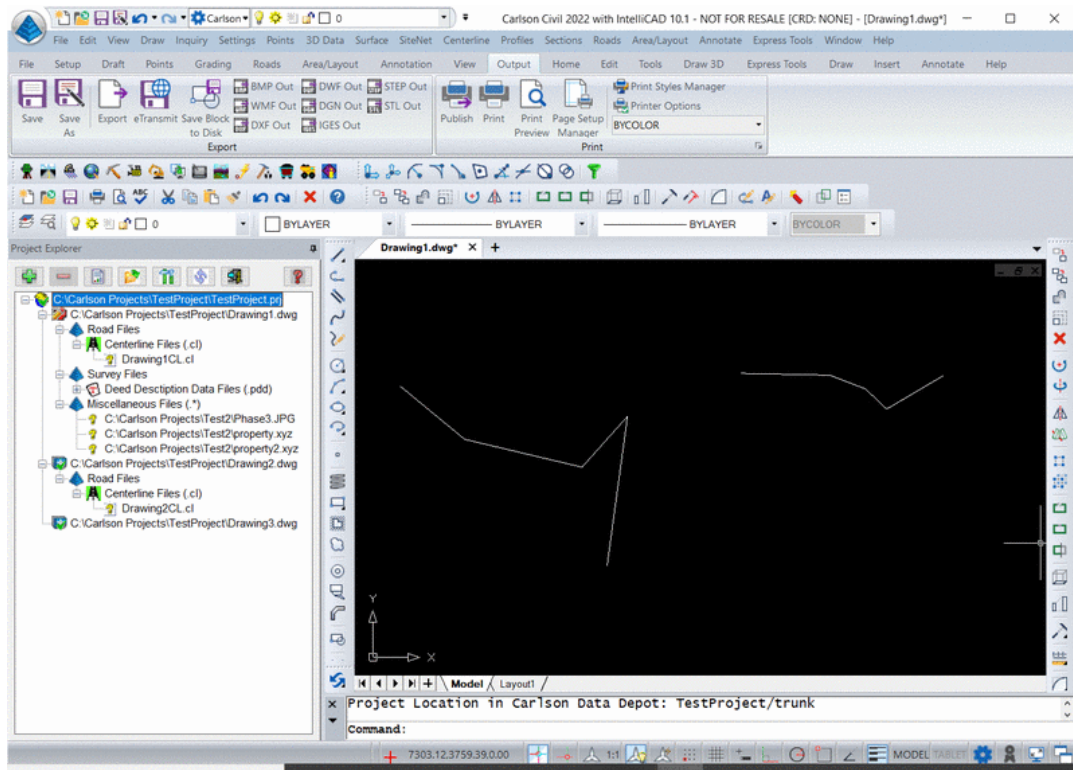
Project Explorer Controls

The project history/log report file can be generated using Report button. The Project Explorer tracks all the data files used with the project. The report function also uses the data file information logged with the Carlson Data

Depot. The program allows you to report all revisions or only the last commit on all project files from Carlson Data Depot. The information reported include project file, drawing name, file category, file path, file name, file type, date, time, size, revision, author, date committed and commit message along with project properties (i.e. company name, project name etc). Here's a sample report.

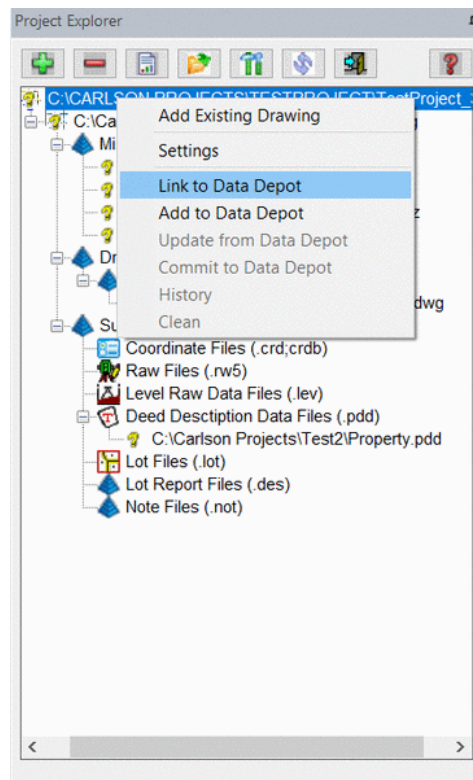


Linking a project to the Carlson Data Depot requires that you properly configure the Data Depot through the Set Project/Data Folders command and then open a drawing and assign it to a Project (.PRJ) file. If a project is not linked to the Data Depot, a question mark is overlaid on all the files and folders under that project as shown:



To link a project to the Data Depot, perform the following steps:

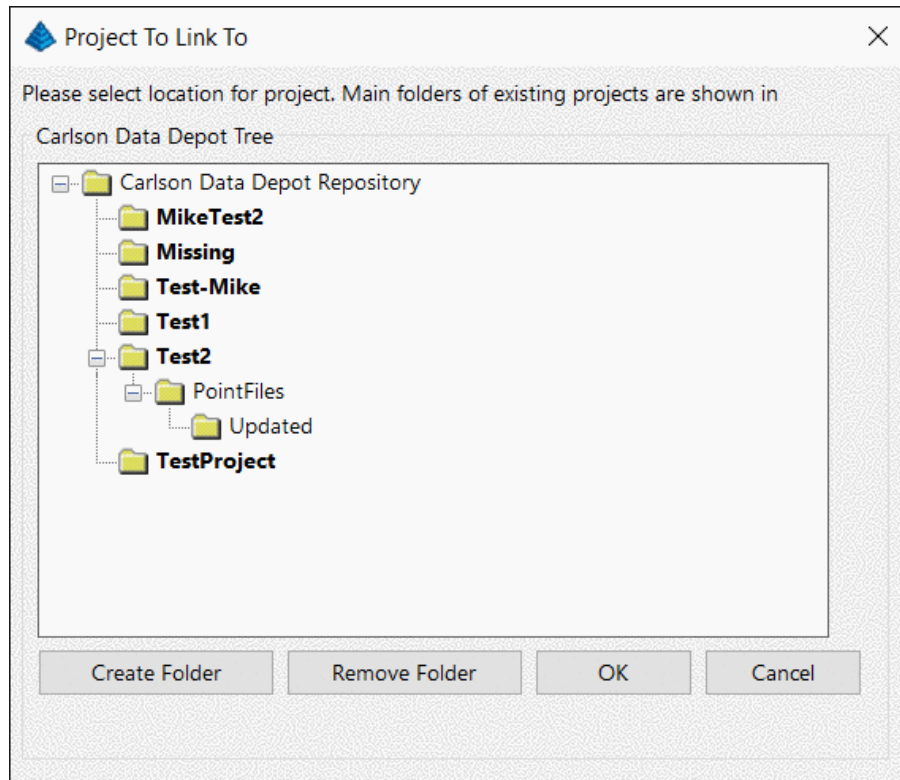
1. Right click on the Project File name (the "trunk" of the Project Explorer tree) and select Link to Data Depot command:



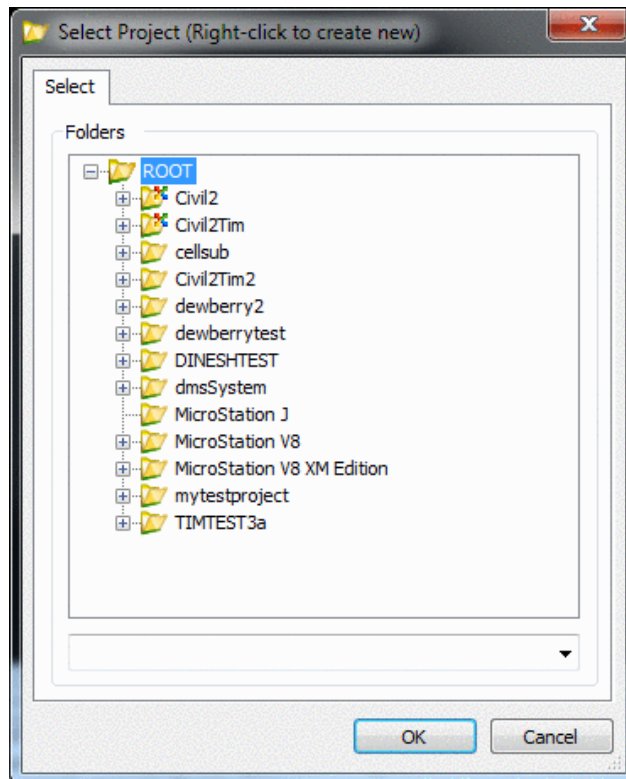
2. This will bring up the "Project to Link to" window. The window you see depends upon whether you are using SVN or ProjectWise as your Data Depot Repository.

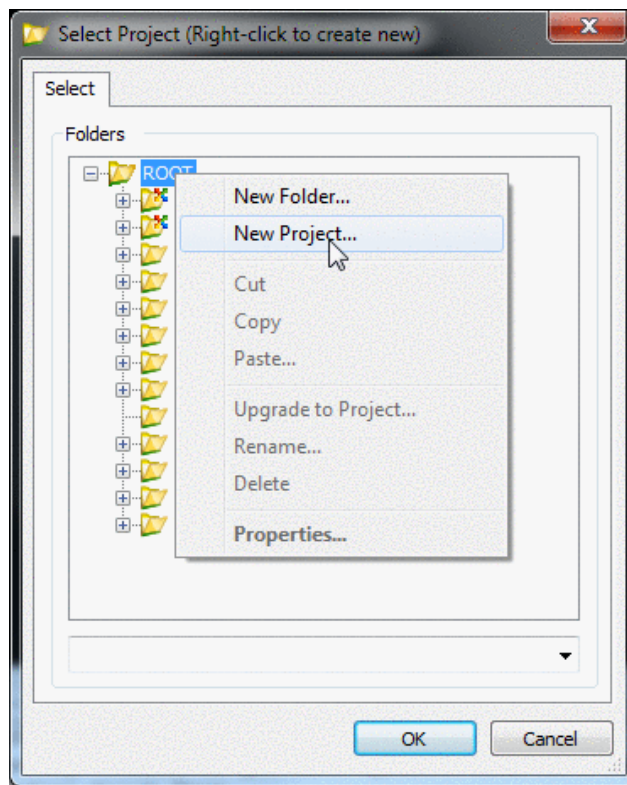
- (a) **SVN:** This window shows the existing projects under the repository, if any. A new project folder can be created under Carlson Data Depot Repository by using the Create Folder button. Sub-folders can be

created by using the Create Folder button while top folder selected. Select the folder you want to create the project under and click OK:

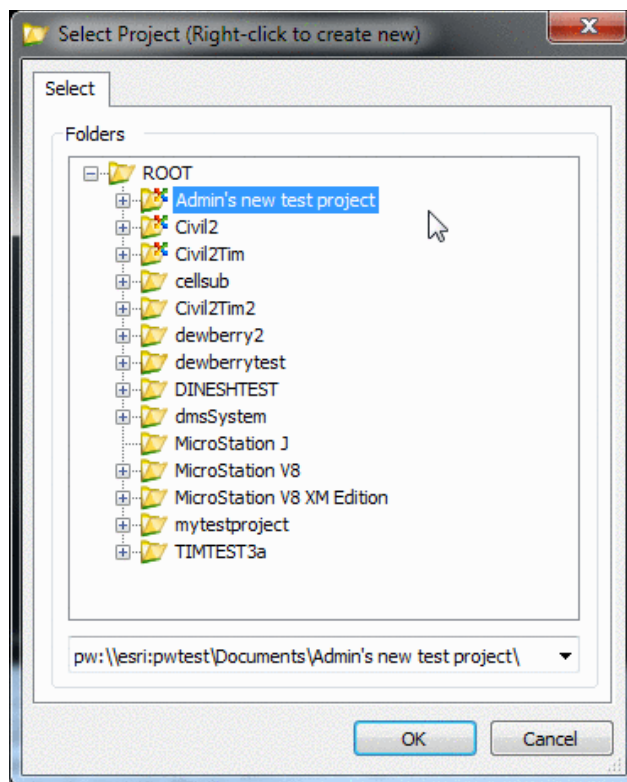


- (b) **ProjectWise - Step 1:** This window shows the existing projects under the repository, if any. A new project folder can be created by right clicking on the ROOT entry of the repository tree:

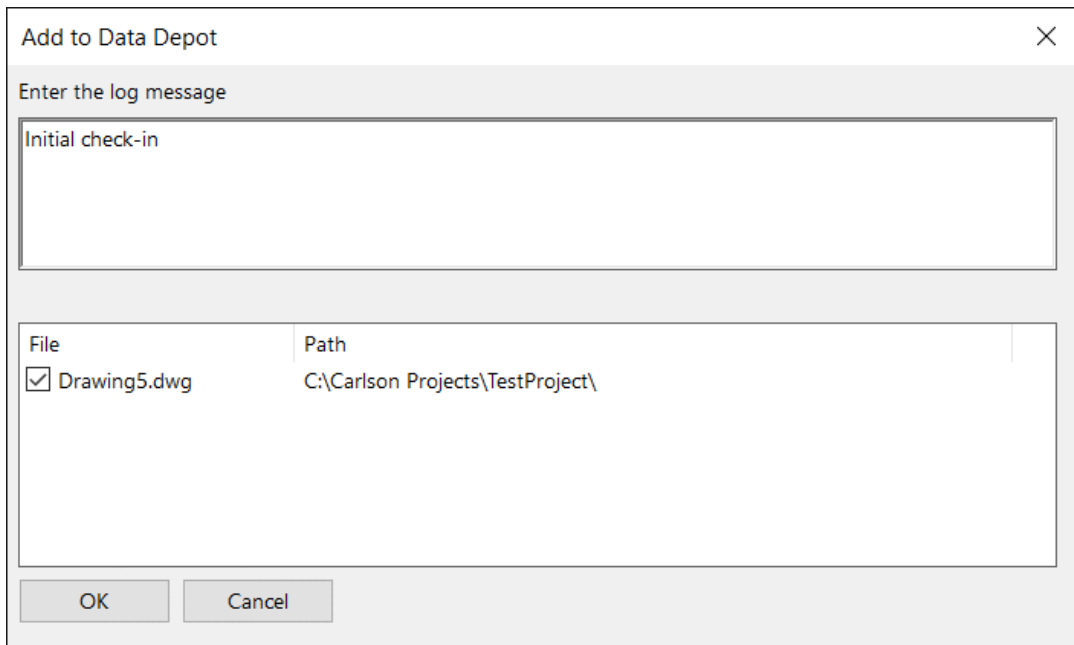




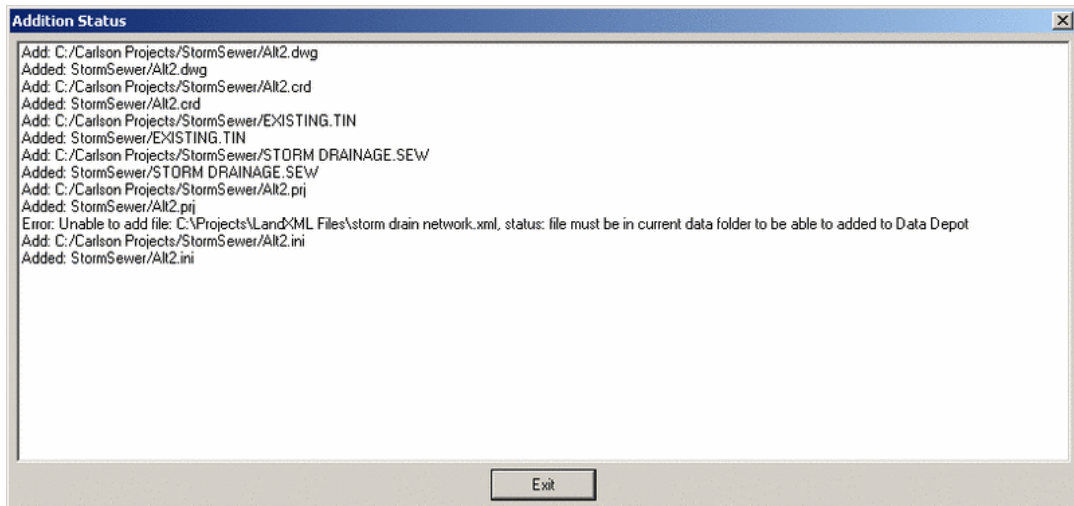
(c) **ProjectWise - Step 2:** Name the new project, select it from the list and click the OK button.



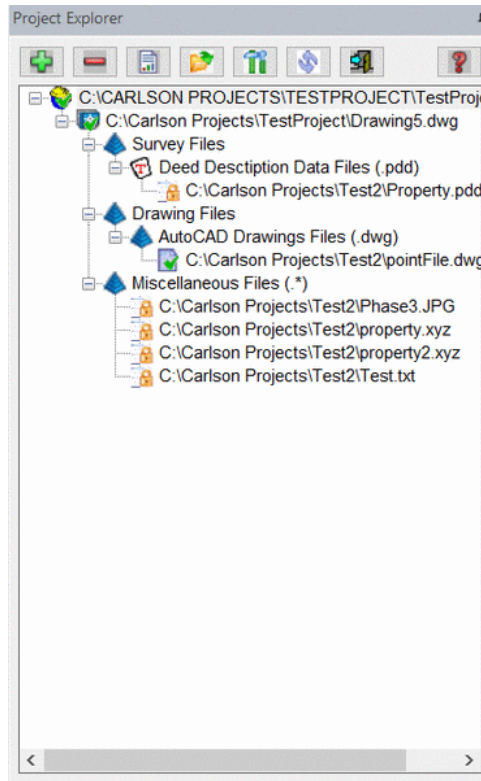
3. This will prompt you to Add to Data Depot window where you can enter a log message to identify what you are doing. This information will be added to the history log. Click OK to start adding files to the project repository:



4. A status window will appear showing the message for files that have been tasked to be added to the repository:



5. Once the files are linked to the Data Depot, the status/icon in the tree view is changed:

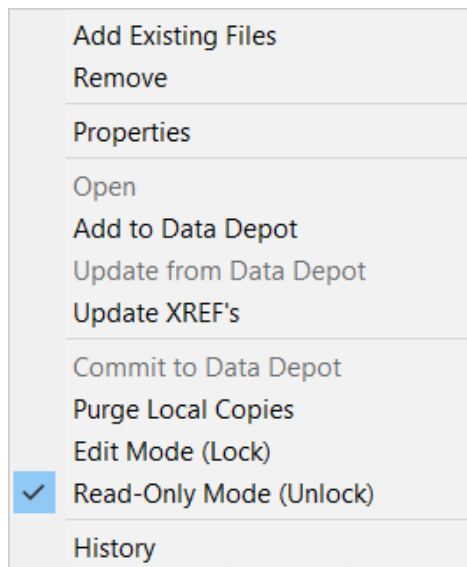


When new files are created inside the project, they are shown as not linked to the Data Depot (using the question mark icon) and can later be linked to the Data Depot by right clicking and using the Add to Data Depot command.

Data Depot File Status

The following shows the icons used in the Project Explorer tree for representing the state of files in the Data Depot:

Data Depot Commands



Add Existing Files: Allows the user to select an existing data file from local storage which will be added to the project and the Data Depot. The data file will be associated (placed under) the currently selected drawing.

Remove: Removes the selected file from the project without removing it from the Data Depot.

Properties: Brings up the system file properties dialog.

Add to Data Depot: Incorporates the selected file(s) into the Data Depot.

Update from Data Depot: Gets the last committed version of the selected file(s) from the Data Depot.

Update XREF's: Gets the last committed version of any XREF files associated with the selected drawing.

Commit to Data Depot: Incorporates locally modified/locked file(s) that are already part of the Data Depot into the Data Depot.

Purge Local Copies: Deletes the selected file from the local storage leaving it in the Data Depot and the project. The local copy can be restored later by using the "Update from Data Depot" command.

Edit Mode (Lock): Locks the selected file(s) for local editing and prevents others from modifying the file(s). If the drawing or data file is newer in the Data Depot than on local storage this command will be disabled.

Read-Only Mode (Unlock): Unlocks the selected file(s) so that they may be edited by others.

History: Displays the history log of changes made to the selected file.

Clean (project level only): "Cleans" the project of items that don't exist in both the Data Depot and local storage.

Once the project has been added to the Data Depot, it can be quickly updated or accessed by other users and be ready for use via the Get Project from Data Depot command:

Notes on locking/unlocking project files when using Subversion repository

Subversion locking of the file is specific to the specific working folder - the file in a particular checked out set of files. Same user working on another copy of file in different folder or on different computer will not own the lock. External subversion clients have additional lock functionality which not exposed through Carlson Data Depot interface for purposes of simplicity. These clients will have options for stealing lock (placing the lock on file already locked at different location or by different user) and forcing lock (unlocking a lock placed at different location or by different user). In a productive collaborative environment such design allows on one hand to prevent accidental editing of the file which is locked by another user and causing a situation when one of the users has to abandon his changes, but on the other it allows to move forward if file is locked accidentally and file needs to be re-locked elsewhere.

In the case when this design is not sufficiently strict, it is possible to override this behavior by implementing server-side hooks which change the default behavior. Here is the example of pre-unlock script for VisualSVN which limits lock overrides to original user or to user called Administrator:

```
@echo off
setlocal
set REPOS=%1
set RPATH=%2
set USER=%3
set SVNLOOK=%PROGRAMFILES%\VisualSVN Server\bin\svnlook.exe
set TMPFILE=%temp%/lockinfo
```

```
rem Creating a temporary file with output of the lock, filtering everything
except owner out
''%SVNLOOK%' lock ''%REPOS%' ''%RPATH%' | find 'Owner' > %TMPFILE%
```

```

rem Parsing the file looking for owner name only
FOR /F 'usebackq tokens=2' %%A IN (%TMPFILE%) DO set LOCK_OWNER=%%A

del %tmpfile%

if '%LOCK_OWNER%'=='' goto GOOD
if '%LOCK_OWNER%'=='%USER%' goto GOOD
rem Only Administrator can break the lock
if 'Administrator'=='%USER%' goto GOOD

echo 'Permission denied! Ask Administrator for assistance' 1>&2
endlocal
exit 1

:GOOD
endlocal
exit 0

```

Simply paste this script into the hook window by going into VisualSVN Server Manager, right-clicking over repository, All Tasks, Manage Hooks, double-click pre-unlock hook.

Pulldown Menu Location: File > Project

Keyboard Command: prjxplore

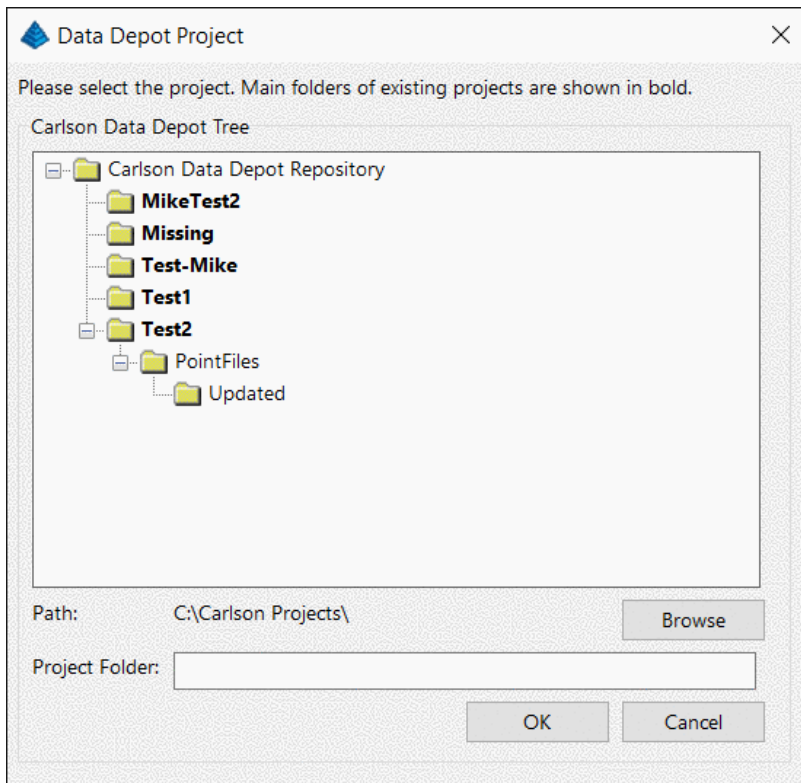
Prerequisite: None

Get Project from Data Depot

The Carlson Data Depot is a document management system to allow tracking of the changing states of files and projects over time and manage the contributions from multiple users providing data integrity, productivity and accountability for the managed products. Once a project has been successfully registered with the Data Depot using the Project Explorer, it can be retrieved through the Data Depot for subsequent editing.

Subversion Example:

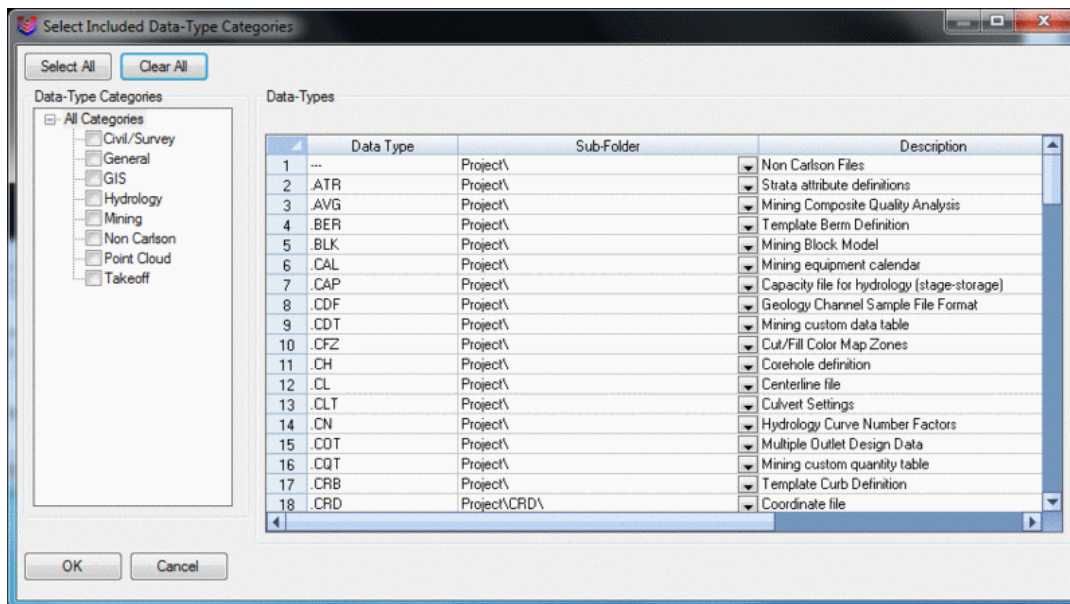
1. Select the project folder from the Repository that you wish to check out:



2. Enter the destination path to check the project out to on your local machine. You can optionally specify a project subfolder to save the project to in the desired path.

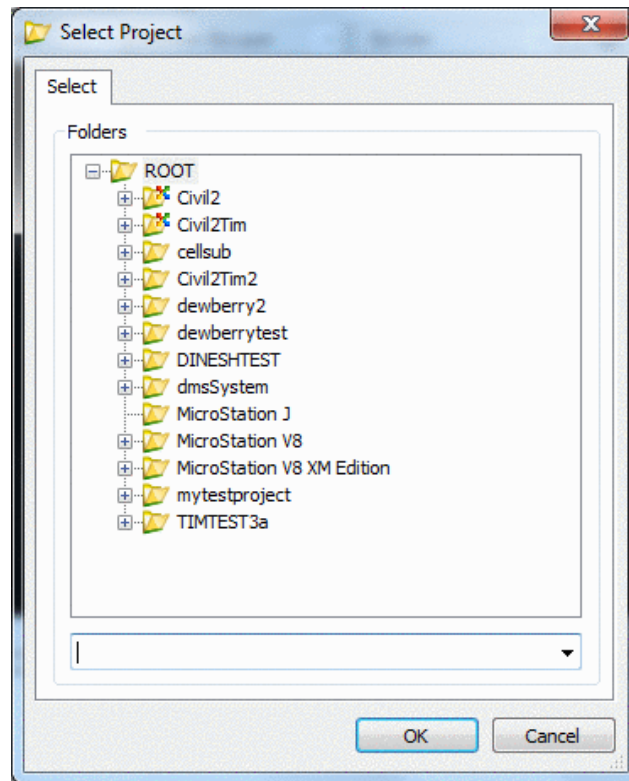
Continue below to the "Select the Project" section.

ProjectWise Example:



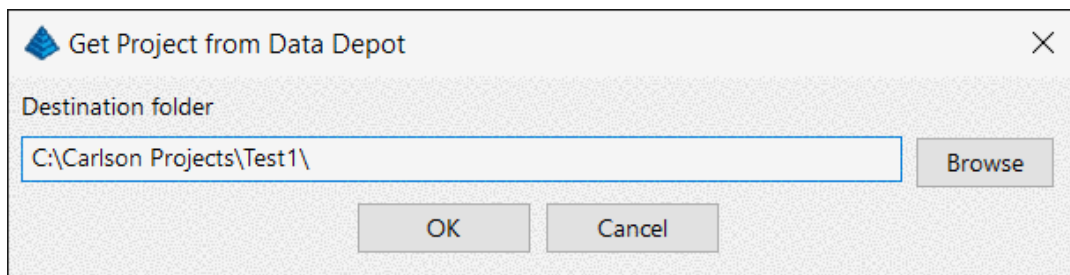
For ProjectWise the "Select Included Data Type Categories" dialog will appear. Here the user can select which types of files to download from the repository. A special category "Non Carlson" is used to determine whether files not associated with Carlson Software should be downloaded or not (e.g. Word .DOC documents or Microstation .DGN

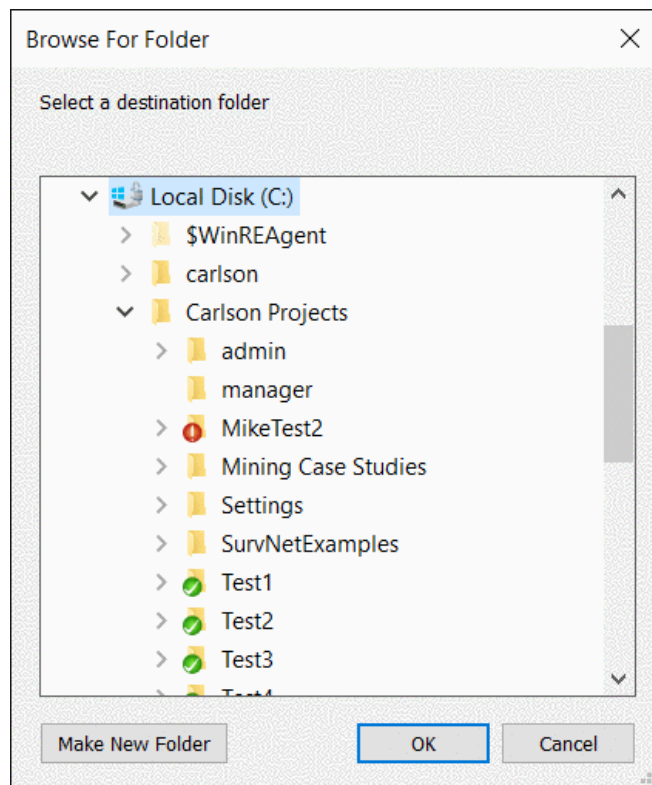
files). If the user wants even more control over what files are downloaded, say at the drawing level, he/she can use the "Clear All" button to deselect all categories. In this case, only the directory structure plus the Project File (.PRJ) will be downloaded. This allows the user to select which drawings they want to download from the repository directly from the Carlson Project Explorer.



Selecting the Project:

Select the project you would like to open and click OK. Unless you have selected "Use Automatic Project Folder Name" (see Set Project/Data Folder for more information), you will be prompted to identify a "working" folder where the checked-out copies of your project files will reside:





Identify the destination folder location and click OK. The checked-out copies of the project files will be placed into the specified folder at which point further editing can begin.

Pulldown Menu Location: File > Project

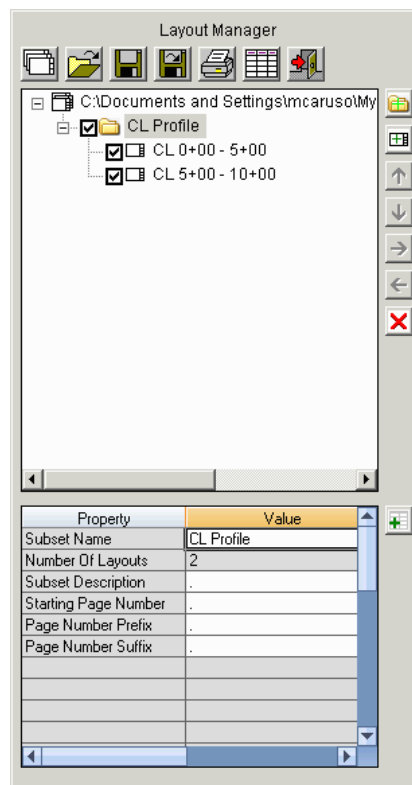
Keyboard Command: get_prj_from_depot

Prerequisite: A properly installed, configured and support content management system

Layout Manager

Note: This command is only available in AutoCAD platform

The Layout Manager is a docked dialog that displays the contents of a Layout Set. Layout Sets (.set files) contain information on layouts that come from a single or multiple drawings. Once you build a layout set, the Layout Manager facilitates printing via AutoCAD's PUBLISH functionality.



The Layout Manager contains two sections. A Treeview, which displays the items of the Layout Set. Items are the root node, which is the topmost node representing the Layout Set. Subsets, which act as folders and can contain Layouts and additional Subsets.

The bottom section contains a grid view, which will display built-in and custom properties of the selected Layout Set, Subset or Layout item. The first column shows the property name, and the second column displays its value. Some properties are read-only and are calculated by the Layout Manager.

New: Use the New button to create a new empty Layout Set. You can also specify an existing layout set to use as a Template. If a Template is specified, only the Subsets (folders) structure will be copied to your new Layout Set. This is convenient if you have a standardized Subset structure to use for all Layout Sets.

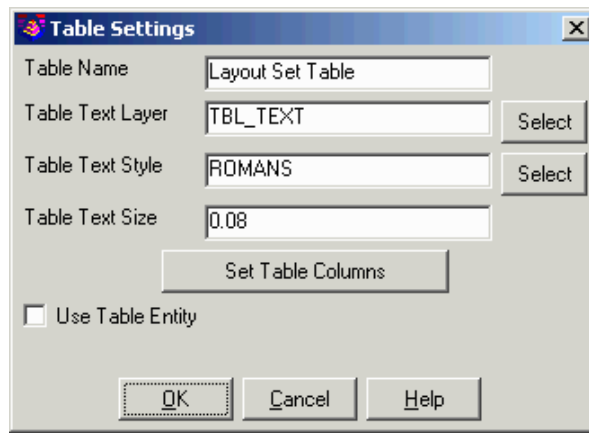
Open: Use the Open button to open an existing Layout Set. Only one Layout Set is allowed open per session. If a Layout Set is currently open in the Layout Manager, you will be prompted to save any changes if necessary.

Save: Use the Save button to save your current Layout Set file. Layout Sets are saved as .set files.

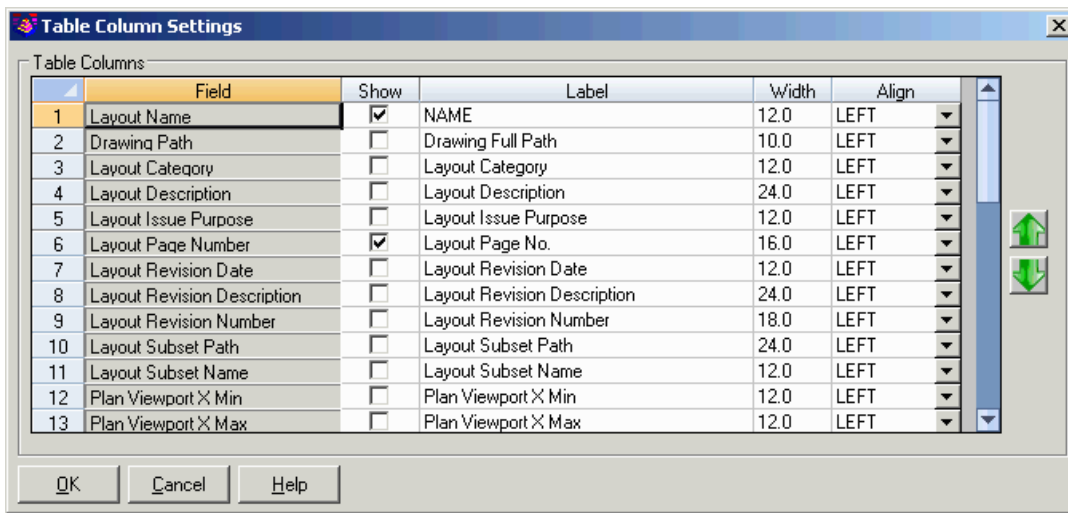
Save As: Use the Save As button to save a copy of an existing Layout Set file with a new name.

Print: Use the Print button print all checked layouts using AutoCAD's Publish functionality. All Subsets and Layouts in the Layout Manager contain a checkbox. If the Subset or Layout is checked, it will be included when the print command is run. When a user checks a Subset, all the child items of that Subset are checked. Similarly, when a Subset is unchecked, all the child items of that Subset are unchecked.

Insert Table: Use the Insert Table button to insert a Table of Contents. This feature allows you to draw a Table containing all the layouts of the Layout Set, including their file name, drawing name, page number, etc.



Use the Set Table Columns button to define which columns are to be displayed in the Layout Set table. Toggle the Use Table Entity to insert the table as a block, or leave it unchecked to draw each row as an individual block.



You can edit the Label column, which will be displayed as the column header in your table. The Width and Text Alignment can also be set for each column.

Exit: Use the Exit button to exit and close the Layout Manager.

Add Subset: Use the Add Subset button to create a new Subset folder under the selected Layout Manger item.

Add Layout: Use the Add Layout button to add a Layout to the current Layout Set file in the selected Subset or root of the Layout Set. Layouts may be selected from the current drawing, or from another drawing on your computer or network.

Move Up: Use the Save As button to move the selected Layout (or Subset and child Layouts) above the previous sibling item.

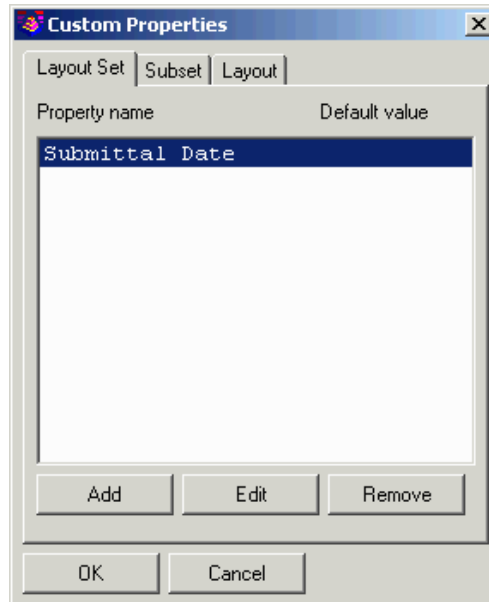
Move Down: Use the Save As button to move the selected Layout (or Subset and child Layouts) below the next sibling Subset or Layout.

Move In: Use the Move In button to move the selected Layout (or Subset and child Layouts) under the next sibling Subset.

Move Out: Use the Move Out button to move the selected Layout (or Subset and child Layouts) above the selected parent Subset.

Remove Item: Use the Remove Item button to remove the selected Layout (or Subset and child Layouts).

Custom Properties: Use the Custom Properties button to Add, Edit or Remove custom properties to a Layout, Subset or Layout Set.



Select the appropriate tab to add a custom property. Then use the Add, Edit or Remover buttons to edit properties for the selected item type.

Pulldown Menu Location: File > Plot

Keyboard Command: layoutmgr

Prerequisite: None

Output Layouts to PDF

This command creates a PDF from layouts in the current drawing. The command starts by prompting for the output PDF file to create. In the dialog, the **Title**, **Subject**, **Keywords** and **Author** are optional properties for the PDF file.

The **Print Area** controls whether to use the page size defined in the Page Setup Manager for each layout, or set the print area by zoom extents on the entities for each layout.

The **Store Layers in PDF** creates a PDF with the CAD layers.

The **Include Off Layers** outputs all layers to the PDF.

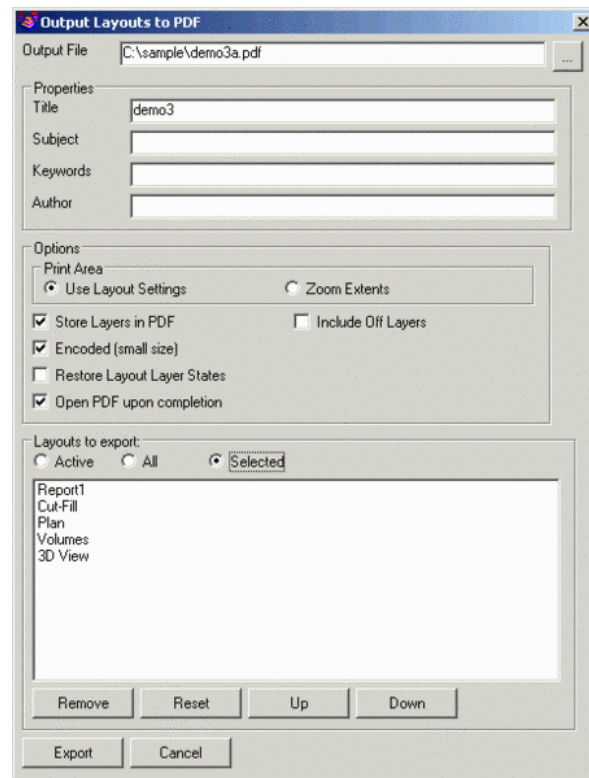
The **Encoded (small size)** option creates a PDF with encoding which should be used unless working with an old Adobe Reader.

Each layout can store the layer state for printing for which layers to thaw and which to freeze. The **Restore Layout Layer States** will set the layer state for the PDF on each layout. This option works together with the Save and Restore Layout Layer commands in the File > PDF menu.

The **Open PDF Upon Completion** option opens the PDF in the viewer for PDFs assigned in Windows which is typically the Adobe Reader.

For **Layouts to Export**, choosing Active will output the current layout to the PDF. The All option outputs all the layouts to the PDF. The Selected option lets you control which layouts to output. When in Selected mode, you can use the Remove button to remove a layout from the output. Use the Up and Down button to change the order of the layouts. The Reset button puts the list of layout back to the default.

Pick the **Export** button to create the PDF.



Pulldown Menu Location: File > PDF

Keyboard Command: pdfexport

Prerequisite: Layouts

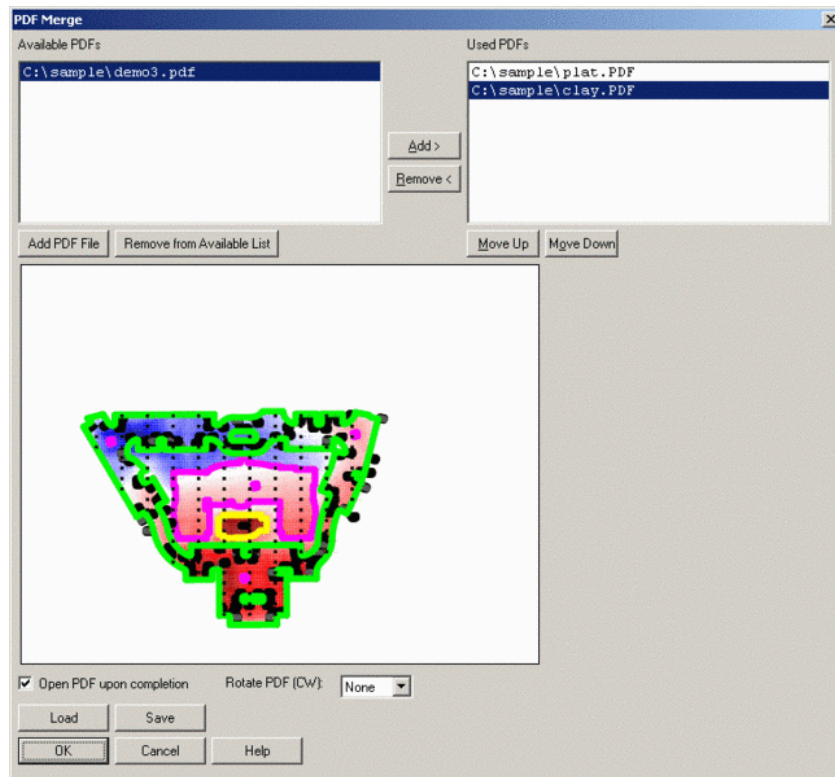
Merge PDF Files

This command combines separate PDF files into a single PDF file. The dialog has a list of available PDF files on the left and there are functions to add or remove PDF files from this list. The list of PDF files on the right and the PDFs to output to the single PDF. Use the Add and Remove buttons to move PDF files between these Available and Used lists. Use the Move Up and Down to change the order of the PDF files for the output PDF. When a PDF is highlighted in the list, a graphic preview is shown.

The Open PDF Upon Completion option opens the PDF in the viewer for PDFs assigned in Windows which is typically the Adobe Reader.

The Rotate PDF CW setting controls how much to rotate the source PDF files for the output PDF.

The Load and Save functions are for storing and recalling a set of PDF files in a .PDQ file.



Pulldown Menu Location: File > PDF

Keyboard Command: pdfmerge

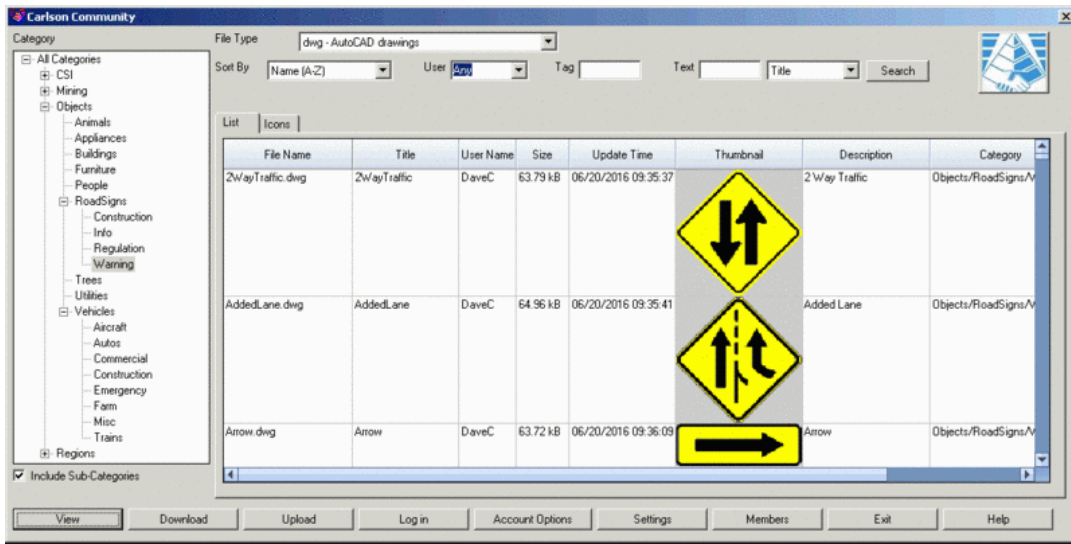
Prerequisite: .pdf files

Carlson Community

This command displays files from Carlson Community where Carlson Software customers can post files to share. This file warehouse is hosted on a Carlson Software server. Types of files on Carlson Community include 3D models for visualization (mdl), point symbol files (dwg), Field-to-Finish code tables for standards in different regions (fld), and road design templates (tpl). To view and download files, you don't need an account log in.

To find a file, first choose the File Type to search for. The files are organized by category and you can browse by category using the category tree on the left. The Include Sub-Categories option shows the files in the current category as well as any sub-categories. The results can be filtered by User, Tag and Text.

There are two views for displaying the file results. The List view shows a spreadsheet. Under the Settings button, you can control the columns to show for the List view. The Icons view shows a series of buttons with images for each file.



To see details about a file, highlight the file and pick the View button, or double-click on the file. The View file dialog shows the file name, title, image, description, tags, category, number of downloads, average rating by users, file size, user name who uploaded the file, date/time of the upload and user comments.



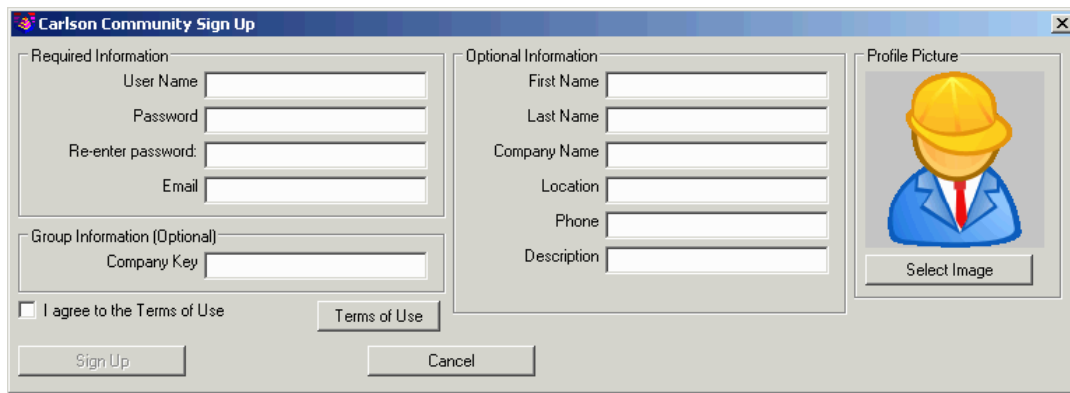
To download the file to your computer, pick the Download button on the View dialog or highlight the file on the main dialog and pick the main dialog Download button. The download routine will then prompt where to save the file on your computer.

To do more on Carlson Community, you need to have an account and log in. To log in, pick the Log In button and then enter either your User Name for Carlson Community or your email and then your password.

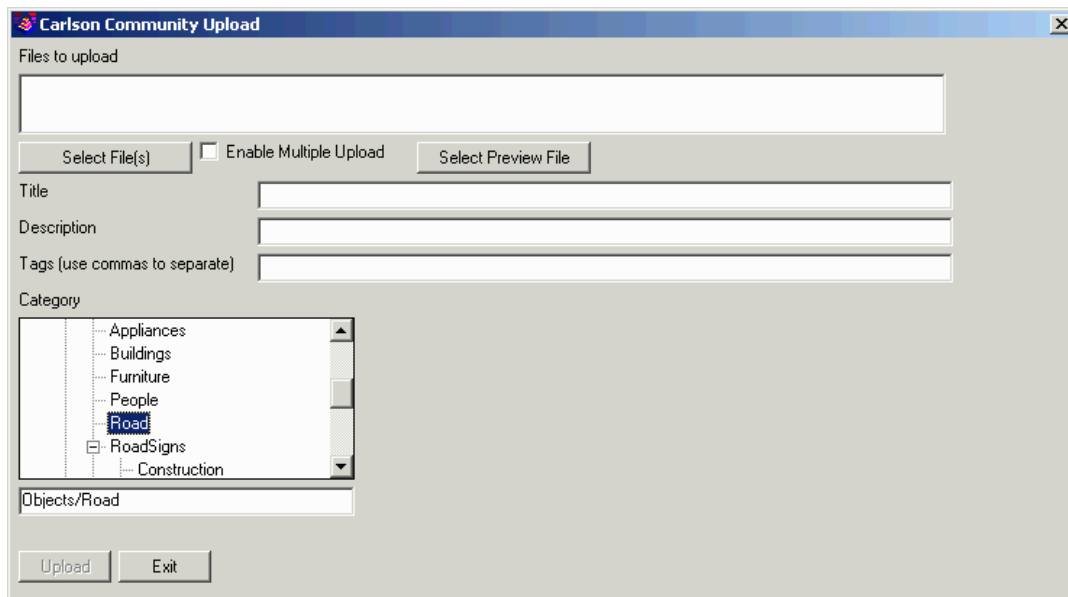


To create a new account, pick the Log In button and then the Account Sign Up button. You need to create a unique User Name for Carlson Community and a password. Your email is also required and needs to be in the Carlson Software records to create a Carlson Community account. If your email is not in the records, then you will receive an email that needs to be confirmed before your Carlson Community account is activated.

Also on the sign up, you must agree to the Terms of Use to create an account. The rest of the information is optional. The Company Key is a way to group users by company which is used by the Carlson Cloud routine for data exchange with SurvCE.



After logging in, you can upload files by picking the Upload button. On the Upload dialog, pick the Select File button and choose the file to upload. Currently the file size limit is 10mb. To upload several files at once, turn on the Enable Multiple Upload toggle before picking Select File. The Select Preview File allows you to pick an image file to use as the preview for your file. Without selecting a preview file, the program will automatically make a preview. The Title, Description and Tags are all optional but helpful ways for users to search for your file. Select a Category for the file from the list or create a new category by entering the category name in the edit box.



Under Account Options, you can update your email, image and other account settings. The List Users shows users by the Company/Group Key name. The Delete Account removes your account from Carlson Community.

The Members routine shows a list of Carlson Community accounts along with activity on uploading.

Files available on Carlson Community include:

MUTCD symbol drawings (dwg): Manual on Uniform Traffic Control Devices from FHWA.

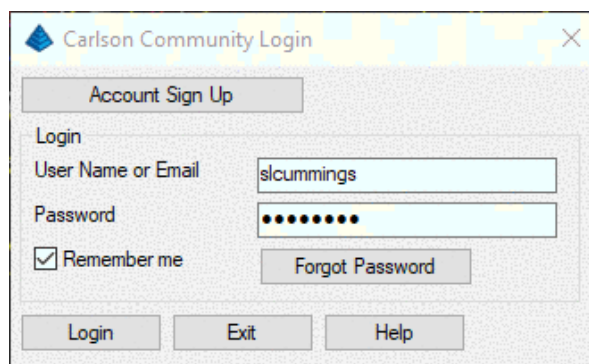
Pulldown Menu Location: File

Keyboard Command: community

Prerequisite: None

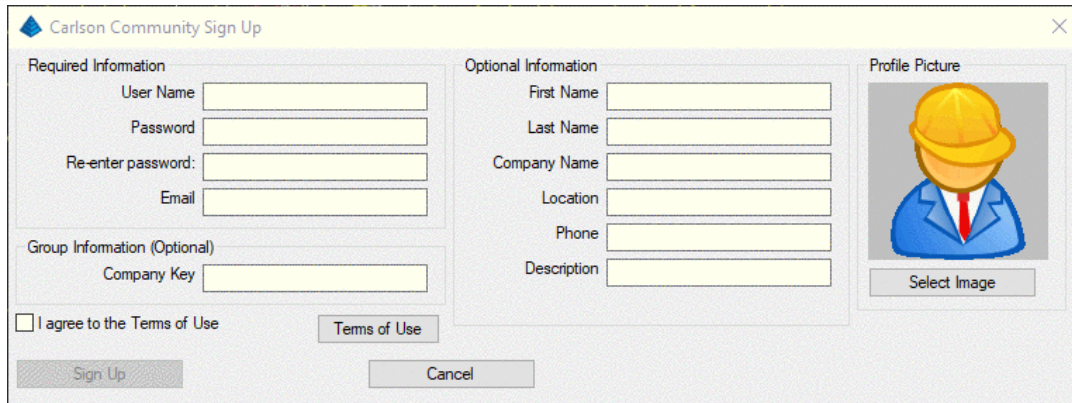
Carlson PhotoCapture

This command is used to download files from Carlson PhotoCapture directly. Found under the CAD engine "File" dropdown, clicking the command opens the Carlson PhotoCapture dialog.



If you do not have an account - these same login credentials are used for Carlson Community, Carlson Cloud, Carlson Academy and Carlson PhotoCapture - Click "Account Sign Up", fill in the information, click "Sign

Up". You will then receive a verification email that you must respond to to activate the account. Should you encounter problems with sign up, contact Carlson Support. If you have forgotten your password, click the "Forgot Password" button to reset it.



The dialog box is titled "Carlson Community Sign Up". It contains three main sections: "Required Information", "Optional Information", and "Profile Picture".

Required Information:

- User Name: [Text Field]
- Password: [Text Field]
- Re-enter password: [Text Field]
- Email: [Text Field]

Optional Information:

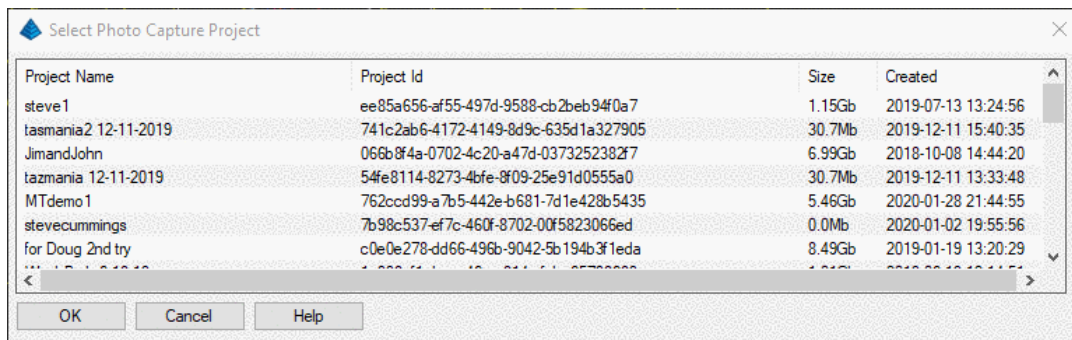
- First Name: [Text Field]
- Last Name: [Text Field]
- Company Name: [Text Field]
- Location: [Text Field]
- Phone: [Text Field]
- Description: [Text Field]

Profile Picture:

- Profile Picture: [Image of a person wearing a yellow hard hat and a blue suit]
- Select Image: [Button]

At the bottom, there is a checkbox for "I agree to the Terms of Use" and a "Terms of Use" button. Below that are "Sign Up" and "Cancel" buttons.

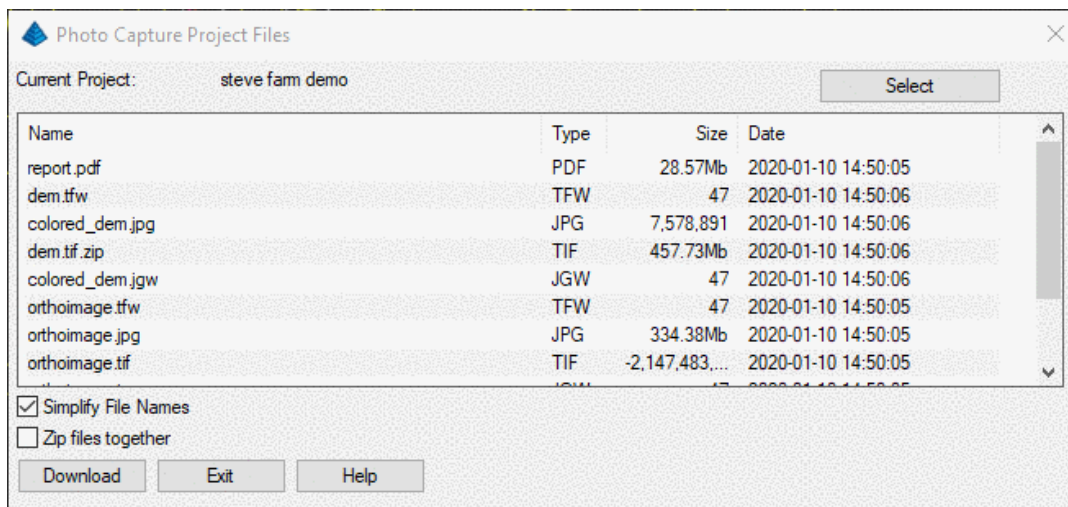
Fill in your User Name and Password, click "Remember Me" checkbox to store this for the next time or not, and click "Login". The following dialog will appear showing all jobs you have processed in Carlson PhotoCapture. Select the job you wish to download and click "OK".



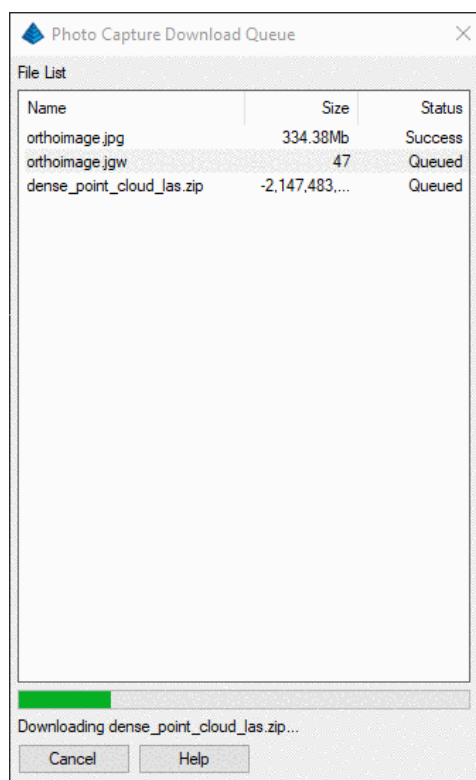
The dialog box is titled "Select Photo Capture Project". It displays a table of project information with columns for Project Name, Project Id, Size, and Created. Below the table are "OK", "Cancel", and "Help" buttons.

Project Name	Project Id	Size	Created
steve1	ee85a656-af55-497d-9588-cb2beb94f0a7	1.15Gb	2019-07-13 13:24:56
tasmania2 12-11-2019	741c2ab6-4172-4149-8d9c-635d1a327905	30.7Mb	2019-12-11 15:40:35
JimandJohn	066b8f4a-0702-4c20-a47d-0373252382f7	6.99Gb	2018-10-08 14:44:20
tazmania 12-11-2019	54fe8114-8273-4bfe-9f09-25e91d0555a0	30.7Mb	2019-12-11 13:33:48
MTdemo1	762ccd99-a7b5-442e-b681-7d1e428b5435	5.46Gb	2020-01-28 21:44:55
stevecummings	7b98c537-ef7c-460f-8702-00f5823066ed	0.0Mb	2020-01-02 19:55:56
for Doug 2nd try	c0e0e278-dd66-496b-9042-5b194b3f1eda	8.49Gb	2019-01-19 13:20:29

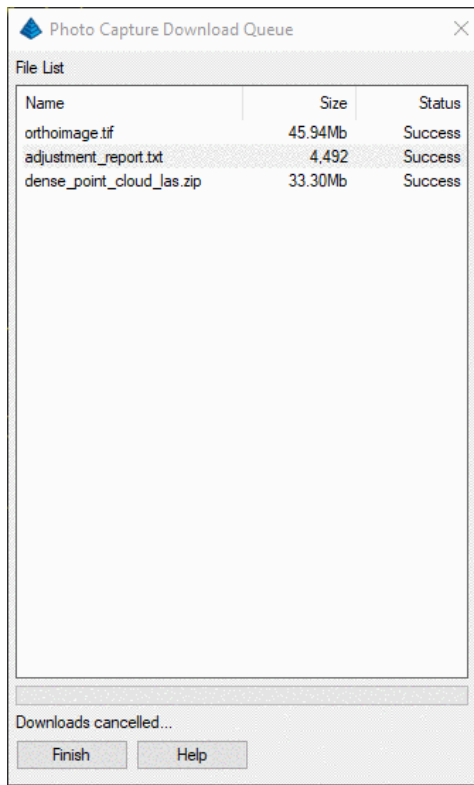
Now the Project Files dialog appears. Click "Select" to return to the Project list. Click the "Simplify File Names" checkbox to make the file names more easily readable. Click "Zip files together" checkbox to download one .zip file instead of multiple files. Using typical Windows Explorer selections tools such as click, ctrl-click, shift-click, select any or all files to be downloaded.



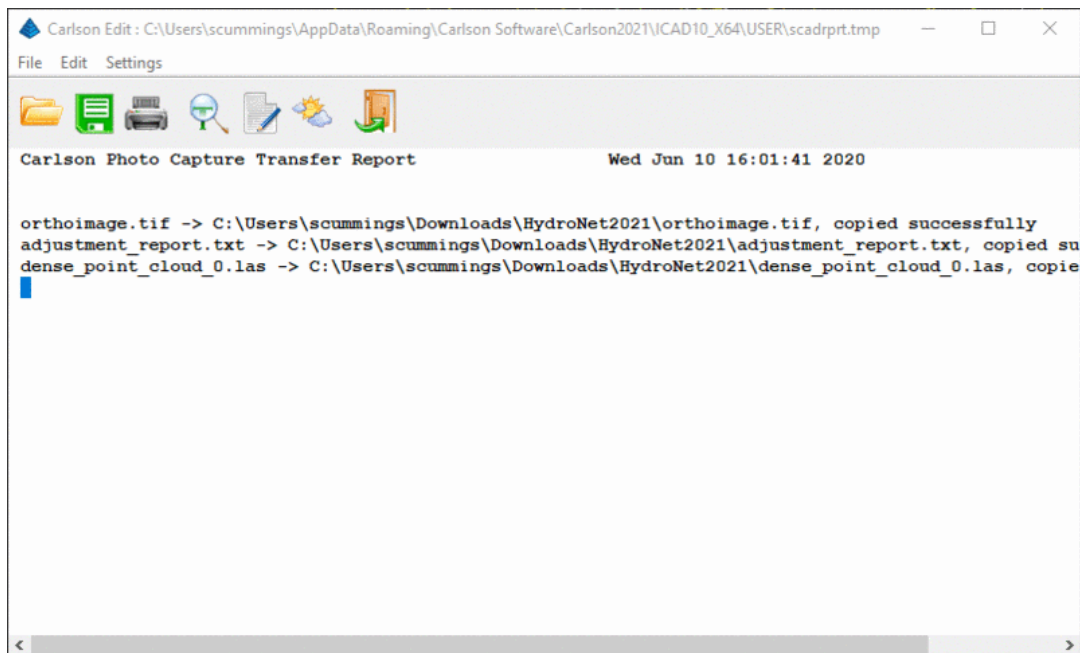
Clicking "Download" begins the download process and the Photo Capture Download Queue dialog appears. You can cancel the download process with the "Cancel" button.



Once the downloads are complete, the "Download" button is replaced by a "Finish" button.



Once the "Finish" button is clicked, a Carlson Photo Capture Download Report is generated. Note that the files are downloaded to the same location where the .DWG file is located. Now the files are available for use. Use the Red X, top right of the report dialog, to close the PhotoCapture interface.



Prerequisites: A Carlson Photo Capture account with at least one project in the account.

Import 3D Viewer File

This command selects a 3DX file and draws the objects from the scene including points, lines and 3D faces. The 3DX file can be created in 3D viewer commands including Surface 3D Flyover and 3D Viewer Drawing.

Pulldown Menu Location: File > Import

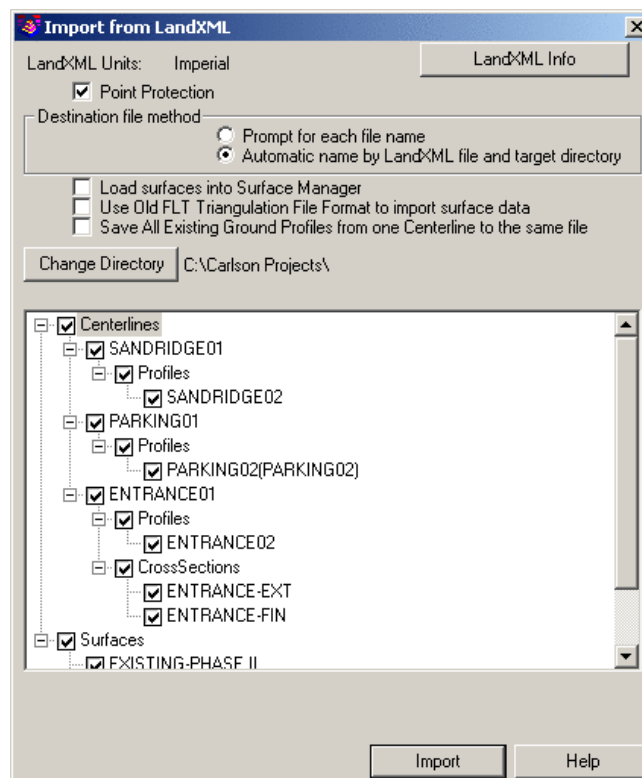
Keyboard Command: import3dx

Prerequisite: 3DX file

Import LandXML File

This command imports project data from the industry standard LandXML file format version 2.0 and earlier. Land development software applications including Carlson Software can exchange project data using LandXML. The project data in LandXML includes survey measurements, coordinates, centerlines, profiles, cross sections, surfaces, lots, plan view linework and pipe networks.

To import a LandXML file, the program first prompts to select a LandXML file to import. Then there is a dialog with import options along with a view of the contents of the LandXML file. You can toggle on which data from the file that you want to import.



LandXML Units: Indicates the Units of Measure associated with the incoming LandXML file (see the Unit Differences item below).

Point Protection: When enabled, you are prompted for a course of action if an existing LandXML file you've selected contains COGO points that have the same number(s) as those that already exist in the drawing. When disabled, existing point data in the project is updated with the values from the LandXML file.

Destination File Method: This option allows you to indicate how the incoming data file(s) are named as they are imported.

Load Surfaces into Surface Manager: When enabled, this option will automatically add surface model (TIN) data into the Surface Manager and graphically represents (draws) the surface model/contours according to the current settings found in the Triangulate & Contour command.

Skip Invisible Triangles: This option skips triangles flagged as invisible when importing a triangulation surface.

Use Old FLT Triangulation File Format to Import Surface Data: When enabled, the older ASCII-based Carlson *.FLT file format will be used in place of the newer and more efficient *.TIN file format.

Save All Existing Ground Profiles from One Centerline to the Same File: When enabled, collections of existing ground profiles associated with a particular centerline are combined into a single *.PRO file.

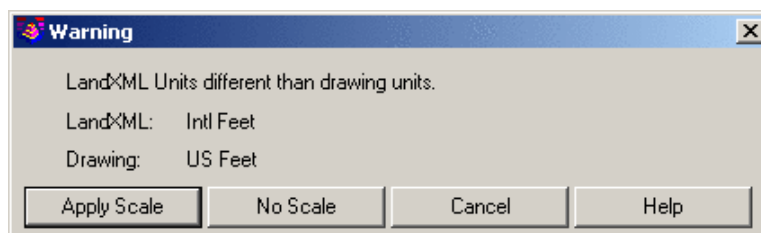
Draw PlanFeatures to current drawing: When the LandXML contains PlanFeatures, this option will automatically draw this linework. Otherwise, the program stores a .pln file which can be drawn using the File > Import > Polyline File routine.

Save Point Descriptions: For coordinate records, this option controls whether to use the LandXML description or code record for the coordinate file point descriptions.

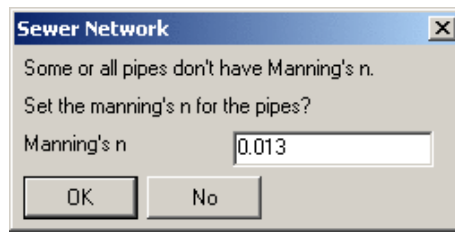
Add F2F Codes to Points from PlanFeatures: When the LandXML file contains both coordinates and plan features, this option adds Field-to-Finish linework codes to the descriptions of the coordinates. For example, if a plan feature line ends at point 25, then the Field-to-Finish end linework code of END is appended to the description of point 25.

Change Directory: This option allows you to adjust the folder location where the new data files will be written.

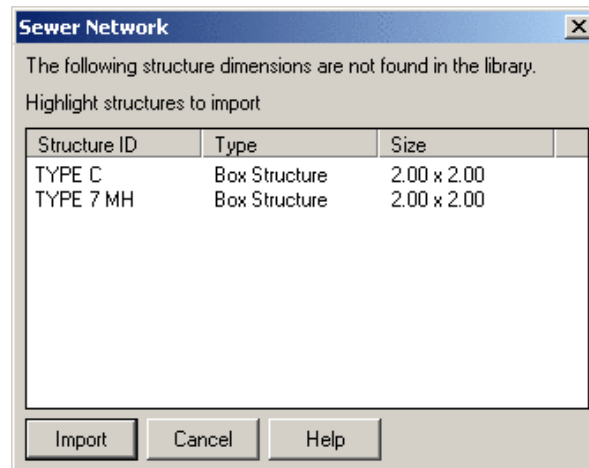
Import from LandXML: Enable or disable various entries that should be used to produce the data files found within the LandXML file.



Unit Differences: If the Units of Measure specified in the LandXML file are different than those found in Drawing Setup, you will be prompted for a course of action.



Manning's "n": If you are importing sewer data from a LandXML file and if the LandXML file does not carry Manning's "n" values, you will be prompted to specify a default Manning's "n" value for all incoming sewer entities that don't already have a Mannings "n" value.



Import Structures: If you are importing sewer data from a LandXML file and structure values specified in the LandXML file do not exist in the Structure Library, you will be prompted to indicate the structure(s) that should be imported into the Structure Library. Use standard Windows **click**, **shift+click** and/or **ctrl+click** functionality to select multiple structures at the same time.

Skip Invisible Triangles: This option applies to importing TIN surfaces from Civil 3D. When this option is active, triangles marked by Civil 3D as invisible or excluded are not imported.

Note:

- The LandXML initiative is being driven by the land development industry as an acceptable means to share and transfer land data rather than the traditional graphical representation of that data. It also provides an effective means for transferring a variety data (points, centerlines, profiles, surface models, sewer data, *etc*). Another advantage of LandXML is that the LandXML data structure is CAD and software vendor neutral (meaning you don't have to own or use the CAD or software product used by your data provider).
- Visit <http://www.landxml.org> for additional information on the uses and acceptance of the LandXML project.

Pulldown Menu Location(s): File > LandXML

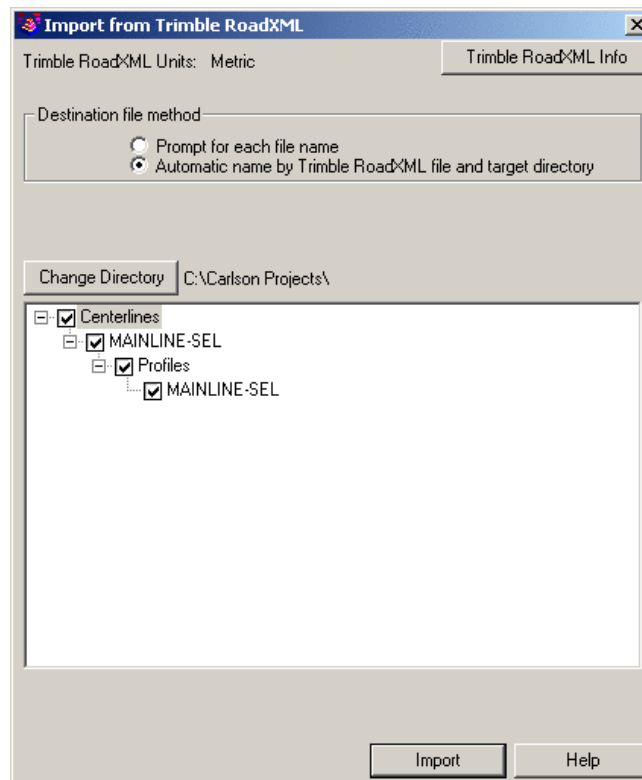
Keyboard Command: landxml_import

Prerequisite: A LandXML file to import

Import RoadXML File

The Import RoadXML File routine provides a mechanism where road-based data from other software applications (including Carlson Software) can be brought into a project and used for analysis and/or design purposes. The program supports centerline and profile data in Trimble style RoadXML format. To import a RoadXML file, a series of dialog boxes are presented:

Select RoadXML File: The standard File Selector dialog box prompts you identify an existing RoadXML (*.RXL) file you wish to import. The following dialog box is then displayed:

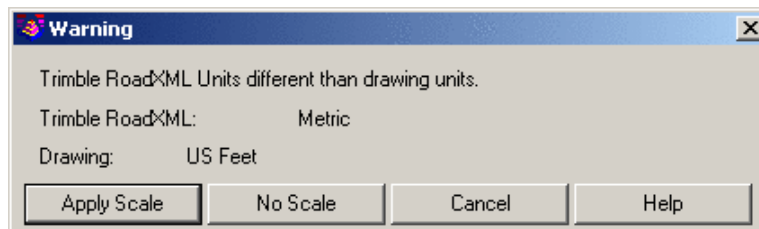


RoadXML Units: Indicates the Units of Measure associated with the incoming RoadXML file (see the Unit Differences item below).

Destination File Method: This option allows you to indicate how the incoming data file(s) are named as they are imported.

Change Directory: This option allows you to adjust the folder location where the new data files will be written.

Import from RoadXML: Enable or disable various entries that should be used to produce the data files found within the RoadXML file.



RoadXML Units: RoadXML files are always in metric units. If the current drawing units as set in Drawing Setup are not metric, then you will be prompted whether to apply a scale factor. Note: Visit <http://www.road-xml.org> for additional information on the RoadXML initiative.

Pulldown Menu Location: File > Import

Keyboard Command: roadxml.import

Prerequisite: A RoadXML file to import

Import 12D File

This function imports project data from the 12D modeling program into Carlson format. Linework geometry from the 12D file is drawn as polylines. The routine also draws any text, symbols and points. Project data for centerlines,

profiles, sections, pipe networks and surfaces are saved into Carlson project files.

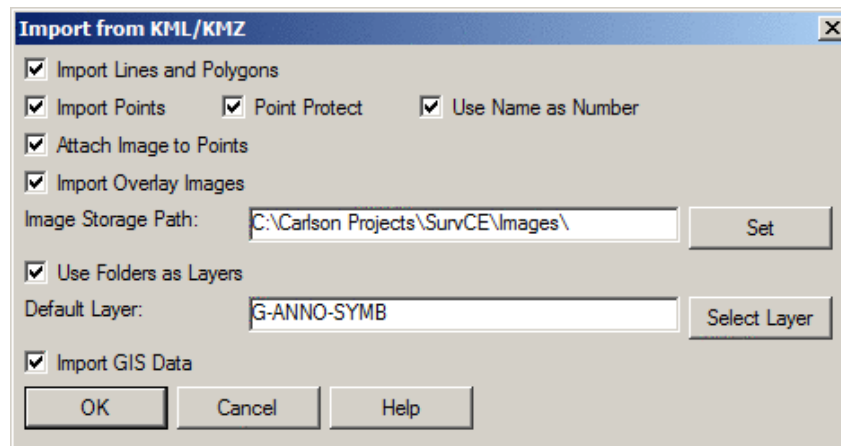
Pulldown Menu Location: File > Import

Keyboard Command: 12d_import

Prerequisite: A 12DA file to import

Import Google Earth File

The Import Google Earth File command allows you to insert a KML (Keyhole Markup Language or alternatively a KMZ) file of points (KML *Placemark*), polylines (KML *Path*) and closed polylines (KML *Polygon*) into your drawing. Throughout this discussion, *KML* will be used to also describe KMZ files unless explicitly noted.



Import Lines and Polygons: When this option is selected, KML Path and Polygon entries will be placed into the drawing as open or closed polylines, respectively.

Import Points: When this option is selected, KML Placemark entries will be placed into the drawing and active coordinate file.

Point Protect: When enabled, existing points in the active coordinate file will not be over-written.

Use Name as Number: When enabled, the names of KML Placemark entries will be utilized as Carlson point numbers.

Attach Image To Points: This option applies to KML files created by Carlson SurvCE when the points are stored together with an image. Use this option to attach the image to the point symbol. The images can be viewed using the Image/Document Inspector command. The image files are extracted from the KML and stored in the **Image Storage Path** folder.

Import Overlay Images: When enabled, any *Overlay Image* placemarks in the KML file will be inserted into the CAD drawing.

Use Folders as Layers: When enabled, KML Folder entries will be used to create layer names in CAD and the supported KML options described above will be placed onto the layer that conforms the the Folder to which they belong.

Default Layer: The supported KML options described above that are not contained in a KML folder will be placed into the specified layer.

Import GIS Data: When enabled, this routine will populate GIS data into the drawing such as that written from the drawing with the Export Google Earth File command.

Note:

- Placemarks, paths or polygon entries that have an altitude value specified will be imported at the proper "Z" elevation in the CAD drawing.
- KML or KMZ files can be specified for the import process.

Prompts

Google Earth File to Read: *Select a previously saved KML or KMZ file.*

- To import a Google Earth image into your drawing, use the Place Google Earth Image command.
- To import a Google Earth terrain data into a Carlson TIN (surface model), use the Place Google Earth Image command.
- To export content from your drawing to a KML file, use the Export Google Earth File command.

Pulldown Menu Location: File > LandXML/RoadXML/Google Earth

Keyboard Command: kmlread

Prerequisite: A KML or KMZ file with Placemark, Path and/or Polygon information, an active coordinate file with an established projection zone through Drawing Setup.

Civil 3D All

This command converts custom AEC objects from Civil 3D into standard CAD entities. The supported objects include points, surfaces, centerlines, pipe networks, feature lines and labels.

Pulldown Menu Location: File > Import

Keyboard Command: c3d_all

Prerequisite: AEC objects

Civil 3D Labels

This command converts custom AEC label objects from Civil 3D into standard CAD text entities.

Pulldown Menu Location: File > Import

Keyboard Command: c3d_text

Prerequisite: AEC label objects

Civil 3D Linework

This command converts custom AEC linework objects such as feature lines from Civil 3D into standard CAD entities.

Pulldown Menu Location: File > Import

Keyboard Command: c3d_lwork

Prerequisite: AEC linework objects

Civil 3D Pipes

This command converts custom AEC pipe and structure objects from Civil 3D pipe networks into standard CAD entities.

Pulldown Menu Location: File > Import

Keyboard Command: c3d_pipe

Prerequisite: AEC pipe objects

Import MxRoad GENIO

This command draws the linework and entities from an MxRoad GENIO file (.inp) into the current drawing. The File > Export > Polyline File command can be used to create an .inp file from the drawing.

Pulldown Menu Location: File > Import

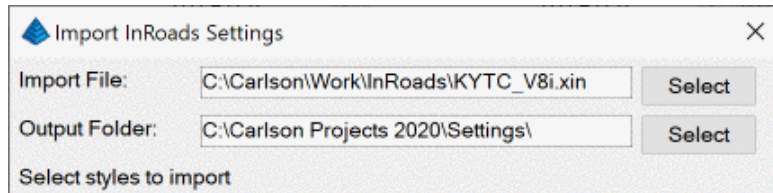
Keyboard Command: genio_read

Prerequisite: None

Import InRoads Settings

The Import InRoads Settings routine provides a mechanism where the InRoads software application settings can be used in Carlson Software. The program currently supports Surface and Profile styles from the InRoads settings file. To import InRoads settings, the following dialog box is presented:

Select InRoads File: The standard File Selector dialog box prompts you to identify an existing InRoads (*.XIN) file you wish to import.

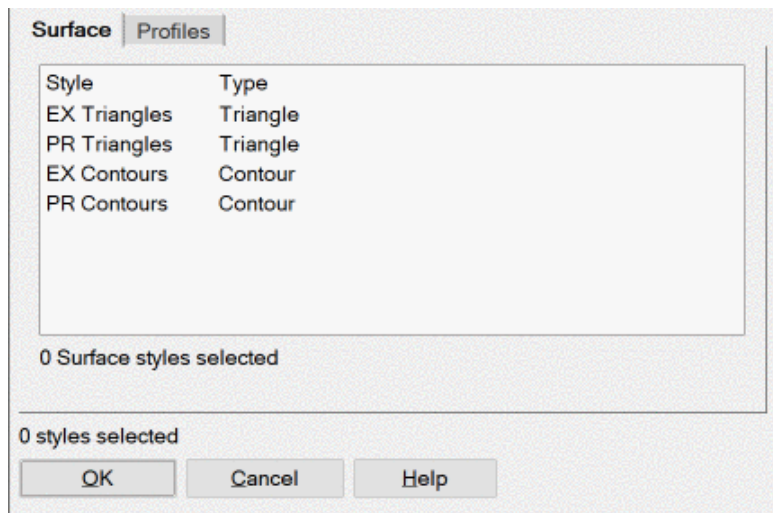


Import File: InRoads settings file to import styles from.

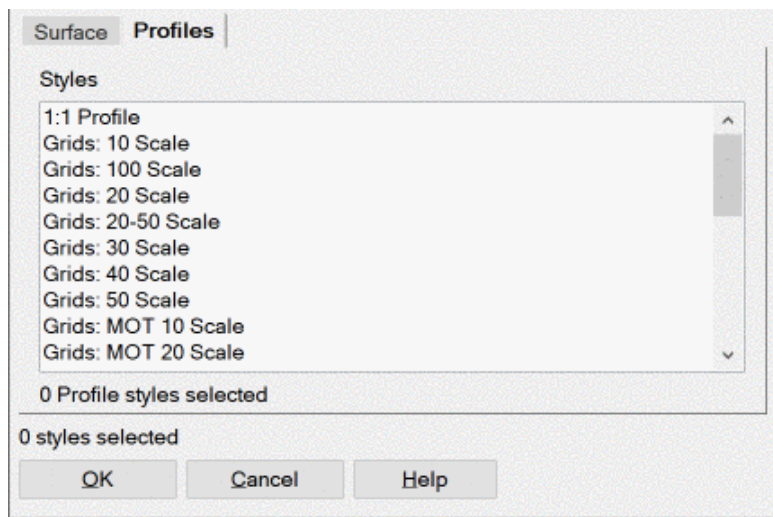
Select: Select a new InRoads settings file.

Output Folder: Output location for setting files to be created.

Select: Select a different output folder location.



Surface Style List: The Triangle and Contour settings display and are multi-selectable. Use the Select and Shift key or Select and Ctrl key for multiple selection of settings.



Profile Style List: The Profile settings display and are multi-selectable. Use the Select and Shift key or Select and Ctrl key for multiple selection of settings.

Each selected style will generate its name .TRI and/or .PFS Carlson setting file in the output folder.

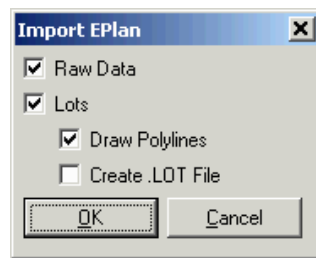
Pulldown Menu Location: File > Import

Keyboard Command: importxin

Prerequisite: An InRoads file to import

Import ePlan File

This command reads an ePlan project file and creates Carlson project files. Point data is stored to a Carlson coordinate file. There is an option to store measurements to a Carlson RW5 file. There is an option to import lot data to a Carlson LOT file and to draw the lots as polylines.



Pulldown Menu Location: File > Import

Keyboard Command: eplan_import

Prerequisite: ePlan XML file

Export LandXML File

The Export LandXML File routine provides a mechanism where data can be sent from Carlson Software into a LandXML file for use in other applications that support the LandXML data specification. LandXML version 2.0 is used which includes the ability to output triangulation surfaces with colors and materials defined for the triangles.

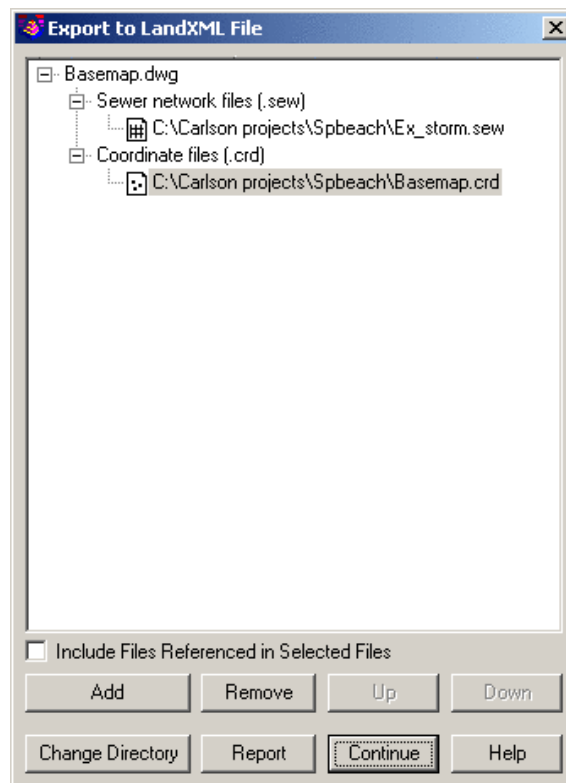
To generate a LandXML file, a series of dialog boxes are presented:



Export to LandXML: This option allows you to individually select the desired Carlson Software data file(s) that should be included in the LandXML file.

Project Data Files: This option allows you to quickly select the various data files associated with, and defined by a Carlson Project (*.prj) file.

Select LandXML File: Specify the name of a LandXML file you wish to create.

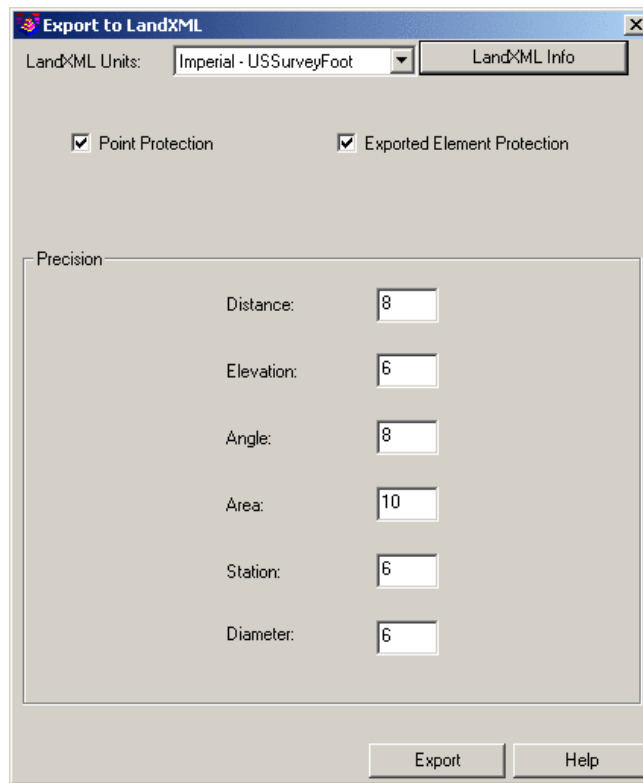


Include Files Referenced in Select Files: When enabled, this option will automatically add other files that are referenced by the selected file. As an example, the file produced by the Carlson Road Network command references TINs, Centerlines, Profiles, *etc.*, and adding the single Road Network file will also add the referenced file(s) into the Export to LandXML File dialog box.

Export to LandXML File: Add, remove (using standard Windows **click**, **shift+click** and/or **ctrl+click** functionality) or otherwise organize the data file(s) that is to be incorporated into the LandXML file.

Change Directory: This option allows you to adjust the folder location from where selected data files should be referenced (often used for project revision purposes).

Report: Create a report (suitable for file transmission or archival purposes) of the file(s) selected to be incorporated into the LandXML file.



LandXML Units: Specify the desired Units of Measure that reflect the outgoing data.

Point Protection: When enabled, you are prompted for a course of action if an existing LandXML file you've selected contains COGO points that have the same number(s) as those being selected for the LandXML file. When disabled, point data you've selected for the LandXML file are automatically written to (or updated into) the existing LandXML file.

Exported Element Protection: When enabled, you are prompted if existing data (such as a centerline) in a LandXML file should be updated with data of the same name that you have selected for the LandXML file.

Precision: Set the desired level of precision (number of decimals) for each of the various measurement categories.

Profiles:

There are two major different types of profiles in LandXML: ProfSurf and ProfAlign. ProfSurf is typically an existing surface that is usually created using existing surface data. The data for this type of profile it is stored in a series of station-elevation values as a representation of a PntList2D list. ProfAlign is for a design profile. The data for this type of profile is stored in LandXML elements starting from the simplest one: PVI element, CircCurve element, ParaCurve element, etc.

Carlson differentiates the two types mentioned above by using the profile type in the Carlson .pro file: Generic = ProSurf, Road = ProAlign.

Note:

- The LandXML initiative is being driven by the land development industry as an acceptable means to share and transfer land data rather than the traditional graphical representation of that data. It also provides an effective means for transferring a variety data (points, centerlines, profiles, surface models, sewer data, *etc*). Another advantage of LandXML is that the LandXML data structure is CAD and software vendor neutral (meaning you don't have to own or use the CAD or software product used by your data provider).

- Visit <http://www.landxml.org> for additional information on the uses and acceptance of the LandXML initiative.

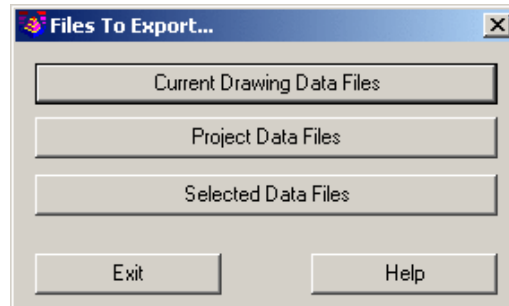
Pulldown Menu Location(s): File > LandXML

Keyboard Command: landxml_export

Prerequisite: Carlson project data files to convert

Export RoadXML File

The Export RoadXML File routine creates a RoadXML RXL file using Carlson format centerline and profile files. This RoadXML file can be used for data exchange with other applications that support the RoadXML data specification such as Trimble. To generate a RoadXML file, a series of dialog boxes are presented:

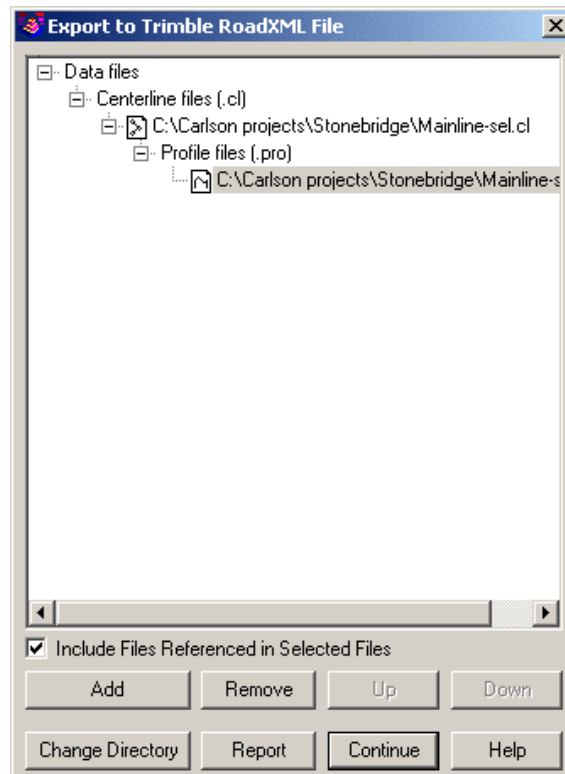


Current Drawing Data Files: This option selects the various data files associated with , and defined by the Drawing Explorer command.

Project Data Files: This option allows you to quickly select the various data files associated with, and defined by a Carlson Project (*.prj) file.

Selected Data Files: This option allows you to individually select the desired Carlson Software data file(s) that should be included in the RoadXML file. This is followed by:

Select RoadXML File: Use the standard File Selector dialog box to specify a new or append to an existing RoadXML file. This is followed by:

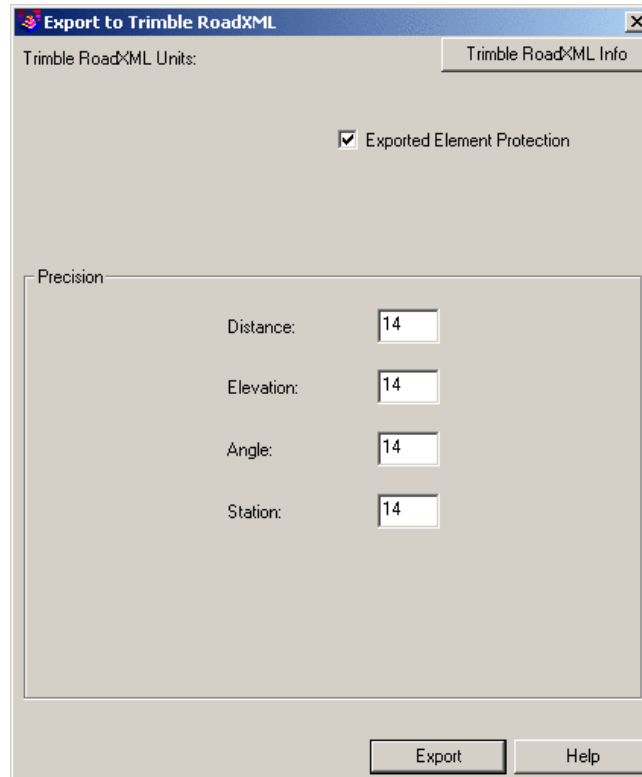


Include Files Referenced in Select Files: When enabled, this option will automatically add other files that are referenced by the selected file. As an example, the file produced by the Carlson Road Network command references TINs, Centerlines, Profiles, *etc.*, and adding the single Road Network file will also add the referenced file(s) into the Export to RoadXML File dialog box.

Export to RoadXML File: Add, remove (using standard Windows **click**, **shift+click** and/or **ctrl+click** functionality) or otherwise organize the data file(s) that is to be incorporated into the RoadXML file.

Change Directory: This option allows you to adjust the folder location from where selected data files should be referenced (often used for project revision purposes).

Report: Create a report (suitable for file transmission or archival purposes) of the file(s) selected to be incorporated into the RoadXML file.



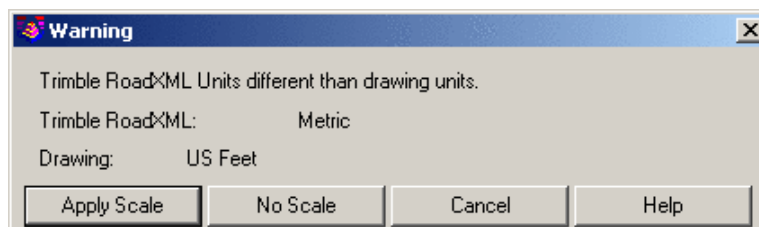
RoadXML Units: The Units of Measure are displayed for the RoadXML file about to be created.

Exported Element Protection: When enabled, you are prompted if existing data (such as a centerline) in a RoadXML file should be updated with data of the same name that you have selected for the RoadXML file.

Precision: Set the desired level of precision for each of the various measurement categories.

Pick the Export button to complete the creation of the RoadXML RXL file.

RoadXML Units: RoadXML files are always in metric units. If the current drawing units as set in Drawing Setup are not metric, then you will be prompted whether to apply a scale factor.



Indicate the desired action of what should occur if the units of the RoadXML do not match those of the current drawing.

Note: Visit <http://www.road-xml.org> for additional information on the RoadXML initiative.

Pulldown Menu Location(s): File > LandXML/RoadXML

Keyboard Command: roadxml.export

Prerequisite: Carlson project data files to convert

Export Polyline File

This command creates a polyline file that contains the point data of the select polylines. The objects supported by this tool include polylines, arcs and lines. If you want to include text, you must use the *Text Explode To Polylines* command found in the Edit menu to convert the text to polylines before running this command. This polyline file is a text file that has three formats. The Carlson format (.PLN) is used by machine control (Carlson Grade, Dozer 2000, GradeStar) for the plan view. Each polyline begins with a line of "POLYLINE, Color number". Then the points for the polyline are listed on separate lines in X,Y,Z format. Here is a list of the available color numbers:

0 = Black	8 = Dark Gray
1 = Blue	9 = Light Blue
2 = Green	10 = Light Green
3 = Cyan	11 = Light Cyan
4 = Red	12 = Light Red
5 = Magenta	13 = Light Magenta
6 = Brown	14 = Yellow
7 = Light Gray	15 = White

The MicroStation format (.txt) can be imported into MicroStation. This format has the coordinates as space delimited for each polyline point. There is an extra column with a 1 or 0 where 1 specifies the start of a new polyline. The DTM and Idan formats create linework files for the DTM and Idan programs.

Prompts

Polyline file format [<Carlson>/DTM/Idan/MicroStation]? *press Enter for Carlson format*

Specify File to Write dialog *create a new file or append to existing*

Polyline file for Grid File Utilities macro [Yes/<No>]? *press Enter* The option will write a polyline file that can be used with Grid File Utilities for inclusion/exclusion perimeters.

Include Z coordinate in polyline file [Yes/<No>]? *press Enter* This option controls whether the polyline vertices are written in 2D or 3D.

Specify Exclusion/Warning Polylines [Yes/<No>]? *press Enter* This option applies to machine control for warning areas.

Specify WorkZone Polylines [Yes/<No>]? *press Enter* This option applies to machine control for working areas.

Reduce Polyline Vertices [<Yes>/No]? *press Enter* This option applies Reduce Polyline to the polyline vertices before writing the file.

Enter reduce offset cutoff <0.1>: *press Enter*

Decimal places for coordinates <2>: *press Enter*

Select polylines, lines and arcs to write.

Select objects: *pick the entities to process*

Done.

Sample Polyline File:

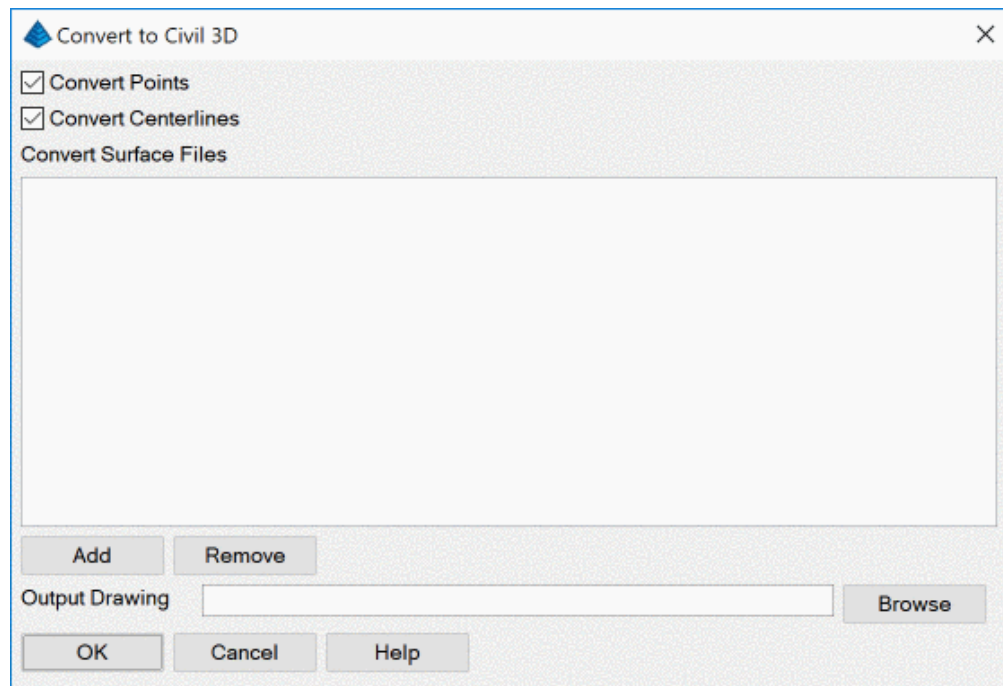
POLYLINE,15
47639.82,74540.11,0.00
47670.49,74565.79,0.00
47701.08,74591.49,0.00
49375.61,76358.47,0.00
50066.86,76846.75,0.00
POLYLINE,15
47633.24,74547.97,0.00
47663.90,74573.65,0.00
etc...

Keyboard Command: polywrite

Prerequisite: Polylines in the drawing

Export Civil 3D Drawing

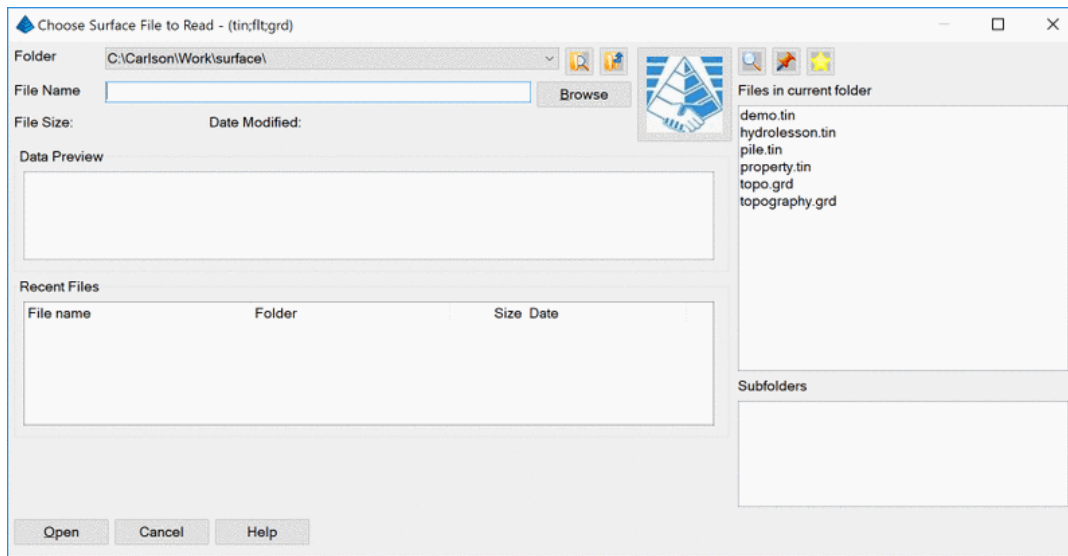
This command saves the current drawing to a dwg file and converts the optional Carlson points, Carlson centerlines and selected Carlson TIN files into Civil 3D objects.



Convert Points: When enabled, the Carlson points in the drawing will be converted into Civil 3D point objects.

Convert Centerlines: When enabled, the Carlson centerlines in the drawing will be converted into Civil 3D alignment objects.

Clicking the **Add** button yields the Choose Surface File to Read dialog box.



Clicking the **Remove** button removes the selected surface file from the list of Surface Files to convert.

Output Drawing *select a .DWG file to create Civil 3D objects within.*

Pulldown Menu Location: File > Export

Keyboard Command: convert_c3d

Prerequisite: None

Export Civil 3D Alignments

This command saves the current drawing to a dwg file and converts the Carlson centerlines into Civil 3D alignment objects.

Pulldown Menu Location: File > Export

Keyboard Command: cl_c3d

Prerequisite: None

Export Civil 3D Points

This command saves the current drawing to a dwg file and converts the Carlson points into Civil 3D point objects.

Pulldown Menu Location: File > Export

Keyboard Command: crd_c3d

Prerequisite: None

Export Civil 3D Surface

This command saves the selected TIN file to a dwg file and converts the Carlson TIN into Civil 3D surface object.

Pulldown Menu Location: File > Export

Keyboard Command: tin_c3d

Prerequisite: None

Export 12D File

This command creates a 12DA project file for the 12D modeling program. You can export drawing entities and Carlson project files. For the drawing entities, the program supports linework, text and point entities. For Carlson project files, the export handles triangulation surfaces (TIN), centerlines (CL), pipe networks (SEW), profiles (PRO) and cross sections (SCT).

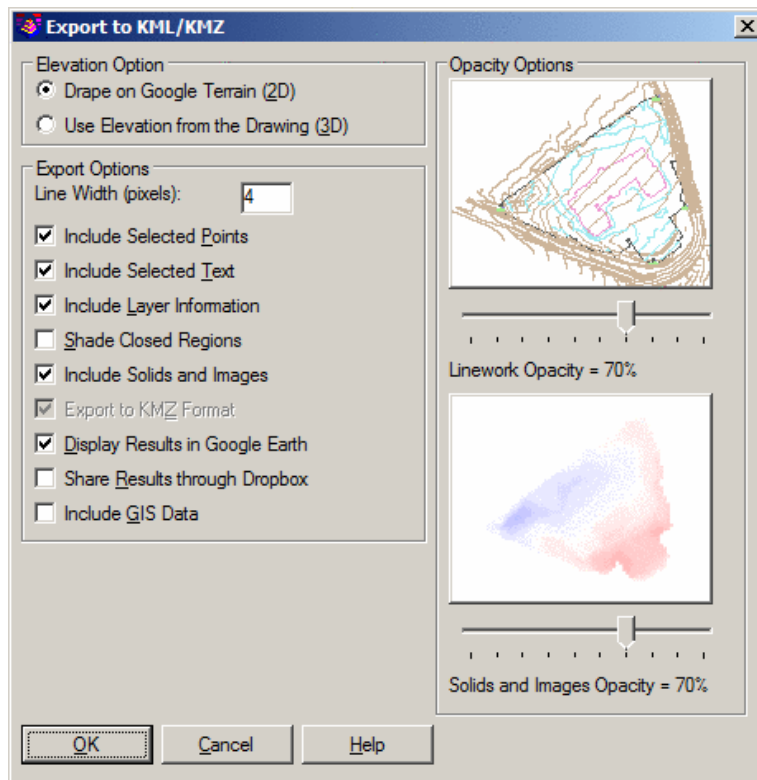
Pulldown Menu Location: File > Export

Keyboard Command: 12d_export

Prerequisite: None

Export Google Earth File

The Export Google Earth File allows you to produce a KML (Keyhole Markup Language or alternatively a KMZ) file of points, polylines, text, solids, images, lines and arcs for rendering in other mapping and GIS applications such as Google Earth and Google Maps. Throughout this discussion, *KML* will be used to also describe KMZ files unless explicitly noted.



Drape on Google Terrain (2D): When this option is selected, entities written to the KML file will have an Altitude setting of "Clamped to ground."

Use Elevation from the Drawing (3D): When this option is selected, entities written to the KML file will have an Altitude setting of "Absolute."

Line Width (pixels): Indicate how wide selected linework should be when viewed in Google Earth. The higher the value, the thicker the line.

Include Selected Points: When enabled, this option exports selected Carlson point information to the KML <Placemark><Point>...</Point></Placemark> tag structure. For the Google placemark, the **Placemark Name** chooses between using the point number, elevation or description.

Include Selected Text: When enabled, this option exports selected Text and MText entities to the KML <Placemark><Point>...</Point></Placemark> tag structure.

Include Layer Information: When enabled, this option organizes exported information based on the layer of each entity, with each CAD layer becoming a KML <Folder>...</Folder> entry with the color of the group taking the general color of the CAD layer.

Shade Closed Regions: When enabled, all closed polyline regions (*e.g.* building pads, ponds, *etc*) will be fill-shaded.

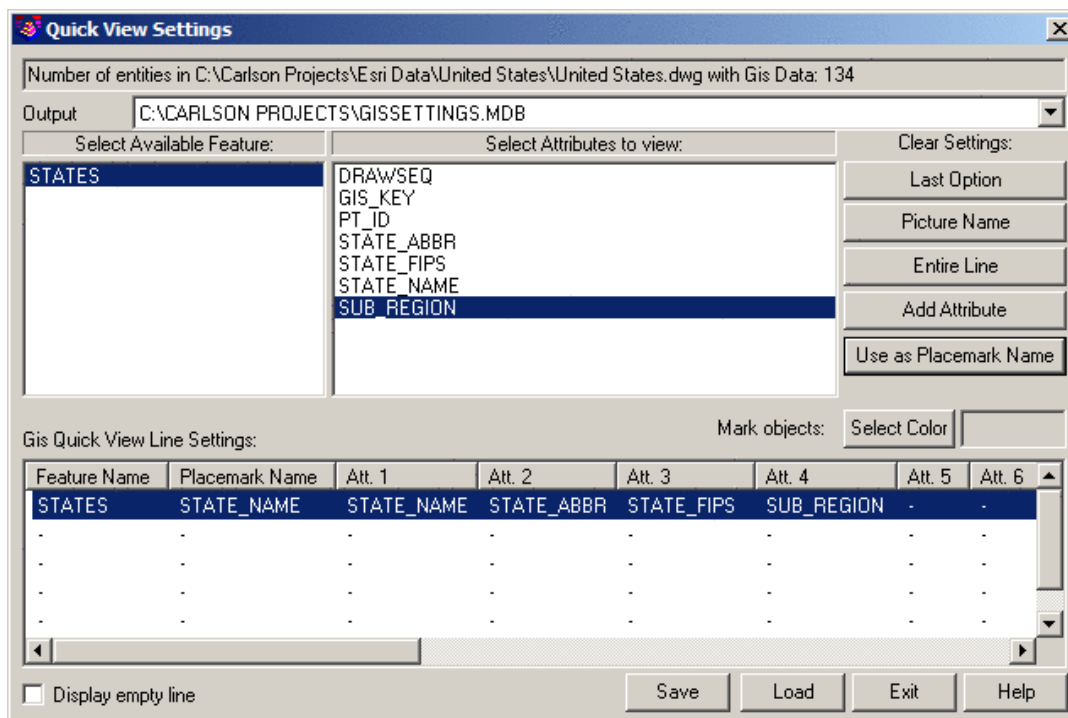
Include Solids and Images: When enabled, Solid entities and Images are included in the KML as <Placemark><Polygon>...</Polygon></Placemark> and/or <GroundOverlay>...</GroundOverlay> tags, respectively.

Export to KMZ Format: When enabled, the KML file is written to the more compact (zipped) KMZ version of the standard KML file format.

Display Results in Google Earth: When enabled, the results of the KML are passed to and automatically opened with Google Earth. To setup for running Google Earth Pro, go to Windows Start > Default Programs > Associate File Type With Program and assign KML and KMZ file types to Google Earth Pro.

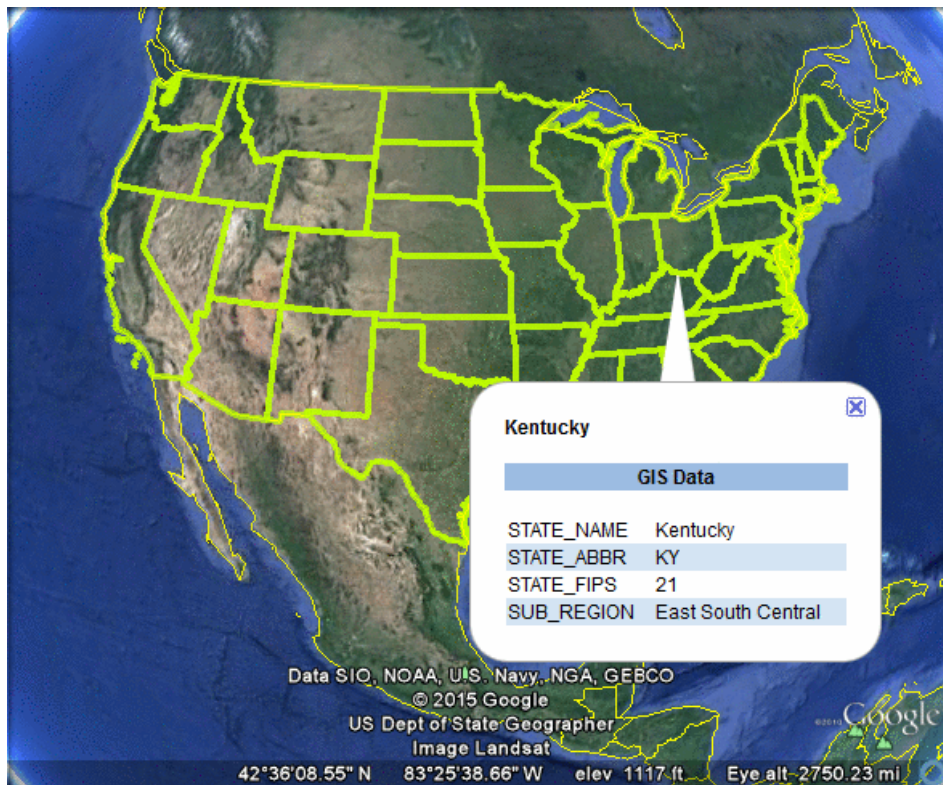
Share Results through Dropbox: When enabled, the results of the KML are passed to a personal DropBox account.

Include GIS Data: When enabled, GIS attribute data associated with selected entities will be populated into the KML file. GIS attribute data can be imported via the Import SHP File command (see the Sample Esri Data note below) or manually assigned with the GIS Data Editor command or the Set Google Tags command. Upon specifying the KML file to output, the **Quick View Settings** dialog box of the GIS Inspector command may display:



1. Select a desired GIS Feature from the upper-left of the dialog box.
2. Add the desired Attributes to the display list using a double-click action or clicking the **Add Attribute** button.
3. Indicate a desired Attribute to serve as the *Placemark Name* and use the **Use as Placemark Name** button to set it.

Upon exiting from the dialog box, available GIS Data on a per-entity basis will be populated into the KML file:



Linework Opacity: Use the horizontal slider control to indicate the desired level of opacity that should be applied to linework entities. A lower opacity results in increased entity transparency and is helpful for viewing underlying map data found in applications such as Google Earth.

Solids and Images Opacity: Use the horizontal slider control to indicate the desired level of opacity that should be applied to Solids and Image entities. A lower opacity results in increased entity transparency and is helpful for viewing underlying map data found in applications such as Google Earth.

Note:

- When the *Use Elevations from the Drawing (3D)* option is selected, be aware that elevation values lower than the Google Earth terrain may be obstructed in the Google Earth display.
- Attribute information (*e.g.* Number, Elevation, Description) of selected Carlson points are also written to the KML and will display in the "balloon" when a point is picked in the Google Earth display or data hierarchy.
- The formatting of any selected MText entities is not propagated into the KML/KMZ file.
- When the *Shade Closed Regions* toggle is enabled, note that **all** closed polyline regions will become fill shaded and may lead to undesired results for items such as closed contours.
- When *Image* entities are included, the size of the image itself is incorporated into the KMZ file and may significantly swell the size of the KMZ file which may result in lengthy load times into other applications.
- To have results posted to Dropbox, the Dropbox Application for Windows must be first installed to your PC using its default folder specification for the location of shared/synchronized folders/files.
- When prompted for the name of the KML/KMZ file to write, the appropriate KML or KMZ file extension based on the *Export to KMZ Format* toggle will be added to the file if the file extension is not specified.
- Arcs and polylines with arcs are converted into chord segments that closely approximate the arc(s).
- Other entities not supported for direct export to a KML file (*e.g.* circles, ellipses, splines, multilines, *etc.*), can be first turned into polylines with the Entities to Polylines command.
- Sample Esri data of the United States in Shape File form can be freely downloaded and explored.
- The graphical symbology of any/all items sent to the KML file can be manually modified via the Google Earth interface.
- When sending closed polylines to Google Earth, they will be sent as closed polygons.

Prompts

Select points, polylines, text, solids, images, lines and arcs to write.

FILter/⟨Select entities⟩: *Select the desired entities and press Enter when complete.*

- To import a Google Earth image into your drawing, use the Place Google Earth Image command.
- To import a Google Earth terrain data into a Carlson TIN (surface model), use the Place Google Earth Image command.
- To import KML content into your drawing, use the Import Google Earth File command.

Pulldown Menu Location(s): File > Export

Keyboard Command: kmlwrite

Prerequisite: Points, lines or polylines in the drawing with an established projection zone through Drawing Setup.

Export ePlan File

This command creates an ePlan project file from Carlson lot and coordinate files. The program prompts for the Carlson .LOT and coordinate files to read and then the ePlan .XML file to create.

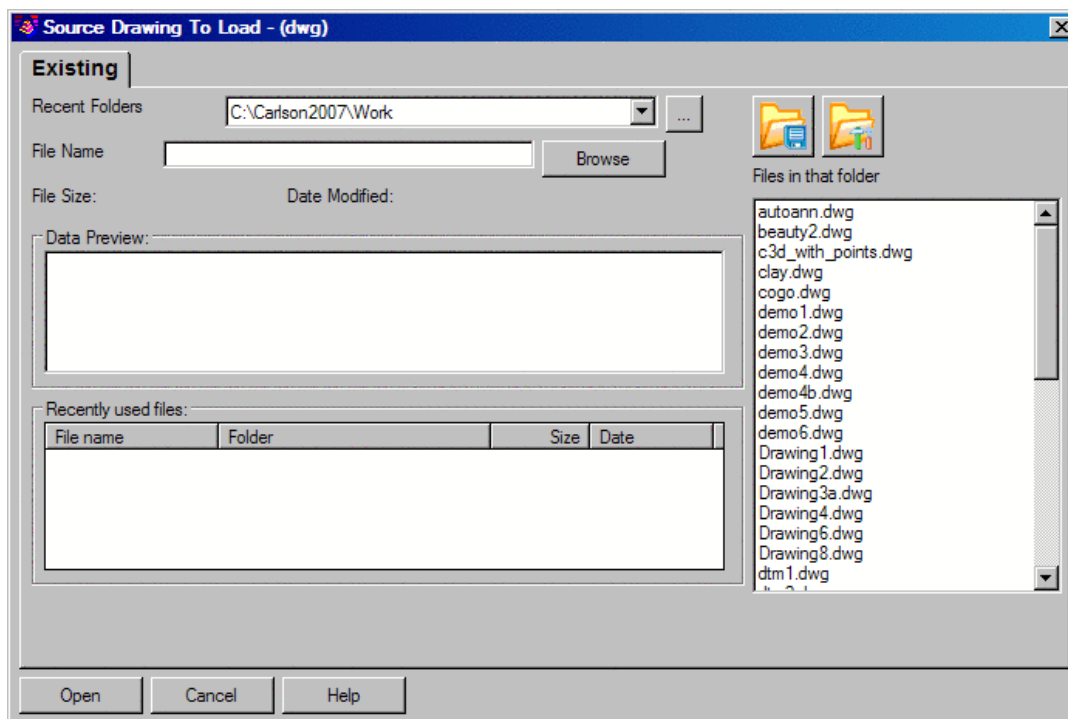
Pulldown Menu Location: File > Export

Keyboard Command: eplan_export

Prerequisite: Carlson lot and coordinate files

Export Drawing to AutoCAD 14

This command will save an existing Carlson drawing to AutoCAD R14 format. This command is for Carlson in AutoCAD 2004 and Carlson working in AutoCAD 2005.

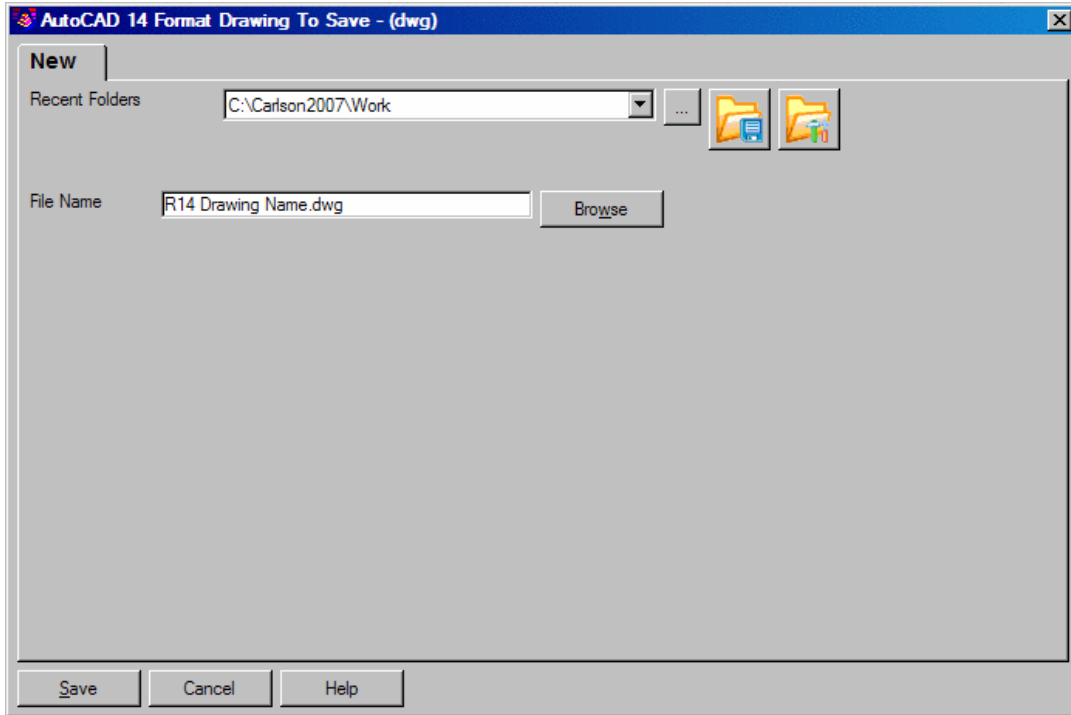


Prompts

Source Drawing To Load dialog *select a .DWG file*

AutoCAD R14 Format Drawing To Save dialog *select name for a new .DWG file*

Files saves to R14.



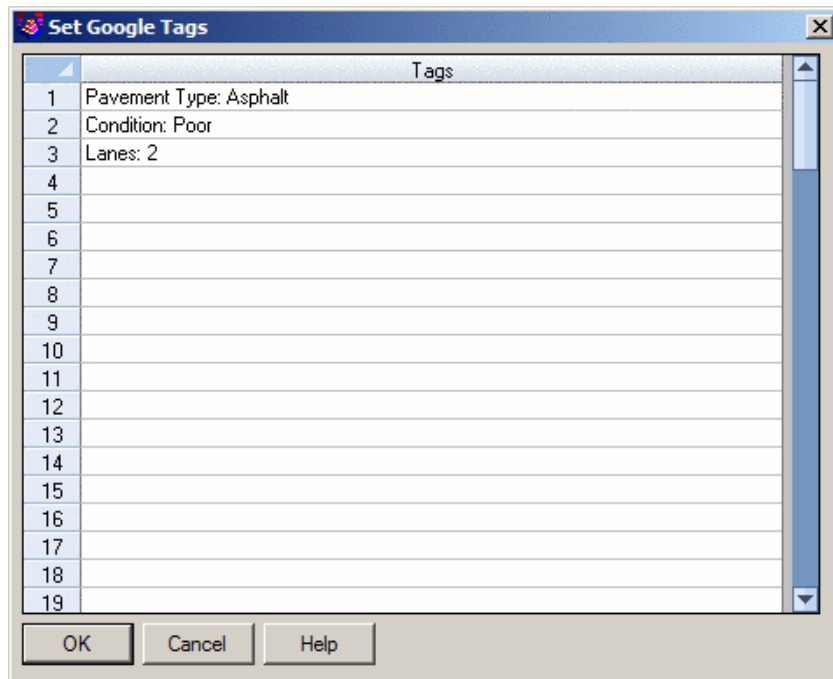
Pulldown Menu Location: File

Keyboard Command: dwg2r14

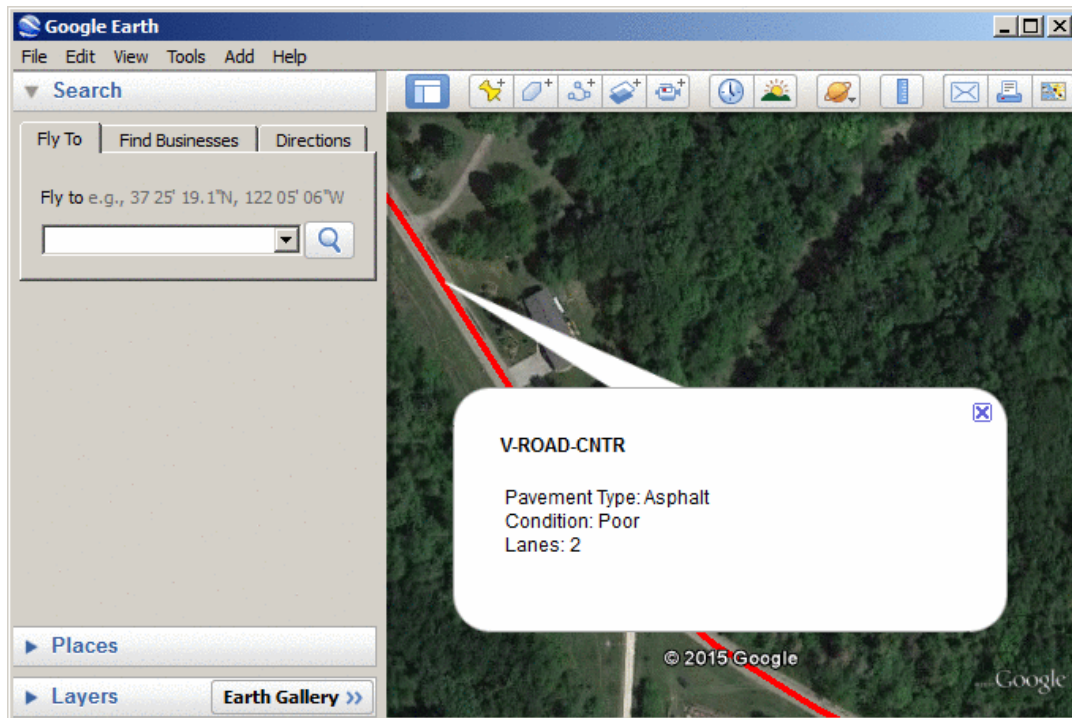
Prerequisite: An existing Carlson .DWG file, using Carlson in AutoCAD 2004 or Carlson in AutoCAD 2005

Set Google Tags

This command adds labels to drawing entities that are shown as "balloon" labels in Google Earth when the Include GIS Data option of the Export Google Earth File command is enabled.



After selecting the entity to tag, the dialog box above displays that permits various forms of tag data to be associated with the entity. An example result is shown below for reference purposes (the first "tag" being the layer upon which the entity resides):



Prompts

Select entity: *Select a point, symbol, text or linework*

Pulldown Menu Location: File > Export

Keyboard Command: tagkml

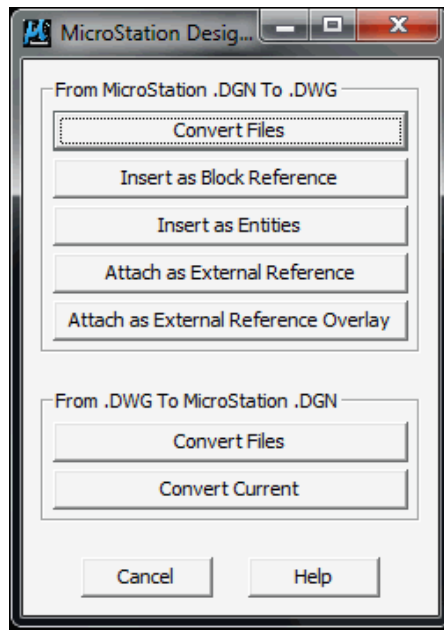
Prerequisite: Entities to attach tags

Microstation DGN

This command converts drawing files between .DWG and .DGN file formats. This command is only available when running on IntelliCAD. When running on AutoCAD, the AutoCAD Import and Export commands can be used to convert .DGN files.

When converting DGN files to DWG file format, there are several processing methods. In all cases, multiple files can be selected and processed at a time. The Convert Files method creates DWG files from the selected DGN files. The DWG files automatically have the same file name as the source DGN files except with the different file extension of .DWG. The Insert as Block Reference method inserts the selected DGN files into the current drawing as blocks. The Insert as Entities method imports the entities from the DGN files into the current drawing. The Attach as External Reference brings the selected DGN files into the current drawing as Xrefs. The Overlay option for Attach as External Reference is the same except that this method is not nested.

For converting DWG files to DGN format, the Convert Files method creates DGN files from the selected DWG files. The Convert Current exports the current drawing. In both cases, the DGN files are automatically named after the source DWG files except with the different file extension of .DGN.



Pulldown Menu Location: File

Keyboard Command: dgnio

Prerequisite: A DWG or DGN to convert

Write Polyline File

This command creates a polyline file that contains the point data of the select polylines. The objects supported by this tool include polylines, arcs and lines. If you want to include text, you must use the Text Explode To Polyline command found in the Edit menu to convert the text to polylines before running this command. Several different output formats are supported.

The Carlson format (.PLN) is a text file format that is used by some Carlson commands and by machine control (Carlson Grade, Dozer 2000, GradeStar) for the plan view. Each polyline begins with a line of "POLYLINE, Color number, etc". Then the points for the polyline are listed on separate lines in X,Y,Z format.

The DTM and Idan formats create linework files for the DTM and Idan programs.

The KOF method creates a .KOF format file.

The MicroStation format (.txt) can be imported into MicroStation. This format has the coordinates as space delimited for each polyline point. There is an extra column with a 1 or 0 where 1 specifies the start of a new polyline.

The Moss format creates a INP file for the MXROAD/MOSS Genio program.

The Peabody format is a company specific format for Peabody Energy.

The Topcon format creates a Topcon LN3 file.

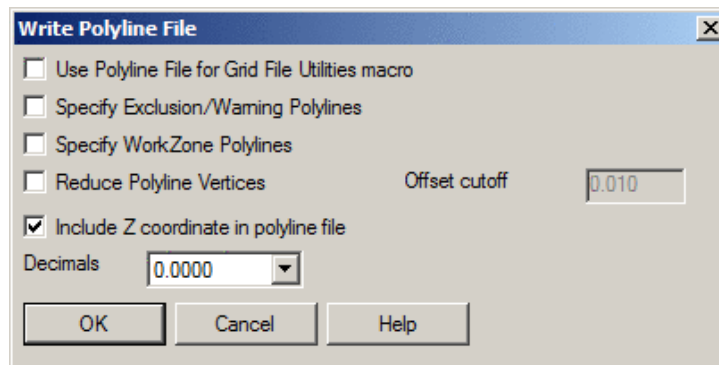
Note:

- The former **Google** (KML) output option has been moved to the dedicated Export Google Earth File command.

Prompts

Polyline file format [<Carlson>/DTM/Idan/MicroStation/MOSS/Peabody/Topcon]? Specify the desired output option by specifying the CAPITALIZED option or press Enter for the <default> option.

Polyline File to Write dialog: Create a new file or Append to Existing. If the **Carlson** option was selected, the following dialog then appears:



Use Polyline File for Grid File Utilities macro: When enabled, the option will write a polyline file that can be used with Grid File Utilities for inclusion/exclusion perimeters.

Specify Exclusion/Warning Polylines: When enabled, this option applies to machine control for warning areas.

Specify WorkZone Polylines: When enabled, this option applies to machine control for working areas.

Reduce Polyline Vertices: When enabled, this option applies the Reduce Polyline Vertices to the polyline vertices before writing the file..

Offset Cutoff: Indicate the allowable offset distance (essentially the *middle ordinate distance* of a 3-point arc) that would allow the middle vertex between two other vertex locations to be removed.

Include Z coordinate in polyline file: When enabled, this option controls whether the elevation(s) (or "Z" value) of the selected polyline vertices are written to the polyline file.

Decimals: Indicate the desired amount of precision for the coordinate values that should be written to the file.

Select polylines, lines and arcs to write.

FILter/<Select entities>: *Pick the entities to process press Enter when complete.*

Sample Polyline File:

```
POLYLINE, 51, 0, 0.0, CONT|V-STRM-PIPE  
5375168.9320, 3932304.7050, 0.0000  
5375193.3310, 3932211.6150, 0.0000  
POLYLINE, 150, 0, 0.0, CONT|V-BRKL  
5375026.8800, 3932090.0480, 962.8334  
5375062.3960, 3932105.7540, 961.5399  
5375075.5640, 3932115.7940, 961.1595  
5375079.0150, 3932128.0920, 961.1532  
5375081.6860, 3932159.7840, 961.6147  
5375086.6920, 3932195.6480, 962.6206  
etc.
```

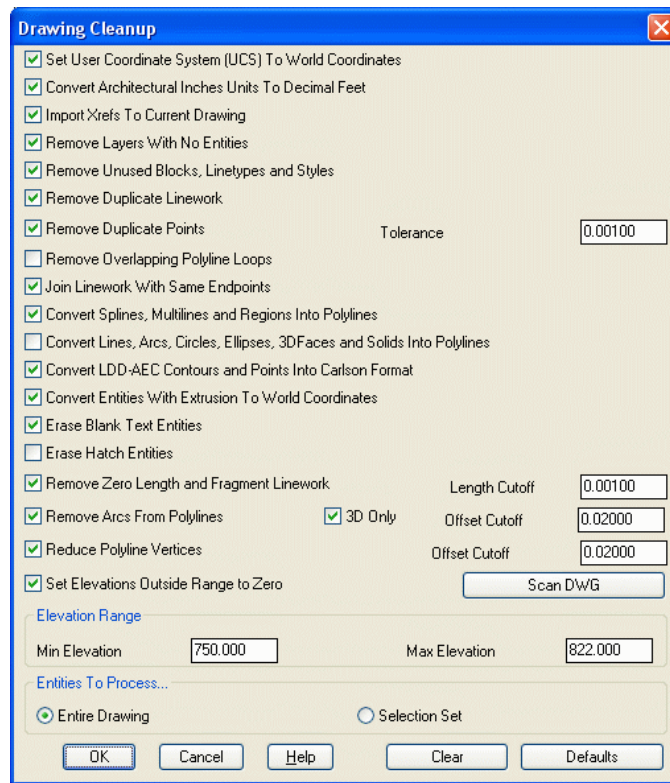
Pulldown Menu Location: File > Export

Keyboard Command: polywrite

Prerequisite: Polylines in the drawing

Drawing Cleanup

The Drawing Cleanup dialog box allows you to perform many functions that fix common errors, and it removes unnecessary data found in many drawing files. It also converts incompatible data into useful entities. This command offers many filters that audit the drawing file and allows you to select which options and settings you want to use. A report of the cleanup results will be displayed upon completion. Always save your file when the drawing cleanup routine is complete.



Set UCS to World Coordinates

This sets the UCS (user coordinate system) to the world coordinate system (WCS). Carlson works exclusively in the world coordinate system and there is no way to change this setting. In CAD, it is possible to change the coordinate system from WCS. If you receive a drawing in which the coordinate system is not set to world, click this on to restore the UCS.

Convert Architectural Inches Units To Decimal Feet

Drawings are sometimes in architectural units, i.e. inches, when the unit of measurement was intended to be in feet. This routine will change the units from inches into feet and then scale the drawing by 1/12.

Import Xrefs To Current Drawing

This routine allows you to import any 'found' external reference files (Xrefs) into the current drawing. If the path is not found, the Xref file will not be brought into the drawing. To set the Path for any unfound Xrefs, run Import Xref to Current Drawing under File.

Remove Layers With No Entities

Drawings work with a "BYLAYER" concept meaning that layer definitions define the drawing. For example, the layer named EOP might be used to display polylines at the Edge Of Pavement in the drawing. Many times extra layers get defined by a user but not used to display any objects. This function removes any layers defined in the drawing that are not being used.

Rename Layers With Wildcards

Layers with wildcard characters such as "*" can interfere with Carlson layer matching functions. This routine renames layers by replacing any wildcard characters with an underscore "_".

Remove Unused Blocks, Linetypes and Styles

This functions removes this unused information from the drawing.

Remove Unused Registered Apps

This function removes Registered Applications that are defined in the drawing but not referenced by any entities.

Remove Zero Length Linework

This function seeks out and removes any linework definition that have zero length. Point nodes are not removed.

Remove Duplicate Linework

This function finds any duplicate linework in the drawing and removes all but one set. The Tolerance setting controls how close the linework vertices must match.

Remove Duplicate Text

This function finds any duplicate text in the drawing and removes all but one set.

Remove Duplicate Points

This function searches the drawing (but not the .CRD file) for points with the same northing, easting and elevation. The tolerances for considering points to have the same coordinate are set to the right. To be counted the same coordinate, both the northing/easting and elevation must be within the tolerance distance.

Remove Overlapping Polyline Loops

Polylines that completely overlap themselves are broken into two different polylines.

Join Linework With Same Endpoint

This function finds common endpoints on linework on common layers with common elevations and joins the linework into a continuous polylines. This is very helpful for future selection sets.

Convert Splines, Multilines and Regions Into Polylines

Some CAD applications utilize Spline Object Definitions and Regions, Carlson utilizes basic polyline/polygon definitions. This function finds any Splines and/or Regions defined in the drawing and re-defines them as simple polylines or polygons.

Convert Lines, Arcs, Circles, Ellipses, 3DFaces and Solids Into Polylines

By converting Lines, Arcs, Circles, Ellipses, 3D Faces, and Solids into Polylines, you can use the variety of Polyline commands available in Carlson.

Convert LDD-AEC Contours and Points Into Carlson Format

Drawings created in the Land Development Desktop CAD program can contain special objects known as LDD-AEC contours that define their topographic contour display. This function locates those special objects and re-defines them as simple 2D polylines retaining their elevation values.

Convert Entities With Extrusion To World Coordinates

This function removes extrusion assigned to entities and puts the entities coordinates into true world coordinates.

Erase Blank Text Entities

This function removes any text boxes defined in the drawing that are not being used.

Erase Hatch Entities

Carlson offers many hatch display options, however hatch entities have no 3D value. This function removes all hatch entities in the original drawing to help reduce the size and clutter of the drawing file.

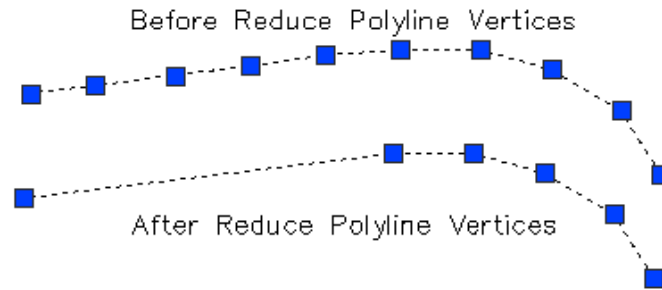
Remove Arcs From Polylines - Offset Cutoff

This function replaces arcs in polylines with a series of short chord segments. The purpose is to prepare the polylines for modeling since arcs need to be converted into segments to be part of the triangulation model. The density of chord segments is controlled by the offset cutoff. This cutoff represents how much the polyline can move horizontally. A smaller cutoff will result in more chord segments. The option for 3D Only controls whether only polylines at zero elevation or both zero and elevated polylines get processed. Sometimes you may want to leave the arcs in zero elevation polylines when these polylines represent road alignments and are not part of the surface model.

Reduce Polyline Vertices - Offset Cutoff

This function utilizes a pre determined offset amount and removes unnecessary polyline vertices that fall within the

offset amount.



Set Negative Polyline Thickness to Zero

This function sets the thickness property of polylines to zero for polylines with negative thickness.

Set Elevations Outside Range to Zero and Elevation Range

This function comes with a "Scan DWG" option that audits the elevation range in the drawing file. Once the minimum and maximum elevation range has been set, manually or by a scan, all objects that fall outside the set range are moved to elevation zero. All objects at zero elevation do not contribute to the 3D model.

Entities To Process...

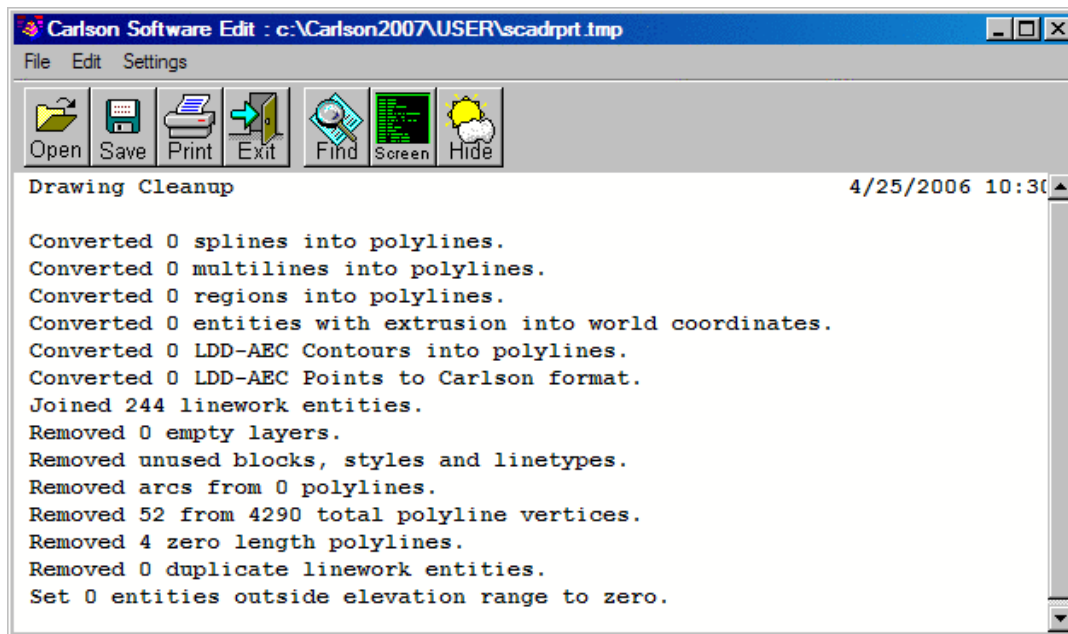
This allows you to run the command for the entire drawing or for a selected set.

Default

This allows you to return to the Carlson Drawing Cleanup default settings.

Final Report

This example report displays the results of drawing cleanup. Like all reports in Carlson, this report can be saved to a text file, sent directly to your printer, or pasted onto the screen as text entities.



Pulldown Menu Location: File

Keyboard Command: `dwg_cleanup`

Prerequisite: None

Save Drawing Named by Current Time

This command does a SaveAs of the current drawing with the dwg file automatically named using the current drawing name plus the current date and time. The date and time format is YearMonthDay.HourMinuteSecond. For example, when the current dwg is `job101.dwg` and the current time is July 4, 2020 at exactly noon, then this command will save using a file name of `job101_20200704_120000.dwg`.

Pulldown Menu Location: File > Drawing Utilities

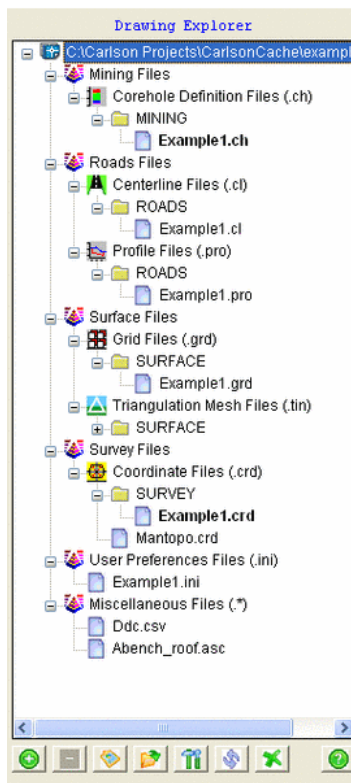
Keyboard Command: `savecurrent`

Prerequisite: None

Drawing Explorer

The Drawing Explorer command presents a list of all Carlson data files that are made in association with a drawing and are tracked in *DrawingName.ini*. If a drawing was not made in Carlson or does not have a companion .INI file, then Drawing Explorer will not display any files. The Drawing Explorer will also not show any data files if the drawing is not saved. Once data files are created such as a coordinate (.CRD) file, then Drawing Explorer will track these files. Drawing Explorer helps manage drawing-related data.

The Drawing Explorer is shown as a docked dialog on CAD window with files shown as "tree view" under different categories. These file categories are fully customizable and can contain multiple file types. The drawing name is shown as root of the tree view with file categories as its children. The file types associated with a category are listed as children of that category. The data files used with the drawing are listed under respective file type or in subfolders of the project folder specified using the Set Project/Data Folders command. The data files used as current files are shown with bold font.



The Drawing Explorer allows user to view/manage data files associated with the currently opened drawing by allowing him to add, remove, report, and change directory of these files. A mouse right-click can also be used to add and remove any data file/file type/file category from the Explorer.



The Add button allows adding a file under the category or file type that is currently selected in the tree view. If the drawing file name is selected, the user is allowed to add any type of file to the Explorer and file will be added to the corresponding category.



Removes the selected file(s) along with any "child" (subordinate) files from the drawing. The underlying file(s) are not physically deleted or removed from the hard-drive, they are merely removed from the Drawing Explorer.



Creates a report through the Report Formatter Options dialog box. The Report Formatter can be used to move to the *Available* entries on the left to the *Used* area on the right. When coupled with the Up/Down options to control the order, highly customized reports can be generated and saved for subsequent use.



Click the **Display** to obtain the report. The Change Directory option allows you to instruct Carlson Software to re-direct the location for files from an old folder location to a new folder location. This option is helpful if project data files are manually moved to a new folder location.



The Settings button allows you to create Categories of file types and assign data files to a particular Category and assign how project data files are presented in the Project Explorer.



The Refresh button re-reads the current Project file along with the various Drawing file settings and updates the Project listing appropriately.

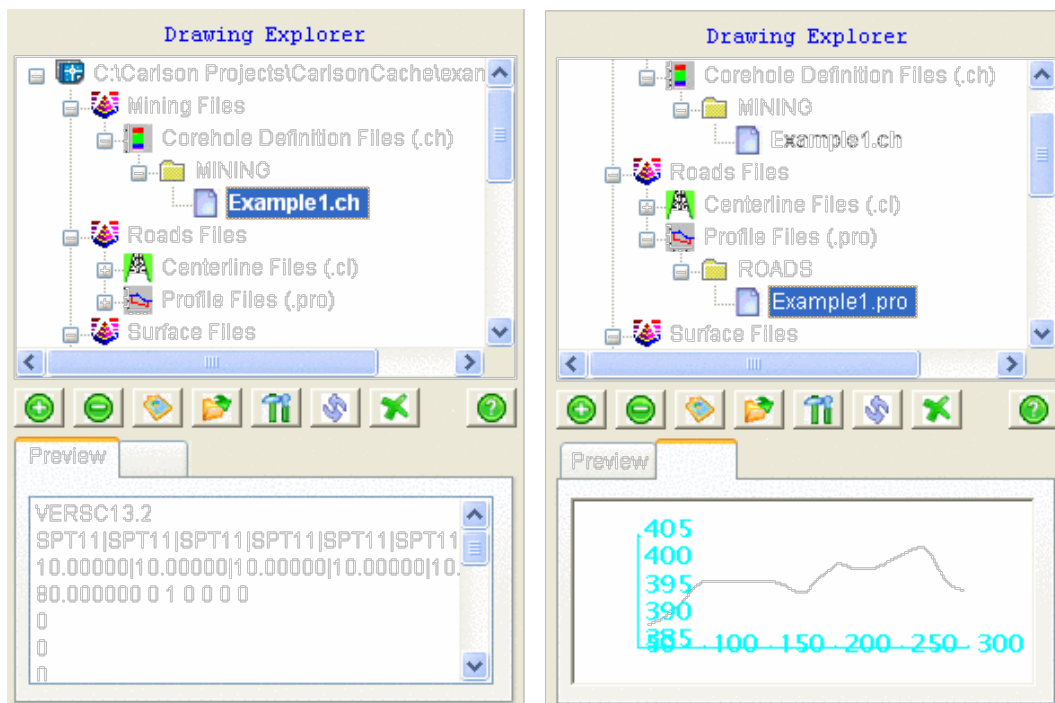


Exits from the Project Explorer command.



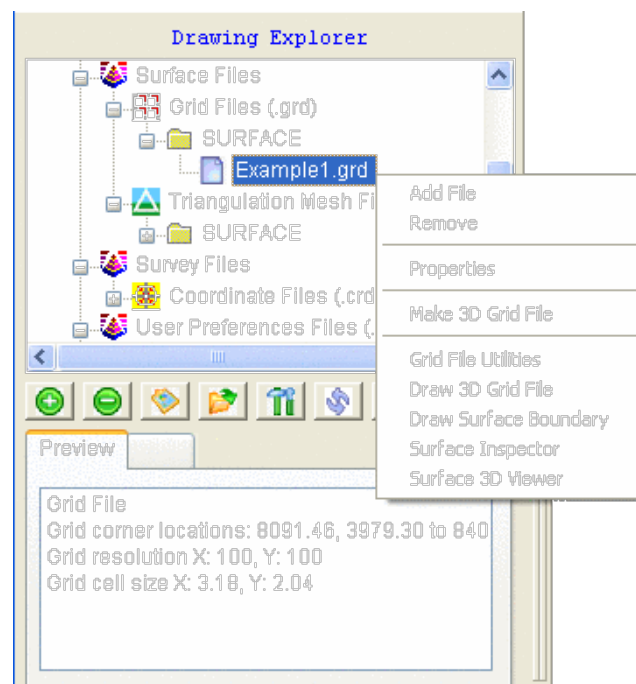
Displays the on-line help.

The option to show preview allows user to see the preview of currently selected data file in a small preview window at the bottom of the Explorer dialog.



The List Data Files settings are used in Project Explorer to list files according to drawing files in the project of by project.

Right-Click Command Execution: The Drawing Explorer also allows execution of functions associated with a file type. Right-clicking on any file type or data file brings up menu for the commands associated with that file. The right-click menu has a Zoom To function for many file types that have plan view coordinates such as centerlines. This Zoom To function zooms and pans the drawing to show the location of the highlighted file. For right-click commands that require the file during execution, the selected data file will be used to run that command. For example, in the figure below, *Example1.grd* will be used to run the Draw 3D Grid File command and the program will not prompt for the grid file to be drawn.



Double-Click Command Execution: Each file type has a default command to run when you double-click on the file. For example, TIN files run the 3D Viewer on double-click.

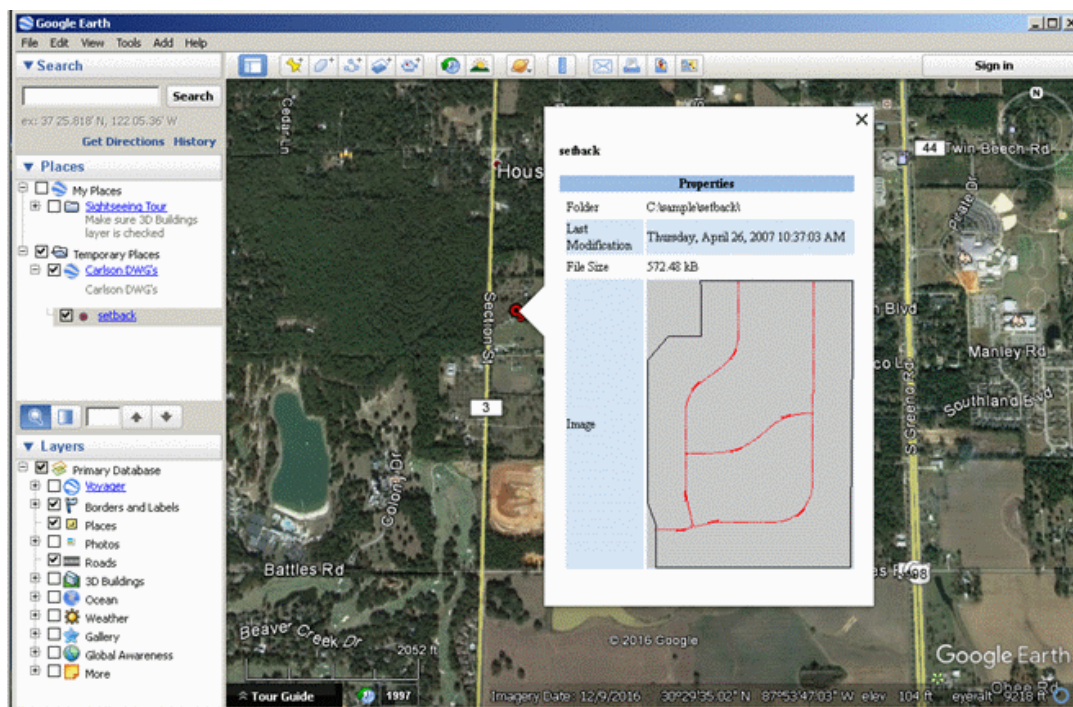
Pulldown Menu Location: File
Keyboard Command: dwgxplore
Prerequisite: None

Geolocate DWG Files

This command displays placemarks in Google Earth for DWG files. The command starts by prompting for a folder. Then all the DWG files in this folder and the sub-folders are used to make the map for Google Earth. In order to use a DWG file, the coordinate system must be set in the DWG using the Drawing Setup command. The command creates a Google Earth file called kml.dwg.kmz in the %appdata%\Carlson Software\version\platform\USER folder.

In Google Earth, the location of each DWG file is shown with a circle marker. When you click on a marker, the DWG properties are shown including the folder location, date, file size and graphic preview.

To load a DWG file from Google Earth, go to Tools > Options in Google Earth. In the Options dialog, go to the General tab and turn on Allow Access To Local Files and Personal Data. Then when you click on a placemark to open it up, you can double-click on the properties window to load that DWG file into CAD.



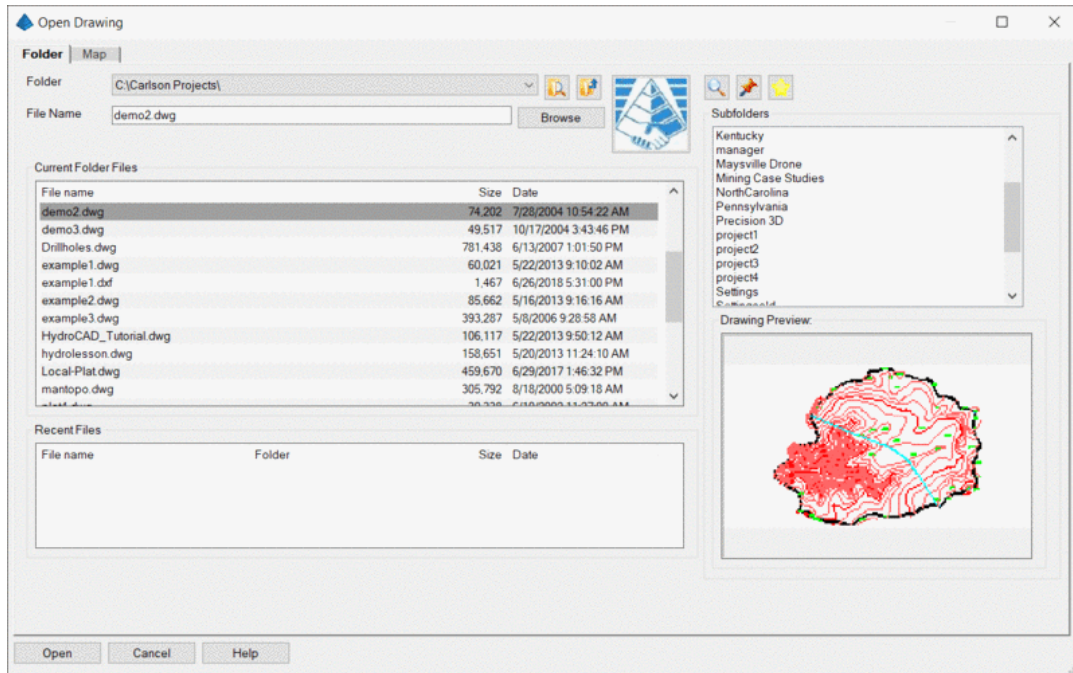
Pulldown Menu Location(s): File > Drawing Utilities
Keyboard Command: dwg2kml
Prerequisite: DWG files with defined coordinate systems

Open Drawing

This command allows you to open drawings either by browsing folders or from a map view. The map view is a visual way to select a drawing. From the map view, you can zoom into an area or type in an address into the Search

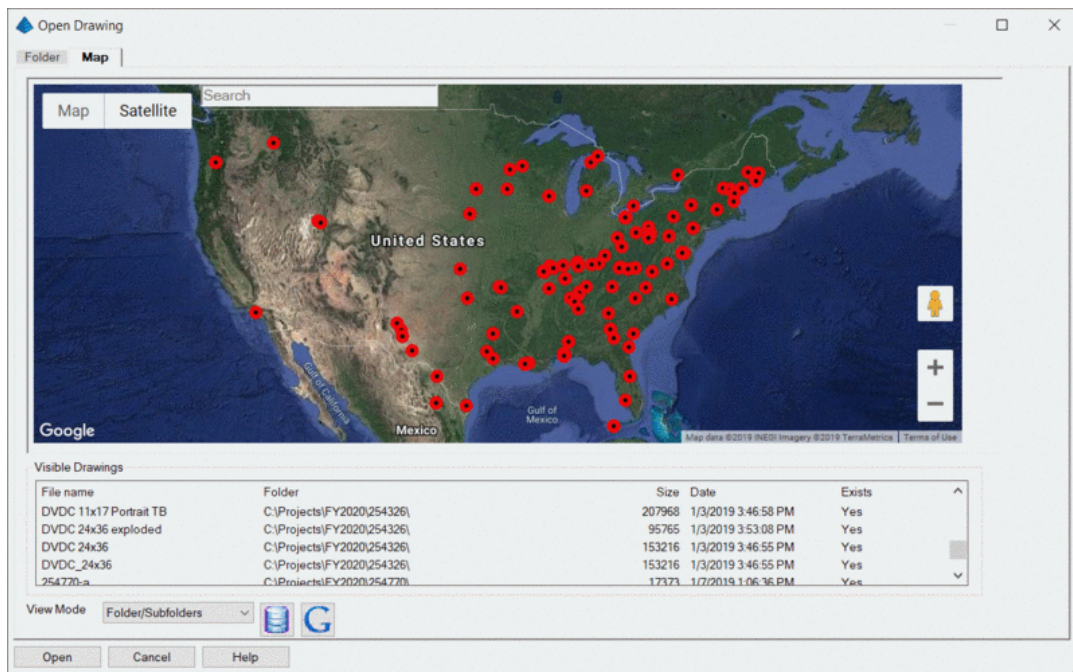
field.

Folder Tab



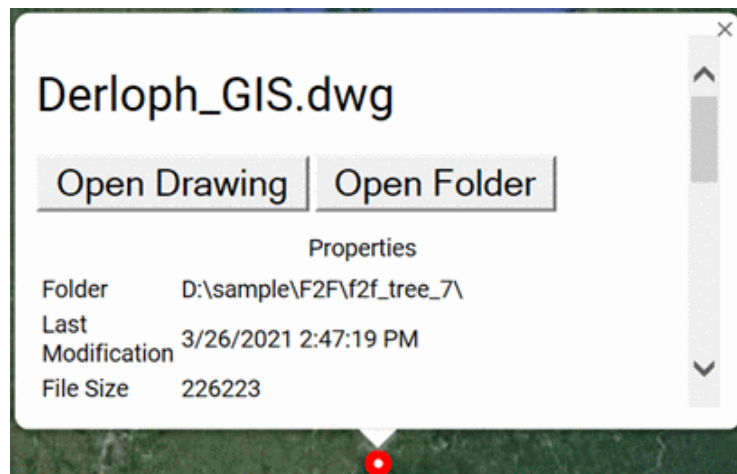
The Folder tab lets you find existing drawings by searching through folders and recent folder locations as in previous versions.

Map Tab



The Map tab displays a Map/Satellite interface that lets you pan, zoom, and browse georeferenced project drawings. In the Visible Drawings location are listed georeferenced drawing files that would appear in the current map and also follow the criteria as selected by the current View Mode listed at the bottom of the dialog box.

To select a drawing, pick on the dot icon on the map. The program will show the drawing name and properties. Pick the Open Drawing button to open the drawing in CAD or pick the Open Folder button to open the drawing folder in Windows Explorer.



For the Map tab to be available, the Use Drawing Database option in Carlson Configure > Project/Data Folders must be turned ON. Please refer to the Set Project/Data Folder command for complete information.

Data Source:

Folder: Displays georeferenced drawings in the current folder on the folder tab that also appear in the current map view.

Folder/Subfolders: Displays georeferenced drawings from the current folder (and all of its sub folders) on the folder tab that also appear in the current map view.

Drawing Database: Displays all the georeferenced drawings that would be visible in the current map display. Only georeferenced files may be added to the drawing database. If you pan or zoom the map so no georeferenced files would be visible, the last visible drawings will still display in the visible drawings list until a new drawing is found.

Recent Drawings: Displays recently opened georeferenced files that would be visible in the current map display.

Use Filter Criteria: Displays all drawings from the drawing database using the current filter criteria. (see Use Filter Criteria below)



Buttons: Next to the Data Source list, there are button icons.

Use Filter Criteria: Same as picking from the Data Source list. (see Use Filter Criteria below)

Drawing Database: Opens the Drawing Database dialog box.

Output to Google Earth: Writes database drawings out to Google Earth Pro (if installed on PC) as temporary places.

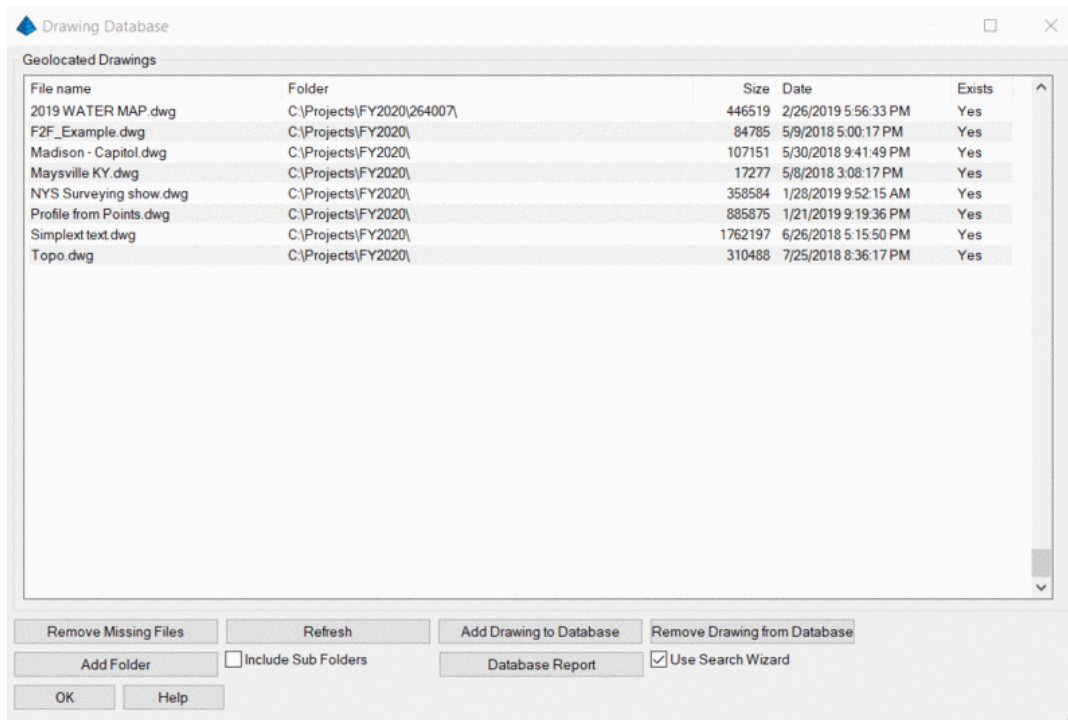
Column Options: Controls which drawing properties to display in the dialog list box.

Use Filter Criteria

Name	Folder	Size	Modified
2019 WATER MAP		446519	2/26/2019 6:56:33 PM
F2F_Example		84785	5/9/2018 6:00:17 PM
Madison - Capitol		107151	5/30/2018 10:41:49 PM
Maysville KY		17277	5/8/2018 4:08:17 PM
NYS Surveying show		358584	1/28/2019 10:52:15 AM
Profile from Points		885875	1/21/2019 10:19:36 PM
Simplext text		1762197	6/26/2018 6:15:50 PM

Drawings that have been added to the Drawing Database may be filtered down using the following criteria in addition to the current map view: Name and Type; Date (Modified) Range; and Size Range.

Drawing Database



Remove Missing Files: Removes files from the database that no longer exist in the location from whence they were added. If you want to remove a large amount of files at once, you can temporarily change the folder name of the root folder of many job files, and then run this command to remove the files, then rename the folder name change back to the original name.

Refresh: Re-reads the database and displays the current list of drawings.

Add Drawing to Database: Prompts you to browse and locate a new georeferenced file to add to the database.

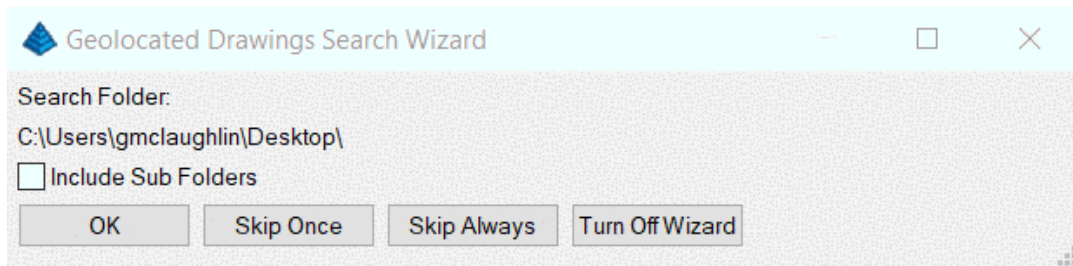
Remove Drawing from Database: Removes the highlighted drawing from the database.

Add Folder: Prompts for a folder to be selected. Any georeferenced drawings contained in that folder (and any included subfolders if this item is checked) are then added to the database. Duplicate named files from different locations or folders may reside together in the database.

Database Report: Reports on all files contained in the database. You can report items such as Full Name (Path and Drawing name); Date Modified; Size; Lat/Lon and if the file still exists in the original location.

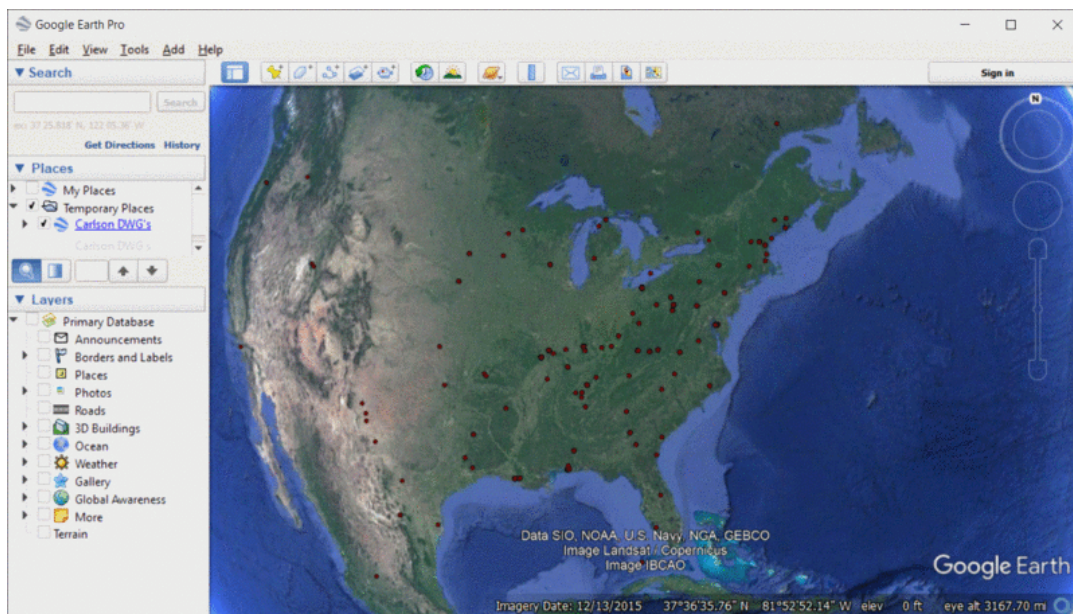
Use Search Wizard: Displays a wizard that will attempt to add georeferenced drawings from the current folder/subfolders to the database when the current folder is set to a new folder.

Geolocated Drawings Search Wizard



When you select a different folder from the folder tab to search or open drawings, then by switching back to the Map tab the Geolated Drawings Search Wizard displays if this is toggled ON in the Drawing Database. By using this search wizard, drawings will be automatically added to the database from the selected folder (and sub folders).

Output to Google Earth



Output to Google Earth writes all the database drawings out to Google Earth Pro (if installed on PC) as temporary places. These places may be added to 'My Places', be used to get directions or to display drawing information from within Google Earth.

Pulldown Menu Location: File

Keyboard Command: opendwg

Prerequisite: none

Draw Polyline File

This command draws polylines from the selected polyline file. This command supports the following formats: Carlson (.PLN), Agtek (.WRL), CAICE (.SRV), Digital Line Graph from USGS (.DLG, .OPT), Idan (.DIS), MicroStation (.TXT), MOSS (.INP, .PRN), Peabody (.PLY) and Topcon (.LN3 and .TXT). For formats that contain

only geometry without layer names, the polylines are drawn in the current layer.

Prompts

Polyline file format [`<Carlson>/Agttek/Caice/DLG/DTM/Idan/MicroStation/MOSS/Peabody/Topcon`]? *press Enter for Carlson default*

Polyline File to Read Dialog *select existing .PLN file*

Pulldown Menu Location: File->Import

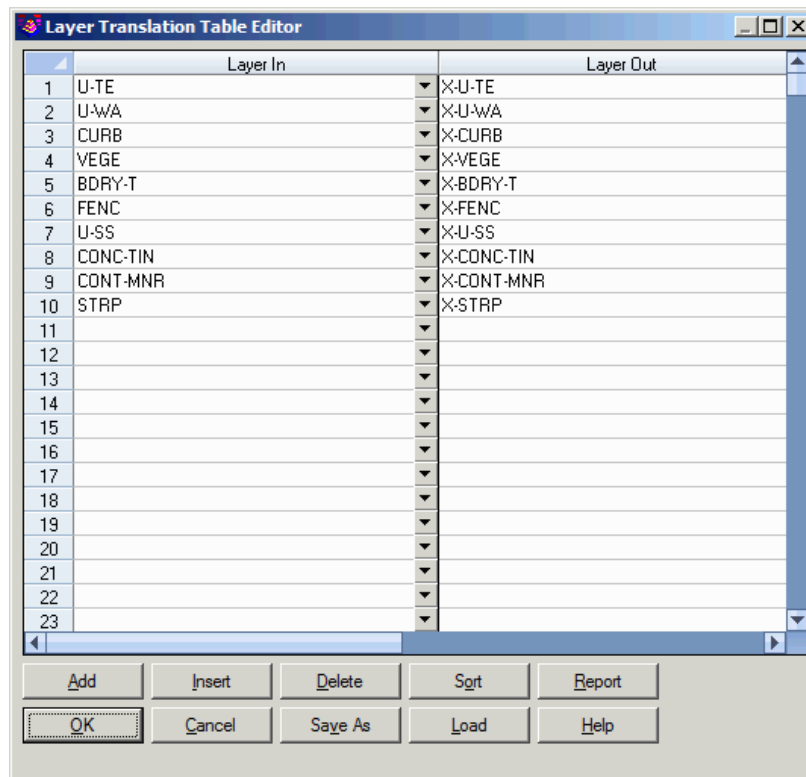
Keyboard Command: polydraw

Prerequisite: A polyline file

Translate Layers

This command renames layers using a lookup table with pairs of original and renamed layer names. This command can be used to convert the layers for a drawing from another source to match your layer standards. The layer names are entered in a spreadsheet. The Add, Insert, Delete and Sort buttons work on the spreadsheet rows. The Report button makes a report for the layer assignments. The SaveAs and Load functions store and recall the layer assignments to a .LTF file for sharing the settings or keeping different sets of layer assignments. The Add From Drawing reads the layers from the current drawing to put under the Layer In fields.

If you use Layer Library and define your target layers with their properties, then the Translate Layers command will use these layer properties. For the example below, if the X-U-TE layer is defined in the Layer Library, then X-U-TE is created using the color, line type, transparency and plot style from the Layer Library.



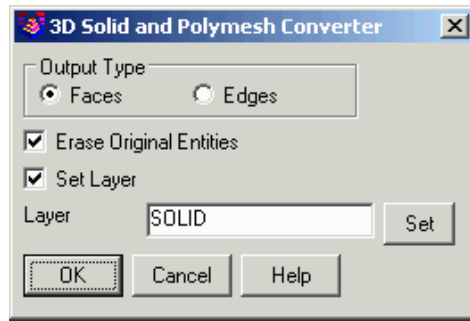
Pulldown Menu Location: File > Drawing Utilities

Keyboard Command: translayers

Prerequisite: None

3DSolid and Polymesh Converter

This command converts 3DSolid and Polymesh entities into 3D Faces or Edges. The Set Layer option allows you to put the new 3D Faces or Edges in a specified layer. Otherwise the layer of the original entities is used.



Prompts

Converter dialog

Select entities to process.

Select objects: *select 3DSolid or Polymesh*

Pulldown Menu Location: File > Drawing Utilities

Keyboard Command: solconvert

Prerequisite: 3DSolid or Polymesh entities

Remove XData

This command removes the xdata (Extended Entity Data) from the selected entities. Many Carlson routines add xdata to entities in order to add extra program specific information to them. Carlson programs use the xdata to make entities more intelligent. For example, when you draw a centerline (.cl) as a polyline, xdata is attached to the polyline that stores the reference of the .cl file name. Then if you double-click the polyline, then the program can read the xdata to know the polyline is a centerline and launch the centerline editor. By removing the xdata, the entities revert to regular CAD entities which is useful if you want to detach these entities from the program links.

Prompts

Select entities to remove extended entity data from.

Select objects: *pick the entities*

Pulldown Menu Location: File > Drawing Utilities

Keyboard Command: xxdata

Prerequisite: Entities with xdata

Remove Civil 3D Custom Objects

This command removes all Civil 3D custom objects (AEC objects) from the current drawing so that the next time the drawing is opened, AutoCAD will not load the AEC functions. The AEC objects can be either erased or exploded into standard AutoCAD entities. The cleaned drawing is saved to a new dwg file.

Prompts

Erase or Explode AEC entities during conversion [<Erase>/Explode]? *press Enter*
Save dwg selection dialog

Pulldown Menu Location: File > Drawing Utilities

Keyboard Command: aeccleaner

Prerequisite: Drawing with AEC data

Remove Reactors

This command removes the reactor links from the selected points, text, polylines and lines. This disables the links for points to the coordinate (.CRD) file, annotation with linework and linework with points. Note that in *General Settings* there is a section called Object Linking. This is the specific section that contains the options for creating these reactors to the drawing entities. Reactors can be turned off for entities created later by clicking off the four link options in *General Settings*. To get to this dialog go to *Settings > Configure > General Settings*.

Prompts

Select entities to remove reactors from:

Select objects: *pick the entities*

Pulldown Menu Location: File > Drawing Utilities

Keyboard Command: delreact

Prerequisite: Entities with reactors

Remove Groups

This command is used to "ungroup" selected entities that, prior to using this command, were part of a group. For our purposes, we might more specifically be referring to Carlson's Point Entity Grouping feature. A group is a named selection set of objects. This routine removes selected entities from groups. It is especially useful when dealing with our Carlson points.

More on Point Entity Grouping: As mentioned in the Points chapter, remember that for each point, the point attribute block, node, and symbol can be bound together. This means that if you choose to use the *Move* command (or other CAD tools) the entire collection moves together. This is done using the grouping functionality in AutoCAD or IntelliCAD. To disable this system altogether, go to *Configure*, choose *General Settings*, and turn off the toggle for Group Point Entities. If you need to temporarily disable grouping in a drawing, you can use the AutoCAD toggle for grouping, which is Ctrl-A. Holding down the Ctrl key, and pressing the letter A on the keyboard, activates this two-way toggle, with the current status echoed to the command prompt area.

Prompts

Select entities to remove from groups.

Select objects: select entities

Pulldown Menu Location: File > Drawing Utilities

Keyboard Command: rmgroup

Prerequisite: Entities in group(s)

Unlock Attributes

This command turns off the lock flag for attributes of the selected block entities.

Prompts

Select attributes to unlock.

Select objects: *pick the entities*

Pulldown Menu Location: File > Drawing Utilities

Keyboard Command: unlockatt

Prerequisite: Blocks with locked attributes

Record Script, End Script, Run Script

These commands process a sequence of commands. First, the Record Script command creates a script file (.scr). When the Record Script is active, the commands that you run and the input data is written to the script file. The End Script command stops the script recording. Then the Run Script command can be used to reprocess the commands and input from the script file. The script file is a text file that you can edit with any text editor such as Notepad or the Display-Edit File command. You can edit input data files as well as the commands.

The script supports the following commands: Draw-Locate Points, Inverse, Traverse, Sideshot, Bearing-Bearing Intersection, Bearing-Distance Intersection, Distance-Distance Intersection.

These scripts can be useful for batch processing a repetitive input. Also, the script can be used as a backup of data input for reprocessing after editing input values.

In the script file, the user-input values are after the "=>". Anything on a line that is after a ";" is a comment. The command prompts are included in the script file for easier reading. Here is a sample SCR file:

```
c:inverse ; Command: INVERSE
Traverse/SideShot/Options/Arc/Pick point or point number: =>26
Traverse/SideShot/Options/Arc/Multiple/Pick point or point number: =>24
Traverse/SideShot/Options/Arc/Multiple/Pick point or point number: =>SS
Exit/Help/Options/Points/Line/Traverse/Inverse/<Angle-Bearing Code <7>: =>
Enter Angle (dd.mmss) <90.0000>: =>45.1515
Points/<Distance>: =>59.2
Enter Zenith Angle (dd.mmss) <90.0000>: =>90.1234
Enter Point Description <>: =>IP
Exit/Help/Options/Points/Line/Traverse/Inverse/<Angle-Bearing Code <7>: =>e
```

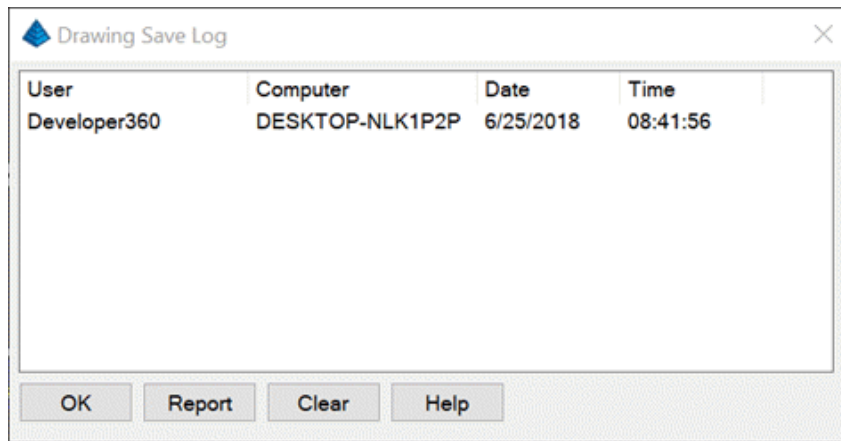
Pulldown Menu Location: File > Scripts

Keyboard Command: recscr, endscr, runscr

Prerequisite: None

Drawing Save Log

The command reports the save history for the current drawing including the date, time, user and computer name for each save. This save history is only recorded when the Generate Drawing Save Log option is turned on in the Carlson Configure > Getting Settings.



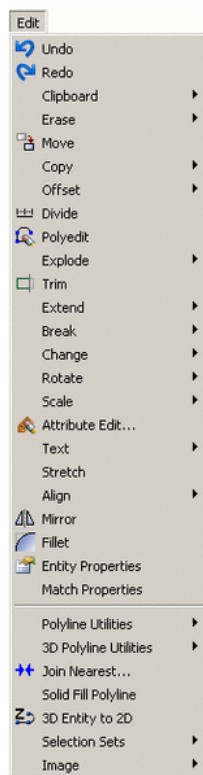
Pulldown Menu Location: File > Drawing Utilities

Keyboard Command: dwg_save_log

Prerequisite: Turn on Generate Drawing Save Log in Carlson Configure

Edit Menu

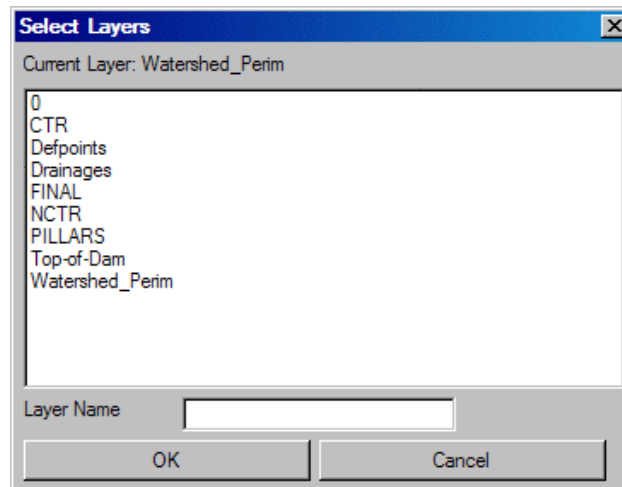
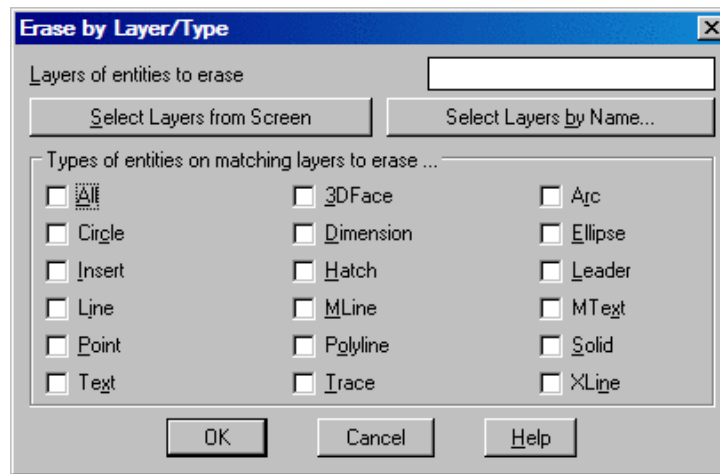
In addition to powerful CAD engine editing commands, the Carlson Edit menu has the additional commands which are explained in this section. Commands that are pure AutoCAD or IntelliCAD are not detailed here. They can be found in the CAD manual.



Erase by Layer

This command will ERASE all the entities on the specified layers but will not delete these layers from the drawing. The command prompts for the layer name to erase and then erases all entities on that layer. In addition to typing in

the layer name, you can also specify a layer to delete by picking an entity on that layer. To select layers by picking, first click the **Select Layers from Screen** button and then select the entities on the layers to be deleted. The **Select Layers by Name** button allows you to choose a layer name from a list of layers in the drawing. You can also specify which types of entities to erase. For instance, if you have both linework and points on the same layer and you want to erase only the linework, you can click off All and check Line and Polyline. The **Save** and **Load** buttons save and recall the layer names.



Pulldown Menu Location: Edit > Erase

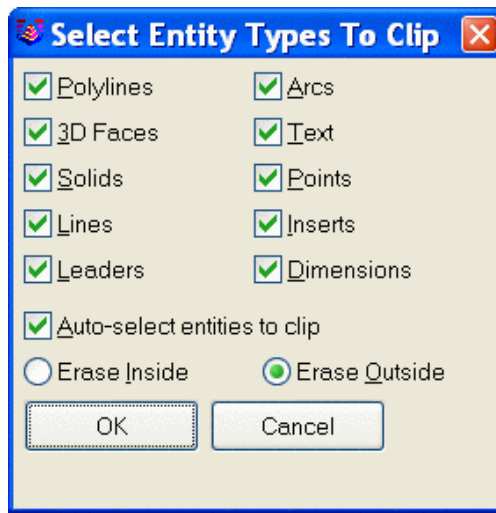
Keyboard Command: ldel

Prerequisite: Something to erase

Erase by Closed Polyline

This tool is used to clean up drawing geometry at the extents of a polyline boundary. It provides options to erase adjacent geometry as well as trim geometry crossing the fence of the polyline.

First, select the boundary polyline. Only one can be selected. Designate the desired options in the following dialog. The top section of the dialog allows you to toggle which object types should be affected by the operation. Note that some of the objects, such as text and inserts, cannot be trimmed.



In the middle of the dialog is a toggle that determines whether to prompt for objects to process. If you want to isolate the drawings contents to that of the selected polyline, turn this toggle on. Note that all geometry in the drawing is effected, even geometry that is outside of the current viewport. Many users will prefer to turn this toggle off, so that they can be prompted to manipulate the geometry.

The bottom row allows you to choose whether to erase all the entities on the inside or outside of the polyline.

Pulldown Menu Location: Edit > Erase

Keyboard Command: erasepline

Prerequisite: Entities and a closed polyline

Erase Outside

This command erases all the entities outside of a user specified window. This can be useful if you somehow place entities way outside your drawing limits and want to easily erase them.

Prompts

Pick 1st corner of window to erase outside of: *Pick point location*

Pick 2nd corner: *Pick second point location*

Pulldown Menu Location: Edit > Erase

Keyboard Command: eraseout

Prerequisite: Entities to erase

Temporary Erase

This command erases the selected entities while keeping track of their data to allow restoring them. To unerase the entities, simply run the command again. The program keeps track of the erased data only during the current drawing session. If you exit the drawing, the entities cannot be restored when the drawing is opened again.

Prompts

Select entities for temporary erase.

Select objects: *pick entities to erase*

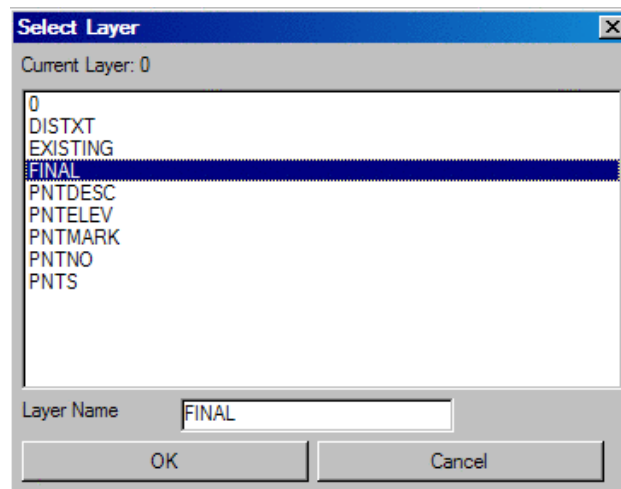
Pulldown Menu Location: Edit > Erase

Keyboard Command: terase

Prerequisite: Entities to erase

Copy To Layer

This command is used to copy a selected entity or entities and put the copy in a specified layer. Once copied to the chosen layer the entity or entities will take on the characteristics of that layer (color, linetype, etc.).



Prompts

Select entities to copy.

Select objects: *select entities*

Select Layer dialog *select a layer from list and click OK*

Pulldown Menu Location: Edit > Copy

Keyboard Command: copy2layer

Prerequisite: Entities to be copied

Copy Polyline Section

This command is used to copy a portion of a polyline, at specified points, and put the copied portion onto another layer. The portion of existing polyline that is being copied still remains as part of the original entity (with no break), while the new portion, with its chosen layer designation, is a new polyline.

Prompts

Select polyline to copy: *Pick a polyline*

Select first break point along polyline: *Pick location on the polyline*

Select second break point along polyline: *Pick the second location on the same polyline*

Layer name <CTR>: wall

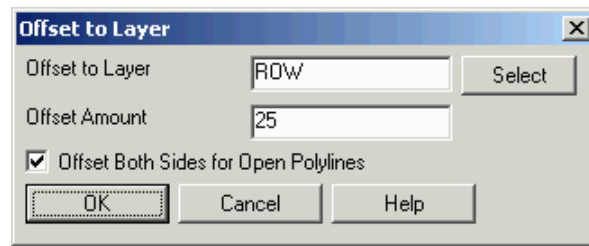
Pulldown Menu Location: Edit > Copy

Keyboard Command: copy_pl

Prerequisite: Polyline to be copied

Offset To Layer

This is a command to offset a polyline and put the offset polyline into a separate layer from the original polyline. The Offset Both Sides For Open Polylines option can be used to create easement perimeters. For example, you can pick a pipe polyline and this option will offset to both sides to create an easement around the pipeline.



Prompts

Offset To Layer dialog

Select object to offset: *pick a polyline to offset*

Specify point on side to offset: *pick a point*

Select a polyline to offset (Enter to end): *press Enter*

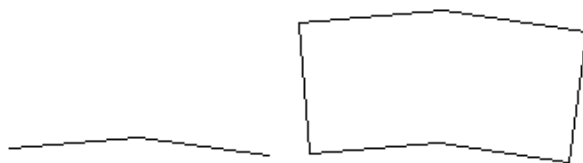
Pulldown Menu Location: Edit > Offset

Keyboard Command: offset_layer

Prerequisite: Linework to offset

Offset to Area

This command offsets a polyline by a distance that results in creating the specified target area. The source polyline should represent the frontage on the area. There is an option to connect the sides between the source and offset polylines to make a closed polyline.



Before and after Offset to Area

Prompts

Pick line or polyline to offset: *pick a polyline*

Select side to offset: *pick a point on the offset side*

Keep existing polyline [Yes/<No>]? *press Enter*

Create closed polyline [<Yes>/No]? *press Enter*

Acres/<Enter target area (s.f.)>: *90000*

Pulldown Menu Location: Edit > Offset

Keyboard Command: offset_area

Prerequisite: polyline to offset

Multiple Offsets

This command applies the same offset multiple times in series from the original polyline.

Prompts

Specify offset distance <20.0000>: 25

Enter Number of Repetitions <1>: 3

Select object to offset or <exit>: *pick a polyline to offset*

Specify point on side to offset: *pick a point*

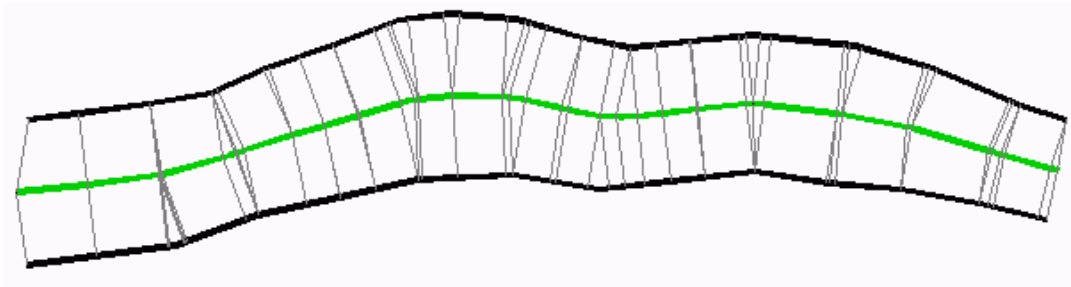
Pulldown Menu Location: Edit > Offset

Keyboard Command: offset_mult

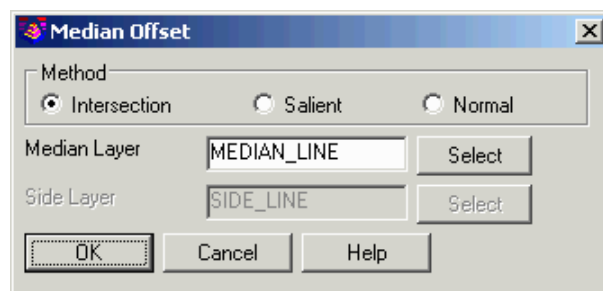
Prerequisite: Linework to offset

Median Offset

This command creates a median polyline that is offset halfway between two existing polylines. For example, this command can be used to get the center of a river given polylines for the left and right river banks. The intersection method finds where the offsets from the two reference polylines intersect. The Salient and Normal methods come from the CAPD program originally developed by the BLM.



Prompts



Select the First Polyline: *pick a polyline*

Select the Second Polyline: *pick a polyline*

Draw Chords of Parabolas [Yes/<No>]? *press Enter*

Draw Perpendiculars for Check [**<Yes>/No**]? *press Enter*

Pulldown Menu Location: Edit > Offset

Keyboard Command: medoff

Prerequisite: two polylines to offset

Variable Offset

This is a command to offset a polyline, with different offset amounts for each polyline segment of the same polyline. The offset distances can be variable, and you choose between a Line or a Point method at the command line.

Prompts

Vary offsets by line segments or at points [**<Line>/Point**]? *press Enter*

Select a polyline to offset (Enter for none): *pick polyline*

Select side to offset: *pick a point on the side to offset to*

As you go from segment to segment, you can enter in different offset values for each line segment.

Enter the segment horizontal offset **<0.000>**: 56

Enter the segment horizontal offset **<56.000>**: 33

Enter the segment horizontal offset **<33.000>**: 12

Select a polyline to offset (Enter for none): *press Enter*

Pulldown Menu Location: Edit > Offset

Keyboard Command: VOFFSET

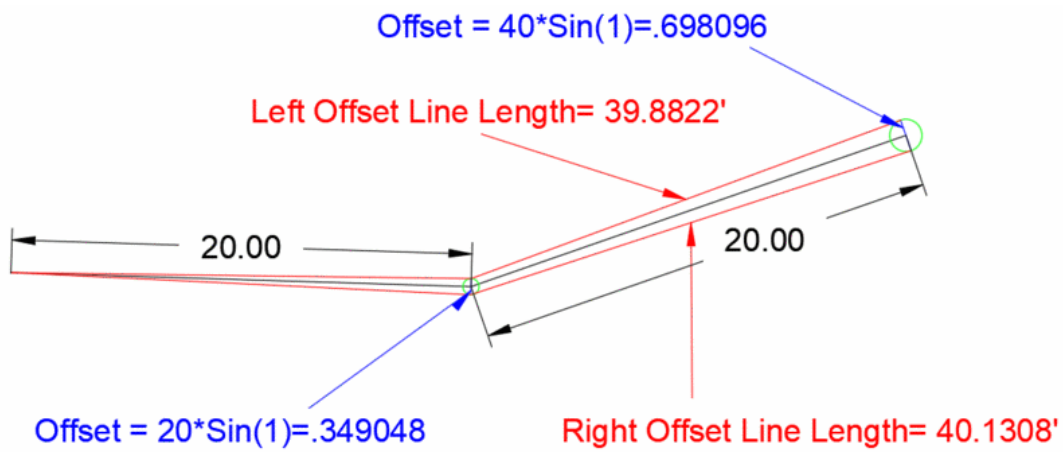
Prerequisite: A polyline to offset

Drift_Offset

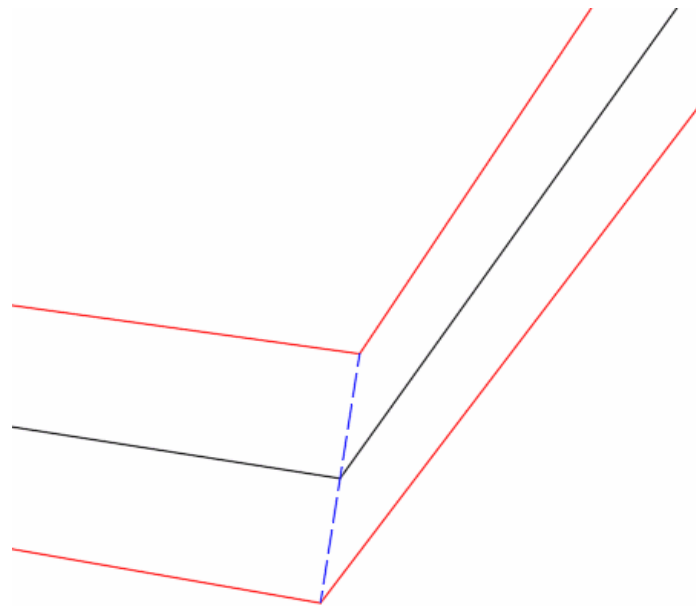
This command draws two offset polylines that offset as a function of the cumulative distance of the polyline. That is, at the beginning of the original polyline, the offset distance will be zero. At the end of the polyline, the offset distance will be equal to the total length of the original polyline multiplied by an offset factor, as described by the below formula:

$$\text{Offset} = \text{Polyline.Length} * \sin(\text{angle})$$

After selecting the polyline and entering the angle, the offsets will be calculated for each vertex along the polyline. It is important to note that the offset polylines are not a simple angle rotation of the original polyline. An example is shown below.



Because the offsets are calculated only at the vertices of the original polylines, sharp angles in the original polyline will produce offset polylines that can actually get closer to the original polyline rather than farther away, as shown below. Note that the offset distances are perpendicular to the original polyline.



It is important to note that although this command can offset either a 2D or a 3D polyline, the resulting offset polylines will only be drawn as 2D polylines.

Pulldown Menu Location: Edit
Keyboard Command: offset_drift

Block Explode

This command retains the values of attributes when a block is exploded. The standard *Explode* command changes the attribute values back to the attribute type. For example, using *Explode*, a Carlson point block would become PNTNO, PNTELEV, PNTDESC. *Block Explode* would keep the point attribute values, such as 10, 1000.0, EP. The

layer names of the exploded block attributes can be either the insert layer of the parent block or the original attribute layers from the block definition.

Pulldown Menu Location: Edit

Keyboard Command: explode2

Prerequisite: A block to be exploded

Extend to Intersection

This command extends the end points of two lines and/or polylines, at the same time, to their intersection point.

Prompts

Select first line or polyline to extend: *pick a line or polyline*

Select second line or polyline to extend: *pick another line or polyline*



Before Extend to Intersection



After Extend to Intersection

Pulldown Menu Location: Edit > Extend

Keyboard Command: extint

Prerequisite: Two lines or polylines

Extend Arc

This command extends an arc entity.

Prompts

Pick arc to extend: *select an arc entity*

Break Arc at Extension [Yes/<No>]? *N* Answering *Yes* will create a new arc starting at the end of the existing arc.

Enter or pick the distance to extend: *5* This extends the arc 5 units

Enter or pick the distance to extend ('U' to Undo): *press Enter to end*

Pulldown Menu Location: Edit > Extend

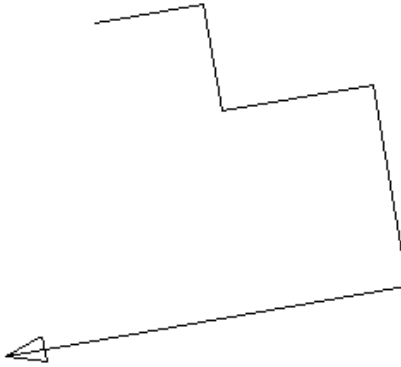
Keyboard Command: extarc

Prerequisite: An arc

Extend by Distance

This command extends a line or polyline, or creates new lines or polylines off of an existing one. By specifying a distance, a new segment of the line or polyline can be drawn from the current position. The current position and direction along the line or polyline is indicated by an arrowhead. Extend by Distance starts by selecting an existing line or polyline. Initially, the current position will be the closest vertex to where the line or polyline was selected. Extending from the endpoint of a polyline will add a new point to that polyline, while extending from any other point will create a new polyline.

There are two modes of operation: draw mode (D) and move mode (M). When in draw mode, extending will draw line or polyline segments. In move mode, the current position arrowhead can be moved without drawing segments. The orientation of the current position arrowhead can be changed with the Right, Left, and Angle commands.

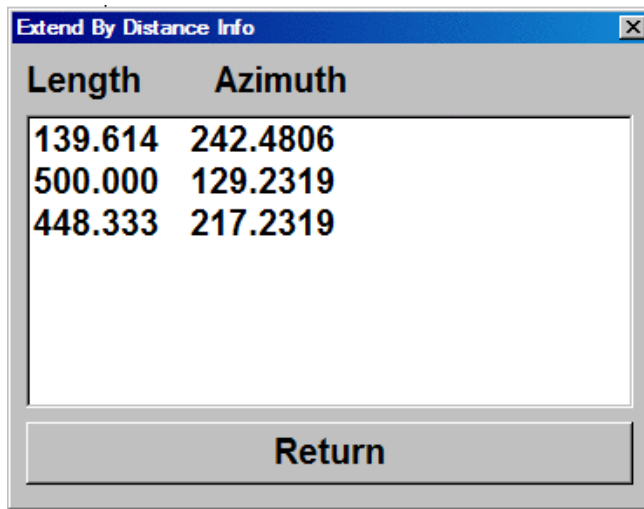


The second prompt for this command offers numerous options in the form of key letters. These key letters are listed below along with their full names and actions. The list of the Extend by Distance commands are:

- # - Number:** Distance to draw or extend
- A# - Angle change:** Rotates pointer by specified number of degrees
- A - Align:** Rotates pointer to align with segment
- B - Bearing:** Sets pointer direction by bearing in format: Qdd.mmss with Q- quadrant, d-degrees, m-minutes, s-seconds (e.g. 130.1005 is NE 30 degrees, 10 minutes, and 5 seconds)
- C - Close:** Closes the polyline
- D - Draw Mode:** Actions draw or extend the line or polyline
- E - Extend to Edge:** Extends to intersection with a selected line or polyline
- I - Input mode:** Toggles distance input between decimal feet and feet-inches
- L - Left rotate:** Rotates counterclockwise 90 degrees
- M - Move Mode:** Actions only move the pointer
- N - Next:** Moves pointer forward to next point
- O - Open:** Opens the polyline
- P - Previous:** Moves pointer backward to previous point
- R - Right rotate:** Rotates clockwise 90 degrees
- S - Switch:** Reverses pointer direction
- T# - Total distance:** Sets current segment to specified distance
- U - Undo:** Undo the last Extend by Distance command
- Z - Zoom mode:** Toggles auto-zoom between on/off
- ? - Info:** Displays lengths of current polyline

H - Help: The Help option also displays this Extend by Distance Commands list.

Press <Enter>: Ends the routine



The result of using the Info (?) feature

Prompts

Select line or polyline to extend: *select line or polyline near the place to extend*

Enter or pick distance to draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,Z,?,Help): 50 The line is extended by 50 units.
Use the Pick option to pick a distance.

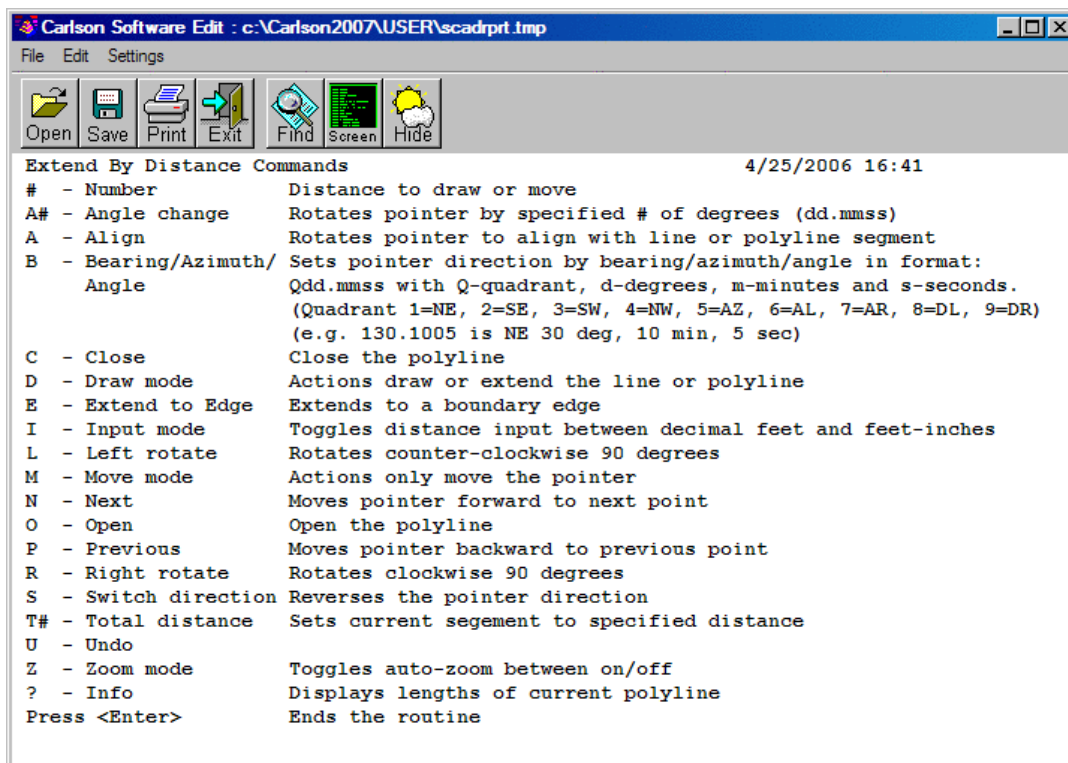
Pick/Horizontal Distance to Extend ([Enter] for new line): R Rotate right 90 degrees.

Enter or pick distance to draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,Z,?,Help): 50 The line is extended by 50 units.
Use the Pick option to pick a distance.

Enter or pick distance to draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,Z,?,Help): *press Enter*

Extend another (<Yes>/No)? No

Note: R50 and L10 can be used to go right 50, left 10, etc.



The result of using the Help (H) option

Pulldown Menu Location: Edit > Extend

Keyboard Command: extender

Prerequisite: An existing line or polyline with at least one segment from which to start.

Break by Crossing Polyline

This tool is used to break drawing geometry at the edge of a polyline boundary. It provides options to change the layers of the interior and exterior geometry after it is broken.

First, select the boundary polyline. Only one can be selected. Then select the polylines and lines to be clipped. You will be prompted for options on specifying the layers for the newly broken geometry. Respond with a "Y" if you want to specify a new layer, then enter the new layer name. If the layer name does not exist, it will be created.

Prompts

Select the clip edge polyline: *pick a closed polyline*

Select the polylines and lines to be clipped.

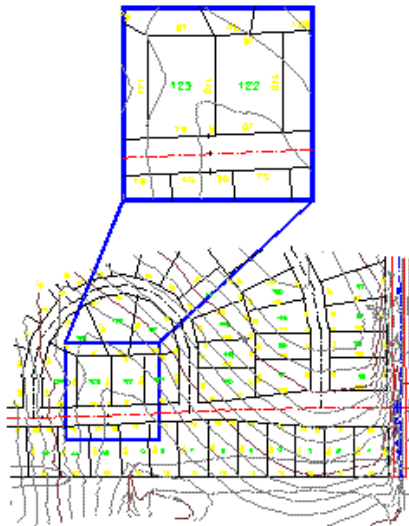
Select Objects: *pick the entities to break*

Specify layer names for Inside segments (Yes/<No>)? *Yes*

Enter a layer name for the Inside segments <0>: *press Enter*

Specify layer names for Outside segments (Yes/<No>)? *Yes*

Enter a layer name for the Outside segments <0>: *Final*



Pulldown Menu Location: Edit > Break

Keyboard Command: clipline

Prerequisite: A closed polyline

Break Polyline at Specified Distances

This command allows you to pick a polyline and break it at a specified distances along the polyline. Following the prompts below, the beginning of the polyline in the illustration was broken into three 55-foot segments.

Prompts

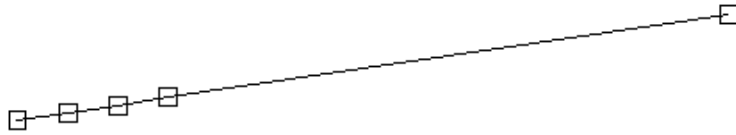
Select polyline to break: *select polyline*

Total Distance: **779.429** This is the length of the polyline reported.

Distance Along Polyline For Break: *55.0*

Distance Along Polyline For Break (Enter to end): *110*

Distance Along Polyline For Break (Enter to end): 165
Distance Along Polyline For Break (Enter to end): *press Enter*
3 polyline breaks created.



Pulldown Menu Location: Edit > Break

Keyboard Command: breakpl

Prerequisite: A polyline

Break at Intersection

This command will break a line, arc or polyline at the intersection of another line, arc or polyline. In many cases this command is used in conjunction with the *Area by Lines & Arcs* command. In order to get the correct area of a figure, it is often necessary to break it from adjoining lines.

Prompts

Select Line, Arc, or Polyline to Break

Select object: *select object to break*

[int on] Pick Intersection to break at: *pick intersection point*

Pulldown Menu Location: Edit > Break

Keyboard Command: breakat

Break at Selected Point

This command breaks the selected linework at the cursor position where you select the linework. This break method has the minimum prompting to create a break since it has only one prompt that is used both to select the entity and specify the break point.

Prompts

Select Line, Arc, or Polyline at break point: *select linework*

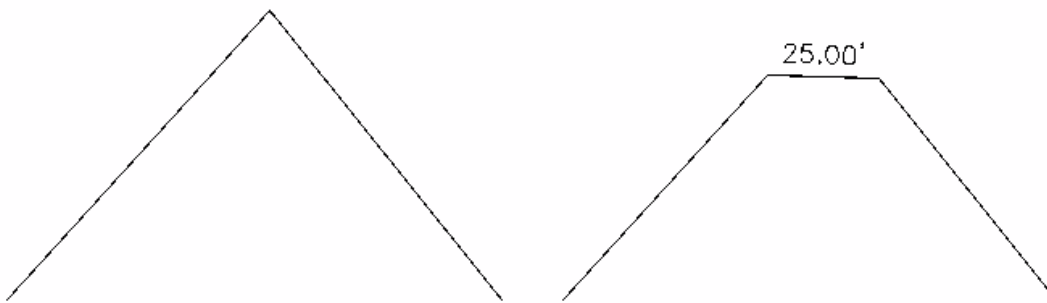
Pulldown Menu Location: Edit > Break

Keyboard Command: breakat1

Prerequisite: A polyline

Chamfer By Chord Length

This command chamfers by a specified chord length. You can have a single chord or fit in a number of chords.



Before and after Chamfer By Chord Length of 25

Prompts

Select Polyline to Chamfer: *pick 1st line segment*

Select Adjoining Polyline Segment: *pick adjoining line segment*

Chord Length: 25

Number of chords <1>: *press Enter*

Pulldown Menu Location: Edit > Chamfer

Keyboard Command: chord_chamfer

Prerequisite: Two connected line segments

Change Elevations

This command will change the elevation of selected entities. It can move the entity to a specified elevation from its current elevation (absolute) or do a differential change by adding or subtracting a value from its current elevation. If Carlson points are selected, their attribute text and z axis coordinate are changed.

Prompts

Ignore zero elevations (<Yes>/No)? *press Enter* If you answer *No*, then entities with elevation 0 will be changed.

[A]bsolute or [D]ifferential Change <A>: A

Elevation to change to: 125 By using the Absolute option all entities selected are changed to the elevation 125.

Select Entities for elevation change.

Select objects: C

First corner: *pick a point*

Other corner: *pick a point*

Select objects: *press Enter*

If Carlson points are selected, the command warns:

This command DOES NOT change the elevations in the Coordinate file!

Use Coordinate File Utilities menu option F to update the file.

Pulldown Menu Location: Edit > Change

Keyboard Command: chgelev

Prerequisite: Something to change

Change Attribute Style

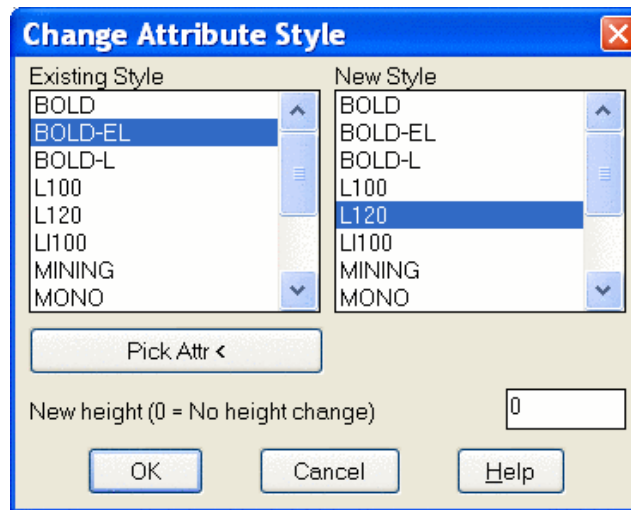
This command will globally change the text style of attributes on the drawing. This can be very useful if all the label styles (such as the point symbol attribute labels) on a drawing must be changed to accommodate a different plotting

specification. The default STYLE used for the point symbol attributes is PTXT.

Under **Existing Style**, select the style that is currently applied to the attributes you want to change. If you are unsure of the existing text style, select the **Pick Attr** button, then pick an existing attribute on the screen. When the dialog returns, the text style applied to that attribute will be selected in the list.

Select the **New Style** that you want to apply to the attributes.

Enter a **New Height** for the attributes. An entry of zero (0) will not modify the existing height.



Pulldown Menu Location: Edit > Change

Keyboard Command: chgattr

Prerequisite: You may want to use the *LIST* command to check the current Text size.

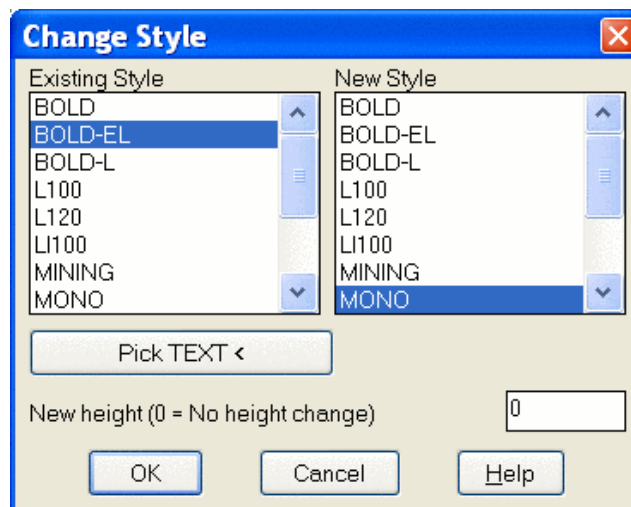
Change Style

This command will globally change the style and height of text on the drawing. This can be very useful if all the text sizes on a drawing must be changed to accommodate a different plotting scale.

Under **Existing Style**, select the style that is currently applied to the text you want to change. If you are unsure of the existing text style, select the **Pick TEXT** button, then pick an existing text entity on the screen. When the dialog returns, the text style applied to that text entity will be selected in the list.

Select the **New Style** that you want to apply to the text.

Enter a **New Height** for the text. An entry of zero (0) will not modify the existing height.



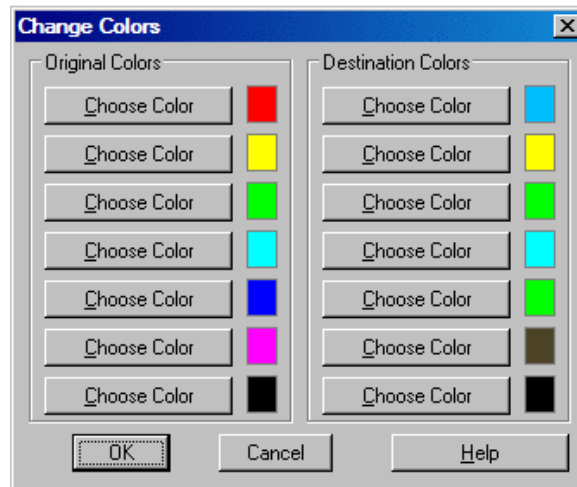
Pulldown Menu Location: Edit > Change

Keyboard Command: chgstyl

Prerequisite: Text entities

Change Colors

This command is designed to change the original color of existing entities in the drawing to a different color. This is done using the Change Colors dialog. You must match up the original colors of original entities to the preferred colors that they will change to. These "destination colors" are directly to the right of the original colors in the dialog (on the same row). You then click OK and select the specific entities on-screen that you want changed. This routine changes all entities in the drawing that you have chosen and that have an original color that has been changed. Do your dialog box color selections and matching up first, followed by *OK*. Then select the entities.



Prompts

Change Colors dialog Create your color change schemes and click OK.

Select entities to change colors.

Select objects: *select entities*

Pulldown Menu Location: Edit > Colors

Keyboard Command: chgcolor

Prerequisite: Entities whose colors are to be changed

Colors From Layer

This command sets selected entities to their current color instead of color BYLAYER. For each entity with color set to BYLAYER, this command looks up the color for the layer of the entity and sets the entity color to that layer color.

Prompts

Select entities to color from layer.

Select objects: *select entities*

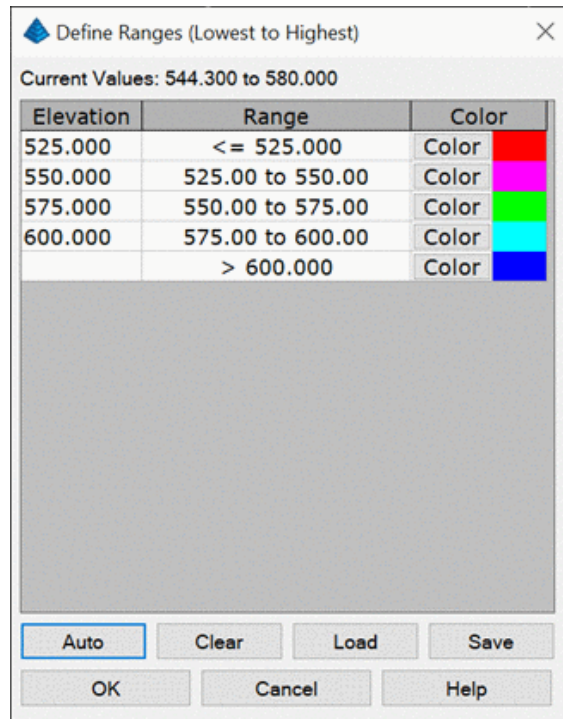
Pulldown Menu Location: Edit > Colors

Keyboard Command: layer2color

Prerequisite: Entities

Colors By Elevation

This command sets the color of entities based on the entity elevations. After selecting the entities, there's a dialog to define the colors for the different elevation ranges. The elevation range of the selected entities is shown at the top of the dialog.



Prompts

Select entities to color.

Select objects: *select entities*

Change Colors dialog Create your color change schemes and click OK.

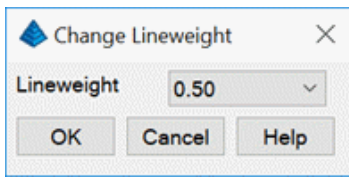
Pulldown Menu Location: Edit > Change

Keyboard Command: ctrcolor

Prerequisite: Entities at elevation

Change Lineweight

This command sets the lineweight for the selected entities.



Pulldown Menu Location: Edit > Change

Keyboard Command: chglwt

Prerequisite: Entities

Change Block/Inserts Rotate

This is a command to set the angle of blocks by various methods. This command optionally can change the rotation of a block by twist screen angle, azimuth, entity segment or by follow polyline. It will work with Carlson point symbol blocks, or any block. For example, you may receive a drawing from another firm, insert it in, and then want to change the rotation.

Prompts

Twist by [<Twist screen>/Azimuth/Entity segment/Follow polyline]? *press Enter*

Enter angle relative to current twist screen <0.0>: 30

Select Symbols to Rotate. *pick symbol*

Select objects: 1 found

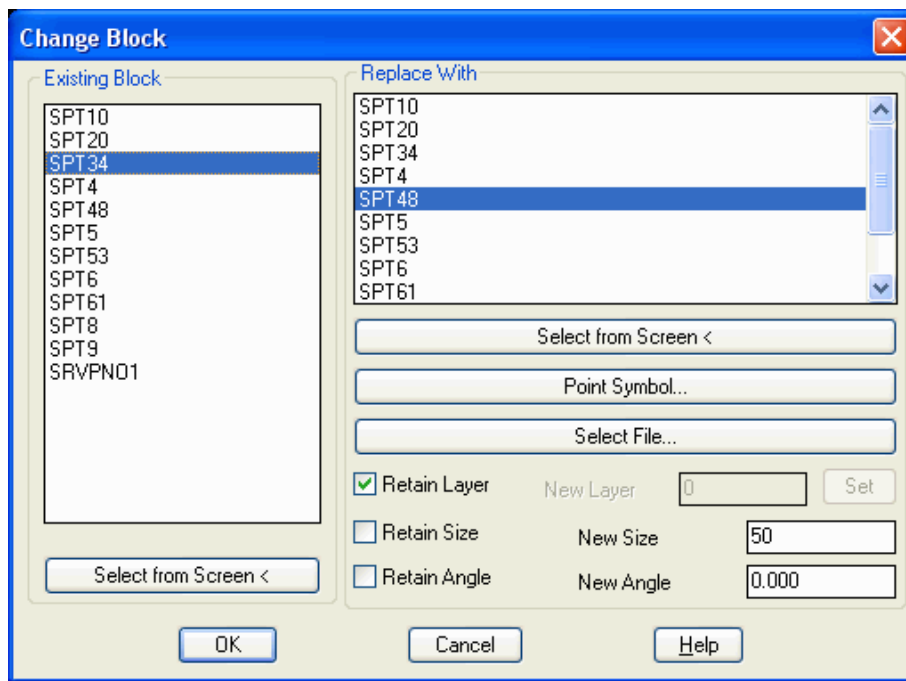
Pulldown Menu Location: Edit > Change > Block/Inserts

Keyboard Command: TWISTSYM

Prerequisite: None

Change Block/Inserts Substitute

This command is used to replace selected block(s) with a different block. The command optionally can change the size and rotation angle. This command will work with Carlson point symbol blocks, or any block. For example, you may receive a drawing from another firm and want to replace certain inserts with inserts of your own specification. In the dialog shown, we are replacing the block named NASTAR with a block named COHNORTH, which will be inserted at 50 scale and zero rotation.



Existing Block: Select the block name to be replaced. If the block name is unknown, choose the Select from Screen button, then select the block from the current drawing.

Replace With: Select the block that will replace the existing block. You may choose from the list of defined blocks, select an existing block from the current drawing, choose a point symbol from the standard Carlson point library, or select a drawing file.

Retain Size and Rotation: When checked, the new block will retain the size and rotation values from the old block.

New Size: Available if Retain Size and Rotation is not checked. Enter the size for the new block.

New Rotation Angle: Available if Retain Size and Rotation is not checked. Enter the rotation angle for the new block.

Pulldown Menu Location: Edit > Change > Block/Inserts

Keyboard Command: chgblk

Prerequisite: None

Change Block/Inserts Resize

This command resizes blocks inserts while maintaining their insertion position. When prompted to select objects, choose the inserts to resize. Note that this routine does not rescale attributes that may be associated with the selected inserts.

Prompts

Scaling Multiplier <0.5>: Enter the size scale factor.

Select symbols and blocks to scale.

Select objects: *select entities*

Pulldown Menu Location: Edit > Change > Block/Inserts

Keyboard Command: sizeblk

Prerequisite: block/inserts in drawing

Pivot Point Rotate by Bearing

This command allows you to rotate the selected entities from the drawing. The rotation angle is defined by the difference between a reference line and an entered bearing or azimuth. The reference line is defined by two points

that can be picked on the screen or entered by point number.

Prompts

Select entities to rotate.

Select objects: *select the entities*

Base pivot point ?

Pick point or point number: 2 The program then reads the coordinate value for pt#2 from the current CRD file.

Reference Bearing point ?

Pick point or point number: *pick a point*

Reference Bearing N 44d31'1" E The program then displays the reference bearing defined by the two points selected.

Azimuth/<Bearing (Qdd.mmss)>: 245.3030 Enter an A to input an Azimuth or enter the bearing. The above response is a bearing of South 45 degrees, 30 minutes, and 30 seconds East. The program then rotates the database to the new bearing.

Update viewports [Yes/<No>]? *press Enter.* This option rotates all the viewports in all the layouts of the drawing.

If Carlson Points are selected the program warns:

This command DOES NOT change the coordinates in the CoORDinate file!

Use CoORDinate File Utilities menu, Update CRD from Drawing.

This warning applies if the points entities are not linked to the CRD file. This link option is set in the Configure command.

Pulldown Menu Location: Edit > Rotate

Keyboard Command: brot

Prerequisite: None

Entity Insertion Point Rotate

This command allows you to rotate the selected entities where the rotation pivot point for each entity is the insertion point of the entity. The rotation angle will follow one of the following alignments: Twist screen, Azimuth, Entity Segment, Follow or Pick. This routine processes TEXT, MTEXT and INSERT entities only.

Prompts

Rotate by [<Twist screen>/Azimuth/Entity segment/Follow/Pick]? *F*

Select polyline to follow: *pick a polyline*

Select Entities to Rotate.

Select objects: *pick entities to rotate*

Flip text for twist screen [Yes/<No>]? *Y*

Rotating

Pulldown Menu Location: Edit > Rotate

Keyboard Command: ss_twist

Prerequisite: Entities to rotate

Select by Length

This command builds a selection set of linework objects in the drawing based on linework length. The length filter can be setup to get linework greater than or less than the specified value, or between two length values. After

specifying the length criteria, the program prompts for selecting the linework to check. The program then builds a selection set of those objects that pass the length filter. Then to use this selection set in other commands, enter "P" for previous at the "Select objects:" prompt.

Prompts

Select by greater, less or between lengths [<Greater>/Less/Between]? *press Enter*

Enter length for greater than: *1000*

Select objects to build selection set.

Select objects: *pick linework to filter*

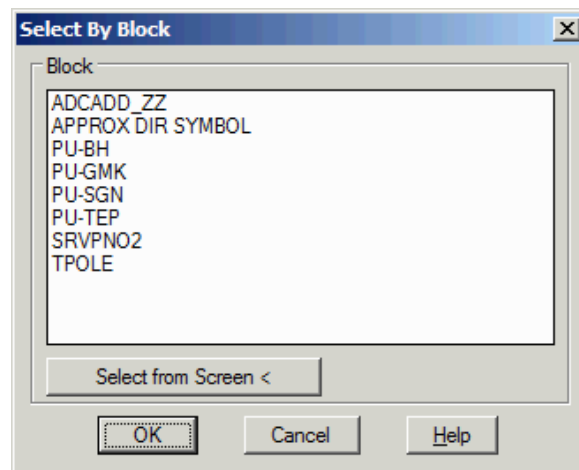
Pulldown Menu Location: Edit > Selection Sets

Keyboard Command: lenselect

Prerequisite: None

Select by Block

This command builds a selection set of blocks by using a block name filter. The block name to match is specified in a dialog with a list of all the block names in the drawing. Either pick from the list or use the Select From Screen button to get the block name by picking a block in the drawing. After selecting the block name, pick OK and the program will report how many of those blocks were found in the drawing and put into the selection set. This selection set is then ready to use at the next command with a select objects prompt. To use the selection set, type 'P' at the select objects prompt.



Pulldown Menu Location: Edit > Selection Sets

Keyboard Command: selblk

Prerequisite: Blocks

Select by Area

This command builds a selection set using inclusion and/or exclusion closed polylines. Entities within the inclusion polylines are selected and entities within the exclusion polylines are not selected. With the Window selection method, the entity must be entirely inside the inclusion area and entirely outside the exclusion area to be included in the selection set. With the Crossing selection method, an entity is added to the selection set if any part of the entity is inside the inclusion area.

Prompts

Select the Inclusion perimeter polylines or ENTER for none:

Select objects: *pick the closed polyline*

Select objects: *press Enter*

Select the Exclusion perimeter polylines or ENTER for none.

Select objects: *press Enter*

Type of selection (Window/<Crossing>)? *press Enter*

Select objects to build selection set.

Select objects: *All These selected objects are checked with the inclusion/exclusion polylines.*

Select objects: *press Enter*

Built selection set with 43 objects.

Command: *Erase*

Select objects: *P* To use previous selection set created by Select by Area.

43 found

Select objects: *press Enter*

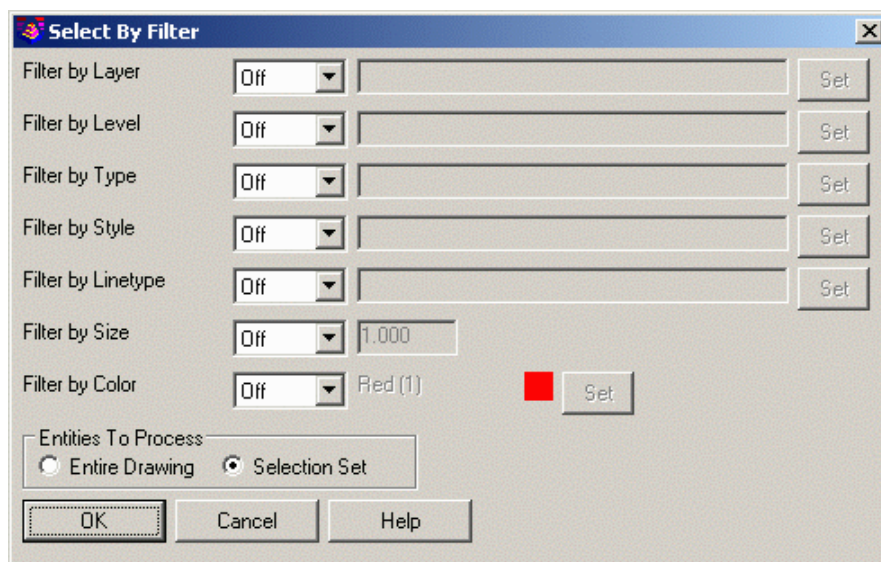
Pulldown Menu Location: Edit > Selection Sets

Keyboard Command: *ssgetarea*

Prerequisite: Closed perimeter polylines

Select by Filter

This command can be used to build a selection set of objects inside a drawing based on properties of layer, level, entity type, style, linetype, size and color. The open/closed filter applies to polylines. There is a dialog to define the filter of which properties to include and exclude. The Entities To Process option chooses between checking the entire drawing or prompting for a selection to process with the filter. The program then builds a selection set of those objects that resides on those layers. Then to use this selection set in other commands, enter "P" for previous at the "Select objects:" prompt.



Pulldown Menu Location: Edit > Selection Sets

Keyboard Command: *fsel*

Prerequisite: None

Select Similar

This command creates a selection set of all entities in the drawing with properties that match the selected entity. The properties filter uses the entity type and layer name. To use this selection set in other commands, enter "P" for

previous at the "Select objects:" prompt.

Pulldown Menu Location: Edit > Selection Sets

Keyboard Command: selectsim

Prerequisite: None

Text Enlarge/Reduce

This command will scale text entities up or down in size. The routine prompts for a scale multiplier and a selection set of text objects. If you want to enlarge the text enter a value greater than one. If you want to reduce text enter a decimal fraction such as .5. This would reduce the text size by 50%. This command is very useful if you have set up your drawing for one plotting scale and decide to change to a new plotting scale. The Change Text Size command can alternatively be used to set the text size to a specific value.

Pulldown Menu Location: Edit > Text

Prerequisite: Text entities to be changed

Keyboard Command: txtenl

Rotate Text

This command sets the rotation of the selected text to the current twist screen, an entered azimuth, or to align with a line or polyline. The text keeps the same insertion point and justification. The *Twist Screen* option sets the text rotation to align horizontal with the current twist screen. With the *Azimuth* option you can enter the angle or pick two points to define the text rotation. The *Entity segment* aligns the text with a selected line or polyline segment. The *Follow* option aligns the text with the closest polyline segment.

Prompts

Rotate by (<Twist Screen>/Azimuth/Entity segment/Follow/Pick)? *press Enter*

Enter angle relative to current twist screen <0.0>: 23

Select Text to rotate.

Select objects: *select the text*

Pulldown Menu Location: Edit > Text

Keyboard Command: twisttxt

Prerequisite: Text

Move Text

This command moves existing text entities by sliding at the text angle or perpendicular. This sliding method is equivalent to setting the crosshairs to the text angle and then moving with ORTHO on.

Prompts

Select Text to slide:

Select objects: *pick text entities*

Pick starting point for slide: *pick a point to begin sliding and then pick a second point for the new location*

Pulldown Menu Location: Edit > Text

Keyboard Command: annslide

Prerequisite: Text entity to move

Move Text with Leader

This command moves an existing text entity and creates a leader from a picked point to the new text location. The routine keeps track of the original text location and has an option to restore the text to the original without the leader. To use the Restore function, type R at the Command prompt. Also, to access the options for this command, type O for Options at the Command prompt.

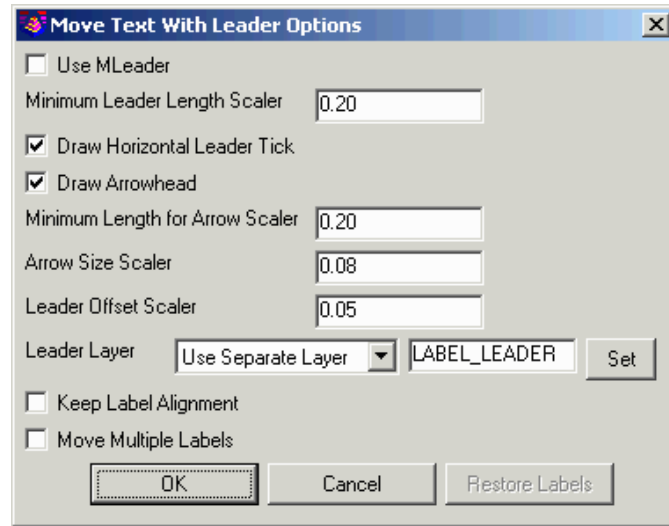
Prompts

Select Label to Move (O for Options,R for Restore): pick any text entity

Pick start point for leader: pick the point where to draw the leader arrowhead

Pick end point for move: pick the end of the leader where to move the text

Select Label to Move (O for Options,R for Restore): O



When Options is chosen the "Move Text With Leader Options" dialog allows the user to customize the leader and label drawing settings:

Use MLeader: This option uses an MLeader for a combined label and leader. Otherwise a standard Leader entity is drawn with separate label.

Minimum Leader Length Scaler: If the distance of the move is less than this value, a leader will not be drawn.

Draw Horizontal Leader Tick: When checked, a horizontal leader tick will be drawn from the end of the leader towards the annotation.

Draw Arrowhead: Controls whether to draw an arrowhead at the end of the leader.

Curved Leader: Chooses between making a straight line or curved leader.

Draw Box Around Text: Creates a box around the text.

Minimum Length for Arrow Scaler: When the leader length is more than this amount, the arrowhead is drawn.

Arrow Size Scaler: Sets the size for the arrowhead.

Leader Offset Scaler: This is used to set the distance from the end of the leader and the annotation.

Leader Layer: The Use Separate Layer method places the leader on the specified layer from the dialog. The Use Current Layer method places the leader on the current drawing layer. The Use Text Layer method uses the label layer for the leader layer.

Keep Label Alignment: This option keeps the original text angle. Otherwise the leadered text is orientated horizontally to the current twist screen.

Move Multiple Labels: Use this option to select multiple text entities as a group for moving.

NOTE: The leader scaler units (Arrow Size Scaler, Minimum Length for Arrow Scaler, Minimum Leader Length Scaler and Leader Offset Scaler) are multiplied by the current horizontal scale value which is set under Drawings Setup.

Select Label to Move (O for Options,R for Restore): R

Select Label to Restore: pick a text that had been moved with the "Move with Leader" command previously. The selected label will be restored to its previous state.

Pulldown Menu Location: Edit > Text

Keyboard Command: movetext

Prerequisite: Text entity to move.

Move Attributes with Leader

This command allows you to move block attributes and to draw a dynamic leader to block insertion point. Leaders and arrowheads may be customized by selecting Options at the command line. The attributes are always justified left or right depending on which side the leader starts. The Restore option prompts to select a moved attribute label and then puts that block attribute back to the default position.

The Options dialog has these settings.

Minimum Leader Length Scaler: Specifies the minimum length, in terms of multiples of the attribute block's height, that the leader must be.

Prompt for Curved Leader Points: This option prompts for additional leader points and creates a smooth leader that goes through all the leader points.

Draw Horizontal Leader Tick: Specifies whether or not to draw a terminating tick (a short horizontal line segment sometimes referred to as a "hook line").

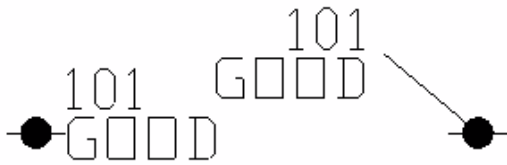
Draw Arrowhead: Specifies whether or not to draw an arrowhead at the end of the leader that points to the point entity.

Minimum Leader for Arrow Scaler: Specifies the minimum length of the leader, in terms of multiples of the attribute block's height, that the leader must be before an arrowhead is placed on it.

Arrow Size Scaler: A scale factor to apply to resize the arrowhead symbol.

Leader Offset Scaler: A distance indicating the desired offset from the point node to the tip of the leader.

Use Separate Leader Layer: Specifies whether or not to use a layer other than that of the identified point for the leader. Use the **Select** button to choose an alternative layer for the leader.



Block attributes before and after move with leader

Prompts

Select Label to Move [Options/Restore]: *select block attribute*

Pick label position: *pick point*

Select Label to Move [Options/Restore]: *press Enter to end*

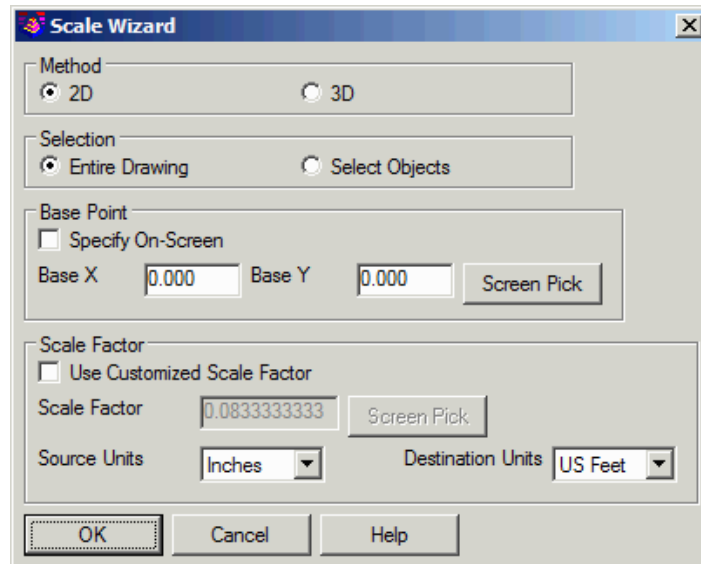
Pulldown Menu Location: Edit > Text

Keyboard Command: moveattrleader

Prerequisite: Block with attributes

2D Scale

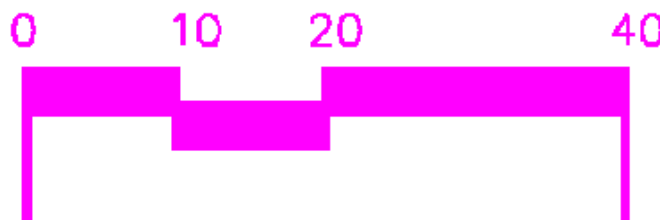
This command will scale selected entities using a specified scale factor and base point. This 2D Scale method differs from the 3D Scale method in that it only scales the entities in the x,y coordinates and does not change the elevations of the entities. A case for using 2D Scale is when the x,y coordinates are in architectural units of inches and the elevation is in feet and you want to convert the x,y coordinates to feet. When the entities are at zero elevation, then 2D Scale makes no difference and it is better to use 3D Scale because it is faster.



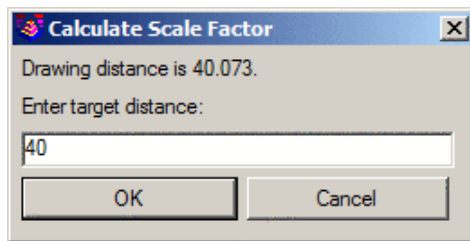
In the dialog shown here, you have the ability to determine what is scaled: the entire drawing or a selection set. If you choose Select Objects, you will be prompted to select the entities to scale after clicking the OK button. The Base Point acts as the center of the scaling operation and remains stationary. The base point you specify identifies the point that remains in the same location as the selected objects change size.

There are two methods for scaling entities: by Units Conversion or by a Customized Scale Factor. The dialog above shows one application of this routine, converting a drawing from architectural (Inches) to decimal units (US Feet) when the architectural units have the drawing x,y coordinates in inches and the elevations in feet. In this case, 2D Scale can be used to apply a 1/12 scale factor (0.08333333) to convert the inches to feet for the x,y coordinates and leave the elevations unchanged.

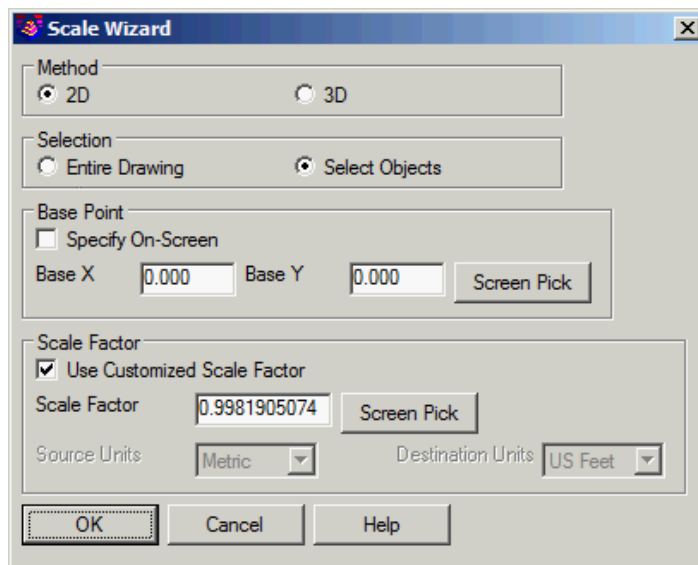
If the scale you want to apply is not a standard conversion, a manual scale can be entered by checking on the Use Customized Scale Factor checkbox. A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object.



To scale a drawing by a known distance on the plan (which is often the case when working with PDF imports) select the Screen Pick button. This will prompt you to pick the beginning and ending points along a known distance (like the bar scale above). The program will then report the current distance of the segment (in this example 40.073) and allow you to enter in the desired distance (which is 40 in this case).



The program will then calculate the proper Scale Factor to apply to the selection set.



Select entities to scale.

FILter/⟨Select entities⟩: *Select the entities whose size should change and press Enter to complete the selection process.*

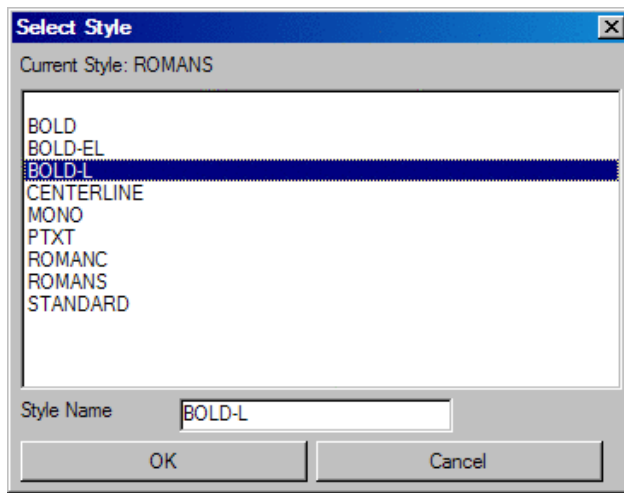
Pulldown Menu Location(s): Edit > Scale

Keyboard Command: scscale

Prerequisite: None

Change Text Font

This command can change multiple text entities to a user specified style. The routine prompts for a selection set of TEXT and/or MTEXT objects. Once the selection is made, the Select Style dialog appears. You can then select a text Style Name, such as MONO or ROMANS, that you would like to change to. Click OK. To the right on Style Name, you can enter a style name that does not exist. If you do, it will be created for you using the font with the same name.



Pulldown Menu Location: Edit > Text

Keyboard Command: chgtxtstyle

Prerequisite: Text entities to be changed

Change Text Size

This command will change the size of the selected text objects to the user specified size. The *Text Enlarge/Reduce* command also changes text size. The difference is that this routine sets the text to an absolute size whereas *Text Enlarge/Reduce* scales, or relatively changes, the text size.

Prompts

Select the text to size.

Select objects: *select the text*

Enter new text size: *enter value*

Pulldown Menu Location: Edit > Text

Keyboard Command: chgtxtsize

Prerequisite: Text entities to be changed

Change Text Width

This command changes the width of the selected text entities, after a new width factor is entered. The insertion point of each text entity is maintained as the routine lengthens or shortens the text.

Iron Pin	Text width = 1
Iron Pin	Text width = 0.75
Iron Pin	Text width = 1.5

Effect of different width factors on the same text line

Prompts

Select the text to change.

Select objects: *select text entities*

Enter new width factor <1.0>: *enter new width factor*

Pulldown Menu Location: Edit > Text

Keyboard Command: chgtxtwidth

Prerequisite: Text entities to be changed

Change Text Oblique Angle

This command allows you to change the text oblique angle on existing text in the drawing. The oblique angle for a specific text style is defined during the creation of the style. The default value for the oblique angle for text styles is 0 until defined to another value by the user. When changing the oblique angle, a minus (-) sign in front of the angle indicates a backward slant and a positive value results in a forward slant. Remember that the reference base point for the oblique change is always 0 degree. This means that if an existing text string has an oblique angle of 20, changing the oblique angle to 25 will not add 25 degrees to the existing 20 degree oblique resulting in a text oblique angle of 45 degrees, but rather a 25 degree oblique will be established by referencing 0 oblique as the base, and then slanting the text to 25 degrees. This works the same for slanting text backward as well as forward. Below is an example showing original text created with the default oblique angle of zero, then changed to a backward slant of 20 and a forward slant of 25 degrees.

Prompts

Select the text to change.

Select objects: *Select text to change oblique angle on.* Note that one or more text strings can be selected. When all desired text has been selected, press Enter.

Enter new oblique angle <0.0>: Enter the desired oblique angle.

Iron Pin	Oblique Angle = 0
Iron Pin	Oblique Angle = -20
Iron Pin	Oblique Angle = 25

Pulldown Menu Location: Edit > Text

Prerequisite: Text entities to be changed

Keyboard Command: chgtxtoblique

Convert Text To MText

This command converts regular text entities (DTEXT) into multi-line text entities (MTEXT). There are two processing methods. The Individual method converts each selected text entity into a separate MText entity. The Group method combines all the selected text entities into a single MText entity. The Group method applies to text entities that belong together such as text lines for a paragraph of text.

Prompts

Group text or process individually [Group/<Individual>]? press Enter

Select objects: *select the text entities to convert*

Pulldown Menu Location: Edit > Text

Keyboard Command: text2mtext

Prerequisite: Text entities

Fix Text Overlaps

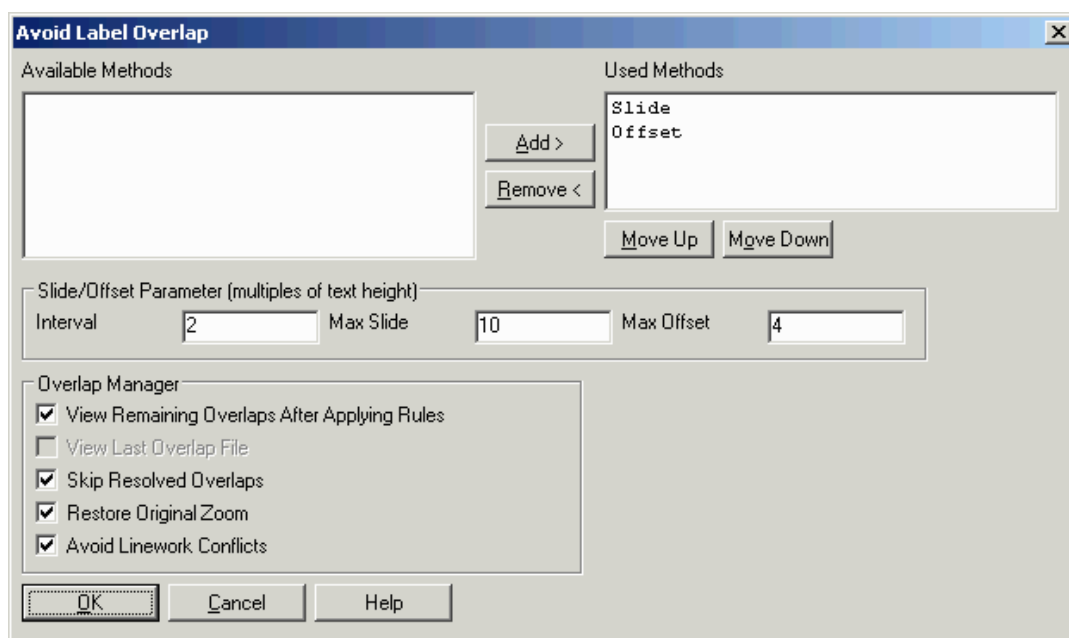
This command allows you to fix text overlaps for where different text overlap each other or where text crosses linework entities. You can either have the program automatically fix overlaps by using rules or just find the overlaps and step-through them to review and fix with CAD edits as needed.

In the options dialog, you can choose which rules to try for fixing the overlaps. The rules are applied in the order of the Used Methods list. If using a rule doesn't clear the text from overlaps, then the program tries the next rule.

The **Slide** rule moves the text parallel to text angle. The labels will not move past the **Max Slide** amount.

The **Offset** rule moves the text perpendicular to text angle. The labels will not move past the **Max Offset** amount.

The **Avoid Linework Conflicts** option checks for text overlaps with linework entities. Otherwise only overlaps with other text entities are checked.

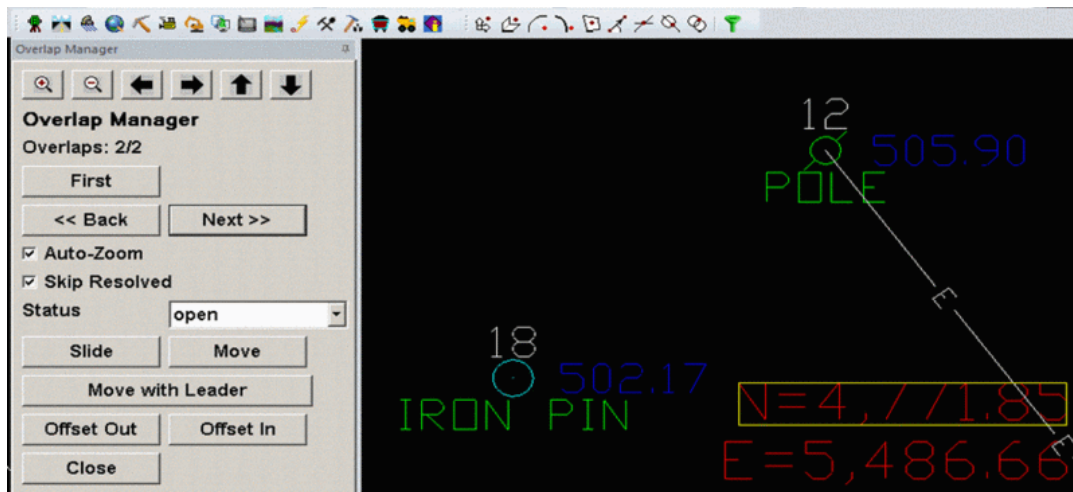


The **Overlap Manager** can be used to manually check and change label overlaps that aren't solved by the automatic rules. With the **View Remaining Overlaps After Applying Rules** option, any remaining overlaps will show in the Overlap Manager. Use the Back and Next buttons to step through each text conflict. The Auto-Zoom option zooms the dwg to show the current overlap. The Skip Resolved option shows only the unresolved conflicts. The Status option shows whether a conflict is resolved and lets you change the status. The Slide function moves text along the text angle. The Move function does an unrestricted move of the text. The Move With Leader function moves the text and creates a leader between the original and new locations. The Offset functions move the text perpendicular to the text angle.

When **View Last Overlap File** is checked, the Overlap Manager will return to the previous labels that were under review.

When **Skip Resolved Overlaps** is unchecked, the Overlap Manager will display all the labels that were moved by the command as a final check for you.

Restore Original Zoom will restore the zoom you were previously at before running the command.



Prompts

Avoid Label Overlap dialog

Select Labels for which to resolve annotation conflicts:

Select objects: *select entities*

Pulldown Menu Location: Edit > Text

Keyboard Command: textconf

Prerequisite: Text entities

Flip Text

This command will change the alignment of text entities by 180 degrees.

Pulldown Menu Location: Edit > Text

Keyboard Command: fliptext

Prerequisite: Text entities to be changed

Flip Text By Twist Screen

This command will change the alignment of text entities by 180 degrees for any selected text that are upside-down relative to the current drawing twist screen (dview twist).

Prompts

Select Text to Flip for Twist Screen.

Select objects: *pick the entities*

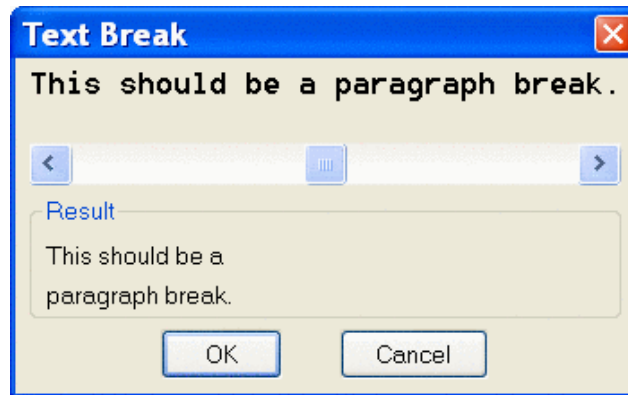
Pulldown Menu Location: Edit > Text

Keyboard Command: annflip

Prerequisite: Text entities to be changed

Split Text into Two Lines

This tool allows you to break a single line of TEXT into two separate lines. First, select the text string you would like to break. The Text Break dialog then appears. Initially, the slider is all the way to the right. Begin dragging it toward the left until it reaches the point where the split is at the desired position. Then choose OK to complete the break operation.



Pulldown Menu Location: Edit > Text

Keyboard Command: txtbrk

Prerequisite: Text entity to break

Text Explode To Polyline

This command converts the selected text into polylines. This function is generally used when preparing a plan view file for machine control, before using the *Write Polyline File* command.

Prompts

Select text to be EXPLODED.

Select objects: *select the text*

Substitute With Simple Font [<Yes>/No]? *Y*

1 text object(s) have been exploded to lines.

The line objects have been placed on layer 0.

Reading the selection set ...

Joining ...

Converting ...

Pulldown Menu Location: Edit > Text

Keyboard Command: textexp

Prerequisite: Text

Text Capitalization

This command sets the capitalization for the selected text. There are three options. The Proper method capitalizes the first character. The Upper method makes all the characters upper case. The Lower method makes all the

characters lower case.

Prompts

Set capitalization to [<Proper>/Upper/Lower]? *press Enter*

Select text to process.

Select objects: *pick text to set*

Pulldown Menu Location: Edit > Text

Keyboard Command: txtcaps

Prerequisite: Text Entities

Text Math

This command updates number labels by a math operation. After selecting the text to update, choose to add, subtract, multiply or divide, and then enter the amount.

Prompts

Select text entities.

Select objects: *pick the number labels*

Enter the operator (+ - * /): +

Enter a real number: 100

Enter number of decimals: 1

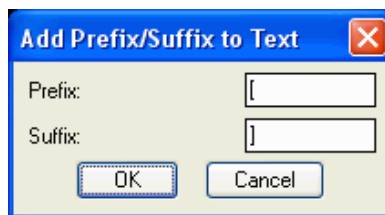
Pulldown Menu Location: Edit > Text

Keyboard Command: txtmath

Prerequisite: Text with numbers

Add Prefix/Suffix To Text

This command simply adds a prefix and/or suffix to the selected text entities. The strings to add are specified in a dialog. Then you select the text entities to update.



Prompts

Add Prefix/Suffix To Text dialog

Select text to process.

Select objects: *pick the text entities*

Pulldown Menu Location: Edit > Text

Prerequisite: Text entities to be changed

Keyboard Command: txtwrap

Remove Spaces From Text

This command removes leading and/or trailing spaces from the selected text entities.

Prompts

Trim all spaces from text on [Right/Left/<Both>]: *press Enter*

Select text to process.

Select objects: *pick the text entities to process*

Trimmed spaces from 1 text entities.

Pulldown Menu Location: Edit > Text

Keyboard Command: txtrmspace

Prerequisite: Text entities

Line Up Text

This command lines up the selected text entities along either a horizontal or vertical line position, an angle defined by two picked points, an existing text or following a polyline. There is an option to respace the text to prevent overlaps when the text is lined up.

19.80TC
19.30FS

Before and after Line Up Text

19.80TC
19.30FS

Prompts

Line up text on [Horizontal/<Vertical>/Linework/Angle/Text]: *press Enter for Vertical*

Pick vertical position: *pick a point*

Respace to clear overlaps [<Yes>/No]?: *press Enter*

Select text to process.

Select objects: *select the text to process*

Pulldown Menu Location: Edit > Text

Keyboard Command: txtlineup

Prerequisite: Text

Join Text Entities

This command combines two text entities by appending the second text to the first. The Words join method puts a space between each text. The Letter join method appends without a space.

Prompts

Select first text line: *pick a text entity*

Select text to add to first text line: *pick a text entity*

Join type as [Words/<Letters>]: *press Enter*

Pulldown Menu Location: Edit > Text

Keyboard Command: txtjoin

Prerequisite: Text entities

Replace Text

This command will replace one text string with another. For example, if the text LEGEL is on a drawing, you could use this command to replace it with LEGAL. This command has a couple method. The Copy method prompts to select an existing text label to copy and then select the text entities to replace with this copy. The Type method prompts for the old string to replace and the new string.

There is also the CAD command *Find and Replace Text* (FIND) includes more options, including replacing partial strings and searching attributes and MTEXT.

Prompts

Type in replacement string or copy another text label [<Type>/Copy]? *press Enter*

Select Text to Change

Select objects: *pick text*

Old string: *LEGEL*

New string: *LEGAL*

Replace all or prompt [All/<Prompt>]? *All*

Pulldown Menu Location: Edit > Text

Keyboard Command: chgtext

Prerequisite: Text entities to be changed

Move Attributes with Leader

This command allows you to move block attributes and to draw a dynamic leader to block insertion point. Leaders and arrowheads may be customized by selecting Options at the command line. The attributes are always justified left or right depending on which side the leader starts. The Restore option prompts to select a moved attribute label and then puts that block attribute back to the default position.

The Options dialog has these settings.

Minimum Leader Length Scaler: Specifies the minimum length, in terms of multiples of the attribute block's height, that the leader must be.

Prompt for Curved Leader Points: This option prompts for additional leader points and creates a smooth leader that goes through all the leader points.

Draw Horizontal Leader Tick: Specifies whether or not to draw a terminating tick (a short horizontal line segment sometimes referred to as a "hook line").

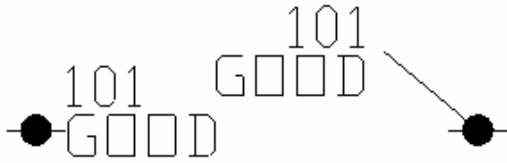
Draw Arrowhead: Specifies whether or not to draw an arrowhead at the end of the leader that points to the point entity.

Minimum Leader for Arrow Scaler: Specifies the minimum length of the leader, in terms of multiples of the attribute block's height, that the leader must be before an arrowhead is placed on it.

Arrow Size Scaler: A scale factor to apply to resize the arrowhead symbol.

Leader Offset Scaler: A distance indicating the desired offset from the point node to the tip of the leader.

Use Separate Leader Layer: Specifies whether or not to use a layer other than that of the identified point for the leader. Use the **Select** button to choose an alternative layer for the leader.



Block attributes before and after move with leader

Prompts

Select Label to Move [Options/Restore]: *select block attribute*

Pick label position: *pick point*

Select Label to Move [Options/Restore]: *press Enter to end*

Pulldown Menu Location: Edit > Text

Keyboard Command: moveattrleader

Prerequisite: Block with attributes

Consolidate Table

This command renumbers the first column of a table to remove any gaps in the numbering. For example, a table with rows labeled 1,3,4 would get renumbered to 1,2,3.

Prompts

Select Table to consolidate: *pick table entity*

Pulldown Menu Location: Edit > Table

Keyboard Command: crunchtbl

Prerequisite: table entity

Select by Elevation

This command builds a selection set of entities that are greater than, less than or in between a specified elevation that you enter in on the command line. Entities selected, based upon this elevation criteria, go into a selection set. With the Window selection method, the entities must be entirely inside of the inclusion area to be included in the selection set. With the Crossing selection method, an entity is added to the selection set if any part of the entity is inside the inclusion area.

Prompts

Select by greater, less or between elevations [<Greater>/Less/Between]? *press Enter*

Enter elevation for greater than: *19*

Ignore zero elevations [<Yes>/No]? *press Enter*

Select objects to build selection set. *pick objects*

Processing selection set ...

Built selection of 120 objects for elev more than 19.00.

To use type 'P' at Select objects: prompt.

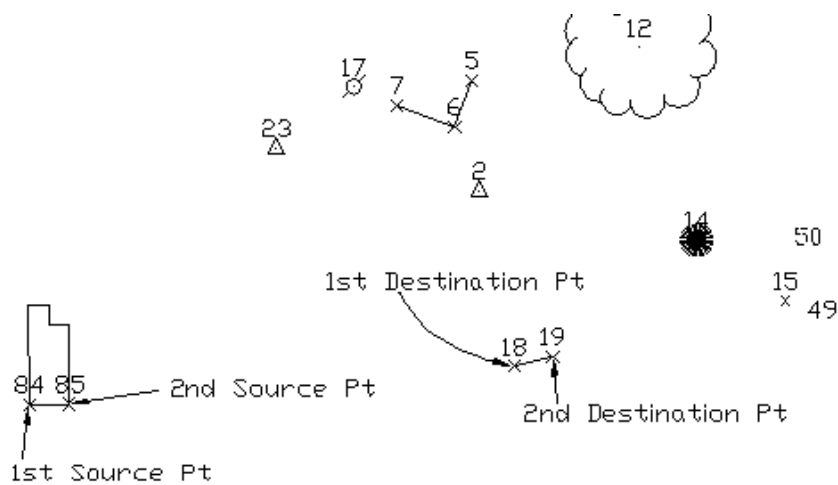
Pulldown Menu Location: Edit > Selection Sets

Keyboard Command: zselect

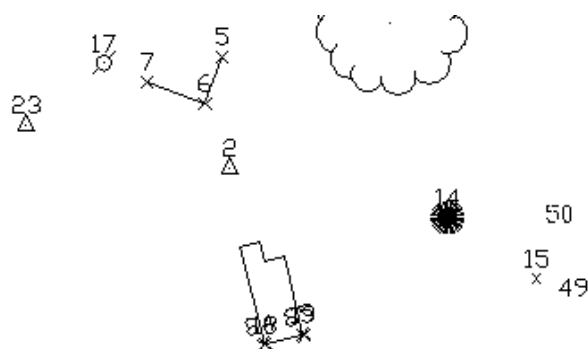
Prerequisite: Entities

2D Align

This command will align (translate, rotate and scale) the selected objects using two pairs of source and destination control points. The difference between the first source point and first destination point determines the translation amount. The difference between the angle and distance from the first and second source points compared to the angle and distance from the first and second destination points determines the rotation and scale. The scale part of the alignment is optional. This 2D Align function is the same as the standard Align function except that this 2D Align function does not use elevations so that the alignment is always in 2D. The control points can be screen picked or entered by point numbers.



Before and after 2D Align



Prompts

Select entities to align.

Select objects: *pick entities to process*

First Source Point?

Pick point or point number: *pick point 84*

First Destination Point?**Pick point or point number:** *pick point 18***Second Source Point?****Pick point or point number:** *pick point 85***Second Destination Point?****Pick point or point number:** *pick point 19***Scale factor:** 1.00434258**Scale objects based on alignment points [Yes/<No>]?** *Y***This command DOES NOT change the coordinates in the CoORDinate file!****Use Coordinate File Utilities menu, Update CRD File from Drawing.****Pulldown Menu Location:** Edit > Align**Keyboard Command:** scalign**Prerequisite:** None

Entities to Polylines

This command converts selected lines, arcs, circles, 3DFaces, ellipses, splines, multilines, regions and solids into individual polylines. Use *Join Nearest* to convert adjoining lines and arcs into continuous polylines.

Prompts

Select lines, arcs, circles, 3DFaces, ellipses, splines, multilines, regions and solids to convert.

Select objects: *select entities***Pulldown Menu Location:** Edit > Polyline Utilities**Keyboard Command:** topline**Prerequisite:** lines, arcs or other entities to convert

Reverse Polyline

This command reverses the order of the line and/or arc segments of a *POLYLINE*. This can be useful in conjunction with the commands *Station Polyline*, *MXS by Polyline*, *Profile from Surface Model* or *CL File from Polyline*, since the polyline must be plotted in the direction of increasing stations. If it is more convenient to draft a polyline in one direction do so and then use the *Reverse Polyline* command to change its order. Temporary arrows along the polyline are drawn to graphically show the new polyline direction.

Prompts

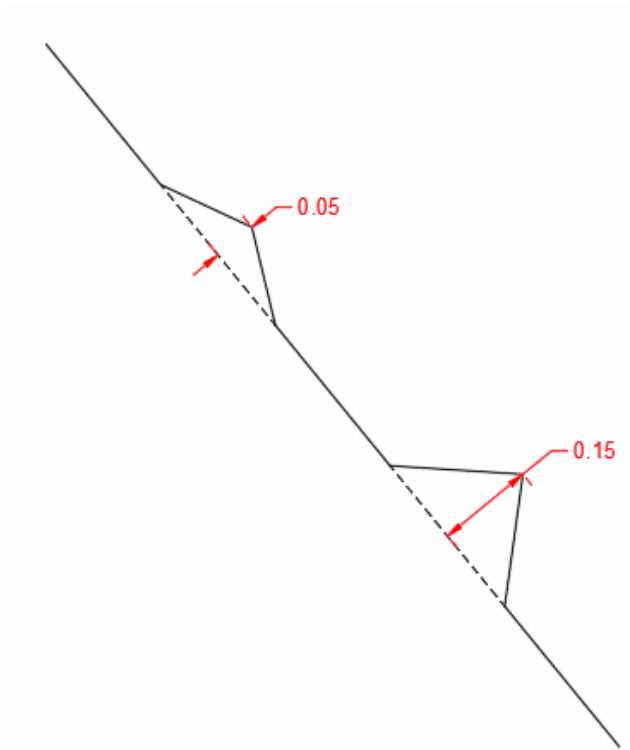
Select the Polyline to Reverse: *pick a point on polyline*

Pulldown Menu Location: Edit > Polyline Utilities**Keyboard Command:** revpline**Prerequisite:** A polyline

Reduce Polyline Vertices

This command removes points from a polyline, without significantly changing the polyline. The offset cutoff is the maximum amount that the polyline can move horizontally and vertically when removing a point. For example, in a polyline with three points in a straight line, the middle point can be removed without changing the polyline. The

below image shows an example. If the offset cutoff were set to 0.10, the upper-left wedge would be removed, but the lower-right wedge would not.



Prompts

Enter the offset cutoff <0.1>: .5

Select polylines to reduce.

Select objects: *pick polylines*

Processed polylines: 1

Total number of vertices: 10

Number of vertices removed: 1

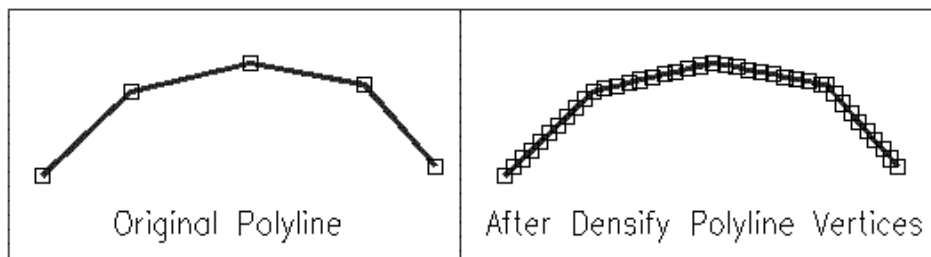
Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: reduce

Prerequisite: A polyline

Densify Polyline Vertices

This command adds vertices to the selected polylines at the specified interval. These points are interpolated between existing points in the polyline. This command is the opposite of Reduce Polyline Vertices.



Prompts

Select polylines to densify.

Select objects: *select polylines*

Point interval <10.0>: *press Enter*

Testing Entity > 1

Added 17 points to 1 polyline.

Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: densepl

Prerequisite: A polyline

Join Two Polylines

This command combines two linework entities to create a polyline. This command is simple variation of the Join Nearest command. The difference is that there is no options dialog and only two linework entities are processed. The linework entities can be polylines, lines or arcs. At the first entity prompt, you can enter Options to set the connection method for how to bridge a gap between the linework. The Direct method creates a segment across the gap. The Average method averages the endpoints into a single vertex. The Fillet method does a fillet with radius of zero.

Prompts

Select 1st entity to join [Options]: *pick polyline*

Select 2nd entity to join: *pick polyline*

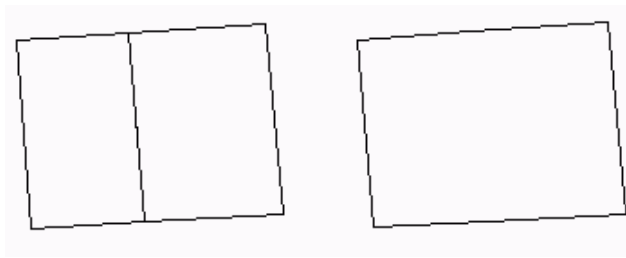
Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: join2

Prerequisite: Linework to join

Merge Closed Polylines

This command combines two closed polylines that share a common boundary to create a single closed polyline. For example, this command can be used to merge a polyline for a lot that is too small with a polyline for a neighboring lot. There are two methods for merging. One method is to pick the two polylines to merge. The other method takes an area size and selection set of polylines and merges all the polylines with an area smaller than the specified amount.



Before and after Merge Closed Polylines

Prompts

Select polylines by individual pick or size filter [**<Pick>/Size**]? *press Enter*

Pick 1st closed polyline to merge: *pick a polyline*

Pick 2nd closed polyline to merge: *pick a polyline*

Done.

Pick 1st closed polyline to merge (Enter to end): *press Enter*

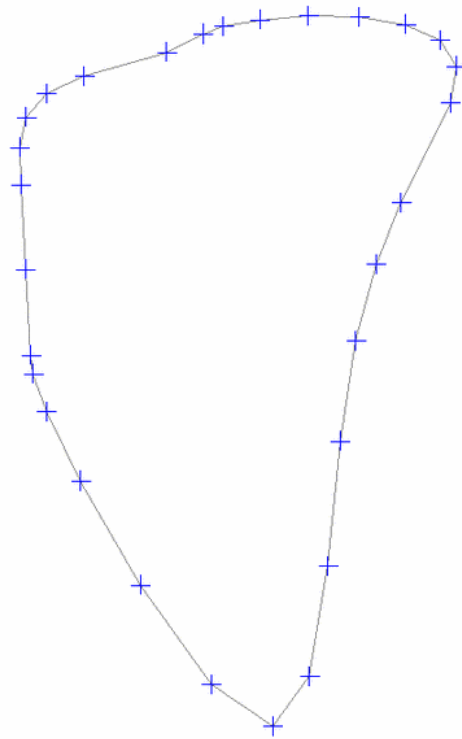
Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: mergepl

Prerequisite: two polylines to offset

Draw Polyline Blips

This command will draw temporary markers, "blips", at each polyline vertex. This allows you to identify the actual location of each vertex. The Blips can be drawn as temporary marks or lines. The lines method creates line entities that can be cleared from the drawing using Remove Polyline Blips. For temporary marks, any change to the viewport (pan, zoom, regen) will make the blips disappear. In later versions of AutoCAD, you can also click on the polyline to activate the grips which will remain visible during and after viewport changes.



Prompts

Draw as temporary marks or lines [**<Temporary>/Lines**]? *press Enter*

Select polylines to draw blips.

Select objects: *select polyline(s)*

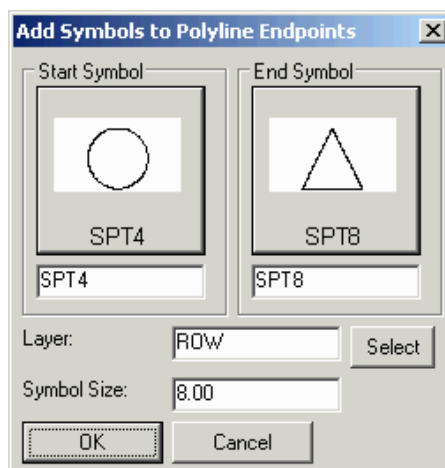
Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: plblip

Prerequisite: A polyline

Draw Polyline Start/End

This command simply draws symbols at the start and end vertices of a polyline to give a visual indication of the polyline direction. The routine starts with a dialog to select the different symbols for the start and end, and to select the layer and size for the symbols. Then you select the polylines and the program draws the symbols.



Prompts

Polyline Start/End Settings dialog

Select polylines.

Select objects: *select polyline(s)*

Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: plends

Prerequisite: A polyline

Set Polyline Origin

This command sets the starting vertex of a closed polyline. Simply pick the polyline and then pick near the point to set as the starting point.

Prompts

Select Polyline: *pick a polyline*

Pick Near New Origin Point: *pick a point on the polyline to be the starting point*

Processing ...

Select Polyline: *press Enter*

Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline

Keyboard Command: plchgorg

Prerequisite: A closed polyline

Add Intersection Points

This command adds points into lines or polylines where there are intersections. This can be useful for other commands such as Auto-Annotate. For example in the drawing shown, Add Intersection Points adds points to the boundary polyline where the lot lines intersect. Then Auto Annotate for the boundary polyline will label the boundary distance along each lot. This routine does not add intersection points on arcs.

This command can also be helpful for surface modeling with breakline polylines. At crossings, breakline polylines must have the same elevations for surface modeling. This command has an option to average the elevations of the polyline vertices at the crossing when the elevation difference is smaller than the specified tolerance.

Prompts

Select lines and polylines to check.

Select objects: *pick lines or polylines*

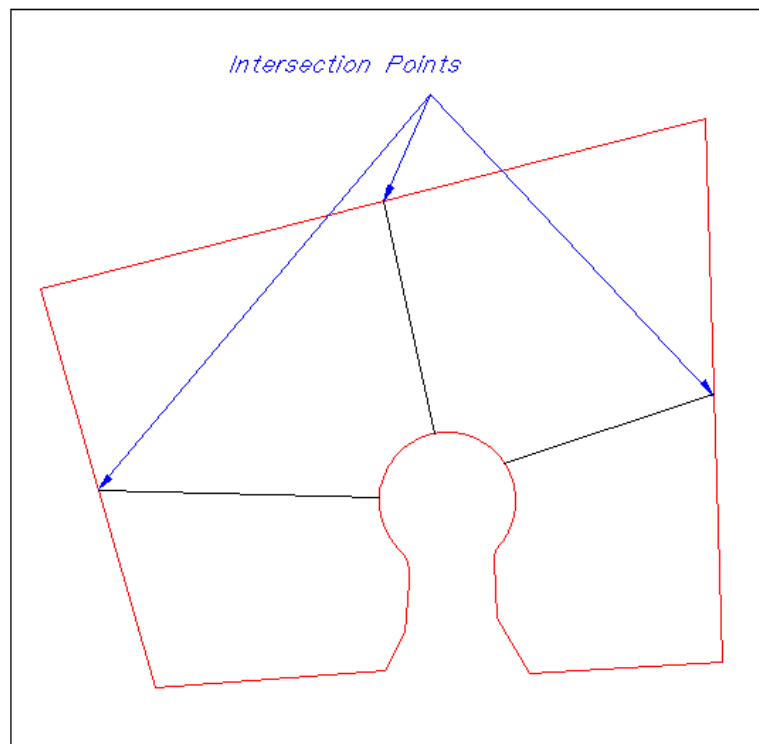
Average intersection elevations within tolerance [*<Yes>/No*]? *press Enter*

Elevation Tolerance *<0.1>*: *press Enter*

Reading the selection set ...

Adding intersection points ...

Added 3 intersection points.



Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline

Keyboard Command: addint

Prerequisite: Polylines or lines

Add Polyline Vertex

This command adds points into a polyline. First you select the polyline to modify. The existing polyline vertices are marked and then you can pick or enter the coordinates for the new points. A new point is inserted into the polyline at the nearest polyline segment. On a 3D polyline, the elevation of the new vertex will be calculated for you. You can continue to pick points to add or use the Undo option to remove the last point added. Press Enter when you are done.

There are two connection method. The Direct method adds the new point at the picked position. The Projected method finds the nearest perpendicular point along the polyline from the picked point and adds a vertex on the polyline segment at this projected position.

Prompts

Select polyline to add to: *pick a polyline*

Mode: Direct connection

Pick or enter point to add [Options]: *pick a point*

Pick or enter point to add (Enter to end) [Undo/Options]: *press Enter to end*

Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline

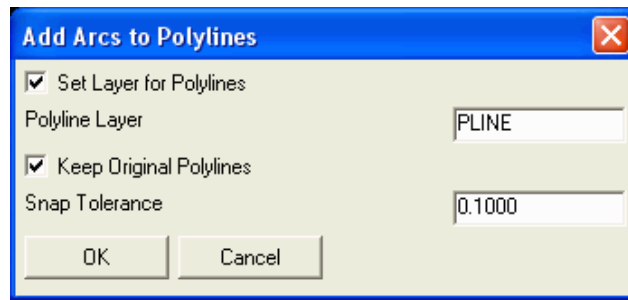
Keyboard Command: addpl

Prerequisite: A polyline

Add Polyline Arcs

This command replaces a series of short chord line segments in a polyline with an arc segment. This applies where you want to have a true arc instead of a series of line segments. In some cases, the CAD drafting has the arcs drawn as a series of short chords. Another application is to create an arc out of a series of connected survey points along the curve. The routine works by searching for a series of polyline vertices that fit within the specified tolerance with a best fit curve.

The options dialog allows you to set the layer for the new polylines. Otherwise the original polyline layer is used. There is an option whether to keep or erase the original polylines. The Snap Tolerance is the maximum offset allowed between the original points and the arc.



Prompts

Add Arcs to Polylines dialog

Select polylines to process.

Select entities: *pick the polylines*

Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline

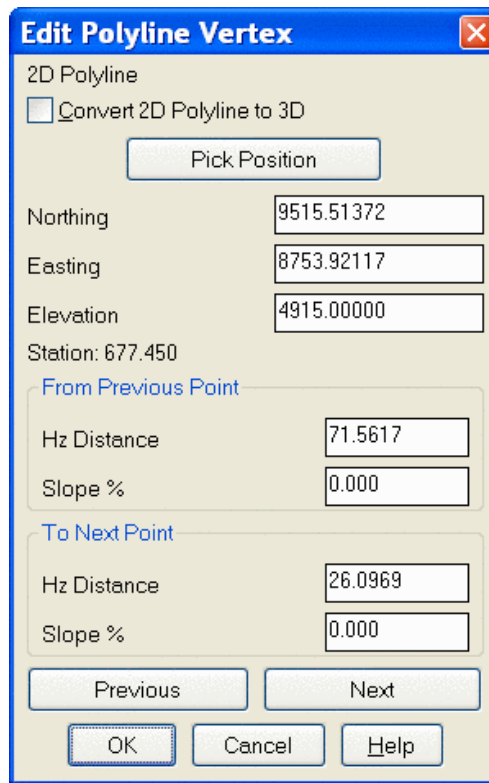
Keyboard Command: addplarc

Prerequisite: polyline

Edit Polyline Vertex

This tool allows you to make changes in the coordinates of vertices on all polyline types. Upon execution, you will be asked to select a polyline to edit. Upon selection, a temporary marker will be placed at all of the vertices of the polyline, making them easy to distinguish. You must then pick near the vertex you wish to edit. The following dialog appears.

At the top of the dialog it identifies the type of polyline as being 2D or 3D. In the case of 2D polylines, it allows you convert the polyline. You have the ability to type in new northing, easting or elevation values. You can also determine the 3D coordinate position by using distances and slope to/from adjacent points. As you change the values in the dialog, new values for derivatives are being calculated. For example, if you change the horizontal distances, the coordinates will change.



Prompts

Select polyline to edit: *pick a polyline*

Pick point on polyline to edit: *pick a point to be modified*

Edit Polyline Vertex dialog *click "Pick Position"*

Pick vertex position: *pick a new location for the vertex*

Edit Polyline Vertex dialog *click OK*

Make changes as needed. You will see the polyline vertices relocated based upon the new picked positions and coordinate changes. Use Previous and Next to move along the polyline. Note the dialog values changing.

Select polyline to edit (Enter to end): *press Enter to end*

Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: editpl

Prerequisite: A polyline

Edit Polyline Section

This command revises a segment of a polyline. Begin by picking a point on the polyline where you want to start editing. Then pick new points for the polyline. When finished picking new points press Enter, and then pick a point on the polyline to connect with the new points. The polyline segment between the start and end points is then replaced with the new points.

Prompts

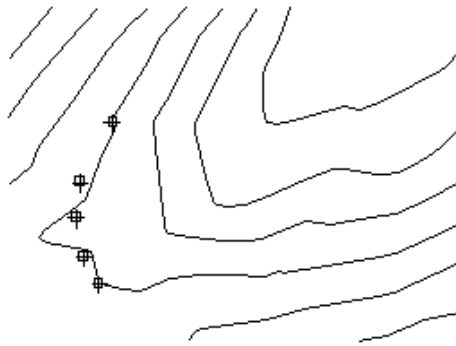
Select polyline to edit: *pick the polyline at the place to start editing*

Pick intermediate point (Enter to End): *pick a point*

Pick intermediate point ('U' to Undo, Enter to End): *pick a point*

Pick intermediate point ('U' to Undo, Enter to End): *press Enter*

Pick reconnection point on polyline: *pick the polyline at the place to join*



Edit this contour by picking new points



Contour with segment replaced with new points

Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline

Keyboard Command: editpl2

Prerequisite: Polylines

Remove Duplicate Polylines

This command analyzes the selected polylines and erases any duplicate polylines found. They must be exactly the same for one to be deleted.

Prompts

Select lines, arcs and polylines to process.

Select objects: *select linework to process*

Reading the selection set ...

Removed 1 duplicate linework entities.

Pulldown Menu Location: Edit > Polyline Utilities > Remove Polyline

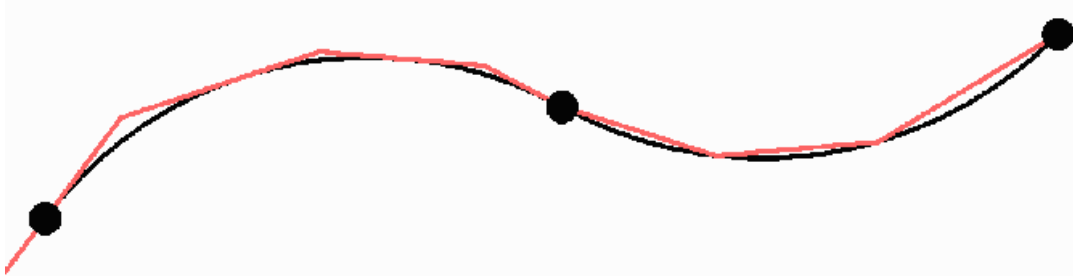
Keyboard Command: rmduplwork

Prerequisite: Polylines that have duplicates

Remove Polyline Arcs

This command replaces arc segments in polylines with a series of chords. Removing arcs is a prerequisite to some Carlson commands that don't handle arcs, such as *Break by Closed Polyline* and *Make 3D Grid File*. This process can add many vertices to the polyline. There are three methods to control the size of the chords and how many vertices are created. The Offset cutoff sets the maximum distance any point on the arc will be allowed to shift from the true arc to the chord position. The Length method sets the chord length. With the Length method, you can choose to create the chords entirely on the left or right side. Without setting the side, the program puts the chords

on the inside of the arcs. The Count method sets how many chords to replace each arc with.



Original polyline with reverse curve and polyline with arcs removed using Length method with chords put on top side

Prompts

Select polylines to remove arcs from.

Select objects: *pick polylines*

Use max offset or chord length method [<Offset>/Length]? *press Enter for Offset method*

Offset cutoff <0.5>: *press Enter*

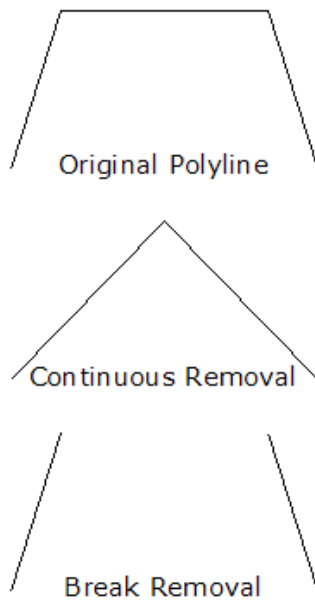
Pulldown Menu Location: Edit > Polyline Utilities > Remove Polyline

Keyboard Command: rmarc

Prerequisite: polyline with arcs

Remove Polyline Segment

This command removes the user specified segment from a polyline. A polyline segment is the section between two vertices of the polyline. There are two options for removing the segment. Either the two vertices of the removed segments are averaged together to keep polyline continuous, or the segment is left missing in the polyline, which creates two separate polylines. The keywords Continuous and Break respectively identify these two options. The first image is of the Original Polyline. The second is with the Continuous Removal option. The third is using the Break Removal option.



Prompts

Break polyline at removal or keep continuous (Break/<Continuous>)? *press Enter*

Select polyline segment to remove: *pick point on polyline*

Select polyline segment to remove: *press Enter to end*

Pulldown Menu Location: Edit > Polyline Utilities > Remove Polyline

Keyboard Command: removepl

Prerequisite: A polyline

Remove Polyline Vertex

This command removes vertices from a polyline. First you select the polyline to modify. The existing polyline vertices are marked and then you pick near the vertex you wish to delete. You can continue to pick vertices to delete, press Enter when you are done.

Prompts

Select polyline to remove from: *pick point on polyline*

Pick point to remove: *pick point*

Pick point to remove (Enter to end): *press Enter to end*

Pulldown Menu Location: Edit > Polyline Utilities > Remove Polyline

Keyboard Command: rmvertex

Prerequisite: A polyline

Remove Polyline Blips

This command erases polyline blip lines created by Draw Polyline Blips.

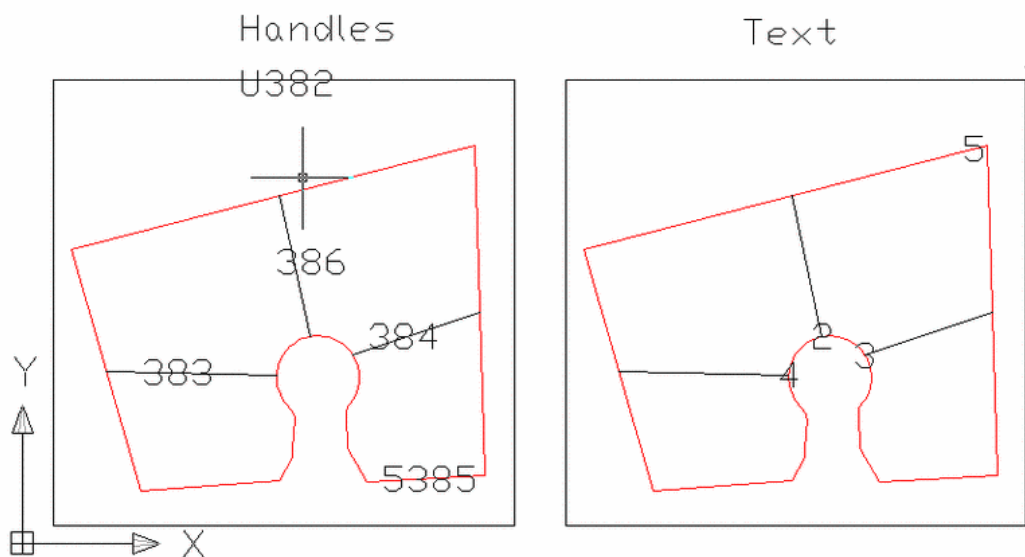
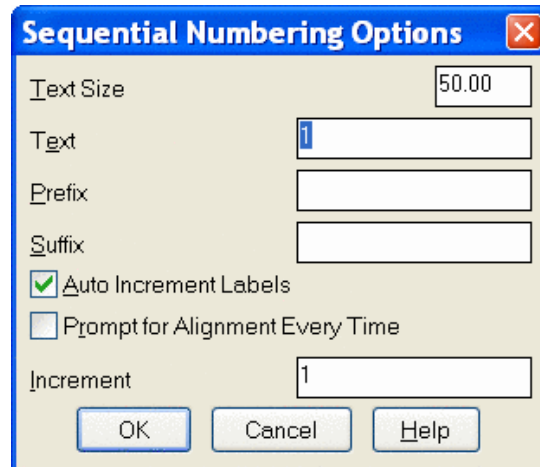
Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: rmbliip

Prerequisite: Polyline blips

Create Polyline ID Labels

This command labels the selected polylines with either the entity "Handle", which can be seen with a list, or with unique text numbers, such as 1, 2, 3, 4, etc.. When using the Text option, the following window appears to choose the text settings.



Prompts

Select Polylines to label.

Select objects: *pick polyline*

Label polylines by Text or Handles [Handles/<Text>]? *press Enter*

Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: label_polys

Prerequisite: A polyline

Join Polyline By Trace

This command combines linework entities to create a polyline. This command is similar to Join Nearest. The difference is that this command processes a single polyline at a time instead of a selection set of polylines. This command prompts for a linework entity to start joining from. Then the program looks in the drawing at other linework to join to this polyline using the filter options from the dialog. All the options in the dialog are the same as Join Nearest except the Prompt For Linework To Join option which prompts for other linework to join to the polyline once the program has finished tracing. See the Join Nearest section of the manual for descriptions of the other options.

Prompts

Join Polyline By Trace dialog

Select line, arc or polyline to join: *pick an entity*

Select entity to join to start (Enter for none): *press Enter*

Select entity to join to end (Enter for none): *press Enter*

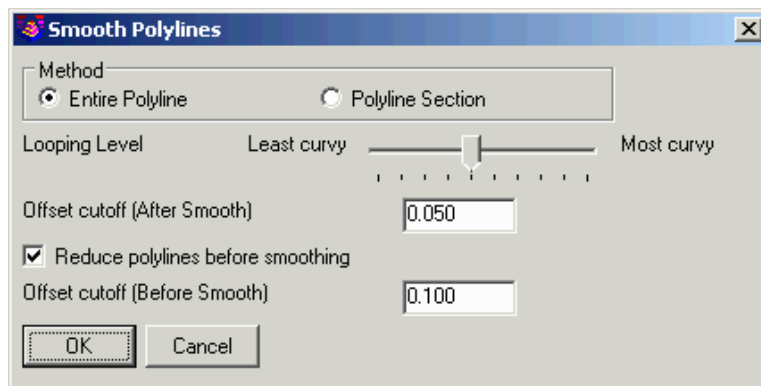
Pull-down Menu Location: Edit > Polyline Utilities

Keyboard Command: joinpoly

Prerequisite: Linework to join

Smooth Polyline

This command smooths the selected polylines using a modified Bezier method that makes the smooth polyline pass through all the original points and only smooths between the original points. Either the entire polyline can be smoothed or only a section of the polyline. When using Polyline Section, the program will prompt for points along the polyline to start and end the smoothing. The Looping Level controls smoothing amount. A higher factor gives more looping. The Offset Cutoff (After Smooth) is used to reduce the number of vertices in the final polylines. To not reduce vertices, set this Offset Cutoff to zero. The Offset Cutoff is the maximum amount that the polyline can move horizontally when removing a point. The Reduce Polylines Before Smoothing option removes extra vertices from the contours before smoothing. Removing points before smoothing gives the Bezier smoothing more freedom to make the polylines curvy.



Prompts

Smooth Polylines dialog

Select polylines to smooth.

Select objects: *pick polylines*

Smoothed 1 PolyLines

Total original vertices: 9 Total final vertices: 50

Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: smoothpl

Prerequisite: A polyline

Make Arcs Tangential

This command adjusts polylines to make arcs tangential. The program holds the tangents and adjusts the radius as needed.

Prompts

Select polylines to process.

Select objects: *pick polylines*

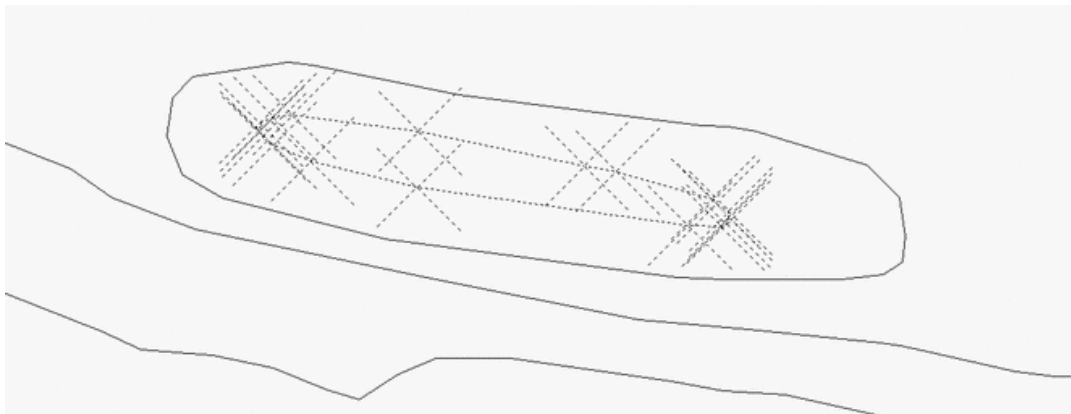
Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: tang_pl

Prerequisite: Polylines with arcs

Check Elevation Range

This command analyzes a selection set of polylines, and highlights the ones that fall outside of a specified elevation range. There is an option to set the polylines that are outside of the range to zero. Every polyline vertex that is outside of the range will be highlighted with an X.



Prompts

Enter elevation range minimum: 0

Enter elevation range maximum: 4900

Select polylines to check.

Select objects: *pick polylines to process*

Found 1 polylines outside of elevation range.

Set polylines outside elevation range to zero elevation [Yes/<No>]? N

Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: checkpl

Prerequisite: Polylines with elevations

Highlight Non-Perpendicular Intersections

This command highlights selected polylines that have T-intersections with other polylines that are non-perpendicular. For example, this command can be used to check that side lines for lots are perpendicular to the frontage polyline. For every non-perpendicular intersection, a temporary graphic arrow is drawn and the angle and the coordinates of the point are reported at the command line.

Prompts

Select the polylines to check.

Select objects: *pick polylines to check*

Warning: Polyline non-perpendicular by 0°00'47" at 5477.08,5047.53

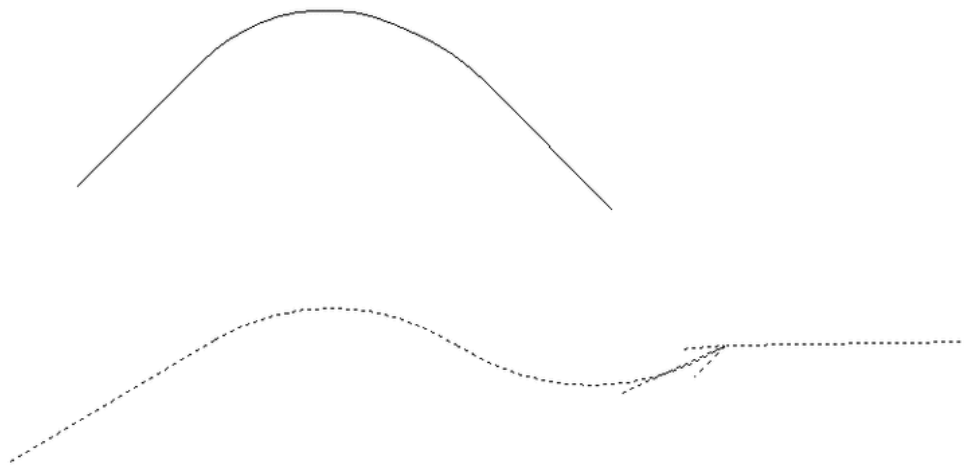
Pulldown Menu Location: Edit > Polyline Utilities > Check Polylines

Keyboard Command: highlight_nonperp

Prerequisite: Polylines

Highlight Non-Tangent Polylines

This command highlights selected polylines that have non-tangent lineworks. For every non-tangent polyline, an arrow is pointed to the first non-tangent point, and the non-tangent angle and the coordinates of the point are reported at the command line.



Prompts

Select polylines to check.

Select objects: 1 found

Select objects: 1 found, 1 total

Select objects: *press Enter to end*

Polyline non-tangent by 32°15'26" at 1540.41,-182.05
Highlighted 1 non-tangent polylines.

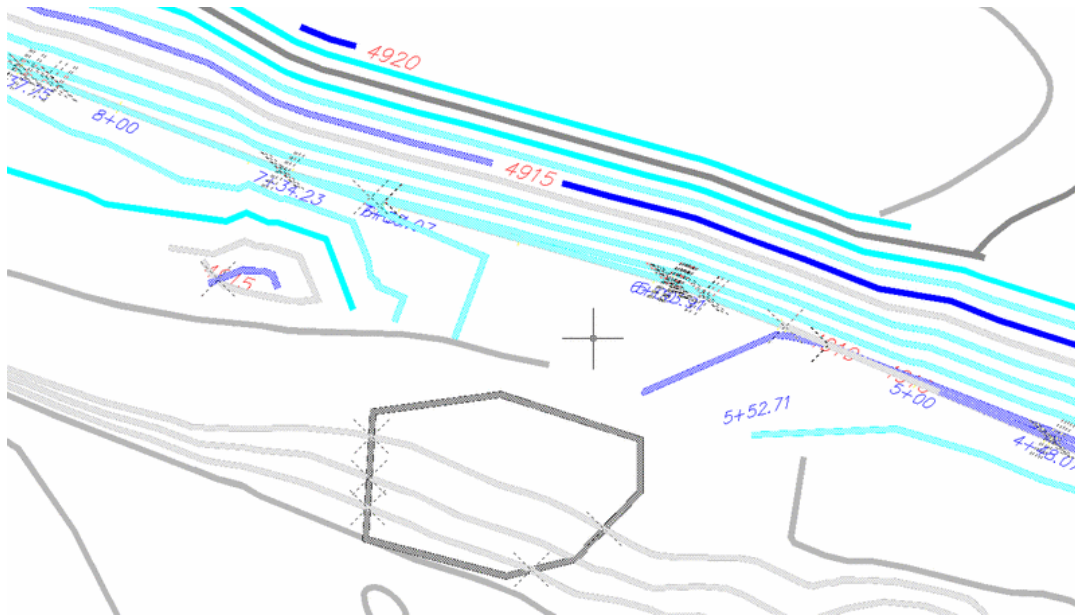
Pulldown Menu Location: Edit > Polyline Utilities > Check Polylines

Keyboard Command: highlight_nontangent

Prerequisite: Polylines

Highlight Crossing Plines

This command highlights selected polylines that are crossing in the drawing and have different elevations at the crossing. Every intersection point where the polylines cross are marked with a temporary X. A report is provided at the end where the X and Y of the intersection points are displayed with the two Z values and the Z difference. The command has the ability to repair crossing polylines by inserting a vertex in each polyline at the intersection and assigning a common elevation at this intersection.



Carlson Software Edit : c:\Carlson2007\USER\scaopr1.tmp

File Edit Settings

Open Save Print Exit Find Screen Hide

Crossing Breaklines Report 4/4/2006 15:00

Intersection	X	Y	Z1	Z2	Delta Z
1	544093.6085	160501.4266	1540.0000	0.0000	1540.0000
2	544244.8235	160540.6909	1540.0000	0.0000	1540.0000
3	544244.7077	160538.8089	1540.0000	0.0000	1540.0000
4	544158.7792	160696.2540	1540.0000	0.0000	1540.0000
5	544249.8746	160816.5767	1520.0000	0.0000	1520.0000
6	544332.9374	160495.1421	1520.0000	0.0000	1520.0000
7	543998.8958	160456.7193	1520.0000	0.0000	1520.0000
8	544154.6199	160751.1884	1520.0000	0.0000	1520.0000
9	544191.4615	160816.5767	1520.0000	0.0000	1520.0000
10	544990.2285	160281.4517	1500.0000	0.0000	1500.0000
11	544146.4027	160797.0493	1500.0000	0.0000	1500.0000
12	543927.0323	160425.3034	1500.0000	0.0000	1500.0000
13	544426.2473	160413.1184	1500.0000	0.0000	1500.0000

Prompts

Select polylines to check.

Select objects: *pick polylines to process*

Ignore zero elevations [**<Yes>/No**]? *press Enter for Yes to filter out polylines at zero elevation*

Reading points ... 1677

Finding points on breaklines ...

19 crossing polylines are highlighted.

Use Report Formatter [**Yes/<No>**]? *press Enter for No.* Use the Report Formatter to customize the report layout or export to Excel.

Minimum delta Z to report **<0.0>**: 2

Add polyline vertices at intersections [**Yes/<No>**]? *Y*

Set 3D polyline to crossing contour elev or average elevs [**Set/<Average>**]? *press Enter for Average.* The Set option applies to crossing polylines where one polyline is a 3D polyline with varying elevations and the other polyline is a contour polyline with a fixed elevation. For this case, the Set method will hold the elevation of the contour polyline and set the 3D polyline elevation to match the contour. The Average method sets the elevation of the intersection point as the average of the crossing polyline elevations at that point.

Maximum delta Z to average **<1.0>**: *press Enter.* This option will only add the intersection point with the averaged elevation if the elevation difference is less than this tolerance.

Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: xing_plines

Prerequisite: Polylines with elevations

Highlight Unclosed Polylines

This tool will evaluate polylines you select and highlight those that are open. It also provides options to close all or selected polylines from those found.

First select all polylines to evaluate. The tool will then display those that are open in a highlighted appearance. You will be offered an option to close all or selected polylines. If you wish to close all the open polylines, choose the All option. If you choose the Selected option you will be prompted to pick which polylines you want to close. As you pick each polyline it will be closed.

Prompts

Select the polylines to check.

Select objects: *pick polylines to process*

Open polylines are highlighted.

Close all or selected polylines [**All/<Selected>**]? *S*

Pick polyline to close: *press Enter to end or select polylines*

Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: unclosed

Prerequisite: A polyline

Close/Open Polylines

These commands allow you close or open multiple polylines respectively. You can also use the PEDIT command to open or close polylines individually.

Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline

Keyboard Command: closepl, openpl

Prerequisite: A polyline

Buffer Offset

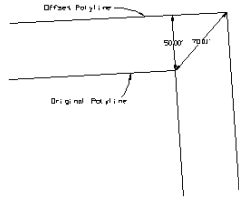
This command offsets a polyline, and maintains a fixed distance from the original polyline by placing an arc on convex corners. The standard *Offset* command can actually have a distance greater than the offset at corners. In the example shown, the distance between the corners of the original and offset polylines is 70.01, while the offset distance is 50.0. *Buffer Offset* makes an offset polyline that doesn't exceed the offset distance. This is useful when you want an offset that goes no further than the offset distance, such as wetland offsets. Later versions of AutoCAD can achieve the same effect using the standard *Offset* command by changing the system variable `OFFSETGAPTYPE` to 1.

Prompts

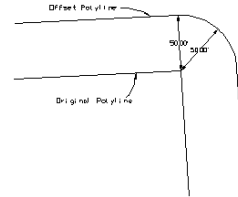
Enter the offset amount: 50

Select object to offset: *pick the original polyline*

Specify point on side to offset: *pick a point on the side to offset to*



Regular Offset



Buffer Offset

Pulldown Menu Location: Edit > Offset

Keyboard Command: boffset

Prerequisite: A polyline to offset

Change Polyline Width

This command sets the width of the selected polylines. In later versions of AutoCAD, the command *PEDIT* can also modify the width of multiple polylines.

Prompts

New width <1.0>: 2

Select Polylines/Contours to change width of:

Select objects: *pick polylines*

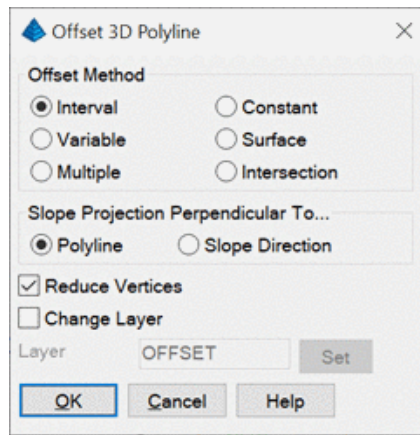
Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline

Keyboard Command: cwidth

Prerequisite: A polyline

Offset 3D Polyline

This command offsets a 3D polyline entity in both the horizontal and vertical directions. There are six offset methods.



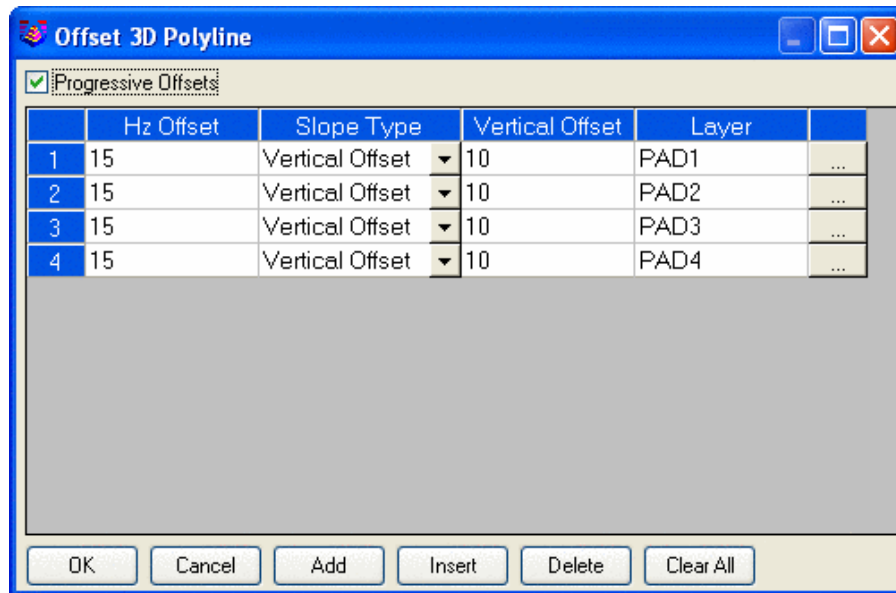
The **Interval** method applies one horizontal and one vertical offset to all the vertices of the polyline.

The **Constant** method has a horizontal offset and sets the elevation of the polyline to one constant elevation.

The **Variable** method allows you to specify each horizontal and vertical offset individually either by polyline segment or for each point. The vertical offset can be specified by actual vertical distance, percent slope or slope ratio.

The **Surface** method allows to offset/project a 3D polyline entity on to a surface (tin;flt;grd) based on cut and fill slope ratios.

The **Multiple** method allows multiple offsets of a 3D polyline with separate layers. User can add, insert and delete offsets rows and set individual layers. The option Progressive Offsets draws offsets progressively, i.e. successive offsets uses last drawn offset as base.



The **Intersection** method creates a 3D polyline at the intersection of two slopes from two reference 3D polylines.

The **Slope Projection Perpendicular To** option applies to sloping 3D polylines. The Polyline method creates the user-specified slope perpendicular to the polyline. The Slope Direction method accounts for the slope of the 3D polyline and makes the surface between the original and offset polylines to match the user-specified slope. For example, if the 3D polyline is at a 10% slope and the offset slope is at 2:1, then the Polyline method would create

surface slopes that are 2:1 perpendicular to the polyline while slightly steeper (1.96:1) for the actual slope that goes in the slope direction with the effect of the sloping 3D polyline. For the same case except with the Slope Direction method, the resulting slope perpendicular to the polyline is less steep (2.04:1) while the actual slope in the slope direction is exactly 2:1.

The **Reduce Vertices** option removes collinear points from the offset polyline.

The **Change Layer** option sets the layer for the offset polyline. Otherwise the offset polyline will use the layer of the original polyline.

Prompts

Enter the offset method [<Interval>/Constant/Variable/Surface/Multiple]: *press Enter*

Vertical/<Horizontal offset amount>: *15*

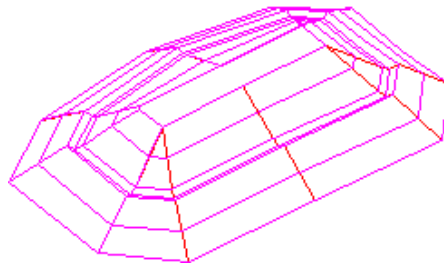
Percent/Ratio/Vertical offset amount <0>: *10*

Select a polyline to offset (Enter for none): *select a 3D poly*

Select side to offset: *pick a point*

Select a point on the graphics screen that is in the direction of the side of line to offset.

Select a polyline to offset (Enter for none): *press Enter*



Pulldown Menu Location: Edit > 3D Polyline Utilities

Keyboard Command: offset3d

Prerequisite: 3D polyline

Fillet 3D Polyline

This command fillets two segments of a 3D polyline with the given radius. The standard *FILLET* command does not support 3D Polyline entities. Since 3D polylines cannot have arcs, this command draws the fillet arc as a series of short chords. The elevations along the curve are interpolated from the 3D polyline.

Prompts

Fillet corner of a polyline or intersection of two polylines [<Corner>/Intersection]? *press Enter*

Enter fillet radius <10.00>: *press Enter*

Select a corner point on polyline: *pick 3D polyline near meeting point of two segments*

Select a corner point on polyline: *pick 3D polyline near meeting point of two segments*

Select a corner point on polyline: *press Enter* (to end command)

Pulldown Menu Location: Edit > 3D Polyline Utilities

Keyboard Command: fillet3d

Prerequisite: 3D polyline

Join 3D Polyline

This command joins *3DPOLY* entities into a single 3D polyline entity.

Prompts

Select the 3D polyline to join: *pick a 3D polyline*

Select the other 3D polyline to join: *pick a 3D polyline that has a common endpoint with the first 3 segments added to the polyline.*

Pulldown Menu Location: Edit > 3D Polyline Utilities

Keyboard Command: join3d

Prerequisite: Plot the *3DPoly* lines to use for selection

Trim 3D Polyline

This command trims lines and polylines. The intersection for the trim location is calculated in 2D. The elevation at the intersection of the trimmed polyline is interpolated from the existing polyline vertices.

Prompts

Select trim edge polylines.

Select objects: *pick polyline for trim boundary*

Select polyline to trim: *pick polyline*

Select polyline to trim (Enter to end): *press Enter*

Pulldown Menu Location: Edit > 3D Polyline Utilities

Keyboard Command: trim3d

Prerequisite: linework

Add Point by Two Slopes

This command inserts a vertex into a 3D Polyline between two points based on the slopes specified for these two points on polyline.

Prompts

Select polyline to process: *select a polyline*

Select first point on polyline: *select a point on polyline*

Enter percent slope from first point: *-1.0*

Select second point on polyline: *select a second point on polyline*

Enter percent slope from second point: *-1.0*

Pulldown Menu Location: Edit > 3D Polyline Utilities

Keyboard Command: plzslopes

Prerequisite: 3D Polylines

Add Points At Elevation

This command inserts vertices into a 3D Polyline at a specific elevation, or elevation interval, by interpolating between existing elevations in the polyline.

Prompts

Add single elevation or elevation interval [Single/<Interval>]? *press Enter*

Enter Elevation Interval: 50

Select 3D polylines to process. *pick 3D polyline(s)*

Select objects: 1 found

Select objects:

Processing polylines ...

Added 10 points to polylines.

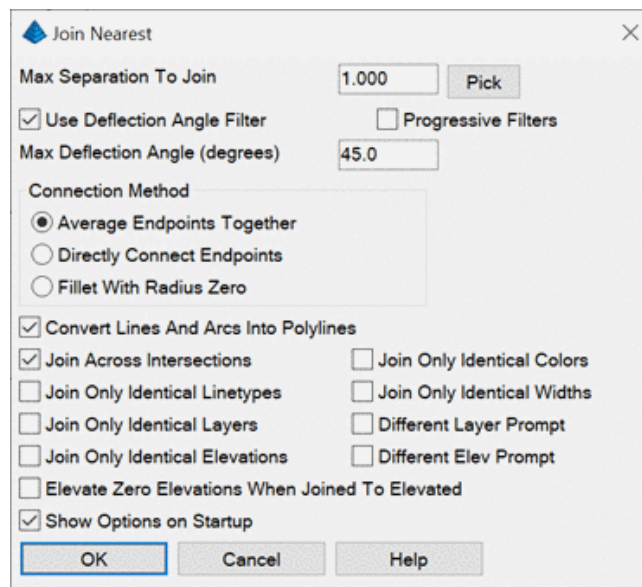
Pulldown Menu Location: Edit > 3D Polyline Utilities

Keyboard Command: addplz

Prerequisite: 3D Polylines

Join Nearest

This command joins lines, arcs and/or polylines together. While the *PEDIT-Join* command requires the endpoints to match, Join Nearest will allow you to join entities whose endpoints do not exactly meet. You specify the maximum separation distance to join, along with other options, in the dialog box shown below. Also you can join many entities at once.



Max Separation to Join: Entities whose endpoints are spaced apart greater than this value will not be joined. You may use the pick button to specify this value by picking two points on the screen.

Max Deflection Angle (degrees): This option will not join any lines if the angle between them is greater than this angle in degrees.

Progressive Filters: This option automatically runs Join Nearest multiple times with greater tolerances starting with tight tolerances and going up to the specified limits. This method helps ensure that the best matches get priority.

Connection Method: Determines how to connect the endpoints. See the illustration below.

1. **Average Endpoints Together:** New vertex will be located at midpoint between two original endpoints (see illustration below on left).
2. **Directly Connect Endpoints:** Original endpoints are connected with new segment (see the middle illustration below).
3. **Fillet with Radius Zero:** Same as the *FILLET* command using zero radius (see the illustration on right).

Convert Lines and Arcs Into Polylines: When checked, automatically converts lines and arcs into polylines. If not checked, lines and arcs are joined but remain separate entities.

Join Across Intersections: This option applies to cases where more than two linework endpoints come together such as a Y intersection. In these cases, there are multiple possible connections. When this option is on, the program will automatically choose one of the possible connections. Otherwise, the program will not connect any of them.

Join Only Identical Widths: When checked, only polylines with the same width will be joined.

Join Only Identical Layers: When checked, only entities on the same layer will be joined.

Join Only Identical Colors: When checked, only entities with the same color will be joined.

Join Only Identical Linetypes: When checked, only polylines with the same linetype will be joined.

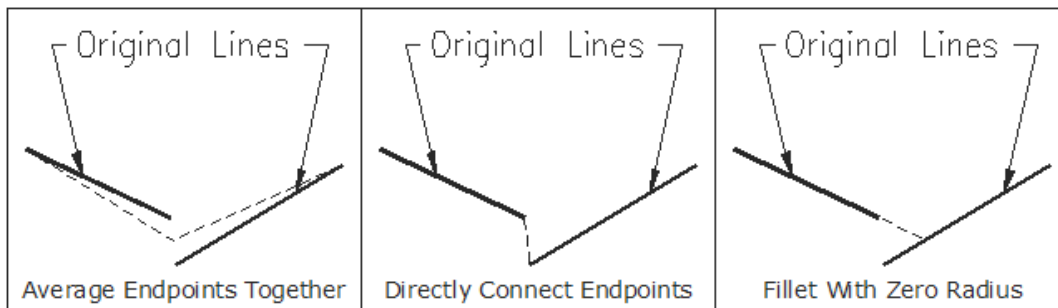
Join Only Common Elevations: When checked, only endpoints located on the same elevation will be joined.

Different Layer Prompt: When Join Only Identical Layers is off, then this option will prompt for which layer to use when it finds a connection between two different layer names.

Different Elevation Prompt: When Join Only Common Elevations is off, then this option will prompt for which elevation to use when it finds a connection between two different elevations.

Elevate Zero Elevations When Joined To Elevated: This option applies when joining a combination of linework at elevation and linework at zero elevation. When checked, zero elevation vertices will get assigned the elevations from connected neighboring vertices.

Show Options on Startup: Controls whether to show the options dialog at the start of the command. When this option is turned off, you can type S for Settings at the command line to bring up the dialog.



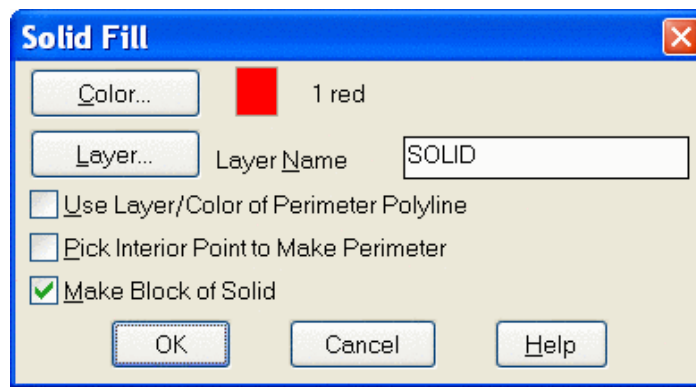
Pulldown Menu Location: Edit

Keyboard Command: nearjoin

Prerequisite: Lines or polylines to be joined

Solid Fill Polyline

This command fills the interior of closed polylines with 3D Faces to make the polyline areas appear solid. Closed polylines for exclusion areas can be used to exclude areas from the fill. Text can also be selected to exclude the text area from the fill. As an alternative, you can use the *HATCH* command, which creates an associative link between the hatch object and its boundary, interior boundary and any text that is excluded.



Prompts

Solid Fill Dialog Box

Use Layer/Color of Perimeter Polyline This option uses the layer and color of the perimeter polyline for the solid fill faces instead of those in the dialog box.

Pick Interior Point to Make Perimeter Instead of requiring a closed perimeter polyline, this option defines the perimeter by the boundary of the area around a picked point.

Make Block of Solid The solid is created by adjoining 3D faces. This option groups the 3D faces into a block.

Select Inclusion perimeter polylines.

Select objects: *pick closed polyline*

Select Exclusion perimeter polylines.

Select objects: *press Enter*

Select Text to Exclude from fill.

Select objects: *press Enter*

Pulldown Menu Location: Edit

Keyboard Command: solidfill

Prerequisite: A closed polyline

3D Entity to 2D

This command changes a 3D Line, Arc, Circle, Polyline, Hatch, Leader, Solid, 3D Face, Insert or Point to 2D. In 2D, the entity has the same Z coordinate for all vertices. When the program detects a 3D polyline with all vertices with the same elevation, there is an option to convert to a 2D polyline with this elevation. Otherwise, the entered elevation is used.

Prompts

Select/<Enter Elevation <0.00>: *press Enter*

Select entities for elevation change.

Select objects: *pick a 3D polyline*

3DPOLY to 2DPOLYLINE

Number of entities changed > 1

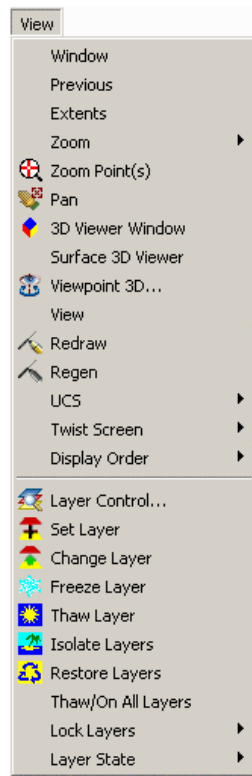
Pulldown Menu Location: Edit

Keyboard Command: 3dto2d

Prerequisite: None

View Menu

In addition to powerful CAD display and view commands, the Carlson View menu has some additional commands. The commands in the top section effect the screen display size and location, and the bottom section commands change layers.



Zoom Points

This command centers the screen to a user-specified point. The point can be specified by either the point number or description. The command searches the current coordinate (.CRD) file. Besides centering the screen, the magnification can also be changed. The default value is the current magnification. To zoom in, enter a smaller value and to zoom out, enter a greater value.

Prompts

Find by point number or description [<Number>/Desc]? N

Point number or range of point numbers to find <1>: 2079

We want to find point number 2079

Magnification or Height <179.50>: press Enter

Accept the default zoom magnification

Pulldown Menu Location: View

Keyboard Command: zoompnt

Prerequisite: A .CRD file

Zoom Selection

This command zooms the display to fit the selected entities. For example, if you run *Viewpoint 3D* and your viewport only shows two small dots of entities that are far apart, then you can use *Zoom Selection* to select the entities of one of these dots and quickly zoom the display to these entities.

Prompts

Select objects to zoom onto:

Select objects: *select entities*

Pulldown Menu Location: View

Keyboard Command: zoom_on

Prerequisite: Entities

Zoom Most

This command zooms to show the bulk of the entities in the drawing and avoid outliers. This zoom method is helpful when the drawing has a few entities far off and regular zoom extents results in showing small dots because it zooms out so far. When outliers are detected, there is a choice to either zoom to the bulk of the entities so you can see your main drawing, or to zoom to the outliers so you can see the entities that are far away. When no outliers are found, this command reports that none are found and does a regular zoom extents.

Prompts

Zoom to [<Most>/Outliers]? *press Enter for Most*

Pulldown Menu Location: View > Zoom

Keyboard Command: zoommost

Prerequisite: Entities

Zoom Extents on All Layouts

This command does a zoom extents for all the layouts in the drawing.

Pulldown Menu Location: View > Zoom

Keyboard Command: zoom_layout

Prerequisite: Layout

Align Paper With Model Space

This command works in paper space for a layout that has a viewport. The command aligns a point from the model view in the viewport to a point in paper space. The command prompts for the point in model space and the point in paper space. Then the command pans the model view to align with the picked point in paper space.

Prompts

Pick point in model space: *pick a point*

Pick point in paper space: *pick a point*

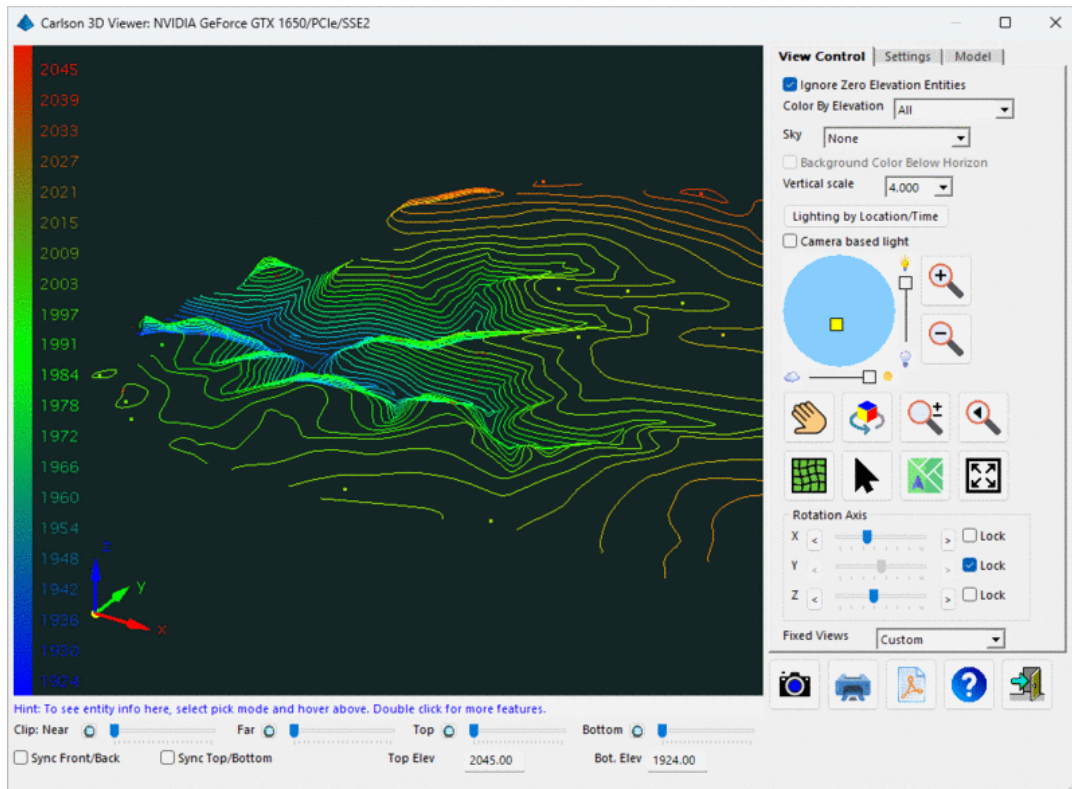
Pulldown Menu Location: View > Pan

Keyboard Command: alignps

Prerequisite: None

3D Viewer Window

This command loads the selected 3D faces, blocks, polylines, lines and points into a separate 3D Viewer window. Some of the features of this viewer include the ability to zoom in and out, pan, rotate around the X-Y-Z axes, and shade entities with user-positioned lighting.



The 3D Viewer Window has several mouse and keyboard controls that can assist with navigating the scene. These are listed below:

- Hold left-mouse button: Rotates/pans/zooms the scene based on the current mode. You can change the mode by clicking one of the control icons.
- Hold right-mouse button: Zooms in/out.
- Hold middle-mouse button: Pans around the scene.
- W key: Zooms in.
- S key: Zooms out.
- A key: Pans left.
- D key: Pans right.
- Q key: Pans up.
- Z key: Pans down.
- E key: rotates down.
- C key: rotates up.
- X key: rotates left.
- V key: rotates right.

Ignore Zero Elevations: When enabled, the 3D viewer will not display entities at zero elevation.

Color By Elevation: This will color the entities by elevation.

The *None* option will not modify the color of entities in the viewer.

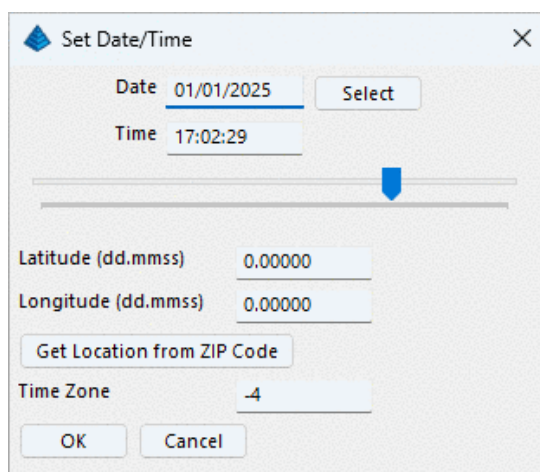
The *All* option will color all entities by elevation. The elevation color legend will be displayed on the left of the window and can be adjusted via the Color By Elevation Scale controls.

The *Surface Only* option will only color surface files by elevation. Other entities such as polylines and 3D faces will not be colored by elevation.

Sky: This option controls the background image for the 3D viewer. By default, no background image will be displayed. When a background image is displayed, you can toggle the **Background Color Below Horizon** checkbox to hide the bottom half of the background image.

Vertical Scale: This option sets the vertical scale factor for the 3D viewer.

Lighting by Location/Time: This button will display the below dialog, which allows you to set the lighting as it would appear at a specific location at a specific time.



Date/Time: These values set the date and time for the lighting. The **Select** button will allow you to pick the date from a calendar. The horizontal slider below the time will allow you to set the time of day.

Latitude/Longitude: These values set the location of the viewpoint

Time Zone: This value adjusts for the various time zones. Values should be integers representing GMT adjustments. For example, the Eastern Standard Time (EST) in the USA will use a value of -5.

Control



Action

This control represents the position of the sun in plan view. If the yellow square is in the center of the blue circle, the sun is in a zenith. If the yellow square is near the edge of the circle, the sun is near the horizon. To move the yellow square, simply drag it to a new location, or click on the new location. The slider bars on the sides control the intensity and brightness of the display. Switch to Pan mode. Click and drag to pan.



Switch to Rotation mode. When the cursor is placed near the outer edge of the view, a "Z" cursor is presented that permits rotation around the Z-axis. When the cursor is placed further into the interior of the view, an "X,Y" cursor is presented that permits the tilt angle of the view to be adjusted.



Switch to Dynamic Zoom mode. Click and drag to zoom in and out.



Zoom Previous.



Toggle shading of 3D Faces and Surfaces. The shading of these entities is controlled by the Shading Mode control.



Switch to Pick mode. In this mode, hovering over an entity will display data such as the layer, entity type, elevation, and length. Double-clicking an entity permits additional actions to be performed on the entity including the ability to change the layer of the entity and/or setting the entity to an elevation of zero (0). When hovering over a surface, the coordinates and slope of the surface at the cursor location will be displayed. Clicking a point on the surface will set a new reference point from which the distance and slope to the cursor will be displayed (click a point, move the mouse to measure the distance/slope)



Resets the 3D view to plan.



Zooms to the extents of the view - ensuring that all objects in the viewer will be visible.

Rotation Axis: Permits the use of "slider" controls to orient the view in the X, Y and/or Z axis direction(s). Axes may be locked to prevent rotation about that axis.

Fixed Views: Permits the view to be displayed from one of six different directions:

1. Custom - This option will be displayed after manually rotating the view. Selecting this option will not return to a predefined view.
2. Plan View - Sets the view directly over the site, looking straight down. This is the same as the Reset to Plan button.
3. NE - Sets the view as shown from the Northeast looking to the Southwest in a downward direction.
4. SE - Sets the view as shown from the Southeast looking to the Northwest in a downward direction.
5. SW - Sets the view as shown from the Southwest looking to the Northeast in a downward direction.
6. NW - Sets the view as shown from the Northwest looking to the Southeast in a downward direction.

TIN Edit: This option is only available when using the Surface 3D Viewer Command and when using the Pick Mode. This allows you to modify TIN files by picking points on the surface.

Swap Tin Edge will rotate the edge of two adjacent triangles, but this is not possible for all triangles on the surface.

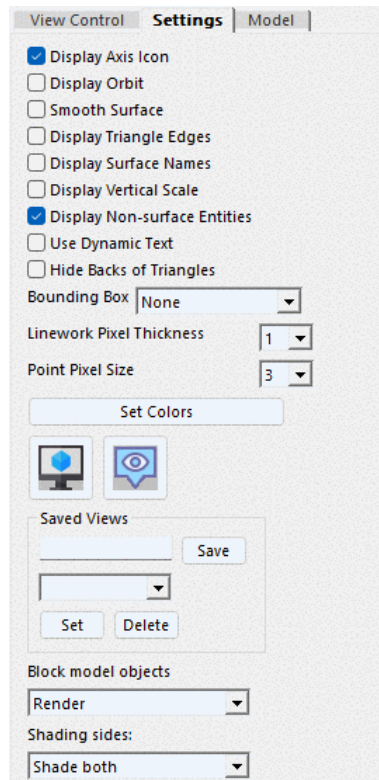
Add Breakline will add a breakline to the surface after selecting the starting and ending points on the surface.

Remove Point will remove a point on the surface and retriangulate the affected area.

Remove Triangle will remove a triangle from the surface, leaving a gap.

Set Point Elevation will allow you double-click a point on the surface and enter a new elevation for that point.

XRay Cursor: This option is only available when using the Surface 3D Viewer Command and when using the Pick Mode. This will hide the surface file near the cursor, allowing you to see objects beneath it.



Display Axis Icon: This option toggles the display of the X-Y-Z axes icon in the lower left corner of the window.

Display Orbit: This option toggles the display of a graphic guide for controlling the view angle and position using the mouse movements similar to the AutoCAD Orbit routine.

Smooth Surface: This option shades 3D faces in a way that appears to remove sharp transitions from one entity to another. This option affects both surfaces and solids.

Display Triangle Edges: This option toggles the edge lines of triangles that make up surfaces. The triangle edge color can be set with the **Set Colors** button.

Display Surface Names: This option toggles the display of the file names for the surfaces currently being viewed.

Display Vertical Scale: This options toggles the display of the vertical scale.

Display Non-Surface Entities: This option toggles the display of entities that have been tagged as "non-surface" by the Tag Non-Surface Entities/Points commands.

Use Dynamic Text: This option automatically resizes and rotates text entities so that they are always readable. When the option is first enabled, you will need to restart the viewer for the changes to take effect.

Hide Backs of Triangles: This option toggles the display of the backs of 3D Faces. When enabled, even the edges of the triangles will be hidden.

Bounding Box: This dropdown menu controls the display of a 3D box around the limits of the data.

The *None* option will not display the bounding box.

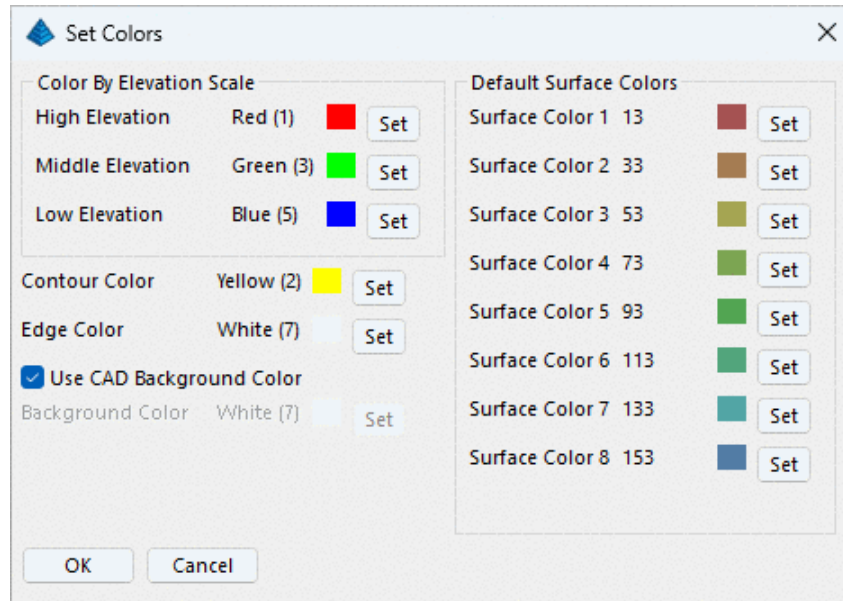
The *Box Only* option will display simple bounding box around the objects in the 3D view.

The *Box with Labels* option will display a bounding box with labels to indicate the range of X, Y, and Z values in the scene.

Linework Pixel Thickness: This value sets the display thickness for polylines.

Point Pixel Size: This value sets the display thickness for points.

Set Colors: This button opens the dialog shown below.



Color By Elevation Scale: These three colors are used for the Color By Elevation option. The program will interpolate between these colors for the color scale.

Contour/Edge color: These colors control the coloring for contours and triangle edges. Note that this does not affect the coloring of contour polylines that are loaded into the viewer, but only contours that are generated within the viewer.

Use CAD Background Color: When enabled, the background color of the 3D viewer will match the CAD background color. If this option is disabled, you can set a different background color.

Default Surface Colors: These colors set the default colors for new surfaces (Grid or TIN files) that are loaded into the 3D viewer. The colors of the surfaces may be changed after they have been loaded.

Control

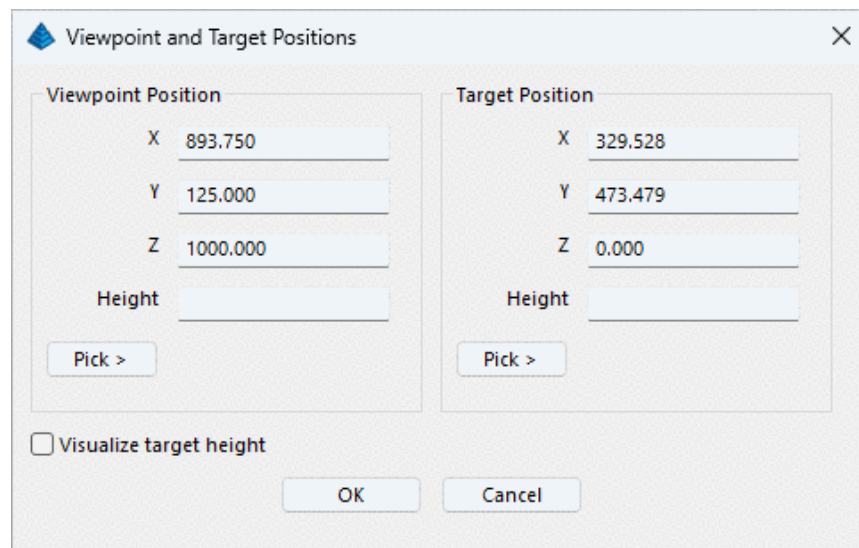


Action

Sets the drawing view to match the view shown in the 3D viewer window.

Sets the view position and target position by coordinates according to the below dialog.

Settings Controls



The positions can be entered in the edit boxes or you can use the respective **Pick >** button to pick a point in the drawing. The program will pick up the elevation of the surface for picked points and then the **Height** above the position can be entered. For example, to check sight distance the view position could be a point on a road at the driver's eye height and the target position could be the distance to check.

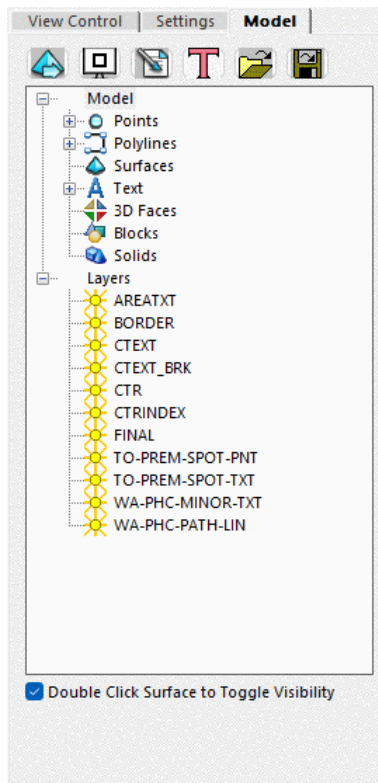
Saved Views: This option allows for naming and saving a 3D view for easy recall later. Named views can be selected from the pull-down and the active view can be deleted from the list.

Block Model Objects: This option has three choices when loading block model entities:

1. Leave as Points - this option will display blocks as points. In this mode, vertical stacks of blocks will be composited by color. For example, if 5 blocks of the same color are on top of one another, only one point will be displayed for those 5 blocks. This greatly reduces the number of points to be displayed in the viewer, but this viewing mode can be difficult to interpret for relatively homogeneous block models.
2. Render - this option will display each block as a solid.
3. Prompt Each Time - this option will prompt you to select the viewing mode each time a block model is loaded into the viewer.

Shading Mode: When the Shading control is enabled, the rendering of the shaded 3DFACE entities (usually used to represent a surface model) will vary based on:

1. Shade Front - Shades only the "positive" normal direction.
2. Shade Back - Shades only the "negative" normal direction.
3. Shade Both - Shades both the "positive" and "negative" normal directions.



Control



Action

Adds a single surface file to the 3D Viewer. You may also add one or multiple surfaces to the viewer by right clicking on "Surfaces" in the tree view.



Allows you to select more entities from the drawing to add to the 3D Viewer.



Allows you to edit properties of the currently selected entity.



Adds a text label into the scene. The program prompts for the text string, position and properties. The text can be drawn to CAD also.



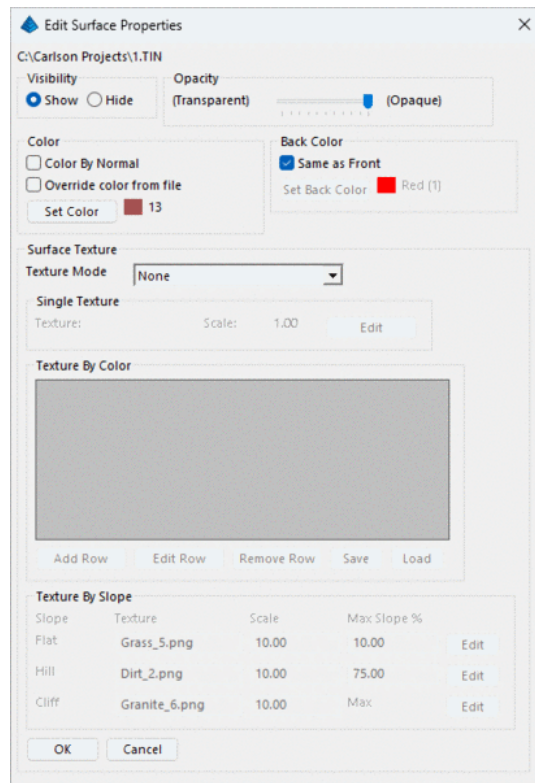
Loads a 3DX file into the viewer. This files contains all entities within a saved 3D View.



Save the entities in the 3D Viewer to a 3DX file.

Model Controls

Within the "model" tab is a "tree-view" of the various entities that comprise the view along with the listing of layers upon which the entities are found. Each "branch" in the tree view may be expanded by clicking the "+" symbol to the left of the branch. Entities may be displayed (thawed), hidden (frozen), or edited by right clicking the entity name and selecting the appropriate option. The below dialog shows the properties of a surface that may be edited.



Visibility: This option toggles the visibility of the entity.

Opacity (Surfaces/Solids): This option controls the opacity of the entity. A lower opacity results in increased surface transparency and is helpful for viewing sub-surface utilities such as Storm Sewer pipes and manholes.

Color By Normal (Surfaces/Solids): When enabled, this option will color the surface/solid by the direction perpendicular to each face. This view can be useful for identifying particularly steep areas of the surface. The Color By Elevation option must be disabled to show the designated color.

Override color from file: When enabled, this option will override any color assignments and display the entity as a single color. The **Set Color** button below this option sets the color for the entity. The Color By Elevation option must be disabled to show the designated color.

Back Color (Surfaces/Solids): This option controls the colors of the backs of triangles. If you disable the Same as Front option, you can set the backs of the triangles to a different color.

Texture Mode (Surfaces/Solids): This dropdown controls how texture images are applied to surfaces and solids.

None will not apply a texture to the surface/solid

Single Texture will apply a single texture image to the surface/solid. The texture image and the Scale of the texture may be set by clicking the Edit button.

By Color will set different texture images for each colored area of the surface. For example, surfaces output from RoadNet, may have a different color for the asphalt compared to ditches. This option will then allow you to apply one texture to the asphalt and another to the ditch. When enabled you can add a row to the Texture by Color table, first selecting a color and then specifying the texture to use in place of that color.

By Color, Single as Default is the same as the *By Color* option, but you are allowed to set a default texture for any colors not included in the list.

By Slope sets the texture of the surface according to the slope of the surface. When enabled, you will be able to set

a texture, scale, and max slope for each zone.

Shading Mode (Solids): This option controls which side of the 3D faces of the solid should be shaded.

These controls are always displayed at the bottom of the window.

Control



Action

The **Clip Plane** slider bars hide a portion of the view. Sliding the bar to the right will clip more of the view. This is helpful for producing quick "section" views of the data.

The **Near** slider bar will clip objects closer to the camera, while the **Far** slider bar will clip objects further away from the camera. Similarly, the **Top** and **Bottom** slider bars will clip higher and lower objects, respectively. Specific elevations for the **Top** and **Bottom** clipping plans may also be set by entering values in the **Top Elev** and **Bottom Elev** text boxes.

Clicking the cyan button to the left of each slider bar will allow you to pick a point in the 3D view to set the clip position.

The **Sync Front/Back** toggle will move both the **Near** and **Far** slider bars by the same amount when enabled. This can be used to hold a specific viewing width while. The **Sync Top/Bottom** toggle has a similar effect on the **Top** and **Bottom** slider bars.

This button takes a screenshot of the 3D Viewer. Several different image file formats are supported including bmp, png, jpg, xpm and gif. The image resolution and color depth may be set with this option.

This button exports the 3D Viewer to a PDF report with project information. The default project information is set under Carlson Configure > General Settings. The dialog that appears for this command is shown below.

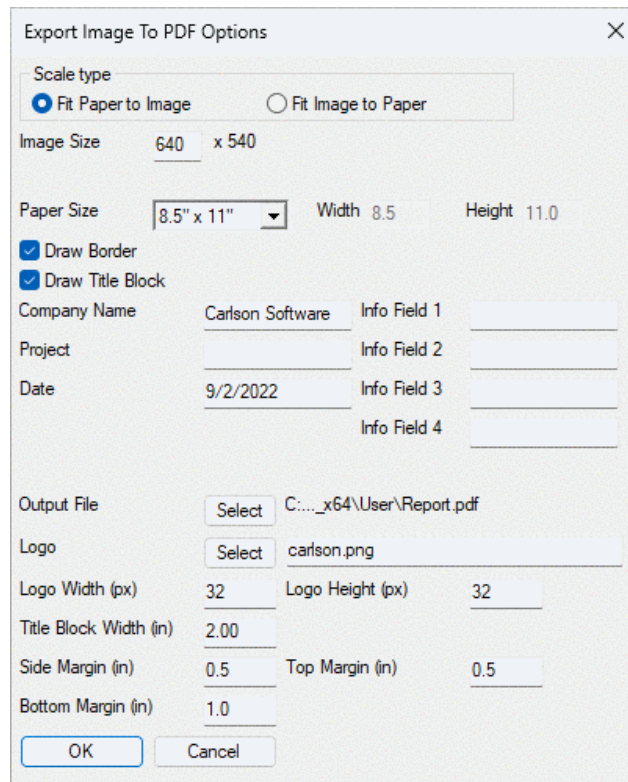
This button exports the 3D Viewer to a 3D PDF. This function is further described in the to the 3D Viewer to 3D PDF help article. Note that you must have access to the CADNET module in order to use this command.

This button opens the help documentation you are currently reading.

Exit the 3D viewer window.



Common Controls



Scale Type: This option controls how the image and the paper will be scaled to match one another. When the *Fit Paper to Image* option is selected, you will be able to manually set the **Image Size**.

Paper Size: This option sets the size of the PDF.

Draw Border: This option toggles the addition of a border around the PDF.

Draw Title Block: This option toggles the addition of a title block in the PDF. Information such as the **Company Name**, **Project**, **Date**, and **Logo** will be included in the title block. The logo may be selected by clicking the **Select** button and the dimensions may be specified in pixel dimensions.

Top/Side/Bottom Margin: These values set the margins around the PDF in inches.

Note regarding speed/performance: If the 3D Viewer Window is responding slowly to camera movements, changes to settings, etc, you may need to toggle a setting to improve performance. The setting is found under the Settings Pulldown Menu > Carlson Configure > General Settings. There you can turn off the "Use Software Rendering" setting. This setting is sometimes enabled after a program crash in an attempt to prevent further crashes. It acts a sort of "safe mode" with greatly reduced performance.

- To quickly view just a surface file, use the Surface 3D Viewer command.
- To visualize a site with animated vehicle controls, use the Surface 3D Fly-Over command.

Pulldown Menu Location: View > 3D View

Keyboard Command: cube

Prerequisite: Entities to display

3D Viewer 3DX Model File

This command selects a 3DX file to view in the 3D Viewer Window. The 3DX file contains all the elements for the scene. Use the 3D Viewer Drawing, 3D Viewer Surface File or Surface 3D FlyOver routines to create a 3DX file.

Pulldown Menu Location: View

Keyboard Command: cube_3dx

Prerequisite: .3DX file

Surface Object View Properties

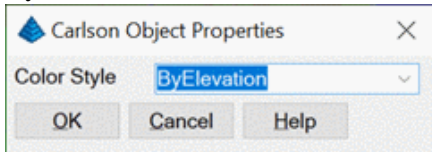
This command sets the display properties for Carlson triangulation, grid and point cloud objects. The Color Style has these choices:

CAD Layer: Uses the color of the layer assigned to the surface object.

Object Colors: Uses the colors assigned to the surface object.

By Elevation: Colors by elevation range with blue for low elevations and red for high elevations.

By Normal: Colors based on the direction the surface is facing.



Pulldown Menu Location: View > 3D View

Keyboard Command: view_prop

Prerequisite: Carlson Surface Object

Draw Model View

This command draws a rectangular polyline in Model space for the outline of a selected viewport in Paper space. Before running this command, switch to a Layout that has the viewport to draw. The polyline is drawn in the drawing current layer. After drawing the polyline, the program switches to Model space.

Prompts

Pick viewport to draw:: *pick a viewport*

Pulldown Menu Location: View > Viewports

Keyboard Command: mview_draw

Prerequisite: Layout with a viewport

Set UCS to World

This command sets the UCS (user coordinate system) to the world coordinate system (WCS). Carlson command work exclusively in the world coordinate system. In AutoCAD, it is possible to change the coordinate system from WCS. If you receive a drawing in which the coordinate system is not set to world, use this command to restore the UCS.

Prerequisite: None

Keyboard Command: ucs_world

Set Level

This command assigns a level name to the selected entities. The level is an optional, additional name that can be assigned to entities and used to filter entities for making selection sets.

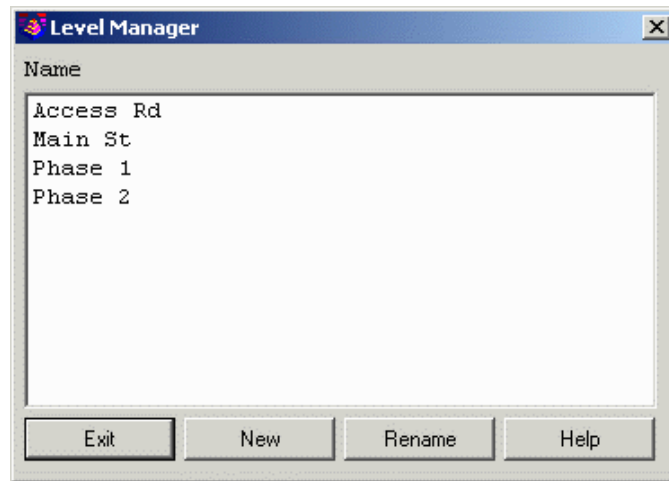
Pulldown Menu Location: View > Levels

Keyboard Command: levelset

Prerequisite: None

Level Manager

This command lists, creates and renames level names. The dialog shows the level names defined in the drawing. Use the New button to add a level name. Use the Rename button to change a level name.



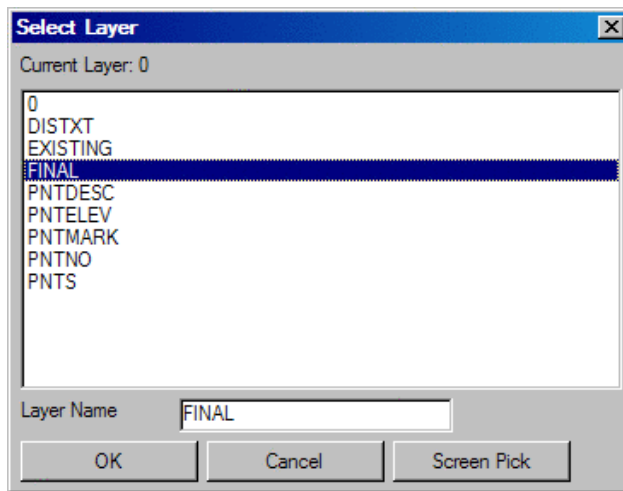
Pulldown Menu Location: View > Levels

Keyboard Command: levelmgr

Prerequisite: None

Change Layer

This command allows you to change the layer of a group of entities by selecting the group of entities. The layer name to assign can be either typed or read from an existing entity by picking an entity that is on the layer that you want to change the group to.



Prompts

Select entities to be changed.

Select objects: *pick entities*

The Select Layer dialog appears *select a layer from the list, or select Screen Pick*

If Screen Pick is chosen,

Pick entity with layer to change to: *pick another entity* This assigns the selected entities to the layer of this entity.
or

Enter new layer name or pick entity with layer (Enter/<Pick>)? *E*

Enter new layer name: *FINAL* This assigns the selected entities to the FINAL layer.

Pulldown Menu Location: View

Keyboard Command: lchg

Prerequisite: None

Freeze Layer By Selection

This command will freeze layers by picking entities on that layer. The entity selection is done by selection set for selecting one or more entities.

Prompts

Select entities on layers to be frozen.

Select objects: *pick entities*

Pulldown Menu Location: View

Keyboard Command: loff

Prerequisite: None

Restore Due North

This command twists the screen to make due north vertical. When the grid projection is specified under the Drawing Setup command, this command will prompt for whether to use grid north or geodetic north. When the grid projection is not set, this routine automatically twists to grid north.

Prompts

Twist to Grid North or True Geodetic North [<Grid>/True]? *press Enter*

Pulldown Menu Location: View > Twist Screen

Keyboard Command: twist4

Prerequisite: None

Restore Layer

This command thaws the layers that were frozen by the *Isolate Layer* command.

Pulldown Menu Location: View

Keyboard Command: restore

Prerequisite: None

Set Layer

This command allows the user to change the current layer to a different layer by picking an entity on that layer.

Pulldown Menu Location: View

Keyboard Command: lset

Prerequisite: None

Turn Off By Selection

This command will turn off layers by picking entities on that layer. The entity selection is done by selection set for selecting one or more entities.

Prompts

Select entities on layers to turn off.

Select objects: *pick entities*

Pulldown Menu Location: View

Keyboard Command: layeroffs

Prerequisite: None

Freeze Layer

This command will freeze layers by picking entities on that layer. The entity selection is done one at a time. As entities are selected, the layers are frozen.

Prompts

Pick entity on layer to be frozen: *pick an entity*

Freezing layer

Pick entity on layer to be frozen (U-Undo,Enter to end): *press Enter*

Pulldown Menu Location: View

Keyboard Command: pickoff

Prerequisite: None

Surface 3D Viewer

This command is identical to the 3D Viewer Window, except that this one loads a Carlson Grid GRD, TIN or FLT file. After the file is selected, the same viewer documented in 3D Viewer Window appears.

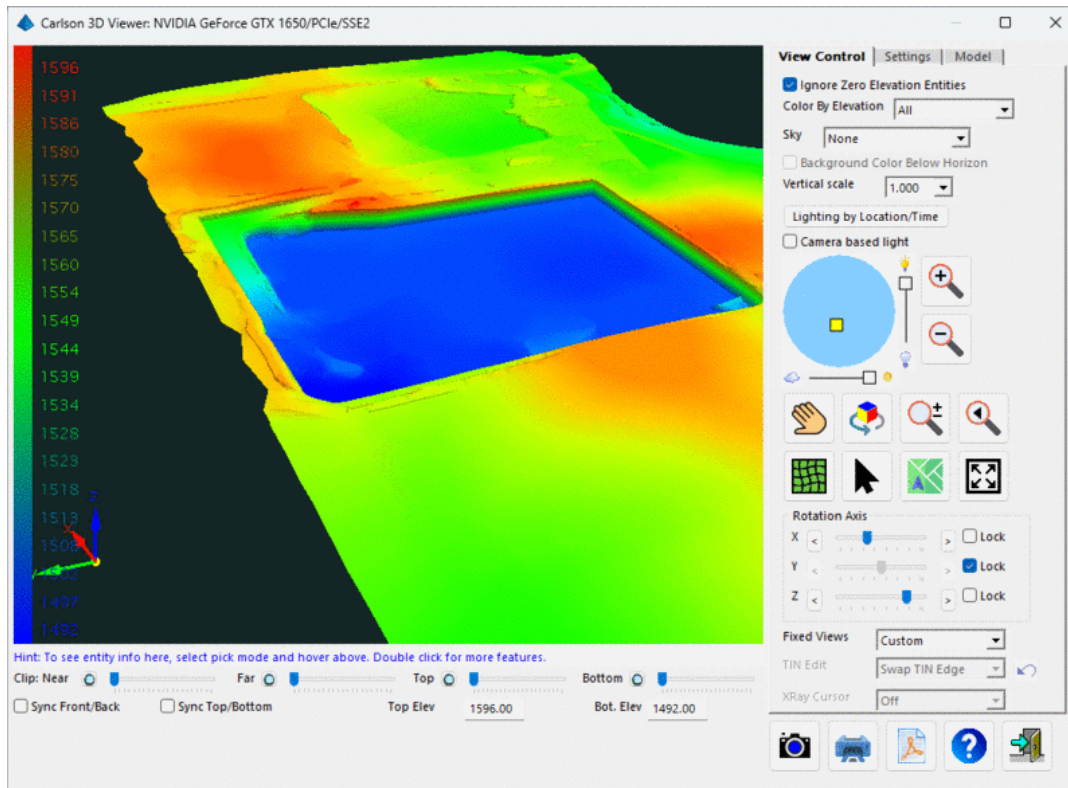
On the Model tab, the surface file is listed as an item in the model. Right-click on the surface name to access functions including:

Add Surface: Add another surface file to the scene

Add Multiple: Add many additional surfaces to the scene

Reload Surface: In case the source surface file is modified, reload the surface in the scene

Edit Surface Properties: Change display properties including colors, transparency, texture



- To view entities within the DWG file (and optionally, Carlson surface files), use the 3D Viewer Window command.
- To visualize a site with animated vehicle controls, use the Surface 3D Fly-Over command.

Pulldown Menu Location: View > 3D View, Civil > Surface > 3D Views

Keyboard Command: cube_surface

Prerequisite: Carlson surface (TIN, FLT, GRD) to display

Change Layer Color

This command changes the color of a layer. The layer is selected by picking an entity on the layer. Then the program prompts for the color to set.

Prompts

Select entity on layer: *pick an entity*
Color dialog

Pulldown Menu Location: View
Keyboard Command: lcolor
Prerequisite: A drawing entity

Add/Replace Layer Prefix

The Add Layer Prefix command adds a string to the beginning of all the layers in the drawing. The Replace Layer Prefix looks for a string at the beginning of layers and replaces this string with a new string.

Add Layer Prefix

Prompts

Layer prefix to add: PRE_

Replace Layer Prefix

Prompts

Old layer prefix to replace: PRE_
New layer prefix to add: POST_

Pulldown Menu Location: View > Layer Utilities
Keyboard Command: addLayerPrefix, replaceLayerPrefix
Prerequisite: None

Freeze Layer By Pick

This command will freeze layers by picking entities on that layer. The entity selection is done one at a time. As entities are selected, the layers are frozen.

Prompts

Pick entity on layer to be frozen: *pick an entity*
Freezing layer
Pick entity on layer to be frozen (U-Undo,Enter to end): *press Enter*

Pulldown Menu Location: View
Keyboard Command: pickoff
Prerequisite: None

Thaw Layer

This command thaws the layers frozen by the Freeze Layer command.

Pulldown Menu Location: View

Keyboard Command: lon

Prerequisite: None

Isolate Layer

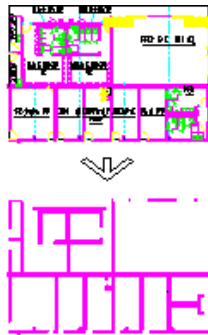
This command freezes all the layers except the ones you select an entity on. The program prompts to see if you would like to retain the POINT layers which keeps the Carlson point layers from freezing. By default, these layers include PNTNO, PNTMARK, PNTDESC, and PNTELEV.

Prompts

Select objects on layers to isolate.

Select objects: *pick entities*

Retain POINT layers [Yes/<No>]? Press Enter



Isolate the wall layer by picking one wall line

Pulldown Menu Location: View

Keyboard Command: isolate

Prerequisite: None

Twist Screen: Standard

This command will twist the screen orientation to where something other than the north direction is toward the top of the screen/drawing. It does not do a coordinate rotation, the drawing coordinates remain unchanged. Use commands on the *Points* menu, such as *Rotate Points* and *Translate Points*, if you want to do a coordinate rotation or translation.

Prompts

This routine prompts for the twist angle then adjusts the screen and cross-hairs to that angle. This is a modification of the *DVIEW* command. The twist angle is always measured counterclockwise with 0 degrees being to the east/right.

Pulldown Menu Location: View > Twist Screen

Keyboard Command: twist1

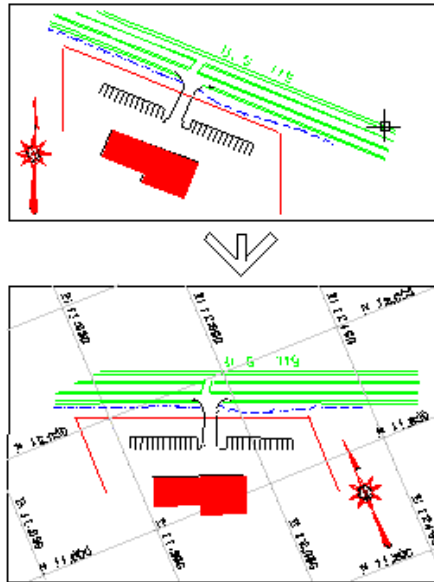
Prerequisite: None

Twist Screen: Line Pline or Text

This is a variation of the previous command that allows you to select a line, polyline, or text in your drawing that you want to be aligned parallel to the east-west direction of the graphics screen. Think of the entity you select as a pointer or arrow that will point in the east direction of the screen after you select it. Select the line, polyline, or text closest to the end point which you want to be the horizontal or east direction of the screen.

Prompts

Pick a line, polyline or text to make horizontal: *pick a line or polyline*



Pulldown Menu Location: View > Twist Screen

Keyboard Command: twist2

Prerequisite: None

Twist Screen: Surveyor

This command is another variation of twisting the screen that allows you to input an angle/azimuth that you want to be aligned parallel to the east-west direction of the graphics screen. Entering zero would align due north with respect to real world coordinates to the east or horizontal direction of the graphics screen. The Grid Projection Angle button prompts for a base point and sets the angle to the grid mapping angle. To use this option, the grid projection



must be assigned in the Drawing Setup command.

Pulldown Menu Location: View > Twist Screen

Prerequisite: None

Keyboard Command: twist3

Twist To 3D View

This command orients selected text, symbols and point attributes to face the current viewpoint. Typically, text and points are drawn to face up to plan view. When viewed in 3D from the side, this text can be hard to read. This command makes this text readable for the current view. Before running this command, the 3D view should be set by commands like Viewpoint 3D or Orbit. The entities are oriented to the current view by setting the extrusion

values for the entities.

Prompts

Select points, symbols and text to twist.

Select objects: *pick entities*

Pulldown Menu Location: View > Twist 3D Entities

Keyboard Command: twist3d

Prerequisite: Entities to view

Restore World View

This command is the companion to the Twist To 3D View command. This command resets entities so that they face up in plan view.

Prompts

Select points, symbols and text to restore.

Select objects: *pick entities*

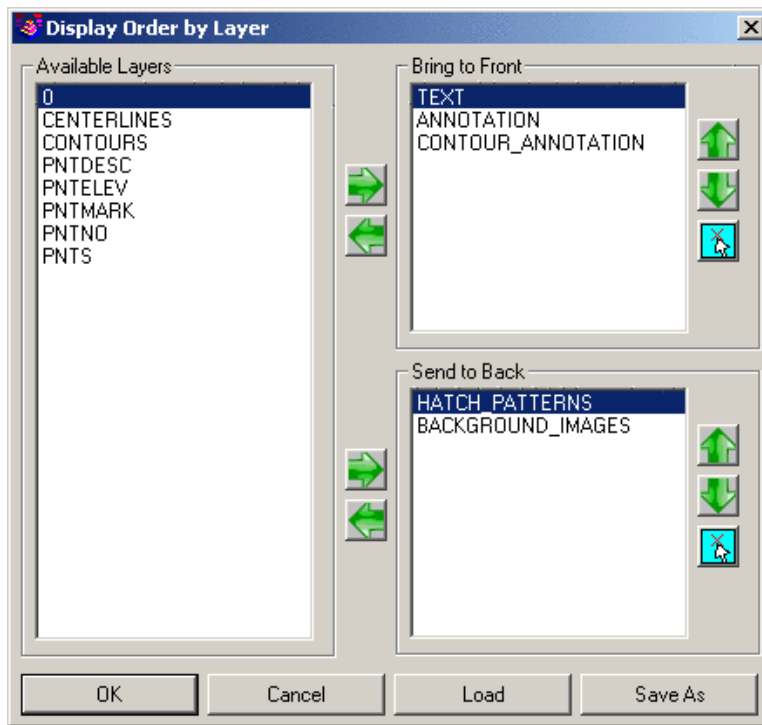
Pulldown Menu Location: View

Keyboard Command: untwist3d

Prerequisite: 3D Entities

Display Order by Layer

The Display Order by Layer command provides a handy method to quickly restore proper visibility to entities in a drawing when they become obscured by other entities (such as hatch patterns or raster image backgrounds). This command provides a combination of the popular *Send to Back* (draworderb) and *Bring to Front* (draworderf) routines (also found under the View>Display Order menu). The display order is applied to the current entities in the drawing at the time the command is run. It does not operate on newly created entities. So you need to run this command each time you want to apply the display order. The following dialog box appears:



Available Layers: The list of layers that are available in the current/active drawing.

Bring to Front: The layers to which the *Bring to Front* routine will be applied.

Send to Back: The layers to which the *Send to Back* routine will be applied.

Control



Action

Moves the selected layer(s) in the *Available Layers* group to either the *Bring to Front* group or the *Send to Back* group.

Moves the selected layer(s) from either the *Bring to Front* group or *Send to Back* group to the *Available Layers* group.

Promotes the selected layer(s) within the *Bring to Front* group or *Send to Back* group.

Demotes the selected layer(s) within the *Bring to Front* group or *Send to Back* group.

Allows the layer of a graphically selected entity to be assigned to either *Bring to Front* group or *Send to Back* group.

Navigation Controls

OK: Sets the display order to the layers in the drawing according to their rank and classification.

Cancel: Does not retain any layer rank or classification changes and does not commit any changes to entities in the drawing.

Load: Allows a previously saved Layer Order (.LO) file to be loaded into memory.

Save As: Saves the current layer rank and classification to a Layer Order (.LO) file.

Note:

- Layers that remain in the *Available Layers* category remain neutral.
- Use standard Windows **click**, **shift+click** and/or **ctrl+click** functionality to select multiple layers at the same time.

Pulldown Menu Location(s): View > Display Order

Keyboard Command: layer_order

Prerequisite: None

Thaw/On All Layers

This command turns on and thaws all layers in the drawing.

Pulldown Menu Location: View

Keyboard Command: loa

Prerequisite: None

Freeze Layer By Name

This command will freeze layers by entering the layer names in a dialog. Multiple layers can be frozen at the same time.

Pulldown Menu Location: View > Layer Control By Name

Keyboard Command: laz

Prerequisite: None

Thaw Layer By Name

This command will thaw layers by entering the layer names in a dialog. Multiple layers can be thawed at the same time.

Pulldown Menu Location: View > Layer Control By Name

Keyboard Command: lat

Prerequisite: Frozen layers

Isolate Layer By Name

This command will isolate layers by entering the layer names in a dialog. Multiple layers can be isolated at the same time. The layers are isolated by freezing all the other layers. Use the Restore Layers command to thaw the layers frozen by Isolate

Pulldown Menu Location: View > Layer Control By Name

Keyboard Command: lai

Prerequisite: None

Turn Off By Pick

This command will turn off layers by picking entities on that layer. The entity selection is done one at a time. As entities are selected, the layers are turned off.

Prompts

Pick entity on layer to turn off: *pick an entity*

Turning off layer

Pick entity on layer to turn off (U-Undo,Enter to end): *press Enter*

Pulldown Menu Location: View

Keyboard Command: layeroffp

Prerequisite: None

Turn Off By Selection

This command will turn off layers by picking entities on that layer. The entity selection is done by selection set for selecting one or more entities.

Prompts

Select entities on layers to turn off.

Select objects: *pick entities*

Pulldown Menu Location: View

Keyboard Command: layeroffs

Prerequisite: None

Turn On Layers

This command turns on the layers turned off by the Turn Off commands.

Pulldown Menu Location: View

Keyboard Command: layeron

Prerequisite: None

Lock Layers

The *Lock Layers* command will lock the layers for the layers of the selected entities.

The *Unlock Layers* command will unlock the layers for the layers of the selected entities.

Pulldown Menu Location: View

Keyboard Command: laylock, layunlock

Prerequisite: None

Show/Hide By Selection

This set of commands to select entities to hide, isolate or make visible.

Hide By Selection

This command prompts to select entities and then makes these entities invisible in the drawing.

Isolate By Selection

This command prompts to select entities and then makes all other entities in the drawing invisible so that only the selected entities are shown.

Show All

This command makes all the entities visible that were hidden by the Hide By Selection or Isolate By Selection commands.

Pulldown Menu Location: View

Prerequisite: None

Keyboard Command: hidess, isolatess, showss

Save/Restore Layer State

The *Save Layer State* command stores to a file all the layers in the drawing and their current status of color, freeze/thaw, on/off, and linetype. The layer state file has a .LAY extension. Later versions of AutoCAD include the ability to save and restore layer states, found in the layer dialog box.

The *Restore Layer State* command sets the drawing layers and their status from the layer information in a layer state file (.LAY file). If a layer from the layer state file does not exist in the drawing, the program will create the layer. Besides the Carlson format, the Land Desktop layer state format, which is also uses a .LAY extension, is supported by this command.

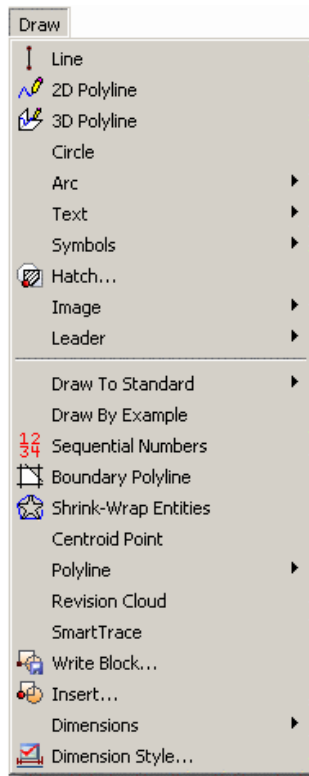
Pulldown Menu Location: View

Keyboard Command: savelay, restlay

Prerequisite: None

Draw Menu

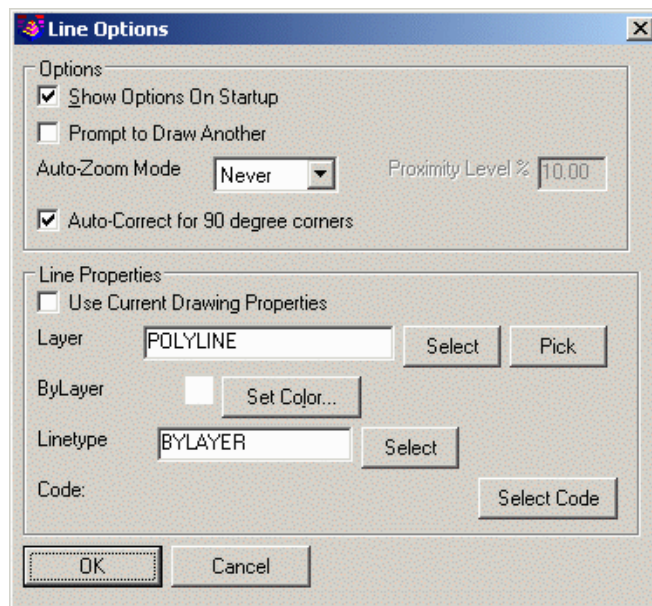
Many of the Draw Menu commands are CAD commands for creating entities in your drawing. Carlson commands that are part of the Draw menu are documented here. Any items not appearing in the Carlson manual are CAD commands that can be referenced in the AutoCAD or IntelliCAD manual.



2D Line

This command allows you to draw a line entity by picking points on the screen or by using the point number and associated coordinates stored in the current coordinate file. This command always draws 2D lines with a zero elevation. The line can also be drawn by entering distances and angles. See the 2D Polyline command for information on using the Distance and Extend input modes.

When the line is drawn using point numbers, the line is linked with the points if the Link Linework with Points option is turned on. This option is set under General Settings in the Configure command in the Settings menu. With links active, changing a point with a command like Move Points automatically updates the line.



Show Options on Startup: When this option is enabled, the dialog box will display automatically upon starting the command. If disabled, you can still get to this dialog by typing O for Options at the command prompt.

Prompt To Draw Another: This option stays in the command after completing the polyline and prompts whether you want to draw another polyline.

Auto-Zoom Mode: This setting provides 3 options for Auto-Zoom: Never, Proximity or Always. The "Never" setting requires you to manually Zoom or Pan to keep the current polyline vertex centered in the drawing screen. The "Proximity" setting will activate the "Proximity Level" setting and will automatically re-center the view only if the current polyline vertex is within a certain distance of the limits of the drawing area. The "Always" option will automatically re-center the view after each new polyline vertex is added.

Auto-Correct For 90 Degree Corners: This option will adjust two line segments to make an exact 90 degree corner when the original lines are nearly 90 degrees.

Use Current Drawing Properties: Select this option if you want the layer, color and linetype of the newly created polyline to match those currently set in the drawing.

Layer: Use this setting to manually assign the layer for the newly created polyline. You can type in the new layer name, use the "Select" button to choose an existing layer from the drawing's layer list or use the "Pick" button to select an entity in the drawing and match its layer.

Set Color: Use this button to manually specify a color for the newly created polyline.

Linetype: Use the "Select" button to manually specify a linetype for the newly created polyline.

Select Code: This option allows you to set the layer, color and linetype of a new polyline by using the properties assigned to a Field to Finish field code. The field code is selected from an existing Field Code table (.FLD) file that has been previously specified in the **Point Defaults** dialog box.

Prompts

Options/⟨Pick point or point numbers⟩: *1-3*

You may enter a single point number or a range of point numbers

Distance/Undo/⟨Pick point or point numbers⟩: *16*

Distance/Extend/Undo/Pick point or point numbers: *35*

Distance/Extend/Undo/Pick point or point numbers: *press Enter*

Pulldown Menu Location: Draw

Keyboard Command: 2DLINE

Prerequisite: None

3D Line

This command draws line entities and is the same as Draw > 2D Line except that this command uses the elevations of the points to make 3D lines.

Pulldown Menu Location: Draw

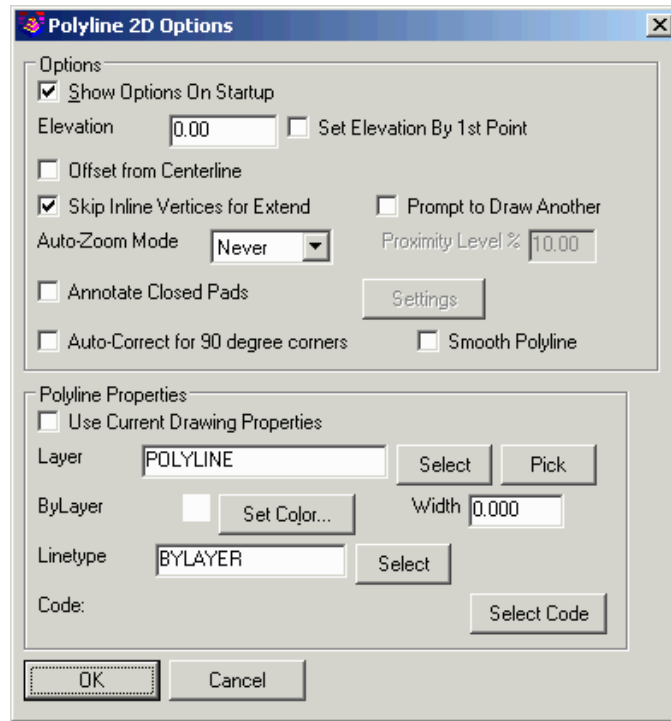
Keyboard Command: 3dline

Prerequisite: None

2D Polyline

This command creates a 2D polyline. A Polyline is a complex CAD entity comprised of one or more line or arc segments. While a 2D polyline elevation isn't necessarily zero, a 2D polyline is flat with all vertices at the same elevation.

This command is available from the **Draw** pulldown menu, from the **Draw** toolbar or at the Command: line (**2DP**) and provides many more options than the standard CAD version of the command. Unless disabled, the **Polyline 2D Options** dialog box will appear after starting Carlson's **2D Polyline** command.



Show Options on Startup: When this option is enabled, the **Polyline 2D Options** dialog box will display automatically upon starting the **2D Polyline** command. If disabled, you can still get to this dialog by typing O for Options at the command prompt.

Elevation: Set the elevation of the polyline to be drawn. The **Set Elevation By 1st Point** option will use the elevation from the first specified point for the whole polyline.

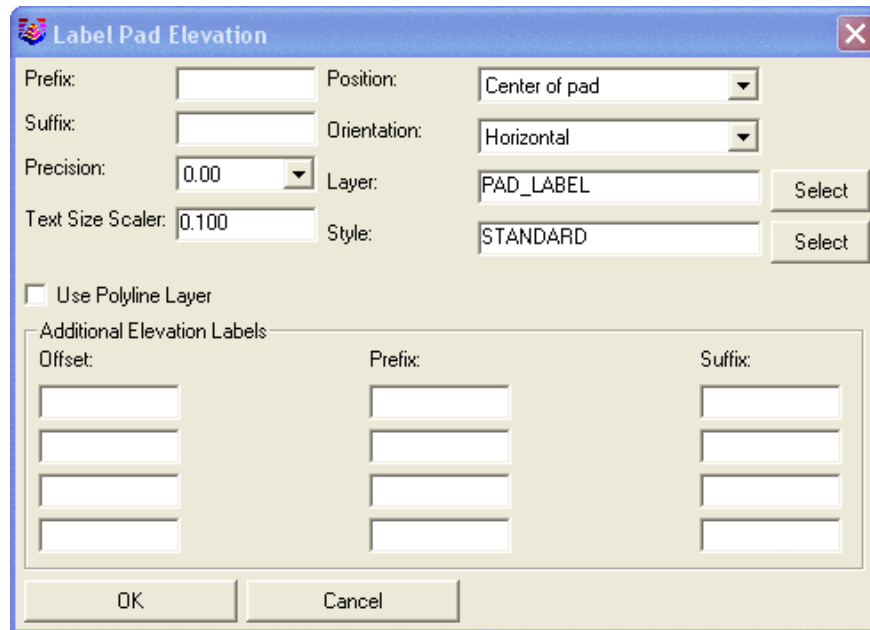
Offset from Centerline: If this option is enabled, an additional option, Offset, is available from the Command: line. Issuing the "Offset" option allows you to draw a new polyline using Station and Offset entry from an existing polyline or existing Centerline (.CL) file.

Skip Inline Vertices for Extend: This setting applies to the "Extend" option with the Total Distance Sub-Menu option. If enabled, an existing vertex will dissolve when lengthening a 2D Polyline segment.

Prompt To Draw Another: This option stays in the command after completing the polyline and prompts whether you want to draw another polyline.

Auto-Zoom Mode: This setting provides 3 options for Auto-Zoom: Never, Proximity or Always. The "Never" setting requires you to manually Zoom or Pan to keep the current polyline vertex centered in the drawing screen. The "Proximity" setting will activate the "Proximity Level" setting and will automatically re-center the view only if the current polyline vertex is within a certain distance of the limits of the drawing area. The "Always" option will automatically re-center the view after each new polyline vertex is added.

Annotate Closed Pads: Enabling this option will activate the "Settings" button. The "Settings" button displays the **Label Pad Elevations** dialog box where you can specify label settings for the pad and other vertical offset elevations. For instance, you can label both the Finished Floor Elevation and the SubGrade elevation of a building pad at the same time using this command. See additional information on the **Label Pad Elevation** command.



In the "Polyline Properties" section of the dialog box you have several alternatives for specifying the layer, color and linetype of the newly created polyline.

Auto-Correct For 90 Degree Corners: This option will adjust two line segments to make an exact 90 degree corner when the original lines are nearly 90 degrees.

Smooth Polyline: This option applies for drawing smooth polylines such as a path for a stream. The Bezier smoothing method is used which passes through all the points and smooths only between the points.

Use Current Drawing Properties: Select this option if you want the layer, color and linetype of the newly created polyline to match those currently set in the drawing.

Layer: Use this setting to manually assign the layer for the newly created polyline. You can type in the new layer name, use the "Select" button to choose an existing layer from the drawing's layer list or use the "Pick" button to select an entity in the drawing and match its layer.

Set Color: Use this button to manually specify a color for the newly created polyline.

Width: Specify the width of the newly created polyline.

Linetype: Use the "Select" button to manually specify a linetype for the newly created polyline.

Select Code: This option allows you to set the layer, color and linetype of a new polyline by using the properties assigned to a Field to Finish field code. The field code is selected from an existing Field Code table (.FLD) file that has been previously specified in the **Point Defaults** dialog box.

Prompts

Command:*2dp*

[Continue/Extend/Follow/Offset/OPTIONS/<Pick point or point numbers>]: *screen pick a point*

[Arc/Close/Distance/Follow/Offset/Undo/<Pick point or point numbers>]: *screen pick a point*

Segment length: 202.55, Total length: 202.55

[Arc/Close/Distance/Extend/Follow/Line/Offset/Undo/<Pick point or point numbers>]: *screen pick a point*

Segment length: 179.73, Total length: 382.28

[Arc/Close/Distance/Extend/Follow/Line/Offset/Undo/<Pick point or point numbers>]: *screen pick a point*

Segment length: 127.45, Total length: 509.73

[Arc/Close/Distance/Extend/Follow/Line/Offset/Undo/<Pick point or point numbers>]: *press Enter*

Command:

Options and SubMenu Options

Once all settings have been specified and the "OK" button is picked, the options shown below are available from the Command: line. To issue any of these options, simply type in the capitalized portion of the Option at the Command: line and press Enter. The default option is always shown between angle brackets < *Default* >.

Continue: This option allows you to select an existing polyline to which you'd like to add more line or arc segments. When prompted to "Select a polyline to continue or extend:", you may pick anywhere on the existing polyline and the new segment will begin at the ending vertex nearest your cursor. New line or arc segments can be added by screen-picking or using the options at the Command: line. Once finished adding segments, they are automatically joined to the original polyline.

Extend: This option gives you many ways to lengthen or shorten an existing polyline using the abbreviated SubMenu options shown below. Some of these options create additional segments at the end of the existing polyline and some allow you to change the length of the ending segment of the polyline. When prompted to "Select a polyline to continue or extend:", you may pick anywhere on the existing polyline and the "Extend" will occur at the ending vertex nearest your cursor. Once finished Extending, the new segments are automatically joined to the original polyline.

[**I / R / L / S / T / A / B / E / U / X / Help / <Enter or Pick Distance>**]

I - Input mode - This option toggles the distance input between feet & inches (will prompt first for feet, then prompt again for inches) and decimal feet.

R - Right rotate - From the ending vertex, turns the pointer 90-degrees to the right and then prompts for a distance.

L - Left rotate - From the ending vertex, turns the pointer 90-degrees to the left and then prompts for a distance.

S - Switch direction - From the ending vertex, turns the pointer 180-degrees and then prompts for a distance.

T - Total distance - Prompts you to "Enter total distance (100.00)" and displays the current length of the segment in parentheses. If a number smaller than the current distance is entered, this option will shorten the existing segment. If a number larger than the current distance is entered, this option will lengthen the existing segment. This option is also affected by the Skip Inline Vertices for Extend setting in the **Polyline 2D Options** dialog box. If "Skip Inline Vertices for Extend" is enabled, then the existing vertex will be dissolved when lengthening a segment. If the setting is not enabled, then the existing vertex will be left intact and an additional segment will be created inline.

A - Angle change - From the ending vertex, prompts you to "Enter Angle (ddd.mmss):" to turn the pointer by a specified angle and then prompts for a distance.

B - Bearing/Azimuth/Turned/Deflection - From the ending vertex, this option allows you to set the pointer direction by specifying an Angle. The Angle format is Qdd.mmss and there are a variety of ways to use the "Q" value to specify the Angle. See here for more.

E - Extend to edge - Extends current segment to another line or entity

U - Undo - Undo last action

X - Quit extend mode - Returns to normal 2D Polyline Draw mode

Help - Displays the descriptions of the Extend options

Enter or Pick Distance - Distance to extend the current segment

Follow: This option allows you to trace all or a portion of an existing polyline. After issuing the "Follow" option, you are prompted to "Select the polyline to Follow:" and then to "Specify the first follow point:". After snapping to a starting point on the polyline, you are asked whether you want to "Interpolate follow vertices elevations?". With this being a 2D Polyline, the answer to this is most likely "NO". You will then be prompted to specify the "Last follow point or follow distance:" where you can snap to another point on the polyline or type in a distance to trace the existing polyline.

Offset: With the "Offset" option, you will first be prompted to select an existing polyline or select an existing Centerline (.CL) file. Next, you will be asked to "Specify starting station:" where you will enter the station number of the first polyline vertex. Then, you will be prompted to "Enter Station" and "Enter Offset" for each vertex of the new polyline. Note: To have this option available, you must place a check next to Offset From Centerline in the **Polyline 2D Options** dialog box.

Pick Point or Point Numbers: This is the default prompt for the command. From here you can set a new polyline vertex by screen picking, entering coordinates in X,Y format or entering a point number from the associated Coordinate (.CRD) file.

Arc/Line: New polyline segments can be either an Arc or a Line segment. If the last polyline segment drawn was a LINE, then the "Arc" option will be shown as an available option; however, if the last polyline segment drawn was an ARC, then the "Line" option will be available.

When in the "Arc" mode, there are many additional SubMenu options available to you for creating an arc segment within the new polyline. The options are generated directly from the standard CAD version of the PLINE command and include Radius Point, Radius Length, Arc Length, Chord and Second Point (Point on Curve).

Close: This option will create a new Line or Arc segment back to the starting vertex of the polyline and results in a closed polyline.

Distance: This option allows you to first enter a distance for the new Line segment and then to specify the direction using one of three methods: Cursor, Line or Angle.

Cursor - This method will draw the polyline segment in the direction of your cursor position.

Line - This method prompts you to select a line or polyline segment to which it will draw a parallel segment.

Angle - This method prompts you for an Angle to determine the direction of your new polyline segment. The Angle format is Qdd.mmss and there are a variety of ways to use the "Q" value to specify the Angle. See here for more.

Undo: Undo the last drawn polyline segment.

Angle Entry Methods

The Angle format is Qdd.mmss where: Q=quadrant/angle, d=degrees, m=minutes and s=seconds.

The Quadrant/Angle can be specified as:

1=NE (NorthEast)

2=SE (SouthEast)

3=SW (SouthWest)

4=NW (NorthWest)

5=AZ (AZimuth)

6=AL (turned Angle-Left)

7=AR (turned Angle-Right)

8=DL (Deflection angle-Left)

9=DR (Deflection angle-Right)

Pulldown Menu Location: Draw

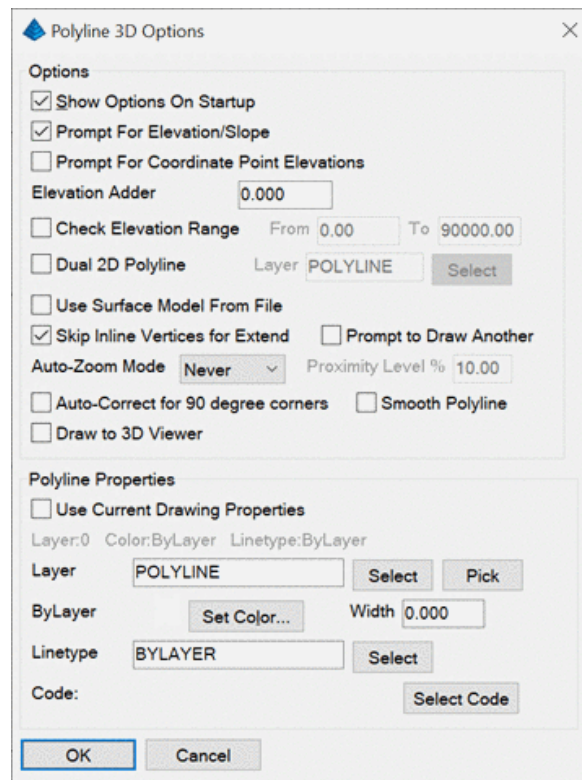
Keyboard Command: 2DP

Prerequisite: None

3D Polyline

This command creates a 3D Polyline. A 3D Polyline is a version of a polyline that can have different elevation at every vertex. 3D Polylines can not have arcs and the program will instead draw a series of short chords to closely follow an arc.

The **3D Polyline** command is available from the **Draw** pulldown menu, from the **Draw** toolbar or at the Command: line (**3DP**). Unless disabled, the **Polyline 3D Options** dialog box will appear after starting Carlson's **3D Polyline** command.



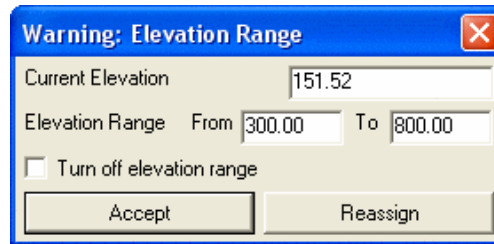
Show Options on Startup: When this option is enabled, the **Polyline 3D Options** dialog box will display automatically upon starting the **3D Polyline** command. If disabled, you can still get to this dialog by typing O for Options at the command prompt.

Prompt for Elevation/Slope: When this option is enabled, the elevation for each new vertex will be displayed as a prompt, giving you an opportunity to override that value by typing in a new elevation. When disabled, the elevation to be assigned to each new vertex is displayed but you are not given a chance to assign a different elevation.

Prompt for Coordinate Point Elevations: This option only applies if you specify a point number from an associated Coordinate (.CRD) file to establish the X,Y,Z values for a new 3D Polyline vertex. When this option is enabled, the elevation for each new vertex will be displayed as a prompt, giving you an opportunity to override that value by typing in a new elevation. When disabled, the elevation to be assigned to each new vertex is displayed but you are not given a chance to assign a different elevation.

Elevation Adder: Use this setting to add a constant elevation value to all default elevation values.

Check Elevation Range: Enabling this option allows you to monitor elevations assigned to 3D Polyline vertices and issue a warning (with options to correct) if the elevation falls outside the specified range. If the proposed elevation of a 3D Polyline vertex falls outside the range specified, the **Warning: Elevation Range** dialog box is displayed. The **Warning: Elevation Range** dialog box allows you to assign a new elevation to the vertex, adjust the acceptable range of elevations or turn OFF monitoring of elevations.



Dual 2D Polyline: This option creates a 2D polyline in addition to the 3D polyline. The 2D polyline is drawn on a separate layer, uses elevation zero and has true arcs instead of the series chord segments used for arcs in 3D polylines.

Use Surface Model From File: Selecting this option allows you to use a Surface Model (.TIN, .GRD, .FLT) file to determine the elevation for each new 3D Polyline vertex.

Skip Inline Vertices for Extend: This setting applies to the "Extend" option with the Total Distance Sub-Menu option. If enabled, an existing vertex will dissolve when lengthening a 3D Polyline segment.

Prompt To Draw Another: This option stays in the command after completing the polyline and prompts whether you want to draw another polyline.

Auto-Zoom Mode: This setting provides 3 options for Auto-Zoom: Never, Proximity or Always. The "Never" setting requires you to manually Zoom or Pan to keep the current polyline vertex centered in the drawing screen. The "Proximity" setting will activate the "Proximity Level" setting and will automatically re-center the view only if the current polyline vertex is within a certain distance of the limits of the drawing area. The "Always" option will automatically re-center the view after each new polyline vertex is added.

In the "Polyline Properties" section of the dialog box you have several alternatives for specifying the layer, color and linetype of the newly created polyline.

Auto-Correct For 90 Degree Corners: This option will adjust two line segments to make an exact 90 degree corner when the original lines are nearly 90 degrees.

Smooth Polyline: This option applies for drawing smooth polylines such as a path for a stream. The Bezier smoothing method is used which passes through all the points and smooths only between the points.

Draw to 3D Viewer: When the 3D Viewer is running during Draw 3D Polyline, then this option will draw the 3D polyline simultaneously in the 3D Viewer and the drawing. This is a way to see the 3D polyline is a 3D scene with other objects while entering the 3D polyline data.

Use Current Drawing Properties: Select this option if you want the layer, color and linetype of the newly created polyline to match those currently set in the drawing.

Layer: Use this setting to manually assign the layer for the newly created polyline. You can type in the new layer name, use the "Select" button to choose an existing layer from the drawing's layer list or use the "Pick" button to select an entity in the drawing and match its layer.

Set Color: Use this button to manually specify a color for the newly created polyline.

Width: Specify the width of the newly created polyline.

Linetype: Use the "Select" button to manually specify a linetype for the newly created polyline.

Select Code: This option allows you to set the layer, color and linetype of a new polyline by using the properties assigned to a Field to Finish field code. The field code is selected from an existing Field Code table (.FLD) file that has been previously specified in the **Point Defaults** dialog box.

Prompts

For A 3D Polyline With A Specified Elevation At Each Vertex:

Command: *3dp*

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: *screen pick a point*

Interpolate/Object/<Elevation> <0.00>: *150.50*

Z: *150.50*

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: *screen pick a point*

Percent/Ratio/Elevation/Degree/Object/Osnap[.]/Next point or elevation<Interpolate>: *155.25*

Z: *155.25*, Hz dist: *324.63*, Slope dist: *324.66*, Slope: *1.5%* Ratio: *68.3:1*

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: *screen pick a point*

Percent/Ratio/Elevation/Degree/Object/Osnap[.]/Next point or elevation<Interpolate>: *148.12*

Z: *148.12*, Hz dist: *272.88*, Slope dist: *272.98*, Slope: *-2.6%* Ratio: *-38.3:1*

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: *press Enter*

Command:

For A 3D Polyline With Interpolated Elevations At One or More Vertices:

Command: *3dp*

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: *screen pick a point*

Interpolate/Object/<Elevation> <0.00>: *91.73*

Z: *91.73*

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: *screen pick a point*

Percent/Ratio/Elevation/Degree/Object/Osnap[.]/Next point or elevation<Interpolate>: *screen pick a point*

This point elevation will be interpolated upon completion.

Percent/Ratio/Elevation/Degree/Object/Osnap[.]/Next point or elevation<Interpolate>: *screen pick a point*

This point elevation will be interpolated upon completion.

Percent/Ratio/Elevation/Degree/Object/Osnap[.]/Next point or elevation<Interpolate>: *screen pick a point*

This point elevation will be interpolated upon completion.

Percent/Ratio/Elevation/Degree/Object/Osnap[.]/Next point or elevation<Interpolate>: *94.44*

Z: *94.44*, Hz dist: *79.39*, Slope dist: *122.88*, Slope: *0.8%* Ratio: *122.4:1*

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: *press Enter*

Command:

Note that the difference between this and the previous example is that, instead of entering an elevation for each vertex, we are screen picking another new vertex. Each time we neglect to enter an elevation we are notified that, "This point elevation will be interpolated upon completion." After we specify "94.44" as the elevation of the last vertex, the slope of the interpolated segments is calculated using the total elevation change and the total length of all interpolated segments. Now, the elevations of all vertices can be determined and set based on the resulting slope.

Options and SubMenu Options

Once all settings have been specified and the "OK" button is picked, the options shown below are available from the

Command: line. To issue any of these options, simply type in the capitalized portion of the Option at the Command: line and press Enter. The default option is always shown between angle brackets *<Default >*.

When starting a new 3D Polyline, the initial set of options assist you in setting the X,Y location of the first vertex:

Continue: This option allows you to select an existing polyline to which you'd like to add more line or arc segments. When prompted to "Select a polyline to continue or extend:", you may pick anywhere on the existing polyline and the new segment will begin at the ending vertex nearest your cursor. New line or arc segments can be added by screen-picking or using the options at the Command: line. Once finished adding segments, they are automatically joined to the original polyline.

Extend: This option gives you many ways to lengthen or shorten an existing polyline using the abbreviated SubMenu options shown below. Some of these options create additional segments at the end of the existing polyline and some allow you to change the length of the ending segment of the polyline. When prompted to "Select a polyline to continue or extend:", you may pick anywhere on the existing polyline and the "Extend" will occur at the ending vertex nearest your cursor. Once finished Extending, the new segments are automatically joined to the original polyline.

[**I / R / L / S / T / A / B / E / U / X / Help / <Enter or Pick Distance>**]

I - Input mode - This option toggles the distance input between feet & inches (will prompt first for feet, then prompt again for inches) and decimal feet.

R - Right rotate - From the ending vertex, turns the pointer 90-degrees to the right and then prompts for a distance.

L - Left rotate - From the ending vertex, turns the pointer 90-degrees to the left and then prompts for a distance.

S - Switch direction - From the ending vertex, turns the pointer 180-degrees and then prompts for a distance.

T - Total distance - Prompts you to "Enter total distance (100.00)" and displays the current length of the segment in parentheses. If a number smaller than the current distance is entered, this option will shorten the existing segment. If a number larger than the current distance is entered, this option will lengthen the existing segment. This option is also affected by the Skip Inline Vertices for Extend setting in the **Polyline 3D Options** dialog box. If "Skip Inline Vertices for Extend" is enabled, then the existing vertex will be dissolved when lengthening a segment. If the setting is not enabled, then the existing vertex will be left intact and an additional segment will be created inline.

A - Angle change - From the ending vertex, prompts you to "Enter Angle (ddd.mmss):" to turn the pointer by a specified angle and then prompts for a distance.

B - Bearing/Azimuth/Turned/Deflection - From the ending vertex, this option allows you to set the pointer direction by specifying an Angle. The Angle format is Qdd.mmss and there are a variety of ways to use the "Q" value to specify the Angle. See here for more.

E - Extend to edge - Extends current segment to another line or entity

U - Undo - Undo last action

X - Quit extend mode - Returns to normal 3D Polyline Draw mode

Help - Displays the descriptions of the Extend options

Enter or Pick Distance - Distance to extend the current segment

Follow: This option allows you to trace all or a portion of an existing polyline. After issuing the "Follow" option, you are prompted to "Select the polyline to Follow:" and then to "Specify the first follow point:". After snapping to a starting point on the polyline, you are asked whether you want to "Interpolate follow vertices elevations?". After answering Yes or No, you will then be prompted to specify the "Last follow point or follow distance:" where you can snap to another point on the polyline or type in a distance to trace the existing polyline.

Options: This will display the **Polyline 3D Options** dialog box.

Pick Point or Point Numbers: This is the default prompt for the command. From here you can set a new polyline vertex by screen picking, entering coordinates in X,Y format or entering a point number from the associated Coordinate (.CRD) file.

After setting its location, the next set of options help you calculate the elevation of the initial vertex:

Interpolate: This option will set the elevation of the vertex by calculating the slope between other vertices of known elevation.

Object: This option allows you to "Select an elevation label or a point on a polyline:" to set the elevation of the vertex. Elevation labels such as "FFE: 124.85" or "Z: 124.85" can be selected.

Adjust: This option allows you to add or subtract an amount from the elevation like for a curb offset.

Elevation: This is the default option and prompts you to type in the elevation for the vertex.

For subsequent 3D Polyline vertices, several options are added to assist you in setting the X,Y location of each new vertex:

Arc/Line: New polyline segments can be either an Arc or a Line segment. If the last polyline segment drawn was a LINE, then the "Arc" option will be shown as an available option; however, if the last polyline segment drawn was an ARC, then the "Line" option will be available.

When in the "Arc" mode, there are many additional SubMenu options available to you for creating an arc segment within the new polyline. The options are generated directly from the standard CAD version of the PLINE command and include Radius Point, Radius Length, Arc Length, Chord and Second Point (Point on Curve).

Close: This option will create a new Line or Arc segment back to the starting vertex of the polyline and results in a closed polyline.

Distance: This option allows you to first enter a distance for the new Line segment and then to specify the direction using one of three methods: Cursor, Line or Angle.

Cursor - This method will draw the polyline segment in the direction of your cursor position.

Line - This method prompts you to select a line or polyline segment to which it will draw a parallel segment.

Angle - This method prompts you for an Angle to determine the direction of your new polyline segment. The Angle format is Qdd.mmss and there are a variety of ways to use the "Q" value to specify the Angle. See here for more.

Undo: Undo the last drawn polyline segment.

After setting subsequent vertices, several more options are added to help you calculate the elevation of each new vertex:

Percent: This option allows you to specify the slope in Percent format (3%) from the previous vertex.

Ratio: This option allows you to specify the slope in Ratio format (for 3:1, enter 3) from the previous vertex.

Degree: This option allows you to specify the slope angle in decimal degree format (dd.dddd) from the previous vertex.

Osnap[.]: Using the [.] will toggle Running OSNAP settings ON or OFF.

Angle Entry Methods

The Angle format is Qdd.mmss where: Q=quadrant/angle, d=degrees, m=minutes and s=seconds.

The Quadrant/Angle can be specified as:

1=NE (NorthEast)

2=SE (SouthEast)

3=SW (SouthWest)

4=NW (NorthWest)

5=AZ (AZimuth)

6=AL (turned Angle-Left)

7=AR (turned Angle-Right)

8=DL (Deflection angle-Left)

9=DR (Deflection angle-Right)

Pulldown Menu Location: Draw

Keyboard Command: 3DP

Prerequisite: None

3 Point

This command draws an arc between three points. The first point is the PC, the second is a point on the arc and the third is the PT. The points can either be picked on-screen or specified by point number.

Prompts

Pick PC point or point numbers: *101* (For point number 101.)

Pick Second point or point number: *102*

Pick PT point or point number: *103*

Pulldown Menu Location: Draw > Arc

Keyboard Command: 3PA

Prerequisite: None

PC, PT, Radius Point

This command draws an arc between the PC point, radius point and PT point. The points can either be picked on-screen or specified by point number. Given these points, the arc can be drawn clockwise or counterclockwise. The program shows one direction and asks if it is correct. If you need the arc to go the other direction, enter No.

Prompts

Pick PC point or point number: *101*

Pick Radius point or point number: *102*

Pick PT point or point number: *103*

Is the direction of this arc correct ? No/<Yes>: *N*

Pulldown Menu Location: Draw > Arc

Keyboard Command: pca

Prerequisite: None

PC, PT, Radius Length

This command draws an arc that is defined by the specified PC point, PT point and radius length. The points can either be picked on-screen or specified by point number. Given these points, the arc can be drawn clockwise or counterclockwise. The program shows one direction and asks if it is correct. If you need the arc to go the other direction, enter No.

Prompts

Pick PC point or point number: *pick a point*

Radius length: *300*

Pick PT point or point number: *pick a point*

Is the direction of this arc correct [<Yes>/No]? press Enter for Yes

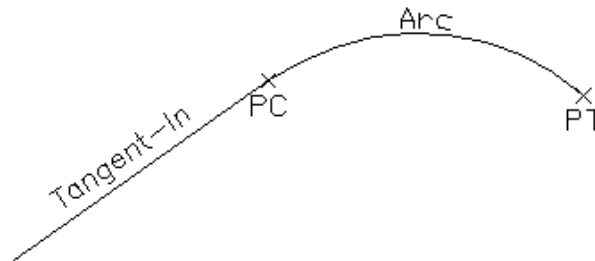
Pulldown Menu Location: Draw > Arc

Keyboard Command: pcptr

Prerequisite: None

PC, PT, Tangent

This command fits a curve between beginning and end points (PC, PT) given a tangent-in. The tangent-in is specified by selecting a line entity. The PC and PT points are screen picked.



Prompts

Pick tangent-in: *pick a line entity*

Pick point PC: *pick a point*

Pick point PT: *pick a point*

Pulldown Menu Location: Draw > Arc

Keyboard Command: tangpcpt

Prerequisite: Tangent line

PC, Radius, Chord

This command draws an arc, given the PC point, radius length, chord length and chord bearing. The PC point can either be picked on-screen or specified by point number. Given these points, the arc can be drawn clockwise or counter-clockwise. The program shows one direction and asks if it is correct. If you need the arc to go the other direction, enter No.

Prompts

Radius of Arc <-40.00>: 500

PC Start Point ?

Pick point or point number: *pick a point*

Chord bearing or chord endpoint (<Bearing>/Point)? Press Enter

Enter Bearing (Qdd.mmss) <90.0000>: 145.1041 (for NE 45d10'41")

Chord Length <200.46>: 200

Is this arc in the correct direction (<Yes>/No)? Press Enter

Pulldown Menu Location: Draw > Arc

Keyboard Command: srcb

Prerequisite: None

PC, Radius, Arc Length

This command draws an arc given the PC point, radius length, and arc length. The PC point can either be picked on-screen or specified by point number. Given these points, the arc can be drawn clockwise or counterclockwise.

The program shows one direction and asks if it is correct. If you need the arc to go the other direction, enter No.

Prompts

Pick PC Point or point number: *pick a point*

Pick Radius point or point number: *pick a point*

Arc length <5.00>: *150*

Is this arc in the correct direction (<Yes>/No)? *press Enter*

Pulldown Menu Location: Draw > Arc

Keyboard Command: pra

Prerequisite: None

2 Tangents, Radius

This command fits a curve between two tangent lines by entering a known radius. It prompts for the radius and then prompts to pick points on the two tangent lines.

Prompts

Radius of Arc <300.000>: *press Enter*

[nea] Pick Point on 1st Tangent Line: *pick a point*

[nea] Pick Point on 2nd Tangent Line: *pick a point*

Pulldown Menu Location: Draw > Arc

Keyboard Command: 2tanlin

Prerequisite: Tangent lines should be drawn before execution

2 Tangents, Arc Length

This command fits a curve between two tangent lines and a known arc length. It prompts for the arc length then pick the P.I. (intersection of tangent lines) and points on the two tangent lines.

Prompts

Arc Length <100.00>: *press Enter or enter distance*

[int on] Pick P.I. of curve: *pick intersection of tangent lines*

[nea on] Pick pnt on 1st Tangent Line: *pick a point*

[nea on] Pick pnt on 2nd Tangent Line: *pick a point*

Pulldown Menu Location: Draw > Arc

Keyboard Command: 2tanlal

Prerequisite: Tangent lines should be drawn before execution

2 Tangents, Chord Length

This command fits a curve between two tangent lines and a known chord length. It prompts for the chord length, the P.I. and points on the two tangent lines.

Prompts

Chord Length <100.00>: *press Enter*
[int on] **Pick P.I. of curve:** *pick a point*
[nea on] **Pick Point on 1st Tangent Line:** *pick a point*
[nea on] **Pick Point on 2nd Tangent Line:** *pick a point*

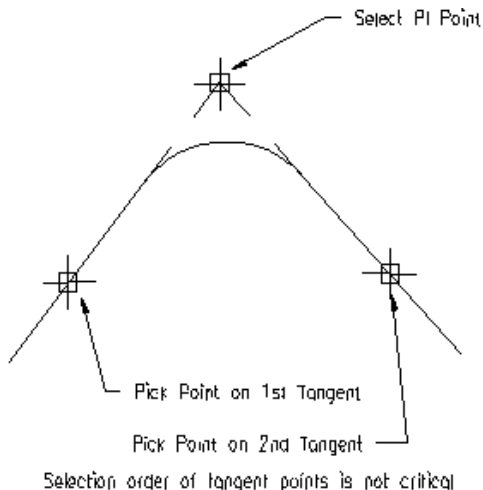
Pulldown Menu Location: Draw > Arc

Keyboard Command: 2tancl

Prerequisite: Tangent lines should be drawn before execution

2 Tangents, Mid-Ordinate

This command fits a curve between two tangent lines and a known middle ordinate. It prompts for the middle ordinate length, the Point of Intersection and points on the two tangent lines.



Prompts

Middle Ordinate <50.00>: *press Enter*
[int on] **Pick P.I. of curve:** *pick a point*
[nea on] **Pick Point on 1st Tangent Line:** *pick a point*
[nea on] **Pick Point on 2nd Tangent Line:** *pick a point*

Pulldown Menu Location: Draw > Arc

Keyboard Command: 2tanlmo

Prerequisite: Tangent lines should be drawn before execution

2 Tangents, External

This command fits a curve between two tangent lines and a known external secant distance. It prompts for the P.I. and points on the two tangent lines then the external distance.

Prompts

[int on] **Pick P.I. of curve:** *pick a point*
[nea on] **Pick Point on 1st Tangent Line:** *pick a point*
[nea on] **Pick Point on 2nd Tangent Line:** *pick a point*
External Distance <50.00>: *press Enter*

Pulldown Menu Location: Draw > Arc

Keyboard Command: 2tanlex

Prerequisite: Tangent lines should be drawn before execution

2 Tangents, Tangent Length

This command fits a curve between two tangent lines and a known curve tangent length. It prompts for the tangent length, P.I. and points on the two tangent lines.

Prompts

Tangent Length <50.00>: *press Enter*

[int on] **Pick P.I. of curve:** *pick a point*

[nea on] **Pick Point on 1st Tangent Line:** *pick a point*

[nea on] **Pick Point on 2nd Tangent Line:** *pick a point*

Pulldown Menu Location: Draw > Arc

Keyboard Command: 2tanltl

Prerequisite: Tangent lines should be drawn before execution

2 Tangents, Degree of Curve

This command fits a curve between two tangent lines by entering a known degree of curve. It prompts for the degree of curve and then prompts to pick points on the two tangent lines.

Prompts

Degree of Curve (ddd.mmss) <5.0000>: *press Enter*

Define by [C]hord or [A]rc length <A>: *press Enter*

[nea on] **Pick Point on 1st Tangent Line:** *pick a point*

[nea on] **Pick Point on 2nd Tangent Line:** *pick a point*

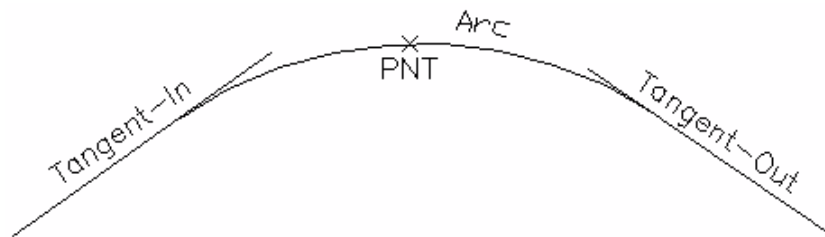
Pulldown Menu Location: Draw > Arc

Keyboard Command: 2tanldo

Prerequisite: Tangent lines should be drawn before execution

2 Tangents, Through Point

This command creates an arc by tangents in/out plus a pass through point on the arc.



Prompts

Pick tangent-in: *pick a line entity*
Pick tangent-out: *pick another line entity*
Pick point on the arc: *pick a point*

Pulldown Menu Location: Draw > Arc
Keyboard Command: 2tanpt
Prerequisite: 2 tangent lines

Tangent, PC, Radius, Arc Length

This command draws a curve from a perpendicular tangent line with a known radius and arc length. It prompts for the radius, the arc length and then to pick the P.C. start point of the curve (endpoint of previously drawn tangent line) and a point along the tangent line.

Prompts

Precede radius with - sign for curve to the left.

Radius of Arc <15.00>: 55

Arc Length <25.00>: 30

PC Start Point ?

Pick point/<point Number>: 14

PtNo. North(y) East(x) Elev(z) Desc

14 4869.06 4390.3 10.00

[nea on] **Pick point along perpendicular tangent line:** *pick a point on tangent line*

Radius Point Coordinates: (4355.2 4911.4 0.0)

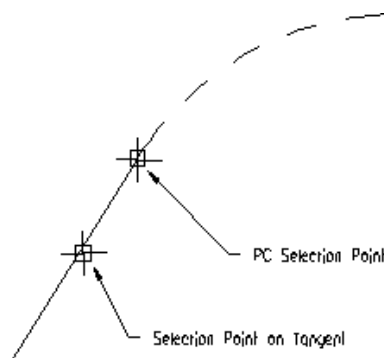
Pulldown Menu Location: Draw > Arc

Keyboard Command: sral

Prerequisite: Tangent lines should be drawn before execution

Tangent, PC, Radius, Tangent Length

This command draws a curve from a perpendicular tangent line with a known radius and tangent length. It prompts for the radius, the tangent length and then to pick the P.C. start point of the curve and a point along the tangent line.



Prompts

Precede radius with - sign for curve to the left.

Radius of Arc <300.0000>: *press Enter*

Tangent Length <236.0000>: *press Enter*

PC Start Point ?

Pick point or point number: *pick a point*

[nea on] Pick point along perpendicular tangent line: *pick a point*

(5270.39 4840.36 0.0)

Radius Point Coordinates: (5251.37 4534.71 0.0)

Pulldown Menu Location: Draw > Arc

Keyboard Command: srlt

Prerequisite: Tangent lines should be drawn before execution

Tang, PC, Radius, Chord Length

This command draws a curve from a perpendicular tangent line with a known radius and chord length. It prompts for the radius, the chord length and then to pick the P.C. start point of the curve and a point along the tangent line.

Prompts

Precede radius with - sign for curve to the left.

Radius of Arc <300.0000>: *press Enter*

Chord Length <25.0000>: *press Enter*

PC Start Point ?

Pick point or point number: *pick a point*

[nea on] Pick point along perpendicular tangent line: *pick a point*

(5142.38 4911.57 0.0)

Radius Point Coordinates: (5221.51 5209.63 0.0)

Pulldown Menu Location: Draw > Arc

Keyboard Command: SRCL

Prerequisite: Tangent lines should be drawn before execution

Tang, PC, Radius, Delta Angle

This command draws a curve from a perpendicular tangent line with a known radius and delta angle. It prompts for the radius, the delta angle and then to pick the P.C. start point of the curve and a point along the tangent line.

Prompts

Precede radius with - sign for curve to the left.

Radius of Arc <300.00>: *press Enter*

Enter Delta Angle <90.00>: *press Enter*

PC Start Point ?

Number/<Pick point>: *pick a point*

[nea on] Pick point along perpendicular tangent line: *pick a point*

Pulldown Menu Location: Draw > Arc

Keyboard Command: srda

Prerequisite: Tangent lines should be drawn before execution

Arc From Last Point

This command draws an arc that is tangent from the last point of the most recent linework or arc entity. The PC point of the arc is automatically set from this last point. This command only prompts for the PT point to create the

arc.

Prompts

Specify end point of arc: *DRAG pick a point*

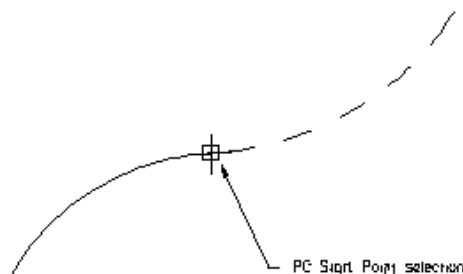
Pulldown Menu Location: Draw > Arc

Keyboard Command: arc

Prerequisite: Linework

Compound or Reverse

This command draws a compound or reverse off an existing curve. It prompts whether the curve is reverse or compound, for the P.C. start point (endpoint of an existing arc) and the known radius. Then the user selects the other known from the choices of tangent length, arc length, chord length or delta angle and enters that value. This command can be confused and malfunction if there is another entity such as a point symbol at the P.C. (If this happens, freeze the PNTMARK layer or temporarily erase the point symbol.)



Reverse curve off an existing curve

Prompts

[end on] Select ARC at PC Start point of the curve: *pick a point*

Type of curve [<Compound>/Reverse]: *press Enter*

Enter the Radius: *300*

Define arc method [Tangent/Chord/Delta/<Length>]: *press Enter*

Enter the arc length: *236*

Pulldown Menu Location: Draw > Arc

Keyboard Command: srcr

Prerequisite: Tangent arc should be drawn before execution

3D Arc

This command draws an arc in 3D using three points for input. The points are the beginning of the arc (PC), a point along the arc, and the end of the arc (PT).

Prompts

Pick PC point or point numbers: *pick a point*

Point on arc.

Pick point or point number: *pick a point*

PT point.

Pick point or point number: *pick a point*

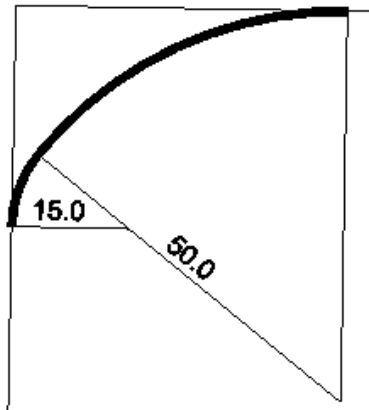
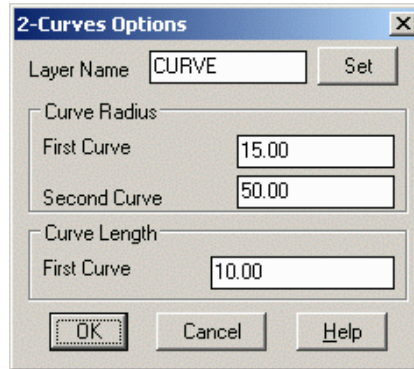
Pulldown Menu Location: Draw > Arc

Keyboard Command: 3darc

Prerequisite: None

2-Radius Curve Series

This command draws a series of two curves with different radii between 2 tangents. In the dialog, set the layer to draw, the radius for each curve and the arc length of the first curve. The program calculates the length of the second curve to fit with the tangents. After the dialog, the program prompts to pick 2 lines to define the tangents in and out.



Prompts

Pick first tangent: *pick a line*

Pick second tangent: *pick a line*

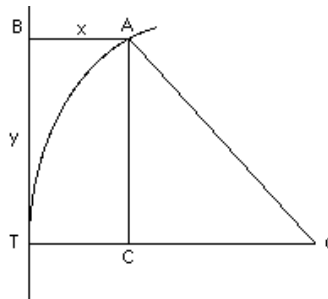
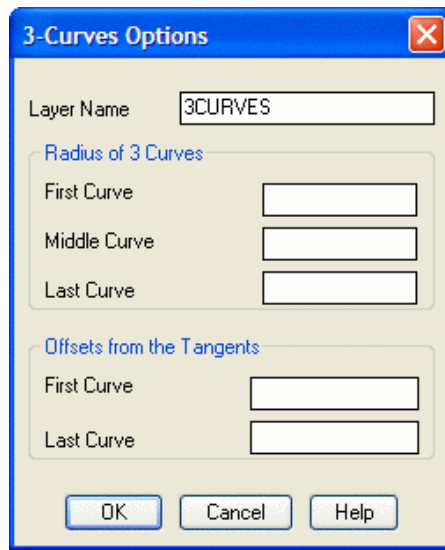
Pulldown Menu Location: Draw > Arc

Keyboard Command: 2curves

Prerequisite: Two tangents

3-Radius Curve Series

This command is used to fit a series of three curves with different radii between 2 tangents. The "Offsets from the Tangents" is the distance perpendicular to the tangent from both ends of the second curve.



Offset from the tangents is the x value

Prompts

Please pick two tangents...

Pick first tangent: *pick a line*

Pick second tangent: *pick a line*

Pulldown Menu Location: Draw > Arc

Keyboard Command: 3curves

Prerequisite: Two tangents

Best Fit Curve

This command draws an arc between two endpoints with a radius that is derived from sampling points. Least-squares is used to find the radius for the closest arc that passes through these points. The Hold End Points option applies additional weight to the end points by the specified weight factor. After specifying the points, the program calculates the best-fit arc and shows the results in the dialog show here. You can toggle each point for whether to include in the calculations. When a point is toggled off for processing, it is not used to calculate the best-fit arc but the residual is still reported. Use the Remove button to remove a point both from calculation and reporting. You can also modify the radius. After picking OK, the arc is drawn in the current layer and there is a report.

Prompts

Starting Point ?

Pick point or point number: 46

Ending point ?

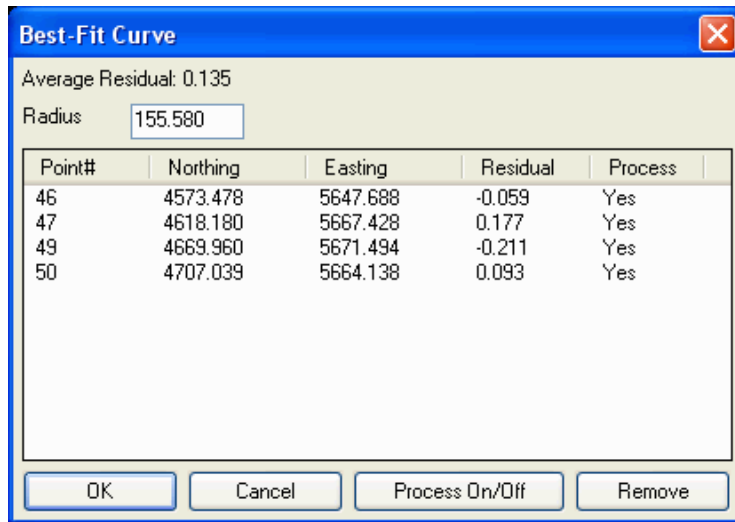
Pick point or point number: 50

Select points from screen, group or by point number [`<Screen>/Group/Number`]? *press Enter*

Select Carlson Software Points.

Select objects: *pick points*

Hold end points [`Yes/No`]? *press Enter*



Best Fit Arc

Coordinate File> C:\sample\PLAT.CRD

Source Coordinates

Point# Northing Easting Residual

46 4573.478 5647.688 -0.059

47 4618.180 5667.428 0.177

49 4669.960 5671.494 -0.211

50 4707.039 5664.138 0.093

Residuals Standard Deviation: 0.148

Average Residual: 0.135

Circle Center: 4657.233,5516.647

Radius: 155.580

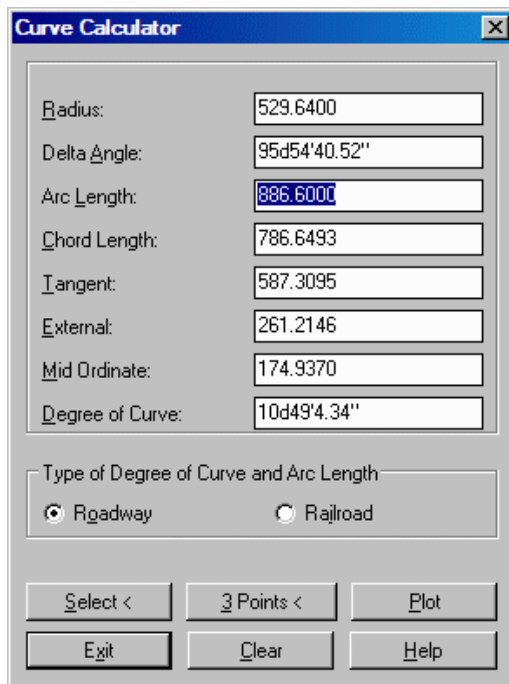
Pulldown Menu Location: Draw > Arc

Keyboard Command: bfitcrv

Prerequisite: Points for sampling should be drawn before execution.

Curve Calc

This Curve Calculator command displays a dialog box with a series of edit boxes that are filled in with the values of a curve. You can input two known values and the program calculates the other values. One of the known values must be the radius or the delta angle. The 3 Points option allows you to simply select three on-screen point locations. All of the fields will immediately be filled in after the picking of the third point. Optionally, you can also input point numbers from a coordinate file.



Roadway or Railroad: Allows you to choose which type of curve you would like information on. Toggling between the two, after data is entered, will reveal different values.

Select: Allows you to select an arc from the drawing. The information for the selected arc is displayed in the dialog box.

3 Points: Allows you to specify three points on the screen to define an arc. The information for this defined arc is displayed in the dialog box.

Plot: Allows you to plot the currently defined arc in the drawing.

Clear: Clears all edit boxes in the dialog.

Prompts

Curve Calculator dialog *Enter at least two values, as described above*

The dialog box first pops up without any data in the fields. The above dialog graphic is a result of entering in the radius and the arc length values of a known curve, then the *Enter* or *Tab* key.

Pulldown Menu Location: Draw > Arc

Keyboard Command: curvecalc

Prerequisite: None

Spiral Curve

This command plots a spiral curve. The user must provide the P.I. (point of intersection), the length of spiral and the radius length of the simple curve. The command will plot a symmetrical spiral or a spiral in or spiral out (choose the S option for the first prompt if you only want to plot a spiral out). If you have an unsymmetrical spiral then plot a spiral in using the T or P option then use the S option to plot the spiral out. The command plots a polyline to represent the spiral as line segments at the resolution specified by the user. You can use the *Calculate Offsets*, *Station Polyline/Centerline* or *Offset Point Entry* commands, found in the Centerline menu, to calculate points and/or stations and offsets from the spiral.

Prompts

Spiral method [TS/ST/<PI>] *press Enter*

PI Point ?

Pick point or point number: *pick intersection of tangent lines*

TS Direction point (tangent in) ?

Pick point or point number: *pick point along tangent in line*

ST Direction point (tangent out) ?

Pick point or point number: *pick point along tangent out line*

Tangent in direction= N 56d24'9" E Azimuth= 56d24'9"

Tangent out direction= S 65d9'1" E Azimuth= 114d50'59"

Overall Delta= 58d26'50"

Point calculating distance resolution <10.0>: *press Enter*

Length of Spiral <350.0>: *press Enter*

Radius of simple curve (precede with - sign if curve to left) <954.93>: *954.93*

Degree of curve: *6d0'0"*

Theta of Spiral= 0.18325951 (radians) 10d30'0" (dd.mmss)

Distance along tangent line from TS to SC= 348.82

Distance offset from tangent line to SC= 21.33

(k) Shift along tangent line of PC= 174.80

(p) Shift offset from tangent line of PC= 5.34

Distance from PI to TS= 712.00

North(Y) of TS= 4583.08 East(X) of TS= 4244.46

North(Y) of SC= 4758.34 East(X) of SC= 4546.82

North(Y) of Offset PC= 4675.36 East(X) of Offset PC= 4393.02

[P]lot spiral or

[I]ntermediate distances for staking (deflection angle calc) <P>: *press Enter*

Point calculating distance resolution <10.0>: *5* Enter the resolution at which you would like the line segments of the representative polyline plotted.

North(Y) of Radius Pt= 3879.96 East(X) of Radius Pt= 4921.44

<press [Enter] for symmetrical spiral out>/[D]elta of simple curve: *press Enter* If you want a spiral in only enter D then input the delta angle of the curve.

Simple Curve Delta= 37d26'50" Length of Arc= 624.12

North(Y) of CS= 4805.10 East(X) of CS= 5158.11

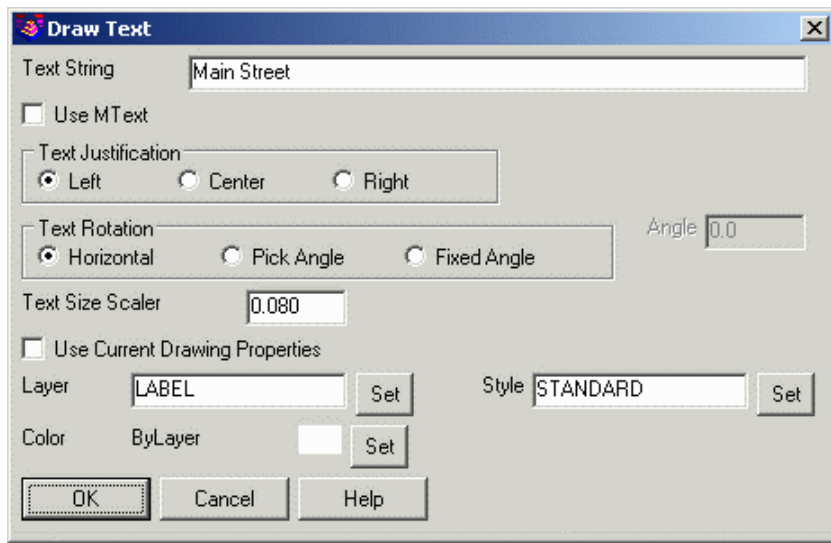
Pulldown Menu Location: Draw > Arc

Keyboard Command: spiral

Prerequisite: For a symmetrical spiral, draw the tangent in and tangent out lines. For spiral in or out only, draw the tangent line in or out.

Text Wizard

This command creates text using a dialog to set the text properties. In the dialog, you set the text string to create, height, justification, layer, style and color. Text Rotation chooses between having the label horizontal to the current view, prompting for the label angle or at a fixed angle specified in the dialog. The Use MText setting controls whether to create the text as a regular Text entity type or as MText. The Use Current Drawing Properties option will use the current layer, style and color from the drawing instead of the dialog.



Prompts

Draw Text dialog

Text location: *pick a point*

Pulldown Menu Location: Draw > Text

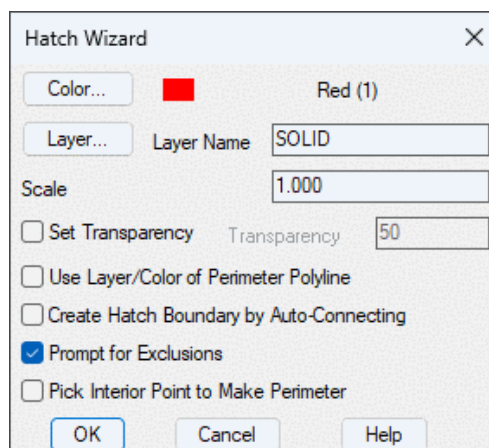
Keyboard Command: drawtext

Prerequisite: None

Hatch Wizard

This command draws a hatch. The dialog has settings for the Color, Layer, Transparency and Scale for the hatch. The **Use Layer/Color of Perimeter Polyline** option sets these properties for the hatch from the polyline used for the hatch perimeter instead of from the dialog. The **Pick Interior Point to Make Perimeter** method defines the hatch area by picking a point in the drawing and having the program find the bounding linework. Otherwise the program prompts to select the inclusion and exclusion perimeter polylines. The **Prompt For Exclusions** controls whether the program prompts to select closed polylines for areas to exclude from the hatch. The **Create Hatch Boundary by Auto-Connecting** method creates a closed area for hatching from unclosed linework by joining the selected linework at their endpoints.

After dialog and selecting the hatch area, the program prompts for the hatch pattern. Then the hatch is created.



Prompts

Pick interior point method:

Pick interior point: *pick a point*

Pick interior point (Enter to end): *press Enter*

Pick polylines method:

Select the Inclusion perimeter polylines.

Select objects: *pick closed polylines to hatch*

Select the Exclusion perimeter polylines or ENTER for none.

Select objects: *press Enter*

Select Text to Exclude from fill or ENTER for none.

Select objects: *press Enter*

For selected text, the program automatically creates an exclusion box around the text to keep the hatch pattern from covering the text.

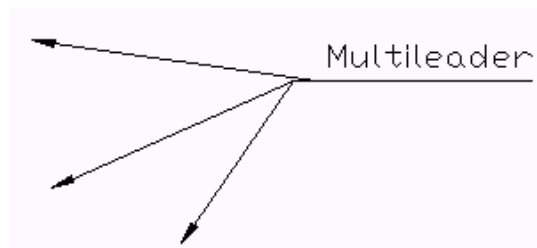
Pulldown Menu Location: Draw

Keyboard Command: hatchwiz

Prerequisite: Perimeter linework

Multileader with Text

This command draws multiple leaders to a label. The style of the leaders is controlled by the current Dimension Style.



Prompts

Beginning point of leader: *Pick point at arrowhead*

End point for leaders: *Pick point at label*

Text: *Multileader*

Text (Enter to end): *press Enter*

Beginning point of leader (Enter to end): *pick a point*

Beginning point of leader (Enter to end): *press Enter*

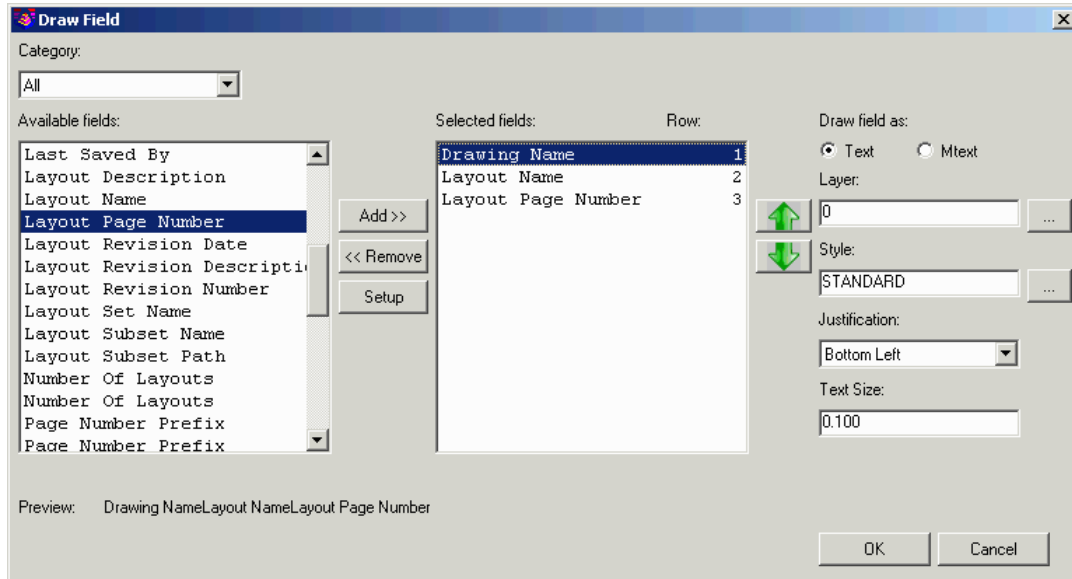
Pulldown Menu Location: Draw > Leader

Keyboard Command: mldr

Prerequisite: None

Field Text

This command adds predefined and custom text objects to your drawing that are updated when a drawing is Regenerated or Saved. Custom fields refresh their data from the Custom Properties defined in the Layout Manager. For example, one of the available fields is Layout Page Number which can be placed as a text entity on a layout. Then if the layout page number changes, the text is updated.

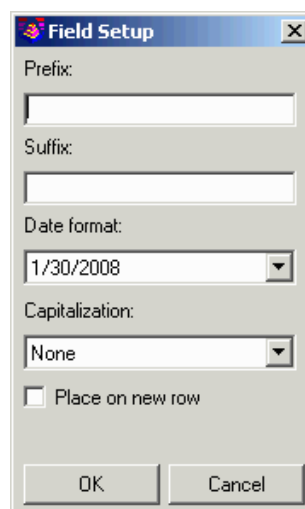


Category: The Category list is used to filter the Available Fields. Selecting All will show all Categories. Some Categories, such as Current Layout Set, Current Layout Subset and Current Layout will only be displayed if the current drawing is attached to a Layout Set file (.set).

Add: Use the Add button to add an Available Field to the Selected Fields list. As you add fields sequentially to the Selected Fields list, the name and row number will be displayed. You can change the row number if you would like the field to appear on a new line. See the Setup button description for more information.

Remove: Use the Remove button to remove a field from the Selected Fields list.

Setup: Use the Setup button to edit the Prefix, Suffix and other contextual properties of the field. For example, Date fields will show a Date Format option, and numeric fields may show a precision format option. Toggle the "Place on new row" checkbox to place the field on a new line. The Field Setup dialog can also be invoked by double clicking a Selected Field list item.



Move Up: Use the Move Up button to move the selected field above the previous field.

Move Down: Use the Move Down button to move the selected field below the next field.

Draw Field As: This setting controls whether the Field is drawn as Text, MText or block attribute objects. The block attribute method can be used to fill out attributes in your title block.

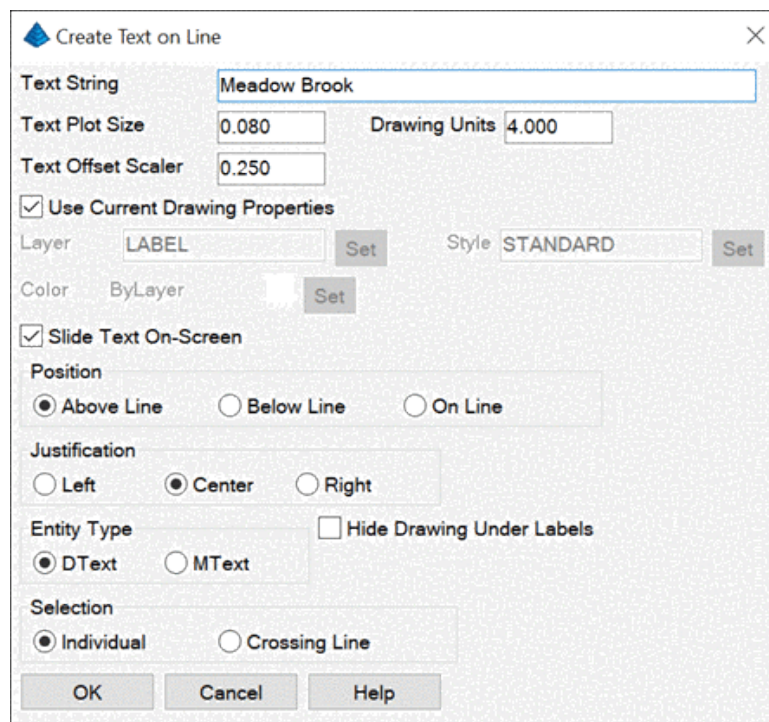
Pulldown Menu Location: Draw > Text

Keyboard Command: drawfield

Prerequisite: None

Text on Line

This command creates text aligned by the selected linework. In the options dialog, you set the text string to create, height, layer, style, color, justification and text entity type. The Text Offset controls the distance between the linework and the text. The Slide option allows you to graphically place the text. The User Current Drawing Properties option will use the current layer, style and color from the drawing instead of the dialog. The Hide Drawing Under Labels creates a Wipeout entity under the text label. The Selection method by Individual prompts for a line or polyline segment to place a single text label at a time. The Crossing Line method prompts to pick two points in the drawing and places the text label at every linework intersection along the line between the two points.



Prompts

Create Text on Line dialog

Pick line or polyline segment: *select linework entity*

Pick point for slide: *pick a point*

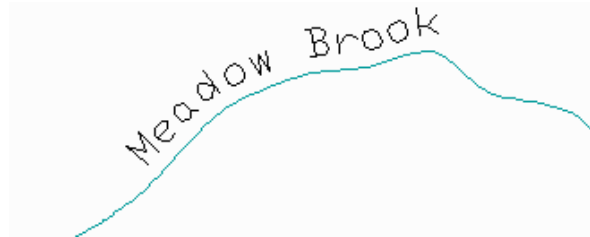
Pulldown Menu Location: Draw > Text

Keyboard Command: textline

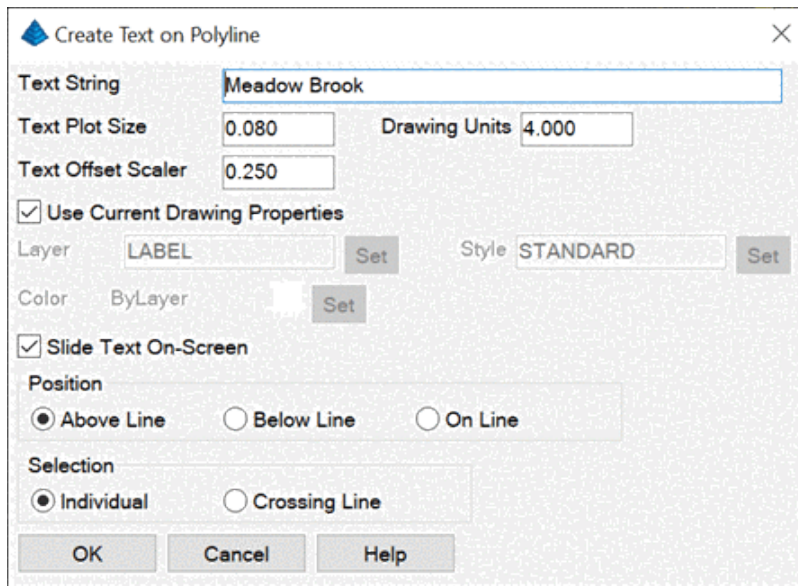
Prerequisite: polyline or line

Text on Polyline

This command creates text with each character aligned by the selected polyline. In the options dialog, you set the text string to create, height and offset. The Text Offset controls the distance between the polyline and the text. The Slide option allows you to graphically place the text. The Hide Drawing Under Labels creates a Wipeout entity under the text label. The Selection method by Individual prompts for a line or polyline segment to place a single text label at a time. The Crossing Line method prompts to pick two points in the drawing and places the text label at every linework intersection along the line between the two points.



Prompts



Create Text on Polyline dialog

Pick polyline: *select polyline entity*

Pick point for slide: *pick a point*

Pulldown Menu Location: Draw > Text

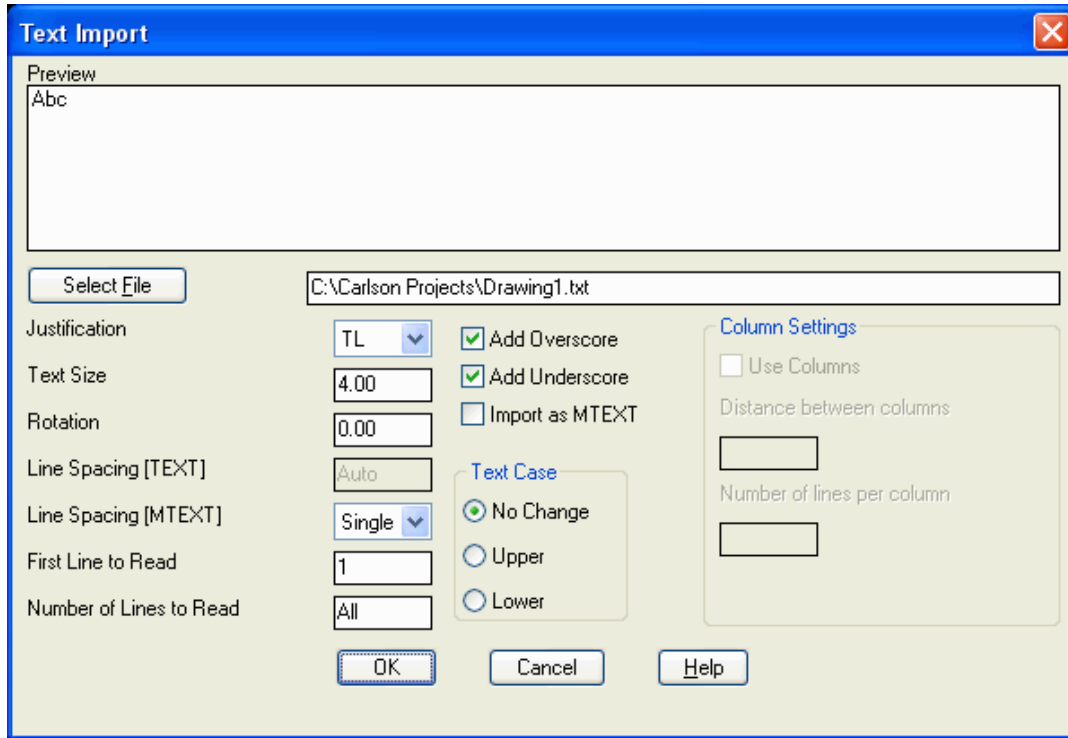
Keyboard Command: textpline

Prerequisite: polyline

Text Import

This command reads a text file and draws the contents as text entities. The file to import and the options are set in the dialog show here.

When using a Microsoft Word .doc or .docx file, the document is drawn as MText using the formatting from the document. There is an option to link the MText to the document so that the MText is updated when the document is changed. This update is done when the drawing is opened or running a REGEN.



Pulldown Menu Location: Draw

Keyboard Command: textin

Prerequisite: text file

Text Export

This command writes a text file from the selected Text and MText entities. The text is sorted to output in top to bottom order based on the entity locations.

Pulldown Menu Location: Draw

Keyboard Command: textout

Prerequisite: text entities

Text Mask

This command hides drawing entities under text by creating a wipeout entity around the text. The Offset is a distance in drawing units to create a buffer around the text.



Prompts

Select text to mask.

Select entities: *pick text*

Offset <0.100>: *press Enter*

Pulldown Menu Location: Draw > Text

Keyboard Command: textmask

Prerequisite: text

Text UnMask

This command removes text masks from the selected text entities.

Prompts

Select text to unmask.

Select entities: *pick text*

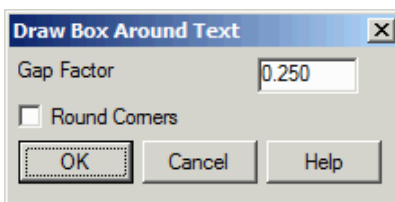
Pulldown Menu Location: Draw > Text

Keyboard Command: textunmask

Prerequisite: masked text

Draw Box Around Text

This command draws a rectangle to enclose the selected text. This rectangle is drawn as a polyline in the current layer. The options dialog has Gap Factor which controls the offset from the text to the polyline. The factor is relative to the text size. The Round Corners option fillets the corners of the box.



Prompts

[Options/<Select text>]: *pick the text to box or type O for the options dialog.*

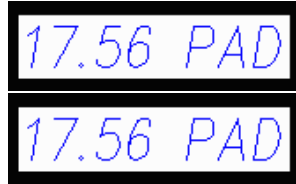
Pulldown Menu Location: Draw > Text

Keyboard Command: txtbox

Prerequisite: Text entity

White Solid Behind Text

This command draws a white solid rectangle to highlight the selected text. The display order for the solid is set behind the text and the solid is drawn on the current layer. This command is only useful when the text itself is not white. At the command prompt, type O for Options to bring up the settings dialog for how to set the layer for the solid.



Prompts

[Options/<Select text>]: *pick the text to highlights*

Pulldown Menu Location: Draw > Text

Keyboard Command: txtwhite

Prerequisite: Text entity

Trim Linework Through Text

This command trims linework that crosses text. After selecting the text to trim with and entering a buffer offset around the text, the program automatically finds any crossing linework and trims.

Prompts

Select text: *pick the text to trim with*

Enter gap <0.5>: *press Enter*

Pulldown Menu Location: Draw > Text

Keyboard Command: txttrim

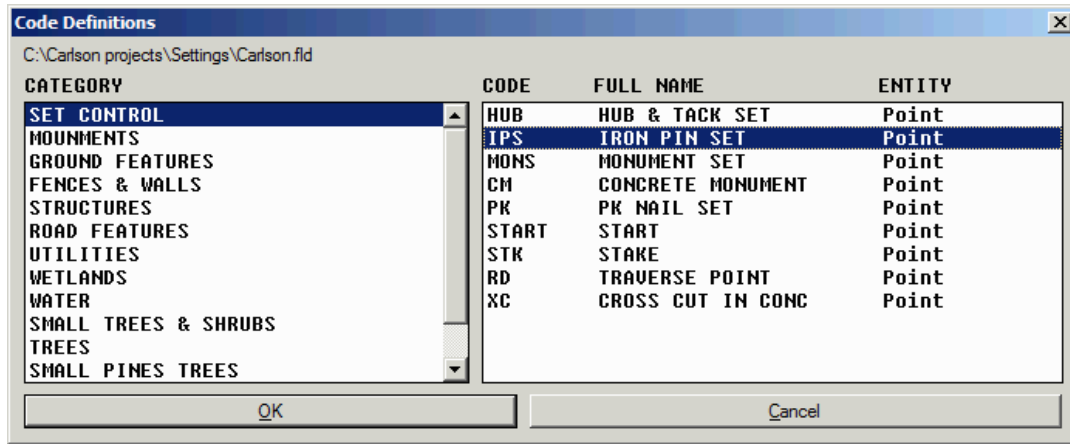
Prerequisite: Text entity and linework

Insert Symbols

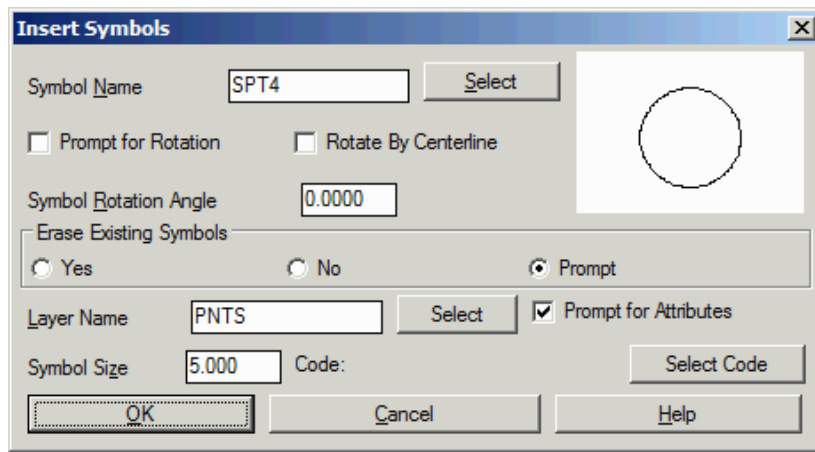
This command inserts symbols from the symbol library into the drawing. The symbol library may be edited using the *Edit Symbol Library* command.

In the Insert Symbols options dialog, choose a symbol by entering the Symbol Name or by picking the Select button which brings up the Select Symbol dialog. The default Symbol Category choices are Points, Trees and Map Symbols. You may select a category by choosing the Symbol Category dropdown list. Within each category, use the scroll bar to view all of the symbols. The Prompt For Rotation option will add a prompt for each symbol rotation. The Rotate By Centerline option will prompt to select linework and then rotate the symbols to make them parallel to the nearest linework. The Symbol Rotation Angle is applied relative to horizontal of the current twist screen or to the nearest linework angle when Rotate By Centerline is active. The Erase Existing Symbols options apply if you specify a symbol location that already has a symbol on it. There are also settings for the symbol layer name and size. The Prompt For Attributes option applies to symbols that have attribute definitions. When active, this option will prompt for the attribute values in a dialog.

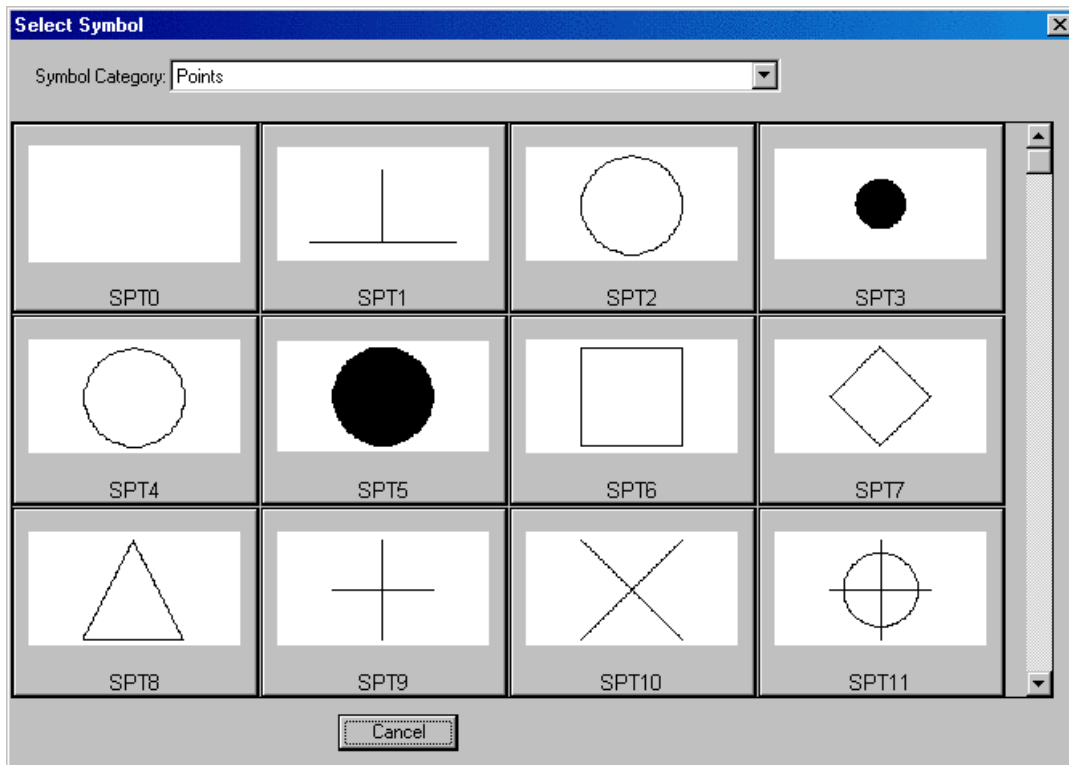
The Select Code option is an alternate way of selecting the symbol by Field-to-Finish code name. The Field-to-Finish code table to use is set with the Points->Point Defaults command. Besides setting the symbol name, the code lookup method also sets the layer. For example, instead of picking a symbol like SPT5 and setting the layer name to "TRAVERSE" for an iron pin symbol, the select code method would set the symbol name and layer by picking the code name/description of "IPS"/"Iron Pin Set" from the code list. So the code method is a way to handle drawing standards.



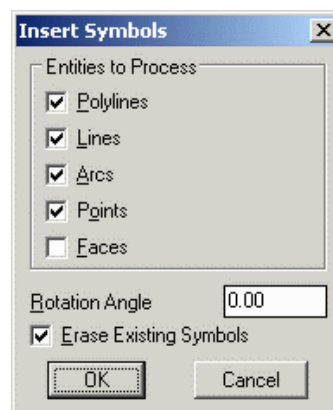
After the options dialog, the program prompts at the Command line for the symbol locations. The locations can be specified by picking points, specifying point numbers in the current coordinate (.CRD) file or by entering the northing and easting. Using the Select entities option, symbols can also be placed on arcs, faces, points, text, lines and polylines. Selecting the Enter coords option allows you to insert the symbol by entering a easting, northing and elevation in x,y,z order.



Appears at start of command



Appears when Select (symbol) is chosen



Select entities dialog box

Prompts

Insert Symbols dialog *Choose parameters and click OK*

Options/Select entities/Enter coords/<Pick point or point numbers>: *pick a point*

Options/Select entities/Enter coords/<Pick point or point numbers>: *5-10* Inserts symbols at points 5-10 from the current coordinate file.

Options/Select entities/Enter coords/<Pick point or point numbers>: *S*

Insert Symbols dialog

Select arcs, faces, points, text, lines and polylines. *select objects*

Options/Select entities/Enter coords/<Pick point or point numbers>: *press Enter to end*

Pulldown Menu Location: Draw > Symbols

Keyboard Command: ptsym

Prerequisite: None

Insert Multi-Point Symbols

This command allows you to locate symbols using multiple insertion points. Up to three insertion points can be defined for an individual symbol. When defining only two insertion points for a particular symbol, the symbol will be scaled and rotated. With three insertion points defined, the symbol is rotated and scaled in both the X and Y directions. The two point insertion definition will aid in the drawing of tree symbols with a specific drip line width. For instance, a surveyor could locate the tree and then locate the drip line, two shots for each tree, and allow the program to size the tree symbol accordingly so that the map will have various tree symbol sizes that reflect the actual field conditions.

The multiple insertion points are defined in the Field to Finish codes. The *Insert Multi-Point Symbols* command reads the Field to Finish code table and finds all of the codes with multi-point symbol definitions. Then you can select from these codes for the symbol to draw. Both the two and three point insertion definitions can aid with the insertion of concretes and buildings symbols during final drawing preparations and design phases of a project.

Here are the various steps to define two point and three point insertion point symbols. First, you must decide on the symbol to use for the desired code, as well as the specific placement points for the symbol. Once a symbol has been chosen, open the desired symbol drawing. To do this, identify the symbol name and then locate the symbol by its drawing name under the SUP sub-directory found under the Carlson installation directory. Next, determine the placement points for the symbol. As shown below, the placement points for the BLD code symbol, which will be explored later in this section, were determined by identifying X and Y values of the desired placement points by using the id command and specifying the end points of the lines.

Next, the symbol insertion points must be defined in the Field to Finish code table (.FLD) file. To do this, open your FLD file by choosing *Draw Field to Finish* under the Survey pulldown. Then select a particular code from the list of codes displayed in the Field to Finish dialog box. Edit it by highlighting the code and picking the Edit button, or define a new code with the Add button. Either choice will display the Edit Field Code Definition dialog. In the Edit Code Definition dialog, choose the desired symbol for the code by pressing the Set Symbol button and selecting the desired symbol. Next, select the Symbol Pts button. This brings up a dialog called Define Symbol Placement Points. Here is where you define the symbol by three points. You do this by entering an X and Y coordinate and a description for the symbol. Enter the X and Y values for each placement point into the appropriate fields. The description fields are used as the prompts when placing the symbol in the drawing. A two insertion point symbol is defined in the same way. An example is the Symbol Pnts definition for the code TREE. The placement points for the Tree code symbol were determined by opening the symbol drawing and finding the X and Y values at the insertion points. The center of the large circle was chosen for Point 1 and the East Quadrant was chosen for point 2. In both cases osnaps were used in picking the points.

Now that we have the codes defined, let's go through the *Insert Multi-Point Symbol* command and see the results. The command starts with a dialog that lists all the codes with Multi-Point Symbols defined. At this point you can select the symbol to draw. The symbol size applies only to using one point to place the symbol. When two or more points are used, the symbol is scaled to fit the points. Let's look at the BLD code three point insertion definition.

Shown below are three points that represent a building pad. We want the building to be exactly the same dimensions defined by the point locations.

The three point PAD and the tree with drip line examples follow. We start by specifying the building pad codes.

Prompts

Insert Multi-Point Symbol Dialog

Choose a symbol to draw. In this example, the Pad symbol is a 3 point multi-symbol.

Specify LTFNT PAD point.

Pick Point or Point Number (Enter to End): 15

Specify LT REAR PAD point.

Pick Point or Point Number (Enter to End): 16

Specify RT REAR PAD point.

Pick Point or Point Number (Enter to End): 17

Insert another BLD symbol [<Yes>/No]? N

Insert Multi-Point Symbol Dialog

Choose a symbol to draw. In this next example, the Tree symbol is a 2 point multi-symbol. Now specify the location of the trunk and the drip line by point number.

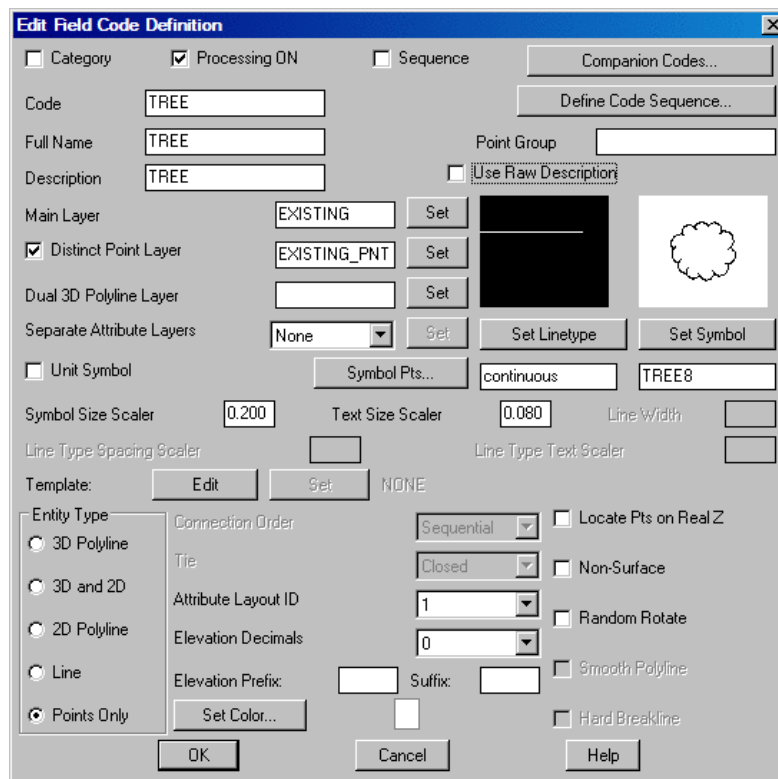
Specify Trunk Location point.

Pick Point or Point Number (Enter to End): 1

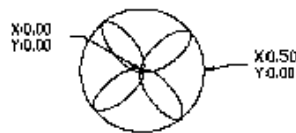
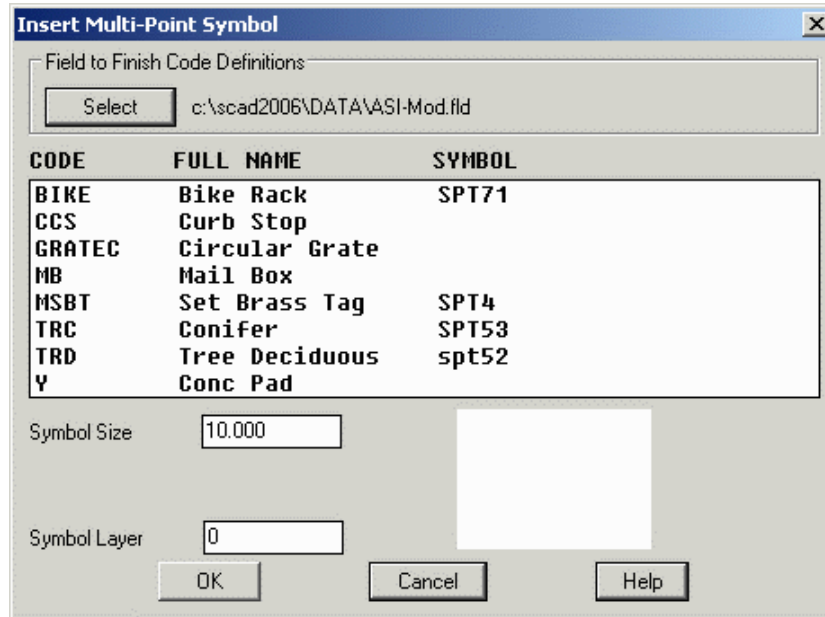
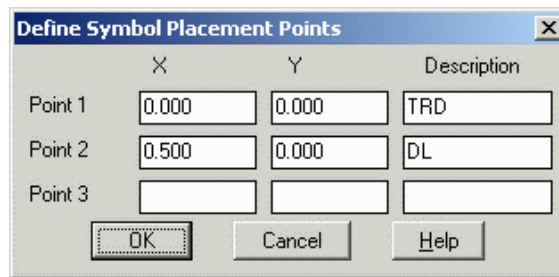
Specify Drip Line Point.

Pick Point or Point Number (Enter to End): 13

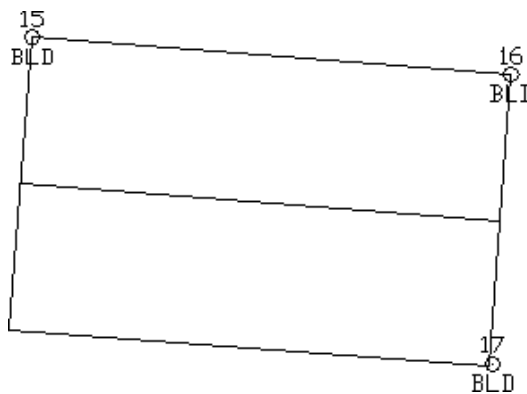
Insert another TREE symbol [<Yes>/No]? N



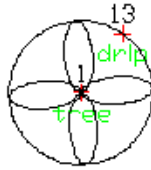
From the Field to Finish routine



Two points symbol placement for TREE



Three points for building PAD



Two point tree with drip line

Pulldown Menu Location: Draw > Symbols

Keyboard Command: multisym

Prerequisite: Field to Finish file (.FLD) with codes defined with Multi-Point Symbols

Draw 3D Model File

This command draws 3D faces for geometry in a 3D model file in .mdl or .obj file format. These graphic files are used in 3D viewing commands like 3D Drive Simulation. The Settings > 3D Model Library manages these graphics files. The .mdl file is also used for Solids in the Underground Mining module.

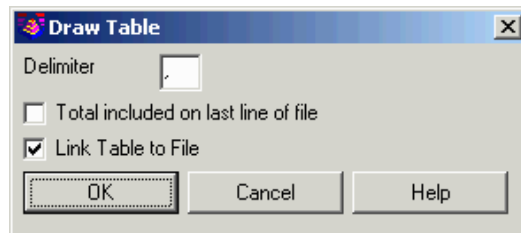
Pulldown Menu Location: Draw > 3D Solids

Keyboard Command: mdl2dwg

Prerequisite: MDL or OBJ file

Draw Table

This command creates a table entity by either reading a data file or by filling in a spreadsheet. For the input by data file, the text file must have rows of data fields separated by a delimiter such as a comma separated file (CSV). In the options dialog, set the Delimiter for the character that separates the values in the text file. Also there is an option for whether to create a totals line at the end of the table. The Link Table option will update the table in the drawing when the text file is updated. This update is done when the drawing is opened or running a REGEN.



For Manual Input method, the data is entered in a spreadsheet that starts out empty. You can manually type in data or copy/paste from the Windows clipboard.

After picking the location to draw the table, there is a dialog for formatting the table. You can control the header names, width, size, alignment and style. There are settings for the table layer, color and title.

Drillhole Log							
DRILLHOLE	NORTH	EAST	SURFACE	STRATA	BOTDEPTH	MOISTURE	PLASTICITY
B-1	2190064.77	6134999.45	818.500	CLAY	5.000	32.000	59.000
B-1	2190064.77	6134999.45	818.500	LIMESTONE	12.000		
B-2	2190068.34	6135073.06	818.400	CLAY	4.400	30.000	57.000
B-2	2190068.34	6135073.06	818.400	LIMESTONE	12.500		
B-3	2190138.64	6135102.68	817.600	CLAY	3.000	34.000	55.000
B-3	2190138.64	6135102.68	817.600	LIMESTONE	11.000		
B-4	2190154.00	6135198.22	818.500	CLAY	2.500	39.000	61.000
B-4	2190154.00	6135198.22	818.500	LIMESTONE	10.300		
B-5	2190283.22	6135182.05	818.100	CLAY	4.250	30.000	58.000
B-5	2190283.22	6135182.05	818.100	LIMESTONE	12.750		
B-6	2189988.08	6135183.40	816.500	CLAY	3.000	29.000	59.000
B-6	2189988.08	6135183.40	816.500	LIMESTONE	12.000		
B-7	2189994.83	6134946.24	817.700	CLAY	5.000	40.000	54.000
B-7	2189994.83	6134946.24	817.700	LIMESTONE	12.500		

Prompts

Select Input Data File

Draw Table options dialog

Pick location for report table: *pick a point*

Table Options dialog

Pulldown Menu Location: Draw

Keyboard Command: csv2tbl

Prerequisite: delimited text file to import

Arrowhead With Line

This command draws a single-segment leader with an arrowhead. The arrow size is specified in drawing units.

Prompts

Arrow size <5.00>: *press Enter*

Pick a starting point: *pick a point*

Pick an ending point: *pick a point*

Pick a starting point (Enter to End): *press Enter*

Pulldown Menu Location: Draw > Leader

Keyboard Command: ldr2

Prerequisite: None

Arrowhead

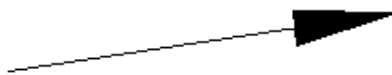
This command draws an arrowhead at the end of the selected line or polyline.

Prompts

Enter the arrow size <5.00>: *press Enter*

Pick a line or pline to add arrow: *pick a line or polyline*

Pick a line or pline to add arrow (Enter to End): *press Enter*



Pulldown Menu Location: Draw

Keyboard Command: arrowhd

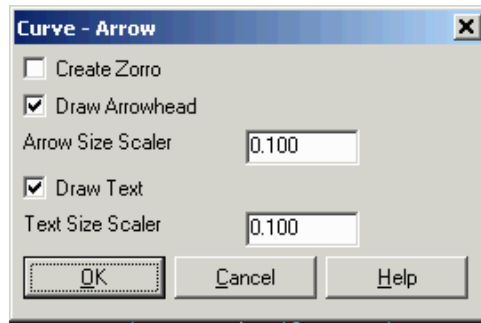
Prerequisite: None

Curve - Arrow

Curve - Arrow can be used to draw a section of contour line or create leader pointer lines. Curve - Arrow draws a Bezier curve through user specified points. After choosing endpoints, each time an intermediate points is picked the curve will be redrawn through all the points. There is an option to draw an arrowhead at the starting point. This

routine also has a Zorro option which creates a Z leader curve. The Draw Text option will make the program prompt after the leader points for a text label to place at the end of the leader.

Prompts



Pick a starting point: *pick a point*

Pick an ending point: *pick a point*

Pick an intermediate point (U to Undo): *pick a point*

Pick an intermediate point (U to Undo): *press Enter*



Examples of Curve - Arrow

Pulldown Menu Location: Draw

Keyboard Command: carrow

Prerequisite: None

Leader With Text

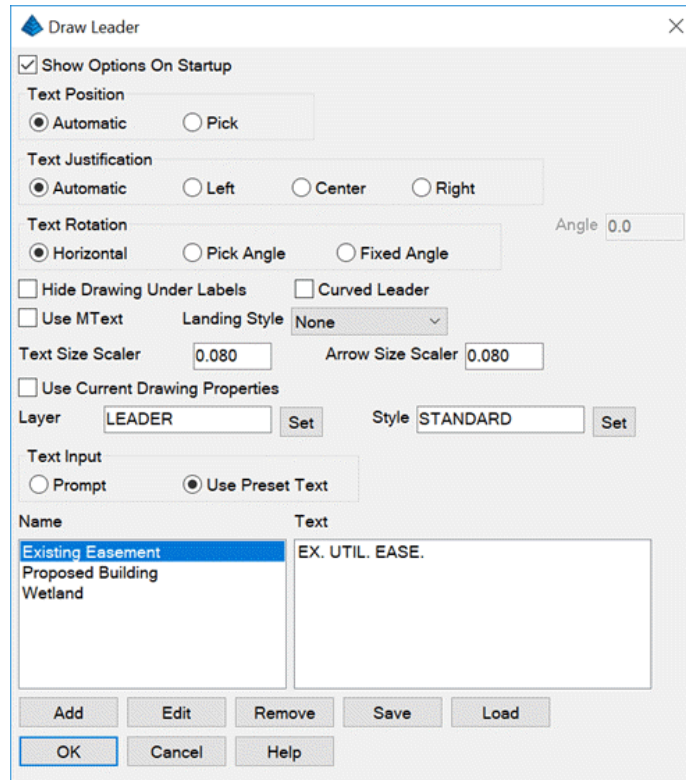
This command will draw a straight or curved leader between two points, with an arrow at one end and optional text at the other. The options dialog is displayed at the start. To skip this dialog, turn off the Show Options On Startup toggle. Then on the command line, entering *O* for Options will show the options dialog. To quickly change the label size, enter *S* for Size at the command line.

In the options dialog, Text Position chooses between automatically placing the label next to the leader end point or picking the label position. For Text Justification, you can set a specific justification or use the Automatic option which uses either Left or Right justification depending on whether the leader end is left or right of the leader start. Text Rotation chooses between having the label horizontal to the current view, prompting for the label angle or at a fixed angle specified in the dialog. The Hide Drawing Under Labels option uses a wipeout to hide drawing entities behind the leader labels. The Curved Leader option curves the leader between the start and end points. Otherwise a straight leader line is created. The Use MText option chooses between creates MText or regular Text entities. The Landing Style of Horizontal Tick draws a short horizontal line segment at the end of the leader. The Landing Style of Underline draws the leader line under the text like an underscore. The Text Size Scaler and Arrow Size Scaler are multiplied by the Horizontal Scale from Drawing Setup to set the drawing size for the label and arrowhead. There

are settings for the layer and style for the label. When the User Current Drawing Properties is on, then the current layer and current style from the drawing will be used.

The Text Input can either be entered with prompts after the leader points at the command line or selected from a prepared list. The prepared list is a way to quickly create leaders for common labels. To prepare the label list, use the Add, Edit and Remove buttons. Use the Save and Load buttons to save the preset labels to a .CALL file.

Prompts

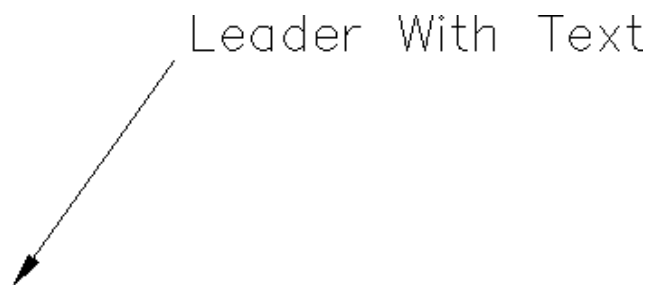


Options/Size/Pick Arrow Location: *pick a point*

Text location: *pick a point*

Text: *Leader With Text*

Text: *press Enter*



Pulldown Menu Location: Draw > Leader

Keyboard Command: LDR

Prerequisite: None

Special Leader

This command draws a curved leader line like the one shown. With this routine you can also choose to enter in multiple lines of text, not just a single line. The arrow size is determined by the Symbol Plot Size setting, found in the *Drawing Setup* command. On the command line, selecting *O* for Options will provide you with more customizing choices to make.

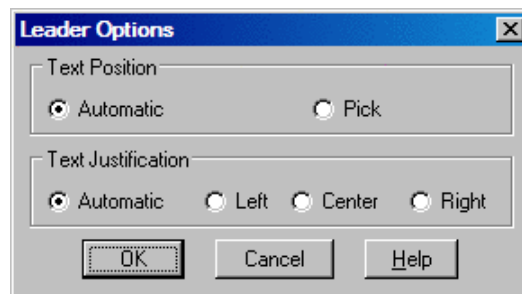
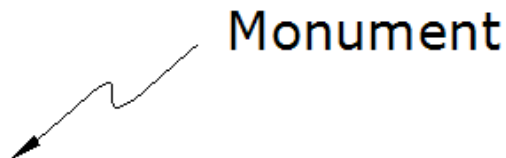
Prompts

Options/Pick Arrow Location: *pick a point* Pick point where leader arrow will start.

Text location: *pick a point*

Text: *Monument*

Text: *press Enter*



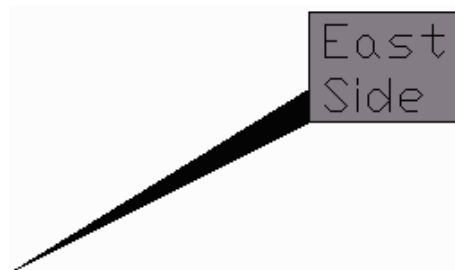
Pulldown Menu Location: Annotate

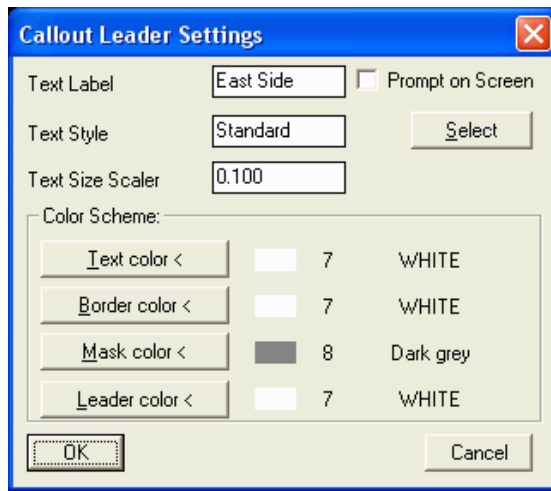
Keyboard Command: sleader

Prerequisite: None

Callout Leader

This command draws a triangle shaped leader and a label inside a box. There is a dialog to enter the label string, style, size and colors. The leader is drawn in the current layer.





Prompts

Callout Leader Settings dialog

Pick callout point: *pick a point for point of leader*

Pick textbox corner: *pick a point for position of label*

Pulldown Menu Location: Draw > Leader

Keyboard Command: callout_ldr

Prerequisite: None

Bold Curve Leader

This command draws a thick curved leader with an arrowhead. This leader is created by picking three points.



Prompts

Starting point: *pick a point*

End of arrowhead: *pick a point*

Pick end point of leader: *pick a point*

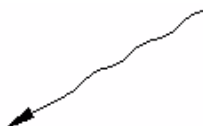
Pulldown Menu Location: Draw > Leader

Keyboard Command: site_leader

Prerequisite: None

Flow Leader

This command draws a wavy leader line with an arrowhead. The size of the arrowhead is set by the symbol size scaler in Drawing Setup.



Prompts

Starting point: *pick a point for arrow end of leader*

Ending point: *pick a point for tail end of leader*

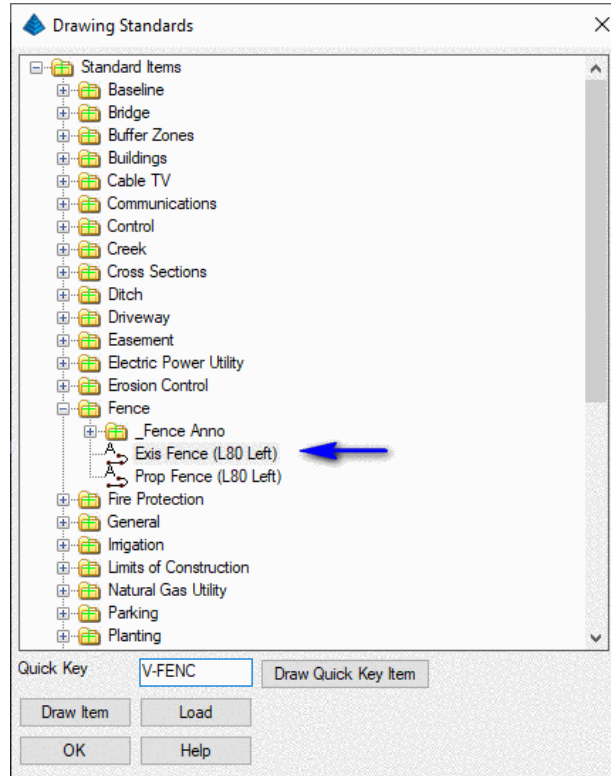
Pulldown Menu Location: Draw > Leader

Keyboard Command: flowline

Prerequisite: None

Draw Standard Item

The **Draw Standard Item** command activates the **Standards Draw** palette:



This palette is essentially the same as the **Standards Draw Manager** palette except that it does not have the Standards management commands available in the right-click short-cut menu.

Draw Quick Key Item: When a valid Standard Item key-in (essentially the layer designation of an item until changed to something more meaningful) has been keyed into the *Quick Key* edit box, you can draw the item without having to navigate for it in the list of Standard Items.

Draw Item: This option will permit you to browse for a Standard Item from the list of items (within their respective Categories, if present) and place the Standard Item into the drawing.

Load: This command will load the content of an XML-structured *.CAD file.

Carlson includes a Standards Database file (.cad) with the installation (**NCS_SurveyCivil.cad** which is a fully populated database based on the **US National CAD Standard**).

The **Draw Standard Item** command permits you to browse through you list of Standard items or draw one based on its "quick key" (essentially its layer name) designation and can be quickly accessed through the toolbar short-cut on the left shown below.





Items in the **Standards Draw** palette are organized into Categories and given commonly referred-to, descriptive names. Defining these Items and other management tasks are done through the Items Standards Manager.


A Standard Item can include any one or a combination of three types of entities:

1. Symbol
2. Linework
3. Annotation (with or without a Leader)

Next to each item in the palette is a graphic icon which indicates the type of entity(ies) defined for that Standard Item.

This icon,  is shown for a Standard Symbol Item.

This icon,  is shown for a Standard Linework Item.

This icon,  is shown for a Standard Annotation Item.

You may also see one of these icons representing a combination of entity types:



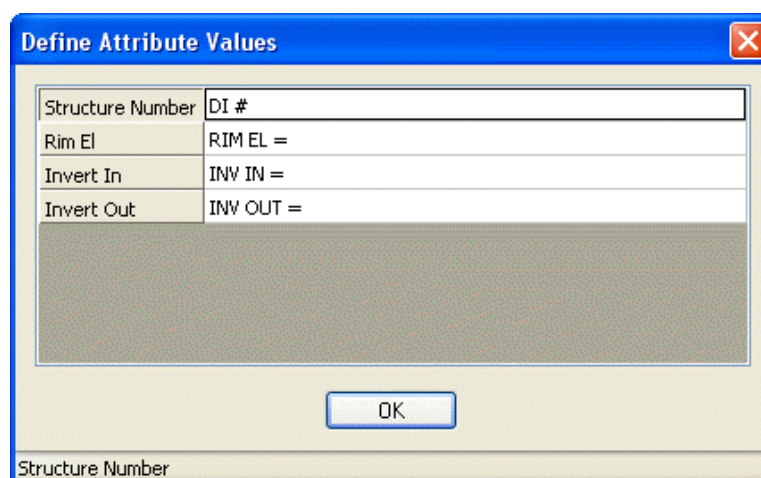
Because many Standard Items will be sized based on the scale of the drawing, it is important to set the "Horizontal Scale" in the **Carlson Drawing Setup** dialog box before drawing Standard Items into your drawing. If Standard Items are drawn and scaled according to the "Horizontal Scale", the "Horizontal Scale" setting is saved with the entity. In other words, changes to the "Horizontal Scale" of the drawing will not affect existing entities. This is true of Symbols, Annotation and Leader components.

To Draw a Standard Item using the Item menu in the palette, navigate through and left-click the Standard Item to be drawn. Assuming the Standard Item has a Symbol, Linework and Annotation component, you will see, generally, the following series of prompts. Note that, depending on the various options that have been set for each Item, the prompts may vary.

Start Point: Using the left-mouse button, select the location for the first Symbol component. This point will also be the first endpoint of the Linework for this Item. If you do not wish to have a Symbol inserted at this point and only wish to draw the associated Linework, hold the **ALT** key when specifying the Start Point to proceed on to the next Endpoint.

Rotation Angle: If you have placed a Symbol, specify its Rotation Angle. If you have set the Symbol Rotation option to "Fixed", you will not be prompted for this Rotation Angle.

If you insert a Symbol having non-Constant attribute values, you will be prompted through a dialog box to define the attribute values:



Halfwidth/Width/CLose/LEngth/Open/Undo/Arc Endpoint of line: Select the next Endpoint of the line segment or specify one of the other Polyline command options before picking the Endpoint. If you do not want to draw a

Linework segment at this time, press **Enter** to skip this step and continue on to place Annotation.

Rotation Angle: Specify the Rotation Angle for the next Symbol. Again, depending on the Symbol Rotation options you have set, you may or may not receive this prompt.

The prompts will continue to alternate between "Rotation Angle" and "Endpoint of Line" until you have reached the end of your Linework. When you have specified your final Endpoint and the Rotation Angle of your final Symbol, right-click to continue on to place Annotation.

Displacement/Identify or <\P>key in alternative text: This is the prompt for your first Annotation entity. The Item's Label is used for default Text content. To override the default Text, simply type in the alternative Text at the Command: Line. If you need a 2nd line of Text, use "\P" to designate the 2nd line of text. For instance, entering "TWO STORY\PWOOD FRAME" would result in the following Text string in the drawing:

```
TWO STORY
WOOD FRAME
```

Also, left-clicking on any other Text entity will update your current Text value to match the Text that was selected. And, holding the **ALT** key while left-clicking on any other Text entity will add the value of that Text entity as a 2nd line of Text to your current Text value.

Once you have finished entering the Text, press **Enter** to finish Text entry. Left-click in the drawing to place the Text.

Rotation angle/Identify or <\P>key in alternative text <0.0000>: Type in a Rotation Angle for the text or left-click to specify the desired angle.

Leader Start Point: Left-click to specify the location of the arrowhead part of the Leader. If you do not want a Leader, you can right-click to skip the Leader and proceed on to place the next Text entity.

Next Leader Point: Left-click to specify the next Endpoint of the Leader. You will continue being prompted for "Next Leader Point:" until you right-click or **Enter** to finish drawing the Leader. After you have finished drawing the first Annotation entity (with or without a Leader), you will continue to be prompted to place additional Annotation and Leaders. Right-click or press **Enter** to finish the command.

Pull-down Menu Location(s): Draw > Draw To Standard

Keyboard Command: drawitem

Prerequisite: Standards Database file (.cad)

Draw Label For Companion Linework

This command creates text using the text properties (layer, style, color, height, etc) defined in Item Standards Manager. This command prompts to select a linework entity from the drawing. The layer from this entity is used to lookup the text properties from the standards manager and then create text. For text items in the standards manager, there is a setting for Companion Linework Layer which needs to be defined using Item Standards Manager.

This command is a way to apply your CAD standards when adding text to linework already in the drawing. For example, if you have polylines for building perimeters drawn on your standardized layer for building perimeters, then you can use this command to add text to the buildings. In this case, you need to have a text item defined in the Item Standards Manager with a Companion Layer set to the layer for the building perimeters.

Pull-down Menu Location(s): Draw > Draw to Standard

Keyboard Command: textitem

Prerequisite: Linework on a layer defined in Item Standards Manager

Draw Standard Item by Quick Key

This command draws standard items defined in Item Standards Manager similar to the Draw Standard Item command. The difference with this command is that the item to draw is selected by typing in the quick key instead of selecting the item from the dialog. This quick key method can be faster than the dialog selection. In Item Standards Manager, each item has a field to specify a quick key.

Pulldown Menu Location(s): Draw > Draw to Standard

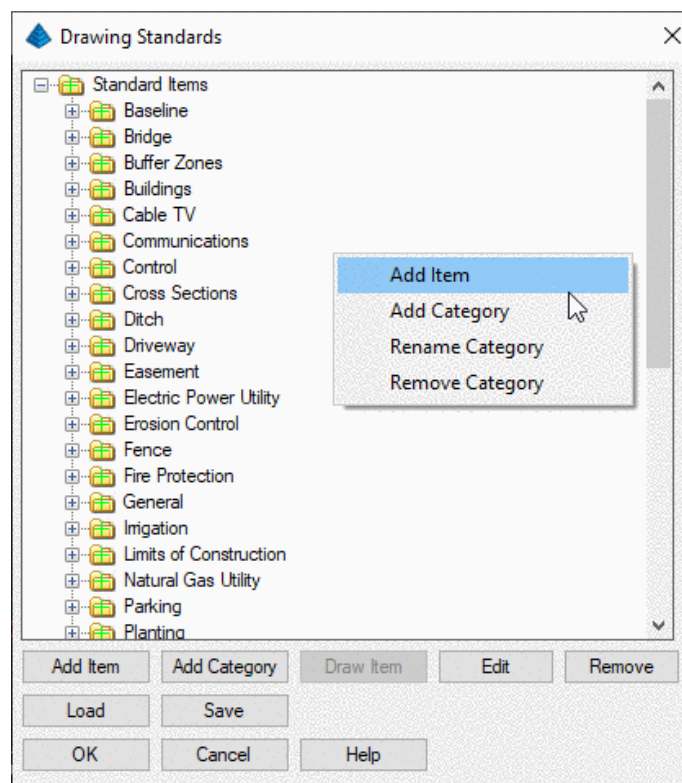
Keyboard Command: qkeyitem

Prerequisite: Items defined with quick keys in Item Standards Manager

Item Standards Manager

The **Item Standards Manager** command launches the **Standards Draw Manager** palette.

This palette has a right-click shortcut menu allowing you to Insert, Modify, Delete and otherwise manage Items stored in the Standards Database file (.cad):



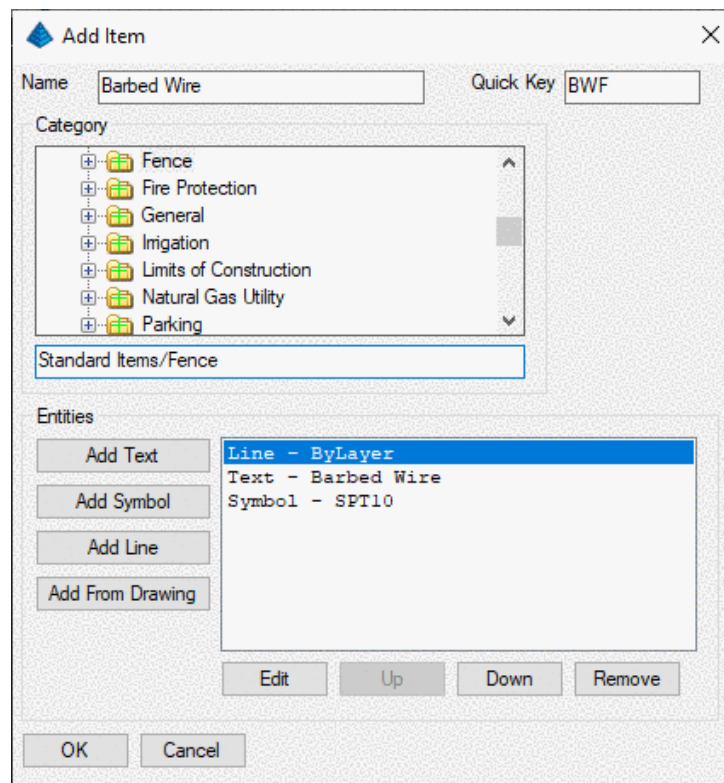
Once displayed, right-clicking inside the **Standards Draw Manager** displays a menu containing the Standards Database management commands. Other than the shortcut menu, this palette is essentially identical to the Draw Standard Item command.

The **Item Standards Manager** command permits you to browse through your list of Standard Items and manage the list of Items you can quickly place into a drawing. The command can be quickly accessed through the toolbar short-cut on the right shown below.



Add Item

When a Category has been selected/highlighted, you can add an Item into it.



Name: Provide a common name for Item in the selected Category.

Quick Key: Indicate a desired short-cut key-stroke (for example, a survey field code might be useful) that would be used to designate the Item. This value is not used or set for Categories.

Category: Select/identify the Category to which the item should belong.

Entities: Indicate the entity(ies) (and the order thereof) that comprise the Standard Item you are creating.

NOTE: The **Add from Drawing** option permits you to select a previously drawn Text, Symbol or Line/Polyline entity and add it to the Standard Item. Each item will have specific properties you can set that will permit you to control its insertion and composition into the Standard Item.

Add Category

When a Category has been selected/highlighted, you can add a sub-Category into it.

Draw Item

With an Item selected from the list, you can draw the item into the drawing. Refer to the Drawing Standard Items discussion for additional details.

Edit

This command allows you to make changes to the composition of the actively selected/highlighted Item or Category.

Remove

This command allows you to delete the the actively selected/highlighted Item or Category.

NOTE: Removing (deleting) a Category will also remove/delete the Items/Sub-Categories which are "housed" within the Category! You will be prompted for confirmation prior to deletion.

Load

This command allows you to replace the content of the current Drawing Standard library content with that of a different *.cad file.

Save

This command allows you to commit the values of the current Drawing Standard library content to a named *.cad file

Pulldown Menu Location(s): Draw – Draw to Standard

Keyboard Command: configItem

Prerequisite: None

Draw By Example

This command prompts you to pick an entity and then starts the appropriate draw command to begin creating another one of the selected type of entity. The properties such as layer and color of the original entity are used for creating the new one. For example, if you pick a polyline, this command will start the *Pline* command. Likewise if you pick text, this command will begin the *Text* command using the layer and style of the selected text.

Prompts

Pick Object for Command: *pick an entity*

The remaining prompts depend on the type of the selected entity.

Pulldown Menu Location: Draw

Keyboard Command: drawbyex

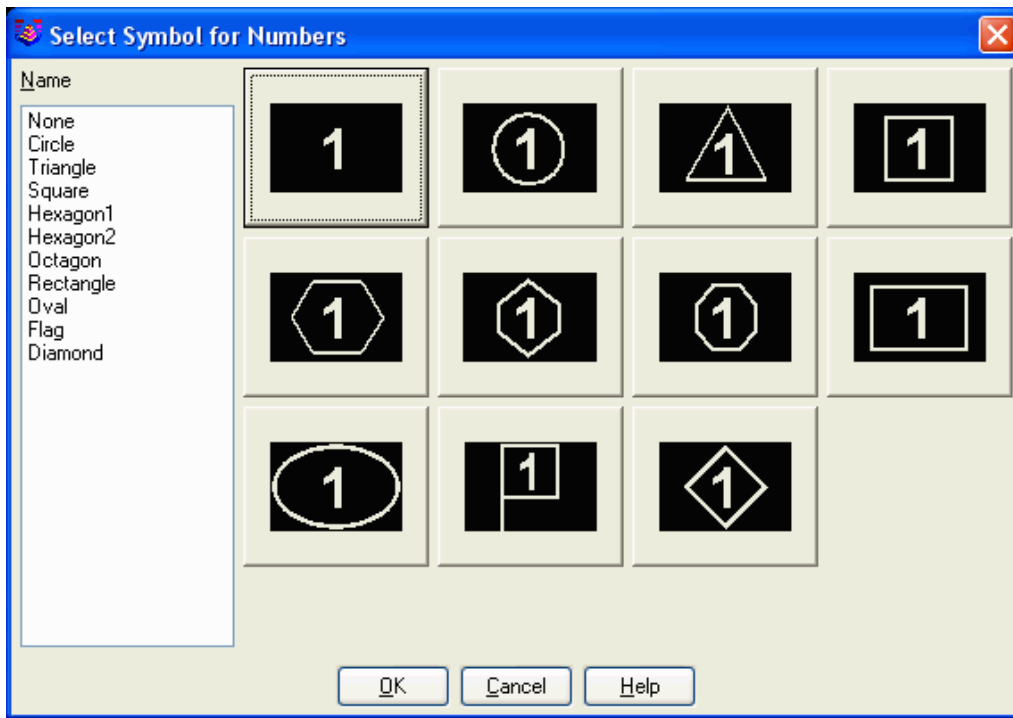
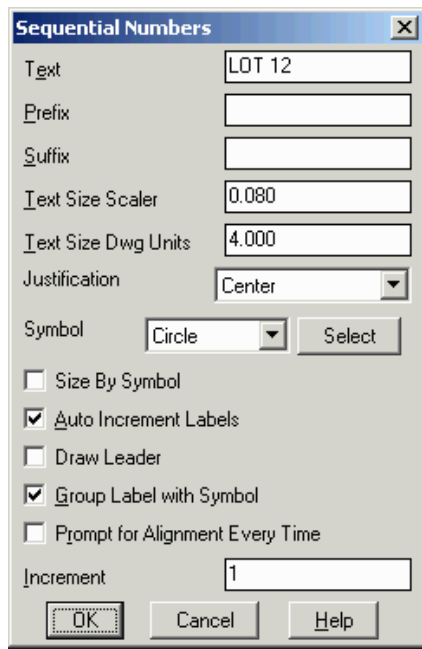
Prerequisite: Entities

Sequential Numbers

This command draws a text label and then increments to the next value for additional labels. The label can optionally be placed inside a circle, square or other symbol. The size of the symbol adjusts to fit the label size.

In the dialog, specify the **Text** label. The text **Prefix** and **Suffix** are optional. The **Text Size Scaler** is the text size in paper units that gets multiplied by the horizontal scale from Drawing Setup to set the text drawing size or **Text Size Dwg Units**. The **Justification** setting controls the text justification mode. The **Size By Symbol** method uses a fixed symbol size and sizes the label to fit within the symbol. When **Auto Increment Labels** is checked, the value entered in the Text field will be incremented by the value in the **Increment** field. The **Group Label With Symbol** option will make a group of the label text and symbol. When **Prompt for Alignment Every Time** is checked, you will be prompted for the alignment angle for each label, otherwise the alignment from the first label is automatically used for the other labels. The **Draw Leader** option creates a leader from the label to a picked point.

The label is drawn by combining the Prefix, Text and then Suffix into one text label. When placing multiple labels, the text portion of the label will increment by the value in the Increment field. For example, this command could be used to quickly label a series of boundaries by setting the Prefix to "Perimeter" and the Text field to the starting number. Then pick points inside the boundaries to label as "Perimeter 1", "Perimeter 2", etc.



Perimeter1	Perimeter5
Perimeter2	Perimeter6
Perimeter3	Perimeter7
Perimeter4	Perimeter8

Prompts

Select Symbol for Numbers dialog *select your symbol*

Sequential Numbering Options dialog *make your choices*

Pick point at beginning of label: *pick a point*

Pick point for label alignment: *pick a point to the right of the first point*

Pick point at beginning of label: *press Enter to end the routine*

Pulldown Menu Location: Draw

Keyboard Command: numbers

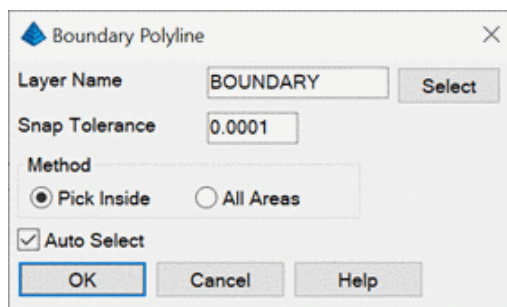
Prerequisite: None

Boundary Polyline

This is a streamlined analog of the AutoCAD command *Boundary*. The Carlson version is faster and works in many cases where *Boundary* fails. *Boundary Polyline* supports a **Snap Tolerance**, which means that you may specify a maximum gap to close when creating a closed polyline.

To create closed polylines from any existing linework, simply select all entities you would like to use and specify desired snap tolerance. There are two methods to create boundary polylines. The **Pick Inside** method creates boundary polylines one at a time. For this method, click inside openings you would like to trace and the routine will generate the corresponding closed polyline. Duplicate polylines are detected and not created, so that clicking more than once in the same area does not change anything. For the **All Areas** method, the program finds all the closed areas from the input linework and creates a boundary polyline for each area. The **Auto Select** option will automatically select the linework in the current drawing view to use for calculating the boundary. Otherwise, the program prompts for you to select the linework to use for the boundary. The **Type of Polyline** controls whether to use the elevations of the existing linework to create a 3D polyline, or ignore the elevations and create a 2D polyline at zero elevation.

These new polylines are created in the layer set in the options dialog. Layers of the original linework do not matter.



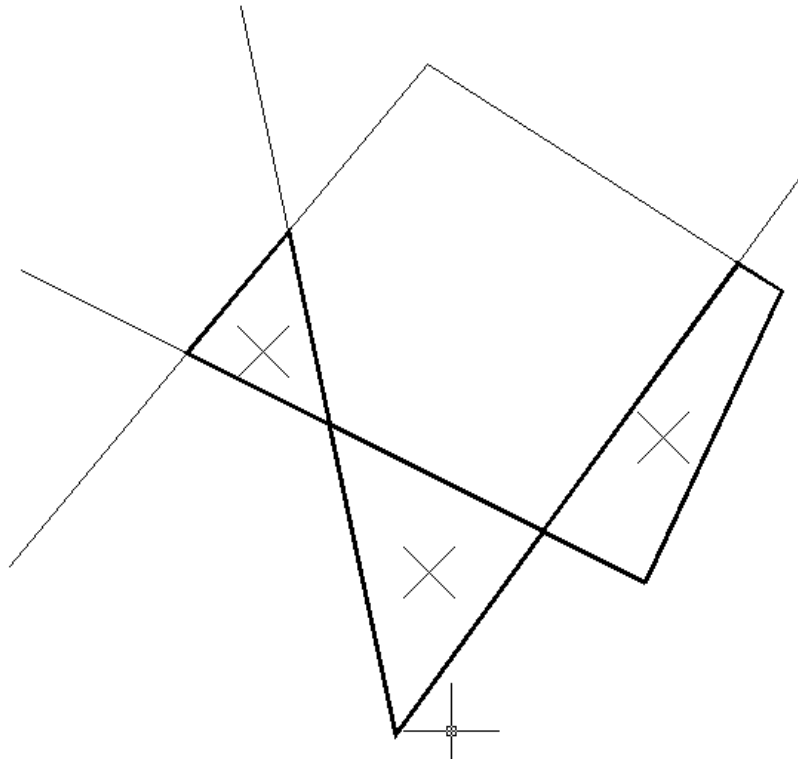
Prompts

Boundary Polyline dialog

Select polylines.

Select objects: *pick entities to be used*

Pick an internal point: *pick the points to enclose*



These three polylines are created from original linework by clicking at shown locations

Pulldown Menu Location: Draw

Keyboard Command: boundpl

Prerequisite: Entities

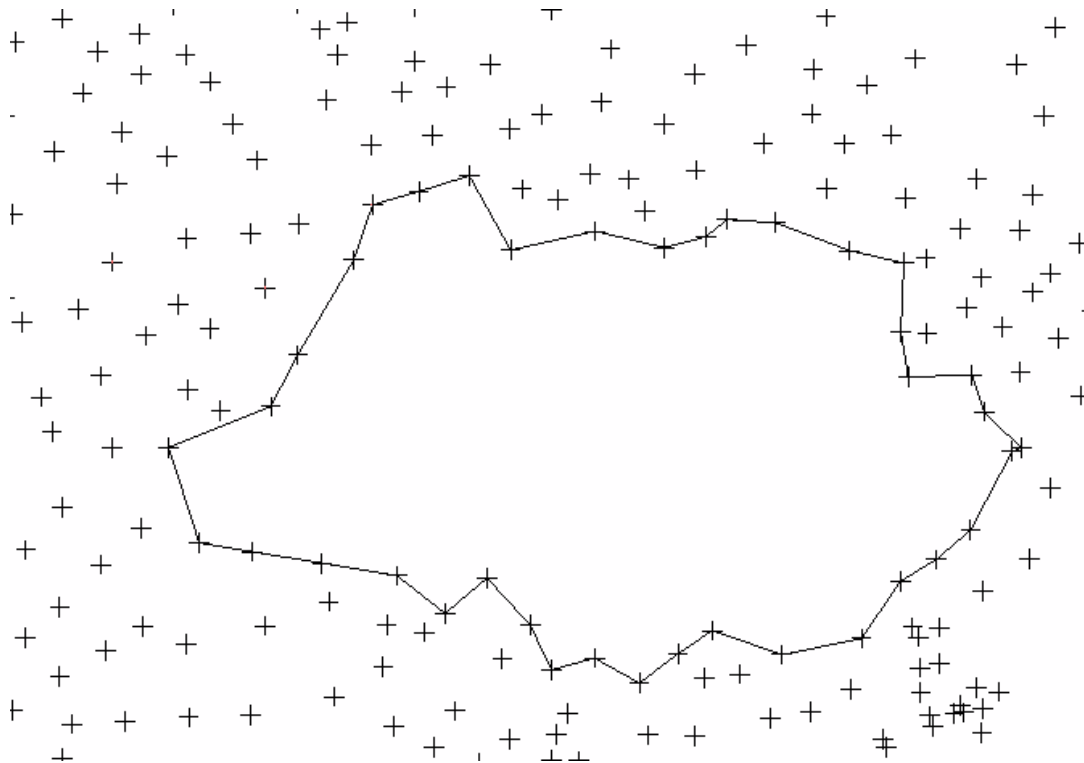
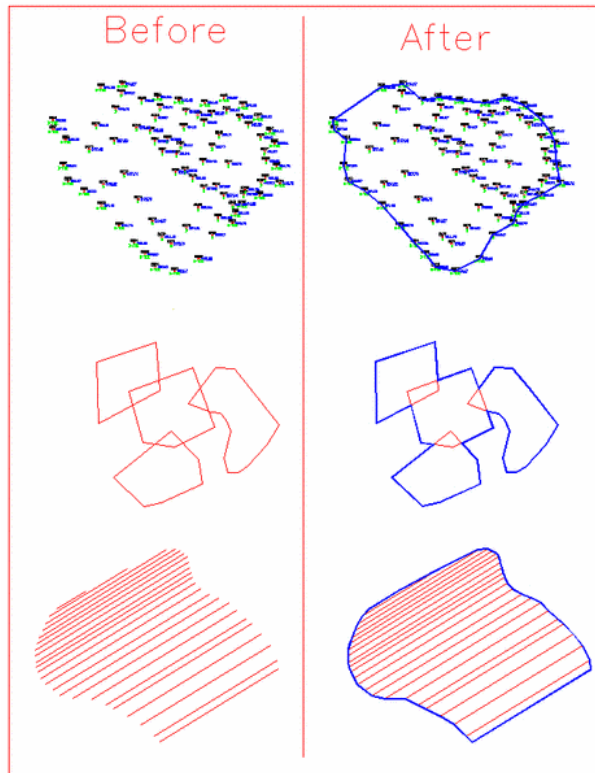
Shrink-Wrap Entities

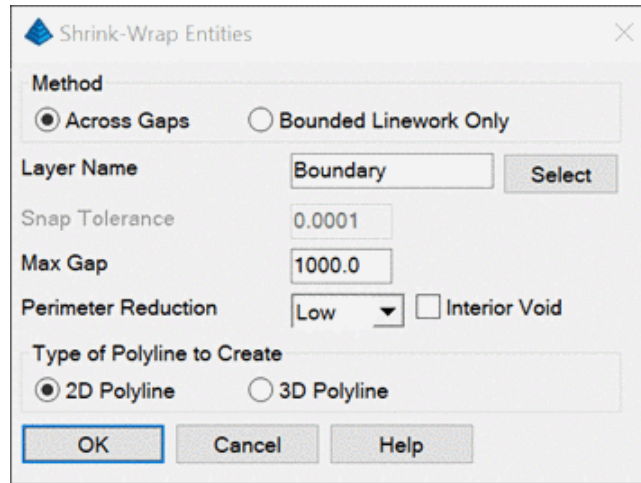
This command creates a closed polyline which encloses a given set of point and linework entities. The resulting polyline is created in the specified layer.

There are two different methods. The Across Gaps method creates a closed polyline through the points around the perimeter of the area defined by the selected entities. The Max Gap limits how far the polyline segments will connect between points. The Perimeter Reduction pulls in the shrink-wrap polyline to make a tighter fit similar to a vacuum suction effect. The 3D Polyline method uses the elevations of the input entities to create a 3D polyline. The 2D Polyline method ignores the elevations of the input entities and creates a polyline at elevation zero. For open polylines, as in the bottom figure, the Gap method works better, as it jumps across the gaps and connects the end points. The Interior Void method creates a closed boundary polyline for an empty area that is surrounded by points such as a pond. The Ignore Zero Elevations option will filter out entities at zero elevation and not use them in creating the perimeter. The Elevate Zero Elevations option will interpolate zero elevation vertices in the perimeter

polyline using other vertices with elevations.

For the Bounded Linework Only method, the shrink-wrap polyline follows the outside border of the selected polylines. The polylines that are processed have to be connected to be shrink-wrapped. The Snap Tolerance is the maximum gap that will be joined to make the closed polyline.





Prompts

Shrink-Wrap Entities dialog

Select points and linework to shrink-wrap.

Select objects: *select entities to process*

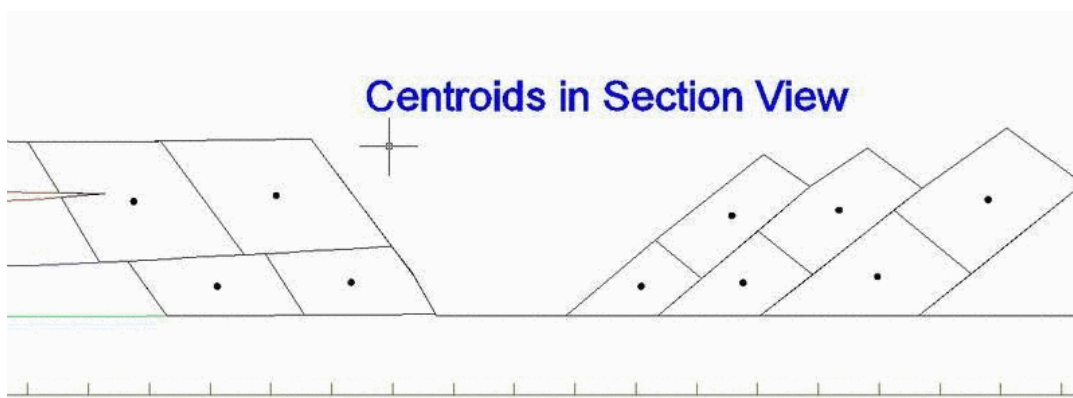
Pulldown Menu Location: Draw

Keyboard Command: swplines

Prerequisite: Entities

Centroid Point

This command draws points at the centroids of the selected polylines. In the section option, the areas need to be closed polylines representing the blocks to calculate the centroids of. This is useful for calculating haul distances and blast distances in section view. When choosing to come from a grid file, it finds the x,y position for the center of mass. Typically this grid would be the difference between existing and design surfaces, represented as a thickness grid. For example, this routine could be used to find the center of mass for a stockpile using a difference grid of the stockpile grid and base grid.



Prompts

Centroid from cross-section or grid file [<Section>/Grid]? S

Select closed polyline(s).

Select objects: 1 found

Select objects:

Center at 1462284.64,1971935.94

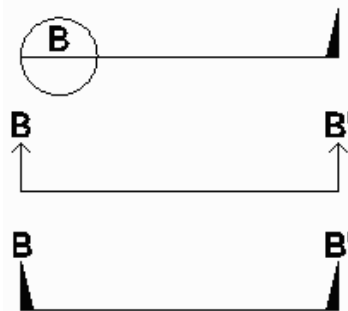
Pulldown Menu Location: Draw

Keyboard Command: plcenter

Prerequisite: Closed polylines for calculating the centroids

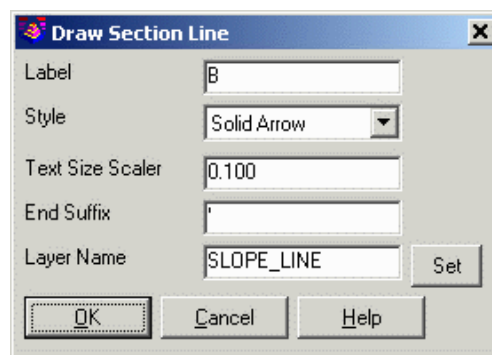
Section Line

This command creates a plan view section line with labels such as for showing the alignment for a cross section. Choose the section line properties in the options dialog. Then the program prompts for the two end points for the line and draws the section line with labels.



Three different styles of section lines

Prompts



Section Line dialog

Pick 1st point: *pick a point*

Pick 2nd point: *pick a point*

Pulldown Menu Location: Draw

Keyboard Command: sectionline

Prerequisite: None

SmarTrace

SmartTrace is an enhanced sketch routine. AutoCAD's *Sketch* command is acceptable for some tracing, but if a large contour map is sketched, it creates a huge file that loads and regenerates slowly. The problem is that the sketch command creates points at user specified increments, this small increment may be appropriate for sharp corners, however, in relatively straight stretches this is very inefficient.

SmartTrace solves this problem. The routine works by reducing the number of vertices created on polyline tracing by using a deflection angle and minimum and maximum distance. In sharp turns, it will create points at the minimum distance specified, and in relatively straight stretches it creates points at the maximum distance specified. If the angle of the polyline turns more than the specified deflection angle, SmartTrace also creates a point.

Prompts

Deflection Angle <4.0>: *press Enter* A range of 3 to 5 degrees is best.

Minimum Distance <5.0>: *press Enter* Usually 5% of map scale, or 1/20 inch.

Maximum Distance <50.0>: *press Enter* Usually 50% of map scale, or 1/2 inch.

Start point: *pick beginning point on polyline*

Begin Tracing ... Press Pick Button to End. Carefully move along polyline. *press the pick button to complete the polyline*

Pull-down Menu Location: Draw

Keyboard Command: TABLET3

Prerequisite: None

Polyline by Slope Ratio

This command allows you to draw or revise a polyline by specifying distance and slope ratio, percent of grade or pick points on the screen. The polyline could represent a section or profile which can be processed with the *Polylines End Area* or *Sections from Polyline*.

Prompts

Horizontal scale <50.0>: 20

Vertical scale <50.0>: 10

Pick point/<Starting elevation of polyline-section <100.0>>: *P* Enter the starting elevation or press Enter to use the default value in brackets and you are prompted for the starting offset or X coordinate. By entering P you are prompted to pick a starting point for the polyline/section.

Pick start point: *pick a point*

Slope ratio + for up slope - for down slope.

Undo/End/switch to Left/Pick point/<Enter Right Distance>: 130

End/Percent slope/<Slope Ratio (?:1) <2.0>>: 5 This enters a slope ratio of 5 to 1 for 130 feet to the right of the starting point.

Undo/End/switch to Left/Pick point/<Enter Right Distance>: L

Undo/End/Pick Point/<Enter Left Distance>: 110

End/Percent slope/<Slope Ratio (?:1) <2.0>>: P

Percent of grade slope: 2 This enters a slope of 2 in 100 for a distance of 110 feet to the left of the starting point.

Undo/End/Pick Point/<Enter Left Distance>: 30

End/Percent slope/<Slope Ratio (?:1) <2.0>>: 4

Undo/End/Pick Point/<Enter Left Distance>: E Entering E ends the command.

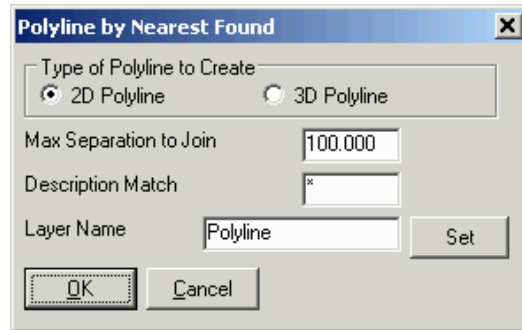
Pull-down Menu Location: Draw

Keyboard Command: PSR

Prerequisite: None

Polyline by Nearest Found

This command draws a polyline by connecting points using a nearest found method. The points to connect can be specified either by entering point numbers or picking POINT entities on the screen. The nearest found method draws a polyline by starting at one of the points and then connecting to the closest of the remaining points. Then a remaining point that is closest to one of the polyline end points is added until all points are part of the polyline.



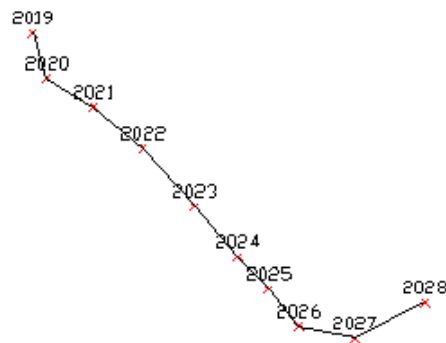
Prompts

Polyline By Nearest Found dialog

Select point from screen or by point number (<Screen>/Number)? *press Enter*

Select points.

Select objects: *pick points*



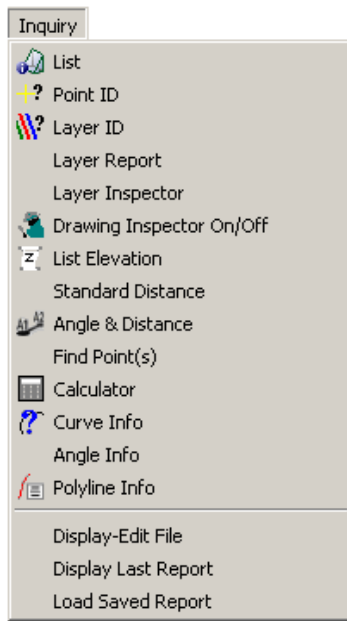
Pulldown Menu Location: Draw

Keyboard Command: plnear

Prerequisite: Points

Inquiry Menu

Shown here is the Carlson Inquiry menu. The top section contains detailed inquiry commands. The lower section of the menu includes report and file editing commands.



Point ID

This command reports complete information pertaining to a Carlson point. Although similar in function to the AutoCAD *ID* command, this routine is much more detailed. With this command, you are given the point number, as well as the northing, easting and elevation coordinates. You also are given the point description, and you are shown the name and the location of the coordinate file for the point.

Prompts

Pick point or point number: 255

```
PointNo.  Northing(Y)  Easting(X)  Elev(Z)  Description
255      4379.83      4265.48    19.01    GROUND/SHOT
N: 4379.83  E: 4265.48  Z: 19.01
PT#: 255  CRD File: c:\Carlson2008\data\mantopo.crd
```

Pulldown Menu Location: Inquiry

Keyboard Command: PT_ID

Prerequisite: None

Layer ID

This command reports the layer name of the selected entity. Under Options, you can choose to draw a label of the layer name.

Prompts

Pick entity to read layer [Options]: *pick an entity*

Layer: FINAL

Pick entity to read layer: *press Enter to end*

Pulldown Menu Location: Inquiry

Keyboard Command: layerid

Prerequisite: None

Layer Report

This command generates a report containing all the layers defined in the drawing. Along with the layer names, the report includes the number of entities on each layer, and the color, linetype and lineweight for each layer.

Layer Report

Drawing: C:\sample\example1

Layer Name	Entity Count	Color	Linetype	Lineweight
0	5	White	CONTINUOUS	
AREA_PERIM	0	White	CONTINUOUS	
AREATXT	0	Magenta	CONTINUOUS	
BL	8	White	DASHED	
BL-LAB	76	Red	CONTINUOUS	

Pulldown Menu Location: Inquiry

Keyboard Command: reportlayer

Prerequisite: None

Layer Inspector

This command is used to inspect and work with layers in the drawing. This command is ideal when you are working on a very dense and complex drawing which has many layers and you want to review the entities on different layers. In some cases, there will be layers that you would want to erase. Another scenario might be that you'd like to highlight a layer that is hard to find and see.

The Layer Inspector command has a dialog that docks to the bottom of the drawing window which keeps the drawing window visible while running the command. On the left of the dialog is a list of all the layers in the drawing. To inspect a layer, highlight the layer name from this list. You can inspect multiple layers at a time by selecting multiple layers in the list using the Shift and Ctrl keys while picking in the list. When a layer is selected, the Entity Count reports how many entities in the drawing are set to that layer. The Zoom toggle will zoom the drawing window to the extents of the entities on the layer. The Isolate toggle will freeze all other layers. The Highlight toggle will highlight all the entities on the layer. The Restore View On Exit will set the drawing window to the original position when Layer Inspector was started. The magnify and arrow buttons are used to zoom in/out and pan the drawing window. The Rename button allows you to rename the layer. The Erase Entities button will erase all the entities on the layer. The Purge button will purge the layer from the drawing which is only available when there are no entities on the layer. The Current button sets the layer as the current layer for the drawing.



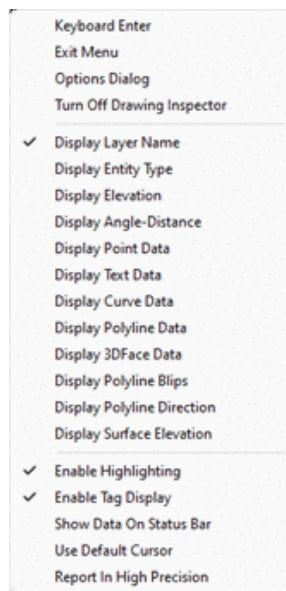
Pulldown Menu Location: Inquiry

Keyboard Command: layer_inspect

Prerequisite: None

Drawing Inspector

This command reports object properties to you as you move the cursor over an entity. You can simply move the pointer over an entity and the selected property will be displayed either in a pop-up window next to the pointer and/or on the status bar, depending on the selected option. Drawing Inspector is a transparent command that can run while other commands are running. Once Drawing Inspector is started, it will stay active even while running other commands until you turn it off. To turn off Drawing Inspector, run the command again to toggle it off by pick Drawing Inspector from the Inquiry pull-down menu or from the toolbar or by typing the command name, or right-click and choose Turn off Drawing Inspector. The options for this command are set in the menu that pops up by clicking the right mouse button. The available properties are: Layer Name, Elevation, Azimuth-Distance, Bearing-Distance, Point Data, Text Data, Curve Data, 3D Face Data, Polyline Data and Polyline Blips.

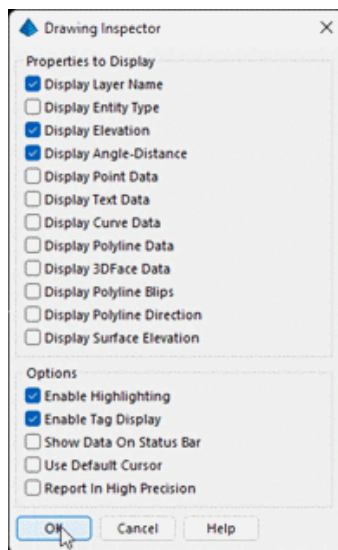


In the *Drawing Inspector* menu, you can choose one or more properties to display.

Keyboard Enter turns off the dialog box and returns to command control

Exit Menu exits the dialog box but leaves Drawing Inspector active

Options Dialog opens a separate dialog box which allows the On/Off toggle of display options. This is useful for turning on or off multiple display options at once.



Turn Off Drawing Inspector turns the routine off completely.

Display Layer Name: displays the layer name of the entity.

Display Entity Type: displays the type of the entity (ie. TEXT or POLYLINE).

Display Elevation: displays the elevation of the entity.

Display Angle-Distance: displays the direction and distance of a line. The angle mode will follow that in the Drawing Setup

Display Point Data: displays the coordinate data of point.

Display Text Data: displays the contents of text.

Display Curve Data: displays the radius, arc length, chord length and delta angle of a curve.

Display Polyline Data: displays the end point elevations, horizontal distance, slope distance, slope ratios and distance along the polyline for the current cursor position.

Display 3D Face Data: displays the Z elevations at the face corners.

Display Polyline Blips: displays temporary blip plus marks at the vertices of polylines.

Display Polyline Direction: displays temporary arrows to show the direction of polylines.

Display Surface Elevation: prompts for a TIN or grid surface file to load and displays the surface elevation at the current cursor location.

In the *Drawing Inspector* menu, you can also choose how the property information is reported.

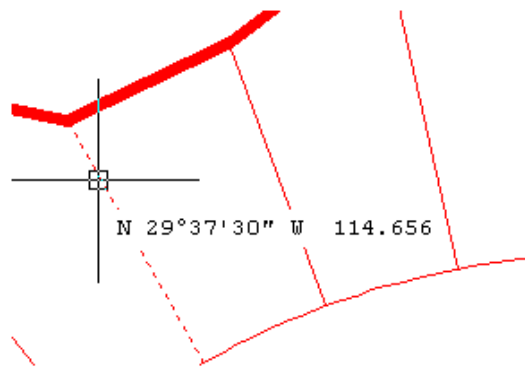
Enable Highlighting: Allows you to highlight the object that the *Drawing Inspector* is reporting.

Enable Tag Display: Enables you to view the information next to the cursor on the screen.

Show Data On Status Bar: Enables you to view the information on the status bar, in the lower corner of the screen.

Use Default Cursor: When enabled, only the drawing cursor shows. When disabled, the mouse pointer is also shown.

Report In High Precision: When enabled, displays 8 decimals on distance and 4 decimal seconds on angles.



Example of Drawing Inspector reporting Angle-Distance using the Tag Display

Pulldown Menu Location: Inquiry

Keyboard Command: inspector

Prerequisite: None

Bearing & 3D Distance

This command reports the slope distance, slope ratio, bearing, azimuth and vertical angle between two 3D points. Pick or enter the coordinates of two points or select a line or polyline segment to calculate between the segment endpoints.

Prompts

Specify bearing-distance from (Line/PLine/<Points>)? *press Enter*

Pick point or enter point number: *pick a point*

Pick second point or enter point number: *pick a point*

Horiz Dist: 233.4 Slope Dist: 233.4 Elev Diff: 0.0 Vert Ang: 0d0'0"

Slope: 0.0% 0.0:1 Bearing: S 71d15'37\"

Pulldown Menu Location: Inquiry

Keyboard Command: 3DIST

Prerequisite: None

Find Point

This command can be used to find a point in the current CRD file with a certain point number or description. For example, if you entered RAD* the command would plot a preview arrow at all the points that have the letters RAD as part of the description. i.e. RADPT1, RADPT2, RADPT3, etc. This command is not case sensitive (test is considered the same as TEST). Matching points are highlighted on the graphics screen with the preview arrow and listed on the text screen.

Prompts

Find by point [N]umber or [D]escription <N>: press Enter
Point number or range of point numbers to find <1>: 8*10
8 4856.75 4747.20 0.00
9 4909.25 4648.37 0.00
10 4223.30 4545.46 0.00 RADPT

If you respond with *D* for the first prompt the program prompts:

Conforms to AutoCAD's wild card matching.
Point Description(s) text to search for <>: rad*
Searching file C:\Carlson\DATA\LOT.CRD for point descriptions matching RAD* ...
7 4817.02 4662.73 0.00 RADPT
10 4223.30 4545.46 0.00 RADPT
Point(s) found 2
Pulldown Menu Location: Inquiry
Keyboard Command: fpnt
Prerequisite: None

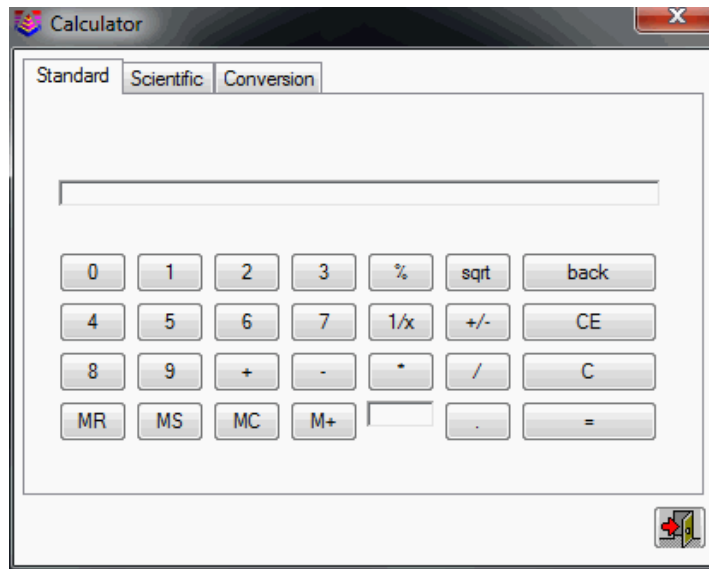
Calculator

The Carlson Calculator command uses a convenient pop-up calculator with three tabs for a standard calculator, scientific calculator and conversion calculator. The standard calculator does basic math calculations using expressions such as +, -, / and *. The scientific calculator has angle and other functions. The conversion calculator has feet-metric and angle conversions including radians. The standard and scientific calculators support RPN. Here is how RPN works:

1+2 = 3
- type value 1 + Enter
- type value 2 + Enter
- type +
X = 3.

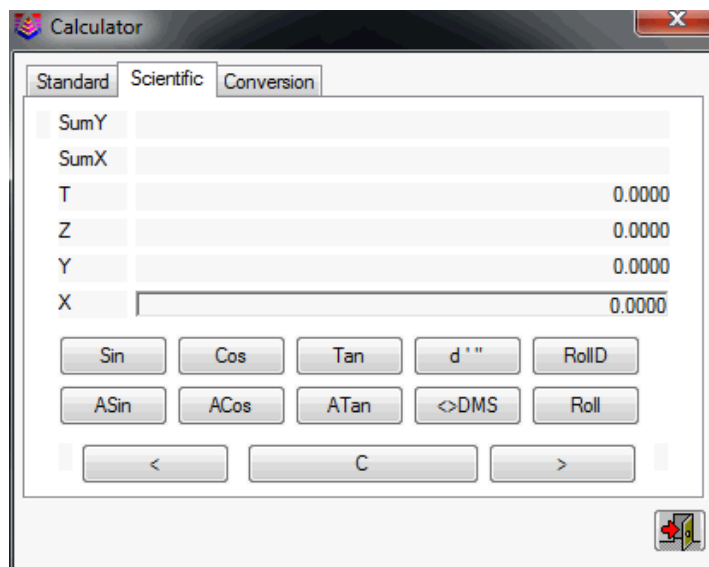
Standard Calculator

Most basic calculations can be performed using this tab in the calculator. Memory functions are also available.



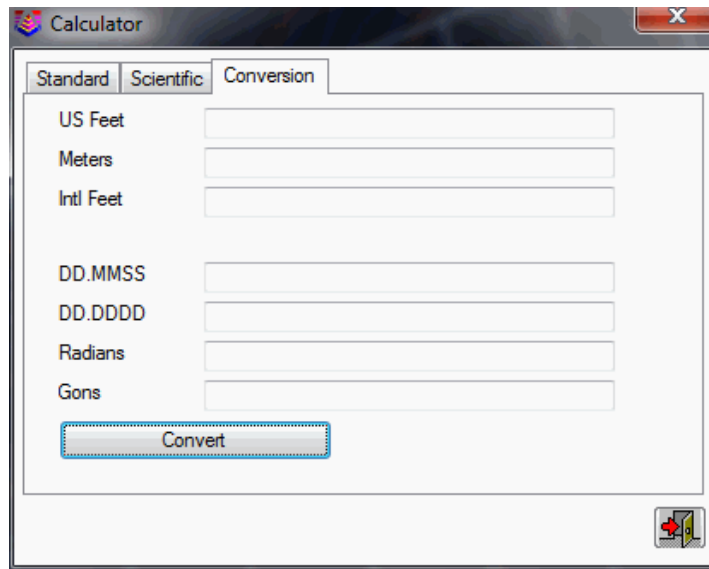
Scientific Calculator

Values can be entered on the X register. The values can be rolled up and down with the up and down arrow keys and the Roll and RollD buttons on the dialog. The Enter key finishes the entry of a number and pushes the stack. The C on the touch screen clears an entry. Additional functions on the screen can be obtained through touching the scroll [<] and [>] area of the screen.



Conversion Calculator

This mode provides for conversion between many units. Enter a value in any field and press Enter to find the conversion value. The following units are available in Feet, Meters and International Feet Degrees, Minutes, Seconds and Gons/Grads and Decimal Degrees.



Pulldown Menu Location: Inquiry

Keyboard Command: cscal

Prerequisite: None

Curve Info

This command displays information about a curve/arc. The curve can be defined by an arc entity or polyline arc segment or by selecting three points on the arc. The three points can be defined by point number or picked on the screen. The curve data is displayed in the text window with an option to be displayed in the Standard Report Viewer. Click Exit to return to the graphics window.

Prompts

Define arc by, Points/<select arc or polyline>: *select the arc entities*

Endpoint: (4923.81 5193.15 0.0)

Other Endpoint: (5168.27 5274.03 0.0)

Radius Point Coords: (5126.6 4990.09 0.0)

Chord Bearing: N 71d41'33" E

Chord Azimuth: 71d41'33"

Delta angle in radians: 0.9304628295

RoadWay Degree of Curve: 19d57'56"

RailRoad Degree of Curve: 20d4'4" **Chord Crv Length:** 265.66 **Excess:** 1.36

External: 34.13 **Mid Ord:** 30.50 **Tangent:** 144.06

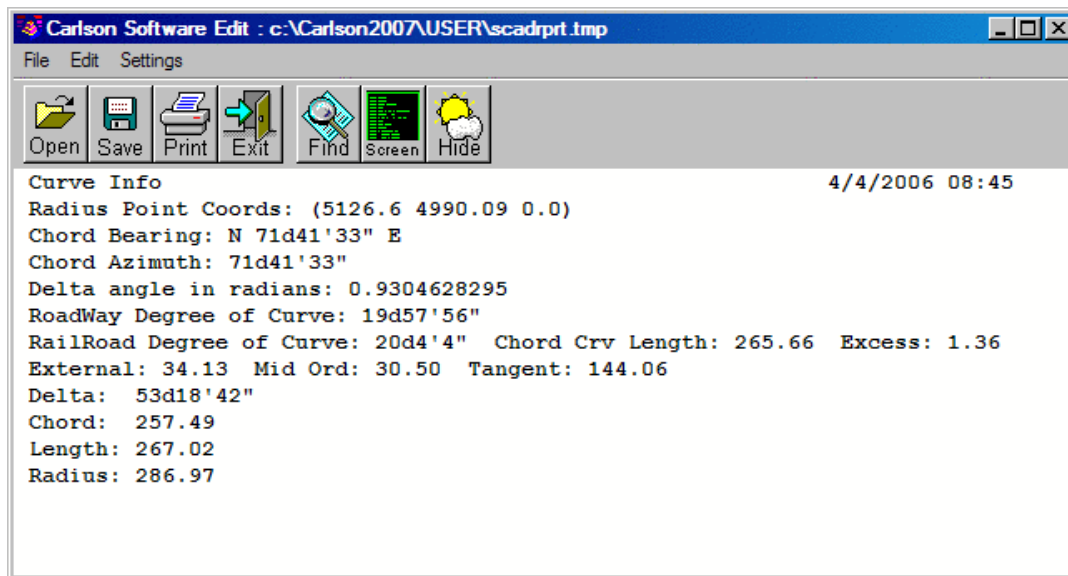
Delta: 53d18'42"

Chord: 257.49

Length: 267.02

Radius: 286.97

Display curve data in report viewer [Yes/<No>]? Y



Pulldown Menu Location: Inquiry

Prerequisite: None

Keyboard Command: cinfo

Polyline Info

This command reports the length and elevation of the selected polyline or line.

Prompts

Pick Polyline or Line: *pick a polyline or line*

Polyline length: 7702.75 **Slope distance:** 7702.75 **Avg elev:** 1700.00 **Avg slope:** 0.00%

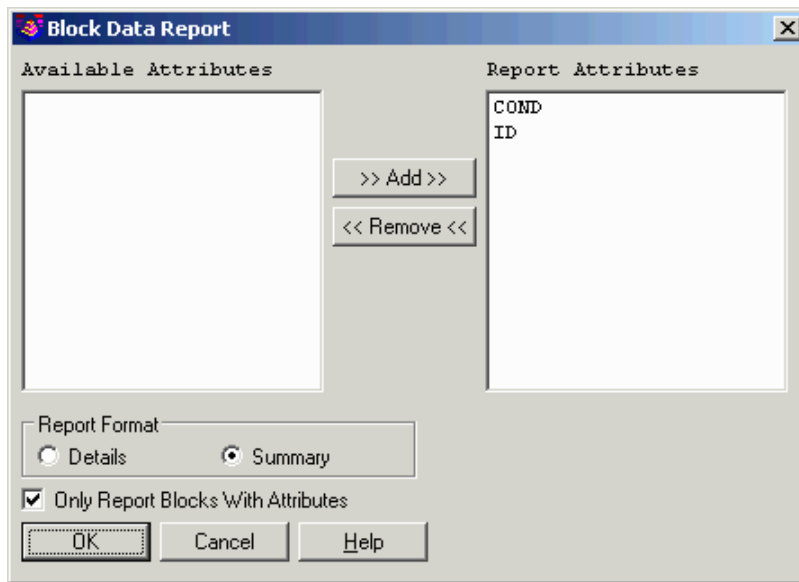
Pulldown Menu Location: Inquiry

Keyboard Command: polylen

Prerequisite: None

Block Data Report

This command reports the number of different types of blocks in the drawing along with the block attributes. After selecting the blocks to report, the program has a dialog with all the block attributes. You can select which attributes to report. The Details option for the Report Format includes the properties for each block including the northing and easting. The Summary Report reports the number of each type of block with matching reported attributes. The Only Report Blocks With Attributes chooses whether to report blocks with no attributes.



Prompts

Select blocks to report.

Select objects: *select blocks*

Block Data Report dialog

Report Formatter



Summary Report

COND Count

POOR 1

GOOD 3

FAIR 2

Total 6

Details Report

ID Northing Easting COND

100 2190229.6 6135223.9 GOOD

101 2190224.1 6135171.4 GOOD

102 2190200.4 6135103.4 GOOD

103 2190227.0 6135117.4 FAIR

104 2190198.9 6135164.8 POOR

105 2190198.9 6135239.4 FAIR

Pulldown Menu Location: Inquiry
Keyboard Command: blk_report
Prerequisite: Blocks with attributes

Layer Report

This command generates a report containing all the styles defined in the drawing. Along with the style names, the report includes the number of entities on each style and the style properties.

Style Report

```
Drawing: C:\files\user\Project.dwg
Style      Font          Text Count Fixed Height Width Factor Oblique Angle
ROMAND     romand.shx     1198      0.0        1.0 0°00'00''
ROMANS     romans.shx     3981      0.0        1.0 0°00'00''
ROMANT     romant.shx     104       0.0        1.0 0°00'00''
```

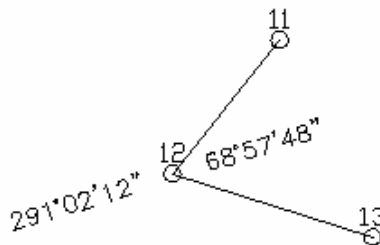
Pulldown Menu Location: Inquiry
Keyboard Command: style_layer
Prerequisite: None

Angle Info

This command reports the interior and exterior angles defined by two joining line segments or by three points. The coordinates, angles and distances of the line segments are also reported. The report is display in the standard report viewer. When the input points have different elevations, there is an option to report the angle in 2D or 3D. For 2D, the elevation are ignored. For 3D, the points are projected onto a plane to report the 3D angle.

```
Angle Information
      Point#      Northing      Easting
Start Point:  11      4728.73      5660.09
Corner Point: 12      4684.89      5624.99
End Point:    13      4664.02      5690.60

      Bearing      Distance
First Side:  S 38°40'56'' W  56.16
Second Side: S 72°21'16'' E  68.85
Interior:    68°57'48''
Exterior:    291°02'12''
```



Prompts

Define angle by, Points/<select line or polyline>: *P for points*

1st Point?

Pick point or point number: *11*

2nd (Corner) Point ?

Pick point or point number: 12

3rd Point?

Pick point or point number: 13

Interior: 68°57'18" **Exterior:** 291°02'42"

Angle Info Report Viewer

1st Point (Enter to end)?

Pick point or point number: *press Enter*

Pulldown Menu Location: Inquiry

Keyboard Command: ainfo

Prerequisite: None

Display-Edit File

This command allows you to edit or review an ASCII/text file. Files are displayed in the *Standard Report Viewer* section of this manual.

Pulldown Menu Location: Inquiry

Keyboard Command: sedit

Prerequisite: A file to edit

Text File To Report Formatter

This command loads data from a comma separated text file into the Report Formatter. The first line in the text file is used for the header names for the report. The rest of the lines in the text files has the field data for the report. Here's an example text file:

```
Point,Northing,Easting,Elevation,Description
11,4684.34317,5652.95193,501.47000,IP
12,4792.02100,5436.00000,505.90000,UP
13,4651.52246,5516.53607,504.03000,BLD
14,4656.32969,5535.94973,504.72000,BLD
```

Pulldown Menu Location: Inquiry

Keyboard Command: csv_report

Prerequisite: Text file

Display Last Report

This brings up the last report generated by any Carlson command that uses the standard report viewer.

Pulldown Menu Location: Inquiry

Keyboard Command: report_up

Prerequisite: A previously viewed report

Load Last Report Formatter

This command brings up the last report generated by any Carlson command that used the Report Formatter.

Pulldown Menu Location: Inquiry

Keyboard Command: prev_report

Prerequisite: A previously run command that used the Report Formatter

Load Saved Report

This brings up the Report Formatter for the report data file saved previously by the Report Formatter.

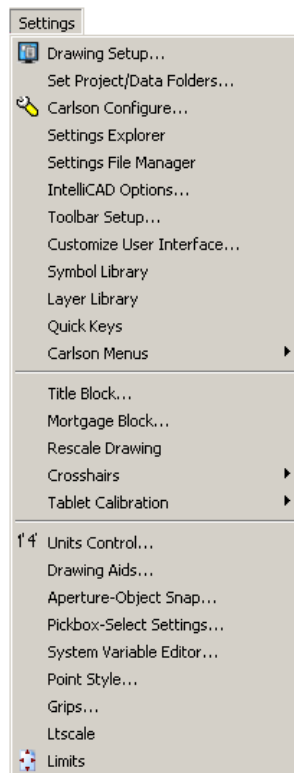
Pulldown Menu Location: Inquiry

Keyboard Command: load_report

Prerequisite: A saved report

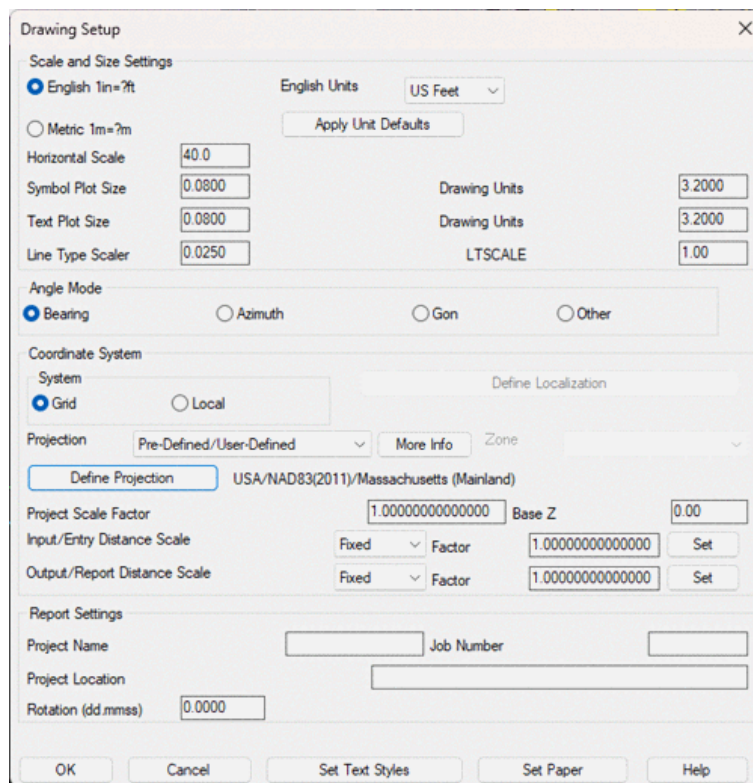
Settings Menu

Shown here is the Carlson Settings menu. The top section contains the commands most important for setting up the drawing. You should run Drawing Setup prior to starting work in your drawing. Additional setup and settings features are found in the middle section. The last section of the menu includes AutoCAD/IntelliCAD settings commands, including the System Variable Editor.



Drawing Setup

This command allows you to specify drawing parameters, including the plotting scale, size of symbols, label annotation size, and the angle mode.



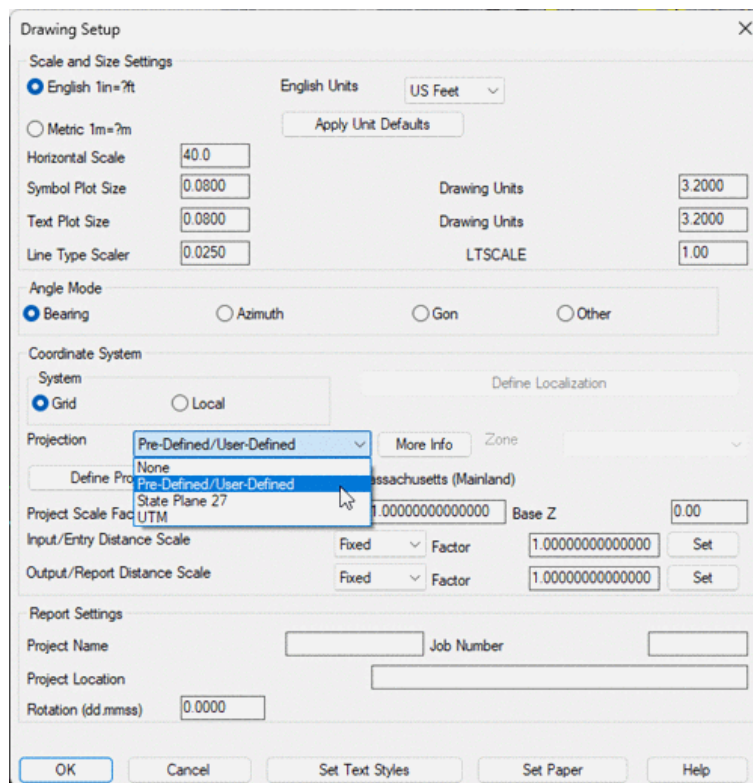
Specify **English 1in=?ft** or **Metric 1m=?m** as the unit mode to use. This affects the prompting and reports. When you are working on a drawing in English units, one unit equals one foot. In metric, one unit equals one meter.

- **Apply Unit Defaults** sets the default units for either English or Metric Units with regards to the Symbol, Text and Line Type Scalers.
- Specify the **Horizontal Scale** of the drawing. For example, if the horizontal scale is set to 50, then 1" = 50' is your drawing scale.
- The **Symbol Plot Size** value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The **Drawing Units** field shows the result of the Symbol Plot Size value (the scaler) multiplied by the horizontal scale.
- The **Text Plot Size** value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Text Plot Size is not entered in Drawing Units. The **Drawing Units** field shows the result of the Text Plot Size value (the scaler) multiplied by the horizontal scale.
- The **Line Type Scaler** option sets the linetype scale by multiplying this scaler by the horizontal scale.
Note: For those utilizing Annotation Scales [SCALELISTEDIT], this value is typically set at 1 for all drawing scales
- **Angle Mode-Bearing** sets reporting to bearing mode for any of the inquiry commands. (Modifies the settings in the AutoCAD *UNITS* command.)
- **Angle Mode-Azimuth** sets reporting to north based azimuth mode for any of the inquiry commands. (Modifies the settings in the AutoCAD *UNITS* command.)
- **Angle Mode-Gon** sets reporting to gon mode for any of the inquiry commands. (Modifies the settings in the AutoCAD *UNITS* command.)
- **Angle Mode-Other** lets the user determine angle mode by using the AutoCAD *UNITS* command.
- **Coordinate System** is an optional setting to define the drawing coordinate system. The coordinate system settings are used in commands like List Points and Label Lat/Lon to report geodetic coordinates from the drawing coordinates. The **Grid** System setting applies to drawing coordinates that are in a grid projection

system such as state plane coordinates. The Projection list selects the grid projection from the list of supported projections. Along with the Projection, there are selections for the zone and datum to use with the projection. When the drawing setup is in English mode, there is a projection setting for whether the feet are in US Feet or International Feet units. The **Local** System setting applies to all other coordinate system beside grid projections. The **Define Localization** button has settings to define the transformation from local coordinates to grid coordinates. With a localization defined, you can work in a drawing in local coordinates and still report lat/lon. The localization definition contains pairs of local and grid coordinates that define the transformation. Once a localization has been defined, it can be saved as a .dat, .loc or gc3 file for use on other projects or exported to SurvCE/PC.

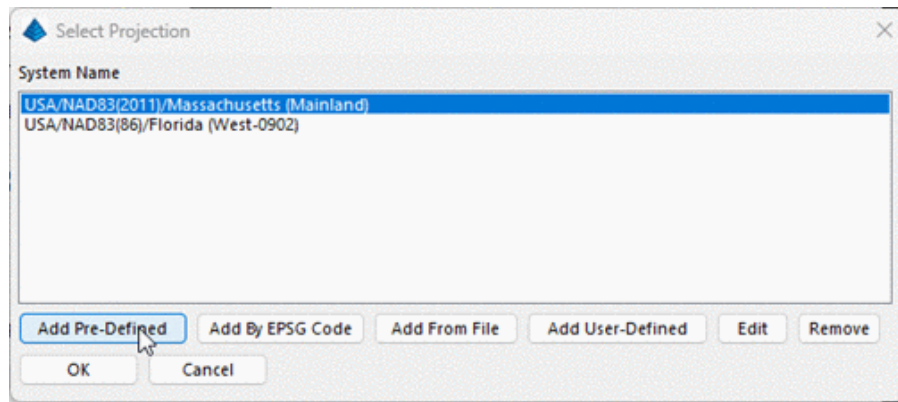
See the *Align To Local Coordinates* command in the *Field Module* for more information.

- **Project Scale Factor** is multiplied by the x,y coordinates when converting between drawing and geodetic coordinates.
- **Base Z** is used for calculating the combined scale factor for calculating geodetic areas.
- **Projection:** There are three choices for selecting a projections. If Pre-Defined/User-Defined is selected, you will be directed to a second dialog box for selecting or creating the projection. If State Plane 27 or UTM are selected, you can select the state or UTM zone respectively from the adjacent drop down menu.

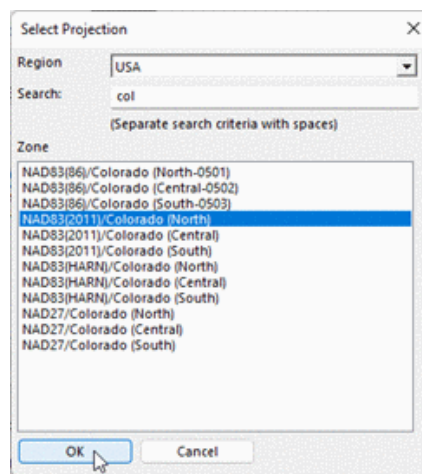


See also *Alternate Drawing Setup* for additional information regarding projections and the previous Lat/Lon datum selection.

- When you pick **Pre-Defined/User Defined**, a dialog shows a list of recently selected Pre-Defined projections.



You can pick from this recently used list, or pick the **Add Pre-Defined** to select from the built-in list.



This expanded Pre-Defined selection includes the projections used in SurvCE which has hundreds of projections including the US County projections for Indiana INDOT INGCS, Iowa Regional Coordinate System, Minnesota, Oregon and Wisconsin (WCCS and WISCRS) as well as from around the world.

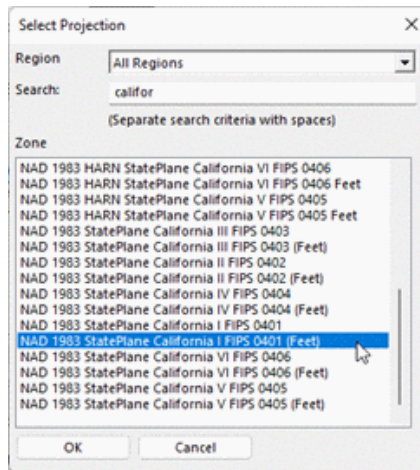
Note: Many projections in the list contain a designation in parentheses () indicating the realization date or specific projection reference. 86, 2011, and HARN are such designations. Pick the datum and realization that your project coordinates are located in. In the Continental US, the most recent realization is 2011 and is suited to most surveys. Changing the projection in the Drawing Setup **DOES NOT** alter the coordinates in the CRD/CRDB file. To transform coordinates from one projection to another, use Coordinate Transformation in the **Coordinate File Utilities**.

The **Search** field shown above, allows you to begin typing the projection name while the program filters available projections based on the entered text characters within that projection name.

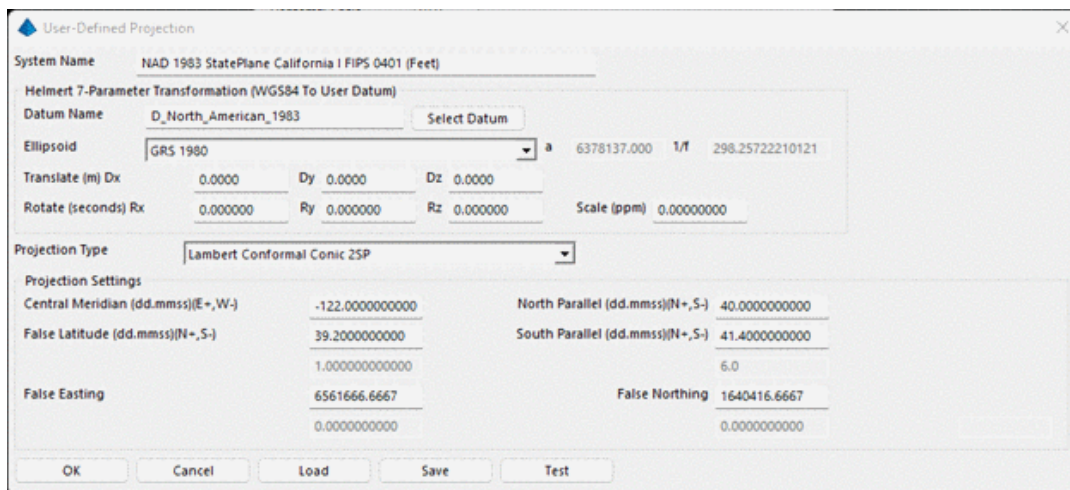
- The **Add by EPSG Code** allows the selection of existing EPSG Codes by entering in the projection number in a separate dialog box.



- The **Add From File** button reads in a projection saved to a file by this routine or by SurvCE CSL or ESRI PRJ.



- The **Add User-Defined** routine defines a projection by setting the ellipsoid, choosing the method and entering the parameters.



There are over 25 built-in ellipsoids to choose from such as Clarke 1880. You can also manually enter the ellipsoid

values. The projection definition includes the 7 parameter Helmert transformation to go from WGS-84 to the user datum. There are over 20 projection types to choose from such as Transverse Mercator. After selecting the projection type, there are edit fields for each of the parameters for the selected projection. The **Load** button allows you to browse and load predefined projections (cls, udp, prj) file formats. The **Test** button brings up a calculator to enter a lat/lon and report the projection coordinates as a way to test that the projection parameters are entered correctly and are working.

- The **Edit** button allows you to change the name or parameters of an existing projection.
- The **Remove** function removes a projection from the list of recently used projections.

Note: Besides Drawing Setup, these projections are used in the Coordinate Transformation function in Coordinate File Utilities.

Alternate Drawing Setup

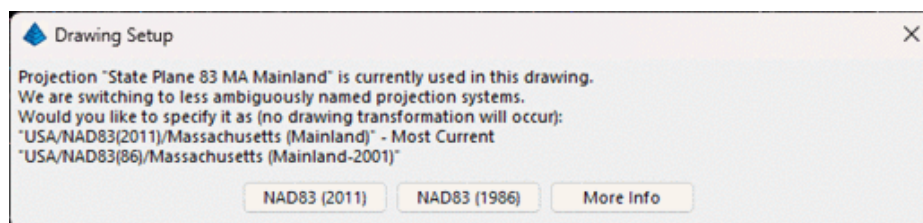
In some cases, usually when opening an older project, you may be presented with a Drawing Setup dialog box with different options:

The above settings, **Perform Geodetic Lat/Lon Transformation** are a replacement to the versions previous to 2024 release. In these earlier versions, there was a drop down menu with options for a Lat/Lon datum. The default for this selection was WGS84. In the Continental US, WGS84 was equivalent to State Plane 83 *or* NAD83(1986) making this setting ambiguous which prompted its removal. When opening an existing drawing, the lat/lon transformation setting in the Drawing Setup is compared to the equivalent settings in Carlson Configure->Survey Settings. If there is a mismatch, the above options will be displayed to allow the user to confirm or correct the transformation setting.

As described above, the newer selection of datum along with any realization date is a more complete selection of a projection. In most cases, users will prefer to have their lat/lon datum on the same datum as their coordinates and not perform any transformation. However, in some cases, especially when opening older drawings, this transformation is detected, and you are prompted to make a choice to rectify these settings.

Example:

You are opening an older drawing that was set to State Plane 83 with the option to use WGS84 as a lat/lon datum. When doing so, you are presented with the following prompt:



You select NAD83 (2011). At this point your projection is not consistent with the previously selected WGS84 lat/lon datum. You will see the above options in the Drawing Setup.

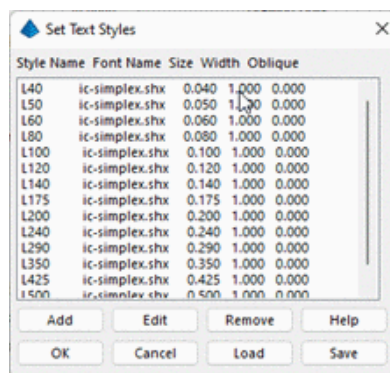
The options for this setting are:

- On - will transform lat/lon coordinates to the previously called WGS84 or other datum
- Off - will perform no transformation. (Preferred for most surveys)

- Use Survey Setting - will follow the options as set in the Survey Settings in Carlson Configure. This setting will remove the above options from the drawing setup placing control in the more global configuration settings.

Note: None of the above settings will alter the existing coordinates in a crd or crdb file. These transformation settings only apply to select routines within Carlson such as the use of geodetic bearings in reports.

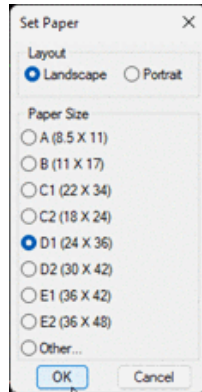
- **Project Name and Job Number** are optional fields that are used in the header for reports.
- **Project Location** is used in the header for reports and used to roughly geolocate the drawing when the Projection isn't set. This location can be used in the OpenDWG command.
- **Input/Entry Distance Scale** is used to adjust distances in a second system besides the drawing units. For example, it can be used to input grid distances when the drawings is in a ground coordinate system. This factor is applied in COGO commands that use distance for input such as Enter Deed Description. The scale factor can be set to a single fixed value or set to go from ground to grid or from grid to ground on-the-fly. The fixed scale factor can be entered directly into the edit box or calculated using the Set button which has feet-meters conversions as well as combined scale factor calculations for grid-ground factors. See the Scale Points command for more information on calculating the combined scale factor. For the on-the-fly factor between grid and ground, the program calculates the combined scale factor for each distance using the two end points for the distance line.
- **Output/Report Distance Scale** is used to show distances in a second system besides the drawing units. For example, this factor can be used to report distances in meters when the drawing is in feet, or it can be used to report grid distances when the drawings is in a ground coordinate system. This factor is applied in commands that have an option to label/report a second scaled distance such as the Inverse command and Annotate Defaults that applies to the angle/distance label routines. The scale factor can be set as described under Input/Entry Distance Scale.
- **Rotation** is used to adjust the bearings and azimuths for labels and reports such as in the Inverse and Auto-Annotate routines. This setting does not affect the coordinate values in the coordinate file. It is used only to query and annotate bearings.
- The **Set Text Styles** button creates text styles in the drawing for the current drawing Horizontal Scale with the specified Font Name and list of Style Names and Text Size Scalars.



For example, when the Horizontal Scale is set to 50 and there is a Style Name of L80 with Text Size Scaler of 0.08 in the list, then this function will create a text style in the drawing called L80 with the text height of 4 ($50 * 0.08$). Besides the font and size, you can also set the width factor and oblique angle for each style. Use the Save and Load buttons to store and recall these style settings to a .FNT file for sharing with co-workers or your own use. the **Add**, **Edit** and **Remove** buttons allow you to add your own text styles, alter the settings of existing ones, or

remove unwanted styles. The **Load** and **Save** buttons allow you to load previously created styles or save the current styles to an external file.

- The **Set Paper** button allows you to draw a rectangle on the screen that represents the edge of your paper. After you have set the horizontal scale, press the Set Paper button and the Set Paper dialog appears.



- The **Layout** option lets you specify landscape or portrait paper orientation. Landscape layout is where the width of the page is greater than the height of the page. Portrait layout is the opposite.
- The **Paper Size** option allows you to specify the paper size. The numbers in parenthesis represent drawing units and will be multiplied by the horizontal scale to determine the rectangle to be drawn. If you select the Other option, you will be prompted on the command line for the horizontal and vertical sizes of the paper.

Prompts (for Set Paper)

Pick or Type lower left corner point for border <(5000.00 5000.00 0.0)>: *pick a point*

Erase existing Set Paper boundary [<Yes>/No]? *Y* This prompt only appears if there is an existing paper boundary in this drawing.

Set Limits [Yes/<No>]? *Y* If you answer Yes to Set Limits, drawing limits are enabled, and AutoCAD restricts the coordinates you can enter to within the paper boundary. Drawing limits also determines the area of the drawing that can display grid dots, and the minimum area displayed by the Zoom All command on the View menu. To turn drawing limits off, type in LIMITS on the command line and set to Off.

Drawing Setup also sets the AutoCAD dimension scale (DIMSCALE) and linetype scale (LTSCALE) to the Horizontal Scale.

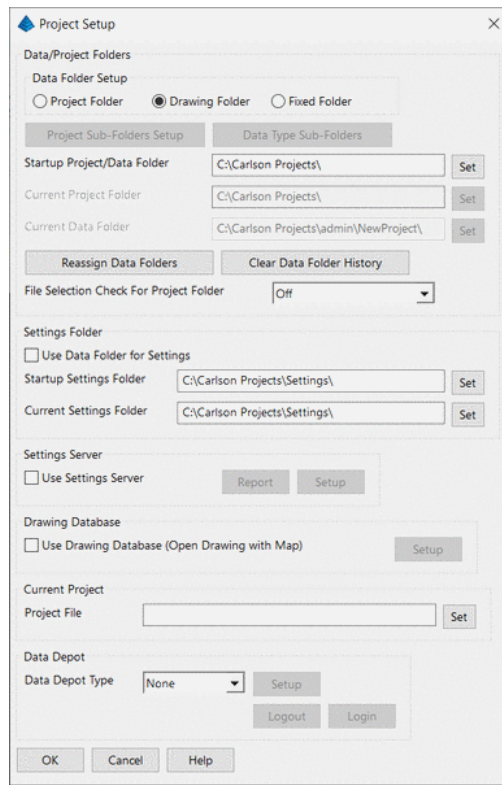
Pulldown Menu Location: Settings

Keyboard Command: setup

Prerequisite: None

Set Project/Data Folders

This command sets the project work folder, the data folder and the settings folder to use as the default folders for your Carlson drawing and data files. The PROJECT folder is the top-level folder for all the data sub-folders with all the files for the project. The DATA folder contains project specific data files such as coordinate (.CRD), profile (.PRO) and centerline (.CL) files. The SETTINGS folder contains program settings files that can apply to multiple projects such as Field-To-Finish Code Tables (.FLD) and Draw Profile Settings (.PFS). These folders are the defaults where the file selection dialogs will start in. When selecting files, you can change to another folder at any time.



Data Folder Setup: This grouping of controls provides varying levels of sophistication towards how data files associated with a given project are stored and organized on your computer system. Three options are provided:

- **Project Folder** - Data files are organized and stored (by default) into a user-definable sub-folder structure and this option is often used by larger organizations that have teams of employees working on a project. Selecting this option enables the Project Sub-Folders Setup and Data Type Sub-Folders buttons.
- **Drawing Folder** - Data files are stored (by default) into the same folder where the current project drawing has been directed and this option is often used by mid-sized or smaller organizations who seek only basic data organization.
- **Fixed Folder** - Data files are stored (by default) into a single folder and might be used by smaller organizations who do not require any type of data organization.

A note regarding file paths: When files are referenced within other files (for example, when a Centerline file is referenced in a RoadNet file), the program may store the file path in one of four ways: Relative path based on project, relative path based on reference file, relative path based on startup folder, or absolute path. As an example, consider the below files:

Startup Project/Data Folder: C:\Carlson Projects

Project Folder: C:\Desktop\Roads

Files within project folder: Road01.rdn, Centerline01.cl, Road Design.dwg

The Road01.rdn is the reference file, the Centerline01.cl file is the target file within the reference file.

Method	Variable	Source of Variable	Example of Target Notes File Path
--------	----------	--------------------	-----------------------------------

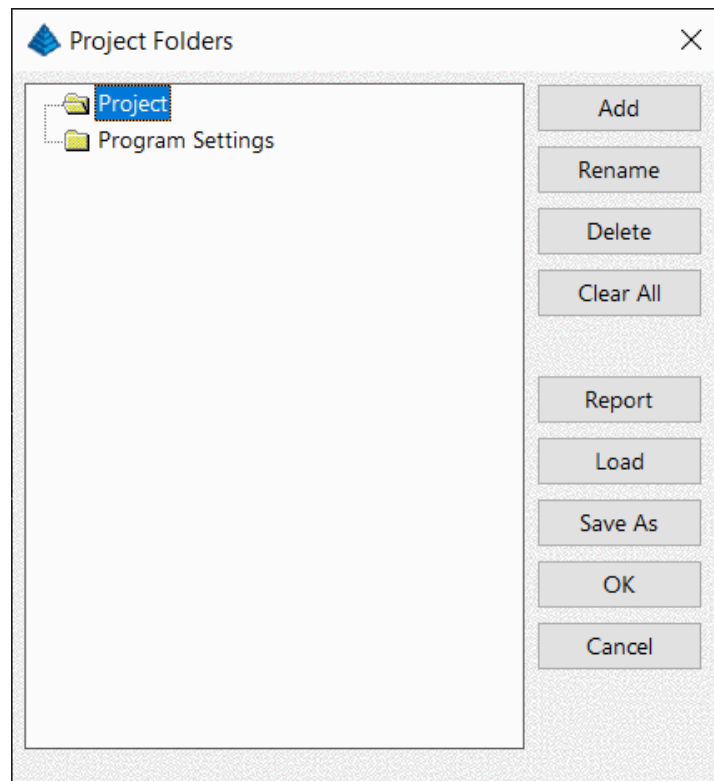
Relative path based on \$PROJECTPATH project folder	Project Folder (set \$PROJECTPATH\Centerline) according to Data Folder Setup)	This is the most commonly used method to store paths to target files. If the target file is stored in the project folder (either directly in the folder or in a subfolder), the path may use this method. The main benefit to this method is that the file paths will remain intact when the project folder is moved/copied to a new location or a different computer.
Relative path based on \$REFPATH reference file	Directory of Reference \$REFPATH\Centerline File	This method is a bit more resilient to folder reorganization. Here, the location of the target file is relative to the reference file (in this case, the RoadNet file). The benefit to this option is that the files do not necessarily have to be stored within the project folder. Therefore, if the reference file and the target file are reorganized within the project folder, or if they are moved outside of the project folder entirely, the paths will still remain intact.
Relative path based on \$WORKPATH startup folder	Startup Folder Project/Data \$WORKPATH\Centerline	This method is used when a target file is stored in the Startup Project/Data Folder.

Absolute path

C:\Desktop\Roads\Certification\ is the least resilient to change, as any degree of reorganization or renaming the folder structure will break the link to the file. This method is only used when the program cannot determine a pattern of association to either the project path or the reference path.

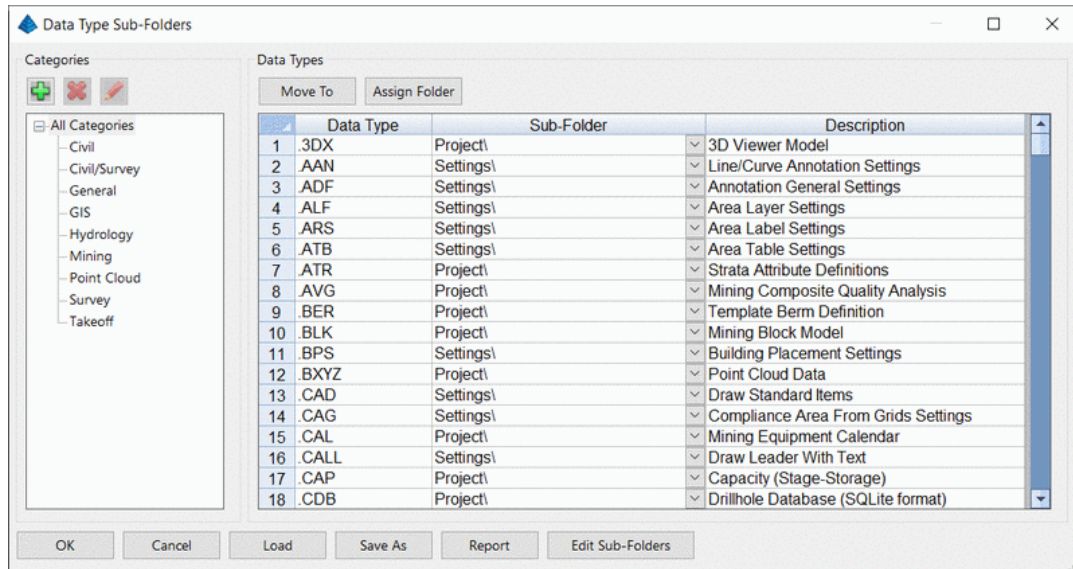
Please note that it is not recommended to manually modify these file paths within reference files. The program will automatically select the method to use for storing file paths.

Project Sub-Folders Setup: Click this button to create a folder structure (see the sample below) that is created when a new project is created. The list of project folders can be customized at any time but modifications to the folder structure will only occur on projects that are created after the modification(s) to the Project Folder list.



Data Type Sub-Folders: Clicking this button allows the various types of data files produced by Carlson to be assigned to a folder identified with the Project Sub-Folders Setup command, as illustrated below. File types set to the "Settings" sub-folder are for program settings files like Triangulate & Contour (.TRI) for settings like contour interval and layer. These settings are often stored for use across the program in the Settings Folder. Files types set to the "Project" sub-folder are for data files that are specific to the project like the Triangulate & Contour (.TIN) for the

surface data file. File types that are not assigned to a sub-folder are stored by default in the current project folder.



The following controls allow you to organize your data file types:

Control



Action

Creates a new data type sub-category.
Removes the selected data type sub-category. Any data types that have been assigned to the sub-category are subsequently migrated into the *Misc Data-Types* category.
Allows the selected data type sub-category to be renamed.

Category Controls

Move To: Associates the selected data type(s) with a data type sub-category. Use standard Windows **click**, **shift+click** and/or **ctrl+click** functionality to select multiple data types at the same time.

Assign Folder: Assigns the selected data type(s) to a project sub-folder. Use standard Windows **click**, **shift+click** and/or **ctrl+click** functionality to select multiple data types at the same time.

Load: Loads a previously saved Data Type Sub-Folder (.DSF) file.

Save As: Saves the current Data Type Sub-folder configuration to a Data Type Sub-Folder (.DSF) file.

Report: Allows the contents of the current Data Type Sub-folder configuration to be sent to a report.

Edit Sub-Folders: Initiates the Project Sub-Folders Setup command.

Startup Project/Data Folder: Indicate or use Set button to assign the start-up (or default) *Project Folder* location (when using the Project Folder option) or the start-up *Data Folder* location when using either the Drawing Folder or Fixed Folder option.

Current Project Folder: This setting is the top-level project folder for the current drawing.

Current Data Folder: This setting is the default data folder for the current drawing.

Reassign Data Folder: This function shows a list of all the folders used for data files associated with the current drawing. You can select a folder from this list and switch to another folder location which re-associates the data files. This function applies to projects that have been moved in the file system.

Clear Data Folder History: This function removes the association of all data files with the current drawing. The effect is to go back to defaults for data file selections.

File Selection Check for Project Folder: This option applies to the standard File Selection dialog for whenever a project data file is selected:

- **Off:** No folder checking is performed.

- **Warn Non-Project:** The program will display a warning if a project file is selected from a folder that is outside the Current Data Folder.
- **Prevent Non-Project:** The File Selection will not accept selecting a data file that is outside the Current Data Folder.

Use Data Folder For Settings: When enabled, this option sets the Settings folder to match the Data folder which, in effect, combines the Settings and Data folders into one folder.

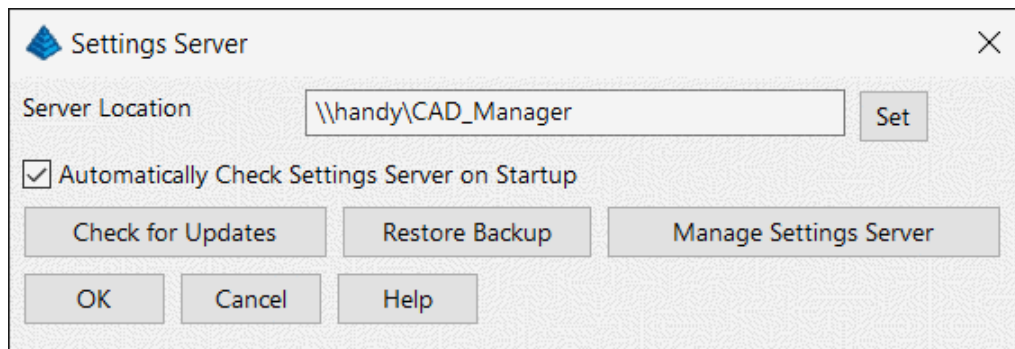
Startup Settings Folder: This folder is the default Settings folder for new drawings.

Current Settings Folder: This folder is the Settings folder for the current drawing.

Use Settings Server: The Settings Server is a system for distributing program settings and files to workstations in the office.

Report: Reports the date and time that files have been copied to your computer.

Setup: Provides an interface to configure the Settings Server as illustrated below:



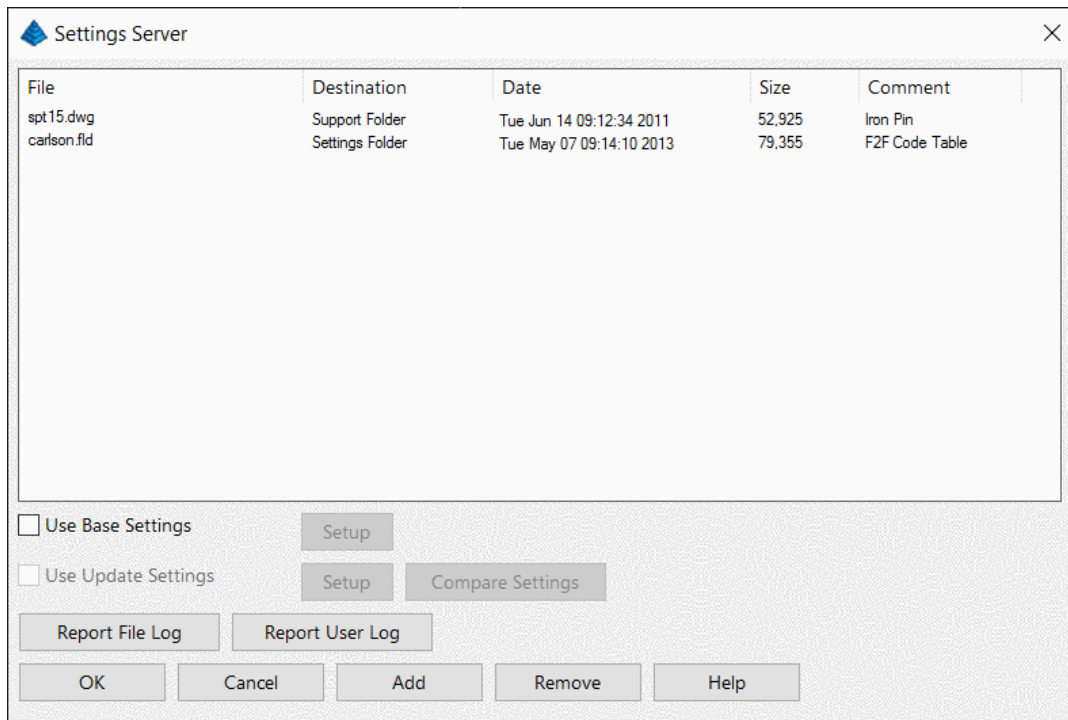
Server Location: Indicate the path to a shared folder on your office network. This folder is where the settings and files to distribute are stored.

Automatically Check Settings Server On Startup: When enabled, the settings are checked each time the program is started.

Check For Updates: This button copies any new files and settings from the Settings Server to your computer. When files are copied from the Settings Server to your computer, your existing files are backed up.

Restore Backup: This option allows you to bring back your original files.

Manage Settings Server: Use this button to setup the files and settings to distribute:



The Manage Settings Server dialog shows a list of the files to distribute. To control access to these settings and files, use Windows permissions to set the folder to *read-only* to all except the Server Manager.

Use Base Settings: Base Settings are copied to each workstation only once. After this initial setup, the workstation settings can be changed by that user. Use the **Setup** button to show the settings in the Settings Explorer command where you can review and edit the settings for each file.

Use Update Settings: Update Settings are copied to each workstation each time the workstation checks the Settings Server. These Update Settings can be used to make sure certain settings are kept set on workstations. Use the **Setup** button to show the settings in the Settings Explorer command where you can review and edit the settings for each file.

Compare Settings: This function reports the difference between the Base Settings and the Update Settings.

Report File Log: This button reports the date and time each file is distributed per user.

Report User Log: This command reports the date and time that users have checked into the Settings Server and it reports the version and build date of Carlson that they are running.

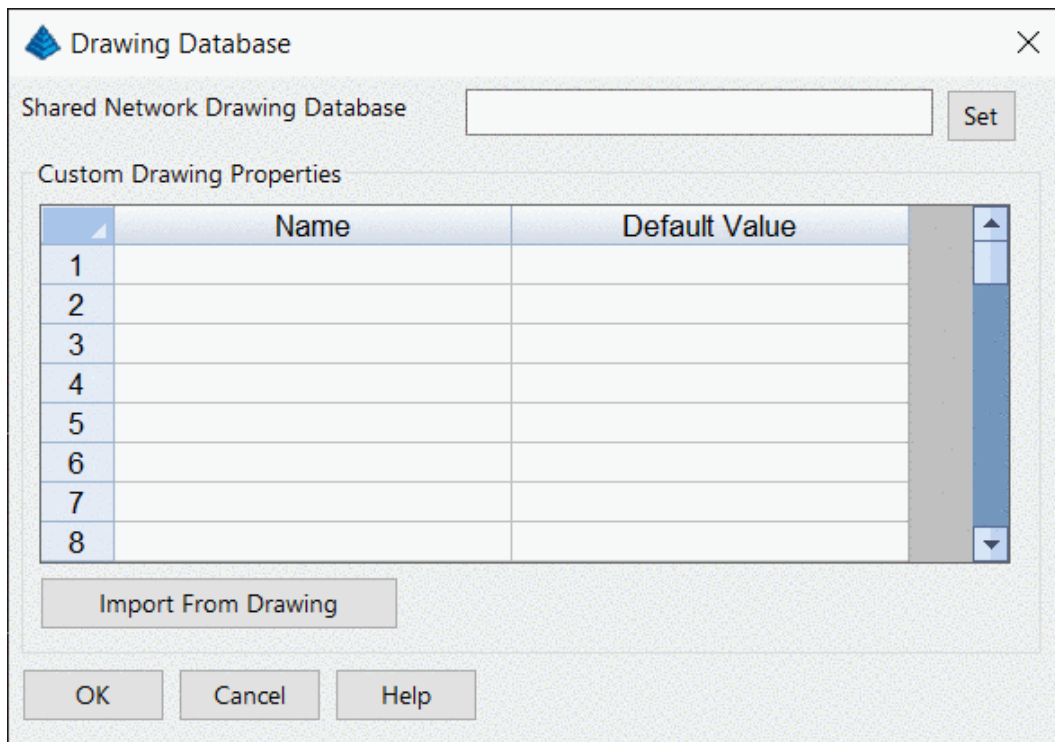
Add: This button permits you to add a file. The Add function prompts for the target folder for the file:

- **Settings Folder:** For Carlson command settings files such as Field-To-Finish code table (.fld), Draw Profile settings (.pfs) or Triangulate and Contour settings (.tri). The target Settings folder is defined in the Project Setup dialog shown above. The default Settings folder is C:\Carlson Projects\Settings.
- **Support Folder:** For CAD support files such as symbol drawings, hatch patterns (.pat) and linetypes (.lin). The target Support Folder is under the Windows AppData folder in this format: \Users\\AppData\Roaming\Carlson Software\\<CAD version>\SUP.
- **User Folder:** For Carlson program files such as Symbol Library (symbols.dta), Quick Keys (quickrun.lsp) and Report Formatter (formats.dta). The target Support Folder is under the Windows AppData folder in this format: \Users\\AppData\Roaming\Carlson Software\\<CAD version>\USER.
- **Data Folder:** For project data files that you want to include in every project. These files can be non-Carlson files such as a Word document.
- **Other Folder:** For files in a user-specified folder.

When adding files, there is an option to assign a comment to the file which is used in reporting.

Remove: This button removes the selected file from the list.

Drawing Database: Controls whether to maintain a database of drawings to use for the Map view in the Open Drawing command. This drawing database stores the drawing file names, latitude/longitude and properties. The drawing database is stored in the Carlson Settings folder and is named DrawingSQLite.db. The Setup button has a dialog with drawing database options. The Shared Network Drawing Database is for a second drawing database besides the drawing database in the Carlson Settings folder. This network drawing database can be used to track the drawing files located on the network drive and shared by your company staff with network drive access. The Drawing Properties are for user-defined properties to store in the drawing database for each drawing. The Import From Drawing buttons fills out the property names from the current drawing which you can edit using the DWGPROPS command.



Project File: Indicate or use **Set** button to specify the Project Settings (.PRJ) file associated with the current drawing. The Project Settings file is a collection of drawing names (e.g. BaseMap.dwg, Roads.dwg, Parcels.dwg, Sewers.dwg, etc) that belong to the same project. This collection of drawings is used by Project Explorer to manage the drawings and data files for the current project and must be specified if the Carlson Data Depot service is to be used. If the current drawing is not associated with a project, then this setting will be blank.

Data Depot Type: The Carlson Data Depot is a document management system to allow tracking of the changing states of files and projects over time and merge the contributions from multiple users providing data integrity, productivity and accountability for the managed products. Carlson Software supports the following *version control systems*:

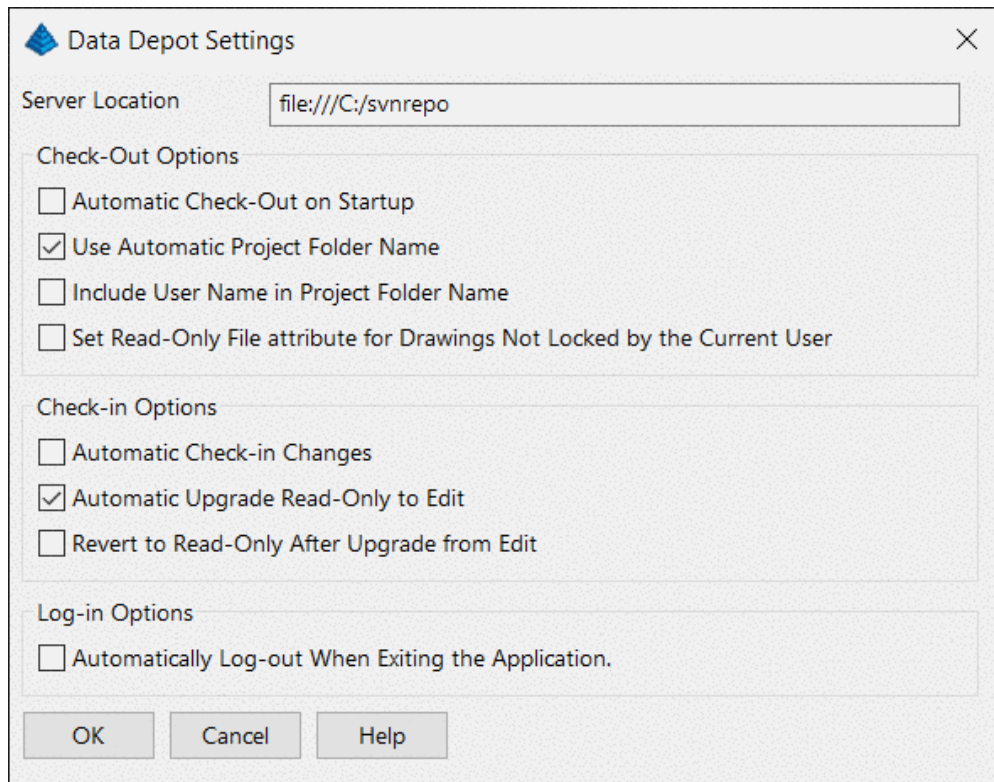
1. Subversion - a free, open-source version control system.
2. ProjectWise - software developed and produced by Bentley Systems.

Setup: Before proceeding, refer to the Carlson Data Depot section for information on how to install and properly prepare your preferred document management system. Once this has been completed, click this button to complete the *Data Depot Settings*.

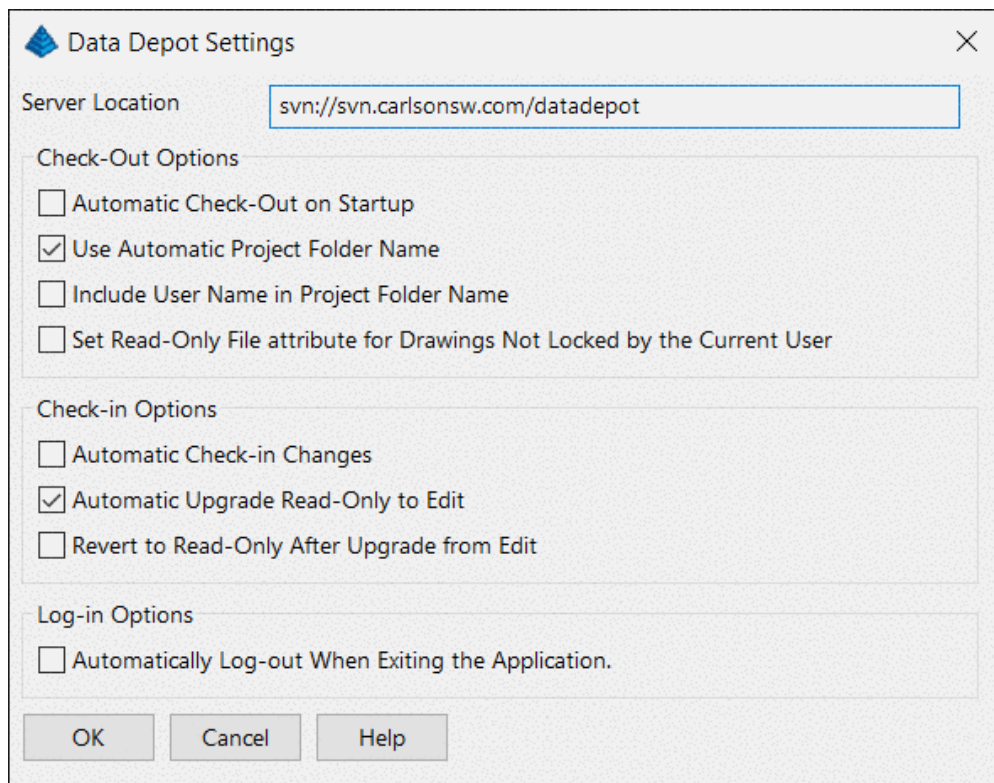
Logout: Log out of the Data Depot server.

Login: Log in to the Data Depot server.

Server Location (Subversion local): Indicate the path for the appropriate location (often a shared server drive) where the file edits and updates are tracked (*e.g.* file:///C:/svnrepo). For Windows users, note the triple-slash convention:

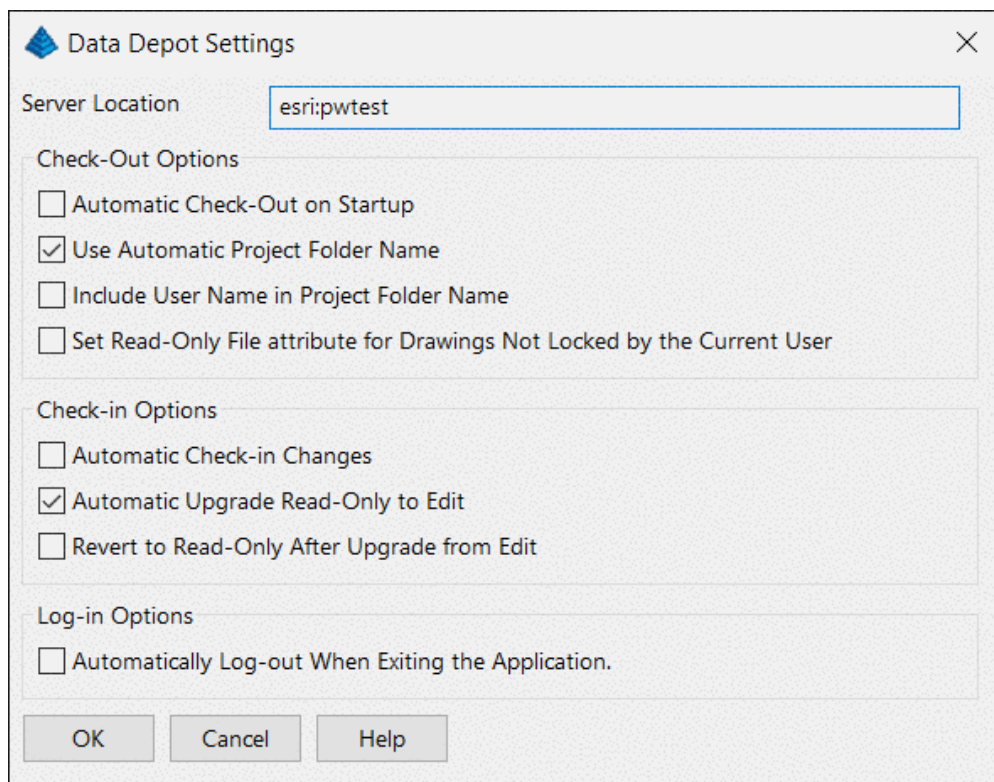


Server Location (Subversion VPN): Indicate the server (proper login name and password may be required) where the file edits and updates are tracked:



For a more extensive write-up on available options, refer to Subversion in Action - Chapter 1. Fundamental Concepts.

Server Location (ProjectWise): Indicate the ProjectWise server and datasource where the file edits and updates are tracked (format is server-name followed by a colon ":", followed by the datasource name, *e.g.* esri:pwtest). See your ProjectWise Administrator for the name of your ProjectWise servers and datasources.



Automatic Check-Out on Startup: This option will check for any updates for the project and associated file on the server while opening the drawing. If there is a new version of a file is found it is automatically updated to match the

current version on the server. User will be prompted if an older version or conflicting versions of file are found.

Use Automatic Project Folder Name: When a project is updated or initially accessed with the `get_prj_from_depot` command, this option will automatically map the project to a folder with the same name as the repository project name under your Startup Project Folder. For example, if your Startup Project Folder is named "C:\Carlson Projects\" and the repository project is named "My Test Project" the `get_prj_from_depot` will use "C:\Carlson Projects\My Test Project\" as the local Current Project Folder.

Include User Name in Project Folder Name: When "User Automatic Project Folder Name" is enabled this option will also be enabled. When a project is updated or initially accessed with the `get_prj_from_depot` command, this option will automatically map the project to a folder with the same name as the repository project name, plus your user name under your Startup Project folder. For example, if your Startup Project Folder is named "C:\Carlson Projects\", your user name is "Engr1" and the repository project is named "My Test Project" the `get_prj_from_depot` will use "C:\Carlson Projects\Engr1\My Test Project" as the local Current Project Folder.

Set Read-Only File Attribute for Drawings Not Locked by the Current User: This option will prevent the user from being able to save drawings for which he/she is not the lock owner. This prevents the user from getting into situations that may cause potential loss of data at check-in time.

Automatic Check-In Changes: If any file under project is updated or edited, it will be automatically checked-in to the repository.

Automatic Upgrade Read-Only to Edit: If a file is checked-in by the current user, the file is upgraded to Edit Mode (locked) for that user for further editing opportunity.

Revert to Read-Only After Upgrade from Edit: If a file is checked-in by the current user, the file is reverted to Read-only Mode (unlocked) so that other users can further edit the file.

Automatically Log-out When Exiting the Application: If a user is logged in to the server upon exiting the application, they will automatically be logged out.

Once the Data Depot has been configured, you can assign a project to the Data Depot via the Project Explorer command.

Pull-down Menu Location: Settings

Keyboard Command: `settmpdir`

Prerequisite: None

Store Project Archive

This command will zip and archive an entire project. The archive contains the current drawing file (.dwg) and all the associated data file such as the surfaces. The data files associated with the current project can be reviewed with the Drawing Explorer command. Besides project data files, images and xref's attached to the dwg are also included. The format of the archive file is a standard .zip file which can be used by WinZip. This file can be sent to someone who can unzip it and use all the same files. The current dwg must have a name before running this command.

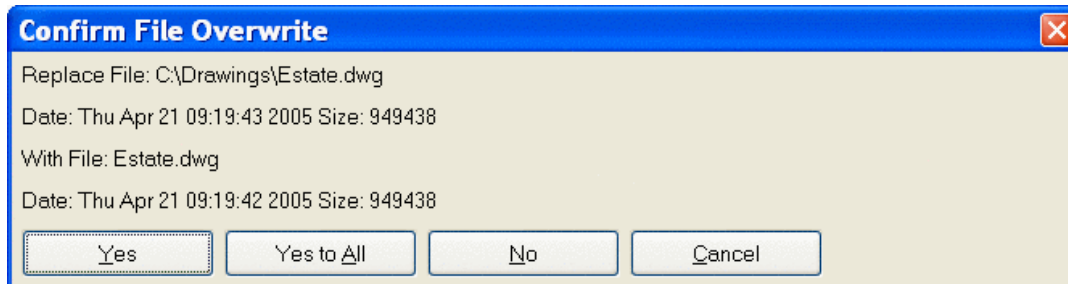
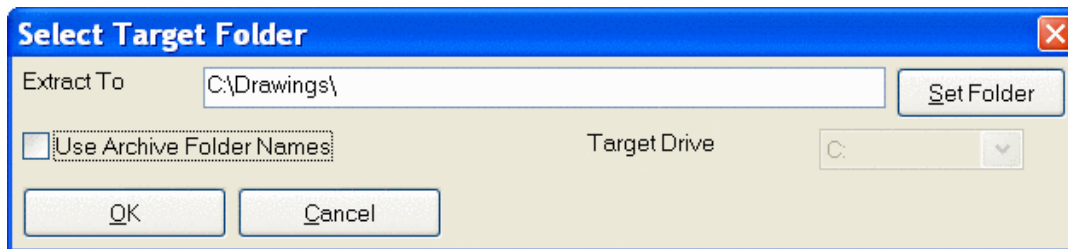
Pull-down Menu Location: Settings > Project

Keyboard Command: `zip_project`

Prerequisite: A named dwg

Extract Project Archive

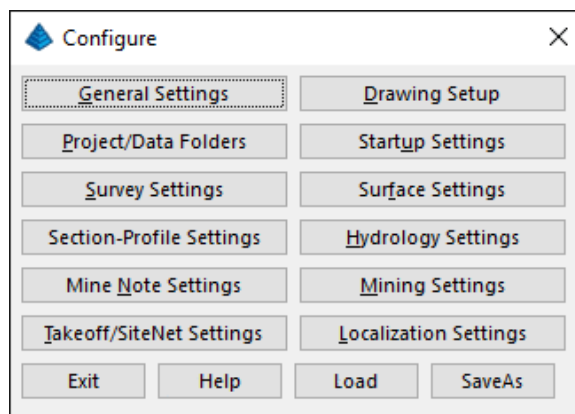
This command will unzip an archive file that has been previously created with the command Store Project Archive. It prompts for the directory to unzip to. If any of the files already exist in the folder it is extracting to, there is a window prompting to overwrite the files.



Pulldown Menu Location: Settings > Project
Keyboard Command: unzip_project
Prerequisite: A project file that has been zipped (ZIP)

Configure Carlson

This command allows you to set up the default settings that are used each time you start a new drawing, or load an existing drawing.



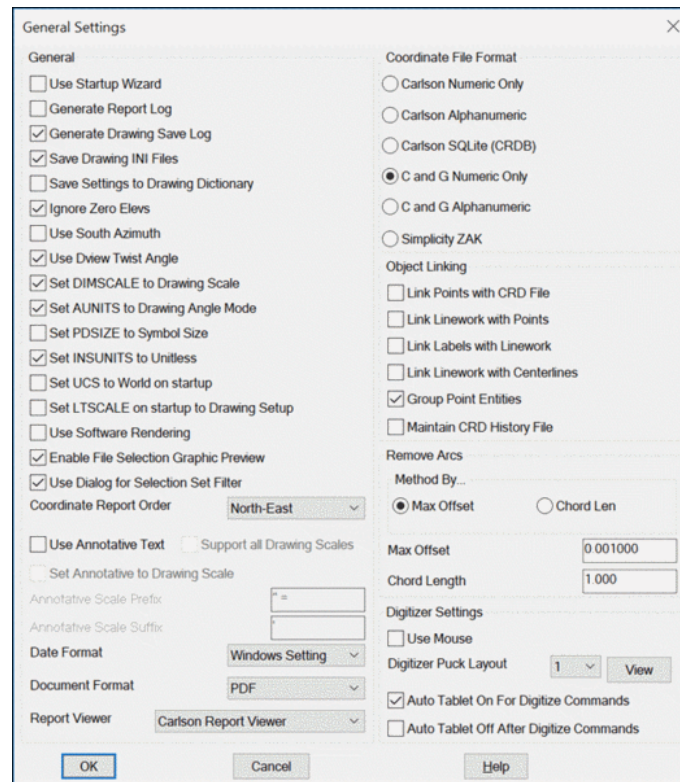
NOTE: When using Carlson products with an "embedded AutoCAD OEM engine" (e.g. *Carlson Survey with Embedded AutoCAD* or *Carlson Takeoff with Embedded AutoCAD*), only a subset of the various configuration commands will be available.

Load: This command permits a previously saved configuration (CFG) file to be loaded into the software and is useful for propagating corporate standards to groups or individuals within an organization.

SaveAs: This command "packages" up all current configuration settings and permits them to be saved to a named configuration (CFG) file that can be shared with users of Carlson Software.

General Settings	Drawing Setup
Project/Data Folders	Startup Settings
Survey Settings	Surface Settings

Table of Contents



Use Startup Wizard: When enabled, a dialog-based "wizard" interface is used for the creation of a new project

Generate Report Log: When enabled, output from several commands will be accumulated in a report buffer. Commands that output to the report log include Inverse, Traverse, Curve Info, *etc.* Also, any report that is displayed in the standard report viewer is also added to the report log. While activated, the report log resides in the lower left corner of the desktop as a minimized title bar that shows how many lines are in the report buffer. To view the report log, pick on the maximize icon on this title bar. You can also view the report log by running the Display Report Log function in the Inquiry menu. The report log can be edited, saved to a file or printed. To quickly turn the report log on and off, you can type REPORT at the command prompt.

Generate Drawing Save Log: This option stores the time stamp and user name in the DWG each time the drawing is saved. This save log can be viewed using the Drawing Save Log command.

Save Drawing INI Files: This option stores the file names of data files used with the current drawing. When enabled, an .INI with the same name as the DWG file will be created to store the data file names. These file names are used for the list of recent files when selecting data files and the Drawing Explorer command manages the list of these data files.

Save Settings To Drawing Dictionary: This option saves the settings for commands within the drawing. Commands always store settings at the program level for recall the next time the commands are run. By storing settings

with the drawing, each drawing will recall the command settings used in the drawing. For example, if you run Triangulate & Contour at an interval of 5 in Drawing_A, then use interval 1 in Drawing_B, then the next time you contour in Drawing_A it will recall interval 5 instead of using the last interval of 1.

Ignore Zero Elevs: This option will ignore any entities with a zero elevation. It is used for many commands, such as Triangulate and Contour or Make Grid File.

Use South Azimuth: Turning on this option will use a South Azimuth instead of a North Azimuth as the basis for 0 degree.

Use Dview Twist Angle: This option makes the program create text entities at an angle to match the current VIEWTWIST system variable so that the labels are horizontal to the screen twist angle. This twist angle can be set with the DVIEW command or the View > Twist Screen commands. In IntelliCAD, this option also controls the USEVIEWTWIST system variable for making MLEADER and MTEXT default rotation to be horizontal to the current screen twist angle.

Set DIMSCALE to Drawing Scale: This will set the dimension scale to match the drawing scale.

Set AUNITS to Drawing Angle Mode: This will set the DWG angular units to match the angle mode established under Drawing Setup.

Set PDSIZE to Symbol Size: This will set the PDSIZE scale to match the symbol size defined in Drawing Setup.

Set INSUNITS to Unitless: This will set the INSUNITS (Insertion Units) CAD system variable to Unitless (INSUNITS=0) when the drawing is opened.

Set MENUBAR on Startup: When enabled, the Carlson Menu associated with the Carlson icon (usually on the PC Desktop) will be loaded. Otherwise, use the Carlson Menus command to select the desired menu.

Set UCS to World on Startup: When enabled, drawings with a User Coordinate System (UCS) other than *World* will have the UCS set to **World**.

Set LTSCALE on Startup to Drawing Setup: When enabled, the *linetype scale* (LTSCALE) variable will be set to the **Horizontal Scale** defined under Drawing Setup.

Use Software Rendering: When enabled, commands that use OpenGL functionality (such as 3D Viewer Window) on computers with older video cards that don't offer extensive hardware acceleration will attempt to render the information with any available random-access memory (RAM).

Use Dialog For Selection Set Filter: For IntelliCAD, this option chooses whether to prompt for selection set filters in a dialog or command prompt at the "Select objects" prompt.

Use Annotative Scale: When enabled, text created by Carlson routines such as Annotation will use scale-able annotative properties. This setting also applies to symbols created by Draw > Insert Symbols. Carlson Points have a separate setting for annotative under the Point Defaults command.

Support All Drawing Scales: When enabled, text placed as annotative entities will make use of all annotation scales currently defined within the drawing.

Set Annotative to Drawing Scale: This option sets the annotative scale to match the Horizontal Scale set in the Drawing Setup command.

Annotative Scale Prefix/Suffix: These strings are used for naming annotative scales when the program creates new annotative scales. For example, when the program creates an annotative scale for a horizontal scale of 50, you can name the annotative scale as 1:50 or 1"=50' depending on these string settings.

Coordinate Report Order: You can choose the traditional North-East format, or reverse these in reports with an East-North format.

Date Format: You can control the display of dates in Carlson reports with this drop-down menu. The default is "*Windows Setting*" which allows you to control it with Windows Control Panel. Several other common formats are

available.

Formatted Document Type: (AutoCAD-based platforms, only) Use this option (for commands such as 3D Viewer Window to establish the type of document produced by the command.

Report Viewer: This option chooses between the Carlson Report Viewer, Windows Notepad and Microsoft Word for the viewer to use for reports that the Carlson commands generate.

Dialog Font: This option chooses which Windows font to use for Carlson dialogs.

AutoCAD Menu: (AutoCAD-based platforms, only) This option chooses which AutoCAD menu to load when picking the AutoCAD menu from the Carlson Menus toolbar or from the Settings > Carlson Menus pull-down menu. When AutoCAD Map is installed, there are different layouts of the Map menu to choose from. When Autodesk LandDesktop is installed, those menus are available.

Object Linking: The Object Linking section contains options for creating additional "intelligence" on Carlson-placed entities:

- **Link Points with CRD File** - (AutoCAD-based platforms, only) When enabled, points placed into the drawing will be given a "reactor" so that any change to the entities such as MOVE or ROTATE will update the coordinates in the CRD file.
- **Link Linework with Points** - (AutoCAD-based platforms, only) Line and polyline entities that are drawn by point number (including those placed via Draw Field-to-Finish such that a positional change to a linked point will automatically update the linework.
- **Link Labels with Linework** - (AutoCAD-based platforms, only) When enabled, direction and/or distance labels assigned to physical linework entities will get updated if the underlying line is changed.
- **Link Linework with Centerlines** - (AutoCAD-based platforms, only) For linework that has been used to create a Carlson Centerline, changes to the linework will update the corresponding centerline (CL) file.
- **Group Point Entities** - When enabled, the three entities of a Carlson point (attribute block, symbol and node) become linked as a single entity. For each point, selecting any one of these entities selects all three entities for the point. **NOTE:** This setting is *not* the same as the "Point Group" functionality as found under the Point Group Manager command.
- **Maintain CRD History File** - When enabled, changes to a point are tracked into a "history" file. These changes can be accessed via the **History** button found under the Edit Point Attributes command.

Coordinate File Format: Carlson can be configured to utilize a variety of coordinate file formats:

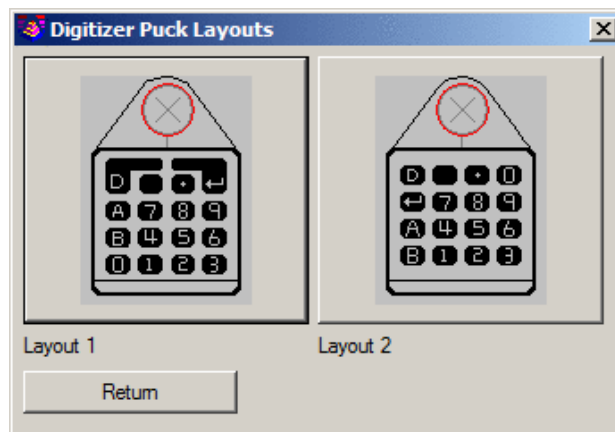
- **Carlson Numeric Only** - Point numbers cannot contain letters and must be in the range from 1 to 99999.
- **Carlson Alphanumeric** - This format allows letters in the point numbers and the point "number" can be up to 9 characters in length. Any combination of letters and numbers is acceptable.
- **Carlson SQLite** - Based on the Structured Query Language (SQL) database format, this CRDB format allows alphanumeric combinations of point number and descriptions up to 255 characters in length.
- **C&G Numeric** - This format of the C&G division supports up to 5 digits, with a 65000 point limit.
- **C&G Alphanumeric** - This format of the C&G division supports up to 10 characters, with no limit to the number of points.
- **Simplicity ZAK:** - This is the Simplicity Systems "Sight Survey" coordinate file format.
- **MS Access Database (LDT)** - This is a Microsoft Access database used by Autodesk Land Desktop. The file is typically named "*points.mdb*" and is typically found in a Land Desktop project \COGO subdirectory. The point identifier limitation is established by the database structure, which has a default of 255 characters.

Remove Arcs: Since 3D polylines do not allow true arcs, the program represents arcs in 3D polylines as a series of short chords. The Remove Arcs settings control the spacing of these arcs:



- **Max Offset** - Sets the maximum difference between the chords and the original arcs as shown in the image above. This method is similar to the Reduce Polyline Vertices command.
- **Chord Len** - Sets the length of the chord segments that replace the original arc.

Digitizer Puck Layout & View: There are two main formats for the digitizer puck. They are numbered 1 and 2. Selecting the View button brings up the window showing the two formats.

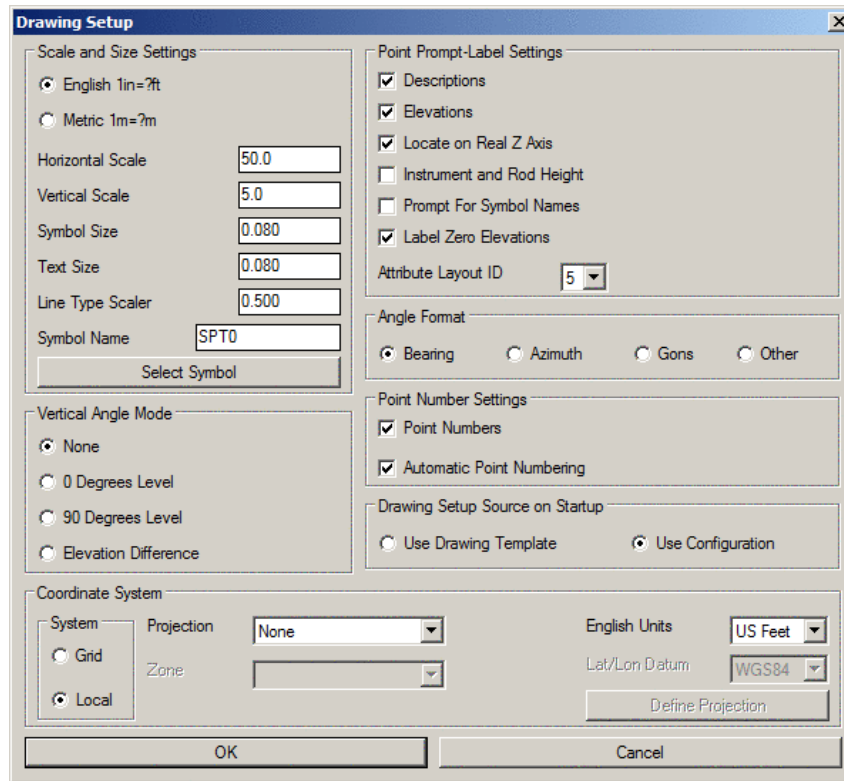


Use Mouse: This option allows you to use the mouse instead of the digitizer puck for the digitize commands.

Auto Tablet On for Digitize Commands: This option will activate the tablet when using the digitize commands.

Auto Tablet Off for Digitize Commands: This option will de-activate the tablet when using the digitize commands.

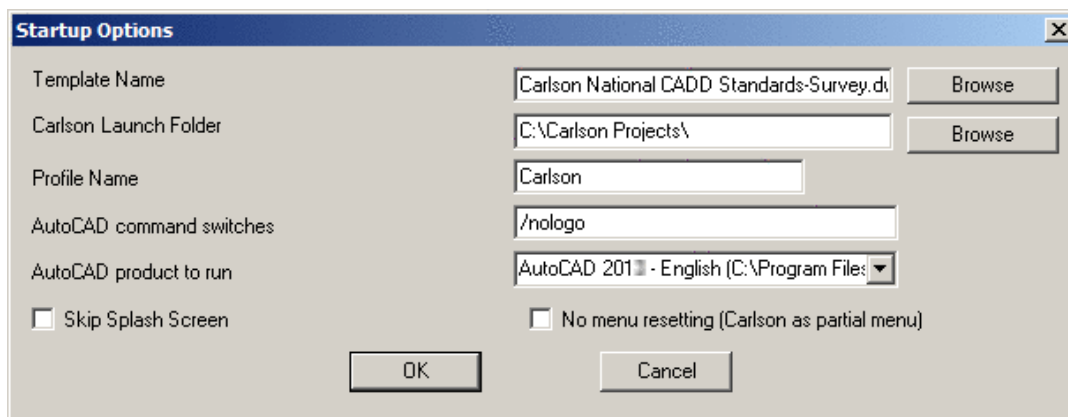
The settings under *Drawing Setup* are used to establish the initial values when creating a new drawing. Most of these settings are identical to those discussed in Drawing Setup. There are a few additional settings. The **Vertical Scale** is used by profile and cross section routines for the grid scale. The Point Prompt-Label Settings, Point Number Settings and Vertical Angle Mode settings are described in the Points > Point Defaults command. The Drawing Setup Source on Startup control the source for the initial settings for a new drawing. The **Use Configuration** option uses the settings from this dialog. The **Use Drawing Template** uses the settings stored in the drawing template (dwt). To store settings to the dwt, open the dwt and run the Drawing Setup command and then save the dwt. The settings are then automatically stored in the dwt.



There is also the ability to maintain two different sets of defaults (English and Metric). You can maintain a separate set of settings for each unit system, especially if you switch back and forth. Also added was support for meters/metres, tons/tonnes and various date representation which can be accessed via the Localization Settings button.

Please refer to the Set Project/Data Folder command for complete information.

These options are used for starting Carlson. Defaults are set here, and will be used at the beginning of each session.



Template Name: This is the drawing template file that will be used when starting a new drawing. The Browse button allows for selecting a new file.

Carlson Launch Folder: This is the folder where Carlson projects would be stored by default. The Browse button allows for selecting a new folder location.

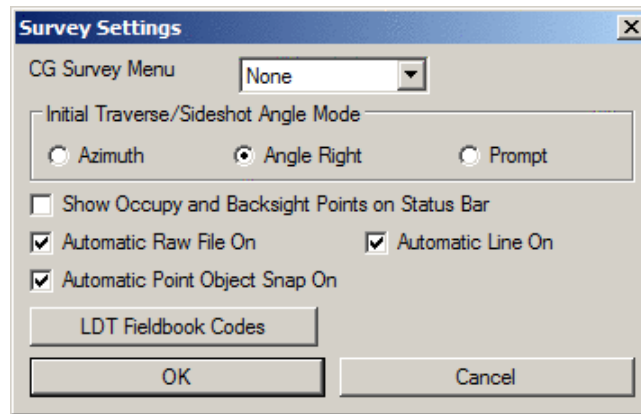
Profile Name: This is the AutoCAD/IntelliCAD Profile that will be used when working in Carlson. If you use a custom profile, be sure that the profile contains the Carlson support folder in the Support File Search path (ie. %appdata%\Carlson Software\Carlson version\CAD version\SUP), and the Carlson main menu must be loaded. If the custom profile doesn't have these requirements, then the program switches to the default Carlson profile.

AutoCAD command switches: This turns off the AutoCAD "splash" screen upon launching the program. The

/nologo takes the splash screen out of the start-up procedure. See AutoCAD documentation for other switches that are available for use.

AutoCAD product to run: (AutoCAD Only) This is the AutoCAD version and flavor (Map or LDT, *etc.*) that Carlson is installed for, and will run with.

No menu resetting: (AutoCAD Only) This controls whether to set the Carlson menu as the main customization file on startup or to keep the current main customization unchanged.



CG Survey Menu: Indicate whether to add-on the C&G Survey pull-down menus to the standard Carlson Survey menus. The Compact mode has all the C&G commands in a single pull-down menu. The Expanded mode has all eight C&G pull-down menus that C&G "stand-alone" used to have.

Initial Traverse/Sideshot Angle Mode: This sets the default angle mode for these COGO commands.

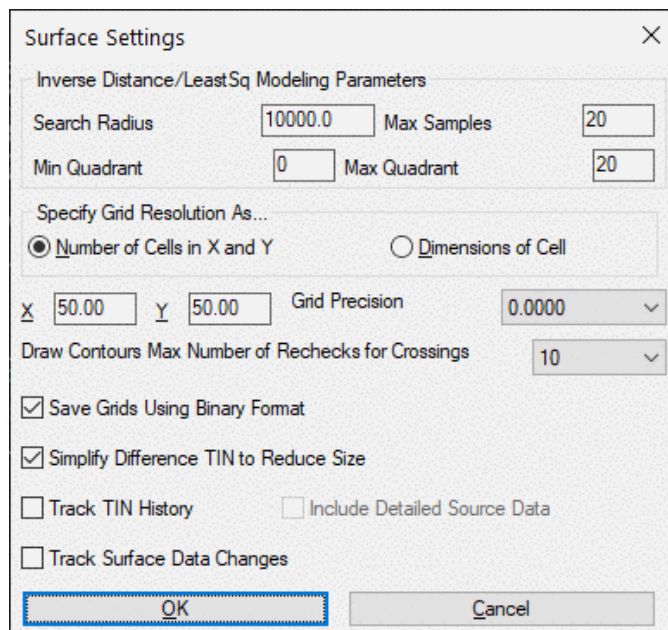
Show Occupy and Backsight Points on Status Bar: This is an option for the COGO Inverse command.

Automatic Raw File On: This is equivalent to toggling on the COGO > Raw File On/Off automatically when the drawing is opened.

Automatic Line On: This is equivalent to toggling on the COGO > Line On/Off automatically when the drawing is opened.

Automatic Point Object Snap On: This is equivalent to toggling on the Settings > Point Object Snap On/Off automatically when the drawing is opened.

Automatic Compare DWG points with Coordinate File on Startup: This option runs the Coordinate File Utilities > Compare Points routine when the DWG file is opened to report any differences between the point entities in the drawing and point coordinates in the coordinate file.



Most of the Surface/Triangulate & Contour commands will remember the settings and parameters used from drawing to drawing. There are some in this screen that will be used for gridding and modeling.

Inverse Distance/LeastSquares Modeling Parameters: The modeling methods of Inverse Distance and Least Squares are similar ways to create a grid from datapoints or drillholes. It is not recommended to use these methods for gridding contour or breaklines. Triangulation is better for that. These methods need a search radius defined. Anything past this distance from one data point to the next will be ignored for influence. The Max Samples are the number of data points that will be used to influence each data point. The area is broken into 4 quadrants. The Min and Max Quadrant are the numbers of data points that will be used in each quadrant.

Specify Grid Resolution As: There are two ways to create a grid file. Once the boundary has been selected, the cells need to be determined. Number of Cells in X and Y will divide the boundary up into the specified number of cells. These will then be odd shaped rectangles, with the size calculated by the boundary dimensions and the number of cells. The Dimensions of Cells is the more commonly used method. This will allow for a set cell size for the X and Y directions. Most of the time the grid cells should be square, where you set the size.

Grid Precision: This is the number of decimals that are stored in the grid file.

Draw Contours Max Number of Rechecks for Crossings: Routines that generate contours check for any crossings that can occur from smoothing or reduction options. When a crossing is found, the smoothing or reduction factors are reduced and then the contours are rechecked in case that adjustment causes a new crossing. This option can be used to decrease the number of rechecks in case your dataset is large and you don't want to take the time for these checks.

Save Grids Using Binary Format: This options chooses between saving .grd files as either text or binary files. This setting applies to all routines that save grid files. The advantage of the text format is the ability to view the grids using any text editor like Notepad and the ability for non-Carlson programs to easily read the grids. The advantage of the binary format is the speed of saving and loading the grids which is several times faster than the text format. Only Carlson 2015 and later versions can read the binary format. When the range of min/max grid values is small enough relative to the grid precision, the program will automatically switch to an indexed binary format which uses half the file size and loads twice as fast.

Simplify Difference TIN to Reduce Size: When difference TIN is calculated, the intersection of individual triangles is calculated. When two source TINs are large and somewhat similar, the resultant TIN can have exceedingly large number of triangles. This option seeks to limit the growth of intersection TIN by keeping the points on the breaklines and in the areas major surface changes (based on the normals) while eliminating extraneous points in flat

areas. While this results in some approximation of the difference TIN, the resulting surface has significantly fewer points and minor slivers.

Track TIN History: When this option is enabled, the information about creation of the TIN and editing operations is stored in the TIN. This allows to review the history, modify the steps and rebuild the TIN in Surface Manager -> History utility.

Include Detailed Source Data: When TIN history is enabled, this option adds additional data about entity data, allowing user to recreate the surface without original drawing or when the source entity is no longer in the drawing. Without this option, only handles (IDs) of entities are stored.

Track Surface Data Changes: When the TIN is made and added as a surface to the Surface Manager, the source selection set used to create the TIN is stored in the drawing. When this option is enabled, the changes to the source entities will be tracked and TIN will be updated accordingly, recreated with same options as originally.

Eye Height and **Object Height** are used for calculating sight distance on vertical curves.

Vertical Curve Drawing Resolution applies to Draw Profile for the segment length for drawing vertical curves.

Use Crest Sight Distance Formula for Sag:When this option is on, the Crest Vertical Curve formula is used to calculate the sight distance for sag vertical curves. Otherwise the the Sag Vertical Curve formula is used.

Parabolic Template Grade Subdivisions applies to Process Road Design and Road Network for how many 3D polylines to draw for grades defined as parabolic in the template.

Station Type controls the format of station labels. For the NO.0 type, there is a setting for the label prefix, and the Use Partial Labels for Intermediate Stations option will skip the label prefix for odd stations.

Section-Profile Settings

Eye Height: 3.500 Object Height: 0.500

Vertical Curve Drawing Resolution: 10.000

Use Crest Sight Distance Formula for Sag

Parabolic Template Grade Subdivisions: 10

Station Type:

1+00 1+000 100 NO.0

NO. Station Prefix: NO. Use Partial Labels for Intermediate Stations

OK Cancel

Stage-Storage File Format: Indicate the format of the Stage-Storage File to be used in Carlson Hydrology. Carlson method should be used as default. SEDCAD 3 and 4 is a comprehensive hydrology and sedimentology package by others that Carlson can output data files to be used.

Rainfall Amount Method: Rainmap TP-40/TP-47, NOAA Atlas 14 and Rainfall Lookup methods are available. The program defaults to the Prompt option and the rainfall amount program will show all three methods when Prompt is on. When users select a method and check the "Skip Method Prompt Next Time", the method is set to the default rainfall amount method.

Rainmap TP-40/TP-47: Get the rainfall amount on the rainfall map.

NOAA Atlas 14: Rainfall amount is download automatically from NOAA website by specifying the location.

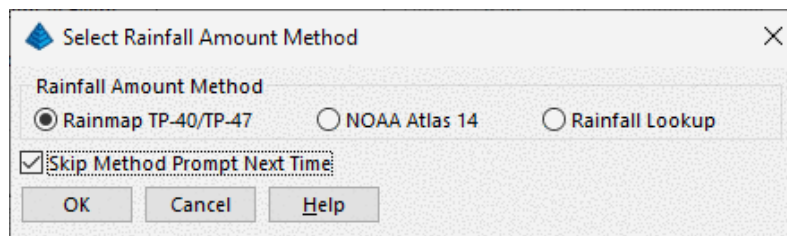
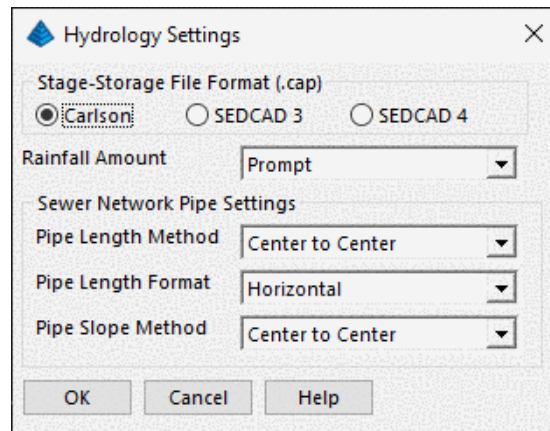
Rainfall Lookup: This is a rainfall lookup table created by users. It can be accessed for all the hydrology functions to get the rainfall amount.

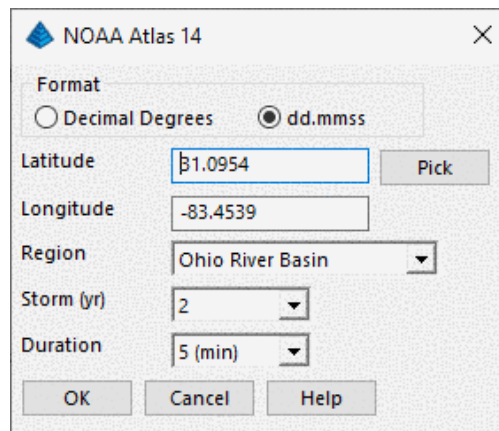
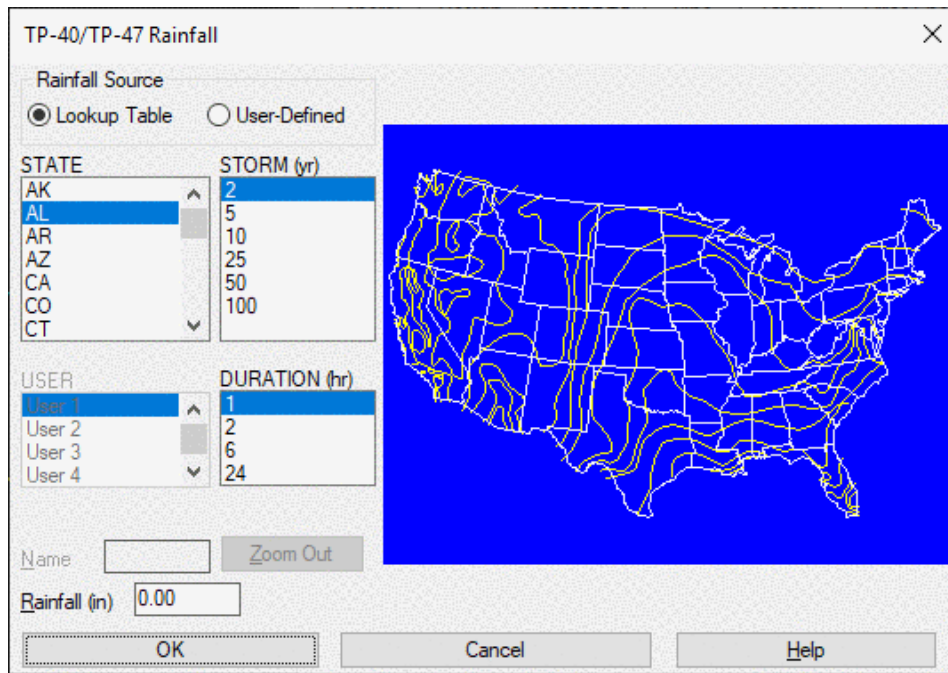
Sewer Network Pipe Settings: Specify the pipe settings based on jurisdictional requirements.

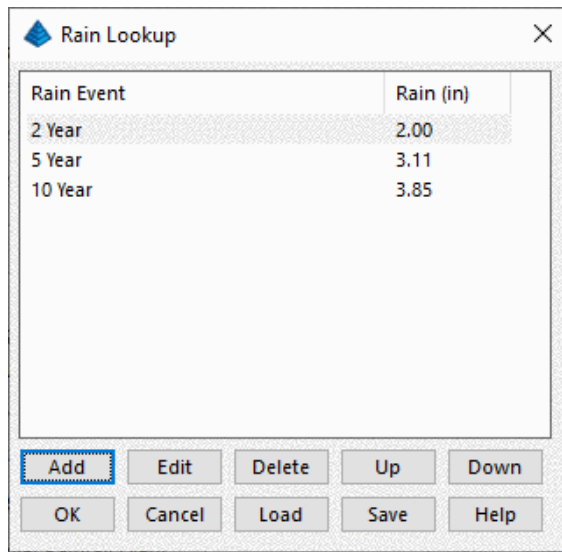
Pipe Length Method can be set for Center to Center or Actual Pipe Length.

Pipe Length Format can be set to Horizontal or Slope Length.

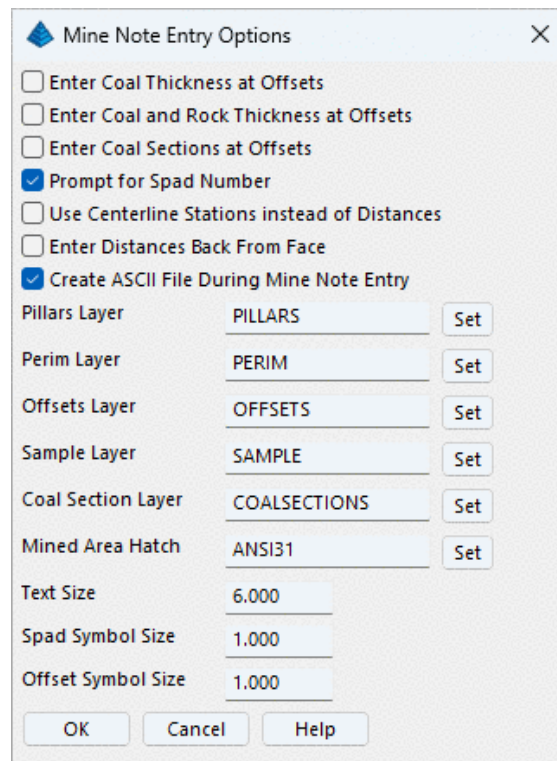
Pipe Slope Format can be Center to Center or Actual Pipe Length.



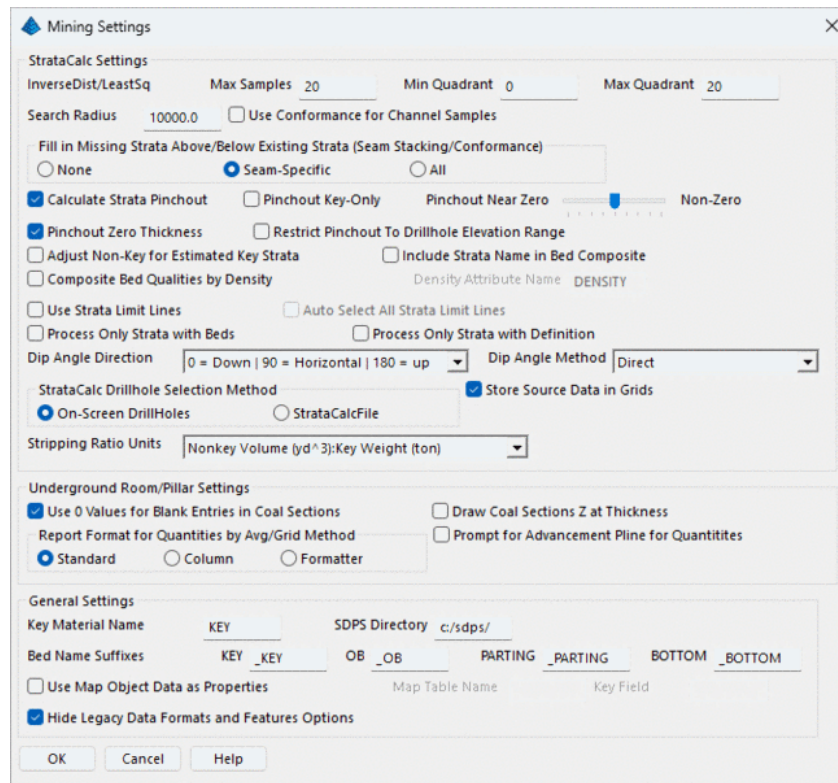




These options will toggle various prompts when entering Mine Notes in the Underground Mining module. You can also set the layers for various linework needed for geologic modeling and mine planning. These options are further described in the documentation of the the Mine Note Defaults command.



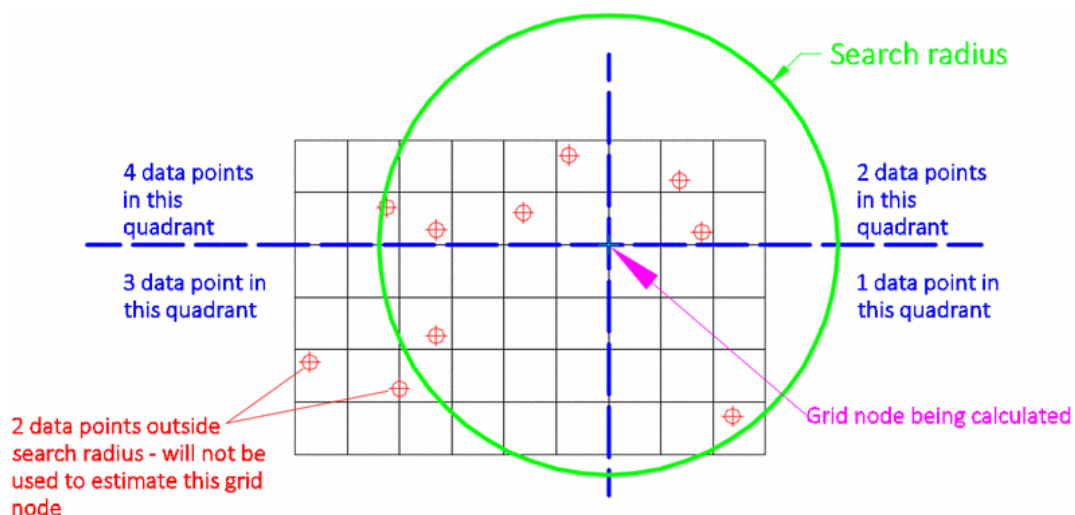
This is the configuration screen for default settings used with the Mining Modules. Each item is detailed below.



InverseDist/LeastSq Max Samples: This value sets the maximum number of data points to use to estimate the value at a grid node. If this value is set to 20, only the 20 data points nearest the grid node being calculated will be used. This option only affects the Inverse Distance and Linear Least Squares modeling methods.

Min/Max Quadrant: When the program calculates the value of a grid node, the area around the grid node is divided into four quadrants, as shown below by the blue dashed line. These Min/Max Quadrant values set the minimum/maximum number of data points to use from each quadrant. If the Min Quadrant value is set to 5, a value will only be assigned to the grid node if at least 5 data points are found within all four quadrants. If the Max Quadrant value is set to 20, only the nearest 20 data points in each quadrant will be used for estimation.

Search Radius: This value sets the maximum search radius for data points. For example, if this value is set to 10,000, any data points beyond 10,000 ft/meters from a given grid node will not be used to estimate the value of that grid node. This concept is shown in the below image.

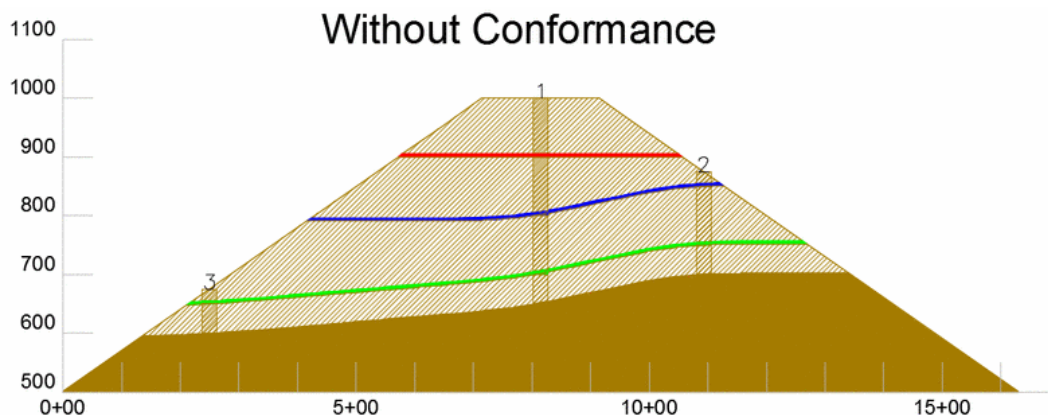


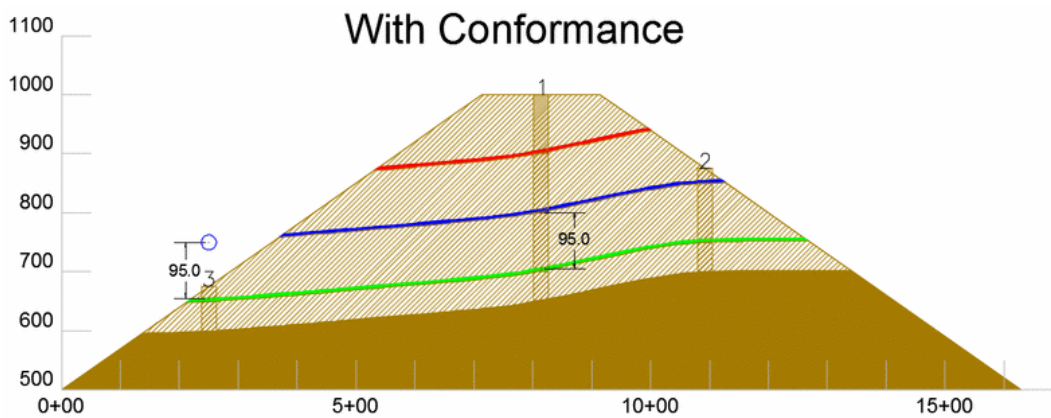
Fill in Missing Strata Above/Below Existing Strata (Seam Stacking/Conformance): This option determines how the program handles drillholes with missing strata. An example of a data set modeled with and without the conformance option is shown below. Notice that when conformance is applied, the red and blue strata will more closely mimic the green strata, even though the red and blue strata do not occur in all three drillholes. The program does this by artificially adding data points to the drillhole with missing strata.

In the below example, consider drillhole #3. This drillhole does not contain the blue strata. To make the blue strata conform to the green strata, the program will search for the nearest drillhole that contains the blue strata (in this case, Drillhole #1). The program will then measure the elevation difference between the blue and green strata and apply that same elevation difference in drillhole #3. In this example, the bottom of the blue strata and the top of the green strata in Drillhole #1 are 800' and 705', respectively (a difference of 95'). The top elevation of the green strata in Drillhole #3 is 655'. So, when the program models the blue strata, an artificial elevation of 750' (that is, 655' + 95') will be used in Drillhole #3. In this case, the green strata is considered to be the "marker" and the blue strata is considered to be the "target".

In drillhole #2, the red strata is not present. In this case, either the blue or the green strata could serve as the "marker" strata. When the "All" option is used, the program will automatically select one of these strata to be the "marker". The program will set the "marker" strata as the strata closest to the "target" strata. In this example, the red strata is closer to the blue strata, so the blue strata will be used as the "marker". If the "Seam-Specific" option is used, you will have to manually define which strata are "markers" and which strata are "targets".

Although this example shows the result of seam stacking/conformance applied to an area with sufficient drilling depth in an area with varying topography, the same effect can be applied to drillholes that are not simply drilled deep enough to intersect all strata.





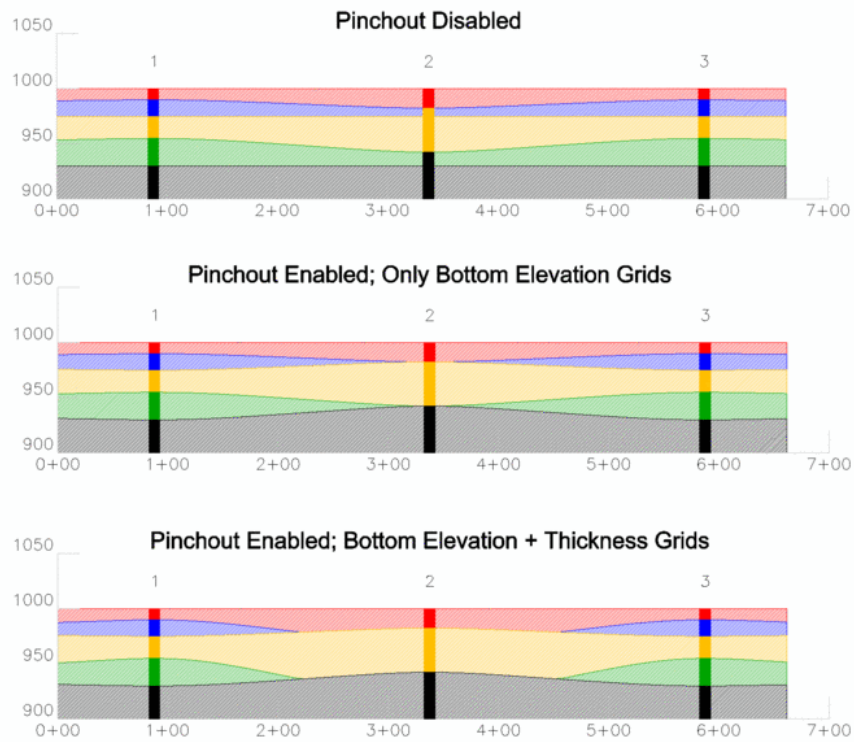
None: This option will not apply seam stacking/conformance. In this scenario, each strata layer will be modeled independently of other strata layers. Only strata data that is actually included in the drillhole will be used to make the strata grid file.

All: This option will apply conformance in any way possible. This can be a useful option when the structure of all strata are similar. However this option does not give you the option to specify which strata layers are "markers" and which strata are "targets".

Seam Specific: This option also applies conformance, but requires that you specify which strata are "markers" and which strata are "targets". The strata are tagged as "markers" and "targets" in the Define Strata/Bed command. It is important to note that when using this option, you can define the "Marker Level", which sets the priority of the marker strata (this is an optional field, but it can have a significant impact on the model when multiple marker strata are available for use).

Use Conformance for Channel Samples: This option controls whether to use channel samples as source data points for conformance of other strata.

Calculate Strata Pinchout: This option determines if the thickness of a seam is pinched out when it does not occur in a drillhole, as shown in the below image.



When pinchout is disabled, as shown in the first cross section in the above image, the middle drillhole is completely ignored when making the grids for the blue and green strata. This results in the blue and green strata suggesting a significant thickness at the location of drillhole 2. In the second cross section, pinchout has been enabled, but the results are misleading because no thickness grids were incorporated into the model. In the third cross section, pinchout has been enabled and the thickness grids have been incorporated into the model to accurately model the pinchout.

It is important to note that elevation grids will not be automatically modified to show the pinchout, although the elevation grids may in fact be adjusted somewhat. It is therefore recommended to only enable this option when making thickness grids to avoid misleading results. When the program creates a thickness grid with this option enabled, the missing strata will be modeled as having a negative thickness at the drillhole without the strata. The thickness value will be equal but opposite to the thickness of the nearest thickness data point for that strata. In the above example, the blue strata has a thickness of 10' in drillhole 1, so the program will use a value of -10' in drillhole 2 when making the blue thickness grid. By incorporating a negative thickness value, the thickness will approach a value of zero approximately halfway between the drillholes 1 and 2. The Pinchout Slider bar to the right of this toggle in the dialog controls where the strata will actually pinchout. Before the thickness grid is written, it will be modified so that all negative values are reset to zero. This results in a thickness grid that begins to decrease in value as it nears the drillhole with the missing strata, but never shows a negative thickness.

Since this option is only intended to be used when making thickness grids, you will need to create modified elevation grids before adding them to the Geologic Model. This can be done quite easily with Grid File Utilities. In the above image, notice that the blue and green strata layers only pinch out properly when the elevation grids have been modified. In this case, the top of the blue strata was created by adding the bottom elevation grid of the blue strata to the thickness grid of the blue strata. Notice that in the second diagram in which only bottom elevation grids were used, the blue strata does not pinchout between the drillholes.

Finally, it is important to note that pinchout will only affect thickness grids when two "sandwiching" strata are detected above and below the missing strata. In the above image, the blue strata is "sandwiched" between the red

and yellow strata. If drillhole 2 did not contain the red strata, then the blue strata would not pinchout. This means that the top and bottom strata in any given model will not be able to pinchout.

Pinchout Zero Thickness: This option determines if a strata with a thickness of zero will be modeled as a pinchout. Pinchout is normally only applied for strata layers that are absent in a hole.

Pinchout Key Only: This option will only apply the pinchout to the Key strata while the Non-Key strata will be modeled as if the pinchout were not applied.

Adjust Non-Key for Estimated Key Strata: This setting controls whether to adjust non-key strata to fit in key strata for pinchout or conformance.

Restrict Pinchout to Drillhole Elevation Range: This option controls where the seam will pinchout. If there is a shallow hole, and a seam is running beneath it, this setting will pinchout the seam if it is off. If it is on, then the seams will only pinchout if they pass through the elevation range of the drillhole. This is useful if it is desired to pinch out a seam that passes above or below the elevation range of the drillholes. For most applications, it is recommended to leave this option disabled.

Include Strata Name in Bed Composite: This option will add the strata name to the bed name when running the bed compositing commands, such as Split Bed by Parameters.

Composite Bed Qualities by Density: When creating grids of bed qualities from drillholes with multiple samples of the bed quality, the quality will be composited to a single data point before the grid file is made. By default, the quality will be averaged by the thickness of the sample. When this option is enabled, the qualities will instead be averaged by weight (thickness * density). The **Density Attribute Name** sets the name of the density attribute to be used for weighting. Density is always expressed as lbs/ft³ or kg/m³.

Use Strata Limit Lines: This option determines if Strata Limit Polylines will be used for modeling. If this option is disabled, you will not be prompted to select Strata Limit Lines when modeling.

Auto Select All Strata Limit Lines: This option will automatically select all Strata Limit Polylines when modeling. You will not be prompted to select the lines when this option is enabled.

Process Only Strata with Beds: This option will ignore all strata layers without a bed name when modeling. This can be useful when overburden/interburden layers are not marked with unique strata names and have not been tagged with a bed name.

Process Only Strata with Definition: This option will ignore all strata that are not listed in the current Strata Definition file when modeling. However, strata layers that are not included in the Strata Definition file will still be used to calculate conformance and pinchout.

Dip Angle Direction: This option determines how drillhole dip values are interpreted. The default option of $0 = \text{Down} - 90 = \text{Horizontal} - 180 = \text{Up}$ will treat dip values of 0 degrees as vertical in the downward direction, dip values of 90 degrees in the horizontal plane, and dip values of 180 degrees as vertical in the upward direction. The other options may be used to accommodate other dip value schemes.

Dip Angle Method: The Direct method simply applies the dip angle of the sample to the depth of the sample. The Minimum Curvature method looks at two adjacent samples at a time and calculates assuming the samples lie on a circular arc. The Min Curvature From Bottom method considers the sample point to be at the bottom of the sample and pairs this point with the dip from the previous sample. The Min Curvature From Top method considers the

sample points to be at the top of the sample and pairs this point with the dip from the next sample.

Store Source Data in Grids: This option will include the source data used to make strata grid files within the grid itself. This includes the X-Y coordinates of the data point, the value used for modeling, and the type of data source (drillhole, channel sample, etc.). This can be useful for understanding how the grid file was generated when the original source data is not available.

StrataCalc Drillhole Selection Method: This option determines how the program will prompt for geologic data when modeling. The *On-Screen Drillholes* option will prompt you to select drillholes, channel samples, etc. when making the geologic model. The *StrataCalc File* option will instead prompt you for a .stc file, which can be saved from the StrataCalc Data Sheet command. The .stc file can be useful when you need to be certain that the same information is being used to create model between iterations.

Stripping Ratio Units: This option sets the units to be used when reporting Stripping Ratio. Some commands, however, will not respect this option, and will instead report Stripping Ratio using the default units of (yd³ waste)/(ton key) or (m³ waste)/(tonne key). Commands that are hard-wired to use the default stripping ratio units are Dragline Pits, Vertical Pit Quantities, and Pit Scheduler. Also, when using Surface Mine Reserves, when the "Type of Strip Ratio Contours/Grid" is set to Accumulative, only the default stripping ratio units will be used for the contour/grid output (be cautious not to confuse the contour/grid output option with the Accumulative Stripping Ratio that is included in the report - all stripping ratio values in the Surface Mine Reserves report will respect the stripping ratio units selected in Carlson Configure).

Underground Room/Pillar Settings: These options apply to the series of commands for placing coal sections (used for calculating end of month volumes).

Use 0 Values for Blank Entries in Coal Sections: This option will replace treat blank values in coal sections as 0 rather than considering it to be null.

Draw Coal Sections Z at Thickness: This option will draw coal section symbols at the Z value of the actual thickness. For example, a coal section with a thickness of 5 feet will be drawn at an elevation of 5. This is useful for contouring or gridding the coal sections with standard commands from the Civil module.

Prompt for Advancement Pline for Quantities: This option will prompt you to select an additional advancement polyline when running the Quantities by Average / Grid / Centerline commands. This polyline represents the direction of mining in a particular area. The length of the polyline will be included in the quantity report.

Report Format for Quantities by Avg/Grid Methods: This option determines the default report format for the Quantities by Average / Grid / Centerline commands. However, each of these commands will also let you set the report format when the command is executed. The *Standard* option will use a simple text editor for the report. The *Columns* option will still use a standard text editor, but with the values aligned into columns. The *Formatter* option will send the report to the Report Formatter for user-defined formats.

General Settings:

Key Material Name: This is the name of the Key material you are mining. This name will be used in several reports to identify Key material, such as the outputs from Surface Mine Reserves and Surface Production Timing. If you have more than one type of key material, you may want to give this a more generic name such as "Key Material" to avoid confusion in the reports.

Bed Name Suffixes: KEY, OB, PARTING, BOTTOM: These suffixes are used to identify the four portions of a bed. An example is shown in the below image. When Bed names are applied to a group of strata layers, all strata layers with matching bed names are treated as a single group. This allows the program to properly correlate groups between drillholes even when the number of samples for each bed is not consistent between drillholes. Although

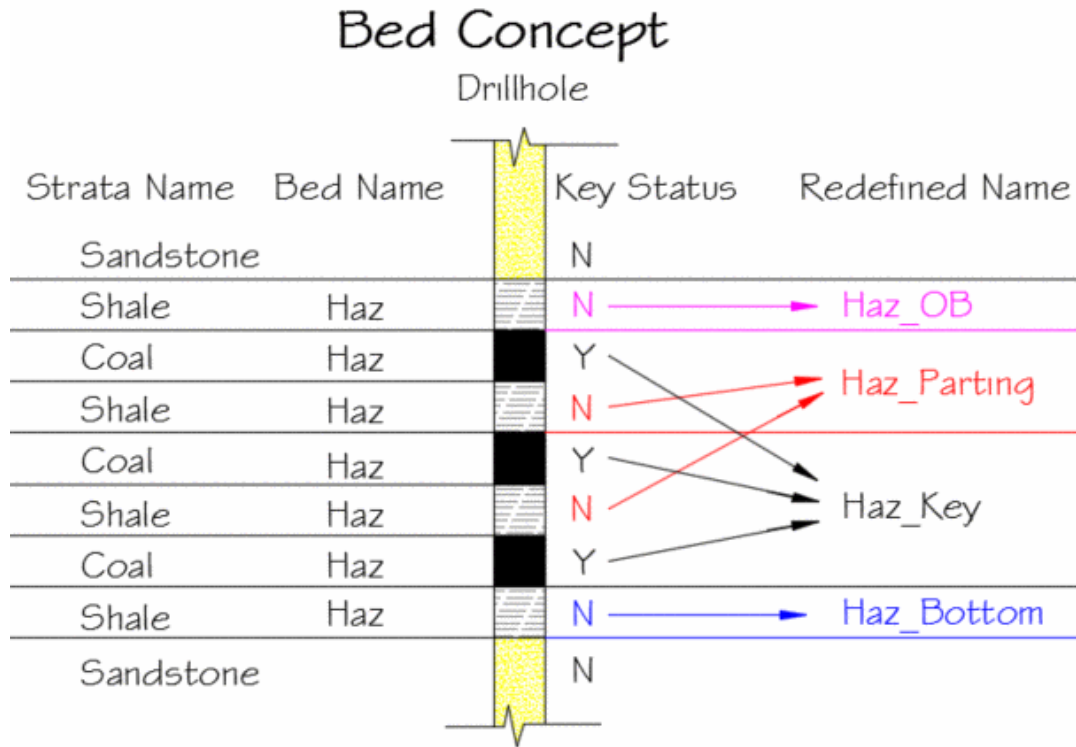
the strata will be treated as one group, there are four portions of the bed available for modeling. Note that not all four portions of the bed may not be present in a data set.

OB: This is the non-key portion of the bed above the first occurrence of Key strata.

Parting: This is the Non-Key portion of the bed bound between Key strata.

Key: This is the Key portion of the bed.

Bottom: This is the non-key portion of the bed below the last occurrence of Key strata.

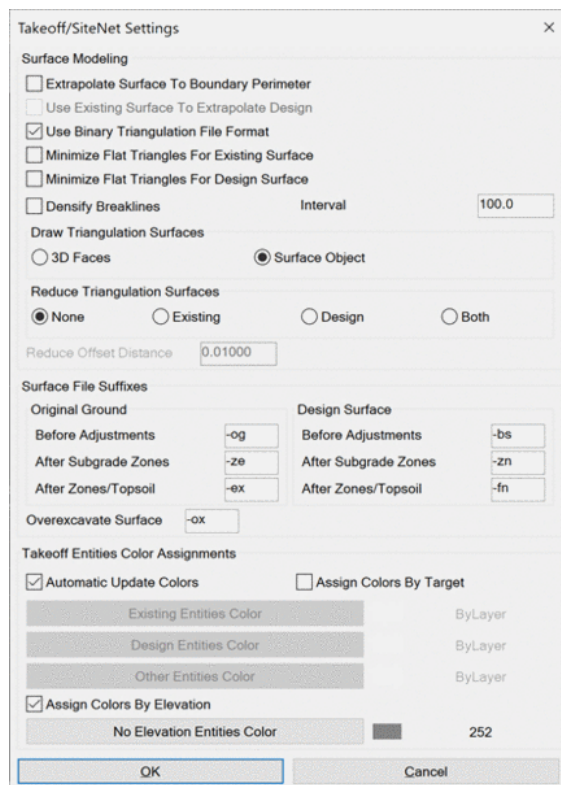


SDPS Directory: This field sets the installation directory for the Subsidence Deformation Prediction System (SDPS) program. The SDPS commands are found under the Subsidence Pulldown Menu of the Underground Mining Module.

Use Map Object Data as Properties: This option will use the AutoCAD Map data to set the Property and Owner names for reserve estimates. When this option is disabled, the program will use the Owner and Property names assigned to polylines from the Assign Property Names command.

Hide Legacy Data Formats and Features Options: When enabled, older settings and commands considered to be outdated will be hidden. The goal with this option is to simplify interfaces by hiding options users are unlikely to use. If you find that this option hides an option/feature you are still using, please contact us at support@carlsonsw.com.

These options are used for the Construction module and SiteNet commands in the Civil module.



Extrapolate Surface To Boundary Perimeter: When this is checked ON surfaces are extended and volumes are calculated out to your boundary perimeter. When this is checked OFF surfaces and calculations end at the extents of your design data.

Use Existing Surface To Extrapolate Design: When this is checked ON surfaces and volumes are calculated to the extents of your existing data.

Use Binary Triangulation File Format: This option sets the format for the surface model files as either binary or ASCII. The binary format has a .tin file name extension and loads about twice as fast and has about 50% less file size than ASCII. The ASCII format has a .flt extension and is the legacy format used by other Carlson products and Softdesk.

Minimize Flat Triangles: This option reduces the occurrence of "flat" (or more precisely, horizontal) triangles. Flat triangles often occur when creating surface models from contour data. The Minimize Flat Triangle option will swap triangulation edges when possible to switch flat triangles to sloped triangles.

Densify Breaklines: This option automatically add vertices on breakline segments for triangulation at the specified Interval.

Draw Triangulation Surfaces: This option sets how the Takeoff surfaces are drawn in the drawing. The 3D Faces option creates 3D Face entities for each triangle in the surface. The Surface Object option creates a single Carlson Surface Object for the surface which is more efficient on memory and drawing size.

Reduce Triangulation Surfaces: This causes edges within the selected surface TIN mesh to be collapsed to reduce the number of triangles, edges, and points within the mesh while having a minimal impact on the overall shape of the mesh.

Reduce Offset Distance: This setting is used by the Reduce Triangulation Surfaces command to set the reduction tolerance. Specify the maximum average distance that any point can be moved outside of the plane of any triangle that connects to that point. Values might range from .01 to .1 for most purposes.

Surface File Suffixes: These settings allow you to change the file names for the surfaces generated by the program:

- **-og** - This is the default name for the original ground surface before adjustments.
- **-ze** - This is the default name for the original ground surface after subgrade zone adjustments.
- **-ex** - This is the default name for the original ground surface after subgrade zone and topsoil adjustments.
- **-fg** - This is the default name for the initial design surface before adjustments.
- **-zn** - This is the default name for the design surface after subgrade zone adjustments.
- **-sg** - This is the default name for the design surface after subgrade zone and topsoil adjustments.
- **-ox** - This is the default name for the over-excavate surface after subgrade zone and topsoil adjustments.

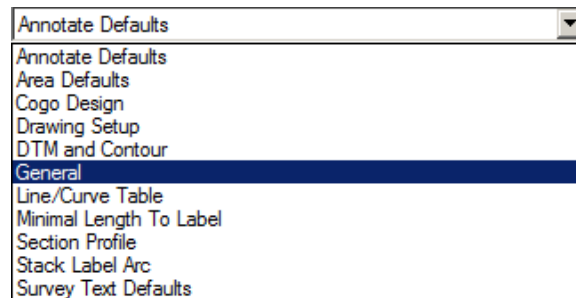
Automatic Update Colors: This refreshes colors in your drawing as they change (*i.e.* elevating entities, setting layers for different Targets, *etc*). If your drawing is very large and is slow to automatically refresh you may want to toggle this off and use the Update Colors For Set Elevations command under View when you want/need to see the color changes.

Assign Colors By Target: This option allows you to set the Existing, Design, and Other layers to any color you define.

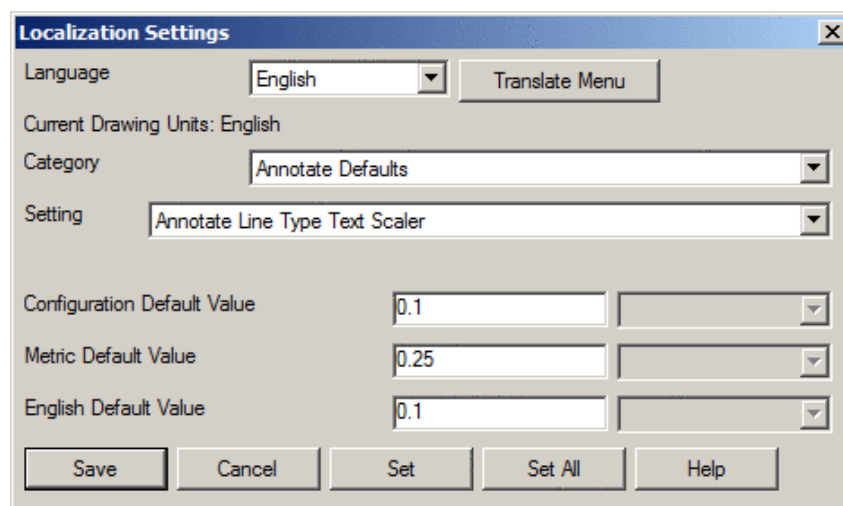
Assign Colors By Elevation: This option allows you to set the color for entities needing elevations.

No Elevation Entities Color: Indicate the color entities with no elevation ($Z=0$) should be assigned to when their layer is classified as "Original" or "Design".

There are literally hundreds of default settings that can be set with this dialog. The categories that can be selected from are:



The Settings for each Category will display all of the items that can be setup for default values. The Default value is set in the Configuration Default Value box. The corresponding Metric or English default values are set here, allowing for easy switching between the two systems.



Pulldown Menu Location(s): Settings

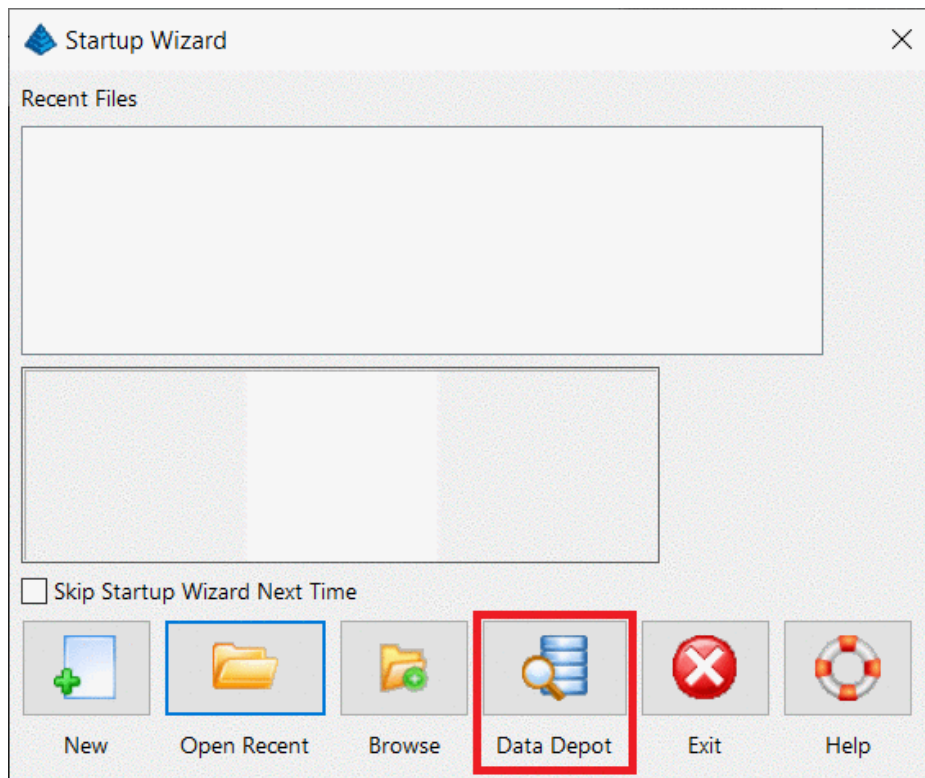
Keyboard Command: config_scad

Prerequisite: None

Data Depot

The Carlson Data Depot is a document management system to allow tracking of the changing states of files and projects over time and merge the contributions from multiple users providing data integrity, productivity and accountability for the managed products. Also, having a central project repository helps with data back-up eliminating the fear of losing data due to local drive failures. The Carlson Data Depot protects against productivity loss due to re-implemented work, not only avoiding losses of data, but also making each user's work readily available to other users on the project. The general process for implementing your preferred document management system is:

1. Install and initialize one or more of the version control software products listed below.
2. Instruct Carlson Software to utilize one of these services by establishing the needed settings in the Set Project/Data Folder command.
3. Assign a project to the Data Depot through the use of the Project Explorer command.
4. Subsequently open or update any Data Depot project through the use of the Get Project from Data Depot command *or* the Data Depot icon on the Start-up Wizard dialog box:



Carlson Software supports the following *version control systems*:

- Subversion - a free, open-source version control system.
- ProjectWise - software developed and produced by Bentley Systems.

Subversion is a powerful revision control system which is actively evolving and is part of Apache web-server project. The home page of this project is <http://subversion.apache.org/>, with a book on setup, use and administration available in print and online at <http://svnbook.red-bean.com>.

Carlson Software supports Subversion release 1.5.6 or newer.

While there multiple ways to setup Subversion repository and connect to it, the most typical scenario is setting up Subversion server and connecting to it through the web-server. Local directory setup is also available, but not useful in group environments. Running the shared directory on the network should not be attempted since it could lead to the repository corruption and permanent data loss.

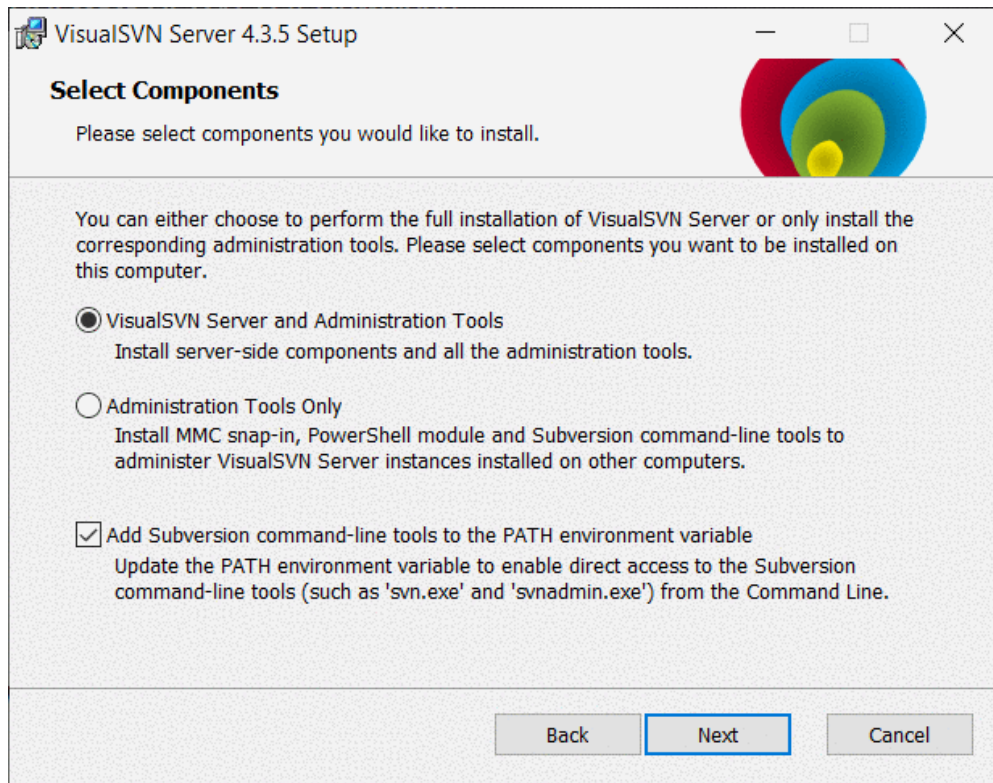
Setting up a Subversion Server on Windows Host

For users who run Windows on their host there is a nicely integrated front-end to a standard Subversion server (included): VisualSVN server (<http://visualsvn.com/server/>). There are two licenses available for commercial users:

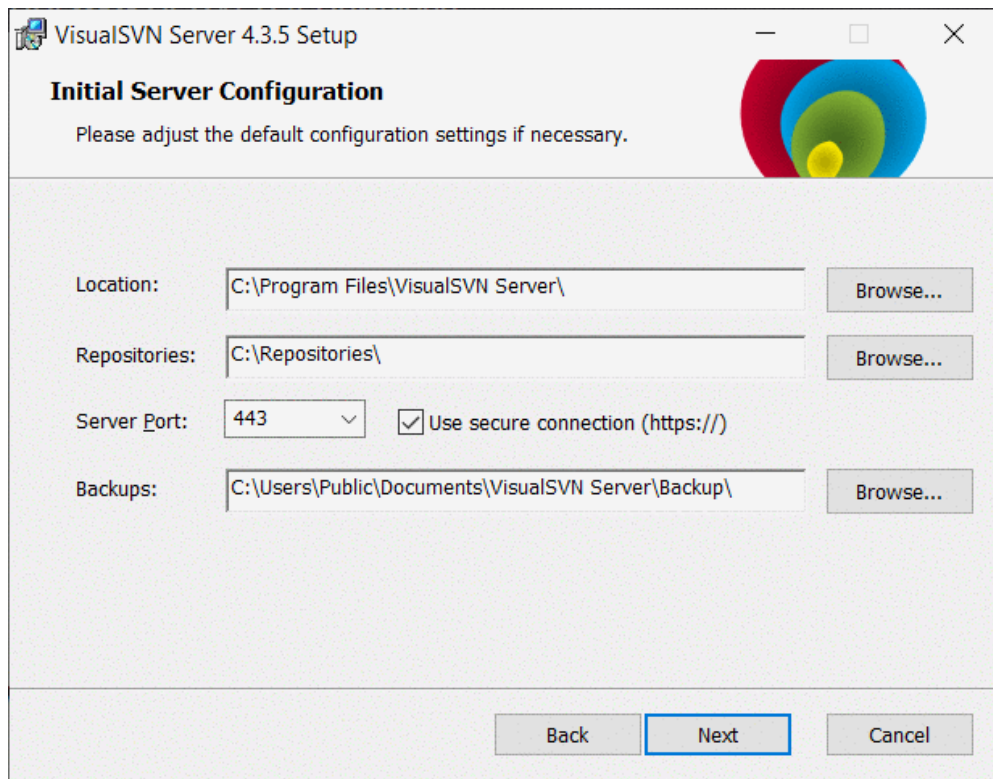
1. a free basic edition
2. a reasonably priced enterprise edition with additional features larger sites will find attractive

This document will concentrate on install of the basic edition.

1. Download and run the VisualSVN install.
2. Select option to install VisualSVN server:

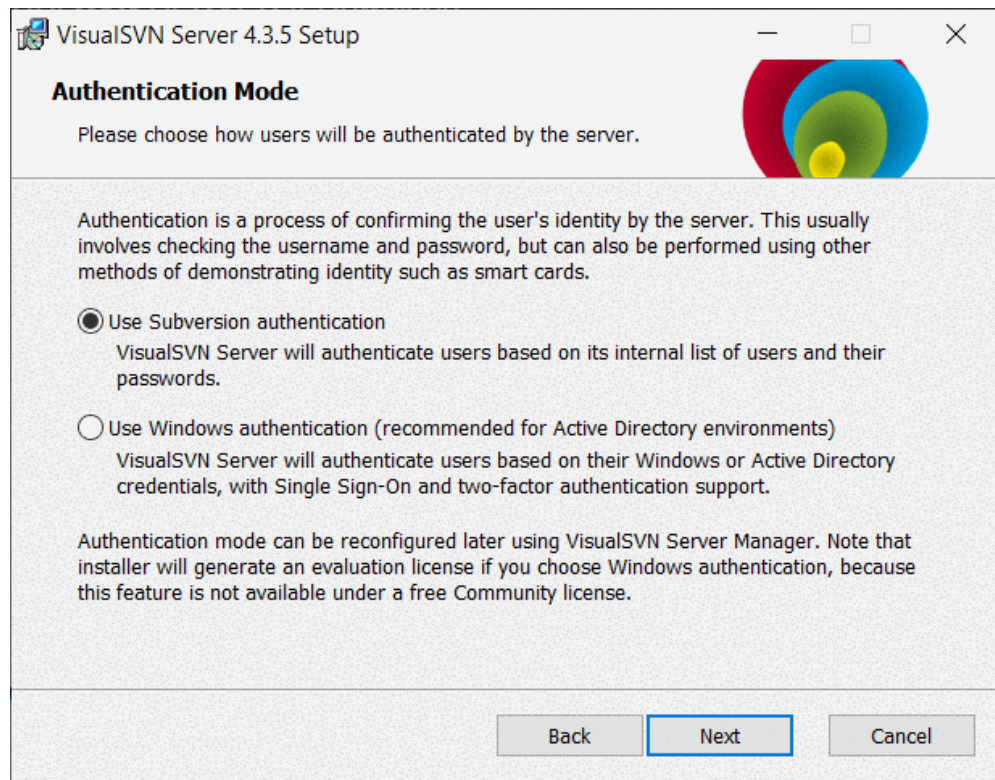


3. Specify repository directory where the data will be stored.

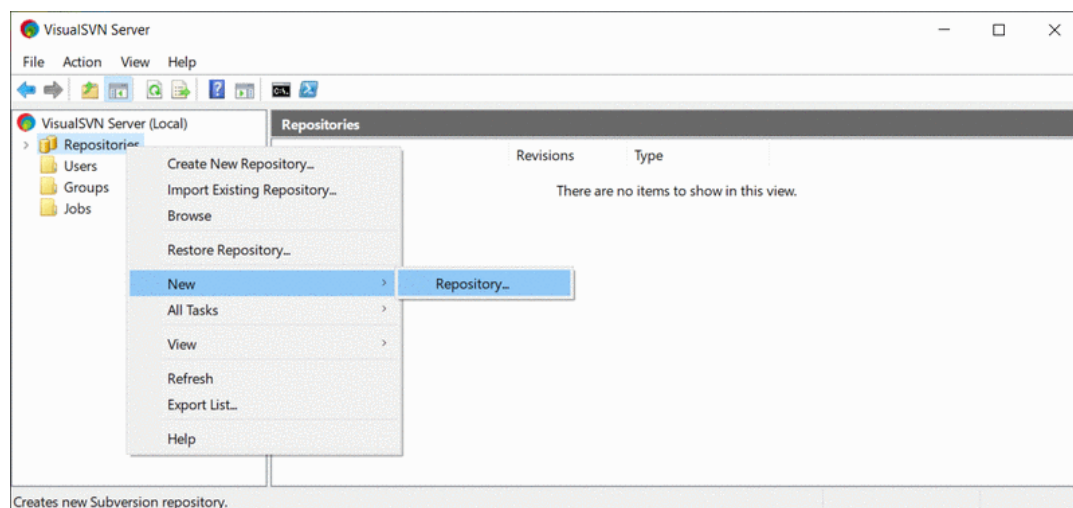


This should be a secure and backed-up location since this is where the data will be stored. The **server port** and **https://** settings are standard and most users will want to leave them alone.

4. The Authentication Mode is a matter of the site preference. Enterprise edition of VisualSVN offers additional authentication options like using Microsoft Active Directory server.




5. Upon completion of the install the manager interface will be displayed. Create a new repository by right-clicking on the Repositories:



6. Select the Repository Type:

Create New Repository ×

Repository Type
Choose the new repository type. 

Select the preferred repository type.

Regular FSFS repository
Create a regular Subversion repository based on the standard FSFS data store.

Distributed VDFS repository
Create a distributed Subversion repository based on the VisualSVN Distributed File System. The VDFS repositories act as standard Subversion repositories and allow data to be replicated between geographically distributed sites.

[Learn more about available repository types](#)

< Back Next > Cancel

7. Give new repository a name. You may choose to have several repositories, each containing data of the similar kind or covering an area of your business:

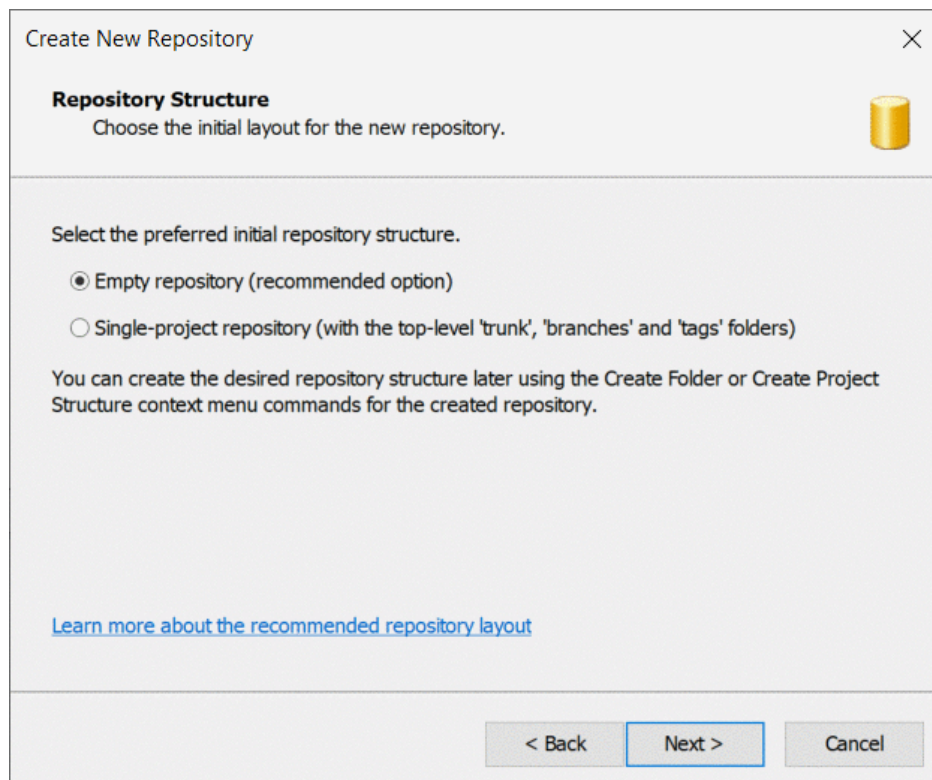
Create New Repository

Repository Name
Specify the name for the new repository.

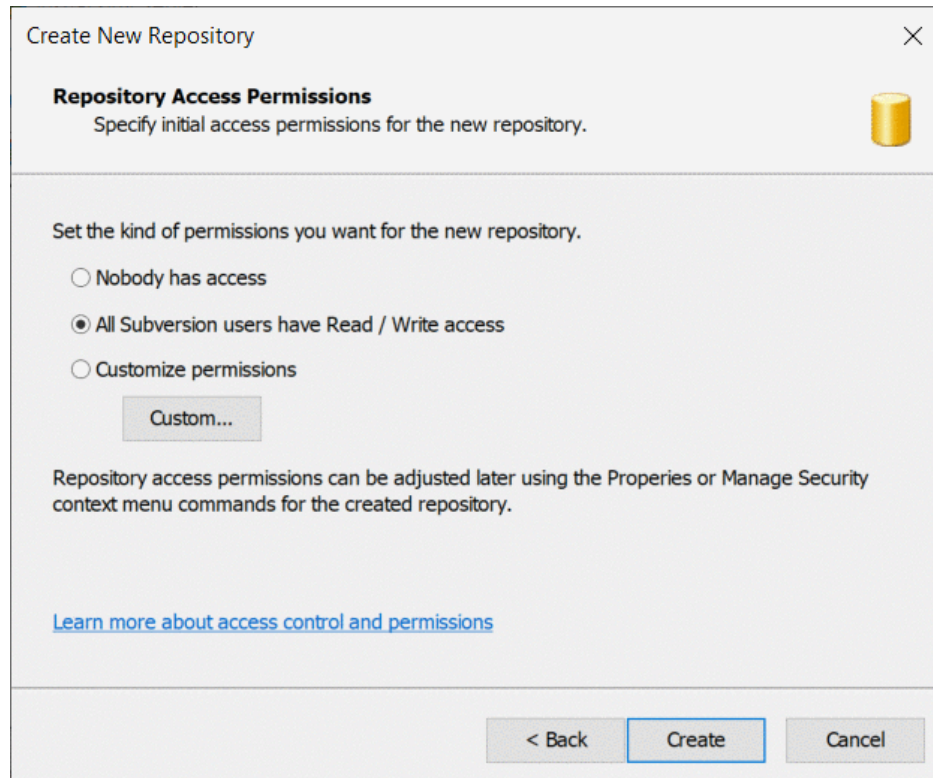
Repository Name:
Projects

< Back Next > Cancel

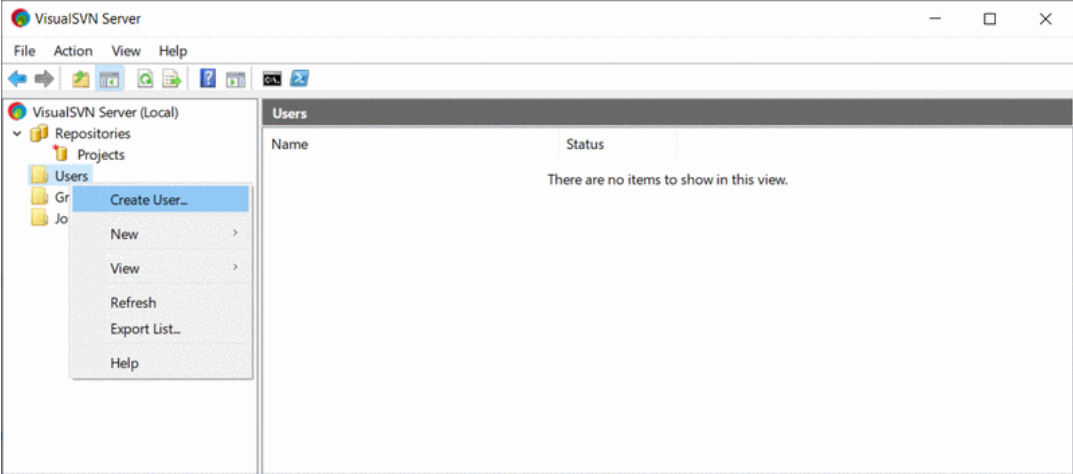
8. Select the Repository Structure:



9. Select the Repository Access Permissions:



10. Create users and set passwords (if using Subversion's own user authentication):



Create New User ✕

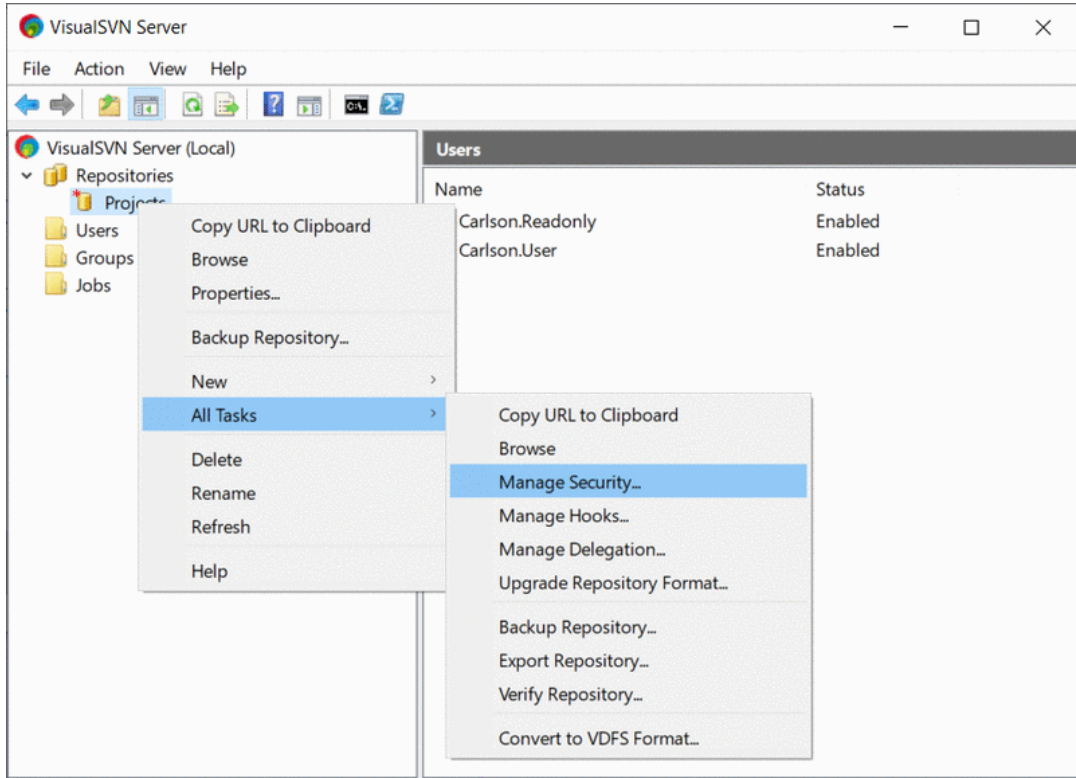
User name:

Password:

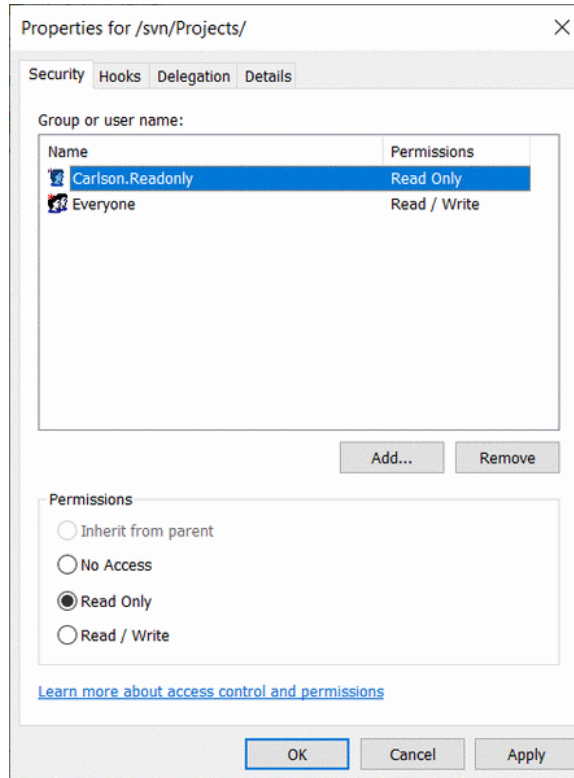
Confirm password:

i User name and password are case sensitive.

11. Run the "Manage security" command by right-clicking over whole repository to set defaults or right-clicking on specific folder to set specific permissions:



12. Add users or user-groups and set specific permissions for the folder you selected:



This completes the server side installation of Subversion.

Setting up a Subversion Server on Linux Host

Subversion is a project which was born on Unix platform, so it is fairly straightforward to set up and run on the Linux host. Specific details may vary for different distributions, but below is a quick summary of steps for Redhat or CentOS based hosts, which should at least point you in the right direction.

Install needed packages

The following packages should be installed:

- **httpd** - Apache web server for access to the subversion
- **subversion** - the Subversion command line and administration tools
- **mod_ssl** - support for secure connections
- **mod_dav_svn** - integration between the Apache and Subversion

Create repository folder

Run the following command to configure the Subversion file structure:

```
svnadmin create --fs-type fsfs /var/lib/subversion/repos
```

where 'fsfs' refers to the type of the file storage being selected and last argument is the future location of the repository data files on the system.

Configure Apache to be handle Subversion calls

The Apache configuration file httpd.conf is typically located in the /etc/httpd/conf folder. Please modify it to contain a section like this:

```
<Location /repos>
```

```
DAV svn
SVNParentPath /var/lib/subversion/repos
AuthType Digest
AuthName 'Subversion'
AuthDigestdomain /repos/
AuthUserFile /var/lib/subversion/svn_passwd
Require valid-user
SSLRequireSSL
</Location>
```

Restart Apache server. This should let you reach your Subversion server with URL like this:

```
https://server_name/repos
```

User control

The configuration above assumes the authentication for the domain "Subversion" handled by Apache itself. Much more powerful options are available, but since these are standard Apache features, plenty of documentation for is readily available.

To add new users, please use the following command:

```
htdigest -c /var/lib/subversion/svn_passwd 'Subversion' user_name
```

where -c is for "create" and only should be used first time. Supply user name for whom the password is being set. The permissions are controlled by the *svnserve.conf* file typically located in /etc/subversion directory. The equivalent configuration to the Windows example above would look like this:

```
[Projects:/]
Carlson.Readonly=r
Carlson.User=rw
```

For further documentation please refer to the book mentioned above or to the numerous Subversion resources online.

Creating a local Subversion Repository

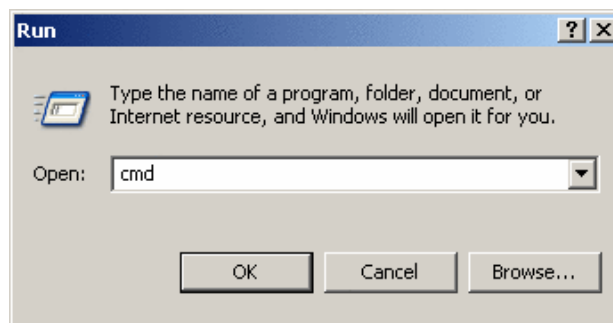
The Windows MSI installer with the basic Win32 client binaries can be downloaded at:

<http://www.collab.net/downloads/subversion/>

Running this installation file installs the required binaries to create the Subversion *repository* on the server or local machine under "**C:\Program Files\Subversion**" using the default options.

To create the Subversion repository:

1. Click the Windows Start > Run and then type **cmd** into the resulting dialog box as shown below and click **OK**:



2. At the DOS prompt, navigate to the Subversion install directory by typing:
cd "C:\Program Files\Subversion" and press the *Enter* button to complete this step as shown above.
3. To create the Subversion repository under the "C" drive, type:
svnadmin create c:\svnrepo and press the *Enter* button to complete this step as shown above. This will create

a "svnrepo" folder under the "C" drive (c:\svnrepo); see the **Notes** section below for important information. The path and the name of the repository could be path to a network drive as well. For setting up a Subversion Repository on the server, refer to the <http://subversion.tigris.org> documentation.

4. Type the word **exit** and press the *Enter* button to dismiss the DOS window and complete the command as shown above.

NOTE: Once you have created a Subversion Repository, do not add, delete, or modify files in the Repository folder unless you know how to avoid corrupting the repository.

Setting Subversion URL in Carlson

In order to connect Carlson to Subversion repository, please go to Settings – Configure Carlson – Project/Data Folders. Set the project type to Subversion and click Setup button. Specify the server location like:

file:///c:/svnrepo

for a local example above, or:

https://server.name/repos

for a network server as described above. Other URL types supported by Subversion for the scenarios beyond the scope of this document are permitted as well.

Accessing Subversion through a GUI Client

There are various graphical-user interface (GUI) client applications available for accessing the SVN repositories on the Internet for free:

- RapidSVN
- SmartSVN
- TortoiseSVN
- ViewVC
- WebSVN

Once a supported document management system has been properly installed and configured for each computer work-station, continue with the Data Depot configuration by initiating the Set Project/Data Folder command.

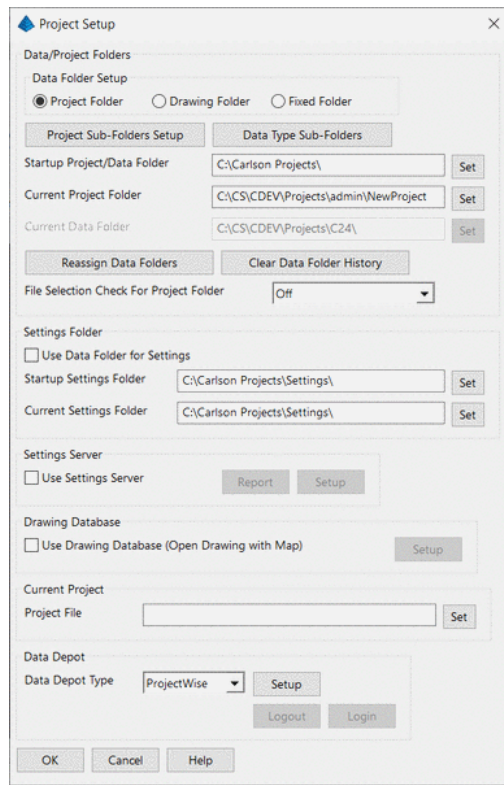
Please see your ProjectWise Administrator for instructions on configuring ProjectWise Datasources for your company. Here are steps to add a sample project to ProjectWise.

- 1) In the ProjectWise Explorer (not to be confused with the Project Explorer, which is the Carlson routine that operates inside of CAD) create a new project under the datasource that you will be connecting to in the "Documents" section (right click on Documents and use "New Project"). Name your project the same name as what you will be calling it in CAD. For example, call it "NewProject".

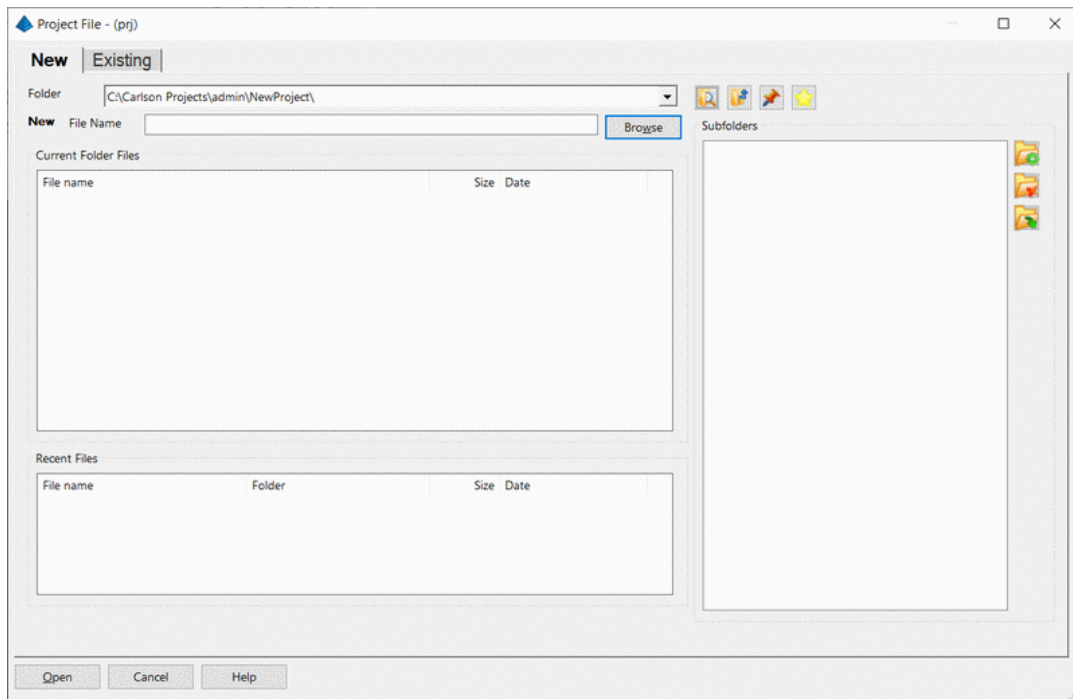
- 2) If you don't have one already, in Windows create a folder where you will store your project and related files. Place any files under that folder. You don't have to place all the files in that one folder. You can use any hierarchy layout that you want but in this sample case for illustration just uses the root folder of the project and a single drawing.

- 3) Open a drawing that belongs to this project.

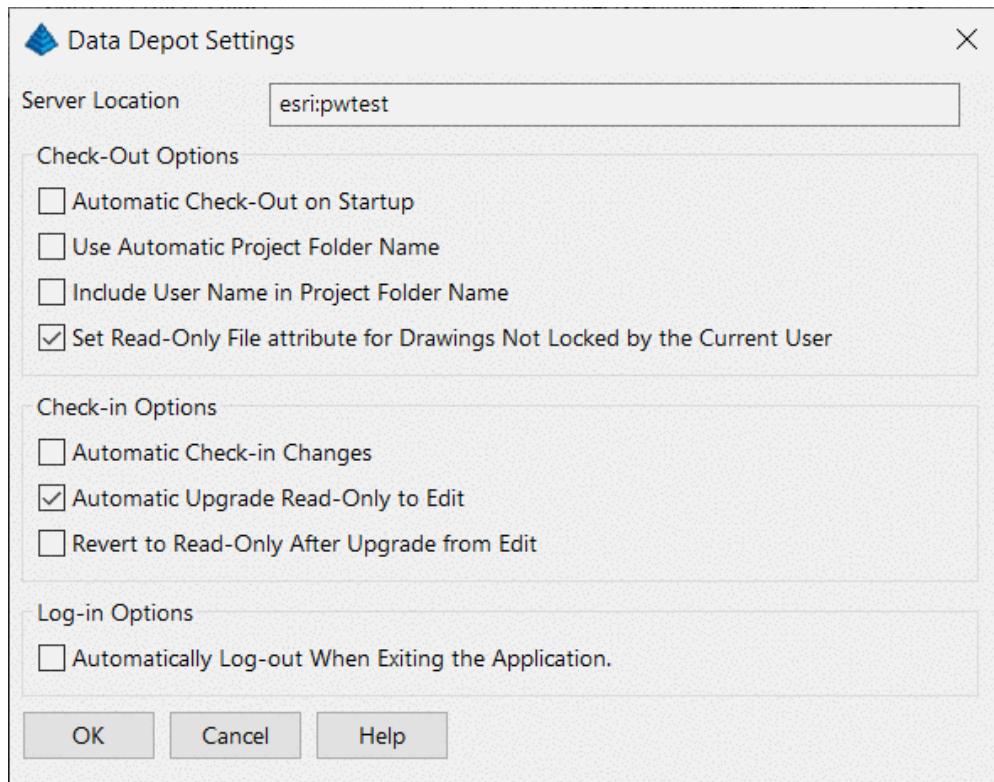
- 4) Go to the menu item Settings->Set Project Data Folders:



Your settings should look something like this. Select the "Project Folder" option. Make sure you set your Project File. When you click on Project File->Set, it will prompt you to either select an existing or create a new project:

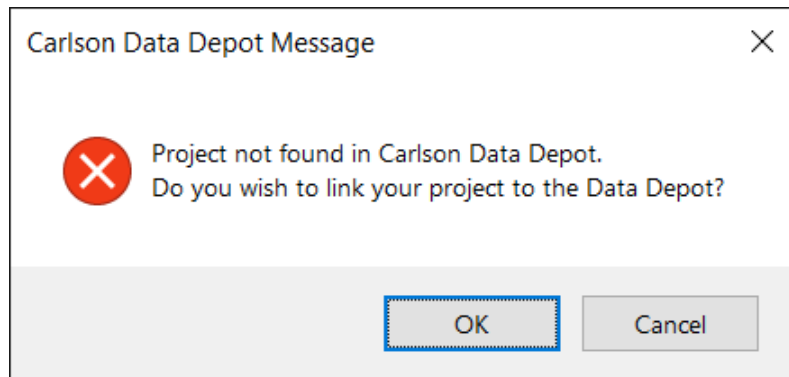


The project folder should have the same name as that you used in ProjectWise. In this case "NewProject". This name is case-sensitive. The .PRJ file name itself can be anything you want to call it. In this case it is called "newproject" as shown above. So the actual project file name is "newproject.prj". If you have multiple drawings that are part of this same project, you will want to make sure they all point to this same project file and have the same settings at this time. In this sample case, there is only the one test.dwg drawing.

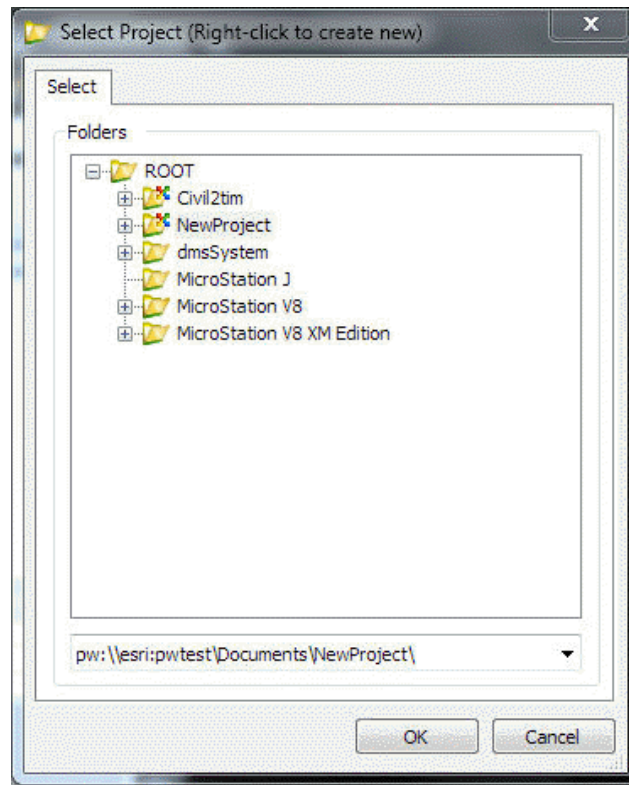


You will notice that the actual folder name shown above is C:\Carlson Projects\admin\NewProject. This is because in the Data Depot Settings the user's name is included in the folder name. In this case the user's name is "admin". This makes it easy to use a single computer with multiple users where they can store their local copy independently of other users.

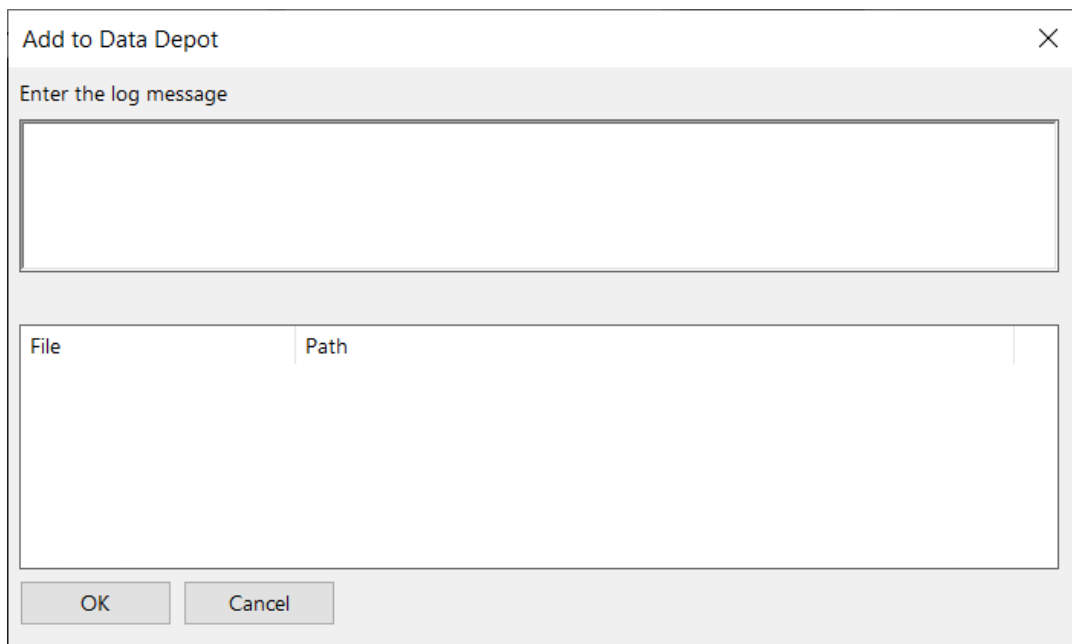
5) Open the Project Explorer in CAD (menu option File->Project->Project Explorer). You should see the following alert:



Select OK. Next you will be asked to select the project in ProjectWise that you want to store the files under:



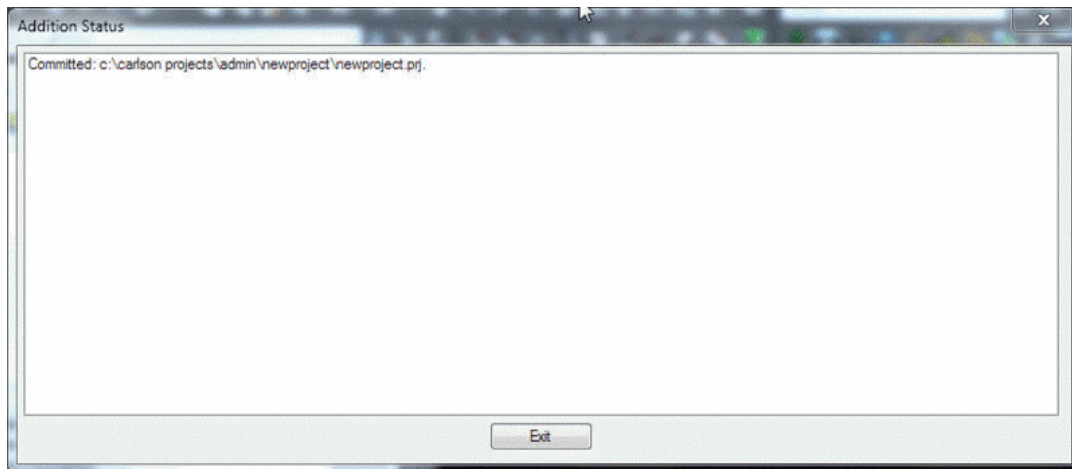
6) Select the new project you created earlier in step 1. In this sample case it is "New Project" as shown above.



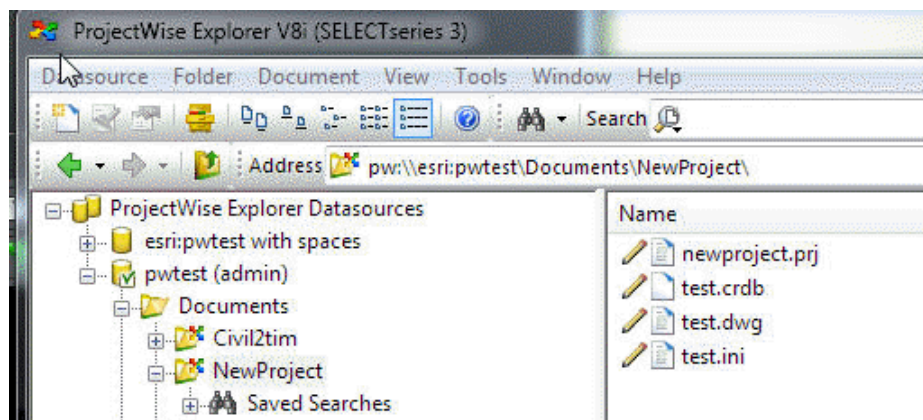
At this dialog, keep the default of "The project and all associated files". Enter any log message you want. This will appear in your ProjectWise audit trail.

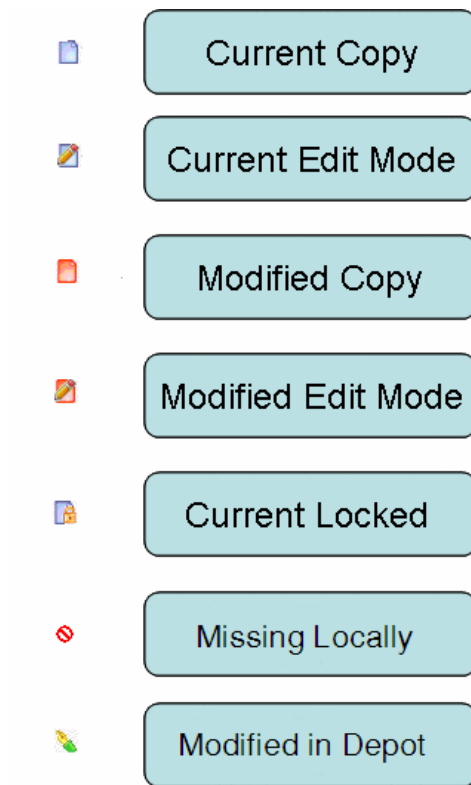
At this point it should upload all of the files that are part of this project. In this sample case that is test.dwg and associated files. If there are any XREFs that have not been uploaded they will also be uploaded at this time.

Once done uploading you will see a message showing that the project file has been uploaded (shown below). In the command window you will also see a list of all of the files that were uploaded to ProjectWise (test.dwg, test.crdp and test.ini in this sample case).



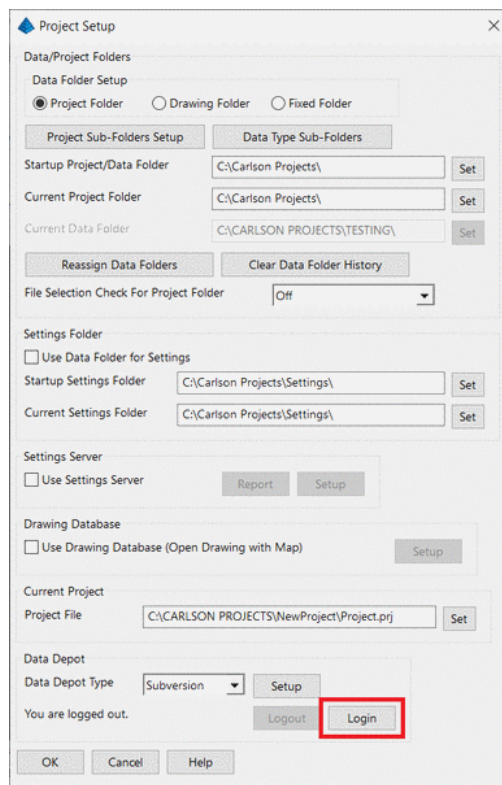
You can now go to your ProjectWise Explorer to see the results:





Logging in or Out of Data Depot in Carlson

1. Go to the menu item Settings->Set Project Data Folders:
2. Click the Login button to log in to the server located at the location specified in the "Data Depot Settings" dialog box(accessed by clicking Setup next to Data Depot Type):



3. Once logged in, the Logout button will be enabled and you will be able to log out from the server.

Pulldown Menu Location(s): None

Keyboard Command: None

Prerequisite: None

Carlson Settings Explorer

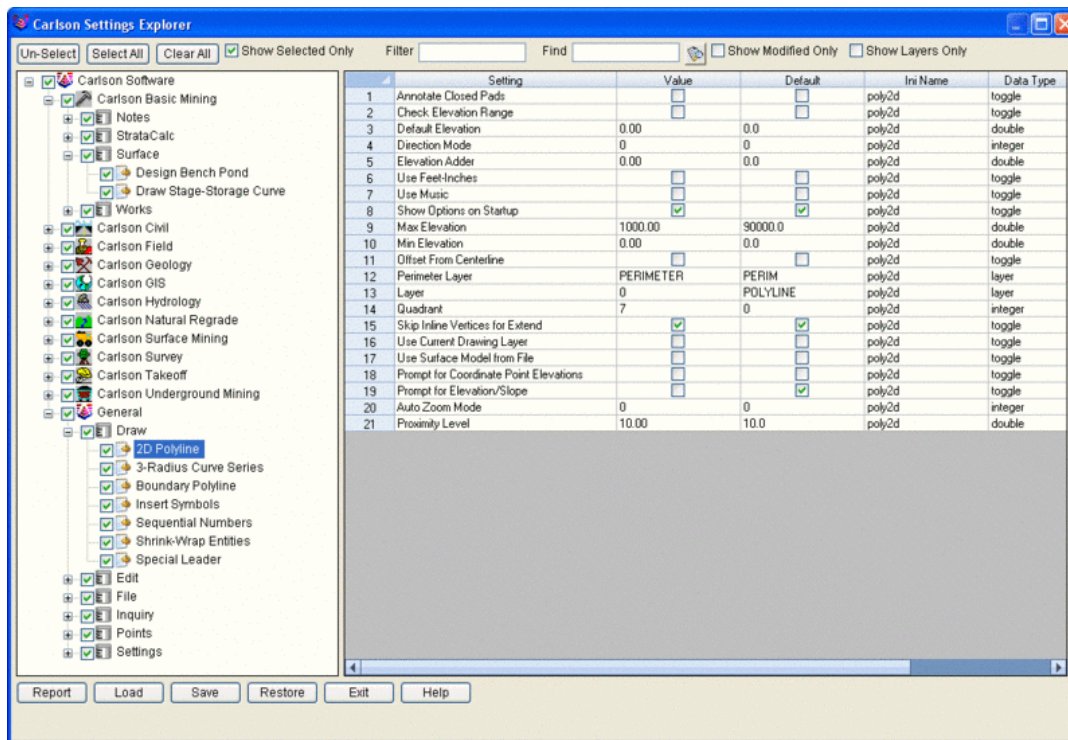
The Carlson Settings Explorer dialog box allows you to view, manage and report the values for all settings in all commands in all Carlson Software programs.

The tree-view on the left of the window shows the four levels of the command structure of Carlson Software, Program, Menu and Function. Selecting (highlighting) any one or many of these levels will show associated settings. The programs level also includes General, that includes the first six menus available in all Carlson programs.

The right-side is a spreadsheet view and is populated depending on the item(s) selected and highlighted in the tree-view. Selecting and placing a checkmark next to an item at one of the upper levels of the tree structure will select that item and all its sub-items.

The spreadsheet view shows the following columns for a command:

- Setting - Prompt for the settings in the dialog or at Command: line
- Value - Current value
- Default - Default value for the setting
- Ini Name - Carlson internal file name for the command
- Data Type - Description of the value for this setting



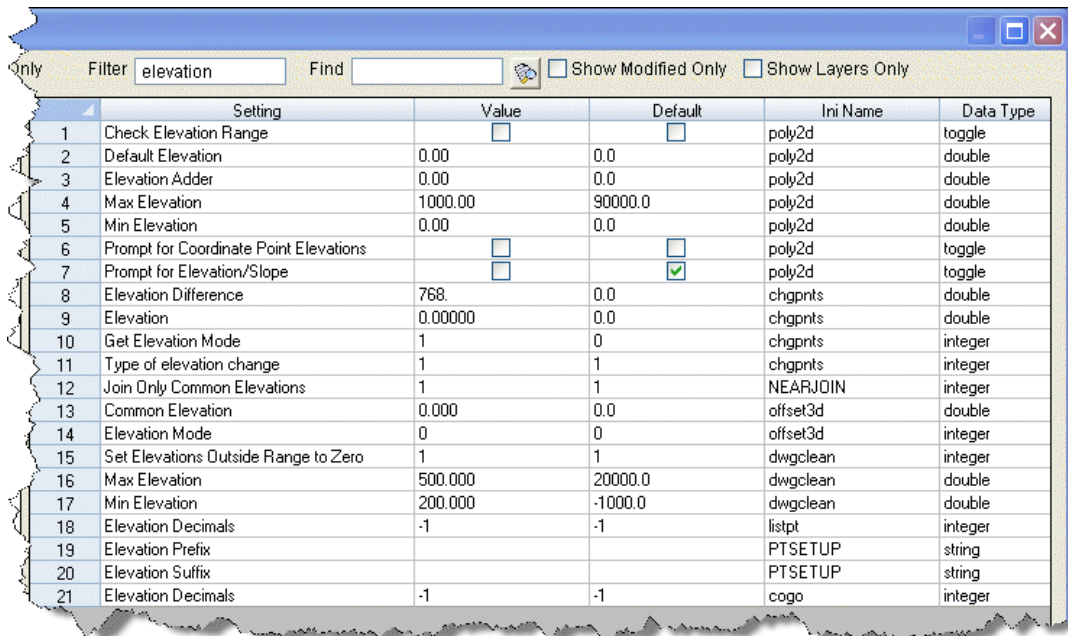
Select/Unselect: Using the square tick boxes you can click this toggle to select or unselect a unique item or all items under the hierarchical list. All ticked items will appear in the spreadsheet view to the right.

Select All: Use this button to place a checkmark next to all items and sub-items in the tree-view.

Clear All: Use this button to clear checkmarks from all items and sub-items in the tree-view.

Show Selected Only: Select this option to display, in spreadsheet view, the tree-view settings that have checkmarks on.

Filter: Using a Filter will display the settings containing the text string specified as the Filter. For instance, filtering with the word "elevation", returns the following results:



Find: Use this option to search for a string of text. First, enter the text to be found and then pick the button to the right to execute the search.

Show Modified Only: Select this option to display the settings whose current value, as shown in the "Value" column, is different from the default value shown in the "Default" column.

Show Layers Only: Select this option to display the settings whose value in the "Data Type" column is specified as "Layer".

Report: Use this button to prepare a report of all selected settings using Carlson's Report Formatter.

Load: Use this button to Load settings from a Carlson Configuration File (.cfg).

Save: Use this button to Save settings to a Carlson Configuration File (.cfg).

Restore: Use this button to Restore selected settings to their Default value.

Pulldown Menu Location: Settings

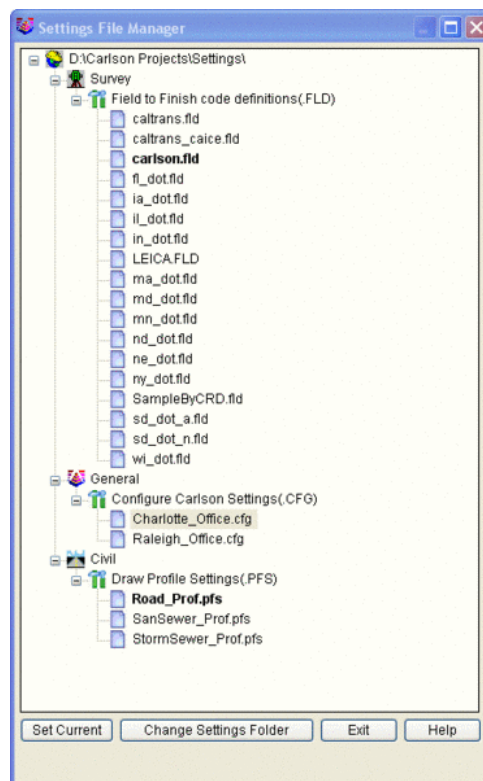
Keyboard Command: setxplore

Prerequisite: None

Settings File Manager

The Settings File Manager provides an organized view of all Carlson Software Settings files that have been saved in a specified "Settings" folder. By default, the initial folder displayed is the "Startup Settings Folder" as specified in the Project Setup dialog box.

The various Settings files are displayed in a tree structure based on their corresponding Carlson Software program. If the current Settings for any command match the Settings found in a Settings file, the file name will show up in a **Bold** font. Because Configuration Files (.cfg) are collections of many Settings files, they cannot be set Current and will not show up in Bold font.



Set Current: After selecting one of the various Settings files, use this button to set that file current. Files that have been set "Current" will show up in a **Bold** font. Only one file of any Setting type can be set Current at one time. And, because Configuration Files (.cfg) are collections of many Settings files, they cannot be set Current and will not show up in Bold.

Change Settings Folder: Use this button to browse to and select a folder in which Carlson Software settings have

been stored.

Compare: This function compares two settings files and reports differences in a spreadsheet. To run this function, select the first settings file to compare from the Settings File Manager dialog. Then pick the Compare button and the program will prompt for the second file of that type to compare with. In the Compare dialog, you can make changes to the settings and use the save buttons to save the edits.

	Setting	test.cfg	NCS_Layers.cfg	Data Type
1	Round Exterior Corners	<input type="checkbox"/>	<input checked="" type="checkbox"/>	toggle
2	Max Depth	1.	1.0	double
3	Min Bench Height	1.	1.0	double
4	Horizontal Interval	25.	25.0	double
5	Vertical Interval	5.	25.0	double
6	Bench Color	1	256	color
7	Create Road	<input checked="" type="checkbox"/>	<input type="checkbox"/>	toggle
8	Berm Height	0.	0.0	double
9	Bench Stop Distance	300.	50.0	double
10	Bench Taper Distance	300.	50.0	double
11	Road Width	150.	10.0	double
12	Road Color	3	256	color
13	Offset Method	<input checked="" type="checkbox"/>	<input type="checkbox"/>	toggle
14	Sequence Method	1	0	integer
15	Slope Direction	0	1	integer
16	Slope Direction	0	1	integer
17	Round Exterior Corners	<input type="checkbox"/>	<input checked="" type="checkbox"/>	toggle
18	Horizontal Interval	25.	25.0	double
19	Vertical Interval	5.	25.0	double
20	Bench Color	1	256	color
21	Create Road	<input checked="" type="checkbox"/>	<input type="checkbox"/>	toggle
22	Road Width	150.	10.0	double
23	Road Color	3	256	color
24	Bench Stop Distance	300.	50.0	double
25	Bench Taper Distance	300.	50.0	double
26	Berm Height	0.	0.0	double
27	Decimals	2	3	integer
28	Decimals	2	3	integer
29	Starting Station	0.	0.0	double

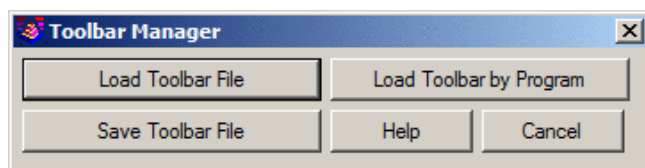
Pulldown Menu Location: Settings

Keyboard Command: setmanager

Prerequisite: None

Toolbar Manager

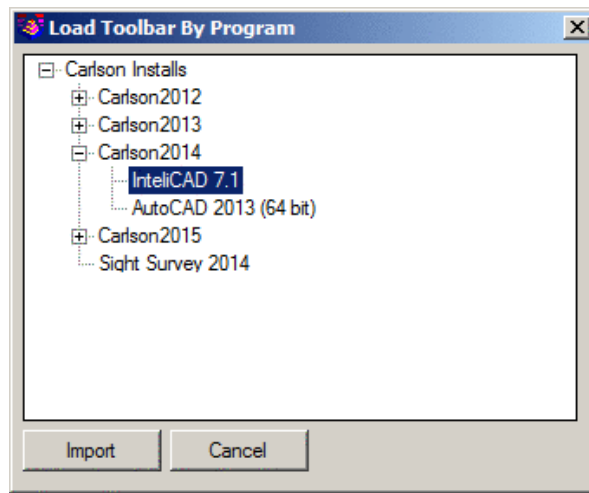
The *Toolbar Manager* command permits the saving and loading of toolbar configurations and allow migrating toolbar settings between versions.



Load Toolbar File: Restores the toolbar settings from a previously saved *.TB file.

Save Toolbar File: Writes the current toolbar settings to a *.TB file.

Load Toolbar by Program: Loads the toolbar configuration associated with an alternate version of Carlson into the current application.



Note:

- The *Load Toolbar by Program* does not allow the toolbar configuration from the current software installation to be loaded into itself.
- The files created and used by this command are stored in an Extensible Markup Language (XML) format but are assigned a *.TB extension.

Pulldown Menu Location: Settings

Keyboard Command: tb_manager

Prerequisite: None

Edit Symbol Library

This command allows you to customize the symbol library. For a printout of the default symbols, get the symbols.pdf in the Carlson Projects folder. The default library has hundreds of 2D and 3D symbols including National CAD Standard (NCS) symbols, MassDOT standard symbols and a set of symbols with wipeouts built-in to hide linework under the symbols.

Categories are a way for grouping symbols by type for your own convenience in symbol selection. There are two levels of categories: top-level and sub-category. Within each top-level category, you can have any number of sub-categories. A new category is added by clicking on the "Add Category" button. An edit field then appears in the tree view on the left and waits for you to enter the category name. The input is finished by pressing the Enter key.

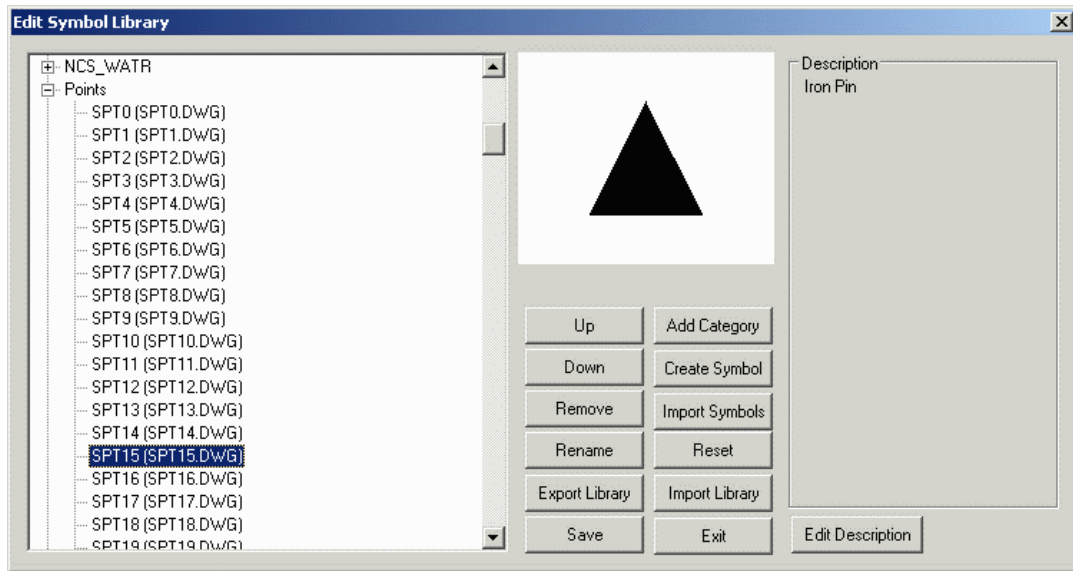
The category may be populated by creating a new symbol from selected entities in the drawing, by specifying drawing (.DWG) files, or by moving existing symbols from one category to another.

Each symbol has an optional description which is shown with the symbol to help choosing in symbol selection like in Draw > Insert Symbols. To change a description, highlight the symbol in the list and then pick Edit Description.

To create a new symbol, open a drawing which has the entities to be used in the symbol. The symbol should be drawn at unit size (scale 1:1) because Carlson will scale the symbol by the current drawing scale when the symbol is used. Highlight the category for the symbol and click on the "Create Symbol" button. A dialog appears for entering the new symbol name. Next, specify the file name for the symbol. The file name has a .DWG extension and would usually reside in the Carlson SUP directory, but you may use another path. Then the program will prompt you to select the entities from the drawing for the symbol. An insertion point for the symbol must also be picked.

The "Import Symbols" button brings up a file selection dialog which allows you to select multiple files to be added to the current category (to select multiple files use Shift or Control keys along with the mouse). If the files you select are not in the Carlson SUP directory, the program will offer an option of copying them there. There are also

Import Library and Export Library buttons.



By default, the symbol description is the same as file name. The description for the symbol or category name may be changed by highlighting that name and clicking on "Rename" button, the name being edited is then placed into edit mode. To move a symbol into a different category, select the symbol to be moved on the tree and click an "Up" or "Down" button as many times as needed to reach the desired category. The symbols are sorted alphabetically within each category, while categories are remaining in the order placed to allow the more frequently accessed categories be on top.

The Reset button adds symbols from the default symbol library into the current library.

Note: The symbol library is stored in an ASCII file named symbols.dta in the Carlson \USER directory.

Pulldown Menu Location: Settings

Keyboard Command: editptsym

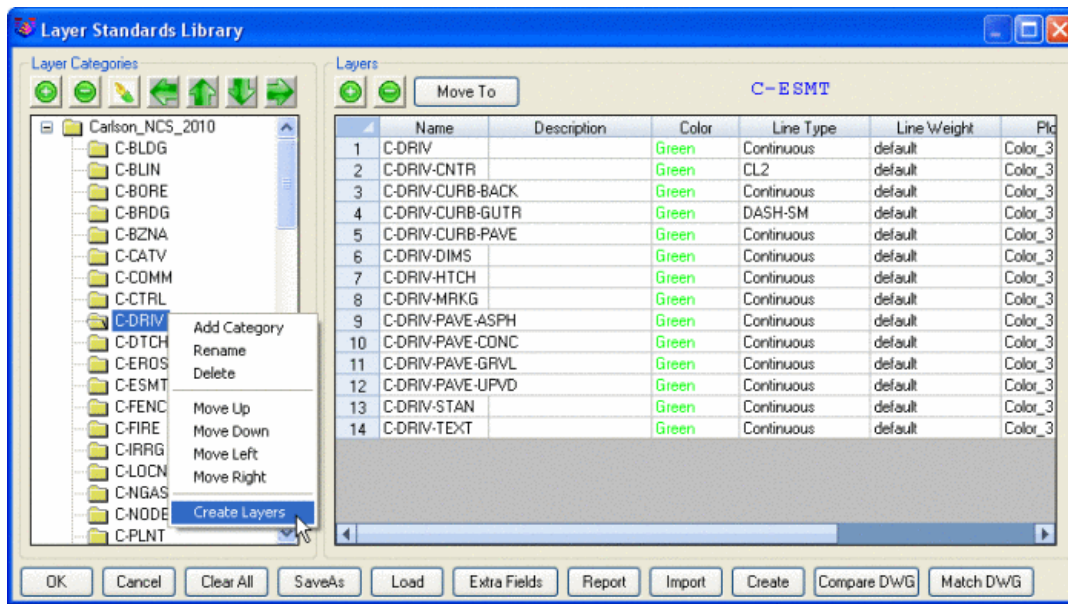
Prerequisite: None

Layer Library

This feature serves as an expanded version of the Layer Manager and also as a layer standards manager. In addition to allowing you to sort layers into easily recognizable groups called **Layer Categories**, this feature can also be used to import layers from a text file and to compare and match layers in the library to the current drawing.

Once populated, layers from the Library can be called from commands such as **2D Polyline** and **3D Polyline** for layer and property assignment.

The **Layer Library** has two areas of the dialog box: the **Layer Category** List on the left and the **Layer** List on the right.



Layer Categories: Layer Categories are shown as a list in a tree view in the left-hand pane of the dialog box.

Categories can be re-ordered by dragging and dropping to a different position in the list or by using the **Move Up**, **Move Down**, **Move Left** and **Move Right** arrow buttons. Other buttons above the Layer Category list also enable you to **Add**, **Remove** and **Rename** Categories.

Also, right-clicking on a Category in the list displays a shortcut menu allowing you to access many of the same commands as the buttons along the top and bottom of the dialog box.

Layers: Layers in a selected Category are shown as a list in spreadsheet view in the right-hand pane of the dialog box.

Layer Properties: Except for Non-Surface, all the layer properties in Layer Library correspond to the layer properties in CAD. The **Non-Surface** property is a specific flag used by the Triangulate & Contour routine to filter out entities. You can set a layer as Non-Surface when you want to exclude all entities on that layer from surface modeling.

The default column-headings for the Layer List are Name, Description, Color, Line Type, Line Weight, Non-Surface, Transparency, Plot Style and Plot/No Plot. Additional column-headings may be added using the **Extra Fields** button at the bottom of the dialog. Using the **Add Layer (plus)** and **Delete Layer (minus)** buttons, layers can be easily added and removed from a particular Category. The **Move To** button can also be used to change a layer to a different Category.

Clear All: This button removes all the Layer Category and Layer definitions.

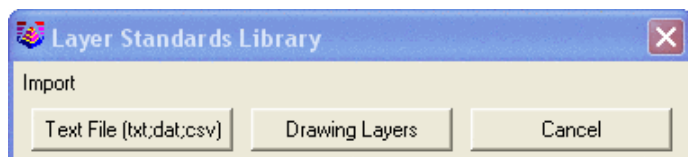
Save As and **Load:** These buttons can be used to create and restore **Layer Library** settings using a Layer Library Settings (.LA) file. There are a few default .LA files in the Carlson Projects\Settings folder that you can load for National CAD Standard (NCS) layers and MassDOT standard layers. The current layer library definitions are stored in the USER folder in a file named **layerstd.dta**.

Extra Fields: This button allows you to define up to ten extra text fields (column headings) for a layer. These fields can then be used as import fields or displayed in a report.

Report: This button uses the Report Formatter to allow you to compile and display a report containing all Layer Categories and Layers in the Library. The Report Formatter can also be used to export the data to a Microsoft Excel

(.XLS or .XLSX) file.

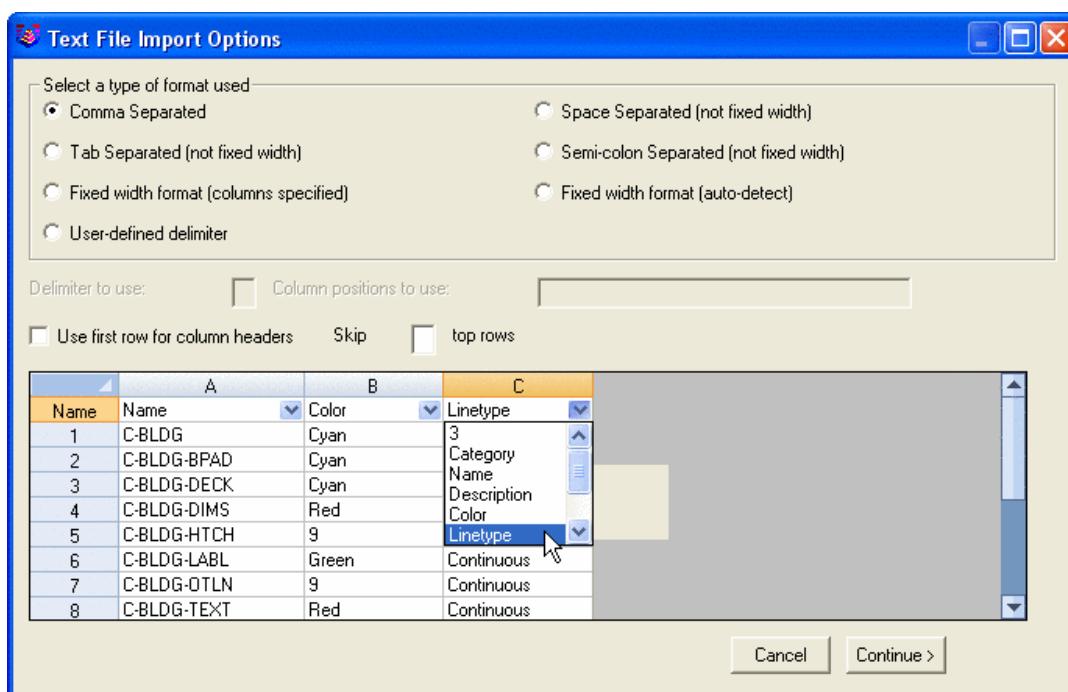
Import: This button gives you two options for Importing layers into the **Layer Library**.



The **Drawing Layers** option simply copies all layer definitions from the current drawing into the Library after prompting you to select the destination Category.

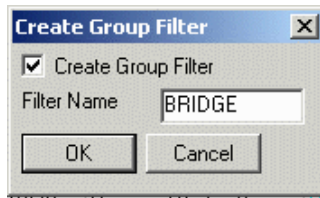
The **Text File** option allows you to select an existing Text (.TXT, .DAT, or .CSV) file containing standard layer definitions to populate the Library. Note that Microsoft Excel provides an option to save an Excel (.XLS or .XLSX) file as a Text file. Follow the steps below to Import layers from a text file.

1. Pick the **Import** button.
2. Pick the **Text File (txt;dat;csv)** button. This opens the **Text File Import Options** dialog box.



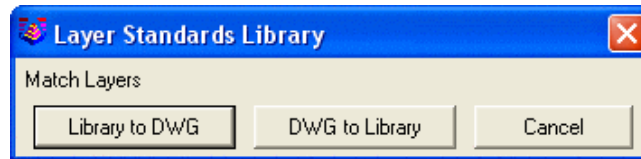
3. At the top of the **Import Options** box, select the file format type such as "Comma-Separated" or "Tab Separated".
4. If the top-line of the Text file contains column headings, pick the option to "Use first row for column headers".
5. If the Text file contains one or more lines of text above the layer and property data, use the text box next to "Skip" to specify the number of rows at the top of the text file to be "Skipped" before importing the list of layers.
6. If headers are not included in the Text file, use the drop-down at the top of each column heading in the spreadsheet view to specify the column's data type such as "Name", "Color" or "Linetype".
7. Pick the *Continue* button and specify the Category into which the new layers are to be Imported.

Create: After selecting a Layer Category, you can pick this button to create all the layers for that Category in the Drawing. When creating the layers, there is an option to create a layer group filter for the CAD Layer command.



Compare DWG: This button is used to Compare drawing layers and their associated properties such as color, linetype, lineweight and plot style to the standard Library definition for those layers. This feature will report how many layers matched exactly, how many had a different set of properties and how many non-Standard layers were found. It will also list the non-standard layers which are those defined in the drawing but not in the Library.

Match DWG: This button is used to alter the properties of drawing layers to match the properties of layers defined in the library, or vice versa. After picking the **Match DWG** button, this dialog box displays:



Pick the **Library to DWG** button to alter the drawing layers to conform to the Library definitions.

Pick the **DWG to Library** button to alter the Library definitions of the layers to conform to those set in the drawing.

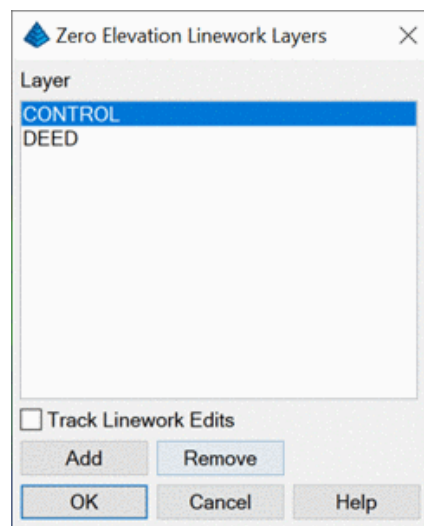
Pulldown Menu Location: Settings → Layer Library

Keyboard Command: layerlib

Prerequisite: None

2D Linework Library

This command sets lines and polylines on specified layers to zero elevation which helps solve the common drafting problem of linework getting moved to elevation such as with a grip edit snapping to a 3D point. In the dialog, you select which layers to check. When you pick OK, the program checks all linework on those layers and sets their elevations to zero. The Track Linework Edits option checks when the linework is modified to make sure the elevations are zero. When this routine prints a message to the command line whenever it moves the linework elevation to zero.



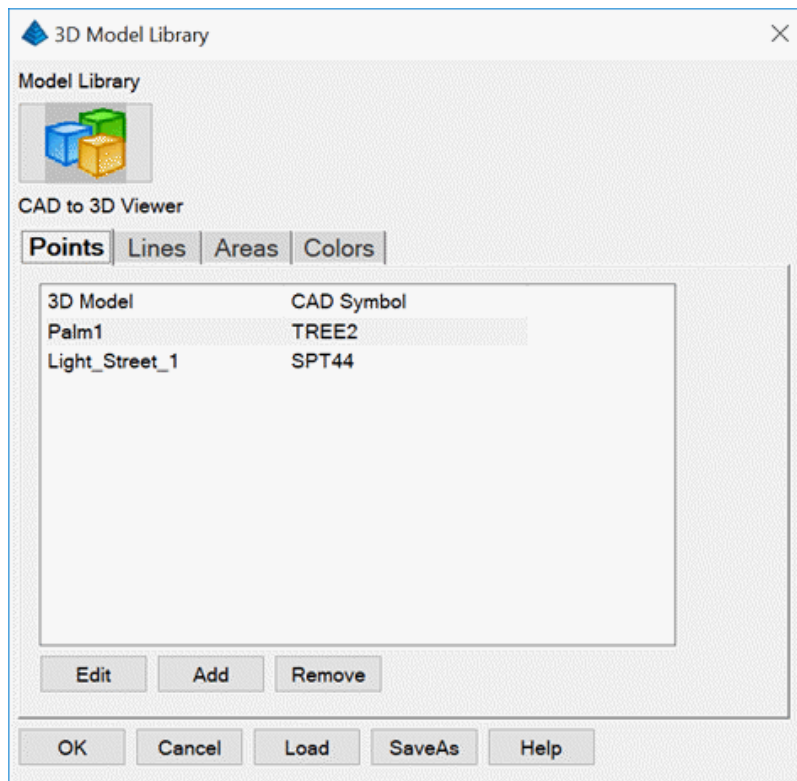
Pulldown Menu Location: Settings

Keyboard Command: 2dlworklib

Prerequisite: None

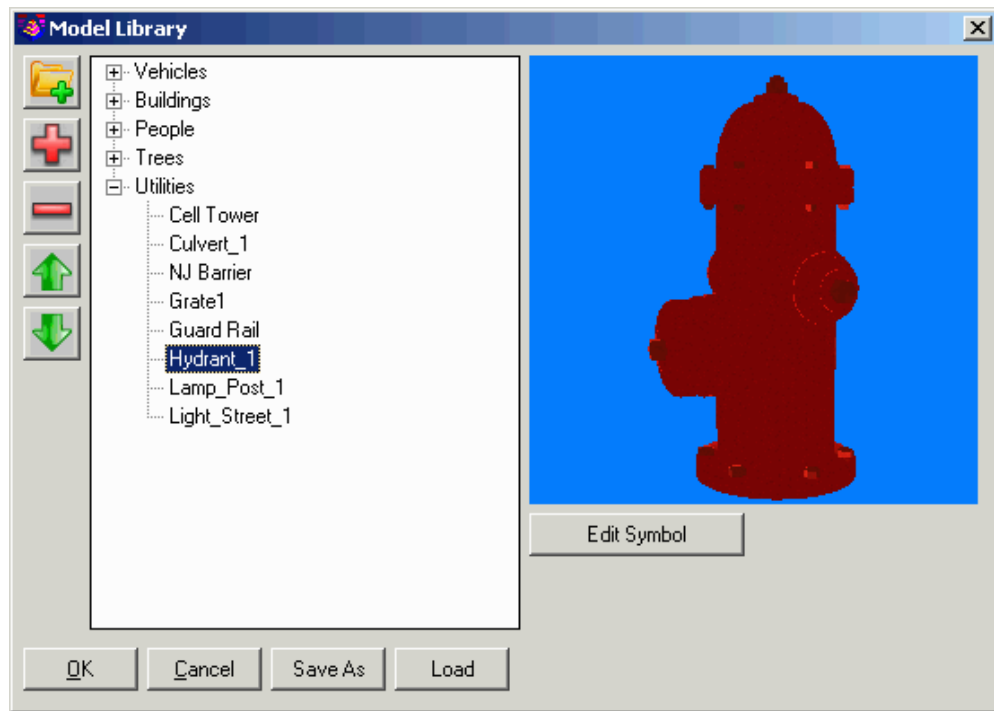
3D Viewer/Model Library

This command manages a collection of 3D graphic files and defines how to render CAD entities in the 3D Viewer. The CAD to 3D Viewer settings define how to translate different types of selected CAD entities into the 3D Viewer. The **Points** tab associates a CAD block name with a 3D model. This way you can have a simple symbol when working in CAD that is rendered as a complex 3D model with viewing in the 3D Viewer. The **Lines** tab defines a 3D model to use for linework on a specified layer. For example, a polyline on a layer of FENCE in CAD can be assigned to render using a fence 3D model. Besides defining the layer and 3D model, there is also a setting for Gap which is the distance between the 3D models along the linework. The **Area** tab defines textures to apply to the surface model by closed CAD polylines on specified layers. You can have different textures using nested closed polylines on different layers and the texture for the inner polyline is used for that inner area. For example, you can have an outer polyline on a Parking layer for a texture of asphalt and inner polylines for parking lot islands on an Island layer for a texture of grass. The **Colors** tab applies to TIN surfaces that have colors assigned to the 3D faces. This settings map TIN colors to textures for rendering the TIN.



The Model Library button has the collection of 3D model files which can be assigned to CAD symbols in the Points tab of CAD to 3D Viewer. Also, these 3D Models can be added to the scene by using the Add Model function in commands such as 3D Drive Simulation.

The files are grouped by user-defined categories. The graphic files can be either .dxf, .mdl or .obj format. SketchUp .skp files can also be selected to add to the library. The program will convert these .skp files into .obj format. The SaveAs and Load functions allow you to save the model library to .mlib files for sharing or backup. The current model library data is stored in the Carlson USER folder in models.dta.



The Edit Symbol function brings up another dialog with edit functions. In the edit dialog, the program shows the dimensions of the model and the number of data points.

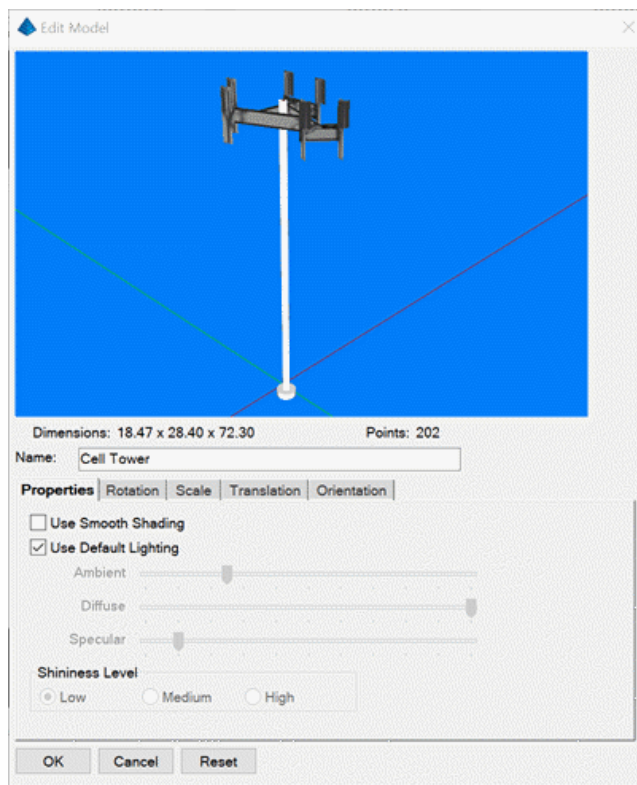
The Properties tab has controls for the lighting of the object and whether to smooth the 3D faces of the object.

The Rotation function sets the default rotation for when the object is inserted. The Random Rotate option makes the rotation random for each time the object is inserted which applies to objects like trees that are inserted multiple times and look better with different rotations.

The Scale function sets the initial insertion scale. Often a scale factor of 0.08333 is needed to convert a model in inches units into feet for inserting into a scene in feet units. The X/Y/Z axis can be scaled independently or uniformly. The scale can be set directly by factors or by entering the model dimensions which is handy for vehicle models when the dimensions of the vehicle are known.

The Translation settings control the insertion point of the model.

The Orientation setting for Force Vertical Orientation makes the model level when inserted into the scene which applies to objects like trees and buildings. Otherwise the models get inserted at the slope of the surface at their position which is good for objects like vehicles.



Pulldown Menu Location: Settings

Keyboard Command: modellib

Prerequisite: None

Quick Keys

Quick Keys can provide an enormous time savings on initiating keyboard commands. One frustration of using CAD is when your command initialization cannot keep up with your train of thought, and you are constantly seeking the fastest way to initiate commands. Quick Keys provides numerous command aliases that are already set up for you, which you can customize easily. You can also add any new commands quickly and easily using the Quick Keys editor, without leaving CAD. The Quick Keys are so productive, that even dedicated menu users appreciate and use them.

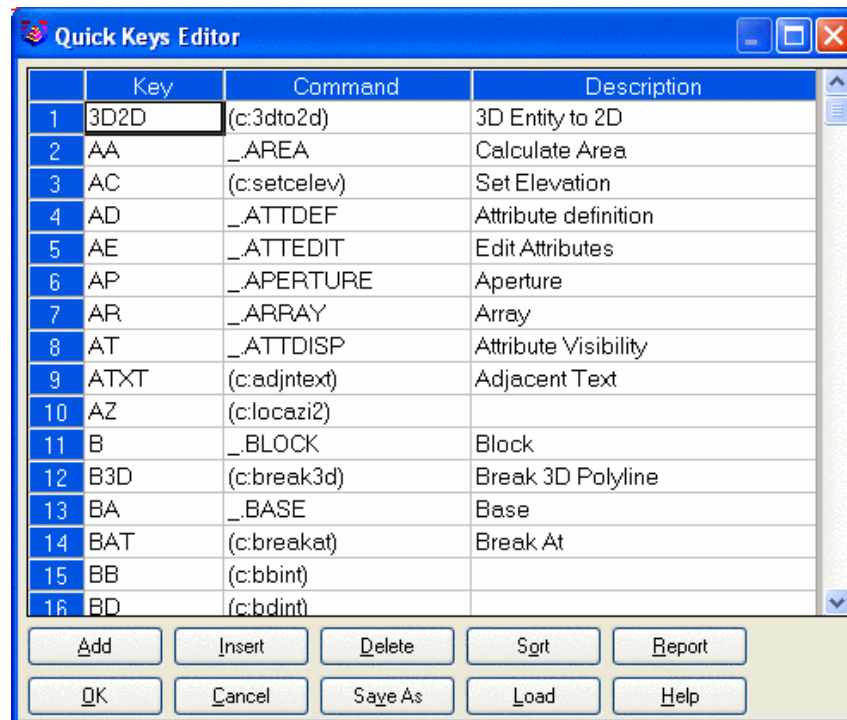
The Quick Keys Editor can be broken down into areas. The spread box at the top of the screen, spread control buttons, and program control buttons. Each area and button is explained below.

Spread Box contains a list of the loaded quick keys. Use the scroll bar to move up and down through the list or maximize the dialog to view more rows. Each item can be edited in its cell.

Each record item consists of up to a 5 character quick key (the portion typed in at the command line), and a long command or AutoLISP expression. Note that if you are adding a shortcut to a lisp function, you must use the following syntax: (C:FUNC) where FUNC is the command name. All Carlson commands use this lisp function syntax. To find the lisp function name for a Carlson command, you can run the command from the menus and look for the command name in Command window. Also, you can find the command name in the Keyboard Command field at the end of each command description in the manual. For example, enter (c:quickkey) for the command to run Quick Keys. For CAD commands, the command name can be entered directly with a prefix of "_.". For example, the CAD Area command name should be entered as _AREA. Some CAD commands will run in command line prompt mode instead of dialog mode when issued by the command name from Quick Keys. For these commands that you want to run with the dialog, enter the command with this syntax of (c:acad_command "name"). For example, enter

(c:acad_command "_plot") for the CAD Plot command in dialog mode. There are several examples of this included with the Quick Keys defaults.

If you intend to make changes in an item, use standard editing procedures, including the use of arrow keys along with the tab key and/or pointer movements to make changes. Pressing enter on either field will have no effect on the item in the list.



Changes made to items are automatically changed in the list, you must use the **OK** button to record changes to be saved.

When a new item has been created in the edit fields, you must click the **Add** or **Insert** button to add the item to the list and type in your key, command and description in respective columns.

Highlight the item you wish to delete in the list box, then press the **Delete** button to remove it from the list. In the event you accidentally delete items, simply choose Cancel to exit without saving changes.

At any time during processing of Quick Keys, you may choose the **Sort** button to sort the list of Quick Keys or left click on respective column header to sort it using key, command or description. The most common use of sort will be after adding several new items to the list.

The buttons on the bottom row are used to control files. Changes made to the Quick Keys table are stored when exiting the dialog with the OK button. The original key definition file supplied is called Carlson.QKS. However, when you make changes for the first time and use the OK button to exit the dialog, the program writes changes to a file called CUSTOM.QKS. The Quick Keys editor looks for the existence of the file CUSTOM.QKS when loading, and will use this file when available. This approach will shelter your (CUSTOM.QKS) from overwrites if you reinstall Carlson.

When all changes are complete, choose the **OK** button to save changes, which will automatically build and load the run time file. Any Quick Keys added or updated will be immediately available at the command prompt. However, if a key definition was deleted, it will not be removed from memory until another drawing is loaded or you begin a new drawing.

The **Cancel** button exits the Quick Keys dialog without making any changes to your system.

This **Report** option is used to obtain a printed list of the Quick Keys currently loaded. First consider sorting the list,

then use this option and enter a filename. The program will write the list to a file that can be brought into any editor or word processor, then printed.

The **Save As** button can be used to save the current quick keys as a QKS file that can be distributed to other computers and can be loaded using **Load** button.

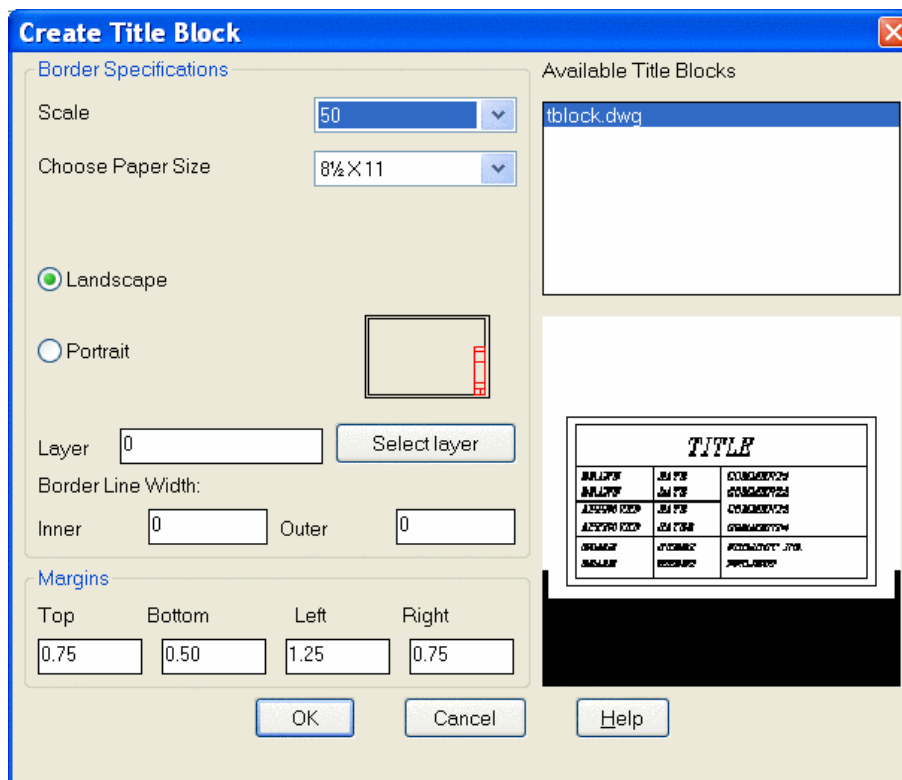
Pulldown Menu Location: Settings

Keyboard Command: quickkey

Prerequisite: None

Title Block

This command draws a border and title block for the selected sheet size. At the top of the dialog, choose your horizontal scale and sheet size. The *other* choice at the bottom of each list will allow you to add your own scale or size if yours is not listed. Anything added to these lists will be retained for future use. Next, choose either "landscape" or "portrait" format. A blue rectangle next to this choice shows you the difference. Below this, you can choose what layer to draw the border and title block on. The margins to use are specified next at the bottom of the dialog. On the right hand side of the dialog, you can choose from several title blocks. As you choose each one, a preview will be shown below this list. This routine looks for all drawings named "tblock" in the \SUP directory. If you want to add your own title block, simply create a new drawing (or copy an existing one) in the \SUP directory and give it a name that starts with tblock. Example: tblock22.dwg and tblock-Jones.dwg are both valid names for this routine, but "MyTitleblk.dwg" is not. After you have made all your decisions in the dialog box, press OK. Depending on your current zoom level, your drawing may be zoomed out to allow you to see the entire area that will be covered by the drawing border. At this point, you have the border attached to your cursor and it is waiting for you to pick a point for insertion. As soon as you do this, a secondary dialog will appear for you to fill out the attributes associated with the particular title block you selected.



Enter Attributes [?] [X]

Block name: tblock

TITLE Job1

COMMENT1 Maysville

COMMENT2 Kentucky

COMMENT3

COMMENT4

PROJECT NUMBER 123

DRAWN BY Carlson

DRAWN DATE 1/1/1

OK Cancel Previous Next Help

<i>Job 1</i>		
<i>DRAWN</i> Carlson	<i>DATE</i> 1/1/1	<i>Subdivision Plat 2</i>
<i>APPROVED</i> 1/1/1	<i>DATE</i> 1/1/1	
<i>SCALE</i> 1:50	<i>SHEET</i> 2	<i>PROJECT NO.</i> 123

Pulldown Menu Location: Settings
Keyboard Command: tblock
Prerequisite: Set horizontal scale in Drawing Setup

Mortgage Block

This command draws a personalized title block for a mortgage survey. You may select an 8½" x 11" sheet, an 8½" x 14" sheet, or define your own sheet size. The dialog box allows the user to edit all block information and input unique data for every layout. The mortgage block drawing is called from the *mortgage.dwg* file located in the \sup directory and can be easily opened and edited within AutoCAD, allowing for the user to alter the size, text, or any other aspect of the drawing to fit the user's particular needs. However, this is usually unnecessary since the original .dwg file places this block for a standard 8½ x 11 ratio drawing. In addition to the block, the user can include the inputs and prescribed text for a Flood Note, which is placed in the bottom left hand corner of the drawing. You may also select a custom drawing file for your flood note. All inputs are saved and recalled from a *mortgage.ini* file located in the \User directory.

The *LIMITS* of the drawing can be set to the lower left and upper right corners of the border. After the title block is drawn, the contents can be edited using the *Attribute Edit* command under the Edit menu.

Title Block	
Scale	1"=50'
Date	04/29/05
LT#	
Drawn	GAW
Checked	ABC
Job#	123
Title	MORTGAGE INSPECTION
Comment1	Prepared by
Comment2	Company Name
Comment3	Company Name
Comment4	Company Name
Comment5	Address
Comment6	City, State ZIP

Paper Size	
<input checked="" type="radio"/>	A1 (8.5x11)
<input type="radio"/>	A1 (11x8.5)
<input type="radio"/>	Legal1 (8.5x14)
<input type="radio"/>	Legal2 (14x8.5)
<input type="radio"/>	User Defined
<input type="checkbox"/>	Set Limits

Flood Insurance Note	
<input type="checkbox"/>	Include Flood Note
Select Flood Note	
Default Flood Note	
Zone	
Panel No.	
Effective Date	
<input type="checkbox"/>	Special Flood Hazard Area

Buttons: OK, Cancel, Help

Pulldown Menu Location: Settings

Keyboard Command: mortgage

Prerequisite: Set horizontal scale in *Drawing Setup*

Rescale Drawing

This command resizes selected text, symbol and block entities within the drawing by comparing the existing drawing scale factor to a new scale factor. Entities are scaled from their individual insertion points. Lines and polylines are not scaled.

Prompts

Old Horizontal Scale: 20

New Horizontal Scale: 30

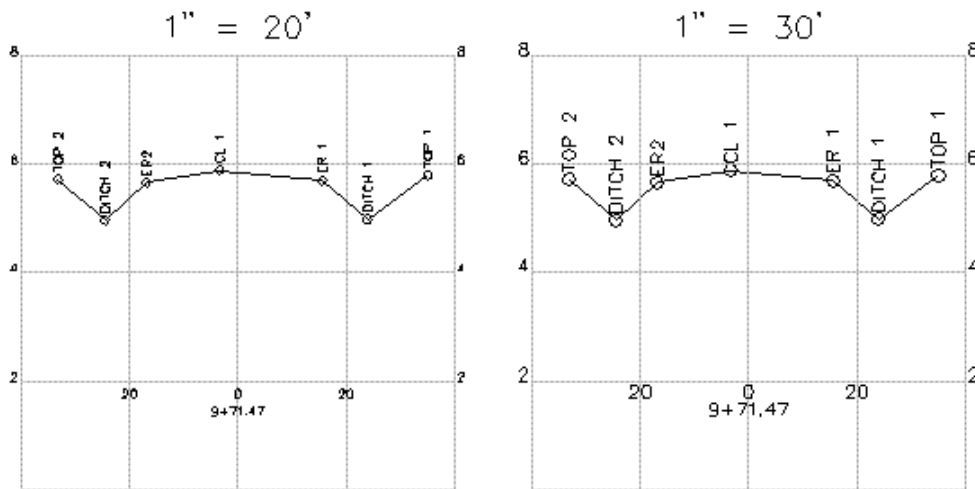
Select text, symbols, dimensions and blocks to scale.

Select objects: select objects by window, crossing or by typing "all" at the command prompt, and press Enter

Number of symbols and blocks changed > 7

Testing Entity > 41

Number of text entities changed > 20



Pulldown Menu Location: Settings

Keyboard Command: scaledwg

Prerequisite: Drawing entities to be scaled

Set/Reset X-Hairs

Set X-Hairs sets the crosshairs either to align with the selected line or polyline or to a user-specified slope. *Reset X-Hairs* restores the crosshairs alignment to horizontal.

Pulldown Menu Location: Settings > Crosshairs

Keyboard Command: setxhairs, resetxhairs

Prerequisite: Line entity

Save/Load Tablet Calibration

A common problem with calibrating maps on a large format digitizer is that if you leave the current drawing session, AutoCAD forgets the tablet calibration. Tablet save can be used to save the calibration when a drawing is taped down properly. This calibration file can be restored at any time later and be accurate so long as the drawing did not move on the tablet.

Save Configuration Procedure:

- 1) Command: TABSAVE
- 2) Designate filename (*.TCF) to save configuration into.

Restore Configuration Procedure:

- 1) Command: TABREST
- 2) Select filename (*.TCF) to restore configuration from.

Pulldown Menu Location: Settings > Tablet Calibration

Keyboard Command: tablet1, tablet2

Prerequisite: None

Create AutoCAD Icon

This command will create an icon on your desktop to launch plain AutoCAD.

Many people assume that the stock AutoCAD icon will launch plain AutoCAD. This is not always true. The problem occurs because, if no profile is specified, AutoCAD always starts with the last used profile. If you run Carlson, then exit and then execute the stock AutoCAD icon (which does not specify a profile), Carlson loads anyway.

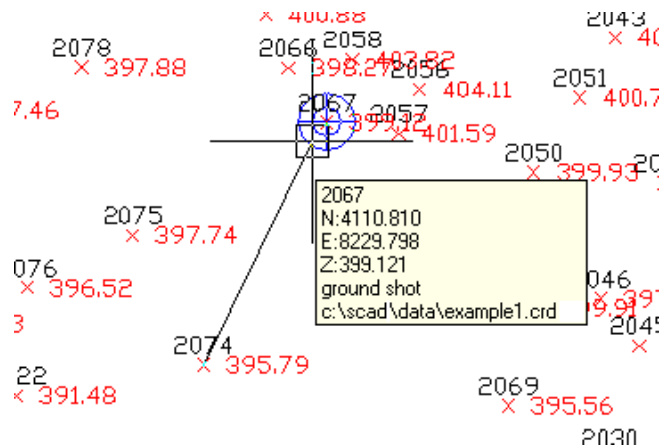
When you run this command, an AutoCAD profile called Vanilla is created, and an AutoCAD icon is created on the desktop that specifies this startup profile.

Technical Note: In its attempt to create the vanilla profile, this routine removes references to Carlson from the support file search path, and replaces the Carlson menu with the stock AutoCAD menu. If the program cannot locate the AutoCAD menu (due to it being deleted or moved), you may have to create the icon manually, as outlined in the Carlson Software Knowledge Base on-line.

Pulldown Menu Location: Settings

Point Object Snap

When this toggle is turned on, you can move your cursor near a Carlson point and snap to the actual coordinates of the point without having to use the AutoCAD *NODE* snap. Point Object Snap can be used alone to display the point information or it can be turned on and used during other commands. In the example illustration, the *2DP* command (2D polyline) has been started and the first point picked was point number 2074. As the cursor nears point number 2067, the point snap marker appears and the point information is displayed, click the mouse and the next polyline vertex will snap to the coordinates of point 2067.



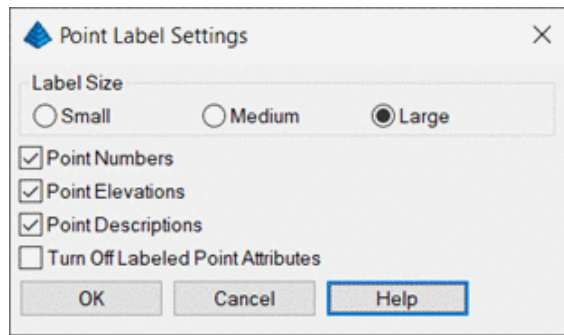
Pulldown Menu Location: Settings

Keyboard Command: 'pointsnap

Prerequisite: None

Point Label Settings

When launching this command, all Carlson points in the drawing that have attributes showing will have those attributes replaced with a dynamic label that retains its height while zooming in and out of the drawing. This is so the text is always readable regardless of what zoom level you are at. The label will display using the font based according to the drawing's current text style.



Label Size There are 3 sizes (Small, Medium and Large) that represent 1, 2 and 3% of the total height of the screen. Choose your size preference that works best for your screen when viewing the labels.

Point Numbers Check this box to hide any point number attributes and to replace with a dynamic point number label.

Point Elevations Check this box to hide any point elevation attributes and to replace with a dynamic point number label. This does not replace elevation labeled as just text.

Point Descriptions Check this box to hide any point description attributes and to replace with a dynamic point number label. This does not replace Carlson Notes.

Turn Off Labeled Point Attributes Check this box to hide the point attributes so you are only seeing one set of labels at a time - either the dynamic point label or the regular point attribute. Unchecking this box will unhide the existing point attributes.

The Point Label is used purely for viewing while in the drawing and will not affect plotting or how points print or export. Certain point attributes may be hidden while in effect, but will still plot based on your layer settings.

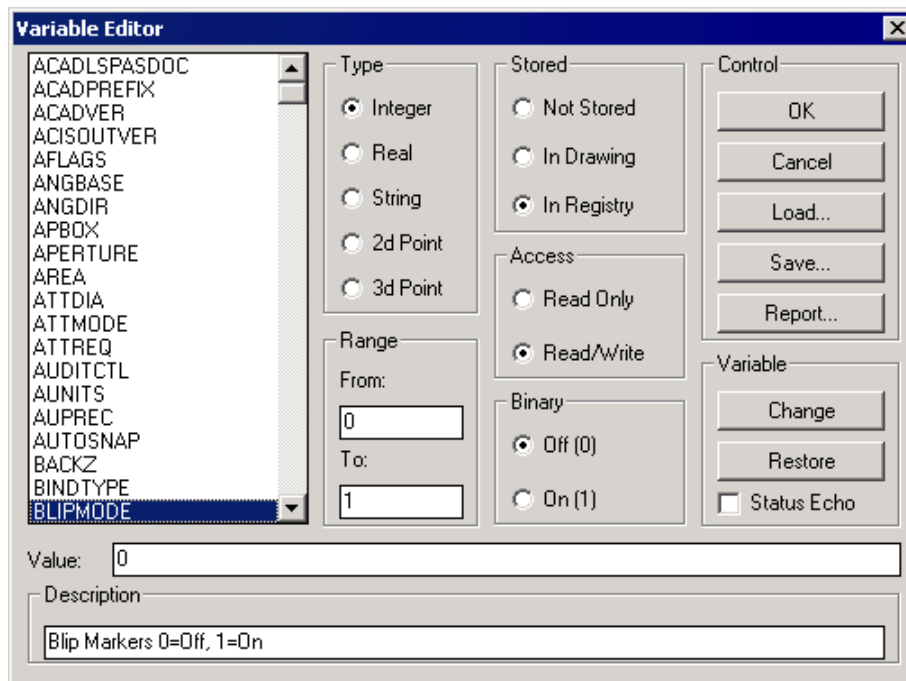
Pulldown Menu Location: Settings

Keyboard Command: pointlabel

Prerequisite: none

System Variable Editor

The AutoCAD/IntelliCAD engine stores the values for its operating environment and some of its commands in system variables. Each system variable has an associated type: integer, real, point, switch, or text string. This command allows you to list or change the values of system variables.



- **List Box:** Contains a list of the variables associated with the currently running version of AutoCAD. There are more items than will display on the list box, use the scroll bar to move up and down through the list. Picking on an item in the list box makes it the current item, causing the information about the item to be displayed, and can be affected by most of the edit commands explained below.
- **Edit Field:** When an item on the list box is picked, its current setting is displayed in the edit field. If you intend to make changes in an item, use standard editing procedures including the use of arrow keys and/or pointer movements to make changes. Once changes have been made, you must use the CHANGE options explained below to effect changes. Pressing enter at the edit field will have no effect on the item in the list. If the item selected is a read-only variable, the edit field will be grayed-out and will not allow input.
- **Description:** When an item on the list box is picked, its definition is referenced and displayed in this field. This can be a benefit in learning the uses of the assorted system variables. This is a display only field, so you can't change the description given.

Under Type Group, the type of variable will be displayed indicated by one of the radio buttons. Each of these types are explained below for your benefit. For additional information on variable types used by AutoCAD, obtain and consult a source of AutoCAD documentation.

- **Integer:** Defined as a whole number in the range from -32767 to +32768, no decimal value accepted.
- **Real:** Defined as a real number in the range from -1.797E+308 to +1.797E+308, with extreme decimal accuracy maintained.. Some real variables have a smaller range than previously stated.
- **String:** Defined as a sequential array of characters in the range from 0 to 65535 characters, with a range of ASCII (0-255). Numbers can be included in strings, even though they have no mathematical significance.
- **2D Point:** Defined as a list of two real numbers in the range from -1.797E+308 to +1.797E+308 separated by a comma, having extreme decimal accuracy maintained. Always maintain the X,Y format, one (and only one) comma must be used, separating the X and Y.
- **3D Point:** Defined as a list of three real numbers in the range from -1.797E+308 to +1.797E+308 separated by commas. While editing a 3D point, you must always maintain the X,Y,Z format, two (no less or no more), commas must be used, separating the X and Y and Z values.

Under Range Group, the variable displayed will usually have a range displayed. The FROM value indicating the minimum, and the TO value being the maximum value accepted.

Under the Store Group, depending on the type of variable, AutoCAD may store the value in the drawing or the configuration file, or it may not be stored. Each of these types are explained below for your benefit.

- **Not Stored:** Some variables, such as PLATFORM and CDATE, are not stored because they are system interdependent.
- **In Drawing:** Most variables are stored in the drawing, making the drawing format more personal than just a database of objects. This allows you to open a drawing and have it behave just as though you had never left it.
- **In Config:** These are variables that remain the same regardless of the drawing opened. APERTURE and PICK-BOX are just two examples of variables stored in the configuration file.

Under Access Group, depending on the type of variable, AutoCAD may not allow you to make changes to it. Each of these types are explained below.

- **Read Only:** Some variables, such as PLATFORM and CDATE, are read-only and therefore cannot be changed. Read-Only variables are marked and the edit field will be grayed indicating that you can't change the variable.
- **Read/Write:** Most variables are read/write and can be changed. These variables are marked and the edit field will be active so you can change the variable.

Under Binary Group, depending on the type of variable, the value may be off or on, yes or no. If the variable type is not binary, this group will be grayed out entirely.

- **Off (0):** Indicate an off condition. Some variables, such as ATTREQ, are simply on or off toggles. You may change a binary item by clicking in this group to change the variable, or changing the value in the edit field.
- **On (1):** Indicate an on condition. Binary variables are simply on or off toggles. Their range is from 0 to 1. You may change a binary item by clicking to change the variable, or changing the value in the edit field.

Control Buttons - These buttons are the main controls in the use of the Variable Editor. Each buttons purpose is explained below.

- **OK:** Used to accept the changes made during the variable editing process, returning you to the command prompt with changes in effect.
- **Cancel:** Used to cancel the changes made during the variable editing process, returning you to the command prompt without the changes in effect.
- **Load:** Used to load a saved set of system variables. This allows you to create a drawing, save the system variables, open a second drawing, and load those variables into that drawing. Read-only variables are skipped.
- **Save:** Used to save the current system variables to a disk file. All system variables are stored to the file, even those that are marked as read-only.
- **Print:** Used to print the current system variables. After choosing this option, you will prompted for an output filename, then the program will proceed to write the system variables to the file. This file can be loaded into any editor or word processor, edited and printed.

Variable Buttons - These buttons are used to control the changes in variables, while using the Variable Editor. Each buttons purpose is explained below.

- **Change:** Used to execute the changes typed into the edit field. You must use this button, simply pressing enter will not make the change.
- **Restore:** Used to cancel the changes typed into the edit field. If you make a mistake or change your mind while making changes in the edit field, press this button to restore the edit field to the value before editing.
- **Status:** Used to determine if the program will echo the status of changes being made to the command area. If this toggle is on, any changes made from the dialog will echo the change. Also if a stream of change commands is being read from a file, and the toggle is on, the changes taking place will be displayed.

Note: This command displays many more system variables than are found in the Systems Variable Chapter, which contains a list of **supported** system variables. Modification of any system variable other than the supported ones found in the Systems Variable Chapter is done at your own risk, and may result in program errors requiring a re-installation of Carlson.

Pulldown Menu Location: Settings

Keyboard Command: VAREDIT

Prerequisite: None

Select CAD Default Settings

This command sets some important CAD user-interface settings. The buttons across the top set all the settings in the dialog.

Carlson Recommended: These are the commonly used settings.

Classic Settings: These have the user-interface like the old versions of the program.

New Standard Settings: These have the new style user-interface settings.

Custom: Let's you choose each setting.

For each user-interface feature, the dialog shows the CAD variable name and value. You can type in these variable names at the Command: prompt to set that feature. Here are the settings in this dialog:

Start Page - Controls whether to use the Start Page tab when the program starts up.

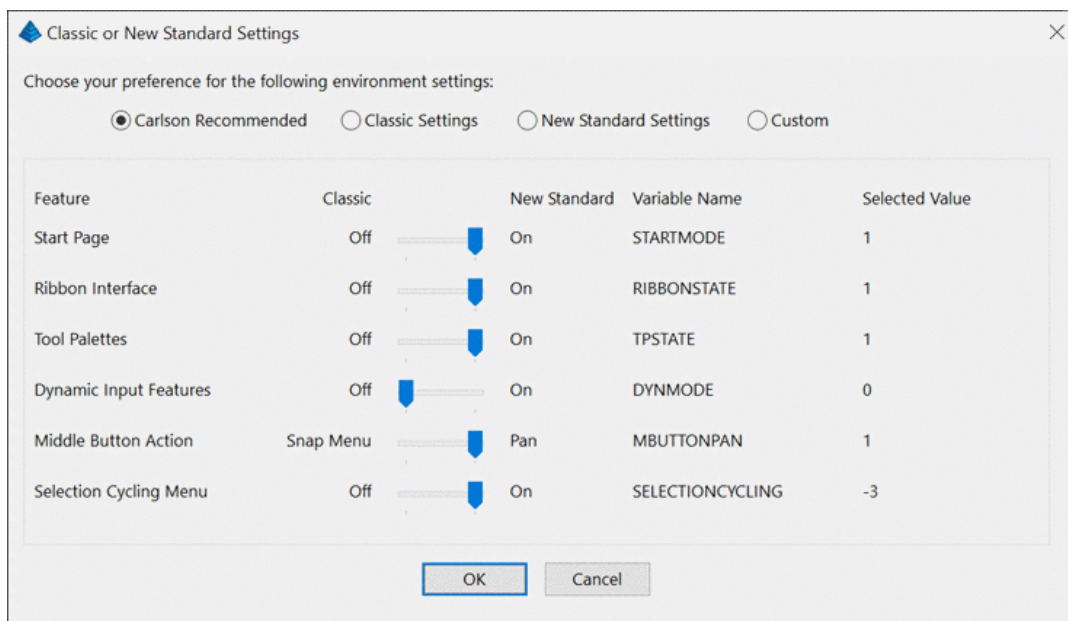
Ribbon Interface - Controls whether to show the ribbon of toolbar icons and tabs.

Tool Palettes - Controls whether to show the tool palettes interface for selecting commands.

Dynamic Input Features - Controls whether the input prompting is show in the display next to the cursor.

Middle Button Action - Choose between using the mouse middle button for the snap menu or pan action.

Selection Cycling Menu - Controls how to select entities when picking a position with multiple entities under the cursor.



Pulldown Menu Location: Settings

Keyboard Command: options_default

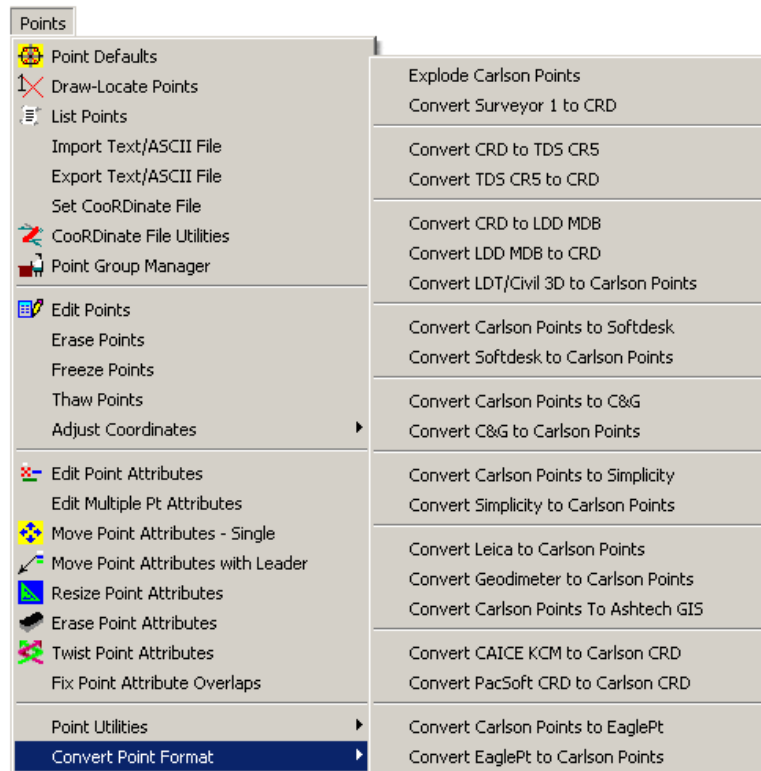
Prerequisite: None

Points Menu

All of the routines in this menu operate on points in a Carlson coordinate (.CRD) file. Coordinate files are binary files that contain point numbers, northings, eastings, elevations and descriptions. The Carlson coordinate database (.CRDB) is based on SQLite and supports point numbers and descriptions up to 255 characters. Alternately, C&G

CRD and CGC files, LandDesktop MDB files or Simplicity Systems ZAK files can be used in place of the Carlson CRD file. All routines in this menu will read from, and write to, these types of point data files. At any given time, there can only be one active coordinate file. If a command is initiated that requires a coordinate file while one is not one set, Carlson will prompt for a coordinate file name. From that point on, this is the current coordinate file. Another coordinate file can be used by choosing Set Coordinate File or *Open CRD File* in Coordinate File Utilities.

Whenever you asked for point numbers, you can enter any combination with commas and dashes or type **ALL** to use all points. For example 1-3,7,20-23 would act on points 1,2,3,7,20,21,22,23. Coordinate files have either numeric or alphanumeric point numbers. Alphanumeric point numbers consist of nine or less digits and letters (*i.e.* point# 7A). The type of point number format is set when the coordinate file is created. This setting is found under *General Settings* in Carlson Configure. This setting only affects new coordinate files.



Each point is drawn by three entities:

1. point block
2. point node
3. symbol

The point block is an INSERT entity with PNTNO, PNTLEV and PNTDESC attributes. These attributes represent the point number, elevation and description respectively. The point node is a POINT entity and is used for picking the point with the NODE snap. The point node is also used as the X, Y, Z coordinate in Triangulate & Contour. The symbol can be any symbol defined in the Symbol Library (use SPT0 for no symbol). Since points use Carlson point symbols, the CAD system variables PDMODE and PDSIZE should usually be set to 0.

The points in the drawing can be linked to their coordinates in the coordinate file. The link updates the coordinate file when a point is modified in the drawing. For example, when points are moved with the Rotate Points command, their coordinates will be automatically updated in the coordinate file. To update the coordinate file without this automatic link, you can run the command *Update CRD File from Drawing* in Coordinate File Utilities. The linking option is called *Link Points with Coordinate File* (currently only available in AutoCAD) and can be set under *General Settings* in Carlson Configure. This setting does not affect points currently in your drawing, only points drawn after you change this setting.

Each point in the coordinate file has room for a 32 character description. To have a longer description, an associated point note file can be used. The note file has the same file name as the coordinate file with a .NOT extension and is

stored in the same directory as the coordinate file. For example, **survey.note** would be the note file for **survey.crd**. The note file is a text file that stores a point number together with the additional notes for the point. There is no limit to the length of the note. Notes can be added to points using the Edit Points command. The List Points command can be used to print out the notes.

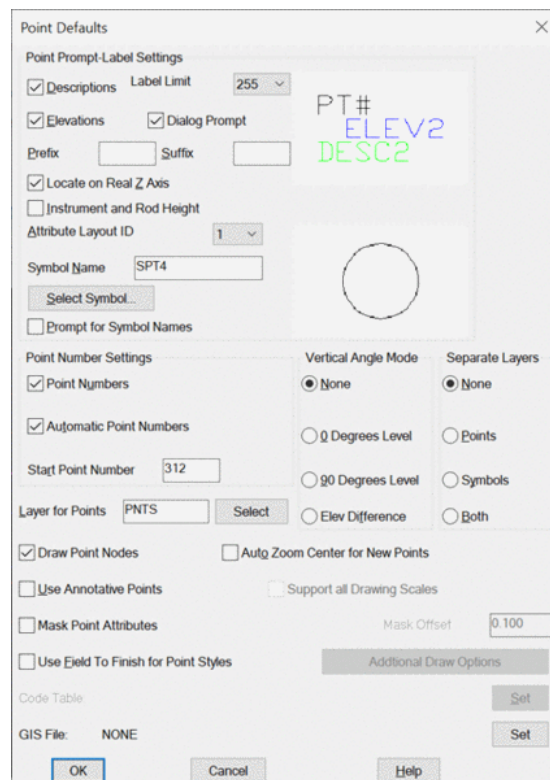
For each point, the point attribute block, node, and symbol can be bound together into a "grouped" entity. This means that if you choose to use the Move command (or other CAD tools) the entire collection moves together. This is done using the grouping functionality.

To disable this system altogether, navigate to Carlson Configure > General Settings and turn off the toggle for *Group Point Entities*. If you need to temporarily disable grouping in a drawing, you can use the AutoCAD toggle for grouping, which is *Ctrl-A* (holding down the 'Ctrl' key and then pressing the letter 'A' on the keyboard activates this two way toggle and the current status will be echoed to the Command prompt area).

Carlson points include additional information on each element that makes up the point collection (attribute block, node and symbol). This information allows Carlson to know such things as which coordinate (.CRD) the point came from. Commands like Drawing Inspector can then display the point information for the point entities. This also makes it easier for Carlson to identify which drawing objects belong to a point, making commands like Edit Point Attributes a "double-click" pick association instead of a selection set.

Point Defaults

This command sets Carlson point options for drawing point entities.



Descriptions: Specify whether you are prompted for a point description when creating points and whether the point descriptions are labeled in the point block.

Label Limit: Sets the maximum length for description attribute labels.

Dialog Prompt: When creating new points, this option controls whether to prompt for descriptions and elevations

in a dialog or at the command line.

Elevations: Specify whether you are prompted for a point elevation when creating points and whether or not the point elevation is labeled with the point.

Prefix/Suffix: Indicate a desired prefix or suffix that should be included with the elevation label.

Locate on Real Z Axis: When checked, points are drawn at their actual elevation, otherwise, points will be drawn at Z=0.

The following tables illustrate the effects of Elevation vs. Real Z settings:

	Elevations Yes	Real Z Yes
Picked Point	Labels point, Prompts for elevation	Uses elevation for z coordinate
Point Number	Labels point, No Prompt	Uses z coordinate from file
Effects of Elevation Settings - Scenario 1		
	Elevations Yes	Real Z No
Picked Point	Labels point, Prompts for elevation	Uses 0 for z coordinate
Point Number	Labels point, No Prompt	Uses 0 for z coordinate
Effects of Elevation Settings - Scenario 2		
	Elevations No	Real Z Yes
Picked Point	Labels point, No Prompt	Uses z coordinate of picked point
Point Number	Labels point, No Prompt	Uses z coordinate from file
Effects of Elevation Settings - Scenario 3		
	Elevations No	Real Z No
Picked Point	No Label, No Prompt	Uses 0 for z coordinate
Point Number	No Label, No Prompt	Uses 0 for z coordinate
Effects of Elevation Settings - Scenario 4		

Attribute Layout ID: Controls the location of the point number, elevation and description. Up to 10 attribute layouts (0 through 9) are available. See Carlson Knowledgebase Article 779 (Customized Carlson Point Attribute Layouts) for examples and steps of different attribute layout configurations.

Symbol Name: Enter the default symbol name to use. You may also pick the *Select Symbol* button to select a symbol from the Symbol Library.

Prompt for Symbol Names: When checked, you will be prompted for each symbol name instead of using the default symbol.

Point Numbers: When this toggle is OFF, no point number will be created and no points will be stored in the active coordinate file.

Automatic Point Numbers: When enabled, point numbers are numbered sequentially from the *Start Point Number*. If the Start Point Number field is set to 0, no point will be plotted. An exception to this is when you use the Draw-Locate Points command and use the Range option, then a point entity is plotted. When disabled, commands that locate new points will prompt for the point number.

Start Point Number: Specify the starting point number to use.

Layer for Points: Specify the default layer name for Carlson points.

Vertical Angle Mode: Specify how Carlson should prompt you for vertical angles. None means "Do not prompt." This applies to creating points with commands such as Traverse. The vertical angle is used to calculate the point elevation.

Separate Layers: Specify settings for point attribute layers:

- **None** - The point number, elevation and description use the layer names PNTNO, PNTELEV and PNTDESC, respectively.
- **Points** - The point number, elevation and description layers are composed by concatenating the point layer and the string NO, ELEV, and DESC. For example, if the point layer is UTIL then the attribute layers will be UTILNO, UTILELEV and UTILDESC, respectively.
- **Symbols** - The point symbol layer is composed by concatenating the point layer and the string MARK. For example, if the point layer is UTIL then the symbol layer will be UTILMARK.
- **Both** - The point symbol, point number, elevation and description layers are composed by concatenating the point layer and the string MARK, NO, ELEV, and DESC respectively. For example, if the point layer is UTIL then the symbol/attribute layers will be UTILMARK, UTILNO, UTILELEV and UTILDESC.

Auto Zoom Center for New Points: When checked, the drawing will perform a "Zoom Center" around new points to keep the display centered around the current working area during the new point creation process.

Draw Point Nodes: This option controls whether to create a CAD point entity. A full Carlson point consists of the CAD point entity, point symbol and point attribute block that has the point number, elevation and description labels. Normally the CAD point entity is part of this group but is not required.

Use Annotative Points: When enabled, points (usually placed through the Draw Field-to-Finish or Draw-Locate Points commands) will use scale-able annotative properties.

Support All Drawing Scales: When enabled, points placed as annotative entities will make use of all annotation scales currently defined within the drawing.

Mask Point Attributes: This option hides the drawing behind the point attributes when the points are drawn. The Mask Offset controls the buffer area around the point attribute. This offset value is scaled by the drawing horizontal scale. You can also leave this option off and apply the mask later by using the Mask Point Attributes command under the Points > Point Utilities menu.

Use Field to Finish for Point Styles: Allows you to use the code definitions from the designated Feature Library Definition (FLD) table specified below. For example, when creating a point with description of "FH" (for Fire Hydrant), Carlson would look up "FH" in the Field to Finish table and will use the field code definitions to establish the parameters of the point being created instead of the definitions defined in within *Point Defaults*. The Field To Finish settings under **Additional Draw Options** are described in the Field To Finish topic in the manual.

Code Table: This option lets you specify the Feature Library Definition file (FLD) used by the Use Field to Finish controls specified above.

GIS File: This option lets you specify a GIS file to be used when creating new points. The GIS file contains a list of fields to prompt for. For each point that is created, the program will prompt for these fields and store the results to the note file (.not) associated with the current coordinate file.

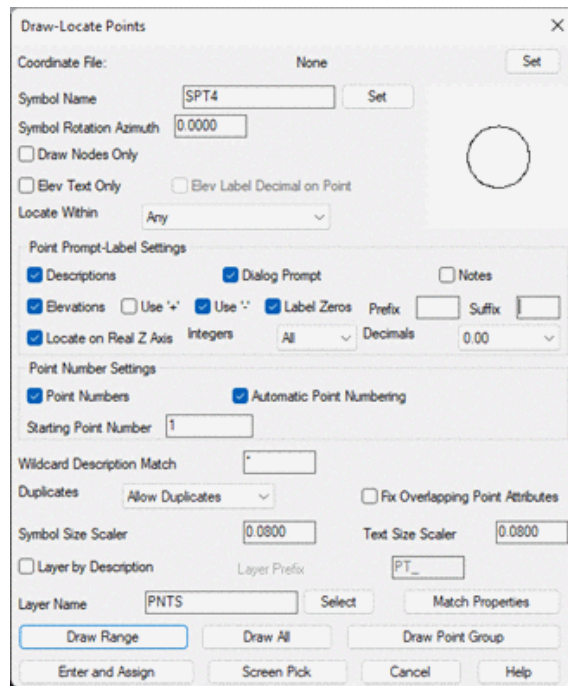
Pulldown Menu Location(s): Points

Keyboard Command: ptsetup

Prerequisite: None

Draw-Locate Points

The Draw-Locate Points dialog box allows you to insert either new or existing points into the drawing. You can create new points either by picking points on the screen, or by entering northing and easting coordinates. You can also place existing points by entering point numbers which reference the current coordinate file. You are prompted to choose a coordinate file if no coordinate file is current.



The **Coordinate File** at the top of the dialog shows the current file which can be changed with the Set button.

The name of the symbol file is displayed in **Symbol Name**. You can choose a different symbol by clicking Select. The selected point symbol is displayed on the right.

Symbol Rotation Azimuth is the rotation angle that is used for the point symbols. This angle is used in a counter-clockwise direction relative to the current twist screen.

Layer by Description inserts the points in the layer named by the point description. Using Layer by Description organizes the points by description and allows for layer management. For example, you can use the Isolate Layers command to show only points on a certain layer. If you include an invalid layer character in the description, the layer name stops at the bad character. A point description of "UP / 105" would use layer "UP", for example. The **Layer Prefix** is added to the beginning of the layer name. For example, a Layer Prefix of "PT_" and a point with the description "EP" would use the layer "PT_EP". Layer Prefix is optional. It allows all the point layers to be grouped.

Draw Nodes Only inserts only a point entity (the node) and not the point block and symbol. This option is most useful when you have a lot of points to insert, because inserting only the nodes is faster than inserting nodes with the point block and symbol. Commands such as Triangulate & Contour and Make 3D Grid File can use these points, and do not need the point block and symbol.

Selecting **Elev Text Only** draws text of the point elevation without the point block, symbol, or node. The decimal place of elevation text is placed at the northing and easting point location.

Locate within Polyline inserts only the points that are inside a closed inclusion polyline. The command prompts you to select a closed inclusion polyline and as well as an optional exclusion polyline. All the points in the current coordinate file are checked. Any points that are located within the inclusion polyline and outside the exclusion polyline are drawn.

Locate within Distance inserts only the points that are within a specified distance from a reference point. The command asks you for the reference point and the search distance. All the points in the current coordinate file are checked. Any points that are located within the search distance of the reference point are drawn.

Locate within Window/Coord Range inserts only the points that are within the specified window or range of northing, easting, and elevation. The command prompts for the minimum and maximum northing, easting, and elevations. These values default to the actual minimum and maximum in the coordinate file. Then the command prompts for the point number range of points to check. The points that fall in both the point number range and the

coordinate range are drawn.

Under **Point Prompt-Label Settings**, you determine attributes for which you will be prompted.

Descriptions determines whether you are prompted for descriptions for each point when creating new points. When you are placing both new and existing points, Descriptions determine whether this attribute is labeled with the point inserts.

Notes works with the note file (.not) associated with the current coordinate file. The note file contains unlimited point descriptions in addition to the fixed 32-character point descriptions in the coordinate file. When you create points with Notes on, the command will prompt for point notes to be stored with the point. When you draw existing points with Notes on, any notes for the points are drawn as text entities below the point description.

Elevations determines whether you are prompted for elevations for each point when creating new points. When you are placing both new and existing points, Elevations determine whether this attribute is labeled with the point inserts.

Use '+' labels the positive elevations with a leading '+'. For example, "+159.43".

Use '-' labels the negative elevations with a leading '-'.

Locate on Real Z Axis determines if the points are placed at their elevations or at zero elevation.

Label Zeros will label points with zero elevation when the Elevations option is on. Otherwise only points with nonzero elevation will be labeled.

Elevation Prefix/Suffix set the prefix and suffix labels to apply for the elevation labels.

Elevation Integers controls the number of digits to display to the left of the decimal point for the elevation label. The All setting will show the full elevation digits. The other settings allow you to limit the number of digits to display for the purpose of reducing the amount of space the elevation labels take up in the drawing. For example, if a site is in the 4000 foot elevation range, then this setting could be set to three digits (000) and an elevation of 4321 would be labeled as 321.

Elevation Decimals sets the number of decimals to the right of the decimal places for the elevation labels.

Under **Point Number Settings**, you determine how points will be numbered.

Point Numbers determines whether the complete point block is drawn or just the symbol and node. When you create new points with Point Numbers off, no points are stored in the current coordinate file, and only the point symbol and node are drawn. When you draw existing points with Point Numbers off, the point attribute block is not drawn and only the point symbol and node are drawn.

Automatic Point Numbering applies to creating new points. With this option active, the command will use the **Starting Point Number** for the first new point. The next point number is automatically incremented. Before storing the point, the command checks whether the point number is used. If the point number is used and point protect is on (set in the Coordinate File Utilities command), then the command will prompt for another point number or to overwrite the point. With Automatic Point Numbering off, the command will prompt for the point numbers.

Determine how the points are to be displayed and in what layer.

With **Wildcard match of pt description**, you can display only points with specific descriptions. This can be thought of as a filter. For example, entering IP would display only points that are labeled with the description IP, or Iron Pin. The default is the asterisk (*). This will display all points regardless of description.

Layer Name allows you to designate a layer for the points to be displayed. You can enter a new name, CLAYER, or choose an existing layer by clicking **Select Layer**. Entry of CLAYER selects the current layer. A Carlson Survey point consists of a block insert with attributes, a point symbol, and a point entity. The point entity is used for picking the point by OSNAP Node in other commands. The block insert includes a point number, elevation, and description. These attributes are in the PNTMARK, PNTNO, PNTELEV, and PNTDESC layers. The points are also in an overall layer as specified in this dialog box. This layer setup allows you to freeze a group of points by the main layer name or freeze point attributes for all the points in the drawing. For example, freezing layer "PNTS"

would freeze all the points in this layer. Freezing layer "PNTELEV" would freeze the point elevation attribute for all the points.

The **Duplicates** option for **Erase and Redraw** will erase existing point entities that match the point numbers currently being drawn. The **Allow Duplicates** option will leave any existing point entities as the specified points are drawn. The **Draw Only New** option will only draw points that don't already exist in the drawing.

Fix Overlapping Point Attributes will detect point number, elevation and description attributes that overlap with other points. Rules can be applied to rearrange the point attributes to avoid the overlaps. A point overlap manager then steps through each overlap for review or manually moving the attributes.

Symbol Size Scaler controls the size of the point symbol and **Text Size Scaler** controls the size of the point attribute labels. The scalers are multiplied by the Horizontal Scale from Drawing Setup to set the size in drawing units.

Match Properties prompts to select an existing point entity and then the program sets the settings in the dialog such as layer and symbol to match the selected point.

Draw Range will draw existing points from the current coordinate file. The Draw Range button will prompt for the point numbers to draw.

Draw All will draw all the points in the coordinate file, and then zoom the extents of the display to show the points.

Draw Point Group will draw a point group with settings that are established in the Point Group Manager.

Enter and Assign can be used to create new points using the point northing and easting. When a grid projection is defined in Drawing Setup, then there is an option to enter the points using latitude/longitude.

Screen Pick allows you to create points by picking the point coordinate on the screen. For example, you could set the Object Snap to EndPoint and pick the end point of a building polyline to create a point at the building corner.

Prompts

To create a new point:

Draw-Locate dialog *choose Screen Pick*

Pick point to create: *pick a point*

Select/<Enter Point Elevation <0.00>: *Enter elevation Press S to select text to set elevation.*

Enter Point Description <>: *Enter*

N: 5106.57 E: 4901.96 Z: 0.00

Enter/<Select text of elevation>: *Select text entity that defines elevation of point.*

To locate a point in the coordinate file (point number 3 in this example):

Draw-Locate Point dialog *choose Draw Range*

Point numbers to draw: *3*

Points Drawn> *1*

Locates point 3.

Point numbers to draw: *1-2*

Points Drawn> *2*

Locates a range of points. From 1 to 2.

Point numbers to draw: *Enter*

Keyboard Command: *lpoint*

Prerequisite: A CRD file and you may want to execute *Drawing Setup* (see the Setting menu) to set the scale and size.

List Points

This command generates a report of point numbers, northings, eastings, elevations and descriptions.

The screenshot shows the 'List Points' dialog box with the following settings:

- Selection method:** Range (selected), Area, Selection Set
- Range Settings:**
 - Highest Point Number: CP14
 - Range of Points: 1-CP14
 - Buttons: All, Point Group
 - Description Match: *
- Report Options:**
 - Report Coordinate Range
 - Report Job Header
 - List Point Notes
 - List CRDB GIS Attributes
 - List SurvCE GIS Attributes
 - Use Report Formatter
 - Include Geodetic Report (with Geodetic Options button)
 - Double Space Between Points
 - Coordinates in Inches: Off
 - Northing/Easting Decimals: 0.00
 - Elevation Decimals: 0.00
- Buttons:** OK, Cancel, Help

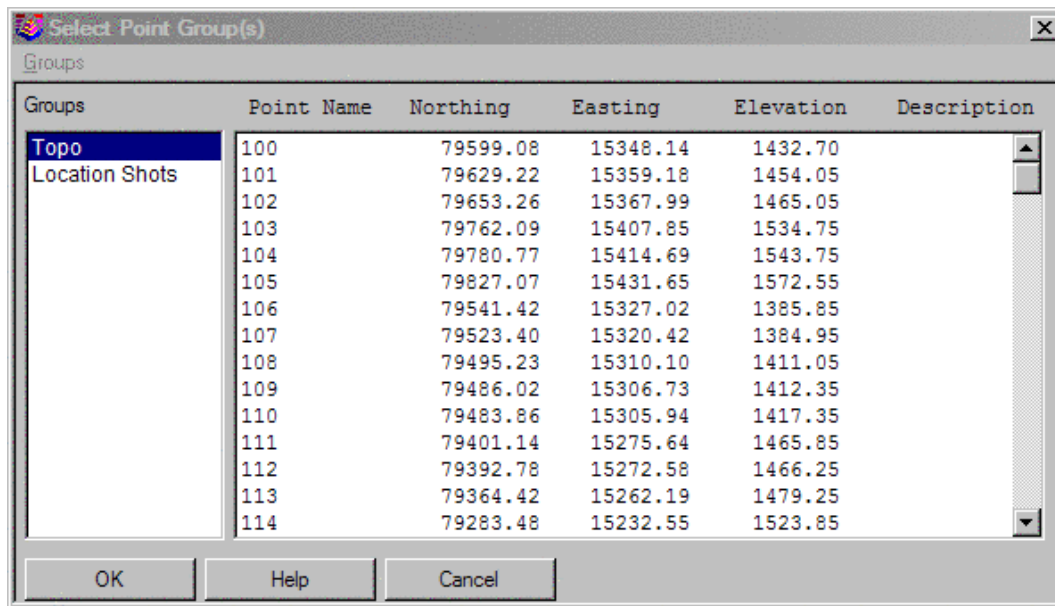
Selection Method-Range allows you to specify the points to list by point number range

Selection Method-Area allows you to select a closed polyline to list all of the points inside of that polyline.

Selection Method-Selection Set allows you to specify the points to list by selecting them from the drawing.

Range of Points: If you are using the Range method, specify the range of points to list here. To quickly specify all points, click the **All** button.

Point Group allows for the selection of a specified group or multiple groups for listing. Standard windows selection tools, ctrl and shift keys, can be utilized for selecting groups.



Description Match: Can be used to filter the point list. For example, entering "EP" for the Description Match would only list those points with a description of "EP". An asterisk (*) is the default setting, it matches any character sequence, meaning no filtering occurs.

Report Coordinate Range: When checked, the point list will include the minimum and maximum northing, easting and elevation.

Report Job Header: Includes the Job Description, Job Number and Job Date in the header of the report. These job values can be set in the Coordinate File Utilities command.

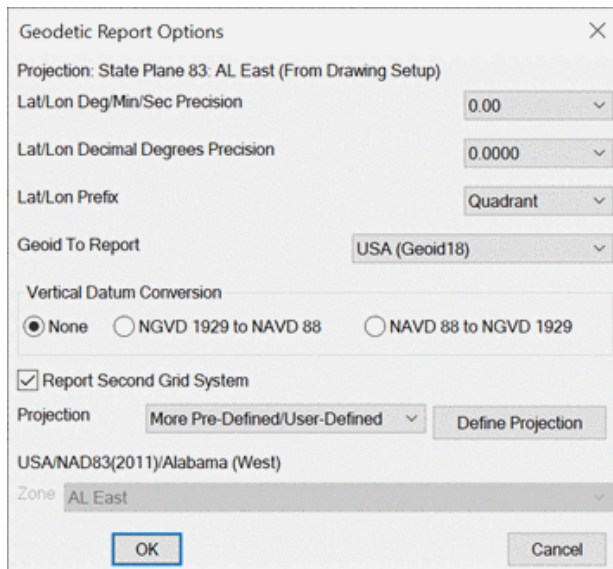
List Point Notes: When checked, any additional point notes assigned to the points will be included in the point list. Point notes can be entered using the *Input-Edit Point* command found in *Coordinate File Utilities*.

List CRDB GIS Attributes: Includes any GIS attributes from CRDB format coordinate files. Use the Point GIS Editor command to edit these GIS attributes.

List SurvCE GIS Attributes: Includes any GIS attributes from the companion VTT file from SurvCE.

Use Report Formatter: When checked, you may customize the fields and layout of the point report using the Report Formatter. The Report Formatter can also be used to export the point report to Excel or Access.

Include Geodetic Report: When Use Report Formatter is on and the Drawing Setup command has a coordinate system defined, this option handles reporting of latitude/longitude and geoids. You can also use this option to report the coordinates in a second grid system.



Double Space Between Points: When checked, the report will be double spaced.

The point list report is displayed in the Standard Report Viewer which can print, draw and save the report file. This report viewer cannot be used to edit the coordinate file. Instead use the *Edit Points* command in the Points menu.

Example of List Points Report:

```

List Points Report
File> C:\Carlson2008\DATA\POINTS.CRD
Job Description>
Job Number> 0.000 Job Date> 06/01/2002
PointNo. Northing(Y) Easting(X) Elev(Z) Description
1          5355.240    5000.000    91.8    CP2
2          5000.000    5000.000    90.0    CP2
1000       5355.236    5000.000    91.8    CK
1001       4941.911    4622.029    91.4    FPC
1002       4952.629    4642.818    90.6    FH
1003       4959.931    4634.440    89.8    TOE1

```

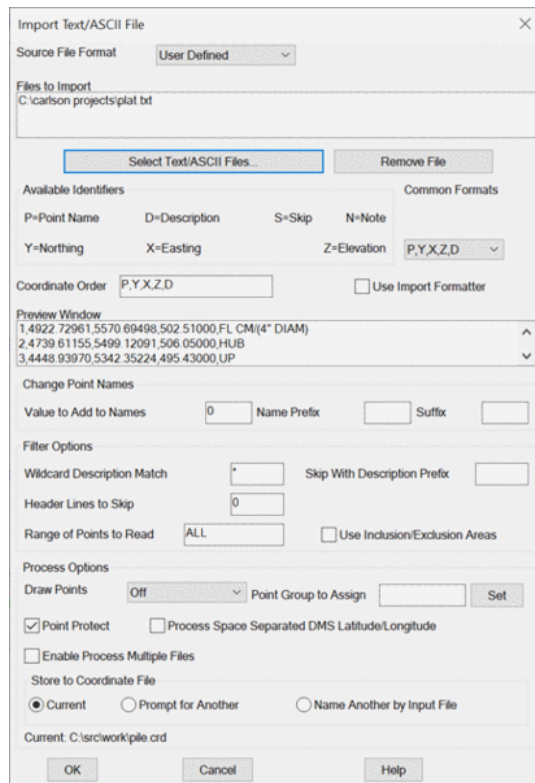
Pulldown Menu Location: Points

Keyboard Command: listpt

Prerequisite: Points in a coordinate file or on the screen

Import Text/ASCII File

This command converts point data from an ASCII text file into a Carlson coordinate (.CRD) file. Each line of the text file can contain any combination of point number, northing, easting, elevation and description. All point information should be on one line with the values separated by a comma, space or other delimiter.



Under the Source File Format setting you can choose from some specific formats or User-Defined. For User-Defined, the format of the text file is specified in the Coordinate Order field where the value identifiers are listed with the appropriate delimiters. For example:

For a text file with northing, easting, elevation and comma delimiters:

5100.0,5150.5,485.1

5127.1,5190.3,487.3

The Coordinate Order would be:

Y,X,Z

For a text file with point number, easting, northing, elevation, description and space delimiters:

1 5000.0 5000.0 490.3 TRAV

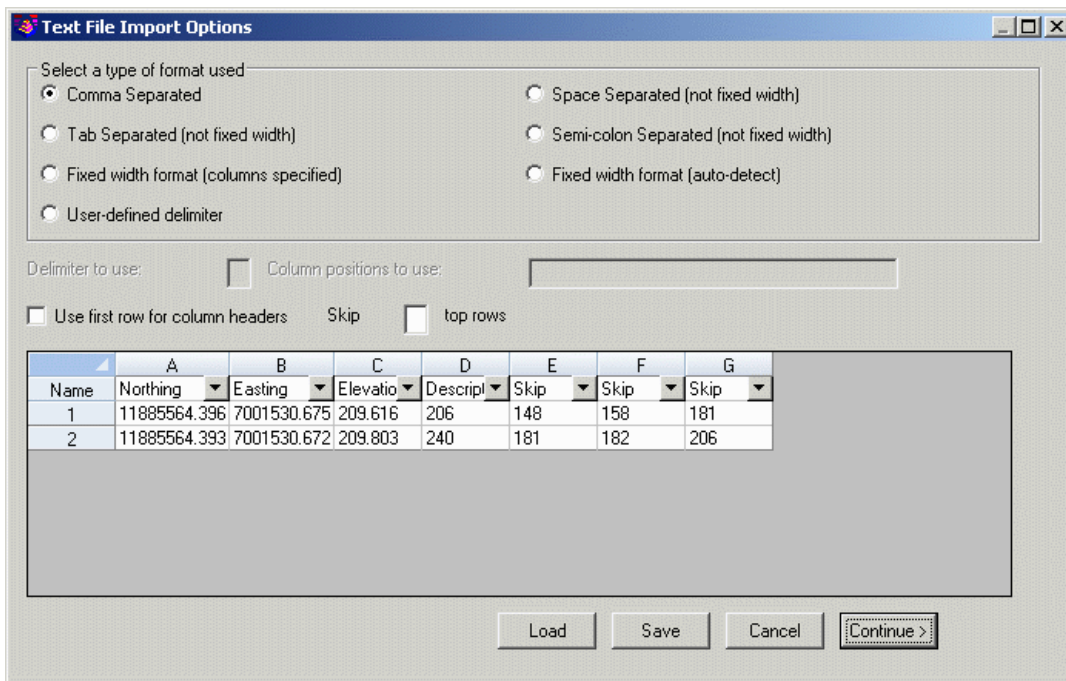
2 5030.4 4930.5 495.5 TRAV

The Coordinate Order would be:

P X Y Z D

Common formats can be selected from the Common Format List. All the lines in the text file should contain only point data and any header lines should be removed. To read the text file, pick the Select Text/ASCII File button and choose the file to read. Then the selected file is displayed in the Preview Window to help with filling out the Coordinate Order. When the Coordinate Order is set, click OK to read the text file.

The Use Import Formatter is an alternative way to define the format of the input file. This method has a heads up interface for selecting the fields for each column of data. This method also supports fixed width, user-defined delimiter and semi-colon delimited files that the Coordinate Order method doesn't handle.

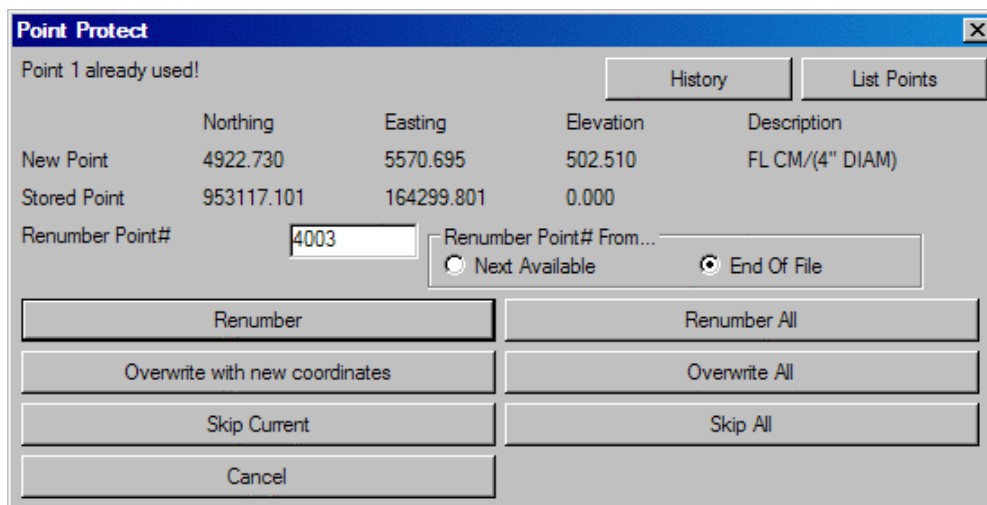


Under Change Point Names, the Value to Add to Point Numbers allows you to renumber the points as they are imported. The Name Prefix and Suffix are added to the point names as they are imported.

Under Filter Options, the Wild Card Descriptions Match allows for only point with matching descriptions to be imported. The Header Lines to Skip value is the number of lines not to be processed at the start of the text file. The Skipt With Description Prefix filters out points with a description that starts with the specified prefix string. The Range of Points to Read is a method to only import points within the specific point number range. The Use Inclusion/Exclusion Areas option prompts for selecting closed polylines to control where to import points.

The Point Group To Assign option will create a point group with the specified name for the coordinate file containing the point numbers imported with Import Text/ASCII File.

With Point Protect active, the program will check if a point number already exists in the CRD before importing the point. If a point conflict is found, you can either assign a new point number or overwrite the old point.



Multiple files can be imported at once. To do this, toggle on the Enable Process Multiple Files option. After selecting the Text/ASCII Files button, you can select multiple files by using the Shift or Ctrl keys while picking files. You can also run Select Text/ASCII Files multiple times allowing for selection of files located in different locations. The files to import are listed in the top scroll display window. The point data from all the import files can be stored to the current CRD file or to separate files for each import file. The separate file option will name the resulting CRD files with the same name as the import file with a .CRD file extension. For example, the import file job125.txt would create job125.crd. The CRD file will be created in the same location as that of the selected text file to import.

Under Process Options, there are choices for selecting the coordinate file to store the imported points. The Current option uses the current coordinate file that is active in the drawing. This coordinate file name is shown at the bottom of the dialog. The Prompt For Another option uses the standard file selection dialog to select the file. The Name Another By Input File uses a coordinate file name with the same name as the input file except for a file extension of .CRD.

The Process Space Separated DMS Latitude/Longitude handles a text file with the degrees, minutes and seconds separated by spaces. The latitude and longitude values get stored to the coordinate file in decimal degrees. Here is an example text file line with the point number (1100), latitude degrees (42), latitude minutes (6), latitude seconds (3.200), longitude degrees (70), longitude minutes (2), longitude seconds (2.090) and elevation (85.245). The Coordinate Order for this example should be set to P Y X Z.

```
1100 42 06 03.200 70 40 02.090 85.245
```

The special formats of SurvCE Job Database .jdb, SDMS .ctl, Leica .d45/.gsi/.raw files, MicroStation .TA2 files, WinCMM .cor and .lev files, TDS .cr5 files, Topobase .ro files, Geodimeter .obs/.raw/.are files, Laser Atlanta .txt files, Trimble .pos files, Zeiss .txt files, Traverse PC .trv files, Maptech, Benchmark .dat files, CAICE/Caltrans .tss files, NLS MMH360 .360 files, EMXS .xng files, and Cadvantage .cog files can be directly imported by choosing that File Format at the top of the dialog.

The Load and Save buttons are for saving and recalling the settings from the dialog using a .rpnt file.

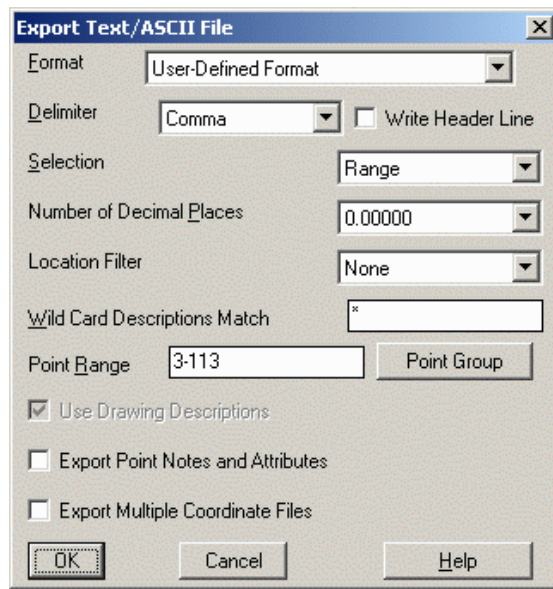
Pulldown Menu Location: Points

Keyboard Command: readpt

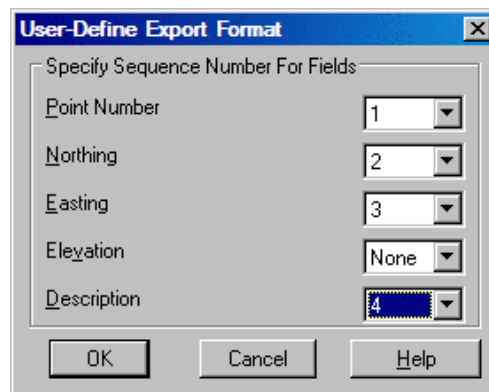
Prerequisite: A text file to read

Export Text/ASCII File

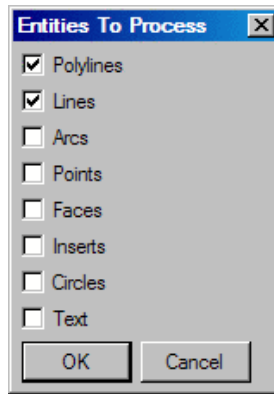
This command outputs point data from the current Carlson coordinate file to an ASCII text file formatted according to a variety of options presented in the form of a general dialog.



Format. Specify the type of file to write from the drop down list. There are several variations on point number, northing, easting, elevation and descriptions as well as specific formats for Leica, Geodimeter, Zeiss, Maptech, Idan REG, NLS MMH360, D45, Cadvantage, Multiplane and SDMS CTL formats. In addition there is a User-Defined Format option to define the order of the fields output. When using the User-Defined format, after selecting OK, the User-Define Export Format dialog will appear. On this dialog, specify the order of the fields by defining a number sequence in each field. You can skip fields and omit data in the output file by leaving None in the sequence field for this data:



Selection. There are four Selection Methods provided for the data to export. Specify either Range, Screen Points, Screen Entities or Pick Points in the Selection Field. A Range selection is a user specified range such as 1-10,30-50. A Screen Points selection is made by selecting points from the screen area. The Pick Points method prompts to select the points to export from the drawing. The Screen Entities option allows for selection of polylines, lines, arcs, points, faces, inserts and text to export point data from. When the Screen Entities option is selected, the following dialog box will display allowing for the specification of the type of entity to export data from:



Delimiter. Select the desired field delimiter of the export file as either Comma or Space from the drop down list. If a header line is to be included, enable the check box.

Number of Decimal Places. Select the desired number of digits to be included in the mantissa of all output ordinates.

Location Filter. Choose from filter methods of within inclusion perimeter polyline, by coordinate window or center within radius from a center point.

Wild Card Descriptions Match. A description filter is also available for exporting only points from the range or selection set with certain descriptions.

Export Point Notes and Attributes. Notes associated with the points may be included in the export by enabling the check box.

Export Multiple Coordinate Files. Enable this check box to specify multiple CRD files to apply the selection criteria against. If enabled, an additional dialog will be presented from which you can browse, select, and remove as many CRD files as desired. In this mode, you can use the Write To Single ASCII File to output the multiple coordinate files into one combined text file.

Point Group. Displays the Point Group manager dialog from which you may define, modify, and select one or more Point Groups to define the points to be included in the export.

After selecting the OK button, a final dialog appears that allows you to specify a new file or to append data into an existing file. The standard file selection dialog allows you to specify the export file name.

Pulldown Menu Location: Points

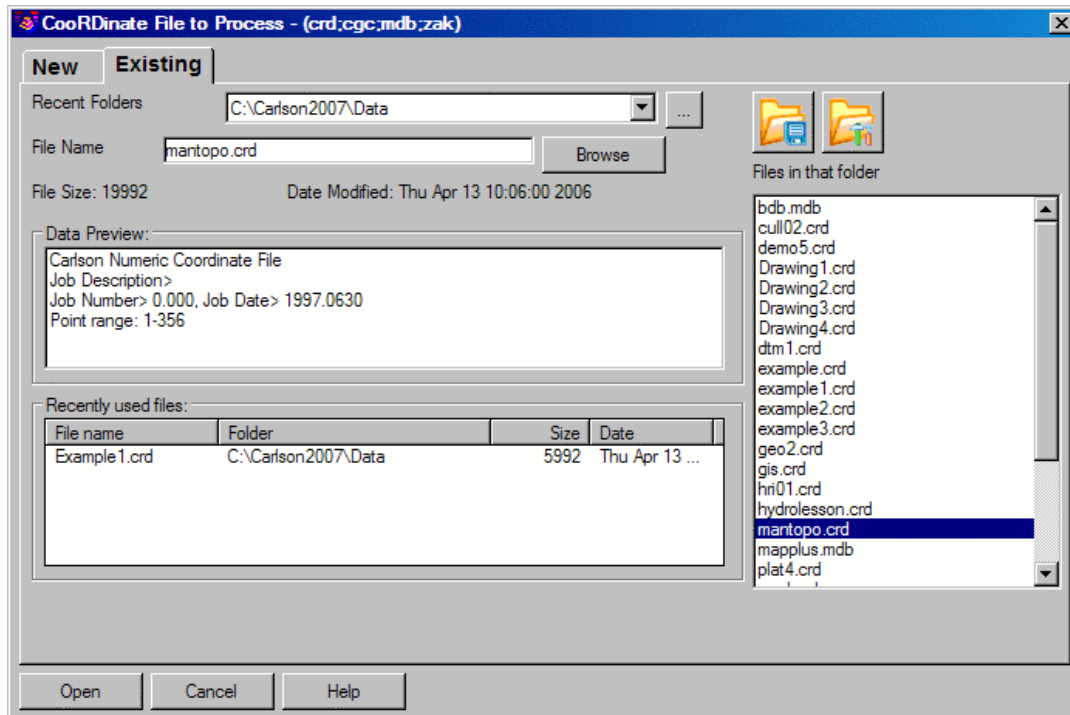
Keyboard Command: writept

Prerequisite: A Coordinate File (.CRD)

Set Coordinate File

This command allows the user to set the name of the active coordinate file. This file is used by different commands that compute, store and recall point coordinates. Carlson coordinate (.CRD) files are binary files that contain point numbers, northings, eastings, elevations and descriptions. Alternately, C&G CRD & CGC files, LandDesktop MDB files or Simplicity Systems ZAK files can be used in place of the Carlson CRD file. These files are stored by default in the configured data subdirectory. When prompted for the name, if you type in a path name the file will be stored in the specified path. If you don't specify a path then the default path that is configured in the *Configure* command, found under Settings, will be used.

When executed, the command defaults to the Existing tab for selection of an existing file. You may select a file from the list of Recent Folders, or choose the Browse button to go to a specific location on your computer. To create a new file, select the New tab and enter the name of the file in the file name field provided. Use the Browse button to specify the desired location to save the file.



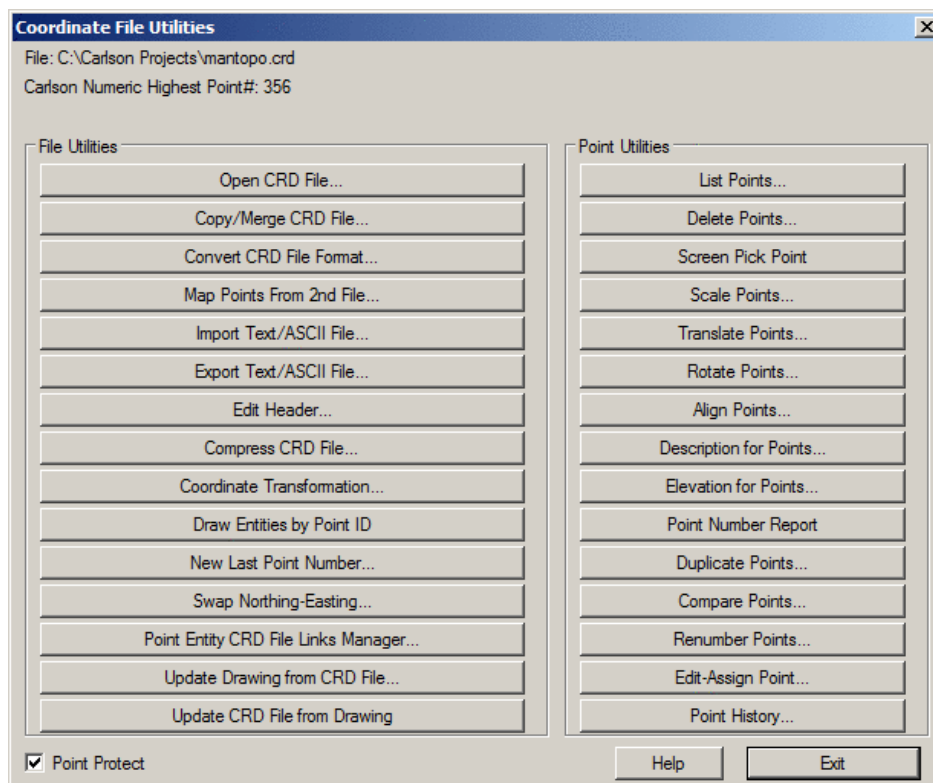
Pulldown Menu Location: Points

Keyboard Command: setcrd

Prerequisite: None

CooRDinate File Utilities

This command organizes a wide variety of coordinate file and point manipulation routines into one central location. The type of coordinate file format is displayed within the dialog box. Another increasingly popular coordinate format is the Carlson coordinate database (.CRDB) which is based on SQLite and supports point numbers and descriptions up to 255 characters in length.



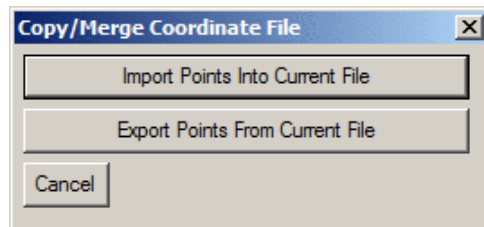
In addition to running the routines through the dialog, many routines have command names that you can enter at the *Command* prompt, create a Quick Key, or put into a toolbar. The table below shows the command names and their corresponding key-in command equivalents:

	File Utilities	Point Utilities	
Open CRD File	setcrd	List Points	listpt
Copy/Merge CRD File	cfucopy	Delete Points	delpt
Convert CRD File Format		Screen Pick Point	
Map Points from 2nd File		Scale Points	scalept
Import Text/ASCII File	readpt	Translate Points	transpt
Export Text/ASCII File	writpt	Rotate Points	rotatept
Edit Header		Align Points	alignpt
Compress CRD File		Description for Points	cfudesc
Coordinate Transformation	cfutransform	Elevation for Points	cfuelev
Draw Entities by Point ID		Point Number Report	cfureport
New Last Point Number		Duplicate Points	cfuduplicate
Swap Northing-Easting		Compare Points	cfucompare
Point Entity CRD File Links Manager		Renumber Points	renumpnt
Update Drawing from CRD File	cfuupdatedwg	Edit-Assign Point	
Update CRD File from Drawing	cfuupdatecrd	Point History	cfuhistory

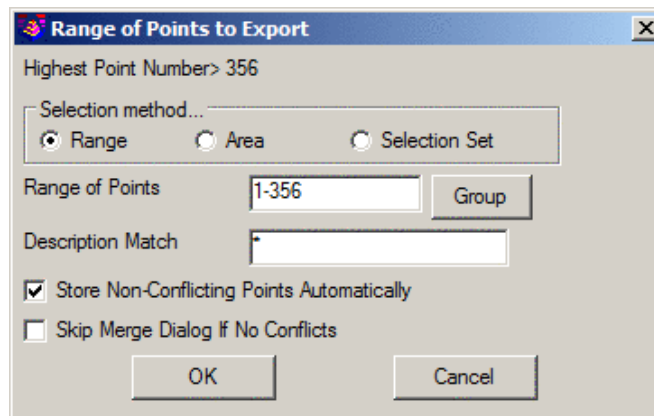
Open CRD File: Allows the user to switch to another file. See Set Coordinate File for additional information. When you exit Coordinate File Utilities this will be the current file that you work with in Carlson.

List Points: List the points stored in the active coordinate file. See List Points for additional information.

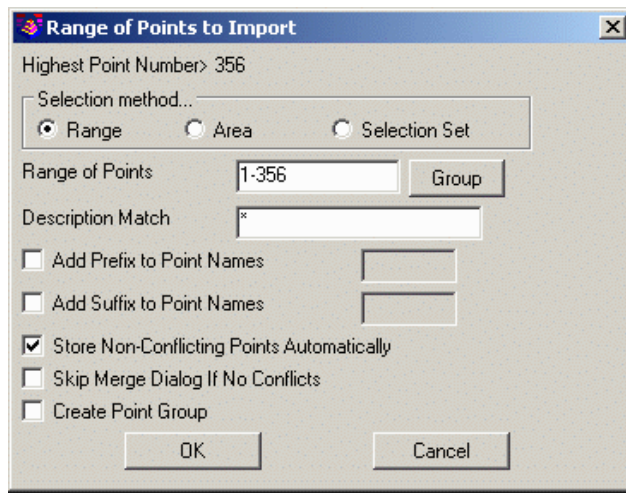
Copy/Merge CRD File: This command allows for the copying of entire CRD files, or parts of CRD files, to a new or existing files. This can be used to make a backup of your coordinate file, and it can also be very valuable in coordinate file manipulation. For example, if a certain range of points from one CRD file was also required in the active CRD file, this command would be used to simply copy the required range into the active CRD file. There are two options when first executing the command. These options are whether to import points from another file to the current (active) CRD file, or to export the current (active) coordinate file to another file.



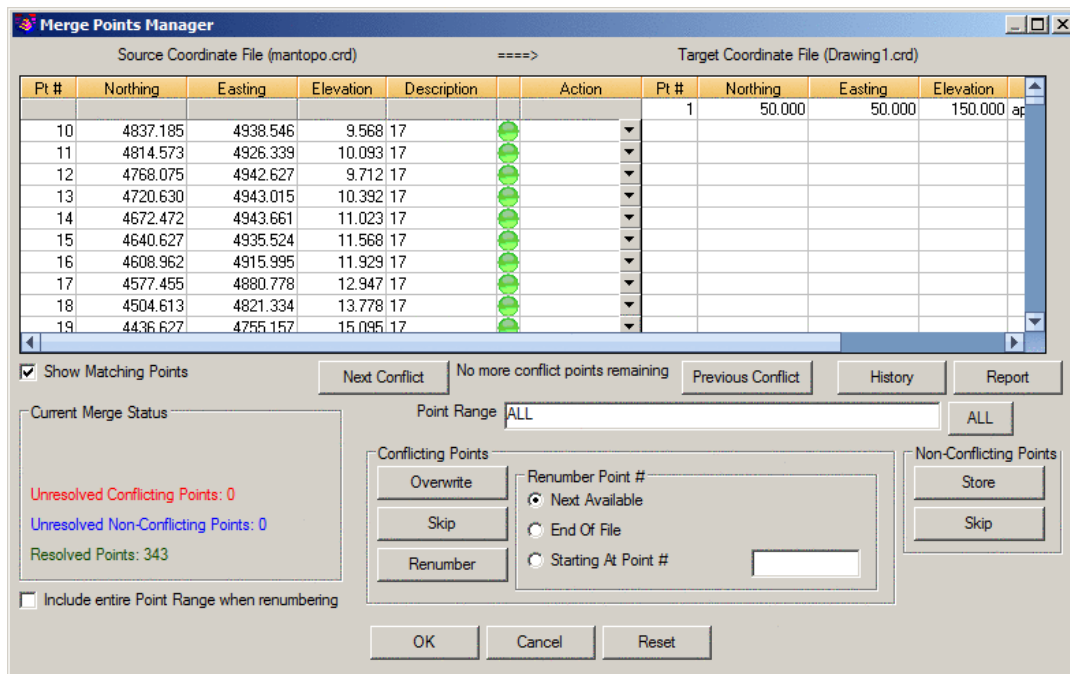
Once this option has been decided, a prompt for the file to copy From or TO, will be displayed. Here simply specify the correct file.



Next there's a dialog to specify the range of points to transfer and some options. Here specify the points to copy. Point numbers and ranges can be entered together. For example, **1-3,10,15** would result in points 1 through 3 and points 10 and 15 being copied. The Description Match can be used to filter the points to transfer only the points with matching description. The default of * will transfer all the points in the range. The Store Non-Conflicting Point Automatically will set the transfer action as Store for all transfer points that don't have a point protect conflict. The Skip Merge Dialog If No Conflicts will skip the next dialog when there are no point protect conflicts.



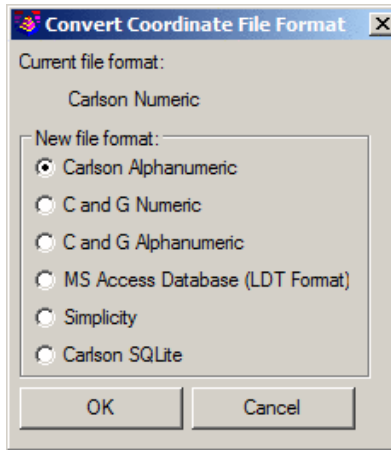
The Import dialog has a few more options than the Export dialog. The Add Prefix/Suffix To Point Names will change the point names as they are imported. The Create Point Group will make a new point group with the imported point names.



Next there's the Merge Points Manager dialog that shows the Source Coordinate File on the left (where the point data is being copied from) and the Target Coordinate File on the right (where the point data is being written to). Conflict cases are when the same point number exists in both files with different coordinates. The action choices for conflicts are to Overwrite, Skip or Renumber. For renumber, you can either renumber with the next available point number in the target file or to the highest point number in the target file plus one. Non-conflict cases are when the source point number does not exist in the target file. The action choices for non-conflicts are to Store or Skip. You can assign actions by picking on the Action field in the spreadsheet or by entering in a Point Range to apply and picking an action button. The Show Matching Points toggle will show points with matching point data in both files. Otherwise only point with differences are shown. The Next Conflict button will highlight the spreadsheet and set the Point Range to the next point that needs an action assigned. Similarly, the Previous Conflict sets focus to a lower point number that needs an action. The History button shows the point history for the selected point. The Report button creates a list points report. The Current Merge Status reports the number of unresolved and resolved points. When all the unresolved points are resolved by assigning actions, you can pick OK.

Delete Points: Deletes points from the active coordinate file. See Erase Points for additional information.

Convert CRD File Format: This allows you to convert the current coordinate file from one format (*e.g.* numeric only) to another (*e.g.* alphanumeric). This routine will also change coordinate files to and from other software formats. These formats include Carlson SQLite (.CRDB), C&G, Microsoft Access (.MDB) in same format as AutoDesk Land Desktop, and Simplicity (.ZAK). The current format of the active coordinate file will be displayed as well as the options for the new file format. This command only changes the format of the active coordinate file.



Screen Pick Point: Pick a point on the graphics screen and its coordinate value is added to the coordinate file. This command does not plot a point, point attributes or point symbol. Use Draw Field-to-Finish or Draw-Locate Points or Update Drawing from CRD File to graphically draw the point(s).

Map Points from 2nd File: This routine adds point to the current coordinate file from points stored in a second coordinate file. The points to copy are specified by numbers one at a time. Prompts for the destination point number (number to create in current coordinate file) and source point number (point number to be copied from second coordinate file) will be displayed.

Scale Points: This option multiplies the point northing, easting, and elevation by the scale conversion factor. You can use this routine for Metric-Imperial unit conversion. See Scale Points for additional information.

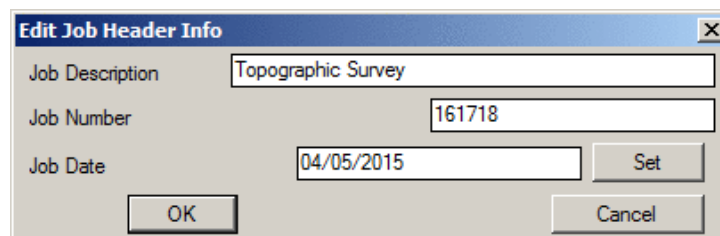
Import Text/ASCII File: This routine converts point data from a text file into the current coordinate file. See Import Text/ASCII File for additional information.

Translate Points: This option translates a range of points based on entered delta x and delta y, entered coordinates or translation point numbers. See Translate Points for additional information.

Export Text/ASCII Text File: This routine outputs point data from the current coordinate (.CRD) file to a ASCII Text file. See Export Text/ASCII File for additional information.

Rotate Points: This option rotates a range of points based on entered degrees or rotation, entered azimuths, entered bearings or rotation point numbers. See Rotate Points for additional information.

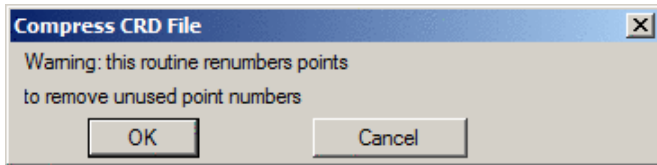
Edit Header: Enter or edit the job information associated with the coordinate file. The fields include Job Description, Job Number and Job Date. This information will appear on the List Point report. Non-digit characters are not allowed in the Job Number field.



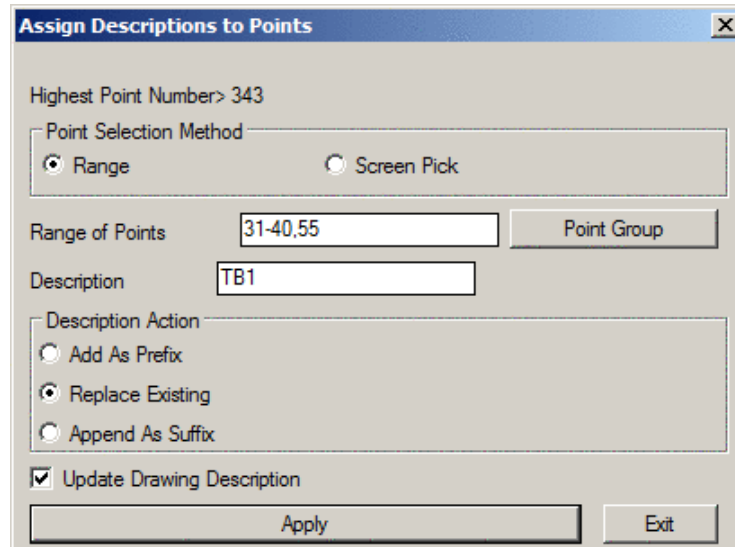
Align Points: This option does a translate based on a source point and destination point and then rotates to align the

first source point and a second source point with the first destination point and a second destination point. See Align Points for additional information.

Compress CRD File: Removes unused point numbers by renumbering high point numbers into the unused spaces. For example, for an original file with points 1,2,105,107,108,109 would be compressed to 1,2,3,4,5,6.

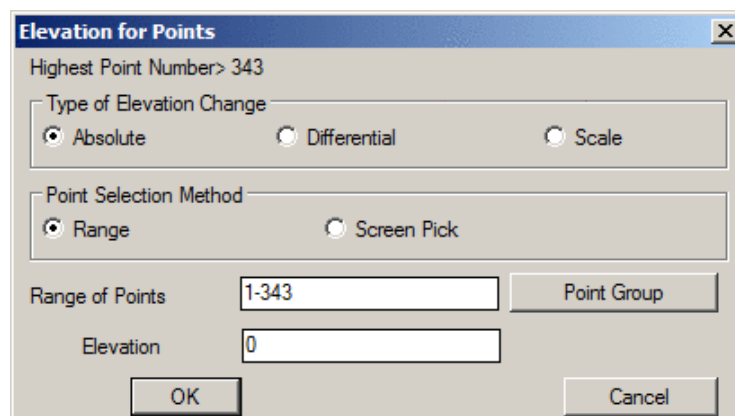


Description for Points: This routine modifies the point description field with the user-specified text for a range of point numbers. There is an option to update the description attributes of the points in the drawing in addition to updating the coordinate file.



Coordinate Transformation: Transforms coordinates between local, State Plane 27, State Plane 83, Latitude/Longitude, Universal Transverse Mercator (UTM) and many other projections, including regional and user-defined projections. This works on individually entered coordinates, by range of point numbers and with on-screen entities. See Coordinate Transformation for additional information.

Elevation for Points: This routine modifies the elevation of the specified points. The Absolute method sets the elevations to the specified value. The Differential method adds the value to the current elevations. The Scale method multiplies the current elevations by the value.



Draw Entities by Point ID: Draw Lines, Arcs, 3DLines, Polylines or 3DPolys by defining a range of point numbers.

Prompts

Plot Entities by Point Number

Type of entity, Arc/Polyline/3dpoly/2dline/Exit/ <Line>: Indicate the type of entity to draw.

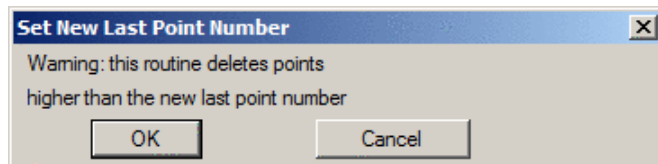
Undo/ <Enter point numbers or ranges>: Indicate the point numbers through which to draw.

Example: **1-4,7,10-12,5,8** would draw the entity from point numbers 1 through 4 then to 7, to 10 through 12, then to 5 to 8 (limit 132 characters)

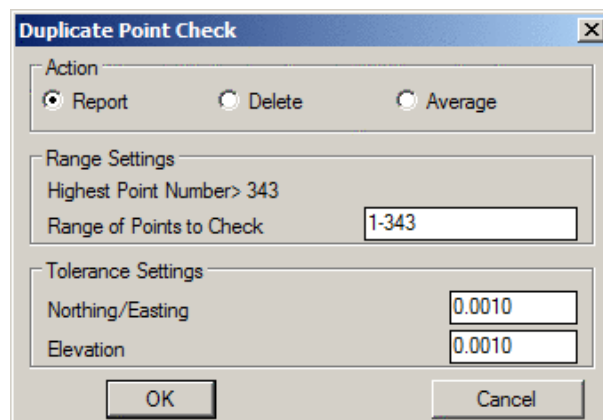
Example: **1-10,20-30** would draw the entity from point number 1 through 10 then to point number 20 through 30.

Point Number Report: This routine lists the used and the unused point numbers in the coordinate file.

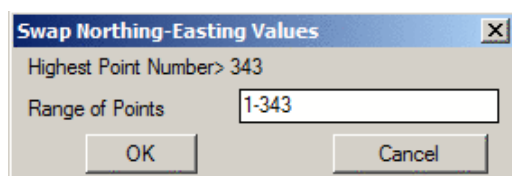
New Last Point Number: This option sets the highest point number in the coordinate file. **NOTE:** All points above this number are erased!



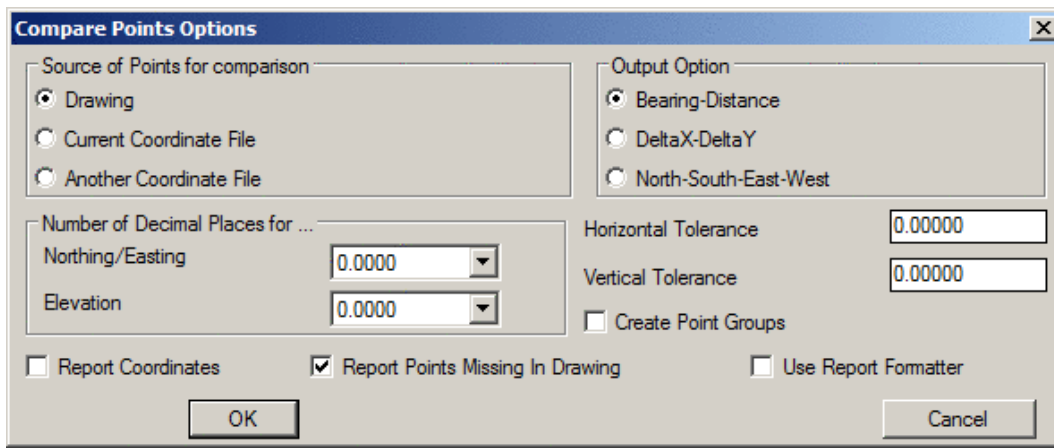
Duplicate Points: This function searches the coordinate file for points with the same northing, easting and elevation. The tolerances for considering points to have the same coordinate are set in the dialog separately for northing/easting and elevation. To be counted the same coordinate, both the northing/easting and elevation must be within the tolerance distance. The duplicate points can be reported, deleted or averaged. For the erase and average options, the first point number is kept and any higher point numbers with duplicate coordinates are processed from the coordinate file.



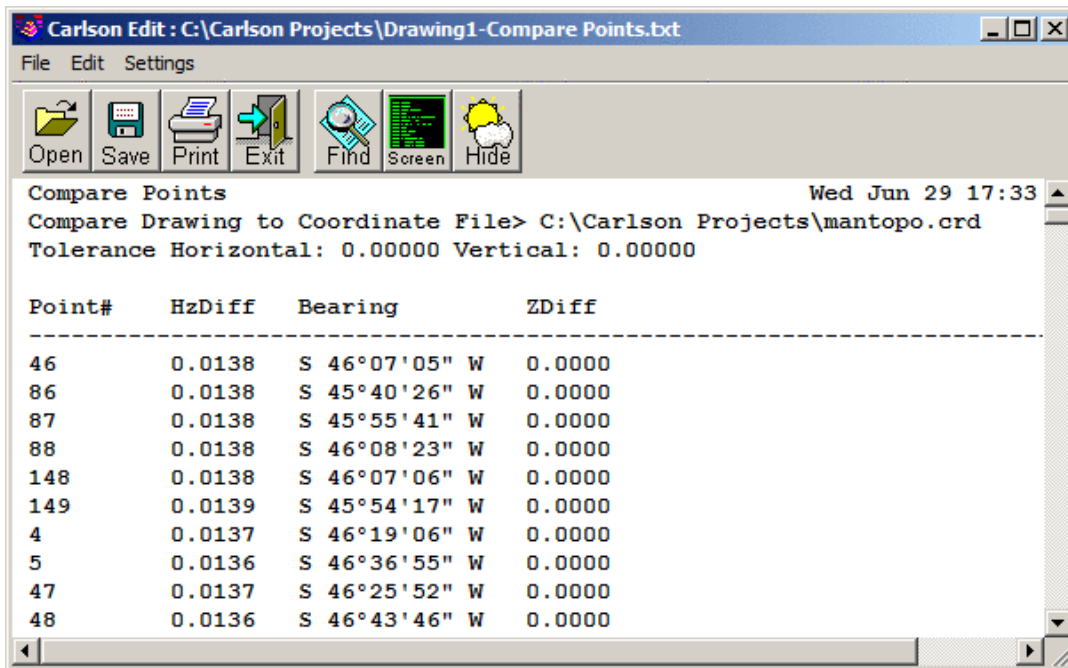
Swap Northing-Easting: This option allows you to swap northing and easting coordinates for any selected range of points. What was the northing of an existing coordinate point, or range of points, becomes the easting. And the easting(s) become the northing(s).



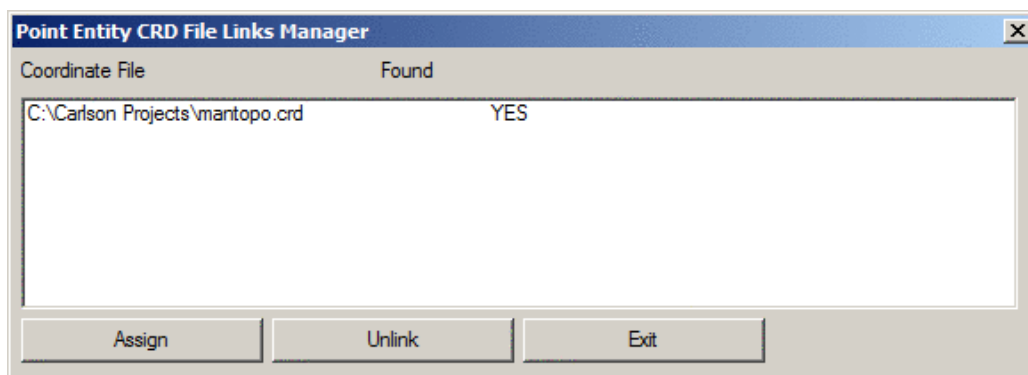
Compare Points: This function compares the coordinates in the coordinate file with either the coordinates for the matching point numbers in the drawing file, with matching point numbers from another coordinate file or with different point numbers from the same coordinate file. A report is created for any differences that shows the point numbers and the differences. The difference can be reported as a bearing and distance between the two points, as distance North/South and East/West or as the delta-X and delta-Y. There is an option whether to include the point coordinates in the report. The Create Point Groups option creates point groups of "Missing From Drawing" and "Changed Points" for any points that have this status. Use the Point Group Manager routine to check on these point groups.



Example Bearing-Distance format Compare Points Report:

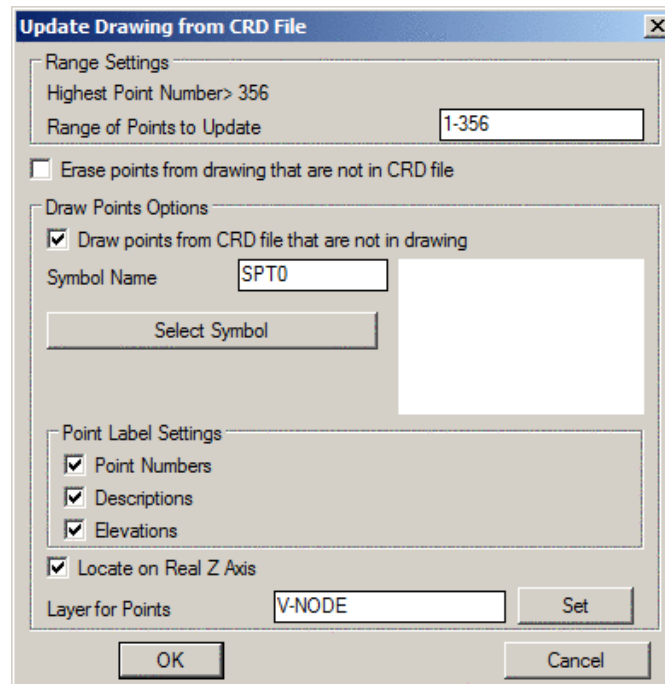


Point Entry CRD File Links Manager: When points are created in the drawing, the program records the source coordinate file for the points. The coordinate file names assigned to the point entities links the point entities back to the coordinate file. These links are used by routines that process the point entities and then need to reference the changes (*e.g.* those generated from the Rotate Points routine) back to the coordinate file. This routine checks all the point entities in the drawing and lists all the linked coordinate files. You can use the Assign button to set the coordinate file assigned to point entities which is useful when the coordinate file has been moved or manually renamed after the points were drawn. Use the Unlink button to remove the link.

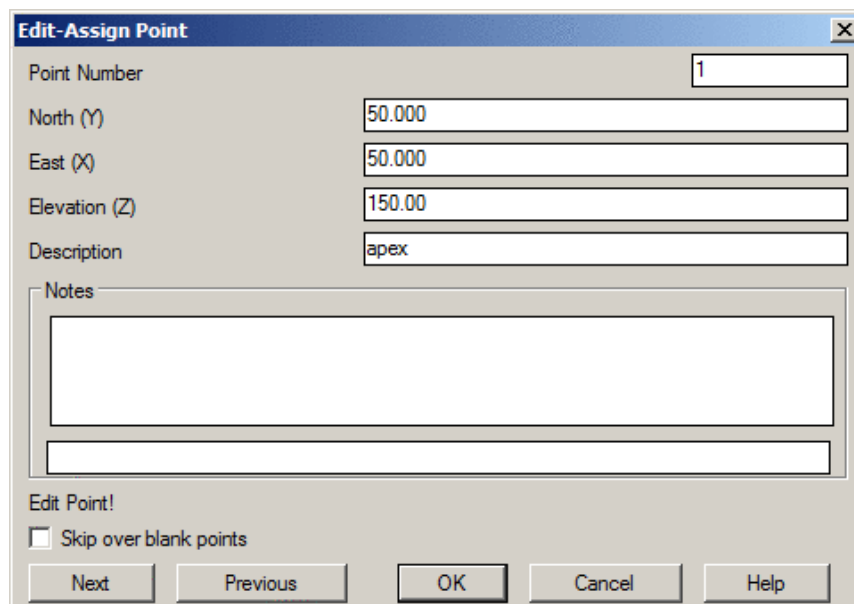


Renumber Points: This option renumbers points in the user-specified range starting from a new point number. See Renumber Points for additional information.

Update Drawing from CRD File: This function updates the position of Carlson points in the drawing to match the position stored in the coordinate file. This command also has options to erase and draw points. For the erase option, points are erased from the drawing if the point number does not exist in the coordinate file. For the draw option, if a point number in the CRD file does not exist in the drawing, then this point is drawn using the settings from the dialog. The number of points modified, erased and drawn is reported at the end of the command.



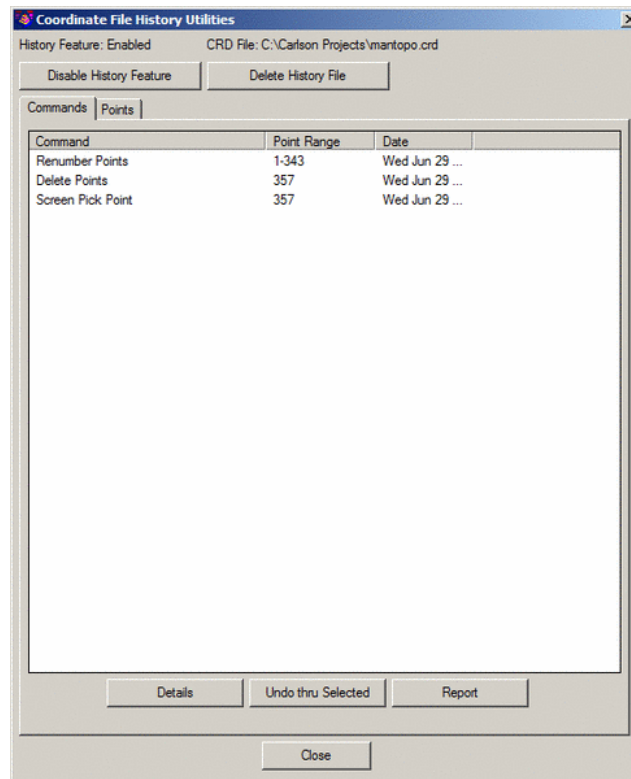
Input-Edit Point: Enter or edit the coordinate values or the description of a point. The Notes section is for adding optional point notes which are additional point descriptions. Under notes, any number of lines of text can be assigned to the point. A list box shows the lines of notes. To add a note line, pick a blank line in the list box and then type in the note in the edit box along the list box and press **Enter**. To edit a note, highlight the line in the list box and edit the text in the edit box.



Update CRD File from Drawing: This function allows you to select all or some of the points in the drawing and add or update them to the coordinate file. The points can be filtered using standard CAD entity selection and/or

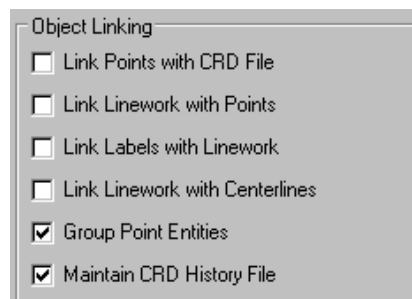
wild-card matching of the point descriptions. The Update Point Descriptions option determines whether the point descriptions from the drawing will be stored to the coordinate file. Use this command to manually update the file after a general CAD edit such as *Move*, *Rotate*, *Erase*, etc. This routine directly reads Leica (Wildsoft), Softdesk, Geodimeter, InRoads, Land Development Desktop, and Eagle Point point blocks.

Point History: All changes to the coordinate file will record the commands performed on this coordinate file and the status of the points themselves. This makes up the coordinate file history. The history can then be reported by point number or by command. All of the changes can be rolled back.

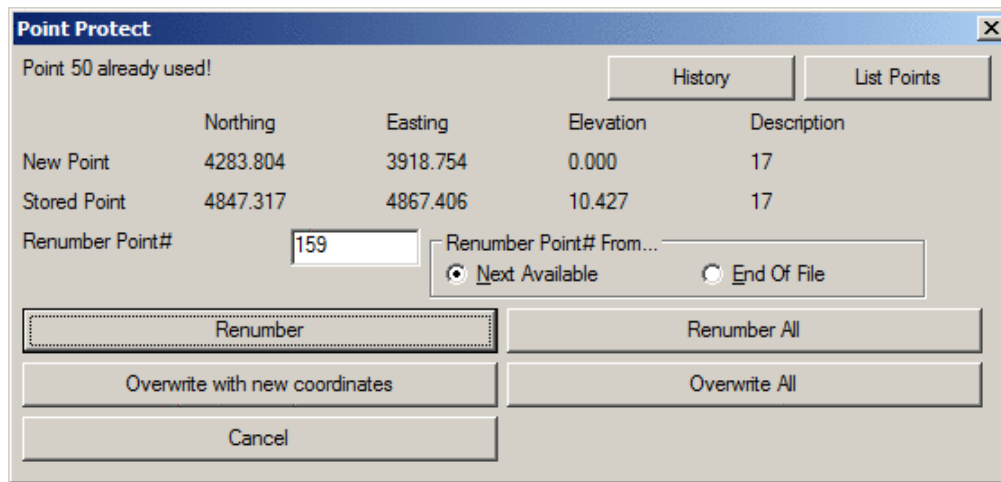


The **Disable History Feature** button at the top of the dialog shown above is a toggle device. It should be clicked if you prefer not to build the point history file. Clicking it a second time changes it back to saying **Enable History Feature**. You can also choose **Delete History File** to delete the file altogether. By clicking any point from the list, as shown in the Points tab example above, and then selecting **History**, you will be given the history for that specific point. Double-clicking on any command shows the details. Clicking on **Details** also shows the selected command details. Undo thru Selected will undo the effect of all of the commands up through and including the selected command. The changes from the undo command are themselves then added to the command list and can be undone in the future.

NOTE: It is important to note that if maintaining such a history file is your objective, make sure that *Maintain CRD History File* option is enabled within the (Carlson) Configure – General Settings command:



Point Protect: When enabled, attempting to store a point with a point identifier (point number) that already exists in the current coordinate file will display the following alert:



Overwrite with new coordinates will update the existing point number with the new location of the point.

The **Renumber Point#** field displays the point number that will be used if the Renumber Point# From – Next Available option is selected (the routine will attempt to fill in any point number gaps. To leave any existing point number gaps in the file, specify the **End of File** option.

The **Overwrite All** and **Renumber All** options apply when more than one point with the same number exists in the coordinate file. These options are helpful when importing points into existing coordinate files.

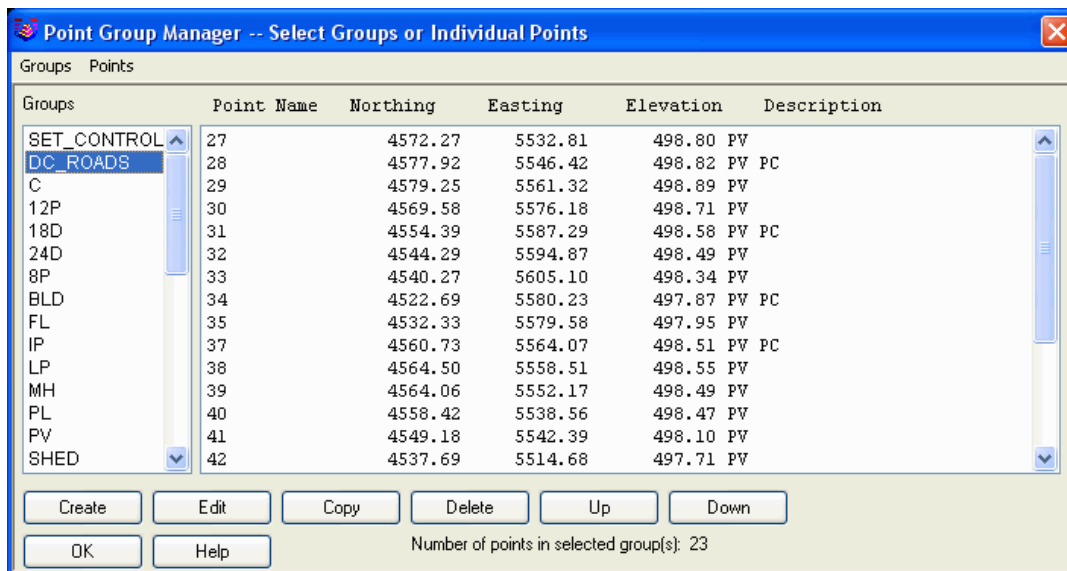
Pulldown Menu Location(s): Points

Keyboard Command: cfu

Prerequisite: None

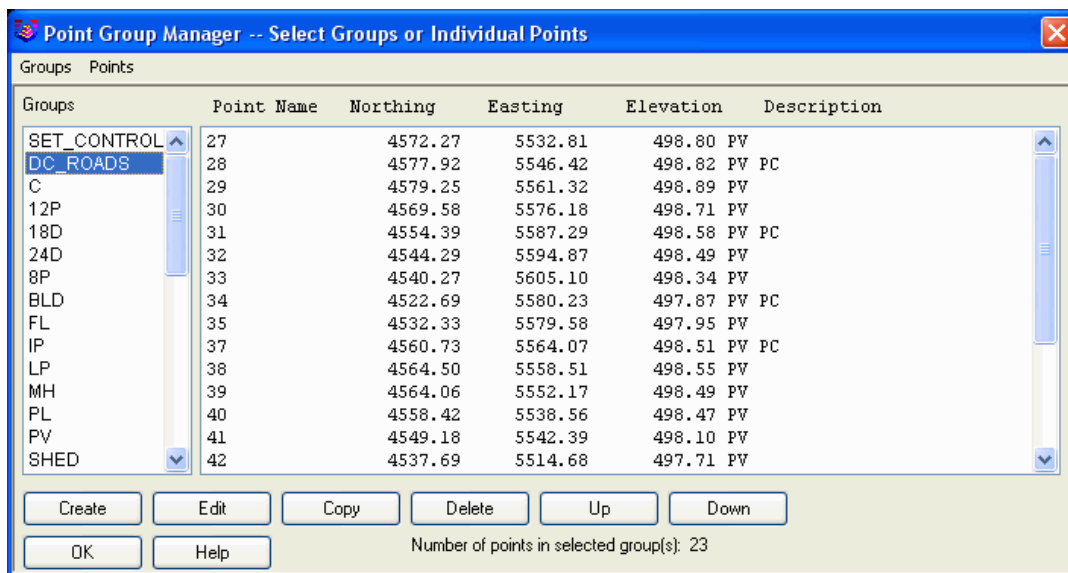
Point Group Manager

This command is used to create point groups based on inclusion and exclusion filters. The manager can perform various functions on these point groups. Also point groups can be referenced by group name in other commands such as Field to Finish and Data Collection.



Groups Pulldown

Create Point Group: This routine creates point groups. When selected, the New Point Group dialog box is displayed.



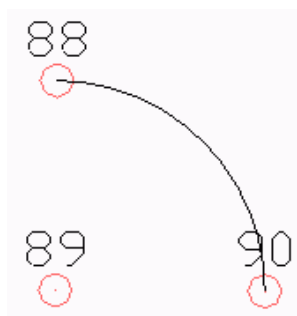
Group Name is the name of Point Group to create.

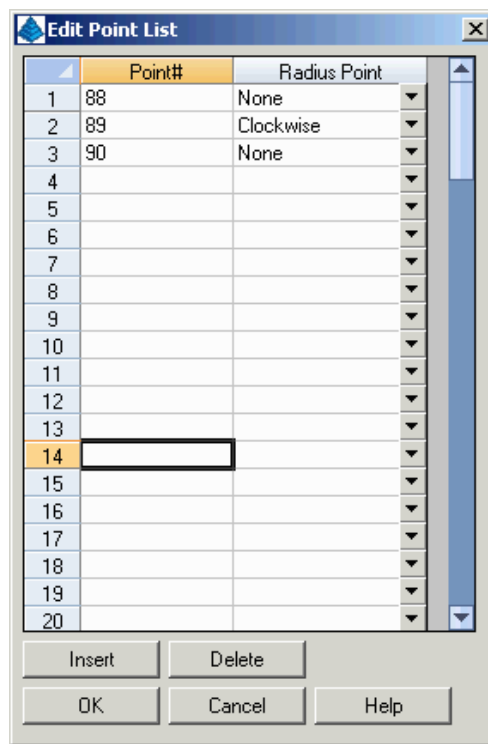
Description is the description of Point Group to create.

Use the **Include Tab** to define the filters to be applied when creating the point group. Inclusion rules are applied before the exclusion rules.

When **Include All** is toggled on, all points in the coordinate file will be included in the selection.

When **Point List** is toggled on, an option of defining the point list can be selected or the point numbers can be manually entered in the edit box. The points retain the order entered in the edit box which can be used in other point functions that process points where the order matters such as Legal Description. The Edit button brings up a spreadsheet editor for the point list and there is a flag for each point for whether the point is a radius point. This radius point flag is used in routines that process points for a polyline or perimeter such as Legal Description. The radius point is used to define an arc in the polyline. The sequence of points for the arc are PC, radius point and PT.





DWG: Select allows for manual selection of the points to include from the drawing. The points must be drawn on the screen prior to using this option. All standard AutoCAD selection tools, are available for selection of the points.

DWG: Inside Circle allows for selection of the points to include by a user defined circle. The circle is defined by specifying the center and radius of the circle. The radius can be defined by entering in a numeric value or by picking on the screen. Points must be drawn to the screen prior to using this option.

DWG: Inside Polyline allows for the selection of points to include by referencing a closed polyline. All points located within the closed polyline will be included in the selection. Prompts for the inclusion polyline and the exclusion polyline will display. The inclusion polyline limits of the selection area. The exclusion polyline defines the area to exclude within the inclusion polyline. Points must be drawn to the screen prior to using this option.

DWG: Along Polyline allows for the selection of points to include by offset from an alignment polyline. All points located within the specified offset tolerance from the polyline will be included in the selection.

CRD: Select allows for manual selection of the points to include from a point list. Standard window selection tools are available for selecting the points to include.

Name	North	East	Elevation	Description
1	4806.66	5980.63	0.00	
2	4950.61	4695.72	102.05	XCUT
3	4785.03	4716.23	96.96	XCUT
4	4852.58	5278.93	109.71	MNSET
5	5154.12	5237.54	106.49	MN-SET
6	5124.84	4999.58	104.90	MN-6
7	5287.14	5049.95	0.00	
8	5297.01	5098.97	0.00	
9	5306.88	5147.98	0.00	
10	5316.75	5197.00	0.00	
11	5326.61	5246.02	0.00	
12	5336.48	5295.03	0.00	
13	5346.35	5344.05	0.00	
14	5356.22	5393.07	0.00	
15	5366.09	5442.08	0.00	

CRD: Inside Circle allows for selection of the points to include by a user defined circle. The circle is defined by specifying the center and radius of the circle. The radius can be defined by entering in a numeric value or by picking on the screen. The points do NOT have to be drawn to the screen prior to selection.

CRD: Inside Polyline allows for the selection of points to include by referencing a closed polyline. All points located within the closed polyline will be included in the selection. Prompts for the inclusion polyline and the exclusion polyline will display. The inclusion polyline limits of the selection area. The exclusion polyline defines the area to exclude within the inclusion polyline. The points do NOT have to be drawn to the screen prior to selection.

CRD: Along Polyline allows for the selection of points to include by offset from an alignment polyline. All points located within the specified offset tolerance from the polyline will be included in the selection.

RW5 File: creates a list of points from all the point numbers used in the selected RW5 raw data file.

History Select: creates a point group by date range using the log stored in the CRD history. See Coordinate File Utility > Point History to review this information.

Point Name Prefix: creates a point group by a specified number of digits from the start of point names. This method applies when your point numbering method uses a fixed beginning string for point names. For example, if point names begin with a code for the crew, then this method can make point groups per crew.

Point Name Suffix: creates point groups using alpha characters that follow numbers in the point names.

Ungrouped: creates a point group including any points that are not already part of another group.

Non-Surface: creates a list of points that are set as Non-Surface points to be skipped in surface modeling.

Elevation Range allows for the selection of points within a specified elevation range to be included in the group. The minimum and maximum elevations can be entered manually in their respective data fields. The minimum and maximum values can also be specified by the Set By Selection and Set From List options.

Set By Selection allows for selection of points to include in the group from the drawing. The points must be drawn to the screen prior to using this selection method. Standard AutoCAD selection methods are available.

Set From List allows for selection of points to include in the group from a point list. Standard Windows selection tools are available with this option.

The **Description** option allows for a selection of points to include based upon the description of the point. The description to filter for can be entered in the data field or by using the Set By Selection and/or the Set From List options described above.

Make Point List converts the group definition to a list of point numbers that currently match the group definition.

The **Exclude Tab** allows for defining rules that pertain to the points to be excluded from the Inclusion selection. After defining the inclusion rules for the group, the options on the Exclude tab can be used to filter for points to exclude from the group. For example, if the inclusion rules call for all points within the elevation range of 8 to 12, an exclusion rule can be set to exclude the points on elevation 9 or with the description tree. The options on this tab work exactly like the options on the Include tab. Please refer to the Include tab definitions for further instruction.

Save Changes saves the point group to the group name specified based upon the Inclusion and Exclusion rules specified.

Cancel Changes discards specified rules and changes and goes back to the Point Group Manager dialog.

Edit Point Group: This function allows for editing of existing point groups. From the list of available groups, highlight the group or groups to edit. When complete with the first group, if more than one is selected, selecting the Save Changes option will save the changes to the active group and switch to the next group in the selection set.

From the Groups pulldown, select Edit Groups, the Edit Group dialog box will now appear.

See Create Point Groups for further definitions of the available options.

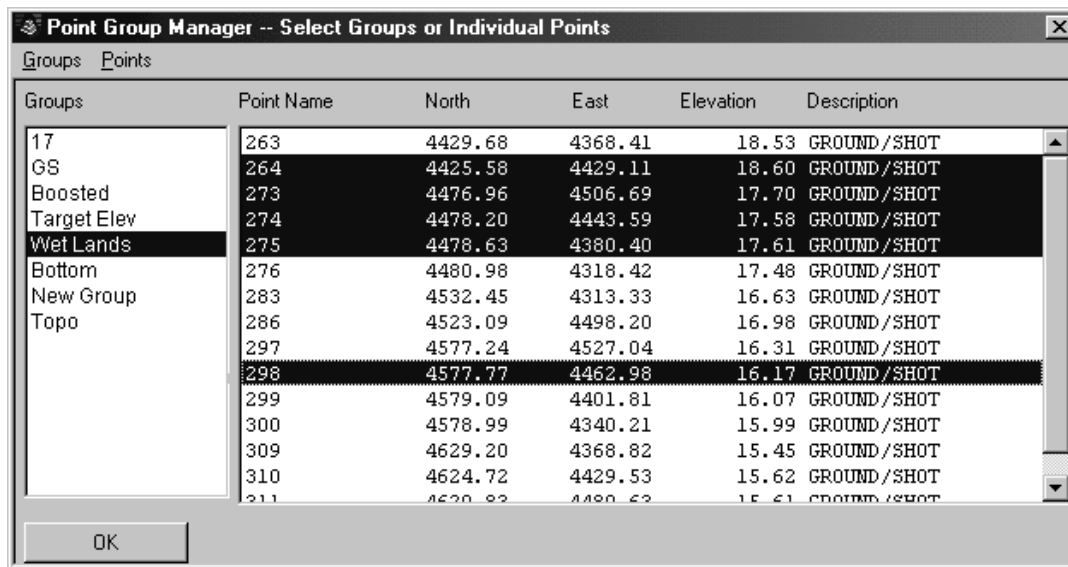
Delete Point Group: This deletes specified groups for the existing group list. One or more groups can be deleted at one time.

Copy Point Group: This routine creates a new point group by copying the currently highlighted group. This allows you to modify an existing group definition and create a new group.

Import Point Groups: This allows for importing filters from point group manager settings of other coordinate files. This is a useful option when coordinate files are going to contain same point group names with the same filters. This option only brings in the filters into the point group manager, it does not import actual points into the coordinate file by group name. Existing points in the active coordinate file that meet the filter definitions of the imported point groups will automatically be added to the corresponding group.

Points Pulldown

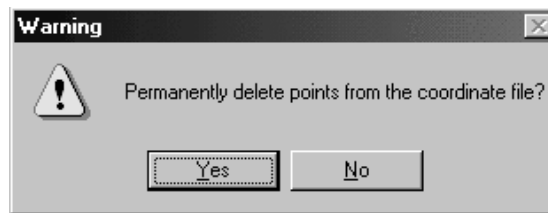
Insert into Drawing: This routine draws the points in the group in the drawing. Individual points or point ranges can be selected from the group to be erased from the drawing. For example points 264-275 and point 298 contained in group Wet Lands are tagged to be erased from the drawing in the following figure.



The symbol to be used and the attribute layout are determined by the Point Default Settings. The symbol size and the point attribute size are determined by the settings in the Drawing Setup routine.

Erase from Drawing: This erases specified point group/groups or specified points from within the group from the drawing.

Erase from Coordinate File and Drawing: This erases the points in the specified group/groups or specified points from within the group from the drawing and will also permanently delete the points from the CRD file. You will be prompted with a warning as follows:



Selecting **Yes** will complete the command and erase the points from the screen and also the coordinate file. Selecting **No** will cancel the command leaving the drawing and the coordinate file unchanged.

Report: The routine will generate a point list of the points contained in the selected group/groups or specified points from within the group.

Highlight: This routine highlights the specified objects in the drawing. This makes them distinguishable from the other points on the screen.

Isolate: This routine freezes all the points except for the current group.

Freeze: This routine freezes the points of the highlighted point group like the Points->Freeze Points command.

Thaw: This routine thaws the points of the highlighted point group like the Points->Thaw Points command.

Thaw All: This routine thaws all the points like the Points->Thaw Points command.

Draw 2D Line: This routine draws a 2d polyline between the points contained in the group/groups or between specified points in a group.

Renumber: Renumbers points in the current group.

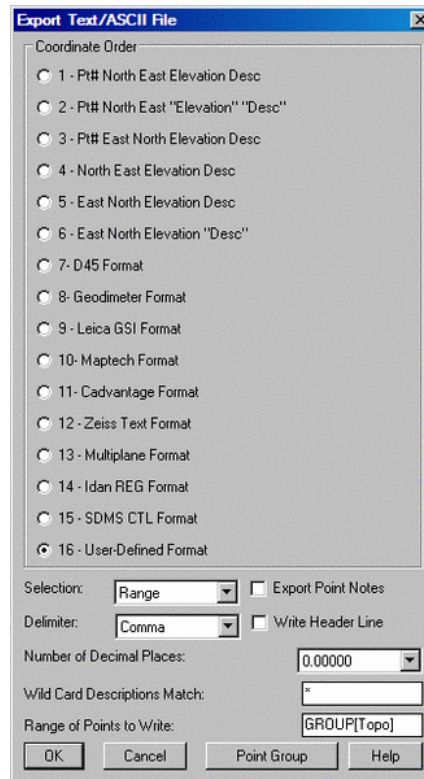
Edit Attributes: Edits the point attributes for the points entities in the drawing.

Edit Descriptions: Sets the description for the points in the current group.

Edit Elevations: Sets the elevation for the points in the current group.

Export: This command exports the selected group/groups or the specified point(s) or range of points from within the group to various formats. The available formats are ASCII/Text, Carlson Software CRD and C&G CRD files.

When **ASCII/Text** is selected, the Export Text/ASCII File dialog box will be displayed. Please refer to the Export Text/ASCII File section of the manual for more information.



The **CRD-Carlson software** command writes the selected group/groups or the specified point(s) or range of points within the group to a new Carlson formatted CRD file.

Specify the file name of the CRD file to create and press save.

CRD-C&G writes the selected group/groups or the specified point(s) or range of points within the group to a new C&G formatted CRD file.

Specify the file name of the CRD file to create and press save.

Button Functions

The series of buttons at the bottom of the main dialog do the same functions as the routines in the Groups pull-down menu except the Move Up and Move Down which are only available as these buttons. The Move **Up/Down** simply change the display order of the groups in the list. The **Import** function brings in group definitions from either another coordinate file or from a C&G Points List File. The **Sort** function sorts the groups by name or by number of points. The **Merge** function combines the definitions from two or more existing groups to create a new group.

To run Merge, highlight the groups to merge in the Groups list by clicking on them while holding the Ctrl key, and then pick the Merge button.

Pulldown Menu Location: Points

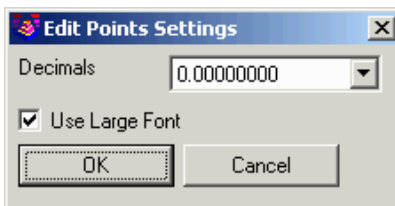
Keyboard Command: pgm

Prerequisite: A coordinate file

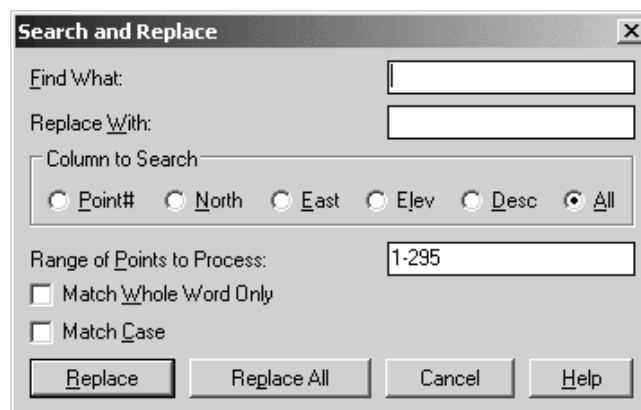
Edit Points

This command edits point data in the current coordinate file or within a point range. The current coordinate file can be set with the Set Coordinate File command. Edit Points shows all the points in the coordinate file. New points can be added and points can be deleted by using the Insert and Delete keys.

In the Edit pull-down menu, there are functions to add and remove rows from the spreadsheet. Also there are clipboard functions to copy and paste spreadsheet cell values. The **Delete Range** function allows you to delete a range of points or all the points by entering "ALL" for the range. The **Hide** function hides the selected range of spreadsheet rows and the **Show** function brings these rows back. The **Settings** function has a dialog to control the decimals to show in the spreadsheet and an option to use a large font in the spreadsheet to make it easier on the eyes to see.



In the Search pull-down menu, the **Find** function searches the spreadsheet. The **Replace** function has settings to limit the replace by point number range or by field type.



The Group > **Point Group Filter** function allows you to edit subset collection of points as defined by Point Group Manager. This Group method is a way to filter the points by point range, elevation range or description.

This tool also lets you edit notes associated with each point. While the standard point description is limited to 32 characters, the drawing notes are not. When you click on a given point, you can add numerous lines of notes about that point in the bottom of the dialog. Keep in mind that these notes are stored in a separate file with the extension ".not" having the same name as the CRD and residing in the same folder.

Point#	Northing	Easting	Elevation	Description
1	4024.91158781	5205.10767843	542.20000000	
2	4062.10394883	5173.57036672	543.10000000	
3	4126.71087104	5180.82246408	543.70000000	
4	4192.64550433	5207.19058280	544.20000000	
5	4219.67569784	5279.05437114	544.30000000	
6	4212.41921401	5369.37699554	544.20000000	
7	4179.45189736	5393.77355838	543.90000000	
8	4165.60696044	5442.55515448	543.70000000	
9	4139.23644727	5467.60890260	543.60000000	
10	4087.15510130	5477.50127071	543.40000000	

Point 1 Notes	
1	
2	
3	

Pulldown Menu Location: Points

Keyboard Command: editpt

Prerequisite: None

Erase Points

This command erases Carlson points inserted from the drawing. The points to erase can either be selected from the screen or specified by point number, point number range or by point group. Erasing a Carlson point will erase the three entities that make up a Carlson point: the point symbol, point attributes, and point node. There is an option to skip erasing the point symbol in case you want to leave the symbols in the drawing. The points may optionally be erased from the coordinate file. As long as the points are not deleted from the coordinate file, they can be redrawn with *Draw-Locate Points* or *Field-to-Finish*.

When the program detects linework associated with the points, there is a prompt for whether to erase this linework too. For example, when *Field-to-Finish* is used to draw both points and linework, then this linework can be erased along with the points.

Prompts

Select points from screen, group or by point number [Screen/Group/<Number>]? *press Enter*

Point numbers to erase: 1-5

Delete points from coordinate file [Yes/<No>]? *press Enter*

Delete point symbols [<Yes>/No]? *press Enter*

Erasing Carlson Points

Number of points erased > 5

Pulldown Menu Location: Points

Keyboard Command: DELPT

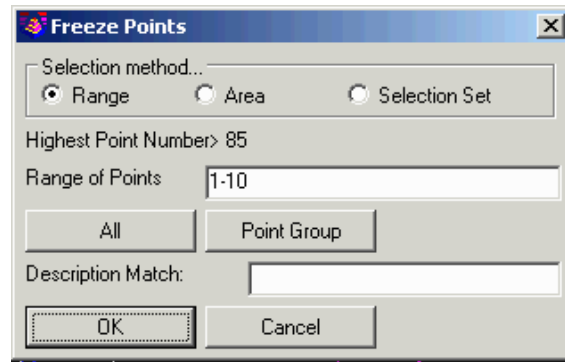
Prerequisite: Carlson points to be erased

Freeze Points

This command freezes Carlson points to hide them from view without erasing them. Use the Thaw Points command to show the points again. This command works similar on points as Freeze Layers works on layers. The points to freeze can be selected by point number range, point group, inclusion/exclusion perimeter polyline areas, or screen selection. There is a dialog to choose the method and specify a description match filter.

When the program detects linework associated with the points, there is a prompt for whether to freeze this linework

too. For example, when Field-to-Finish is used to draw both points and linework, then this linework can be frozen along with the points.



Pulldown Menu Location: Points

Keyboard Command: freezept

Prerequisite: Carlson points to freeze

Thaw Points

This command thaws Carlson points that were frozen with the Freeze Points command to show the points in the drawing again. This command works similar on points as Thaw Layers works on layers.

Pulldown Menu Location: Points

Keyboard Command: thawpt

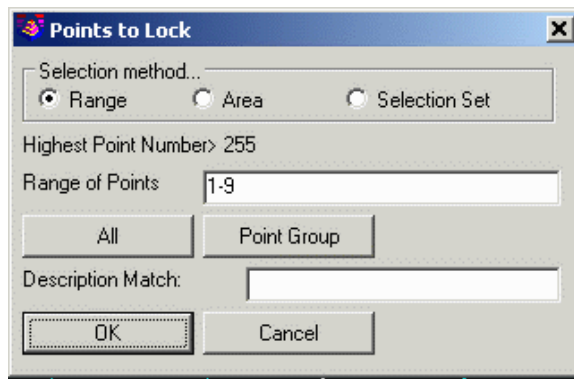
Prerequisite: Frozen Carlson points

Lock Points

This command locks points in the coordinate file so that their names, coordinates and descriptions cannot be modified. There are several types of coordinate files. Lock points only works with the CRDB format because this format is in an SQLite database which has a point attribute for lock status. To check on the format of the current coordinate file, run Coordinate File Utilities (CFU) which shows the format type at the top of the dialog. To change the current coordinate file format to CRDB, run CFU and choose the Convert CRD File Format function.

The only way to modify locked points is to first run the Unlock Points command and then run one of the point edit functions.

The points to lock can be selected by point number range, point group, inclusion/exclusion perimeter polyline areas, or screen selection. There is a dialog to choose the method and specify a description match filter.



Pulldown Menu Location: Points
Keyboard Command: lockpts
Prerequisite: CRDB coordinate file

Unlock Points

This command unlocks points from the current coordinate file that are locked. There is a dialog to choose the method to select which points from the coordinate file to unlock.

Pulldown Menu Location: Points
Keyboard Command: unlockpts
Prerequisite: CRDB coordinate file

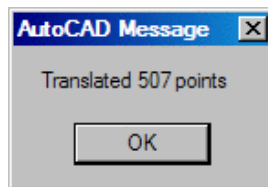
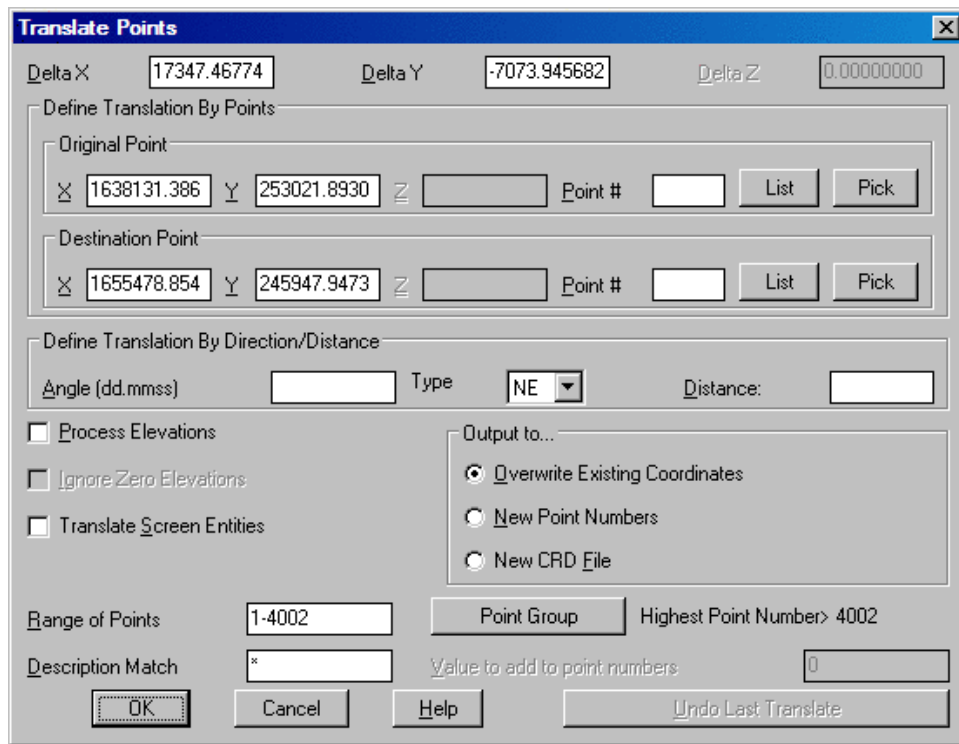
Report Locked Points

This command reports points from the current coordinate file that are locked. There is a dialog to choose the method to select which points from the coordinate file to check for the locked status.

Pulldown Menu Location: Points
Keyboard Command: reportlocks
Prerequisite: CRDB coordinate file

Translate Points

This command translates points in a coordinate file from one coordinate position to another. The delta X, Y, and Z can be entered directly or calculated from original and destination coordinates. The original and destination coordinates can be entered directly, specified by point number, selecting the point number from a point list by selecting the list icon, or selected from the screen by selecting the pick icon. Once these points have been specified, the Delta X,Y,Z, if Process Elevations is checked ON, fields will be filled in with their calculated values. Any points in the drawing will be updated automatically in addition to updating the coordinate file.



Define Translation By Angle/Distance requires a specified direction, Northeast (NE), Southeast (SE), Southwest (SW), Northwest (NW) or Azimuth (AZ) along with a specified distance in order to perform a translation. Once the direction and distance are entered, the Delta X,Y,Z will be calculated. This is a useful command when you know that the job needs to shift, for example, to the Northeast 25 degrees for a distance of 100 feet. Here you would simply type in 25 in the Angle (dd.mmss) field, choose NE in the Type field and then enter the distance of 100 in the Distance field.

With **Process Elevations** checked, all elevations will be translated by the specified or calculated Delta Z value. This option is very useful in correcting point elevations after performing a survey with assumed elevations and then later surveying into a benchmark with known true elevation. In this case only the Delta Z value, use (-) to indicate a lower correction, and the range of points to translate would be required for a translation. For example if the entire job needed to be lowered by 5', the Delta Z would be defined as -5 and the Range of Points defined as ALL.

Ignore Zero Elevations is only available when Process Elevations has been chosen. With this option checked ON, all points with an elevation of 0 will be ignored resulting in no translation taking place on these points.

With **Translate Screen Entities** checked ON, after specifying the point range or group to translate and selecting OK on the dialog box the following command line prompt is displayed:

Select objects to rotate (points excluded):

At this prompt select the objects on the screen, polylines, lines, arc, etc., to also translate and press enter. The translation of the points and screen entities will be completed.

Various **Output Options** for the translated points are available.

Overwrite Existing Coordinates will overwrite the existing coordinate points with the new translation coordinates thus changing the coordinate values in the existing crd file.

New Point Numbers will assign new point numbers to the translated coordinate points and leave the original co-

ordinate points unchanged and present in the coordinate file. When using this option, on the Range of Points to Translate dialog, there is a Value to add to point numbers field. In this field, enter the value to add to the point numbers. For example if the existing point numbers are 1-20, and the value to add is 100, the resulting new point numbers will begin at 101 and end at 120.

New CRD File will place the translated coordinates in a new crd file. After selecting OK to the range of points to translate dialog, the Coordinate File to Create dialog will appear. On this dialog enter the name of the new crd file and select save. The original crd file will remain unchanged and the new file will contain the points with the translated coordinates.

Specifying the points to be translated is accomplished either by specifying a **Range of Points** (1-20,33,36-40....) or by **Point Groups**. If using the Point Group option, the Select Point Group(s) dialog box will be displayed allowing for the selection of the Group(s) to rotate.

The **Description Match** option only translates points with the description(s) specified in this field.

Undo Last Translation restores the points to their previous location before translation. It is important to note that if Translate Screen Entities has been checked to restore the translated objects to their previous location will require the use of the undo command located in the Edit pulldown.

The AutoCAD command *MOVE* can be used to translate points on the screen but this does not update the coordinate file unless you have the option Link Points with CRD File turned ON in *Configure* . (Note: This toggle must have been turned ON prior to locating the points). If you do use the *MOVE* command and the CRD file needs updating, run the command *Update CRD file From Drawing* found in *Coordinate File Utilities*.

Pulldown Menu Location: Points

Keyboard Command: transpt

Prerequisite: points in a coordinate file

Rotate Points

This command rotates points in a coordinate file. The degrees of rotation can be entered directly or calculated from original and destination bearings or azimuths.

Rotate Points

Degrees of Rotation (dd.mmss)

Rotation Point

X: Y: Point #:

Original Bearing(Qdd.mmss)/Azimuth(dd.mmss)

Bearing Azimuth From Pt#: To Pt#:

Destination Bearing/Azimuth

Bearing Azimuth From Pt#: To Pt#:

Rotate Screen Entities

Output to...

Overwrite Existing Coords New Point Numbers

Range of Points Highest Point Number > 356

Description Match Value to add to point numbers

The **Rotation Point** will remain unchanged while the points specified for rotation rotate around it. This point can be specified by using the **List** button to pick from a list of points contained in the coordinate file, or from the screen by using the **Pick** button. The rotation point can also be defined by a coordinate value by manually entering in the X and Y values of the point. This point must be defined before the rotation will take place.

The **Original Bearings/Azimuths** and **Destination Bearings/Azimuths** can be entered directly or specified by point numbers. If using a pair of points to define the original bearing and then specifying the destination bearing by entering in the desired Bearing/Azimuth, the From and To Pt# fields should be left blank in the destination bearing/azimuth settings. Use the From and To Pt# fields in the Destination Bearing/Azimuth when you want to make a direction or Bearing/Azimuth between two existing points match the Bearing/Azimuth between two other existing points within the file. For example, to make the bearing between points 10-12 match the bearing between points 25-26, the Original Bearing/Azimuth could be defined as From Pt#10 To Pt#12 with the Destination Bearing/Azimuth defined as From Pt#25 To Pt#26.

With **Rotate Screen Entities** checked ON, after specifying the point range or group to rotate and selecting OK on the dialog box the following command line prompt is displayed:

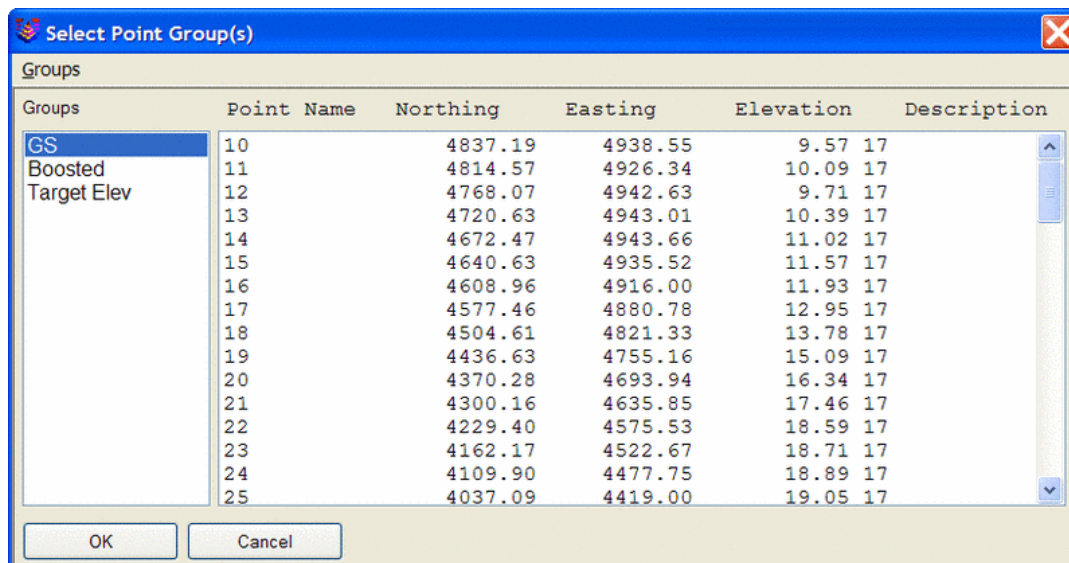
Select objects to rotate (points excluded):

At this prompt select the objects on the screen, polylines, lines, arc, etc., to also rotate and press enter. The rotation of the points and screen entities will be completed.

Various **Output** options for the rotated points are available.

Overwrite Existing Coordinates will overwrite the existing coordinate points with the new translation coordinates thus changing the coordinate values in the existing crd file.

New Point Numbers will assign new point numbers to the translated coordinate points and leave the original coordinate points unchanged and present in the coordinate file. When using this option, on the Range of Points to Translate dialog, there is a Value to add to point numbers field. In this field, enter the value to add to the point numbers. For example if the existing point numbers are 1-20, and the value to add is 100, the resulting new point numbers will begin at 101 and end at 120.



Specifying the points to be rotated is accomplished either by specifying a Range of Points (1-20,33,36-40....) or by Point Groups. If using the Point Group option, the Select Point Group(s) dialog box will be displayed allowing for the selection of the Group(s) to rotate.

The **Description Match** option only rotates points with the description(s) specified in this field.

The points that have been specified for rotation that are present in the drawing will be graphically updated to their new location in addition to an automatic update of the coordinate file.

Undo Last Rotate restores the points to their previous location before rotation. It is important to note that if Rotate Screen Entities has been checked to restore the rotated objects to their previous location will require the use of the undo command located in the Edit pulldown.

Pulldown Menu Location: Points

Keyboard Command: rotatept

Prerequisite: points in a coordinate file

Align Points

This command translates a specified Range of Points or Points Group(s) based on a source point and destination point and then rotates to align the first source point and a second source point with the first destination point and a second destination point. The command basically combines the Translate and Rotate Point commands. To specify a Range of Points to align, enter the range to align or select a point group(s) by selecting the Point Group button. Each of the Translation and Rotation points, both Source and Destination points, can be entered manually or picked from the point list by selecting the List button.

Align Points (Translate and Rotate)

Range of Points: 1-311 Point Group

Translation Points

First Source Point
X: 15502.93972 Y: 79257.05119 Z: 1452.000000 Point#: 2 List Pick

First Destination Point
X: 15488.99039 Y: 79547.48899 Z: 1434.600000 Point#: 45 List Pick

Rotation Points

Second Source Point
X: 15619.72502 Y: 79613.66199 Z: 1481.170000 Point#: 6 List Pick

Second Destination Point
X: 15538.86065 Y: 79683.64316 Z: 1489.730000 Point#: 49 List Pick

Align Screen Entities Ignore Elevations

DX: -13.949 DY: 290.438
Rotation (dd.mmss): 1.5901

OK Cancel Help Undo Last Align

When **Align Screen Entities** is checked, after specifying the point range or group to align and selecting OK on the dialog box the following command line prompt is displayed:

Select objects to rotate (points excluded): At this prompt select the objects on the screen, polylines, lines, arc etc., to also align and press Enter. The alignment of the points and screen entities will be completed.

When **Ignore Zero Elevations** is checked, all points with an elevation of 0 will be ignored in the alignment.

Undo Last Align restores the points to their previous location before alignment. It is important to note that if Align Screen Entities has been checked to restore the aligned objects to their previous location will require the use of the

undo command located in the Edit pulldown.

Pulldown Menu Location: Points

Keyboard Command: alignpt

Prerequisite: Points in a coordinate file

Scale Points

This command scales points in a coordinate file. The northing, easting and optionally the elevation are multiplied by the specified scale factor. You can use this routine for Metric-English conversion or a specific conversion by choosing the Use Customized Scale Factor option and specifying the desired Scale Factor in the edit box.

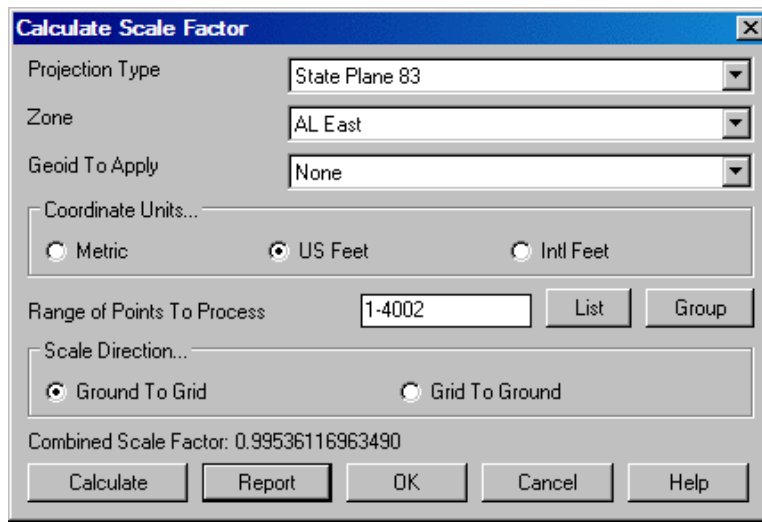
The screenshot shows the 'Scale Points' dialog box with the following settings:

- Points and Entities To Process:**
 - Range of Points: 1-4002
 - Point Group: (button)
 - Description Match: *
 - Highest Point Number: 4002
- Output to...:**
 - Overwrite Existing Coords
 - New Point Numbers
- Scale Screen Entities
- Value to add to point numbers: (text box)
- Scale Factor:**
 - Scale Factor: (text box)
 - Calculate Combined Factor: (button)
 - Ignore Elevations
- Base Point:**
 - N(y): 0.0
 - E(x): 0.0
 - Point #: (text box)
 - List: (button)
 - Pick: (button)
- Use Customized Scale Factor
- Source Coordinates:**
 - Metric
 - US Feet
 - International Feet
- Destination Coordinates:**
 - Metric
 - US Feet
 - International Feet
- Warning: this command scales the coordinates in the CRD file.
- OK: (button)
- Cancel: (button)

Specify the **Range of Points** to scale by entering in a range or group to scale. You can access the group dialog box by typing "group" in the range of points field.

The **Description Match** option only scales points with the description(s) specified in this field.

The **Scale Factor** is to be entered in manually when using a customized scale factor. If converting from standard measurement units, feet to meters, meters to feet, US Feet to International Feet, etc., the scale factor will be calculated and entered automatically. If a combined scale factor is required for converting from ground to grid and grid to ground coordinates, this value can be calculated by using the **Calculate Combined Factor** option. The Combined Factor is the Elevation Factor times the Grid Factor. This calculation process begins with the Calculate Scale Factor dialog shown below.



The **Projection Type** must be specified as either State Plane 83 or State Plane 27 as well as what state plane **Zone** is required.

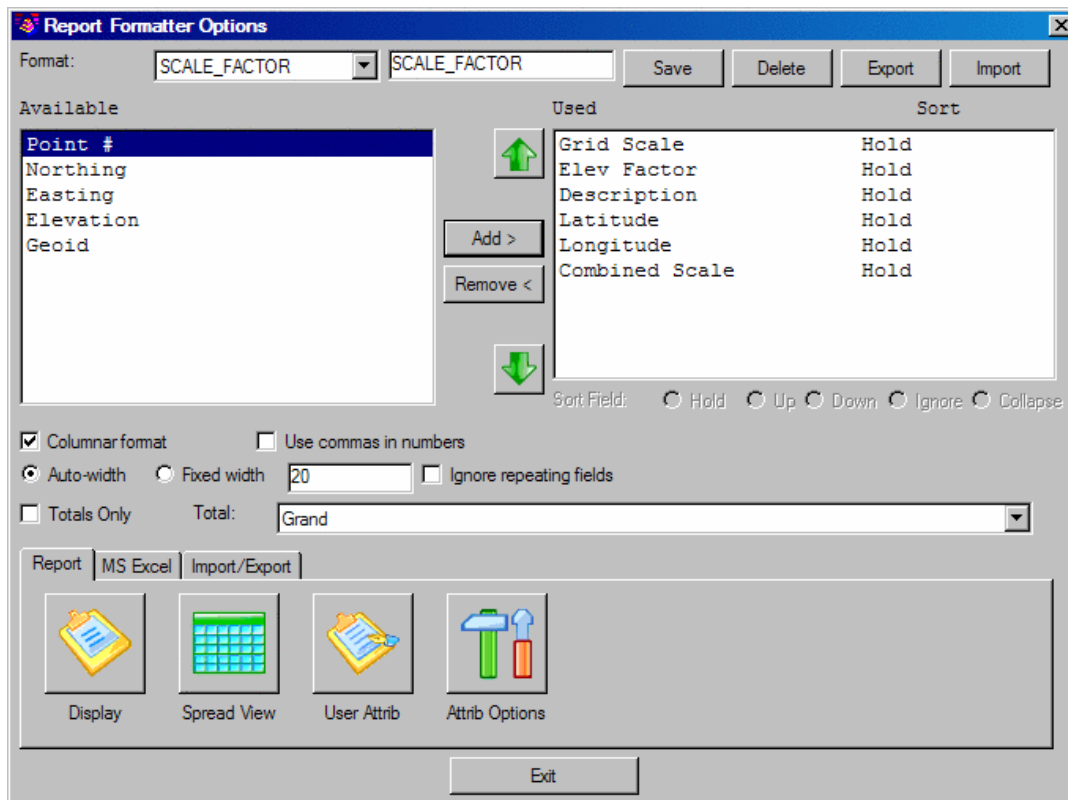
The available **Coordinate Units** are Metric, US Feet and International (Intl) Feet. The correct unit must be specified before calculating the combined scale factor.

The **Range of Numbers** to Process should be used to select the points to be used in order to calculate the combined scale factor. This does not specify what points are going to be scaled by the resulting scale factor. These points can be selected from a list by selecting the **List** button.

Scale Direction determines which way the scale factor will be calculated. A scale for **Ground to Grid** or **Grid to Ground** can be calculated and applied.

Pressing the **Calculate** button will calculate and then display the combined scale factor on the dialog box. To accept this value as the customized scale factor to use to scale the points in the coordinate file, press the **OK** button.

The **Report** option displays a report showing specified information. This information is specified by using the report formatter found throughout the program. Simply choose the information you wish to display and the order to be displayed. For further instruction and information on the Report Formatter please refer to the Report Formatter section of this manual.



With **Align Scale Entities** checked ON, after specifying the point range or group to scale and selecting OK on the dialog box the following command line prompt is displayed:

Select objects to scale (points excluded):

At this prompt select the objects on the screen, polylines, lines, arc, etc., to also scale and press enter. The points and screen entities will be now be scaled and updated graphically and in the active coordinate file.

With **Use Customized Scale Factor** Off, various conversions can be performed by specifying the Source Coordinate units and the Destination Coordinate units. This is a quick and easy way to perform Metric/English conversions.

Pulldown Menu Location: Points

Keyboard Command: scalept

Prerequisite: points in a coordinate file

Move Points

This command allows you to move Carlson points, one at a time by selecting any part of the point. Each Carlson point is made of three entities: a POINT entity, a symbol, and a point attribute block with the point number, elevation and description. This routine updates the X,Y of the point and not the Z. To update the elevation, use commands such as Edit Point Attributes or Translate Points. All these parts of the point are moved together with this routine. Any point moved using this command will result with the original source coordinate file (which is not necessarily the current coordinate file) updated with the new position of the point. Setting the Link Points with CRD File from Carlson Configure->General Settings is not necessary because the coordinate file is always updated since the Move Points command has built-in smarts to lookup the coordinate file for the selected point entity. The Link Points settings applies to generic CAD commands like the regular Move command.

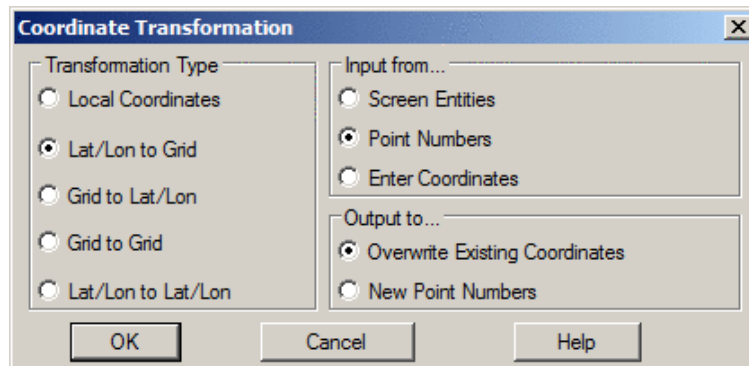
Pulldown Menu Location: Points

Keyboard Command: mpnt

Prerequisite: Carlson points

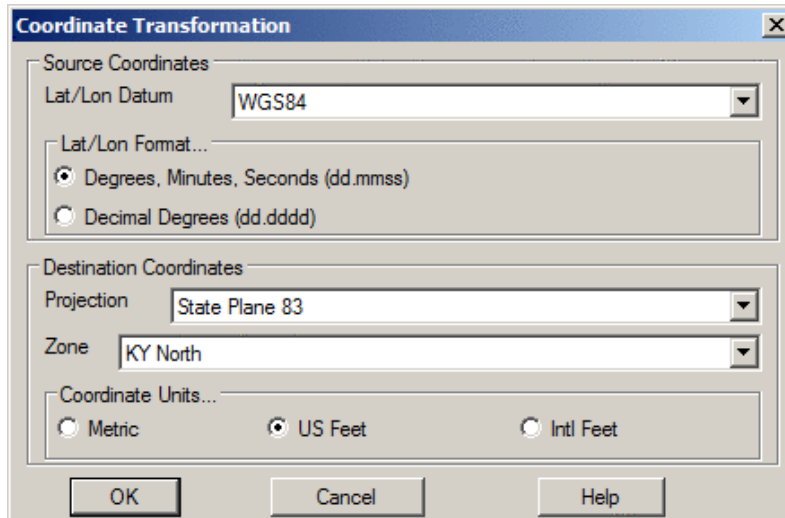
Coordinate Transformation

Transforms coordinates between local, State Plane 27, State Plane 83, Latitude/Longitude, Universal Transverse Mercator (UTM) and many other projections, including regional and user-defined projections. This works on individually entered coordinates, by range of point numbers and with on-screen entities. For converting between State Plane 27 and 83 within the USA, Carlson calls upon NADCON from the National Geodetic Survey to apply the Latitude/Longitude adjustment. Converting between NAD27 and NAD83 for Canada is supported using NTV2 grids.



The Transformation Type is used to define the Source Coordinate and Destination Coordinate formats. Settings for Lat/Long Datum, Lat/Long formats (dd.mmss or dd.dddd), Projections, State Plane Zones and coordinate units are defined in the Transformation Type dialog. The format of this dialog will change depending upon the type of transformation requested.

For Grid to Grid transformations, the program converts between state plane projections as well as other pre-defined and user-defined. When converting between pre-defined/user-defined projections, the program automatically converts the source grid coordinate to latitude/longitude and then to the destination grid coordinate. This method of using latitude/longitude works for converting between projections that share the same datum.



Example Lat/Long to Grid dialog

For all Transformation types, there are three options for inputting the data to be transformed. Data can be selected from the screen by using the **Screen Entities**. If a range of points or a particular point is desired, the **Point Numbers** option would be used. Manual entry of coordinates to transform one at a time is accomplished with the **Enter Coordinates** option. The coordinates can be typed in or use the Input Point Number option. Output Point Number is an option to store the results in the coordinate file.

For all transformations there are two output options when using point numbers as the input data. **Overwrite Existing Coords** replaces the original coordinate values with the new coordinate values after transformation. **New Point Numbers** will retain the original coordinate data and point numbers and create new point numbers with the revised coordinate data after transformation.

When transforming a **Local Coordinate System**, there are three options for defining the transformation as shown in the next dialog.

The **Enter Transformation Parameters** method prompts for the translate, rotate and scale parameters. The Base X and Y are used for the rotation and scale. The Load and Save functions store and recall the parameters to a .DXY file.

The **Align by Two Pairs of Points** option uses two pairs of source and destination coordinates. The first pair defines the translation as the difference between the source and destination northing and easting.

Local Coordinate Transformation

Translation Points

First Source Point
 Northing: 4768.07489136 Easting: 4942.62719383 Point #: 12

First Destination Point
 Northing: 3998.58691151 Easting: 4085.35992418 Point #: 33

Rotation Points

Degrees of Rotation (dd.mmss): 86.205224280

Second Source Point
 Northing: 4763.58402514 Easting: 4304.29082980 Point #: 44

Second Destination Point
 Northing: 4661.03001519 Easting: 4038.39555830 Point #: 129

Scale Points Scale: 1.04034394

OK Cancel Help

This destination point is also the pivot point for rotation. Rotation can be entered directly or defined by a second pair of points where the bearing between the first and second source points is rotated to align with the bearing from the first and second destination points. There is an option to also apply scaling. The scaling holds the angle between points and adjusts the distances by the scale factor. The scale factor is calculated for each point as the elevation factor at the first source point times the grid factor at the first destination point averaged with the elevation factor at the transform point times the grid factor at the transform point.

The **Least-Squares Best-Fit** option is used when there are more than two pairs for translation points. Since two pairs of points are sufficient to define the translation and rotation, more than two pairs of points provides more than enough information.

Local Coordinate Transformation

Scale: 1.0403439565 Rotation: 86°20'52.24" DX: -8528.204 DY: -604.289 Avg HRes: 0.000

Coordinate Display: Source Destination

Average Residual Display: All Process Only

Transformation Type: 2D 3D

Point#	Northing	Easting	Elevation	H Residual	V Residual	Process
12	4768.075	4942.627	9.712	0.000		Y
44	4763.584	4304.291	11.445	0.000		Y

Horizontal Transformation: Over Determination by Plane Similarity Rigid Body (No Scale Factor) Helmert 7-Parameter

Vertical Transformation: Level Plane Tilted Plane

Helmert 7-Parameter Transformation

7-Parameter Values: Calculate From Control Points User Entered

Translate Dx: 0.0000 Dy: 0.0000 Dz: 0.0000

Rotate (seconds) Rx: 0.0000 Ry: 0.0000 Rz: 0.0000 Scale (ppm): 1.0000

Add Edit Delete Optimize Report Process On/Off

OK Cancel Load Save Help

Over Determination by Plane Similarity is used to find the least squares best fit transformation for all the given source and destination points. Besides doing a translation and rotation, this option will also scales the points during the transformation. The **Rigid Body Transformation** also does a best fit least squares transformation, but applies

only translation and rotation with no scale.

The **Helmert 7-Parameter** method can also be used for local transformations. The 7-Parameter Values can be calculated from control points or entered by the user. The program uses the Helmert Transformation, Strict formula:
 $[X_t \ Y_t \ Z_t] = [dX \ dY \ dZ] + M * R * [X_s \ Y_s \ Z_s]$
where R is the rotation matrix.

The Transformation Type chooses between doing a 2D transformation and 3D transformation. For the 3D transformation, the program transforms the x/y using the same method as the 2D transformation, and the z is transformed using an elevation difference model that is modeled by either a best-fit level plane or tilted plane as set by the **Vertical Transformation** setting.

The **Add** button is used to define the source and destination coordinates for the points that define the transformation. Pressing this button brings up the following dialog box.

Source Coordinate	
Point Number	34
Northing	4048.75387
Easting	4088.82307
Elevation	17.97471

Destination Coordinate	
Point Number	243
Northing	4326.12568
Easting	4377.69479
Elevation	20.10343

The **Edit** button is used to edit existing data.

The **Delete** button removes the source and destination pairing from the transformation setup.

The **Process On/Off** button allows source and destination pairings to be turned on and off. This is useful when wanting to inspect different results using different pairings.

The **Optimize** option chooses which point pairings would yield the best transformation results by turning off the processing of pairings with higher residuals. This minimizes the average residual for the control points.

The **Report** option displays a report of the transformation point pairings, their residuals, processing status, transformation scale and avg. residual.

The **Load** and **Save** options allow for saving and recalling local coordinate transformation pairings and settings.

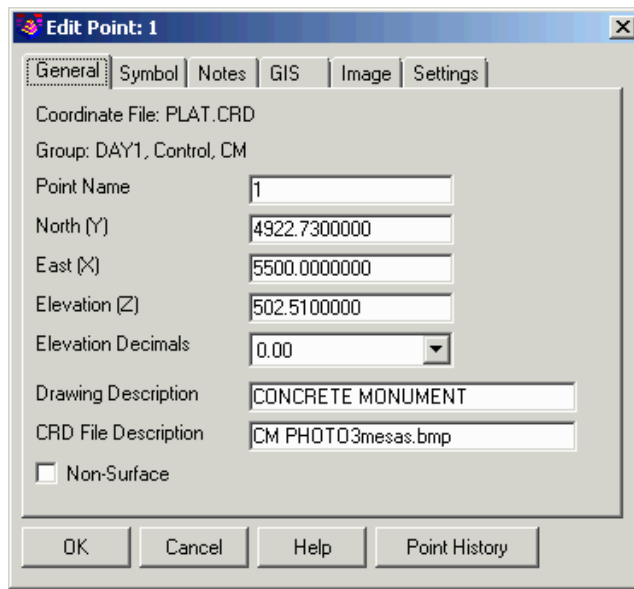
Pulldown Menu Location(s): Points

Keyboard Command: cfutransform

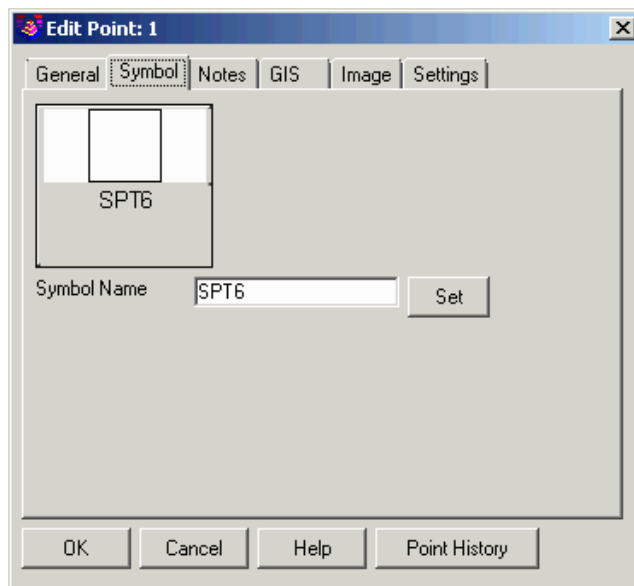
Prerequisite: Drawing entities or coordinate points

Edit Point Attributes

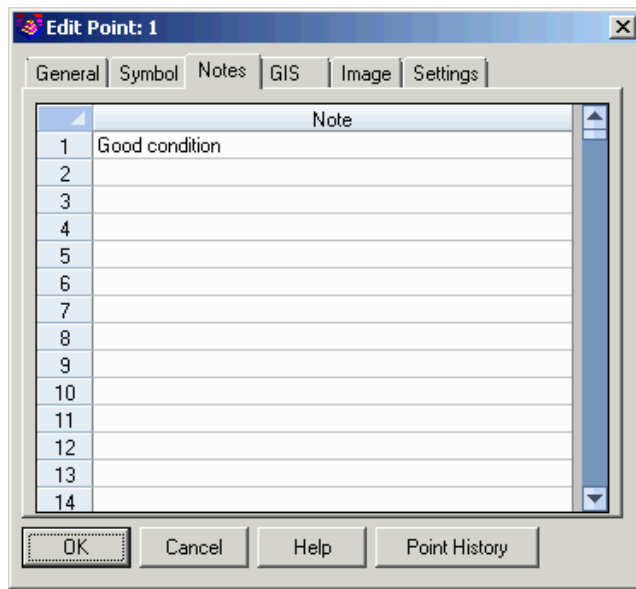
This command will edit the attributes of a Carlson point, such as the symbol type, point number, elevation and description. When this command is invoked, the command line will prompt the user: **Select point to edit (Enter to end)**. At this point, you can select any part of the point including the symbol, elevation, point number or the description. Next, a dialog will appear as shown.



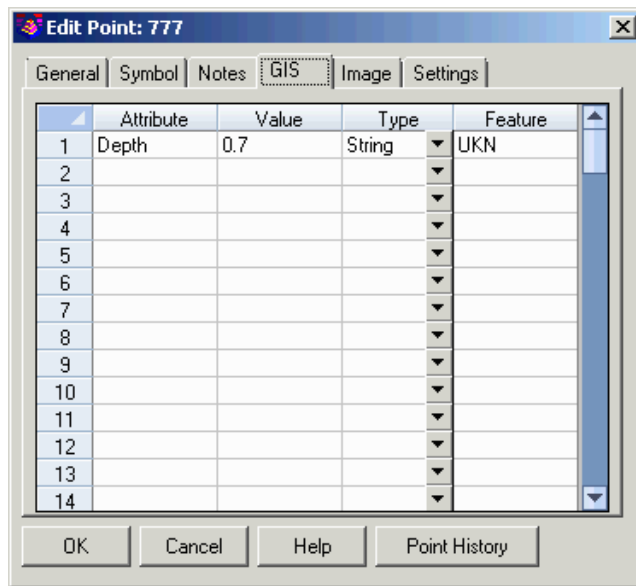
The dialog has several tabs. On the General tab, the name of the coordinate file for the point is displayed at the top along with any point groups that contain this point. The Elevation Decimals setting for the point elevation label. Both Drawing Description and CRD File Descriptions are displayed. When a change to the Drawing description is made, this change will not be reflected in the coordinate file. This allows the change of a description that is defined in the Field to Finish (fld) table for a particular code. If a change is made in the CRD File description, it will be reflected in the coordinate file. Take note that if the CRD file description is changed, running Field to Finish will change the definitions for the point changed. If you change the point number to a number that already exists in the current CRD file, and point protect is ON, you will be prompted whether to overwrite or renumber. The Non-Surface toggle controls whether this point is filtered out in surface modeling with Triangulate & Contour. This Non-Surface setting is the same that the Non-Surface Points routines use in the 3D Data menu. The properties that you modify, with the exception of Drawing Description, will update the current CRD file. All modifications will update the point screen entities. Edits to the elevation and Non-Surface setting will update any triangulation surface created by Triangulate & Contour that used the point for input.



To change the symbol, go to the Symbol tab and either type in a new symbol name in the edit box, or choose the Symbol or Set button where you can choose from a list of symbols from the Symbol Library.



On the Notes tab, you can edit notes for the point. These notes are free-form additional descriptions for the point.



On the GIS tab, you can edit GIS attributes for the point. These GIS attributes are only available for CRDB format coordinate files. See the GIS Data section of the manual for more information on the CRDB GIS data.



The Image tab shows any images assigned to the point. The Open function will open the image in the image program set in Windows for the image file type. The Add and Remove functions are to add and remove images assigned to the point. The Next and Previous buttons are for selecting different images when the point has multiple images.

The Settings tab has a control for the number of decimals to use in the dialog.

Selecting the Point History button will bring up another dialog box that displays the point history of the point chosen. A history of the point will be listed, but only if, under General Setting, the Maintain CRD History File had been set to ON (selected) for the coordinate file that you are working with. With the CRD History feature of Carlson, all point changes can be rolled back.

You may also choose to use the *CAD DDATTE* command to change the attributes of a point. If you do this, then the CRD file will not be updated and if you change the elevation attribute, the point will not change its current Z location.

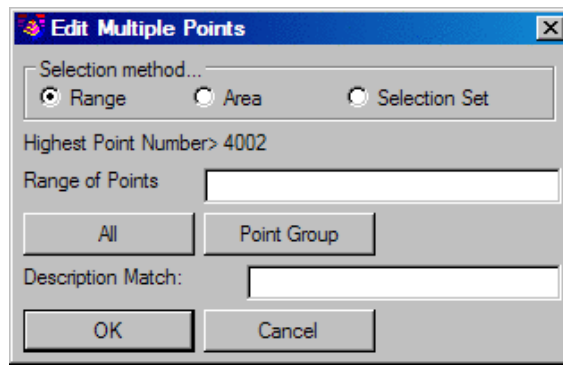
Pulldown Menu Location: Points

Keyboard Command: editpnt

Prerequisite: Carlson points

Edit Multiple Pt Attributes

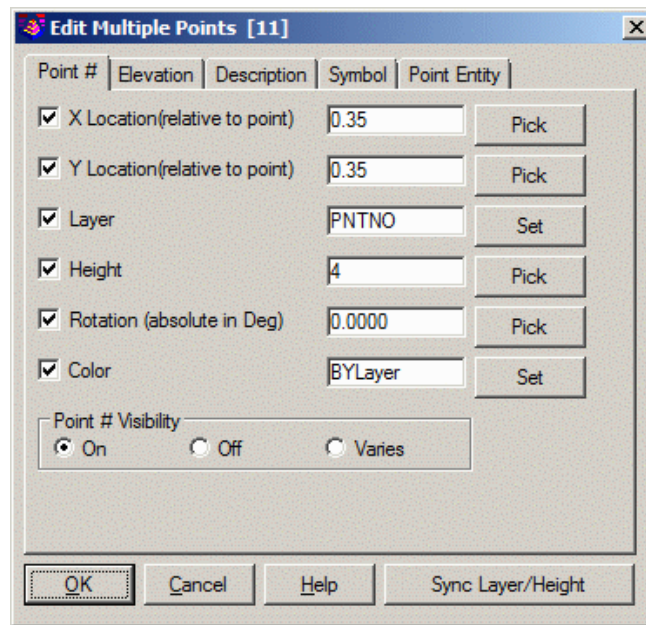
This function allows you to modify the properties of multiple point attributes at the same time. This command gives you complete control over the Carlson point attributes that are present in the drawing. Changes can be made to each attribute – the point number, elevation, description or symbol – all in one motion. For example, you could rotate the elevation text of some points to 45 degrees, change the height of the description text for all the points in the drawing, or change the layer for a particular attribute. Once this command is chosen, the entry Edit Multiple Points dialog, a smaller box, appears. Here you can determine your point selection method. There is also an option for description matching.



After the selection of the points to change, click OK, and the subsequent, larger Edit Multiple Points dialog boxes will appear. The number of points selected will be shown at the top of the dialog boxes.

Edit Multiple Points dialog

For each attribute, you can change any number of the properties, including the layer, height and rotation. These dialogs will reflect the current status of each attributes properties. If, for example, you select 10 points, and 5 of them have the elevation rotation set at 45 degrees, and the other 5 are set at 0 (zero) degrees, then the rotation edit field will say *varies* to let you know that the properties of the points you selected are not the same. Here is an example of the dialog box.



The **X location** refers to the distance in the X direction from the center (or insertion point) of the point symbol. The **Y location** refers to the distance in the Y direction from the center (or insertion point) of the point symbol.

The **Layer** refers to the layer of the individual attribute, not the entire attribute block. To change the layer of the entire attribute block, use the **Attribute Block Layer** option. The **Height** is expressed in real units (generally feet or meters), not plotted size. The **Rotation angle** is expressed in absolute decimal degrees. The **Color** can be set ByLayer or to a specific color. The **Point Entity Layer** refers to the layer that the node of the point resides. The required layers can either be typed in manually, or the Select button can be used to pick from the existing layers in the drawing. If a new layer is desired, simply type in the name of the new layer and it will be created automatically. Use the layer property manager to edit the properties of this new layer, if required. The **Visibility** setting allows for attributes to be shown or hidden in the drawing.

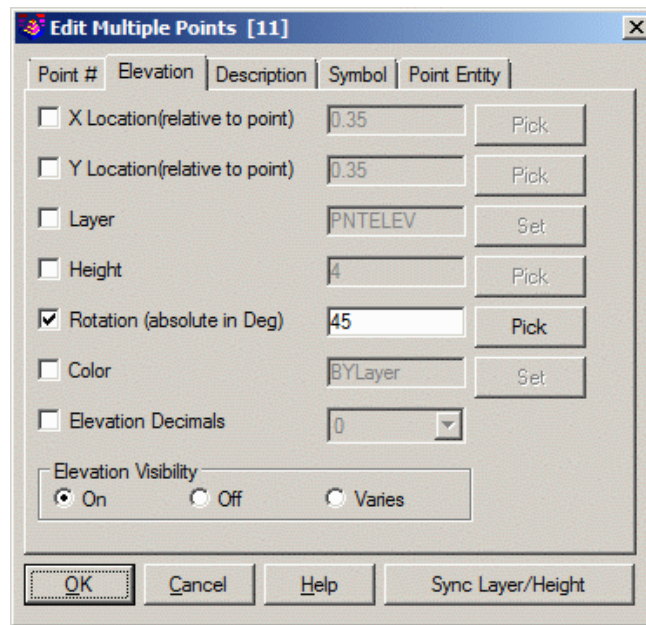
To change a point symbol, check on the **Symbol** tab and use the select button to choose the desired symbol. On the Point Entity tab, the **Attribute Layout ID** refers to the attribute layout style defined in Point Defaults or Field to Finish code definitions. This option allows you to change the particular layout with one of the other available styles or to a customized style if defined. The Pick buttons allow you to pick two points to define a distance (or angle in the case of Rotation). If you want to select a line to define a distance or angle, select two points on the line with the appropriate OSNAP.

Select the attribute to edit, make the necessary changes to this attribute and then move on to the next attribute if required. Changes made to the attributes are remember individually, which allows for switching back and forth though the attributes until the command is completed. After completion the new settings for the point attributes will be retained until changed or redrawn on the screen.

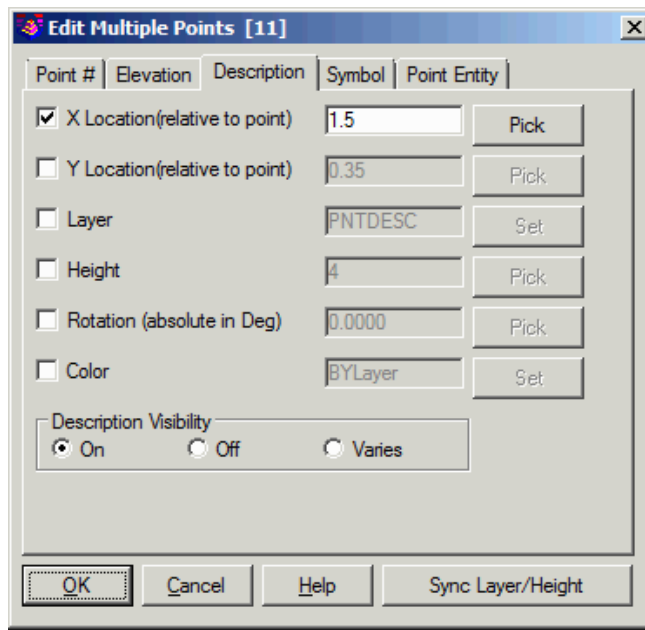
The **Sync Layer/Height** function sets the layer or height for some or all the attributes. The layer and height can be entered manually or pick an existing attribute to get the value. The **Save** and **Load** functions are a way to store and recall all the point display settings to a .PT file for having different point styles to reuse or share.

Example sequential use of Edit Multiple Points dialog

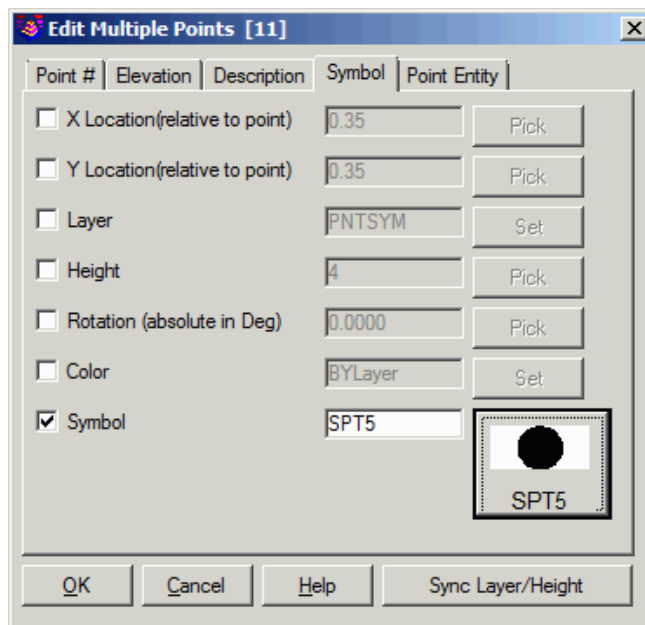
Again, the number of points selected will be shown in the dialog title. Let's now define the changes for each attribute individually. In the following example, suppose we want to rotate the elevation text to a 45 degree angle, move the description to the right and change the symbol. First, click on the Elevation for the Attribute to Edit. Now, select the Rotation option and type in 45. The dialog box should be as below.



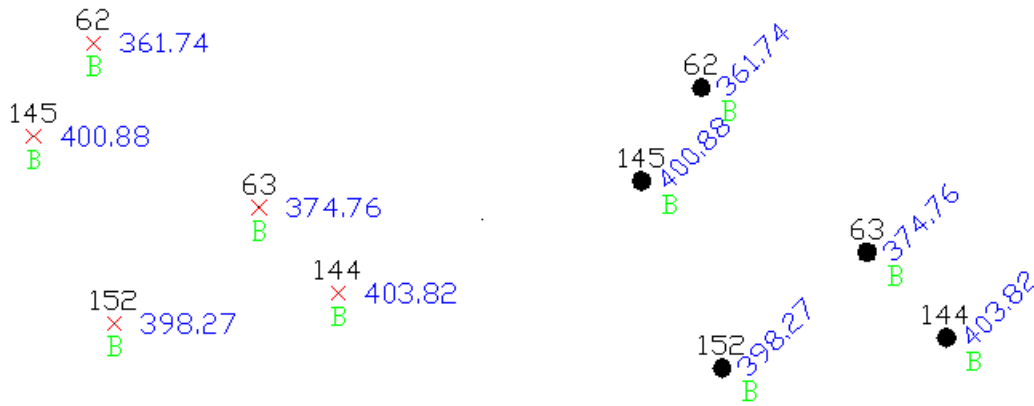
Now, select the Description option for the Attribute to Edit. Select the X location from the Items to Change. Enter 1.50 in the box. This value makes the description line up better with the rotated elevation. The dialog should be as below:



Now, for the final change, select the Symbol for the Attribute to Edit. We want to actually change the point symbol. To do this, toggle on the option to change the symbol by clicking in the box beside the word Symbol. Next, press the Select button and select symbol SPT5. The dialog should be as below:



At this point we are ready to select the OK button to perform the changes. The following image shows the points before and after the changes.



Before and After Changes

Pulldown Menu Location: Points
Keyboard Command: modpnts
Prerequisite: Points drawn on the screen

Dynamic Point Attributes

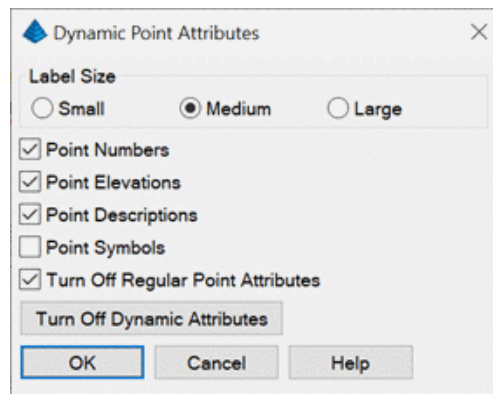
This command creates point attributes that dynamically resize to view in the current zoom level. This way the point labels appear the same size at any drawing zoom level. These point attributes are for viewing only and are not saved to the dwg file.

Label Size: Sets the label size.

Point Numbers, Elevations, Descriptions, Symbols: Choose which labels to view as dynamic.

Turn Off Regular Point Attributes: Hides the regular point entities in the drawing to make viewing the dynamic point attributes clearer.

Turn Off Dynamic Attributes: Turns off all the dynamic point attributes and returns to regular points.



Pulldown Menu Location: Points
Keyboard Command: pntlab
Prerequisite: Points in the drawing

Move Point Attributes Single

This command allows the user to move Carlson point attributes (including the point number, elevation or description) one at a time.

Prompts

Select Point Number, Elevation, or Description to Move: *select point attribute*

Pick new location: *pick point*

Pick new angle: *pick new angle or press Enter*

Pulldown Menu Location: Points

Keyboard Command: movepnt

Prerequisite: Carlson points

Move Point Attributes with Leader

This command allows you to move Carlson point attributes (including the point number, elevation or description), and to draw a dynamic leader to the point node. Leaders and arrowheads may be customized by selecting Options at the command line. If you redraw a point with Draw-Locate Points or Field-To-Finish, the program will recall the moved attribute location to redraw the attributes in the same location as long as the point coordinate is the same.

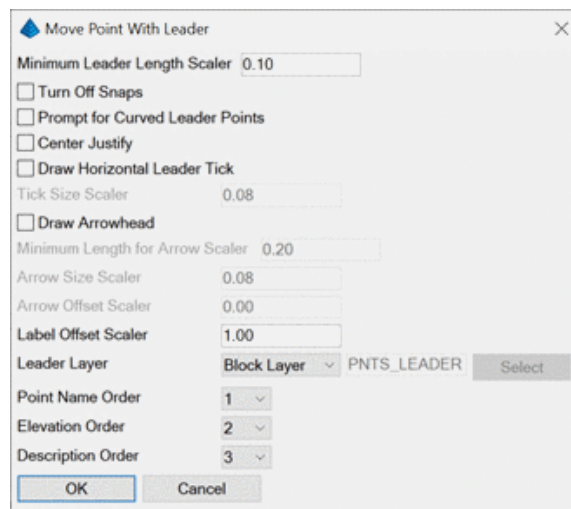
The **Options** keyword brings up the settings dialog shown below. The **Restore** function prompts to pick a point label that has been moved and puts this label back to its default position. The **Multiple** method moves all of the selected points instead of one at a time. The **Group** function selects the points to move by Point Group Name. The **Number** function selects the point to move by point number instead of screen pick.

Prompts

Select Point Label to Move [Options/Restore/Multiple/Group/Number]: *select point attribute*

Pick label position: *pick point*

Select another Point Label to Move [Options/Restore/Multiple/Group/Number]: *O*



Minimum Leader Length Scaler: Specifies the minimum length, in terms of multiples of the attribute block's height, that the leader must be.

Turns Off Snaps: Temporarily turns off any object snaps while picking the label position.

Prompt for Curved Leader Points: This option prompts for additional leader points and creates a smooth leader that goes through all the leader points.

Center Justify: This option uses center justification for the labels. Otherwise the attributes are always justified left or right depending on which side the leader starts.

Draw Horizontal Leader Tick: Specifies whether or not to draw a terminating tick (a short horizontal line segment

sometimes referred to as a "hook line").

Draw Arrowhead: Specifies whether or not to draw an arrowhead at the end of the leader that points to the point entity.

Minimum Leader for Arrow Scaler: Specifies the minimum length of the leader, in terms of multiples of the attribute block's height, that the leader must be before an arrowhead is placed on it.

Arrow Size Scaler: scale factor to apply to resize the arrowhead symbol.

Arrow Offset Scaler: distance from the point node to the tip of the leader.

Leader Offset Scaler: distance from the end of the leader and the labels.

Leader Layer: Specifies whether to use a layer other than that of the identified point for the leader. Use the **Select** button to choose an alternative layer for the leader.

Point Name Order, Elevation Order and Description Order: These settings control the sequence of the attributes for the leader.

Pulldown Menu Location: Points

Keyboard Command: movepntleader

Prerequisite: Carlson points

Scale Point Attributes

This command will scale point attribute text (number, elevation and descriptions) and point symbols up or down in size. The routine prompts for a scale multiplier and a selection set of objects. If you want to enlarge, enter a value greater than one. If you want to reduce, enter a decimal fraction such as .5. This would reduce the text size by 50%. This command is very useful if you have set up your drawing for one plotting scale and decide to change to a new plotting scale. This command has the added benefit that it will adjust the point attributes and symbols to a new screen twist angle.

Prompts

Scaling Multiplier <0.500>: 2.5 This response would enlarge the point attributes and symbols by 250 percent.

Scale symbols only, point labels only or both [Symbols/Labels/<Both>]? *press Enter*

Select points from screen, group or by point number [<Screen>/Group/Number]? *press Enter*

Select Carlson Software points. *pick a point*

Select objects: Specify opposite corner: *pick a point*

Scaling Carlson Software Point Attributes

Number of entities changed> 174

Pulldown Menu Location: Points

Keyboard Command: pntenl

Prerequisite: Carlson points

Erase Point Attributes

This command allows you to erase point attributes like the number, elevation or description individually by picking on the attribute to erase.

Prompts

Select Point No., Elev, or Desc to Erase: *select point attribute*

Pulldown Menu Location: Points

Keyboard Command: erasepnt

Prerequisite: Carlson points

Twist Point Attributes

This command will rotate the orientation of the text of Carlson point attributes (point #, elevation, description) and/or point symbols. The Twist Screen option aligns the point attributes to appear horizontal in the current twist screen. The Azimuth option allows you to enter an azimuth or pick two points to align the point attributes. The Entity Segment option aligns the point attributes by the selected line or polyline segment in the direction the entity is drawn. The Follow Polyline option aligns the point attributes by the polyline segment that is closest to the point.

Prompts

Twist by [<Twist screen>/Azimuth/Entity segment/Follow polyline]? *F* for follow

Attributes to twist [<All>/Symbol/Name/Elevation/Description]? *press Enter*

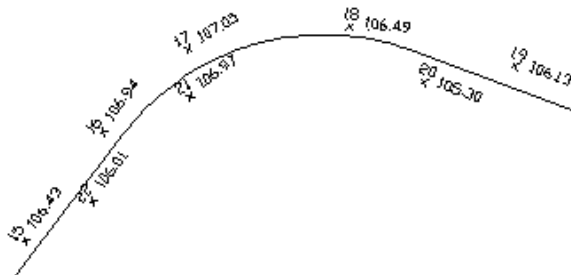
Select reference polylines to follow.

Select objects: *pick a polyline*

Select points from screen, group or by point number [<Screen>/Group/Number]? *select Enter*

Select Carlson Software points.

Select objects: *pick the Carlson point inserts*



Point attributes aligned by Follow Polyline option of Twist Point

Pull-down Menu Location: Points

Keyboard Command: twistpts

Prerequisite: None

Resize Point Attributes

This command sets the size of the selected point attributes (point number, elevation, description) and point symbols. This command is similar to Scale Point Attributes, but instead of scaling the size by a factor, all the select points are set to the same specified size. Points can also be chosen based upon Point Groups.

Prompts

Enter point attribute and symbol size <4.0>: *press Enter*

Scale symbols only, point labels only or both [Symbols/Labels/<Both>]? *press Enter*

Select points from screen, group or by point number [<Screen>/Group/Number]? *press Enter*

Select Carlson Software points.

Select objects: *pick the point entities*

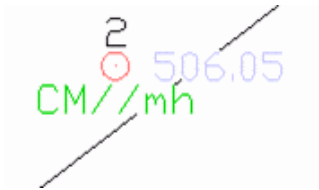
Finding Carlson Software Point Attributes

Number of entities changed > 10

Pulldown Menu Location: Points
Keyboard Command: sizepnt
Prerequisite: Carlson points

Mask Point Attributes

This command creates wipeouts behind the selected point attribute labels (point number, elevation and description) to hide other drawing entities that cross the point attributes.



Prompts

Select point attributes to mask.
Select objects: *pick points*
Offset <0.100>: *press Enter*

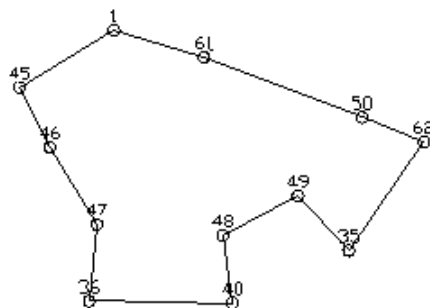
Pulldown Menu Location: Points
Keyboard Command: ptmask
Prerequisite: Points

Trim by Point Symbol

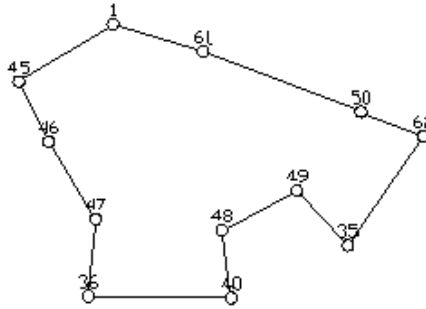
This command will trim lines and polylines that pass through the selected point symbols such that the lines do not appear within the symbol. This should be a last step because this routine explodes the points and modifies the lines and polylines by trimming which makes these entities unusable by some of the other COGO routines.

Prompts

Select Carlson Software point symbols to trim against.
Select objects: *select the point symbols*



Before Trim by Point Symbol



After Trim by Point Symbol

Pulldown Menu Location: Points
Keyboard Command: trimpts
Prerequisite: Carlson point symbols

Freeze Point Attributes

This command freezes the point attribute layers for all Carlson points in the drawing. The attribute layers include the point number, elevation and description. This command is a quick way to hide all point attributes from view without erasing them. Use the Thaw Point Attributes command to show the points again.

Pulldown Menu Location: Points > Point Utilities
Keyboard Command: freeze_ptattr
Prerequisite: Carlson points

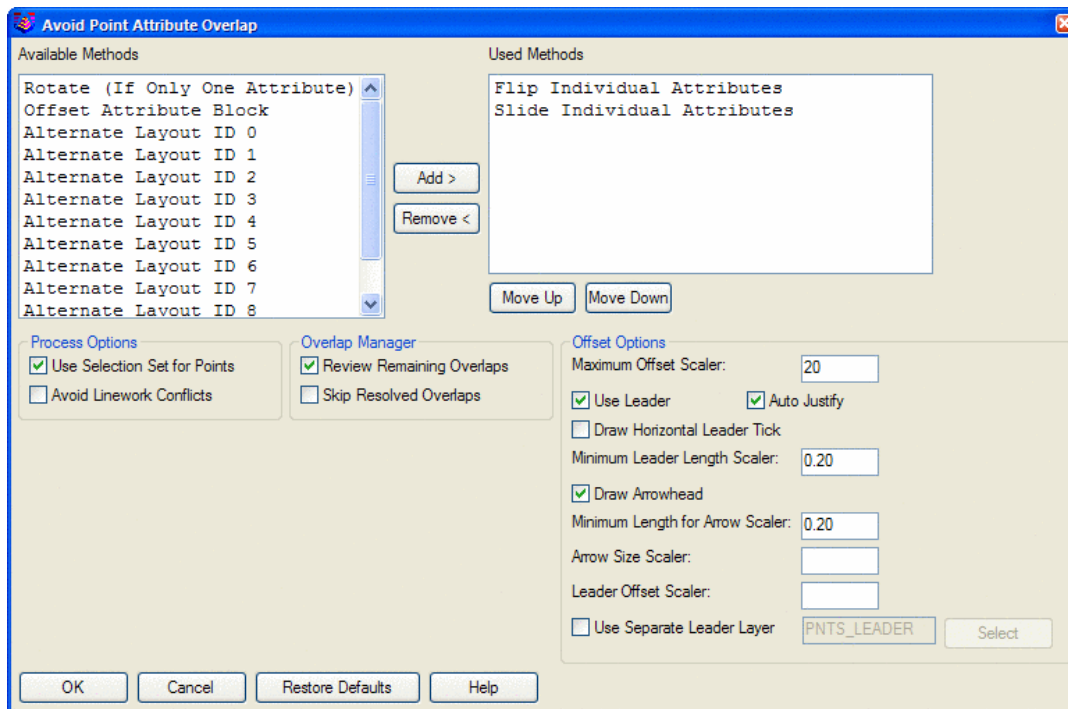
Thaw Point Attributes

This command thaws the point attribute layers for all Carlson points in the drawing to show the point number, elevation and description attributes.

Pulldown Menu Location: Points
Keyboard Command: thaw_ptattr
Prerequisite: Carlson points

Fix Point Attribute Overlaps

This command is to be used to adjust point attribute labels to avoid overlapping labels. It applies adjustment methods based upon user-specified ordering and tolerances. The command steps you through any remaining overlaps in an Overlap Manager, which includes the capability to manually move labels. This point overlap feature is also available within the Draw-Locate Point and Field To Finish commands.



Methods: There are different methods of automatically solving a point attribute overlap. The methods will be applied in order from top to bottom on the Used Methods list. Unused methods appear on the Available Methods list. The methods are:

Alternate Layout ID 0-9

These methods will simply apply the specified attribute layout ID and then check to see if the attributes of the point in question still overlap. The different attribute layout IDs can be seen in the *Point Defaults* command on the Points menu.

Flip Individual Attributes

This method tests each attribute (point #, description, and elevation) by flipping it or mirroring it the other side of the point. The mirror is the vertical axis of the text that goes through the point entity. This method is not applied to points that have a leader.

Slide Individual Attributes

This method tests each attribute (point #, description, and elevation) by sliding it back and forth. The maximum distance the attribute will be moved is the horizontal length of the text. This method is not applied to points that have a leader.

Rotate (If Only One Attribute)

This method is applied if there is only one point attribute, either point #, description, or elevation. The one attribute is rotated around the point entity to see if the point overlap can be fixed.

Offset Attribute Block

This method is arguably the most powerful method and can solve any overlap by moving the attribute block far enough. See *Offset Options* below for a description of the options that can be used with this method.

Offset Options: These are the options that apply to the Offset Attribute Block method of automatically solving point attribute overlaps.

Maximum Offset Scaler: This specifies the maximum distance, in terms of multiples of the whole attribute block's height, that the attribute block may be offset from the point entity.

Use Leader: Specifies whether or not a leader should be drawn when offsetting the attribute block.

Minimum Leader Length Scaler: Specifies the minimum length, in terms of multiples of the height of an attribute's text, that the leader must be.

Draw Arrowhead: Specifies whether or not to draw an arrowhead at the end of the leader that points to the point entity.

Minimum Leader for Arrow Scaler: Specifies the minimum length of the leader, in terms of multiples of the height of an attribute's text, that the leader must be before an arrowhead is placed on it.

Arrow Size Scaler: Specifies a scale factor to be applied to control the size of the arrowhead if drawn.

Leader Offset Scaler: Specifies the length, in terms of multiples of the height of an attribute's text, that the leader arrowhead should be offset from the point.

Use Separate Leader Layer: If enabled, allows the user to define a different layer on which to place the resultant leader.

Use Selection Set for Points: Check this checkbox to be given the option of selecting which points in drawing to fix overlaps with. If not checked, then all the points in the drawing are used.

Avoid Linework Conflicts: Check this checkbox to prevent point attributes from overlapping linework in addition to other point attributes.

Review Remaining Overlaps: Check this checkbox to have the Overlap Reviewer dockable dialog come up after the automated process finishes. The Overlap Reviewer allows for reviewing the automated fixes as well as tools for manually fixing any remaining overlaps. See Overlap Reviewer below for more information.

Skip Resolved Overlaps: Check this checkbox to skip overlaps that were automatically resolved and to only review unresolved overlaps. If not checked, then both resolved overlaps and unresolved overlaps will be available for review. This option only applies if Review Remaining Overlaps is on.

Overlap Reviewer

The Overlap Reviewer will come up after automatic overlap fixing if the Review Remaining Overlaps checkbox was checked. This tool displays how many points were found, how many overlaps were fixed, which overlap is currently being viewed, how many overlaps there were total, and the point # of the current overlap. Use the First, Last, Back, and Next buttons to navigate forwards and backwards through the list of overlaps. Use the Move Block and Move Attrs buttons to manually move either the entire attribute block or individual attributes.



Pan and Zoom Controls: Use the buttons on the top to help zoom in and out and pan the drawing around. You can also use the standard mouse controls for panning and zooming.

First, Last, Back, and Next: These buttons allow you to step through each overlap or to jump to the first or the last.

Status: This drop-down list indicates the status of the current overlap. *open* means that the overlap has not been fixed yet. *resolved* means that the overlap has been fixed. *ignore* can be chosen by you to remove the overlap from the list.

Restore: Restores the attributes of the current point to their original location and rotation from before the Fix Point Attribute Overlaps command was run.

Move Block: Allows you to move one or more attribute blocks in the drawing. See the documentation for *Move Point Attributes with Leader* command in the Points menu.

Erase Attrs: Allows you to erase selected point attributes.

Move Attrs: Allows you to move and rotate one or more individual attributes in the drawing. See the documentation for *Move Point Attributes* command in the Points menu.

Auto-Zoom: Check this checkbox to automatically zoom and pan the view as each overlap is viewed.

Prompts

The following prompt will be displayed if the Use Selection Set for Points checkbox is on and OK is pressed.

Select the points to fix overlaps with: *pick the Carlson point inserts*

Pulldown Menu Location: Points

Keyboard Command: overlapppts

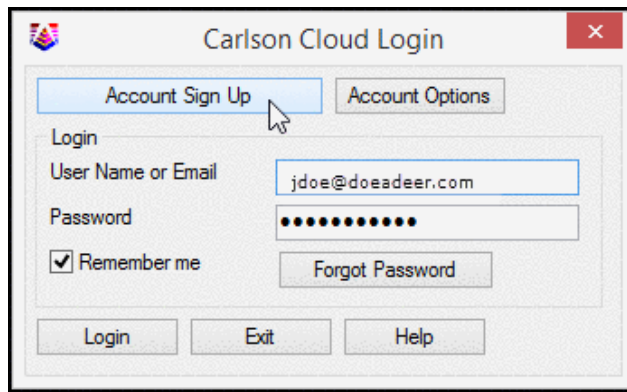
Prerequisite: Points in the drawing

Coordinate Cloud

The Coordinate Cloud command (not to be confused with the Carlson Point Cloud module) is an Internet/cloud-based routine designed to easily facilitate storing and sharing coordinate point information either publicly or privately.

Prior to using the coordinate cloud command, the user must have created a Carlson Cloud account.

Upon launching Coordinate Cloud, the user is challenged to provide login credentials as shown below:



- **Account Sign Up**
- **Account Options**

Username: Supply the user name (Case Sensitive) that you used when creating your Carlson Cloud account.

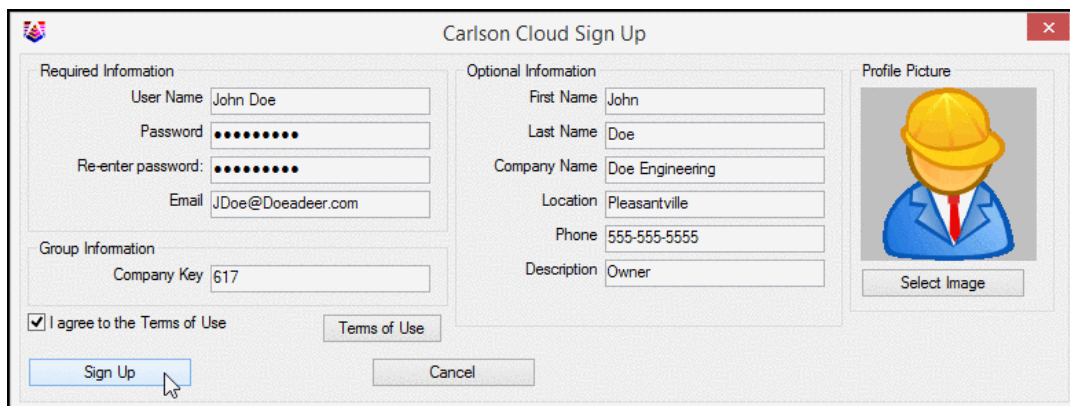
Password: Supply the password (Case Sensitive) that you used when creating your Carlson Cloud account.

Remember Me: When enabled, the last used *Username* and *Password* values are recalled for the next Carlson Cloud login session.

Forgot Password: Use if you have forgotten your password and need it emailed to you or reset.

Once logged in, refer to the Carlson Cloud - Logged In discussion.

The *Account Sign Up* command provides the ability to create a new user account and associate the user account with a "company key" (essentially, a group of people with whom you wish to network within Carlson Cloud). See the *Carlson Cloud* command for more information



Username: Supply the user name (this will be Case Sensitive) that you will want to use for your Carlson Cloud account.

Password: Supply the password (this will be Case Sensitive) that you will want to use for your Carlson Cloud account.

Email: Supply an Email address where password recovery instructions can be sent.

Optional Information: Supply your name, company, location, phone and additional description

Create Company Key: When enabled, this toggle permits you to create a new company "profile" (or communication group).

Existing Company Key: Indicate the name (Case Sensitive) of an existing company "profile" to which the new member should belong.

Read and agree to the **Terms of Use**

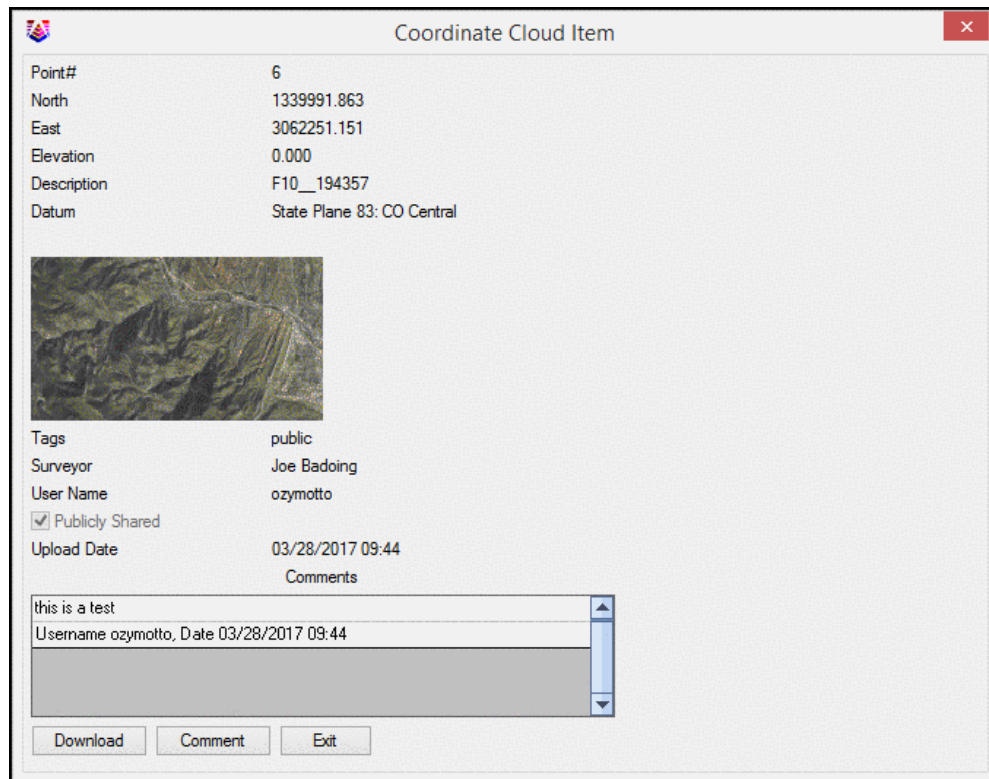
Sign Up: Click this button when complete

Show Public Points: When enabled, all points designated as public will be shown.

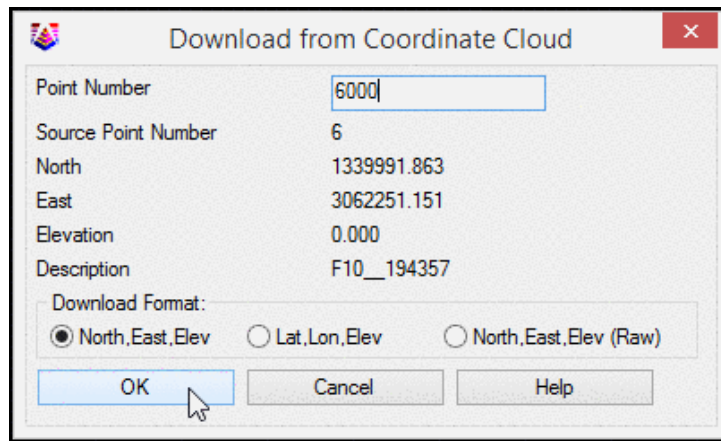
Search by Distance: Will control the search area by zip code, or distance in miles.

Select CRD File: Use this to select a coordinate file from which to upload point data.

View: Highlight a point from the display list and view the detailed information



Download: Download the point data into the current coordinate (CRD)file.



Upload: Upload point data from the current coordinate (CRD) file.

Export to Google Earth: Export the point data to Google Earth

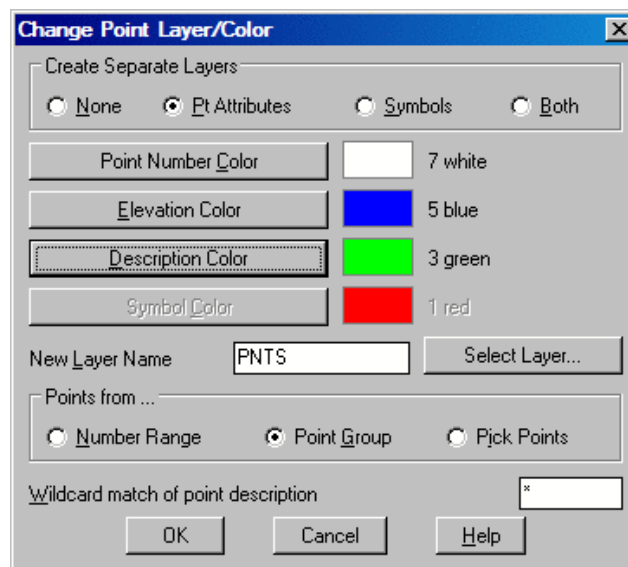
Pulldown Menu Location: Points->Point Utilities

Keyboard Command: crdcloud

Prerequisite: Internet connection, Carlson Cloud account

Change Point LayerColor

This command changes the layer and optionally the color of Carlson points. The points are initially put in the layer set in Point Defaults. The symbol, point number, elevation and description are in the layers PNTMARK, PNTNO, PNTELEV, and PNTDESC. To change the point attribute colors, this routine creates new attribute layers based on the new layer name. For example if the new layer name was TRAV, then the resulting layers would be TRAVMARK, TRAVNO, TRAVELEV and TRAVDESC. These new layers can be given different colors. To select an attribute color, pick on the color button. To permanently change attribute colors, edit the drawing SRVPNO1.DWG in the Carlson SUP directory. To permanently change a symbol color, edit the symbol drawing itself.



The selection of the points to change can be accomplished in three ways. A number range selection would require the input of the range of points to change. An example would be 1-20,25,30, 32-36. Points groups can also be used as a selection method. Simply specify the point group name to change, when prompted, and all the points included

in that group will be changed. The final selection method is that of Pick Points. Using this method a prompt to select objects is displayed. When prompted select the points to change from the screen.

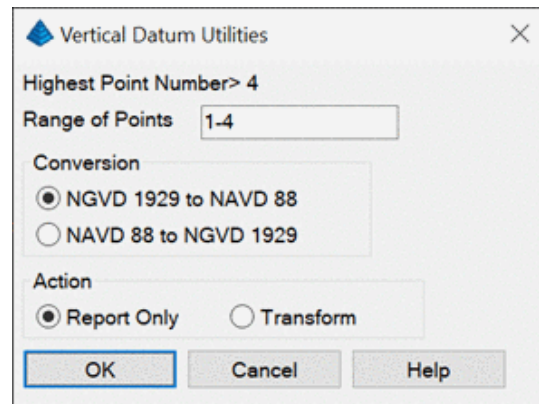
Pulldown Menu Location: Points

Keyboard Command: pntchg

Prerequisite: Carlson points displayed in the graphic drawing window

Vertical Datum Utilities

This command calculates the elevation difference between NGVD 1929 and NAVD 88 for points from the current coordinate file. Before running this command, the grid projection for the drawing must be defined in Settings > Drawing Setup so that the points can be geolocated. The Report Only option uses the Report Formatter to report the elevation differences. The Transform option updates the point elevations in the coordinate file.



Pulldown Menu Location: Points > Point Utilities

Keyboard Command: vdatum_util

Prerequisite: Coordinate file

Link Linework With Points

This command links lines and polylines with points so that the linework updates if the points are updated. This link is the same as option under Carlson Configure > General Settings. The option in Configure is applied when the linework is created. This separate link command allows you to add links on selected linework in case the Configure link option was off when the linework was created.

In order to link points to the linework, the program matches points from the current coordinate file to the linework vertices. The Vertex Tolerance is how close the coordinate point needs to be to the linework vertex to create a link.

To remove point links, run File > Drawing Utilities > Remove Reactors.

Prompts

Enter Point to Vertex Tolerance <0.001>: *press Enter*

Select Lines and Polylines to Create reactors for.

Select objects: *pick the linework*

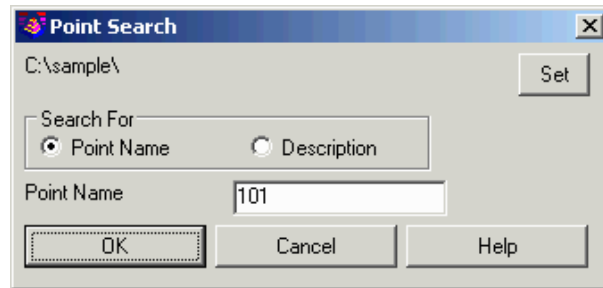
Pulldown Menu Location: Points > Point Utilities

Keyboard Command: linkline

Prerequisite: Points and linework

Search Point Files

This command searches for a point number or description in coordinate and raw files in a folder. The program also searches in all sub-folders of the selected folder. The command reports all files that found a match.



Pulldown Menu Location: Points > Point Utilities

Keyboard Command: ptsearch

Prerequisite: None

Point Attribute Layout Manager

This command helps manage the drawings that are used for Carlson Points for the point attributes of point number, elevation and description. To allow for up to ten different layout styles, there are ten point attribute drawings in the support folder. The support folder path is shown at the top of the dialog. The file names for the point attribute drawings are SRVPNO followed by a digit 0 to 9. For example, SRVPNO1.dwg.

The dialog shows a preview of the point attribute drawing. Select from the Attribute Layout ID list to see different drawings. The Edit function opens the point attribute drawing. Then you can make edits in CAD such as moving or rotating the attributes, and then save the dwg file. The Save and Load functions are a way to store and recall all 10 of the point attribute drawings to a single PBLK file. This provides a way to transfer custom point attribute blocks between computers.



Pulldown Menu Location: Points > Point Utilities

Keyboard Command: pt_attr_mgr

Prerequisite: None

Geoid Utilities

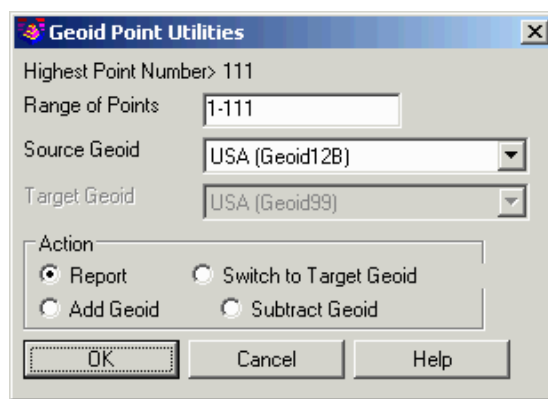
This command has functions for using a geoid with points in the current coordinate file.

Report: Reports the geoid heights at the points.

Add Geoid: Adds the geoid heights to the point elevations.

Subtract Geoid: Subtracts the geoid heights from the point elevations.

Switch To Target Geoid: Allows you to adjust the point elevations to switch from one geoid to another. The geoid used with the original point elevations is specified as the Source Geoid, and the geoid to switch to is set by the Target Geoid.



Pulldown Menu Location: Points > Point Utilities

Keyboard Command: geoid_util
Prerequisite: Points in coordinate file

Renumber Points

This command edits point numbers. This function is also in Coordinate File Utilities. The points to renumber can be selected by point number range, point group, the area inside a closed polyline or by selecting points from the drawing. The Description Match is a way to filter which points to renumber. The Condense Points option removes any gaps in the point numbers to make the renumbered points sequential. There are five renumbering methods:

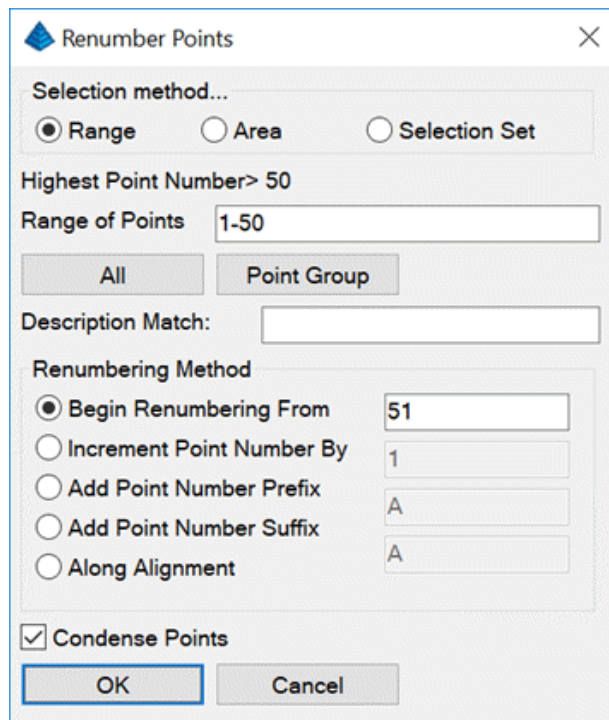
Begin Renumbering From: Points are numbered starting from the specified point number.

Increment Point Number By: Adds this number to the existing point numbers.

Add Point Number Prefix: Adds this string to the start of the existing point numbers.

Add Point Number Suffix: Adds this string to the end of the existing point numbers.

Along Alignment: Sequences the point numbers in station order along a centerline.



Pulldown Menu Location: Points > Point Utilities

Keyboard Command: renumpnt

Prerequisite: Coordinate file

Explode Carlson Points

This command can be useful if you need to send your drawing to another firm who does not have AutoCAD/Carlson. Drawing transfer problems occur when the recipient does not have the same block/inserts defined or available. This command explodes all blocks and replaces the Carlson point attributes with TEXT entities of the same value. After the points have been selected, a prompt for the layer name for each point attribute will be displayed. Point Numbers, Point Elevations and Point Descriptions can be put on user specified layers, or the default for each prompt can be selected. **Caution:** After using this command, the link between the points and the coordinate file are destroyed and

you can no longer extract the attributes from the drawing. If you want to use this command but retain your point information, follow these steps:

1. Save your drawing
2. Run this command to explode the points
3. Execute the SAVEAS command and save the drawing as a different name (you can also choose DXF format if you wish).
4. Exit the drawing **without** saving.

Prompts

This command will explode selected Carlson Software point blocks and replot the attributes as Text entities! The resulting points will NOT be useable by most Carlson Software commands!!!!

Select Carlson Software Points to Explode. *select points*

Layer Name for Point Numbers <PNTNO>: *press Enter*

Layer Name for Point Elevations <PNTELEV>: *press Enter*

Layer Name for Point Descriptions <PNTDESC>: *press Enter*

Number of entities changed> 345

Pulldown Menu Location: Points

Keyboard Command: explode_scad

Prerequisite: Carlson points

Convert Surveyor1 to CRD

This command will convert a Surveyor1 coordinate file to the current Carlson format.

Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: SURVEYOR2CRD

Prerequisite: A Surveyor1 coordinate file

Convert CRD to TDS CR5/Convert TDS CR5 to CRD

These commands convert coordinate file formats between a Carlson CRD file and a TDS CR5 file. Both of these file formats are binary which require these special routines. These commands will prompt for the file names to process.

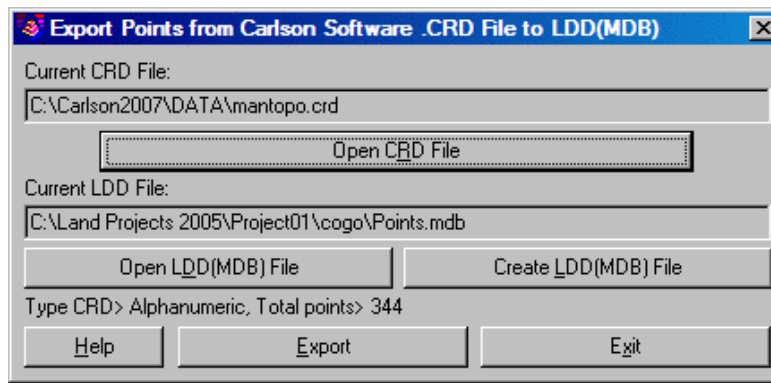
Pulldown Menu Location: Points

Keyboard Command: crd_cr5, cr5_crd

Prerequisite: A CRD or CR5 file

Convert CRD to Land Desktop MDB

This command converts a Carlson CRD file into an Autodesk Land Development Desktop (LDD) point database file in Access MDB format. The LDD point database always has the file name of POINTS.MDB. So, to specify the LDD file to create, you only need to specify the directory/path and not the file name. This path corresponds to the LDD project directory. The conversion program has point protect, so that if a point number from the CRD file already exists in the LDD file, you then will be prompted to skip or replace the point. Once the command is executed, the following dialog is displayed. On this dialog, specify the Carlson CRD file to convert as well as the LDD (MDB) file to append, if existing, or create if creating a new LDD (MDB) file.



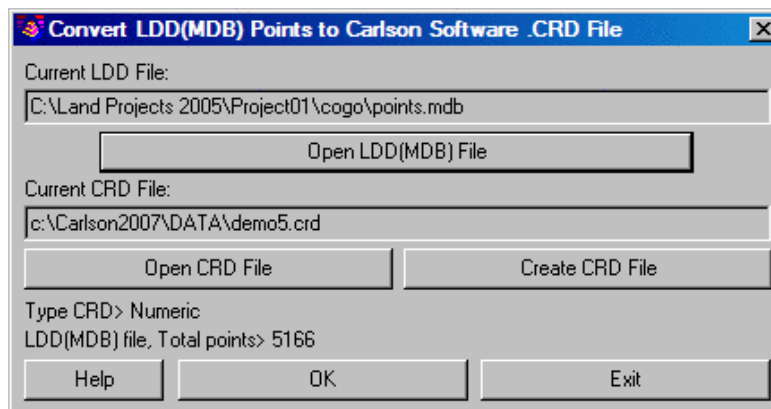
Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: crd_ldd

Prerequisite: A .CRD file

Convert Land Desktop MDB to Carlson Points

This command converts an Autodesk Land Development Desktop (LDD, also referred to as LDT) point database file into a Carlson CRD file. The LDD point database always has the file name of POINTS.MDB and is stored in the LDD project directory. Once the command is executed, the following dialog is displayed. On this dialog, specify the LDD file to convert as well as the Carlson CRD file to append, if existing, or create if creating a new CRD file.



Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: LDD_crd

Prerequisite: An LDD point database file

Convert Civil 3D to Carlson Points

This command converts Civil 3D point entities into Carlson format point entities. When running in AutoCAD, the Civil 3D Object Enabler from Autodesk is used to read the Civil 3D point entities. This object enabler must be installed before running this routine. The installation for the object enabler is located under Support at www.autodesk.com. When running in IntelliCAD, this routine uses a conversion program from the Open Design Alliance to read the Civil 3D point entities.

If you have Civil 3D, another way to make a drawing with standard AutoCAD entities is to use the `aectoacad` command in Civil 3D which converts the custom objects into standard entities.

Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: c3d_crd

Prerequisite: Civil 3D points in the drawing

Convert Carlson Points to Land Desktop

This command converts a Carlson CRD file into a Land Desktop point file. To do this, you must specify the existing Carlson CRD points to convert. You have the option of selecting all points, or selecting on-screen the specific points you'd like to convert.

Prompts

Convert all or selected points [All/<Selected>]? *press Enter*

Select Carlson Software Points to convert:

Select objects: *pick first point for window selection method*

Select objects: *pick second point*

Processing Carlson Software point...

Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: pt_aec

Prerequisite: A Carlson CRD file

Convert Land Desktop to Carlson Points

This command converts Land Desktop point entities into Carlson format point entities. The Land Desktop Object Enabler from Autodesk is used to read the Land Desktop point entities. This object enabler must be installed before running this routine. The installation for the object enabler is located under Support at www.autodesk.com. Be sure to match the version of the object enabler with the Land Desktop version used to create the drawing.

Prompts

Convert all or selected points [All/<Selected>]? *all.* Choose which points to convert.

Point position method [Insertion/<Database>]? *press Enter.* Choose between the drawing insertion points or the point database for the point locations.

Locate points on Real-Z Axis [Yes/<No>]? *press Enter.* Choose between creating the points at their elevation or at zero.

Convert point markers to symbols [<Yes>/No]? *press Enter.* Choose between using a point symbol or the PDMODE.

Pulldown Menu Location: Points

Keyboard Command: ldd_crd

Prerequisite: LDT points in the drawing and the LDT Object Enabler

Convert Softdesk to Carlson Points

This command converts Softdesk point blocks in the drawing to Carlson point blocks. These point block formats are similar and converting only requires reordering and renaming the attributes. Softdesk points can also be read into the current CRD file by using the command *Update CRD File from Drawing* in *Coordinate File Utilities*, this updates the CRD file without modifying the screen entities.

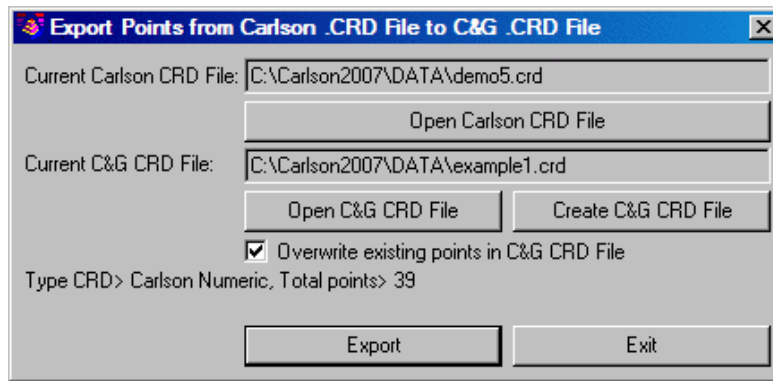
Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: 2surv

Prerequisite: Softdesk points

Convert Carlson Points to C&G

This command converts a Carlson CRD file into a C&G Point file.



Specify the existing Carlson CRD to convert by selecting the Open Carlson CRD File button. Specify the existing C&G CRD file to write to, or the new C&G CRD file to create, by selecting either Open C&G CRD file or Create C&G CRD file. Press OK and the conversion is completed.

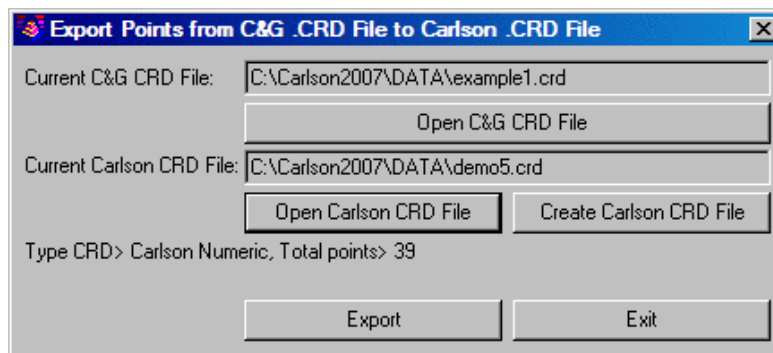
Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: crd2cg

Prerequisite: A Carlson CRD file

Convert C&G to Carlson Points

This command converts C&G Points into a Carlson CRD file.



Specify the existing C&G File to convert by selecting the Open C&G CRD File button. Specify the existing Carlson CRD file to write to, or the new Carlson CRD file to create, by selecting either Open Carlson CRD file or Create Carlson CRD file. Press OK and the conversion is completed.

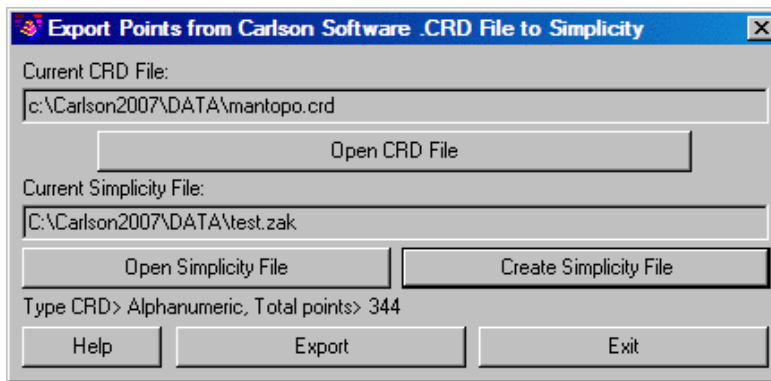
Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: cg2crd

Prerequisite: A C&G point file

Convert Carlson Points to Simplicity

This command will convert Carlson points to Simplicity.



Select Carlson CRD file to convert by selecting the Open CRD file button.

Specify the existing Simplicity file to write to, or the new Simplicity file to create, by selecting either Open Simplicity File or Create Simplicity File. Press Export and the conversion is completed.

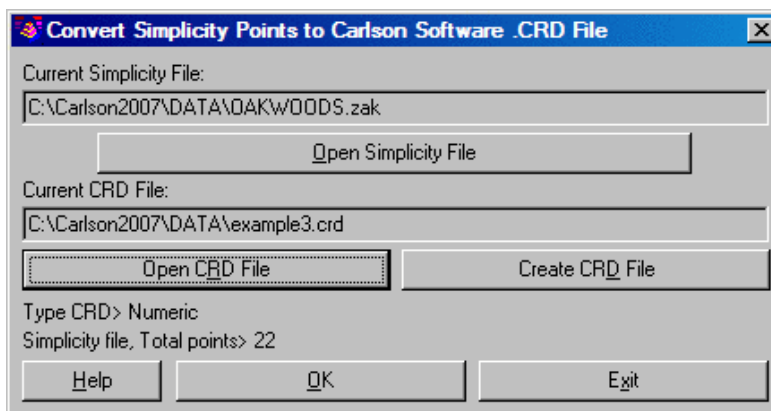
Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: crd_zak

Prerequisite: A Simplicity point file

Convert Simplicity to Carlson Points

This command converts Simplicity Points into a Carlson CRD file.



Specify the existing Simplicity File to convert by selecting the Open Simplicity File button. Specify the existing Carlson CRD file to write to, or the new Carlson CRD file to create, by selecting either Open CRD File or Create CRD File. Press OK and the conversion is completed.

Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: zak_crd

Prerequisite: A Simplicity point file

Convert Leica to Carlson Points

This command converts LisCad or Leica point blocks in the drawing to Carlson point blocks. These point block formats are similar and converting only requires reordering and renaming the attributes. Leica points can also be read into the current CRD file by using the command *Update CRD File from Drawing* in *Coordinate File Utilities*. This updates the CRD file without modifying the screen entities.

Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: 2surv3

Prerequisite: Leica points

Convert Geodimeter to Carlson Points

This command converts Geodimeter point blocks in the drawing to Carlson point blocks. These point block formats are similar, and converting only requires reordering and renaming the attributes. Geodimeter points can also be read into the current CRD file by using the command *Update CRD File from Drawing* in *Coordinate File Utilities*. This updates the CRD file without modifying the screen entities.

Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: 2surv4

Prerequisite: Geodimeter points

Convert Carlson Points to Ashtech GIS

This command converts Carlson point blocks in the drawing to Ashtech GIS point blocks. After executing the command, you will be prompted to select the points to convert. When using this command, the setting "Group Point Entities", found under General Settings of the Configure command (Settings menu) should be unchecked (turned off).

Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: 2ashtech

Prerequisite: Carlson Points

Convert Carlson Points to Softdesk

This command converts Carlson point blocks in the drawing to Softdesk point blocks. These point block formats are similar, and converting only requires reordering and renaming the attributes.

Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: 2soft

Prerequisite: Carlson points

Convert CAICE KCM to Carlson CRD

This command converts a CAICE .KCM point database file to a Carlson CRD file.

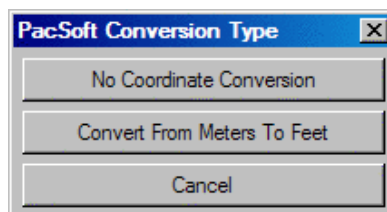
Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: kcm2crd

Prerequisite: CAICE KCM file

Convert PacSoft CRD to Carlson CRD

This command converts a PacSoft CRD file to a Carlson CRD file. PacSoft stores the point descriptions to a separate coordinate descriptor file having an extension of PTD. This file should be present in the same directory as the CRD file to convert. Prompts for the PacSoft CRD file to convert, and the Carlson CRD file to create, will be displayed. Once both files have been specified, the following dialog box will be displayed.



The **No Coordinate Conversion** option converts the file format while leaving the coordinate values unchanged.

Convert From Meters to Feet will assume the coordinates in the selected PacSoft crd file are metric, and will convert the coordinate values to US Feet.

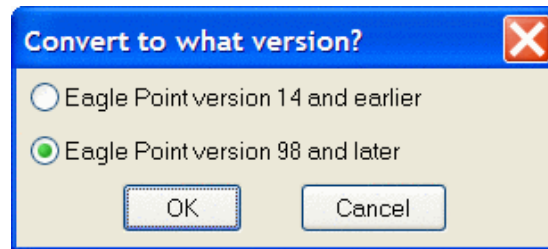
Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: pacsoft2crd

Prerequisite: PacSoft crd file

Convert Carlson Points to Eagle Point

This command converts Carlson point blocks in the drawing to Eagle Point point blocks. A prompt for the Eagle Point version to convert to will be displayed.



Specify the appropriate version and then select the OK button. You will then be prompted to select the Carlson points to convert. These point block formats are similar, and converting only requires reordering and renaming the attributes.

Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: 2eds

Prerequisite: Carlson points

Convert Eagle Point to Carlson Points

This command converts Eagle Point point blocks in the drawing to Carlson point blocks. These point block formats are similar, and converting only requires reordering and renaming the attributes. Eagle Point points can also be read into the current CRD file by using the command *Update CRD File from Drawing*, found in *Coordinate File Utilities*. This updates the CRD file without modifying the screen entities.

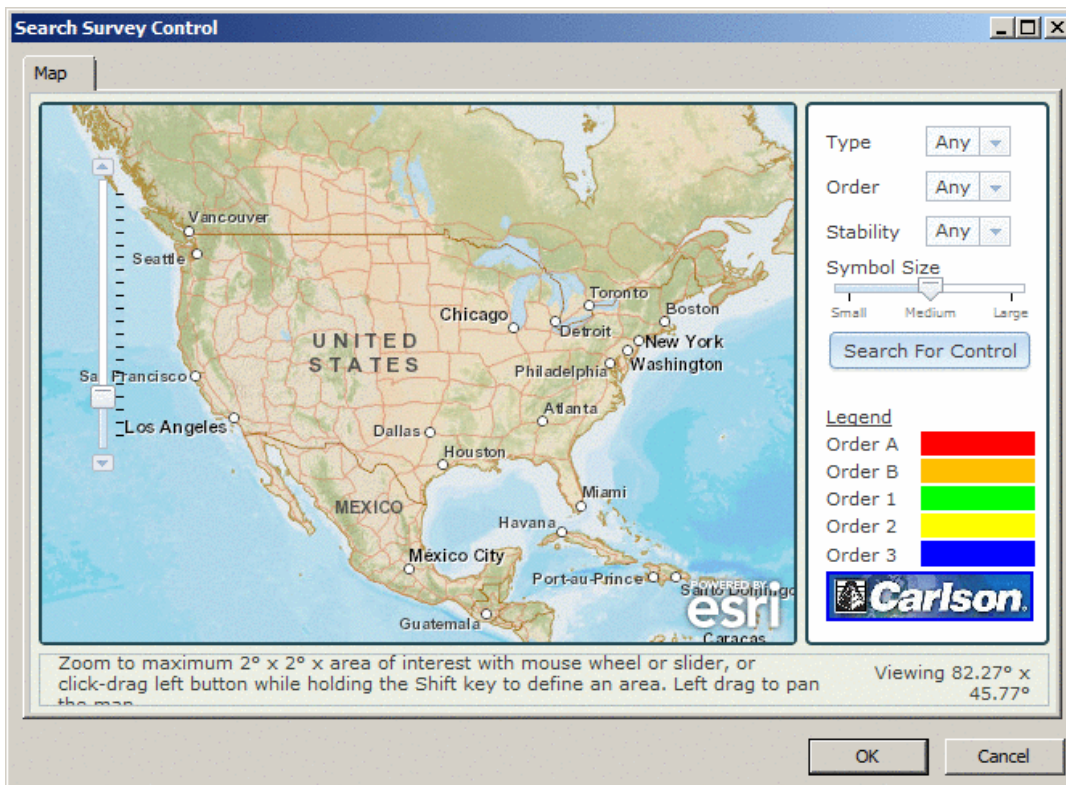
Pulldown Menu Location: Points > Convert Point Format

Keyboard Command: 2surv2

Prerequisite: Eagle Point points

Search Published Control

The Search Published Control command allows you to search published control mark data freely available on the National Geodetic Survey (NGS) web-site (<http://www.ngs.noaa.gov>) and optionally store the retrieved information to the active coordinate file.



Map Tab: Use the Map tab to navigate to a location bounded by a viewing window no greater than 2 degree of latitude by 2 degree of longitude. The limits of the current view are shown at the lower right of the Map tab (see below). To navigate to the area of interest:


- "zoom" using the slider control on the left side of the Map tab, or,
- "zoom" using the mouse wheel, or,
- "zoom window" by holding the Shift key and while left-clicking two points to define a rectangular area, or,
- "pan" by left-clicking and dragging the image to the desired position in the Map tab.

Type: Select the type of the NGS markers that are to be returned.


Order: Select the positional order of accuracy of the NGS markers that are to be returned.

Stability: Select the elevational stability of the NGS markers that are to be returned.

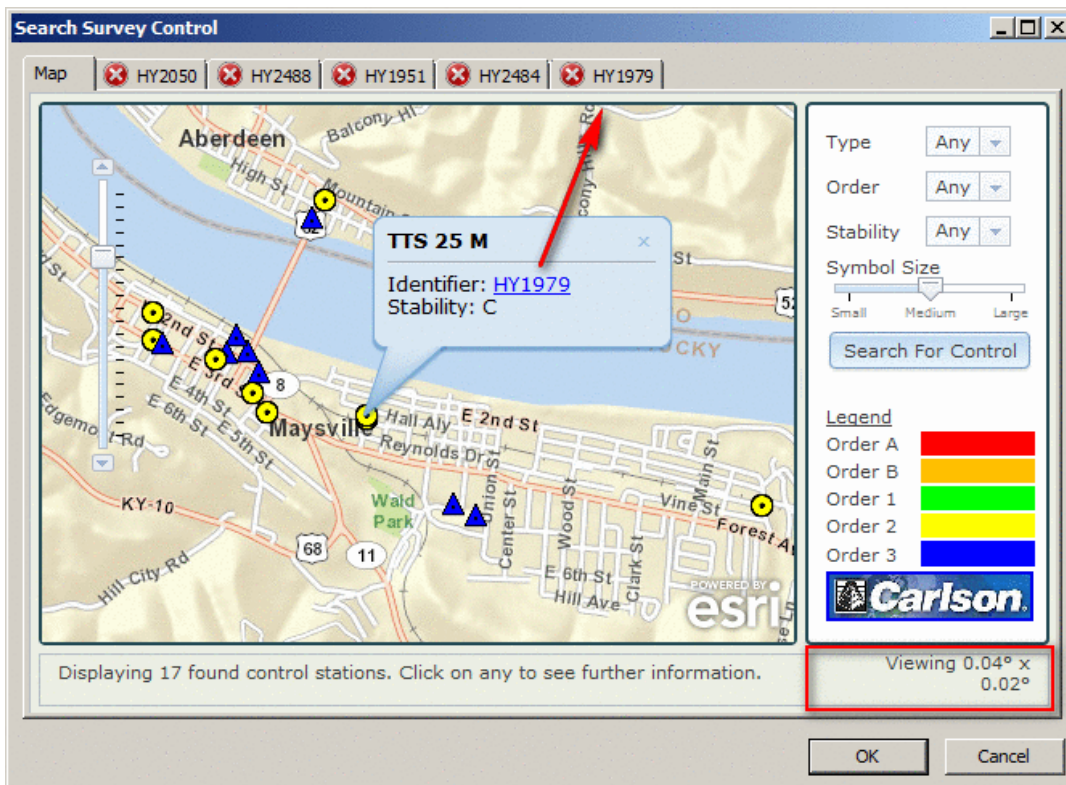
Symbol Size: Use the horizontal slider to adjust the symbol size of the NGS markers.

 Triangle = horizontal control, elevation unknown

 Circle = vertical control (benchmark), hz position usually scaled

 Triangle with circle = 3D position, all dimensions measured.

Search for Control: Click on this button to initiate the search for NGS control markers that satisfy the search criteria.



Note:

- The Search Survey Control dialog box is re-sizeable and contains re-sizeable controls.
- Order - For additional information on Order accuracy, reference <http://www.ngs.noaa.gov/faq.shtml#WhatHARN>: Horizontal A-order stations have a relative accuracy of 5 mm +/- 1:10,000,000 relative to other A-order stations. Horizontal B-order stations have a relative accuracy of 8 mm +/- 1:1,000,000 relative to other A-order and B-order stations. Additional information can also be viewed at <http://gpsinformation.net/main/ngs-accuracy.html>.
- Stability - For additional information on marker Stability, reference http://www.ngs.noaa.gov/AERO/Genspecs_A/Volume%20A_Attachment%201-6.pdf: Stability code A = expected to hold an elevation. Examples: rock outcrops; rock ledges; bedrock; massive structures with deep foundations; large structures with foundations on bedrock; or sleeved deep settings (10 feet or more) with galvanized steel pipe, galvanized steel, stainless steel, or aluminum rods.
Stability code B = probably hold an elevation. Examples: unsleeved deep settings; massive retaining walls; abutments and piers of large bridges or tunnels; unspecified rods or pipe in a sleeve less than 10 feet; or sleeved copper-clad steel rods.
Stability code C = may hold an elevation but subject to ground movement. Examples: Metal rods with base plates less than 10 feet deep; concrete posts (3 feet or more deep); large boulders; retaining walls for culverts or small bridges; footings or foundation walls of small to medium-size structures; or foundations such as landings, platforms, or steps.
- As NGS markers are retrieved, left-click on the marker itself to see summary information about the marker.
- To retrieve the NGS datasheet with additional information about the marker, left-click on the Identifier hyperlink. To close a "balloon" marker on the Map tab or an NGS datasheet, left-click on the "X" icon of the balloon or datasheet tab, respectively.
- If any NGS datasheet tabs are open and the OK button is clicked on the dialog box, you will be prompted if the open NGS markers should be saved to the current coordinate file.

Prompts

Save selected stations to coordinate file? Indicate your preference if the opened NGS control marks should be saved to the current coordinate file.

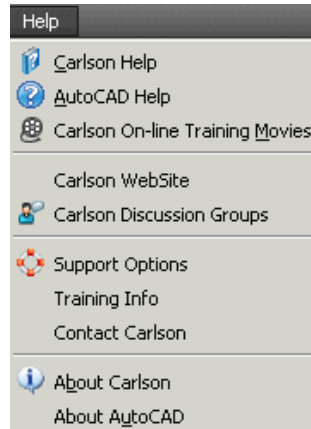
Pulldown Menu Location: Points > Point Utilities

Keyboard Command: searchcontrol

Prerequisite: Internet connection

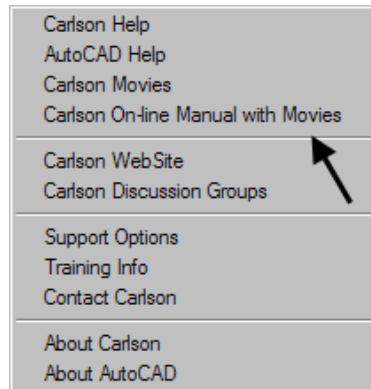
Help Menu

The Help menu has links to the manual and helpful links to on-line resources.



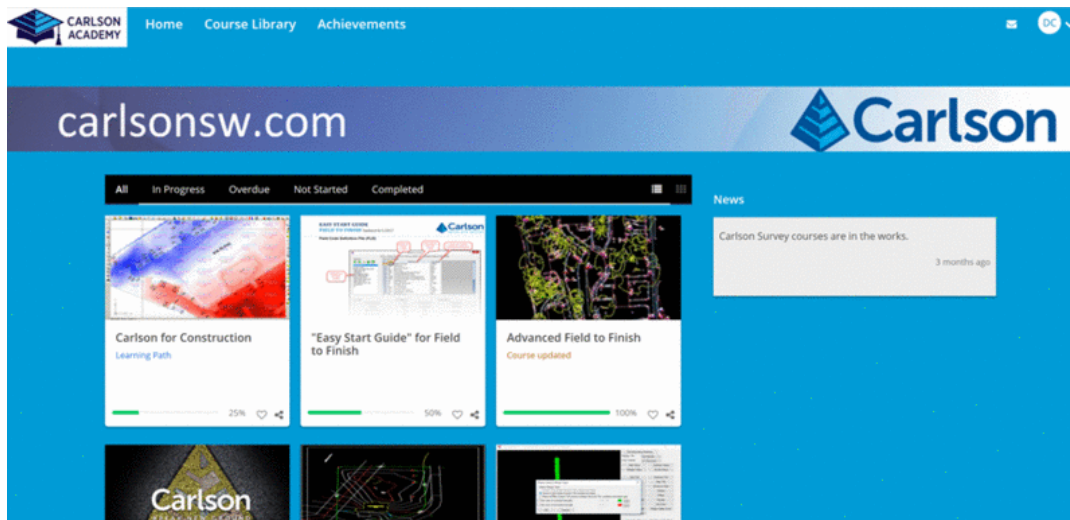
Carlson On-line Manual with Movies

This function launches the Internet browser to view the Carlson manual on-line. This on-line manual also includes training movies. You will need a fast internet connection to use this manual.



Carlson Academy

This command launches your default internet browser to bring up Carlson Academy which is an on-line learning system. There are training movies and example data files that are organized by categories.



Carlson Academy requires a login with username and password. To create an account, go to <https://account.carlsonsw.com/login> and pick Sign Up. Access to Carlson Academy requires a Carlson Software account and your company being current on the maintenance program.

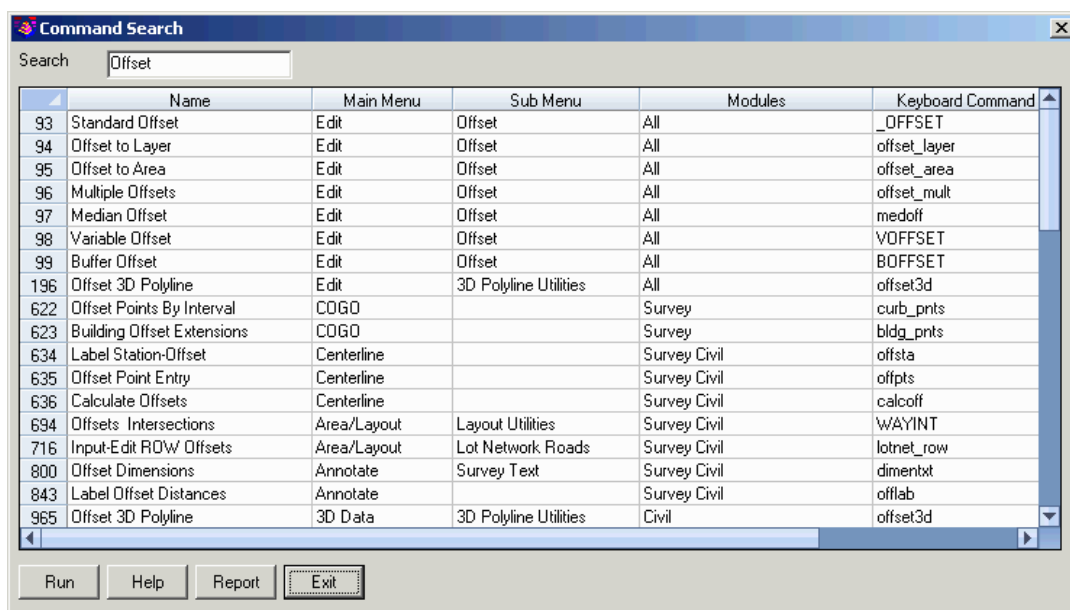
Pulldown Menu Location: Help

Prerequisite: None

Command Search

This command helps find commands in the menu. Enter a command name to search for in the Search edit box. The resulting matches are shown in the spreadsheet. Main Menu and Sub Menu are for the pop-down menu name that contains the command. Modules lists the modules that include the command. Keyboard Command is the name for running the command from the Command: line. Toolbars lists any toolbars that contain the command.

The Run button runs the highlighted command. The Help button brings up the manual help topic for the highlighted command. The Report function uses the Report Formatter to make a report for the current contents of the spreadsheet.



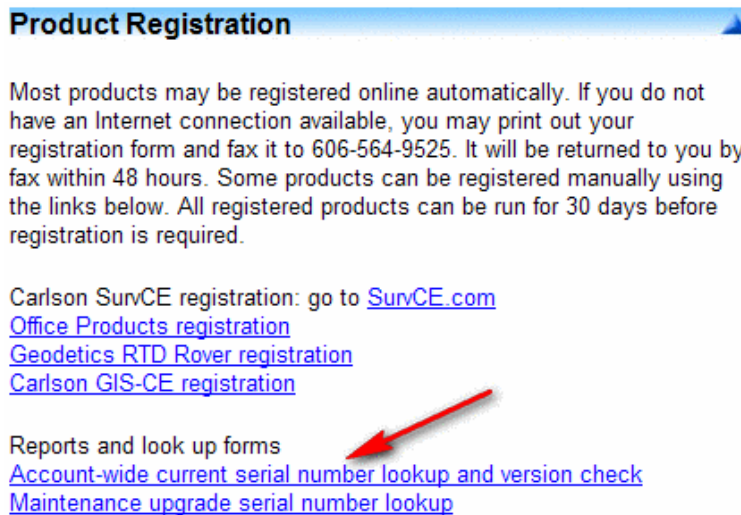
Pulldown Menu Location: Help
Keyboard Command: cmd_search
Prerequisite: None

Serial Number Report

Carlson Software is setup (by default) to check the Carlson server for periodic updated patches (*aka* "service packs"). These checks are logged so that the server collects which build versions a particular machine is running. On the www.carlsonsw.com webpage, under the Support > Product Registration, there is a link for a report of this serial number information:

http://update.carlsonsw.com/current_version_report.php

This tool generates a patch status report for all the computers at your company.



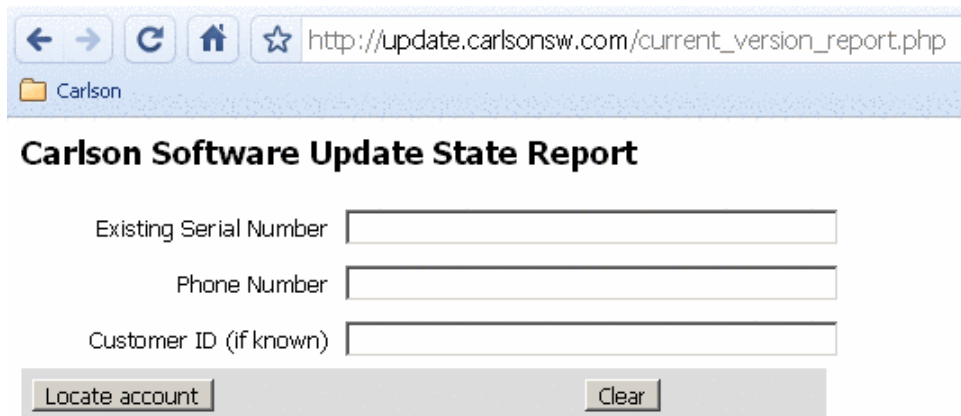
Product Registration

Most products may be registered online automatically. If you do not have an Internet connection available, you may print out your registration form and fax it to 606-564-9525. It will be returned to you by fax within 48 hours. Some products can be registered manually using the links below. All registered products can be run for 30 days before registration is required.

Carlson SurvCE registration: go to [SurvCE.com](#)
[Office Products registration](#)
[Geodetics RTD Rover registration](#)
[Carlson GIS-CE registration](#)

Reports and look up forms
[Account-wide current serial number lookup and version check](#)
[Maintenance upgrade serial number lookup](#)

This link goes to a webpage which prompts for the account to report. A serial number must be supplied along with either a phone number or Carlson customer ID assigned to the serial number.



← → ↻ 🏠 ☆ http://update.carlsonsw.com/current_version_report.php

Carlson

Carlson Software Update State Report

Existing Serial Number

Phone Number

Customer ID (if known)

A report is then generated with all the machines, serial numbers and current build numbers and last update check date for the identified account as shown in the sample below (parts of the serial numbers below have been blurred for security purposes).

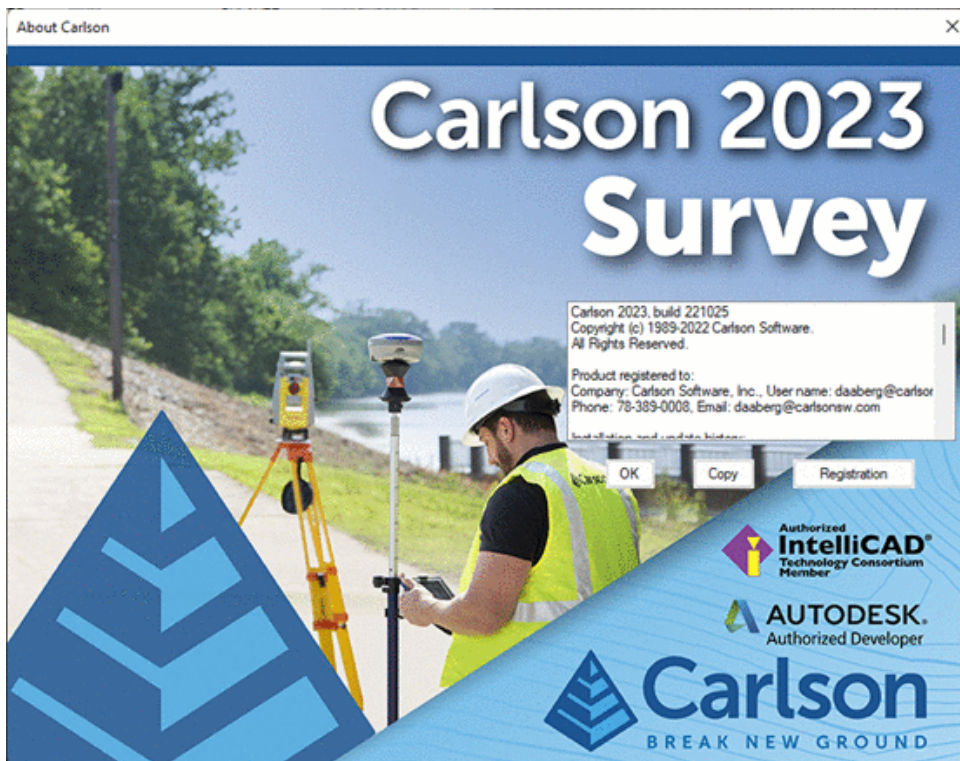
Carlson Software Update State Report

Product Name	Serial Number	Build	Date last seen
Carlson 2010	-4F27AC0-	100713	August 25, 2010, 9:09 am
Carlson 2010	-4F27AC0-	091023	June 2, 2010, 10:40 am
Carlson 2010	-4F27AC0-	100713	August 25, 2010, 7:44 am
Carlson 2010	-4F27AC0-	091023	November 25, 2009, 1:31 pm
Carlson 2010	-4F27AC0-	091023	November 25, 2009, 12:24 pm
Carlson 2010	-4F27AC0-	091023	April 27, 2010, 5:04 pm
Carlson 2010	-4F27AC0-	Unknown	Never
Carlson 2010	-4F27AC0-	091023	December 14, 2009, 8:05 am
Carlson 2010	-4F27AC0-	100318	July 22, 2010, 10:38 am
Carlson 2010	-4F27AC0-	091023	March 17, 2010, 1:52 pm
Carlson 2010	-4F27AC0-	100713	August 11, 2010, 8:59 am
Carlson 2010	-4F27AC0-	100318	July 19, 2010, 2:13 pm
Carlson 2010	-4F27AC0-	091023	December 17, 2009, 9:56 am
Carlson 2010	-4F27AC0-	100318	August 16, 2010, 6:48 am
Carlson 2010	-4F27AC0-	091023	November 18, 2009, 8:01 am
Carlson 2010	-4F27AC0-	100713	August 23, 2010, 9:39 am
Carlson 2010	-4F27AC0-	100713	August 19, 2010, 8:30 am
Carlson 2010	-4F27AC0-	Unknown	Never

- The Carlson *Build Number* is essentially identical to the release date for the build as it is in the form of **YYMMDD**.

About Carlson

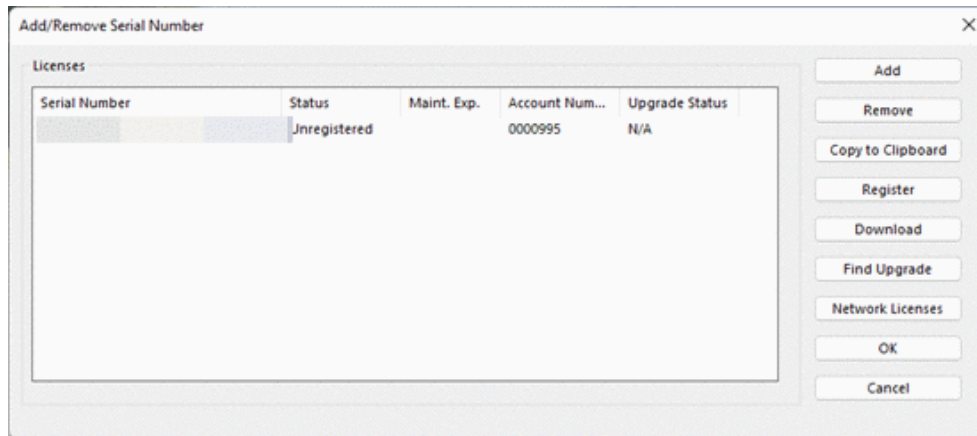
The purpose of this dialog is to inform user about currently installed version of the software, enabled features and to assist with registration and installation issues.



The list box in the dialog shows the copyright notice, when and what version was installed, patches applies and

features currently licensed based on the serial numbers enabled.

Registration - clicking button leads to a dialog containing the list of the serial numbers currently in use in software and their status:



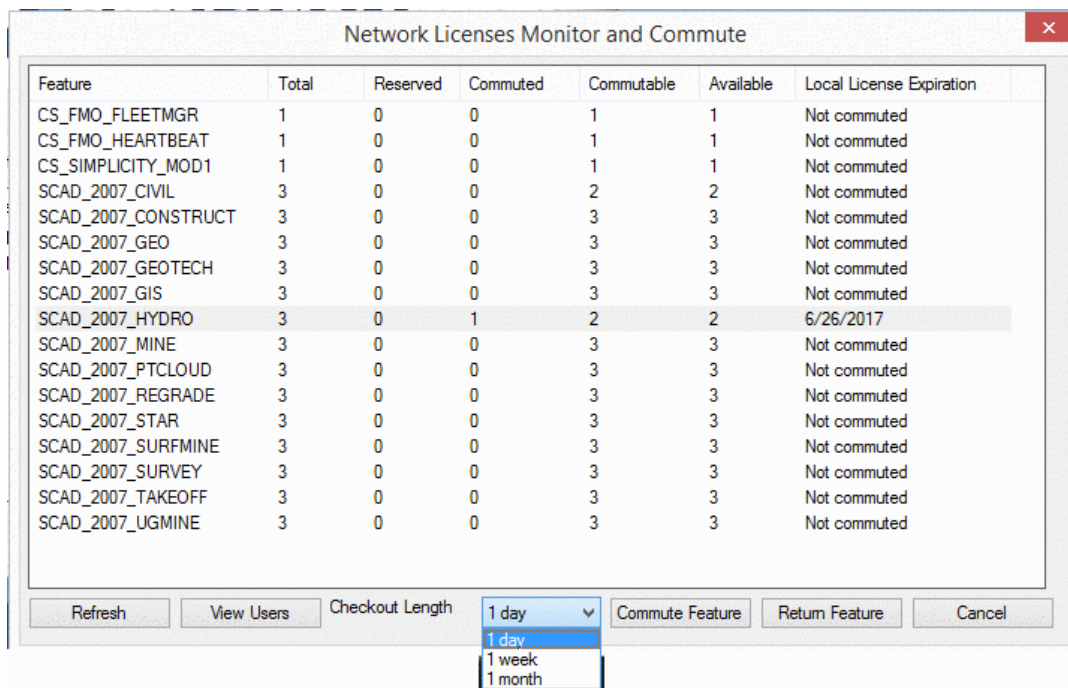
Click **Add** to put in additional serial numbers, or highlight number and click **Remove** to deactivate it. At least one serial number needs to be entered in this dialog for the program to function. Each purchased serial number needs to be registered, which is done at program startup or from this dialog, by highlighting the number and clicking **Register**.

Copy to Clipboard copies the serial number to aid in updated version installation and other applications

Download button takes user to the download page on the Carlson website.

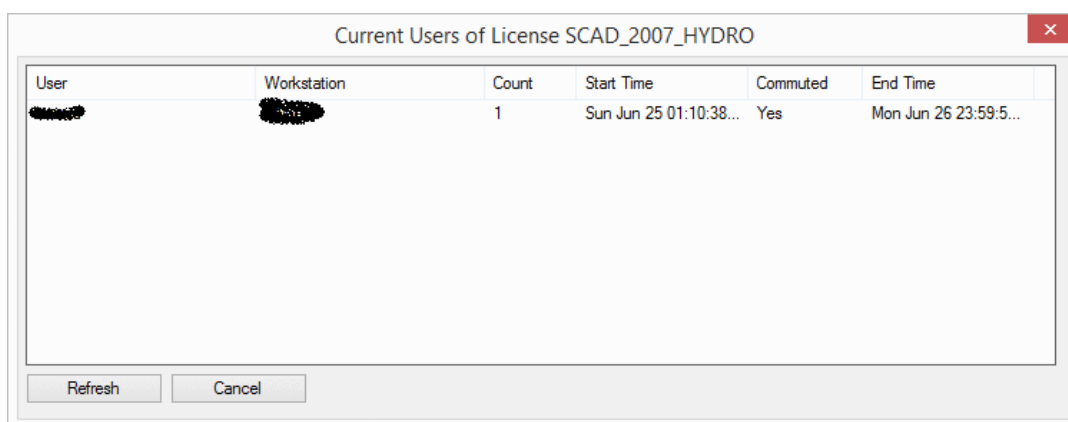
Find Upgrade function opens the page on the Carlson website to lookup the upgrade serial number to the newest version of the product, if the user is on active maintenance agreement.

Network Licenses accesses the integrated Network License Manager tools to display available licenses on the server, show users using the license; commute (borrow) or return committed license.



Refresh updates the list of licenses and license counts.

View Users displays the list of users for the selected feature, along with information on how long the license is commuted for.



user can use this feature to determine the holder of the license he needs and communicate with his coworker about the access to the license.

Commute Feature button requests the commute of the license for the selected feature for the selected length of time. If commute is successful, the display will update to reflect this.

Return Feature button returns the commuted license back to the server. Normally, the commuted license will return to server automatically upon expiration. This function allows to return the license early if it is no longer needed locally.

Extend Demo - this button is only present when software is in trial mode. This provides user with information needed to extend duration of the trial by 14 days or 2000 uses, whichever comes first. Extension is not additive,

so it needs to be performed when software expired or about to expire to give the most benefit.

Revision History - clicking this button causes software to receive the revision notes off the server and to display information about changes contained in individual updates

Prompts

Pulldown Menu Location: Help

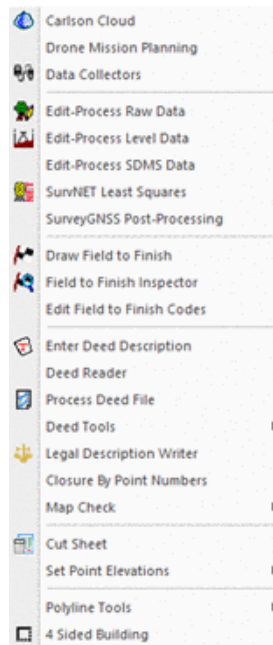
Keyboard Command: `about_scad`

Survey Module

3

Survey Menu

This chapter provides information on using the commands from the Survey pulldown menu, in order to download data from data collectors, process raw data and prepare plats. The first sections of the pulldown provide information on working with data collectors, editing and processing raw data and drawing Field to Finish. Carlson SurvNET is Carlson's Network Least Squares Reduction (NLSA) program. Below that there are complex deed creation and linework commands. The bottom portion of this menu provides features for creating cut sheets, polyline data and other survey important requirements.

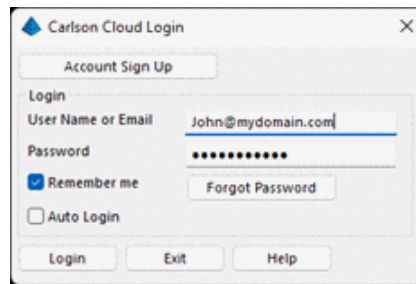


Carlson Cloud

The Carlson Cloud command (not to be confused with the Carlson Point Cloud module) is an Internet/cloud-based routine designed to easily facilitate sending messages, data files and crew locations between the office and Internet-connected field devices running Carlson SurvCE or Carlson SurvPC.



Upon launching Carlson Cloud, the user is challenged to provide login credentials as shown below:



- **Account Sign Up**
- **Account Options**

Username: Supply the user name (Case Sensitive) that you used when creating your Carlson Cloud account.

Password: Supply the password (Case Sensitive) that you used when creating your Carlson Cloud account.

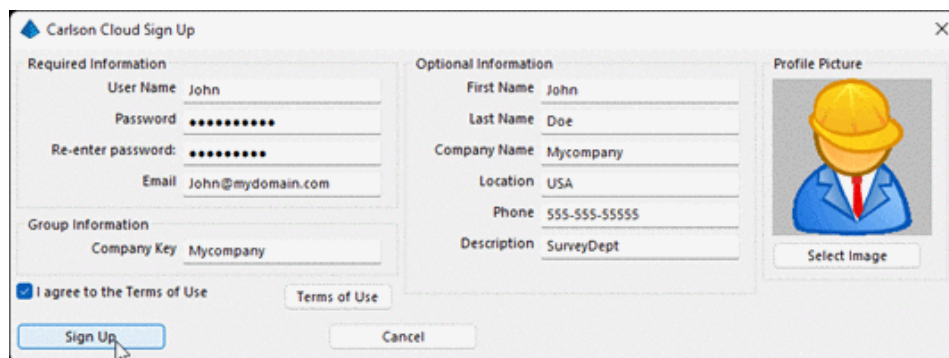
Remember Me: When enabled, the last used *Username* and *Password* values are recalled for the next Carlson Cloud login session.

Forgot Password: Password recovery instructions are sent to the specified Email address.

Auto Login: When enabled, automatically logs in to Carlson Cloud each session.

Once logged in, refer to the Carlson Cloud - Logged In discussion.

The *Account Sign Up* command provides the ability to create a new user account and associate the user account with a "company key" (essentially, a group of people with whom you wish to network within Carlson Cloud).



Username: Supply the user name (this will be Case Sensitive) that you will want to use for your Carlson Cloud account.

Password: Supply the password (this will Case Sensitive) that you will want to use for your Carlson Cloud account.

Phone: Supply your phone number.

Email: Supply an Email address where password recovery instructions can be sent.

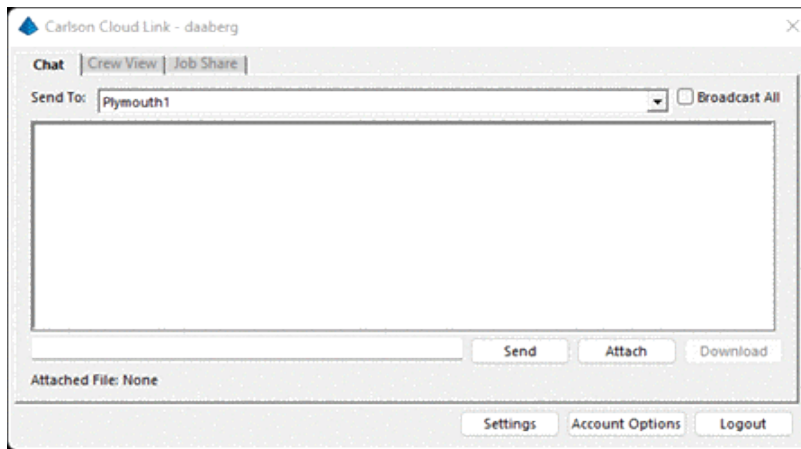
Create Company Key: When enabled, this toggle permits you to create a new company "profile" (or communication group).

Existing Company Key: Indicate the name (Case Sensitive) of an existing company "profile" to which the new member should belong.

Optional Information: Enter in the optional information of name, company, location, phone number and a description.

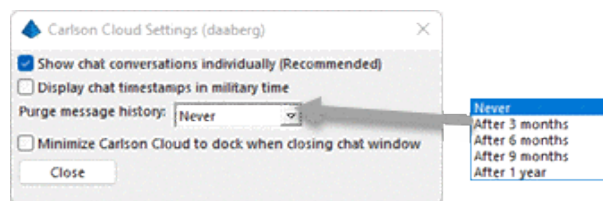
Select **Sign Up** when all information has been entered.

Upon successful login to Carlson Cloud, the user is presented with the communication aspects of Carlson Cloud.



This dialog box contains the Chat Window, Crew View and Job Share Tabs.

Settings:



This dialog box shows options for the Chat View Tab as well as other options.

Show Chat Conversation Individually shows a single conversation at a time in the view window.

Display chat timestamps in military time sets the time view to a 24 hour clock.

Purge message history controls how long to keep the chat history for each user

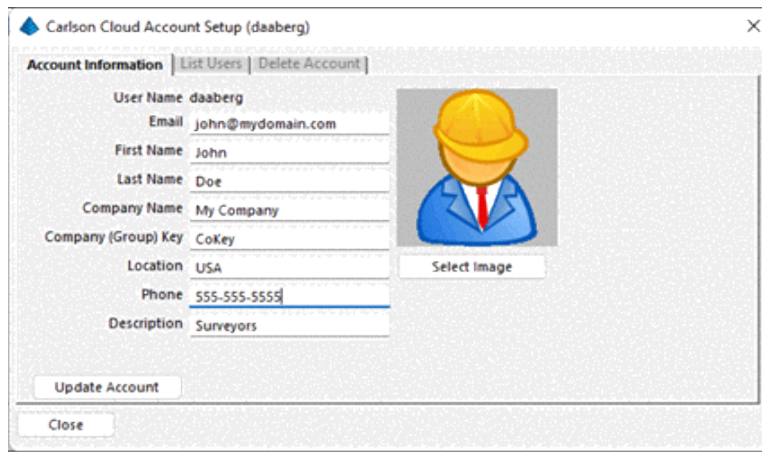
Minimize Carlson Cloud to dock when closing chat window keeps the chat window active but minimizes it to a small icon. This option allows you to keep a chat conversation active while working in the drawing editor without the distraction of the complete dialog box.

Account Options

The *Account Options* of Carlson Cloud permits you to investigate and manage certain aspects of your Carlson Cloud account(s).

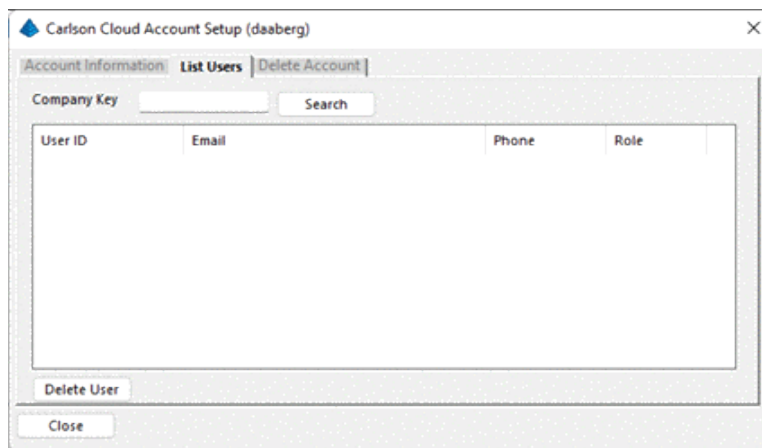
Account Information:

Allows you to review, edit and update your account information.



List Users Tab

Carlson Cloud can display the listing of users assigned to a specified *Company Key*.



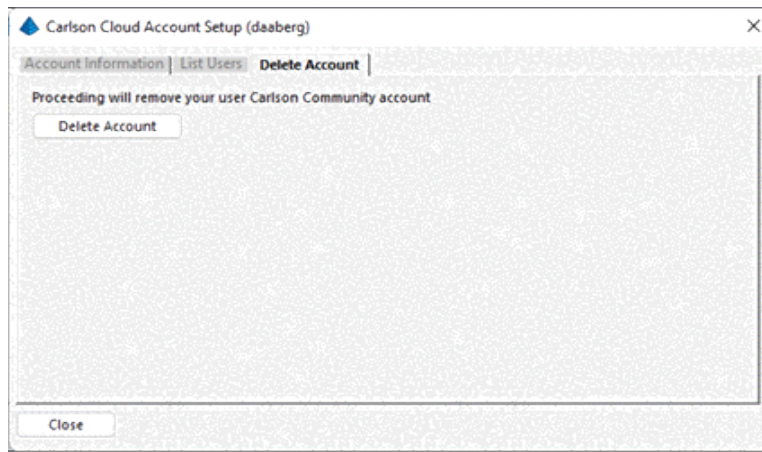
Company Key: Supply the name of the unique Company Key established for your communication group.

Search: The list of users associated with the Company Key is returned.

Delete User: Highlight an existing user and press Delete User to remove a user from the group.

Delete Account

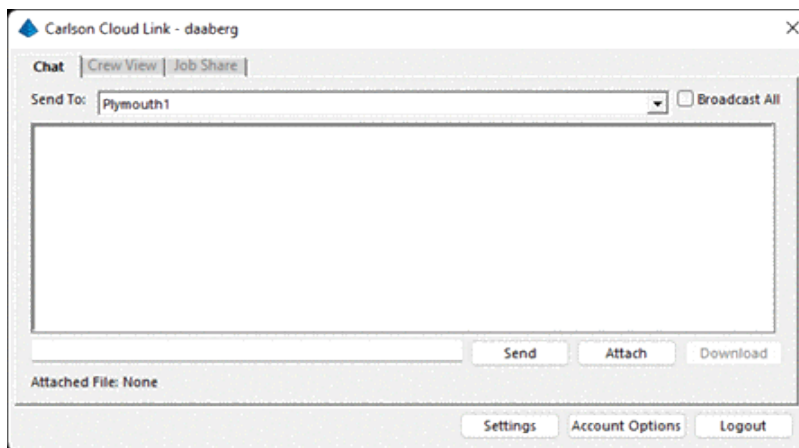
Carlson Cloud accounts that are no longer needed can be removed from the Carlson Cloud service.



Logs the active account out of the Carlson Cloud service.

Chat Tab

While logged in to Carlson Cloud, it is possible to communicate with other Carlson Cloud accounts that belong to the same "Company Key" as that of the active logged in user.



Username Drop-list: Select the individual with whom you would like to chat.

Broadcast All: When enabled, all members listed in the Username Drop-list will receive your chat (and optional attachment) at their next login.

Message Area: The listing of communication chats (both current and received since the last successful login) will appear.

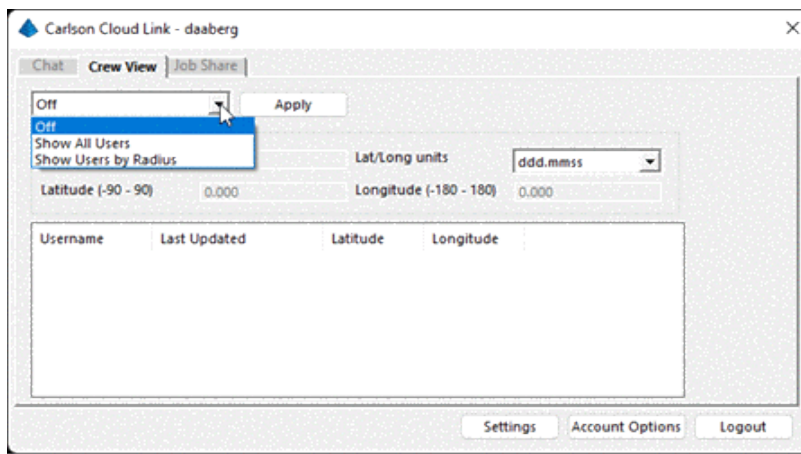
Chat Line: Supply a brief chat question or statement for the intended person or audience.

Send: Delivers the Chat Line content (and optional attachment) to the intended person or audience.

Attach: Specify a Carlson SurvCE, SurvPC or CSI Mobile compatible file (*e.g.* CRD, RW5, NOT, LOC, *etc*) that should be delivered to the intended person or audience.

Crew View Tab

Carlson Cloud permits the viewing of other crew members that can be helpful for site logistics and planning.



Crew to Show Drop-list: Indicate the crew members (if any) that should be shown:

- **Off** - No crew members will be shown.
- **Show All Users** - All crew members will be shown.
- **Show Users by Radius** - Crew members within a prescribed radius of a central Latitude/Longitude position will be shown.

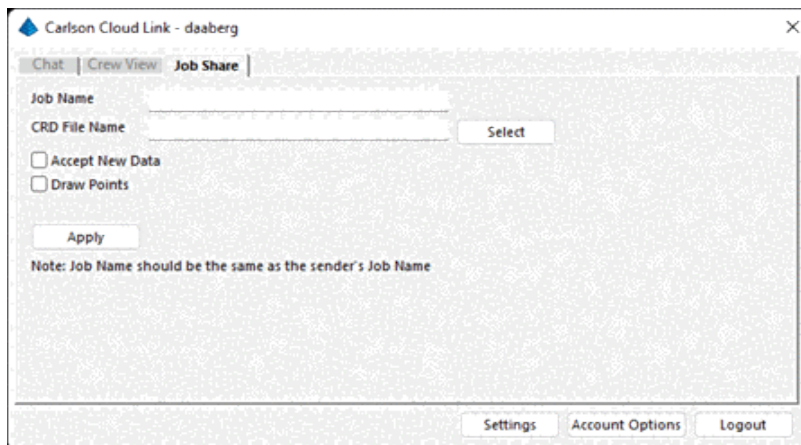
Latitude: Indicate the central latitude.

Longitude: Indicate the central longitude.

Radius: Indicate the radius value in miles from the central Latitude/Longitude position to display crew members.

Job Share Tab

Carlson Cloud permits the sharing of job data (*e.g.* field readings) with other Carlson Cloud members associated with the Company Key.



Job Name: Specify the name as it appears (Case Sensitive) from the sending device (such as Carlson SurvCE).

CRD File Name: Indicate the name of the Carlson Coordinate File that is to receive the job information. **Note:** This CRD does not need to be the same name as that used on the sending device.

Accept New Data: When enabled, incoming data will be permitted into the specified **CRD File Name**.

Draw Points: When enabled, incoming data will be placed into the drawing via the Draw-Locate Points command.

Note:

- When creating a "Company Key," consider a unique combination of your organization name with a portion of your phone number or postal code to help establish "uniqueness."
- When resetting a password, the "From" Email address will be from **customerportal@carlsonsw.com**. Please make sure you add this address to your "Safe Senders" list.
- The positions of crew members will only appear if the crew has enabled their "Allow others to view my position" toggle.
- The positions of crew members update about once every 5 minutes.
- While logged out of Carlson Cloud, any "chats" or data files sent to you are held in-transit until you next log into Carlson Cloud.

Pulldown Menu Location: Survey

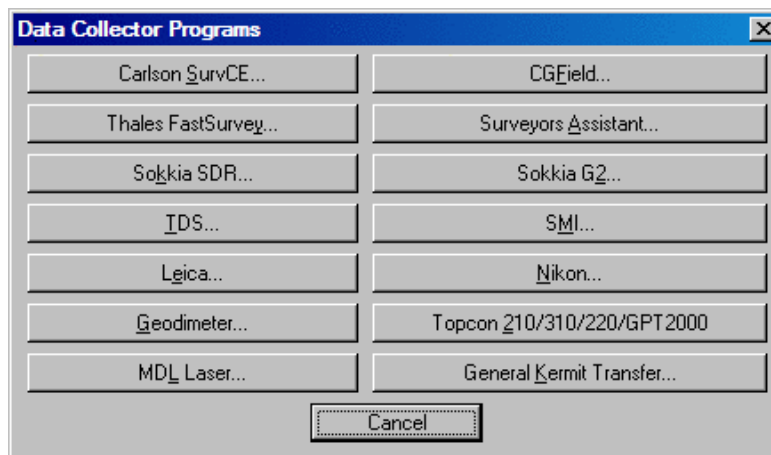
Keyboard Command: ccloud

Prerequisite: Internet connection

Data Collectors

This command does two main functions for a variety of popular data collectors. First, this command transfers (uploads and downloads) data between the data collector and Carlson. Second, this command converts data formats between the data collector format and the Carlson format. So, if you already have the data file on the computer, you can skip the transfer function and just perform the conversion function.

The transfer function does the conversion at the same time. In most cases, the download from the data collector produces a raw (.RW5) file (field notes) and/or a coordinate (.CRD) file (coordinate points). Several of the download programs have an option to automatically run the *Edit-Process Raw Data File* command after downloading raw data. You can also send, or upload, a coordinate (.CRD) file. The dialog shown here appears when the menu command is selected.



Carlson SurvCE: For Carlson Software data collection programs SurvCE and SurvStar. This button produces the SurvCOM dialog and program.

Prepare Geoid for SurvCE: Creates a subset of a geoid as a .GSF file to load into SurvCE.

CG Field: For CG Field programs.

Magellan/Thales FastSurvey: For Magellan and Thales FastSurvey data collectors.

Surveyors Assistant: For data collectors running Surveyors Assistant software (Corvallis MC2, MC5 and Pentax SC5).

Sokkia SDR: For SDR2 through SDR 33 and other collectors that have a SDR format like the Trimble.

Sokkia G2: Specifically for the SDR2.

TDS: For data collectors that use TDS software (Ranger, HP48, HP95, Husky FS-2 & FS-3, Corvallis MC-V and TOPCON FS2, FC95 and FC48).

SMI: For SMI data collectors on the HP48.

Leica: For Leica GIF-10 module, GeoCOM for older Leica and DBX for newer Leica instruments.

Nikon: For Nikon DTM and DR-48 total stations.

Geodimeter: For the Geodimeter Geodat collector.

Topcon 210/310/220/GPT2000: Supports these Topcon models.

MDL Laser: For MDL Laser instruments.

General Kermit Transfer: For general transferring using Kermit.

Carlson SurvCE

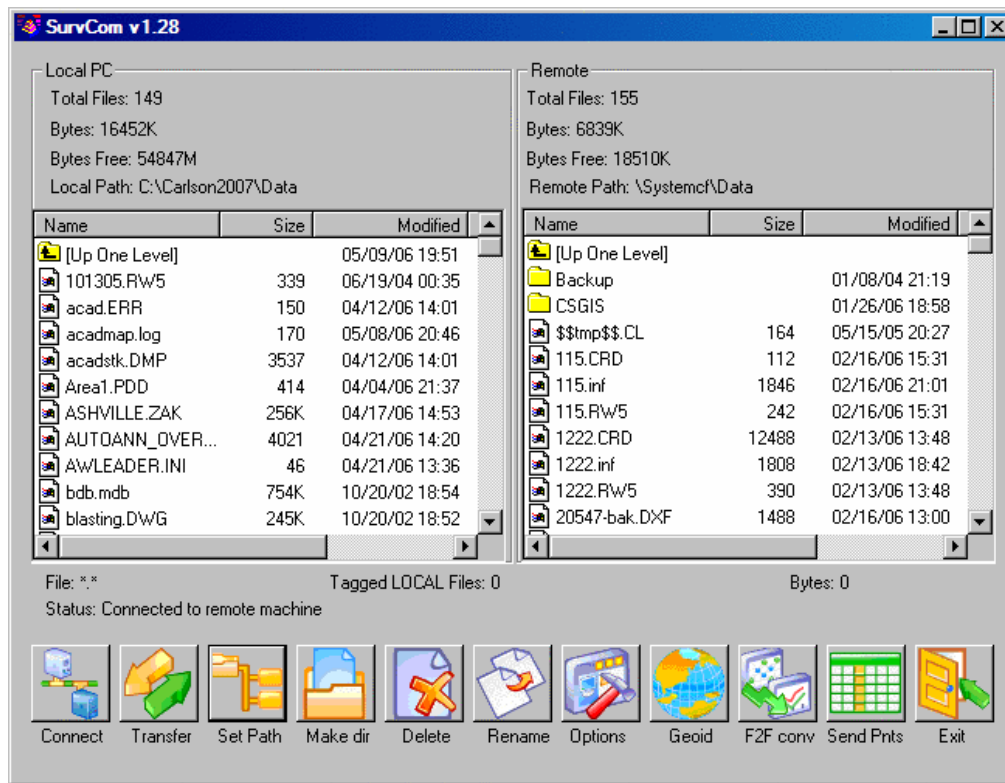
Note: In the following text, the term SurvCE will apply to SurvCE, SurvStar, and Sokkia G2

Connect the serial cable. Select Data Transfer from the on the handheld. Choose Carlson/Carlson Survey Download. This leads to a File Transfer screen on SurvCE, which says "Awaiting Connection". All the action is on the PC side. There is no time delay in this handshake. It will wait for the PC program to catch up. When you connect the cable from SurvCE to the PC, Microsoft ActiveSync may interfere and say "Connect to PC?" If you get this question, say No or otherwise terminate the Microsoft ActiveSync linkage. Start the Carlson portion of this link by choosing Survey, Data Collectors, then the SurvCE option. If connection is automatically established, SurvCE will display, "Connected to PC".

If only the left side of the screen displays data, then you do not yet have a connection. Press the Connect button located at the bottom left of the file transfer dialog. The transfer program will respond with Retrieving File List. Once the file list has been retrieved, the left side of the dialog box will show files located in the specified path on the PC and the right side of the dialog displays the files located in the designated path on the remote. You can change directories by scrolling to the top of the file list and choosing Up One Level (just like in Windows).

To transfer one or more files, simply select or highlight the desired files and select the transfer button. More than one file can be transferred from the remote to the PC or from the PC to the remote during the transfer process. Standard Windows selection options apply. For example, selecting one file and then while pressing the shift key on the PC, selecting another file deeper on the list will select all the files in between the first and last selected. You can also select the first file to transfer and press and hold down the shift key and use the down arrow to specify the range of files to transfer. Pressing and holding the control key on the keyboard allows for the selection of multiple files in any selection order, by picking the files with the left mouse button.

After the files have been selected, press the transfer button. When the transfer is complete, the program will return a "Transfer Complete" message, and will then proceed to update the file lists on the PC and the Remote.

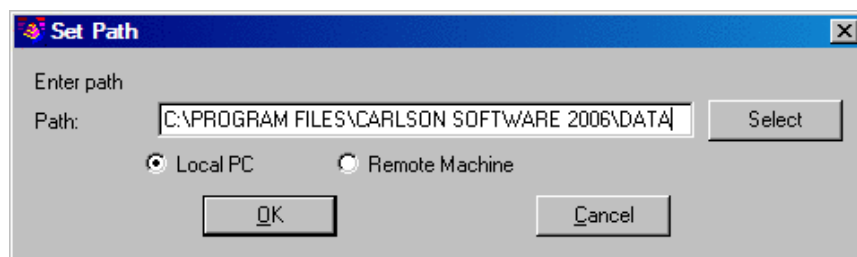


The following information describes the buttons on the bottom row of the SurvCOM dialog box. The button name is on the left in bold:

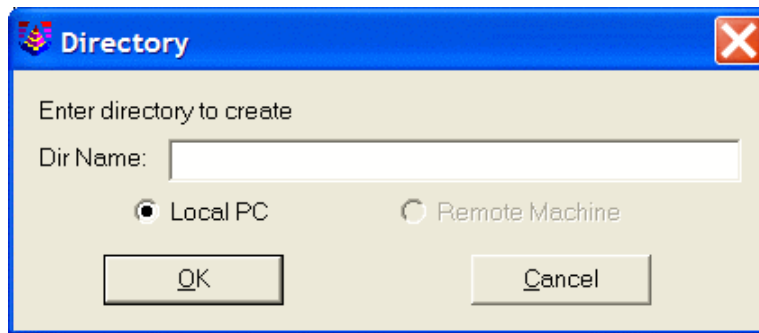
Connect: After selecting Data Transfer in SurvCE, press this button to start the connection. Once connection is made, the status line on the file transfer utility dialog box will show Connected to the remote machine.

Transfer: Pressing this button transfers selected files from either the Remote to the PC, or the PC to the Remote.

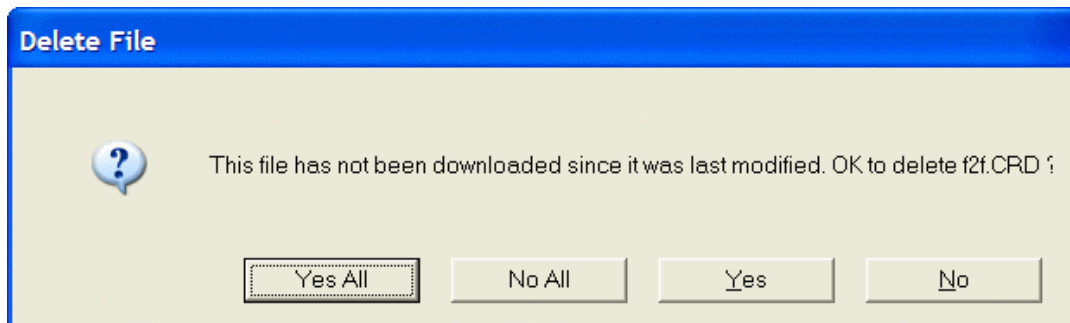
Set Path: This option allows for the specification of the desired source and destination drives and folders for both the PC and the Remote device. For example, if you were downloading, or copying files from the Remote device to the PC, to specify a source path on the remote device, select the Remote Machine toggle and then type in the desired path in the path field. To specify a destination path on the PC, select the Local PC toggle and type in the desired path the path field. When a change to either path is made, the transfer utility will retrieve a new file list from the specified paths.



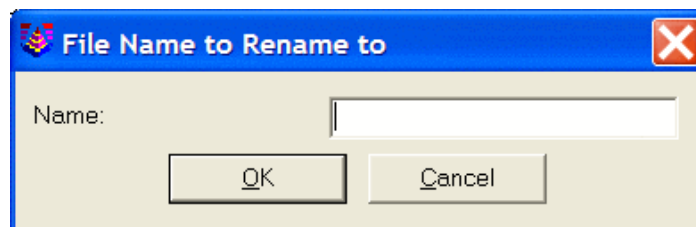
Make dir: This option allows for creation of directories on both the PC and the Remote device. Specify the hardware on which to create the directory and then enter the directory name.



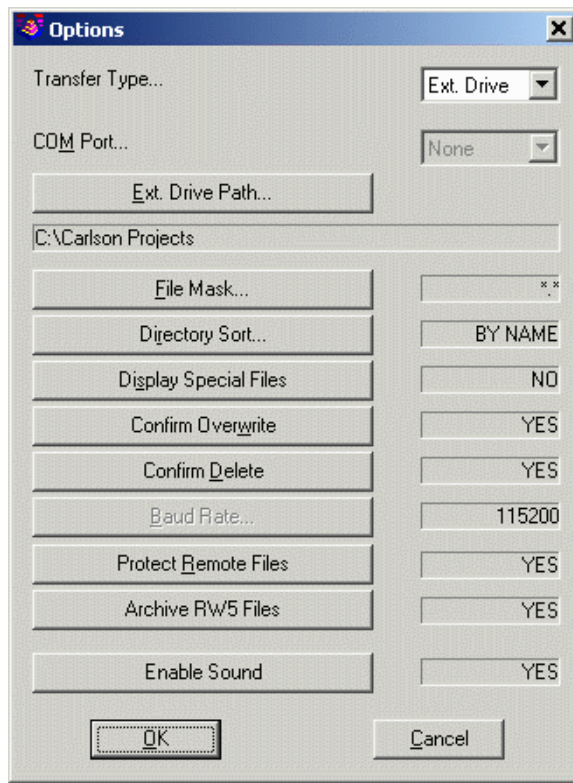
Delete: This option allows you to delete the tagged files. To delete a file, select the file to delete by clicking on the file, press the delete button at the bottom of the dialog. Confirm deletion by selecting the appropriate response on the Delete File dialog.



Rename: To rename a file, click on the file to rename and select the rename button at the bottom of the dialog. On the dialog that displays type in the new name and press the OK button.



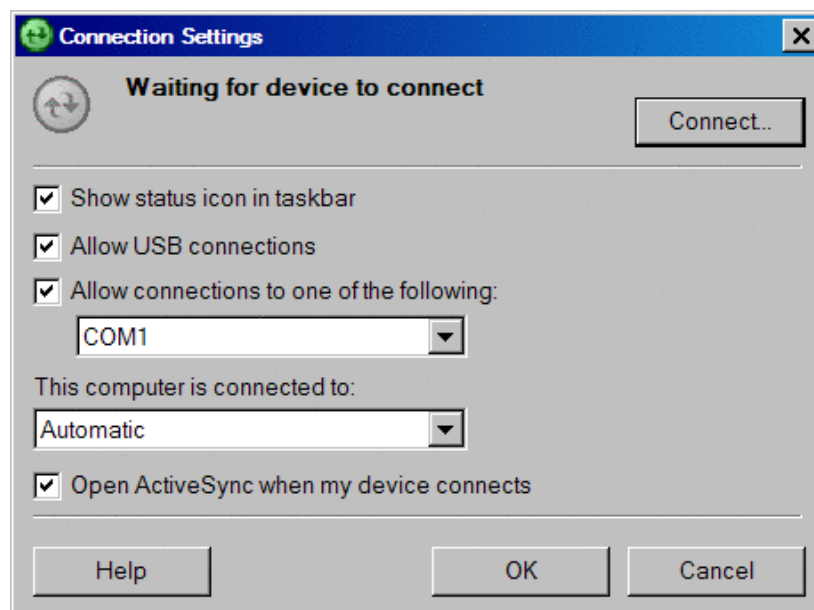
Options: This command allows you to set various options for data transfer. The dialog shown below will appear.



Transfer Type: Choose USB for transferring over a USB cable. Choose 9-Pin for transferring over a 9-pin serial cable. Choose Ext Drive for transferring to another folder on your computer or a drive connected to your computer such as a USB storage drive or memory card.

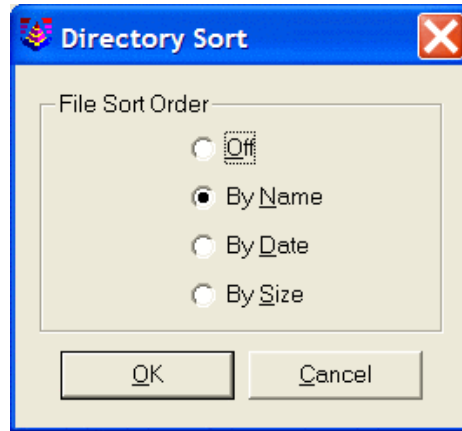
Com Port: You must select which com port on the PC to use when using the 9-Pin transfer type.

If you are transferring data via a USB port, set the com port to ActiveSync, see the Options section below for procedures to change com ports. To transfer data using an USB port a connection between the Remote and PC using ActiveSync is required. In ActiveSync verify that the "Connect Settings" have been set to "Allow serial cable or infrared connection to this Com port" and Allow USB connection with this desktop computer. This will allow for connection using an USB port or a COM port connection. Both will use ActiveSync to transfer data between devices.



File Mask: You must select a file filtering syntax. This filter allows for the setting of specific file types to display. For example if you only wanted to see CRD files the filter would be *.CRD.

Directory Sort: You must select how to sort the list of files.



Display Special Files: Toggle whether or not you should see special files.

Confirm Overwrite: Check this to confirm before overwriting files.

Baud Rate: You must choose the baud rate for transferring data.

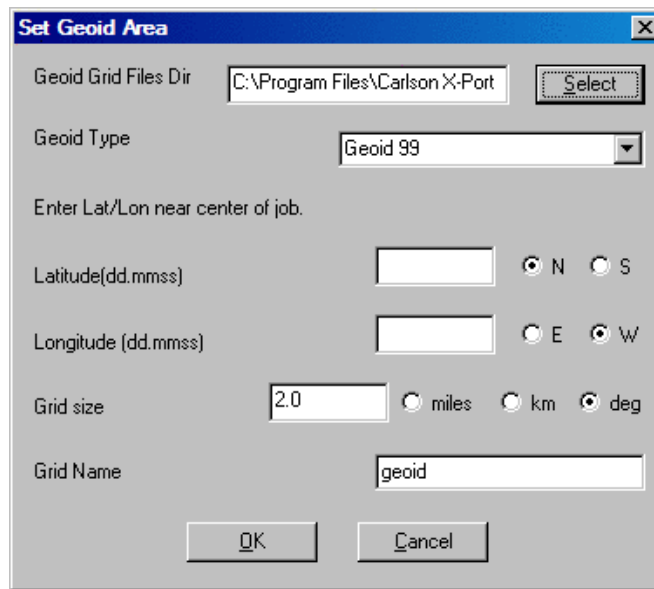
Protect Remote Files: Check this to protect files on the mobile device.

Archive RW5 Files: With this option set to YES, when downloading rw5 files, a second copy of the file will be made with a .SC5 extension to serve as an archive of the original rw5 file.

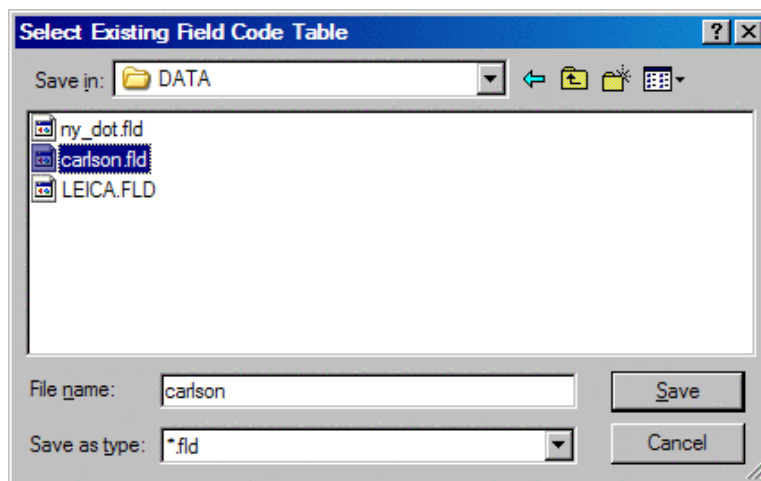
Geoid: This command will carve out a portion of the Geoid 99, EGM96, Canadian CGC2000, Canadian HT2.0, Canadian HT 1.01, Australian GDA94, Great Britain OSG-MO2 and Geoid 2003 grid files, and send it to SurvCE. Since these geoid grids are very large, this carves out a precise portion of it and avoids overloading the memory on the remote device running SurvCE. You will be prompted for the directory on the PC of the source Geoid grid file, the approximate latitude and longitude of the job, and the size of the area desired in miles, kilometers or degrees of latitude and longitude. To define a Geoid area, make sure that this criteria is met:

1. Specify the location of the geoid grid files.
2. Specify the geoid type.
3. Enter the latitude and longitude near the center of the job area.
4. Specify the Grid size either in miles, km (kilometers), or deg (degrees).
5. Name the grid file.

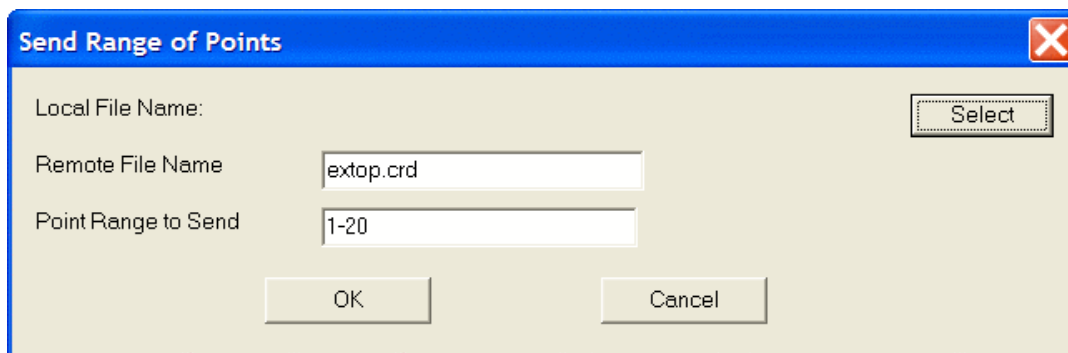
The file will be transferred to the data collector and place in the appropriate place for use.



F2F conv: This converts the more thorough and detailed Carlson field code file (for field-to-finish work, *.FLD) to the more simplified Feature Code List that runs in SurvCE (*.FCL). The Feature Code List in SurvCE (not SurvStar or Field) handles Linework (on or off), Line Type (2D or 3D), Layer (= Code) and Full Text (Description). Select the Carlson field code (*.FLD) to convert, the conversion takes place and the file is transferred and located in the correct location for use in the data collector.



Send Pnts: This option allows for the uploading of a user specified point number range out of the selected crd file to unload. Use the Select button to specify the crd file to upload. The Remote File Name will default to the name of the crd file selected to upload. You can change this name if needed. Specify the Point Range to Send and select the OK button.

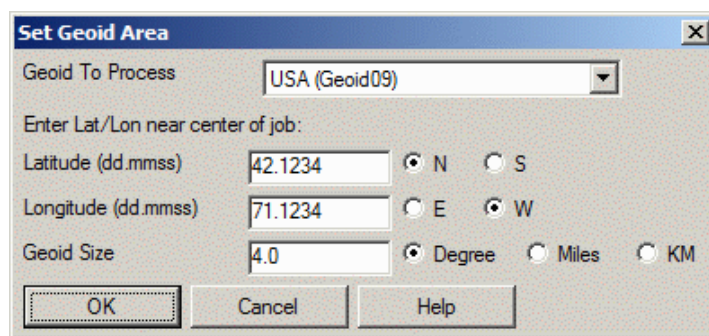


Exit: This command will exit the File Transfer Utility

The following information describes the buttons on the Data Collection Programs dialog box that come after the Carlson SurvCE button, moving from left to right and then from top to bottom. The command/button name is on the far left margin, in bold:

Prepare Geoid for SurvCE

This function creates a .GSF (Geoid Separation File) for SurvCE from a built-in geoid. Most geoids are very large and this routine carves out a subset of the geoid by specifying a center position and area size. The geoid data files are not included in the regular install since they are so large. Instead, the program automatically downloads them as needed from the Carlson server. You can also install them separately by running the CarlsonGeoidGrids.exe from the Support->Other Downloads on www.carlsonsw.com.



CG Field

To transfer data to and from data collectors using CGField software, first make sure that the Baud Rate is set to 9600 and the Parity is set to NONE then follow the steps outlined below.

Receiving a Coordinate File from CGField

CGField:

- 1) Go to the UTILS menu and select Option 1, C&G Transfer.
- 2) Select Option 4, "Send Coords"
- 3) Select the Coordinate file to send.

Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

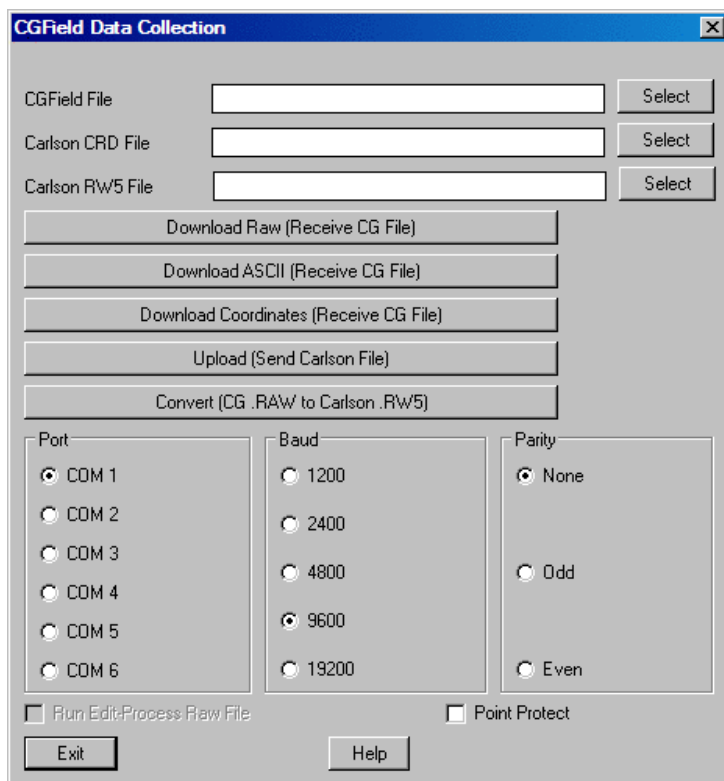
Press the "Download Coordinates" button to ready Carlson to receive the file.
Stop here in Carlson and go back to CGField to complete the transfer process.

CGField:

Select the points to send

- 1) For All points
- 2) To select Blocks of points.
- 3) From .PTS file (the set of points in a Batch Point File).

The coordinates will be transferred. After the transfer is complete, you will be asked for the CRD file name. The C&G CRD file will automatically be converted to a Carlson CRD file. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.



Receiving a Raw Data File from CGField

CGField:

- 1) Go to the UTILS menu and select Option 1, C&G Transfer.
- 2) Select Option 2, "Send Raw Data". Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

Press the "Download Raw" button to ready Carlson to receive the file. Stop here in Carlson and go back to CGField.

CGField:

Select the raw data file to be sent. The transfer will begin.

The C&G .RAW file will be transferred and saved in the data folder. After the transfer is complete, you will be asked for the RW5 file name. The RAW file will be automatically converted to a Carlson RW5 file.

Receiving an ASCII file from CGField

This will allow you to transfer a C&G report file (RPT) or an ASCII NEZ file to Carlson.

CGField:

- 1) Go to the UTILS menu and select Option 1, C&G Transfer.
- 2) Select Option 6, "Send ASCII". Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

Press the "Download ASCII" button to ready Carlson to receive the file. Stop here in Carlson and go back to CGField.

CGField:

Select the ASCII file to send.

After the transfer is complete, you will see the file in the Carlson editor. You can then select FILE and SAVE (or SAVEAS) to save the ASCII file.

Sending a Coordinate File to CGField

CGField:

- 1) Go to the UTILS menu and select Option 1, C&G Transfer.
- 2) Select Option 3, "Receive Coords" to ready the data collector. Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

- 1) Press the "Upload (Send Carlson File)" button.
- 2) Select the Coordinate file.
- 3) Select the points to send.
- 4) Press the "Start Transfer" button.

CGField:

Carlson will send the file name to CGField and a coordinate file with the same name will be automatically created or opened in CGField.

If the file exists you will be asked how you want to handle duplicate points:

- 1) Overwrite
- 2) Don't Overwrite
- 3) Ask for each Point

The point transfer will begin.

Convert CG .RAW to Carlson .RW5

This utility allows you to convert a C&G raw data file to a Carlson raw data file. Select the C&G .RAW file to convert. Then enter the file name of the destination Carlson RW5 file.

Thales/FastSurvey You will be taken directly to the SurvCOM dialog, similar to the Carlson SurvCE process.

Surveyor's Assistant

Download

From the Surveyor's Assistant data collector, go to the Transfer routine from the main menu. Fill out the transfer screen as follows:

Direction: OUTPUT

Format: LIETZ

Data: Coordinate or All Data

Port: COM1 or COM2 Ckh Hold: NO

Protocol: NONE

You should also check the settings under the PORT menu. Typical port settings are baud=9600, parity=none, data=8, stop=1 and handshake=XON/XOFF. Now in Carlson, run *Data Collection* in the Survey menu and choose Surveyor's Assistant. Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to Surveyor's Assistant and press GO. The file transfer should now go. If the All Data option is used, then the Leitz format will contain both coordinate and raw data. The coordinate data is converted to a Carlson coordinate (.CRD) file and the raw data is converted to a Carlson raw data (.RW5) file. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Upload

Point data from the Carlson coordinate (.CRD) file can be uploaded into the Surveyor's Assistant. First go to the Transfer routine on the main menu. Fill out the screen as follows:

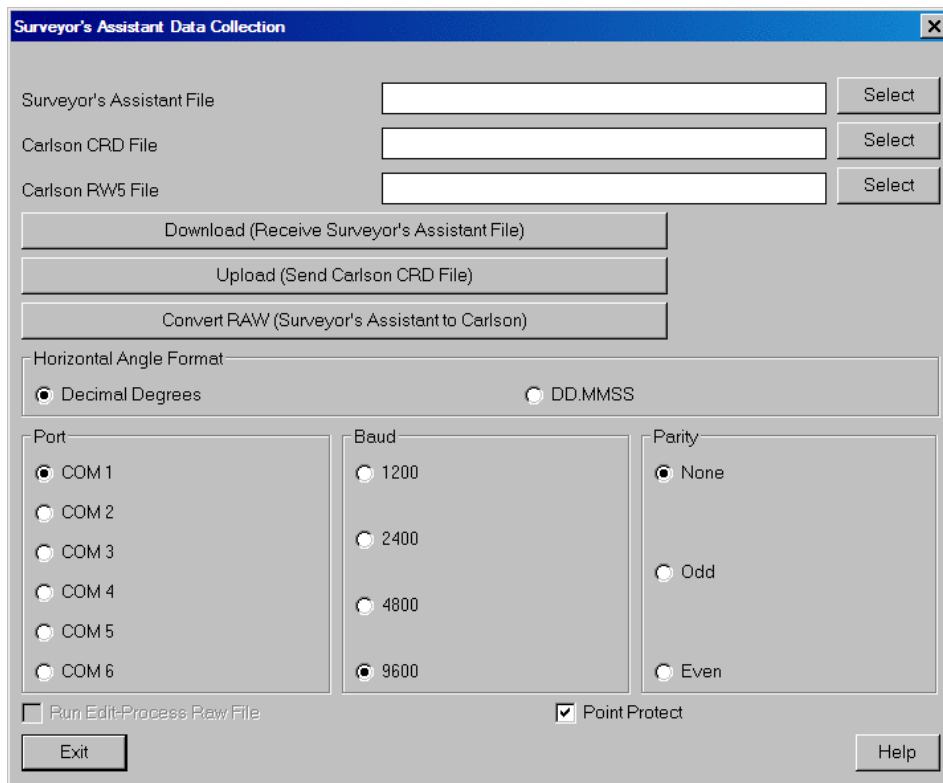
Direction: INPUT

Format: LEITZ

Port: COM1 or COM2

Protocol: NONE

Go back to Carlson and choose Surveyor's Assistant from the *Data Collection* command in the Survey menu. Check that the COM port and baud rate are set correctly. In the Carlson dialog, pick the Select File button next to the Carlson coordinate (.CRD) File edit box and choose the coordinate (.CRD) file to send. Then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Before clicking the OK button for range of points, go to the Surveyor's Assistant and hit the GO function key. The Surveyor's Assistant is now waiting to receive so return to Carlson and click OK on the range of point dialog. The file transfer should now go.



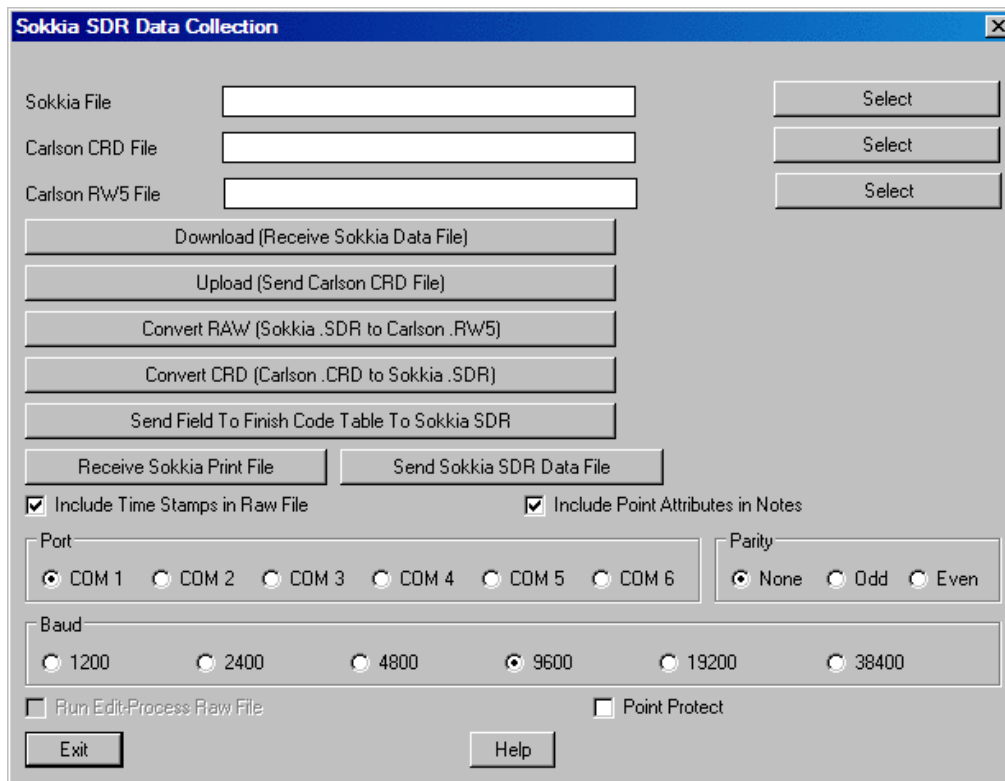
Sokkia SDR

This routine applies to the Sokkia SDR-20, SDR-22, SDR-31 and SDR-33 as well as other collectors that have SDR format transfer such as the Trimble and C & G.

Download

From the SDR data collector, go to the Communications routine from the main menu. Choose Data Format SDR. Next hit the Send function key. Then choose Select Jobs. From the list of jobs, highlight the job to transfer and set it to Yes with the arrow keys. Now in Carlson, run *Data Collection* in the Survey menu and choose Sokkia/SDR. Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to SDR and press OK. The file transfer should now go. The SDR format contains both coordinate and raw data. The coordinate data is converted to a Carlson coordinate (.CRD) file and the raw data is converted to a Carlson raw data (.RW5) file. The original SDR transfer file is stored on the computer as a RAW file. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

The SDR-33 has different modes for storing and transferring data. In coordinate mode, the download will create points in the coordinate (.CRD) file and the raw data (.RW5) file will only contain some basic header lines. In the raw data mode, the download will create all the measurement data in the raw file and no points will be created in the coordinate (.CRD) file. For this raw data mode, you will need to run *Edit-Process Raw Data File* in the Survey menu to calculate the points from the raw data. The third mode in the SDR-33 creates both raw data in the raw data (.RW5) file and points in the coordinate (.CRD) file. The Include Time Stamps in Notes option sets whether all the date-time records for each point are put in the raw data (.RW5) file as description records. The Include Point Attributes in Notes option will store SDR code 13(AT) codes to the point note (.NOT) for the coordinate (.CRD) file.



Upload

Point data from the Carlson coordinate (.CRD) file can be uploaded into the SDR. First go to the Communications routine on the SDR main menu. Choose Data Format SDR. Go back to Carlson and choose Sokkia/SDR from the *Data Collection* command in the Survey menu. Check that the COM port and baud rate are set correctly. In the Carlson dialog, pick the Select File button next to the Carlson CRD File edit box and choose the coordinate (.CRD) file to send. Then click the Upload button. Then a Sokkia Options dialog appears for setting the job parameters for the file to be created on the collector. Be sure to choose the Distance Unit that matches your coordinate (.CRD) file (meters, US feet or international feet). Click OK and the next dialog now allows you to specify the range of point numbers to upload. Before clicking the Start Transfer button for range of points, go to the SDR and hit the Receive function key. The SDR is now waiting to receive so return to Carlson and click Start Transfer on the range of point dialog. The file transfer should now go.

Communication Settings

Besides matching the baud rate between Carlson and the collector, make sure that the collector is set to word length of 8 and 1 stop bit under the communication settings.

Print File

The Receive Sokkia Print File downloads a print report from the SDR33 data collector. This file is only used for printing report purposes in Carlson. This file is not used by Carlson to generate coordinate (.CRD) files or raw files. The first step is to choose Data format=Printed in the Communications menu of the SDR33. Next pick the Receive Print File button in Carlson. Then on the SDR33 choose the Send function and select a job to send. At this point the file is transferred. After downloading, the job report is displayed in the Carlson standard report viewer.

Example of Sokkia Printed Format:

```
SDR33 V04-04.25 (C) Copyright 1998 Sokkia May-29-80 23:39 01/29/1999
      Angle Degrees          Dist Feet
      Temp Farenht          Coord N-E-Elev
JOB      TRAV              Point Id Alpha (14)
      Atmos crn No          C and R crn No
      Record elev Yes      Sea level crn No
POS TP 1      North 10050.000      East 10000.000      Elev 0.000
```

```

POS TP 2      North 10000.000      East 10000.000      Elev 0.000
POS TP 3      North 9515.636      East 9551.975      Elev 37.611
              Code T3
POS TP 403    North 4967.527      East 5074.632      Elev 0.000
NOTE TS      Jan-01-80 00:14
** End of report **

```

Sokkia G2 This routine takes you directly to the SurvCOM dialog, similar to the Carlson SurvCE process.

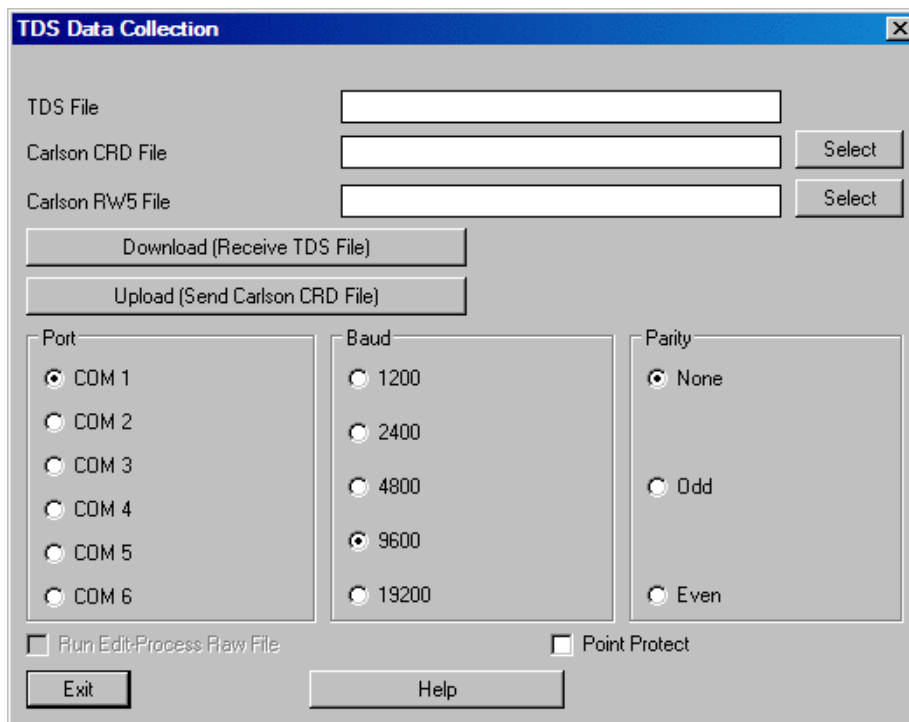
TDS

Download [HP-48 and Husky]

In the TDS program, go to the File Transfer routine. Choose the type of data to transfer (CRD or RAW). Next pick the Send function key. Stop here on the TDS and go to Carlson to run *Data Collection* in the Survey menu and pick TDS. Make sure that the COM port and baud rate are set correctly. Then pick the Download button. The Carlson program will now wait to receive the TDS file. Within 10 seconds select the file to send on the TDS. The file should be transferred now. When the transfer is complete, the program will ask you for the Carlson file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Download [Ranger and Windows CE]

In the TDS program, go to the Transfer routine and pick the Send File function. Set the "Connecting To" field to HP-48. Make sure that the COM port, baud rate and parity are set correctly and then pick OK. In the Type field of the file selection dialog, choose Coordinate Files or Raw Files. Stop here on the TDS and go to Carlson to run *Data Collection* in the Survey menu and pick TDS. Make sure that the COM port and baud rate are set correctly. Then pick the Download button. The Carlson program will now wait to receive the TDS file. Within 10 seconds select the file to send on the TDS and pick OK in the TDS dialog. The file should be transferred now. When the transfer is complete, the program will ask you for the Carlson file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.



Upload [HP-48 and Husky]

A Carlson coordinate (.CRD) file can be converted to a CR5 file and uploaded into TDS. Start in the TDS program, by going to the File Transfer routine. Then move back to Carlson and run *Data Collection* in the Survey menu and pick TDS. In the Carlson dialog, enter a TDS File name. This name should not include the drive and directory path or file extension. For example, if the coordinate (.CRD) file is c:\scadxml\data\simo2.crd then the TDS File name could be just SIMO2. Next pick the Select File button next to the Carlson coordinate (.CRD) File edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Now pick the Carlson Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the range of points but before clicking the Start Transfer button go to TDS and hit the Receive function key. Within 10 seconds go back and click the OK button on the range of points. The file should then transfer.

Upload [Ranger and Windows CE]

A Carlson coordinate (.CRD) file can be converted to a CR5 file and uploaded into TDS. Start in the TDS program, by going to the Transfer routine and pick the Receive File function. Set the "Connecting To" field to HP-48. Make sure that the COM port, baud rate and parity are set correctly and then pick OK. Then move back to Carlson and run *Data Collection* in the Survey menu and pick TDS. In the Carlson dialog, enter a TDS File name. This name should not include the drive and directory path or file extension. For example, if the coordinate (.CRD) file is c:\scadxml\data\simo2.crd then the TDS File name could be just SIMO2. Next pick the Select File button next to the Carlson coordinate (.CRD) file edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Now pick the Carlson Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the range of points and click the Start Transfer button.

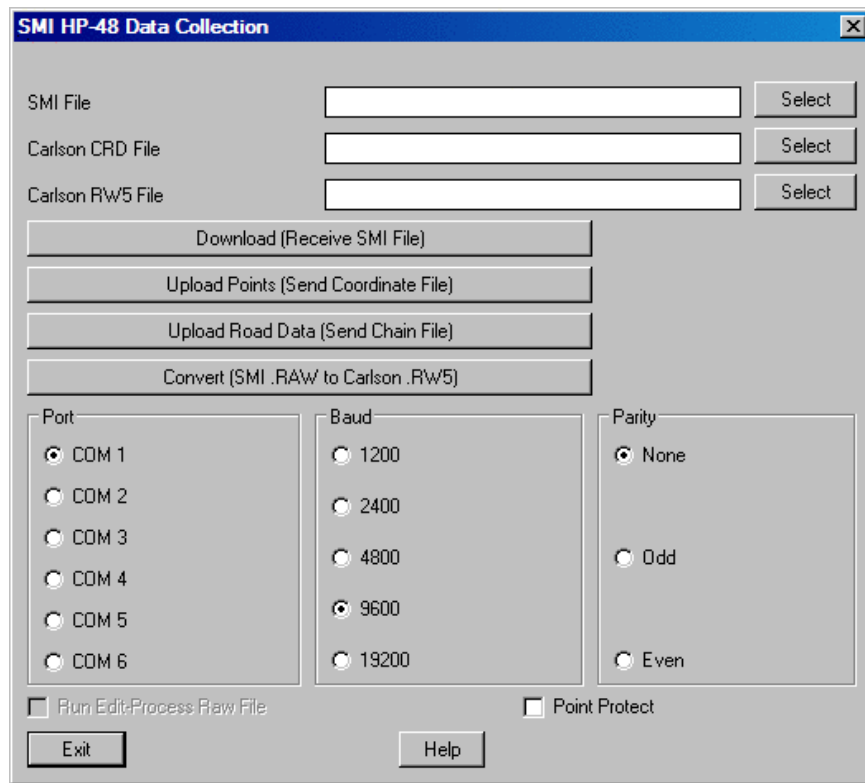
SMI

Download

To send point data from the SMI data collector, go to the file transfer routine by typing [More] [NXT] [TOPC] [COMM]. In SMI version 6 or later, type [Job][KERM][SEND]. Also in version 6, make sure that the first function key reads [NE] and not [XY] in the [Job][KERM] screen. Otherwise the coordinate northing and easting will be reversed. The [NE] stands for North-East coordinate order which is the format that Carlson expects. Also in the [Job][KERM] screen, make sure that the second function key reads [COMM] and not [SPACE]. The [COMM] stands for comma separators. Then enter the first point to send followed by the last point to send but before pressing Enter for the last point go to Carlson. Run *Data Collection* in the Survey menu and choose SMI. Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to SMI and press Enter for the last point to send. The file transfer should now go. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector. To send raw data, use the [Print][Raw] routine in SMI along with the same Carlson procedure used for point data.

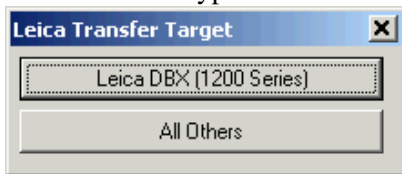
Upload

From the SMI data collector, go to the file transfer routine by typing [More] [NXT] [TO48] [COMM]. In SMI version 6 or later, type [Job][KERM][RECV]. Also in version 6, make sure that first function key reads [NE] and not [XY] in the [Job][KERM] screen. Otherwise the coordinate northing and easting will be reversed. Then enter the first point to send followed by the last point to send. Next enter the job name but before pressing Enter go to Carlson and run SMI under *Data Collection* in the Survey menu. In the Carlson dialog, specify the same job name as entered in SMI. Next pick the Select File button next to the Carlson CRD File edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the same range of points as entered on the SMI. Go back to SMI and hit Enter for job name followed by clicking the OK button for range of points in Carlson. The file transfer should now go.



Leica

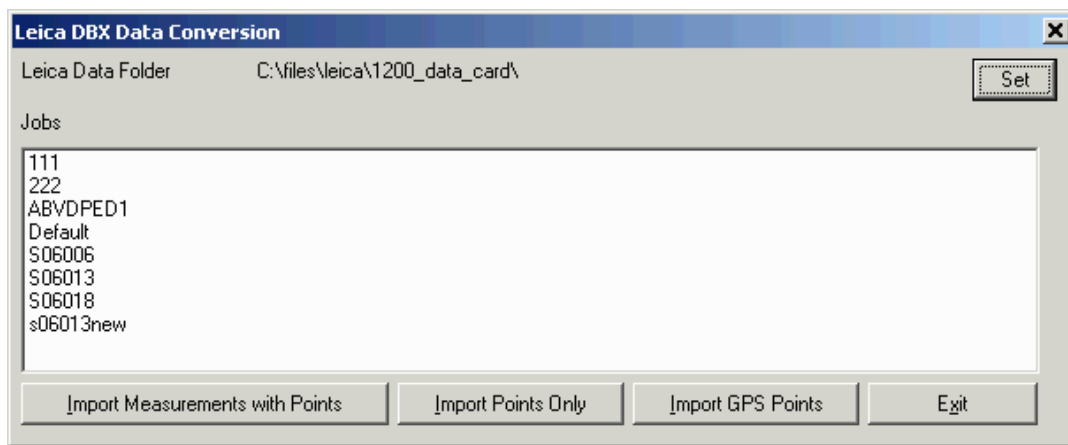
There are three types of Leica transfers: GIF-10, GeoCom and DBX.



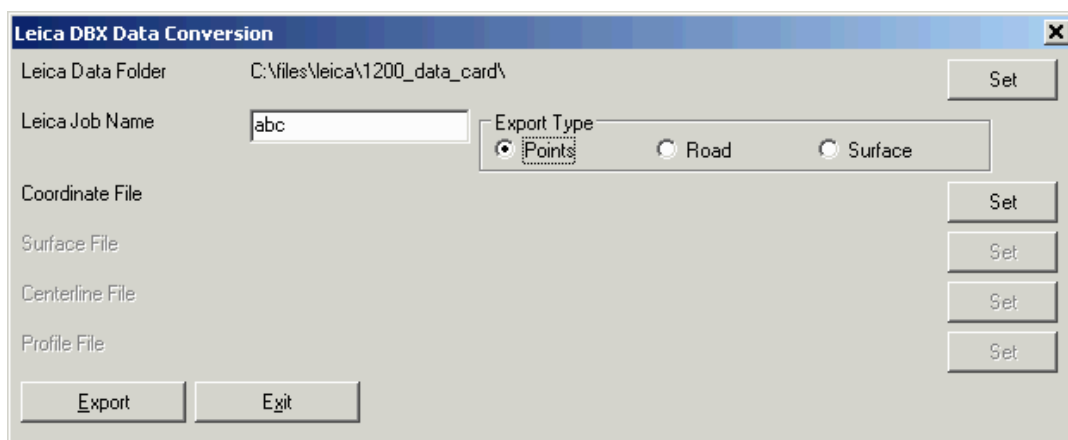
Choose newer Leica instruments, choose Leica DBX on first dialog. Then there is a choice between Import and Export.



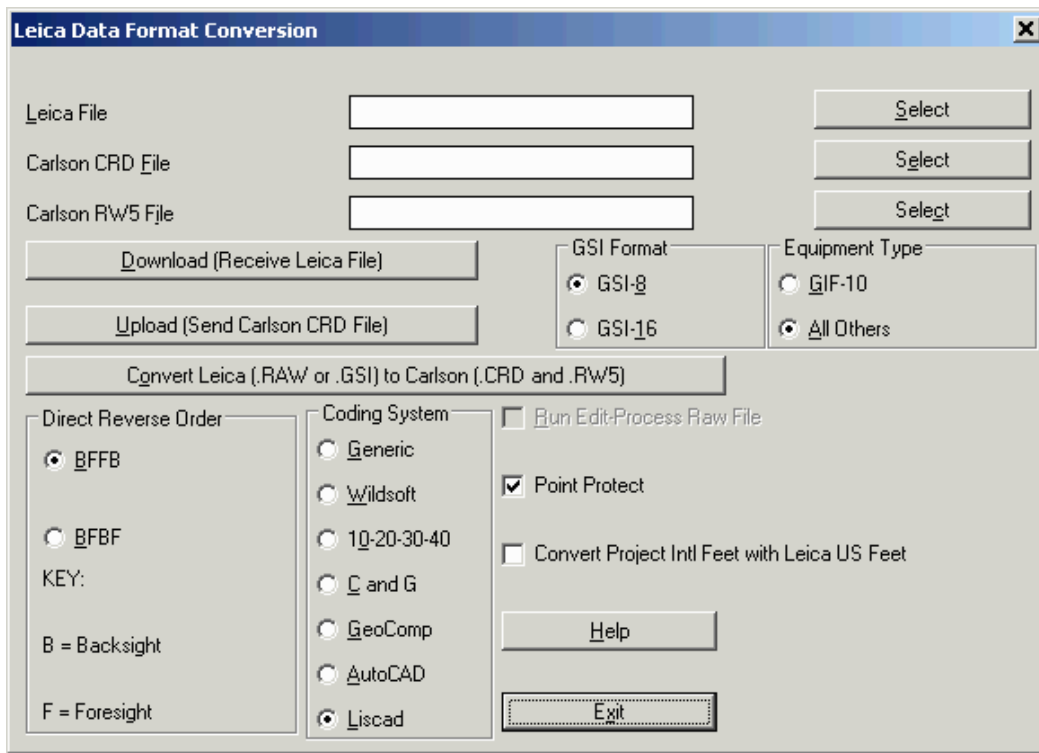
For Import, select the folder that contains the Leica DBX data. Typically the Leica DBX data is on a memory card that is inserted into the computer and gets assigned a drive name by Windows. Use the Set button to browse to this Leica DBX drive or folder. Then the program shows a list of the Leica projects in that folder. The Import Measurements With Points function reads the Leica data into Carlson CRD and RW5 files. The Import Points reads the Leica data into a Carlson CRD/CRDB file. If CRD file is chosen, then it will bring in attribute data to the Carlson NOT file. If CRDB file is chosen, then it will store attribute data to the CRDB file. The Import GPS Points function imports the Leica data into a Carlson RW5 file for GPS measurements.



For Export, select the folder to store the Leica DBX data to using the Set button. Enter in a job name for the new job in the Leica Job Name edit box. There are three types of project data that can be exported. The Points export converts a Carlson coordinate file to Leica format. The Road export converts Carlson profiles and centerlines to Leica format. The Surface export converts a Carlson TIN file to Leica format.



For GIF-10 and GeoCom, choose All Others on the first dialog Then the choice for GIF-10 or GeoCom is set in the Equipment Type field on the main dialog. For transferring with the Leica instruments, the GeoCom program shows a dialog of the available COM ports on your computer. On the first time that you transfer to an instrument, you will need to pick the Instruments button and register the instrument from the list. Pick the Port Settings button to make sure that the communication settings match the instrument.



To download a file with GeoCom, make sure that the instrument is ON and connected to the computer by serial cable. The instrument also needs to be in GeoCom mode. Then pick the Download in the Carlson dialog. In the GeoCom program, open the computer COM port that the instrument is connected to by picking the '+'. Then open the Memory Card and GSI folders. Next select the file to transfer and click the OK button. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

To upload a file with GeoCom, specify the file name to be created on the instrument in the Leica File field and pick the Upload button in the Carlson dialog. Then the program will prompt for the range of points to transfer. Fill out the range and pick the Start Transfer button. Then the GeoCom program will start. Open the computer COM port by picking the '+'. Then open the Memory Card folder and highlight the GSI folder and click OK.

The upload and download file transfer works with the GIF-10 data collector. The GIF-10 communication settings should be the following:

- Baud: 9600
- Parity: NONE
- Protocol: NONE
- Stop Bit: 1
- End Mark: CR/LF
- Connected As: Some computers use DCE and others use DTE

Download

From the GIF-10, go to the file transfer routine. Then go to Carlson and run *Data Collection* in the Survey menu and choose Leica. Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to GIF-10 and select the file to send. The file transfer should now go. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Upload

From the GIF-10 data collector, go to the file transfer routine. Then go to Carlson and run Leica under *Data Collection* in the Survey menu. In the Carlson dialog, specify the job name in the Leica File field. Next pick the Select File button next to the Carlson coordinate (.CRD) File edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Then click the Upload button. A dialog now allows you

to specify the range of point numbers to upload. Before clicking the OK button for range of points, go to GIF-10 and start the receive by highlighting Receive and pressing the Run button. The GIF-10 now shows the available job numbers. Choose a job to receive the transfer using the arrow buttons and then press the Run button.

Converting

Carlson supports raw and coordinate data collected using three different Leica Operation Codes: Wildsoft and 10-20-30-40 as well as the newer LISCAD. Moreover, data could be in the GSI8 format or the newer GSI16 format. Some example files are shown here.

GSI8 format data file using LISCAD Operation codes:

WILD GIF-12

410149+00000001 42....+00005003 43....+00005.42 44....+00005.25 45....+00005000
110150+00005000 21.324+35959480 22.324+09238590 31..01+00228271
410151+00000005 42....+00010100
110152+00005001 21.324+35156390 22.324+09303500 31..01+00133532
410153+00000005 42....+00070100
410154+00000014 42....+00000ELM
110155+00007082 21.324+34739450 22.324+09322050 31..01+00137685
410156+00000005 42....+00070102

GSI16 format data file using LISCAD Operation codes:

*110001+0000000000000001 84..11+0000010000000000 85..11+0000003000000000
86..11+0000000001000000 87..11+0000000000005170
*410002+0000000000000009 42....+0000000000000001 43....+000010000000.000 44....+000003000000.000
45....+000000001000.000
*410003+0000000000000001 42....+0000000000000001 43....+00000000005.330 44....+00000000000.000
*410004+0000000000000004 42....+00000000178.1530
*410005+0000000000000003 42....+0000000000000002 43....+0000000000000001
*110006+00000000000000RO 21.324+0000000017815300 22.324+0000000008424260
31..01+0000000000000000
*410007+0000000000000100
*410008+000000000000012 42....+00000000005.090
*110009+0000000000000002 21.324+0000000000831230 22.324+0000000008130270
31..01+0000000000089996
*110010+0000000000000002 21.324+0000000018831230 22.324+0000000027829250
31..01+0000000000089996
*110011+00000000000000RO 21.324+00000000035815170 22.324+0000000027539300
31..01+0000000000000000

GSI8 format data file using Wildsoft Operation codes:

410001+00000001 42....+00000013 43....+00000000 44....+00000012 45....+00981101
410002+00000002 42....+00000013 43....+00005.42 44....+00000012 45....+00000000
410003+00000032 42....+00000500 43....+00004.26 44....+00000012 45....+00000000
410004+000000TP 42....+00000000 43....+00000000 44....+00000000 45....+00000000
110005+00000501 21.124+00000000 22.104+09136260 31...1+00000000 51..0.+0012+000
110006+00000502 21.124+03741320 22.104+08915570 31...1+00246818 51..0.+0012+000
110007+00000503 21.124+03915180 22.104+08919040 31...1+00251956 51..0.+0012+000
110008+00000504 21.124+06530420 22.104+08839360 31...1+00113998 51..0.+0012+000

Leica raw files usually have a .RAW or .GSI extension. The primary difference in the GSI8 and GSI16 formats is that information is contained in data blocks of 16 characters in the GSI16 format, while it is contained in blocks of 8 characters in the GSI8 format. Leica instruments make it possible to have both the GSI8 as well as GSI16 data formats in the same raw file. However, lines with the GSI16 format data will always start with an asterisk (*) character, to distinguish them from the GSI8 format. There is no distinction between Leica raw files collected in the Wildsoft and LISCAD operation codes.

Supported Wildsoft codes:

- 1: Start Job
- 11: Assign Coords
- 12: Coord Offset
- 13: Target Height
- 14: Add to Tgt Ht
- 15: Add to Meas Dist
- 2: Occupy Point
- 21: Occupy Saved Point
- 3: FS to Trav Pt
- 31: FS to Single Pt
- 32: Radial Sideshots
- 33: Sets of Angles
- 4: Closing Pt
- 41: Closing Angle
- 50: BS to Benchmark
- 51: FS to Turn Pt
- 52: BS to Turn Pt
- 53: FS to Benchmark
- 60: Save Point
- 61: Recall Point
- 62: Compare Point
- 63: Remark

Supported LISCAD codes:

- 1: New instrument setup
- 2: New target height
- 3: Sets of directions
- 4: Fixed azimuth
- 5: Feature code
- 6: Measured offset
- 8: Line creation for sub-codes 1 (straight string), 2 (curved string) and 6 (arc by 3 points)
- 9: Fixed coordinates
- 11: Close string
- 14: Additional description
- 20: Start of job
- 27: Feature code
- 90: Split feature code
- 100+: Descriptions

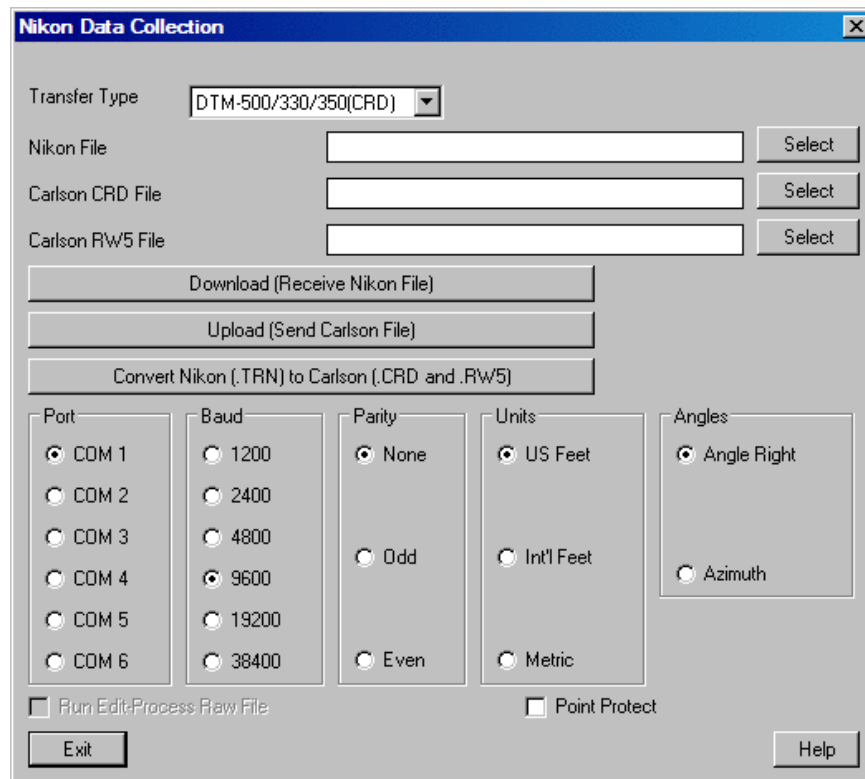
The Convert button can be used to convert any Leica format file into a Carlson format file. For example, if you have a Leica PCMCIA card then there is no serial cable transfer to do. Instead use the Convert routine to make the Carlson raw data (.RW5) and coordinate (.CRD) files. Since there is no distinction between Wildsoft and LISCAD files, the user must know in advance which format has been used in the file. Then, select that particular option (Wildsoft, 10-20-30-40 or LISCAD) under the "Coding System" option at the bottom of the dialog box, as shown in the previous page. Another option that the user needs to choose is the order in which foresight-backsight readings have been recorded in the raw file, BFFB or BFBF, as explained in the dialog box. Then, the user can simply pick the "Convert" button and the program prompts for the input "Wild/Leica File" (raw file), and the output "Carlson RW5 file" and "Carlson CRD file", if they are not already filled.

Nikon**Download**

First choose the equipment and data type under the Transfer Type list. Also check that the communication and data format settings match your collector. Then click the Download button and follow the on-screen directions. When the transfer is complete, the program will ask you for the Carlson coordinate file (.CRD) and raw file (.RW5) to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector. The original data from the collector is stored in a file name with the same name as the coordinate file except with a .TRN extension. For example, job5.crd would have job5.trn.

Upload

Pick the Select File button next to the Carlson CRD File edit box and choose the CRD file to send. Check that the COM port and baud rate are set correctly and then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Set the points and then click the Start Transfer button. The file transfer should now go.



Convert Nikon to Carlson

The Convert button will translate the Nikon raw file format (.TRN or .RAW) into Carlson coordinate (.CRD) and raw (.RW5) files.

Portion of typical Nikon file format:

```
MP,1,,5000.0000,5000.0000,0.0000,T/1
CO,31-Oct-1999 11:42:38
ST,1,,2,,0.0000,0.00000,0.00000
SS,3,0.0000,152.1510,359.59590,90.44100,11:43:38,T/2
SS,4,0.0000,127.5560,0.06040,90.40110,11:44:45,CON
SS,5,0.0000,97.1820,2.19580,90.52460,11:45:43,CON
```

Geodimeter

Download

From the Geodimeter data collector, go to the file transfer routine by pressing the PRG (Program) key and entering

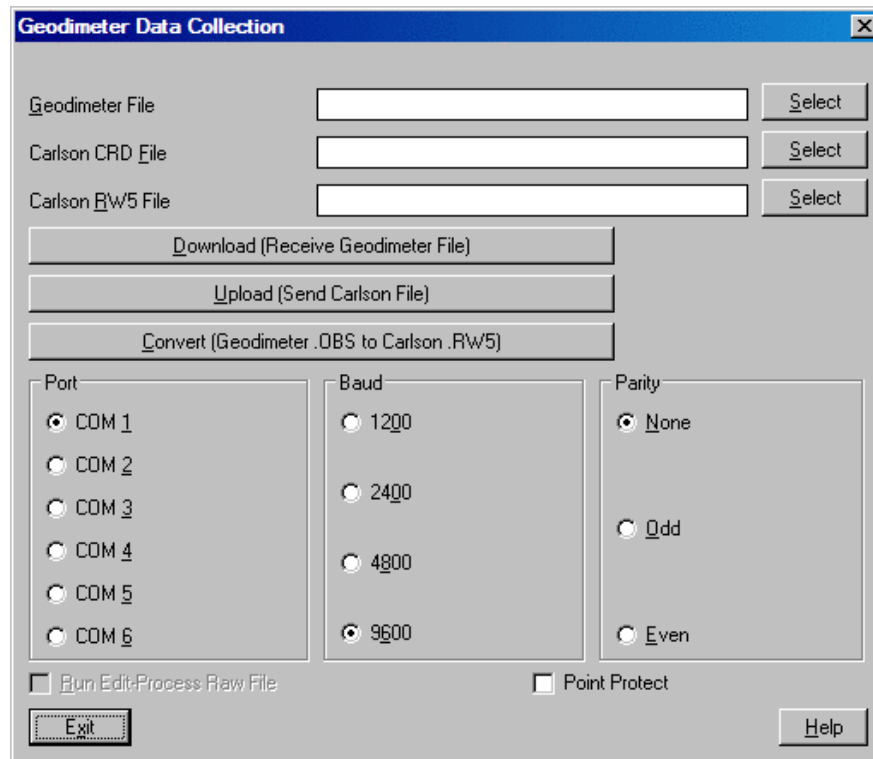
program 54. Then choose Imem (option 1) as the source. Next choose the file type to send as either Job (measurement data) or Area (point data). The Geodimeter will then prompt for the job name. Next enter Serial (option 3) as the destination. A confirmation screen appears showing the serial port settings. Here are some typical settings:

COM=1,8,0,9600

Before pressing enter (ENT key), go to Carlson and run *Data Collection* in the Survey menu and choose Geodimeter. Then click the Download button and within 15 seconds, go back to the Geodimeter and press Enter. The file transfer should now go. When the transfer is complete, the program will ask you for the Carlson coordinate file and raw file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Upload

In Carlson, run Geodimeter under *Data Collection* in the Survey menu. Pick the Select File button next to the Carlson CRD File edit box and choose the CRD file to send. Check that the COM port and baud rate are set correctly and then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the points to send but before clicking OK, go to the Geodimeter data collector. Start the file transfer routine by pressing the PRG key and entering program 54. Then choose Serial (option 3) as the source. The Geodimeter will display the serial port settings. Check these values and press enter. Next choose Area (option 2) as the destination. Then enter the job name. The Geodimeter is now listening for data. Quickly go back to Carlson and click OK on the points to send dialog. The file transfer should now go



Convert

The Convert button will translate the Geodimeter raw file format (.OBS) into Carlson coordinate (.CRD) and raw (.RW5) files.

Communication Settings

If the Geodimeter is not communicating with Carlson, run function 79 on the Geodimeter and make sure that it is set to 4. This setting is for the transfer message end of sequence format.

Supported Geodimeter Codes

The following Geodimeter codes are processed when converting the Geodimeter raw file. All other codes are

recorded as descriptions in the Carlson rw5 file.

0=Info

1=Data

2=Station No

3=Instrument Height

4=Point Code

5=Point Number

6=Signal Height

7=Horizontal Angle

8=Vertical Angle

9=Slope Distance

11=Horizontal Distance

17=Horizontal Angle

18=Vertical Angle

21=Horizontal Reference Angle

30=Atmospheric Correction

37=Northing

38=Easting

39=Elevation

40=Delta North

41=Delta East

42=Delta Elevation

45=Correction To Bearing

46=Standard Deviation

50=Job Number

51=Date

52=Time

53=Operator

54=Project Id

55=Instrument Id

56=Temperature

60=Shot Id

61=Activity Code

62=Reference Object

70=Entered Radial Offset

71=Entered Angle Offset

72=Calculated Radial Offset

73=Calculated Angle Offset

74=Air Pressure

Portion of typical Geodimeter file format

5=108

4=13POC

6=5.000

7=238.0708

8=89.2236

9=440.39

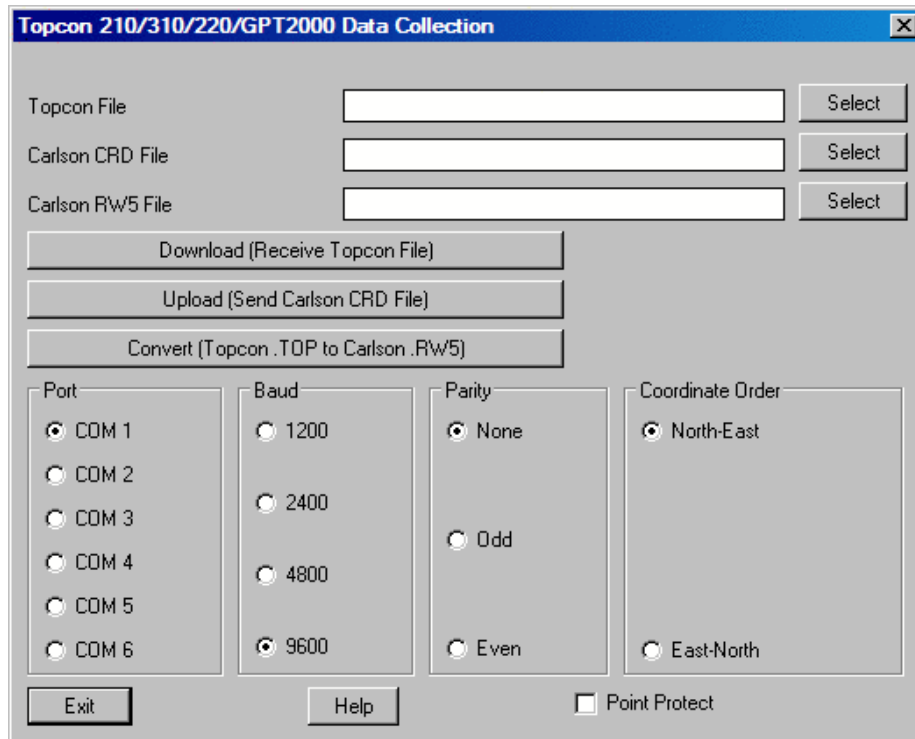
37=767.42

38=4626.07

39=699.795

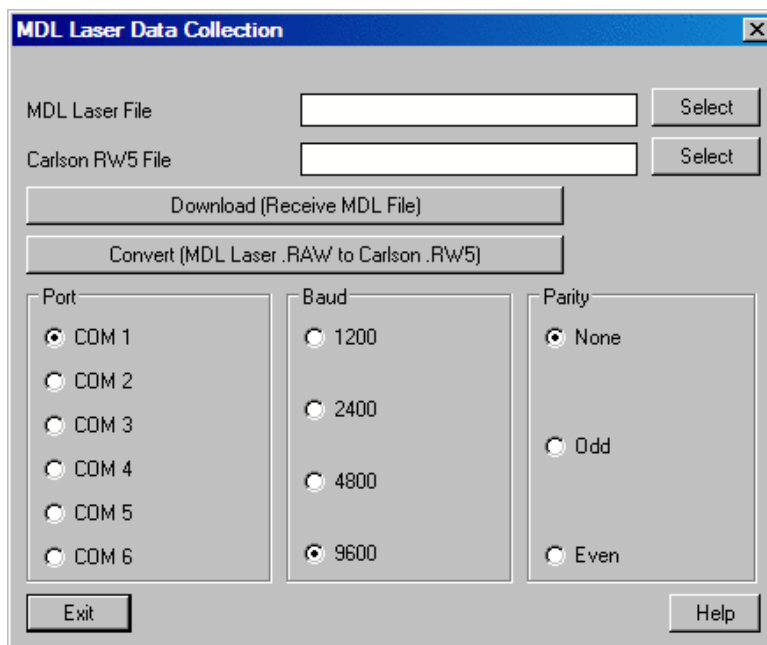
Topcon 210/310/220/GPT2000

This command supports these above Topcon models.



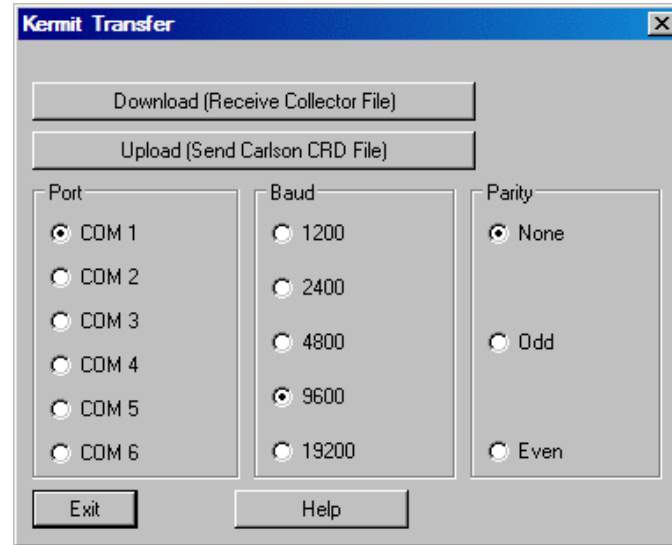
MDL Laser

The MDL Laser outputs a raw file of angles, distances and codes as one long string of data which can be converted into a Carlson raw data (.RW5) file. There is no coordinate data in the MDL raw file. So you need to run *Edit-Process Raw File* to calculate coordinates from the raw data. The Download button will transfer the MDL raw data from a BDI logger.



Kermit

Kermit can be also used for transferring files with accuracy. The dialog looks like this:



Pulldown Menu Location: Survey

Keyboard Command: datacolt

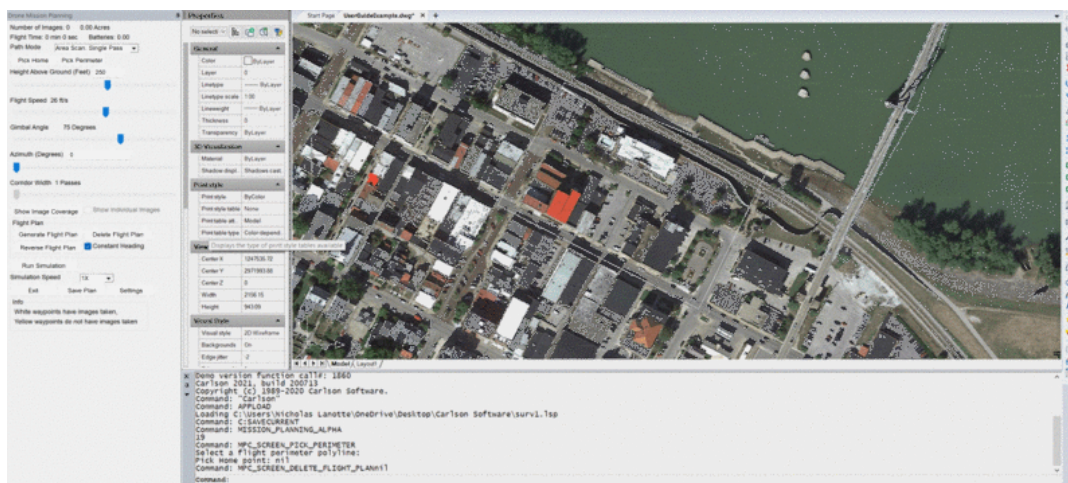
Prerequisite: None

Drone Mission Planning

This command generates a drone flight plan for taking aerial photos. Before running this command, set the Coordinate System for the drawing using the Drawing Setup command. Also before this command, you can optionally load an aerial image into the drawing using the Place Google Earth Image command.

Basic Workflow

1. When the Drone Mission Planning command starts, it places a dialog on the left side of the drawing.



2. Select a path mode. Defaults to Area Scan: Single Pass.
3. Draw a Polyline to perform the Area Scan on. The Polyline must be closed for an area scan or structure scan and must be open for a corridor scan. Structure scans and corridor scans can only be performed on polylines that do not contain arcs.
4. While Area Scan or Structure Scan is the selected Mode there will be a "Pick Perimeter" button to select the polyline. If the Path Mode is Corridor Scan this button becomes "Pick Corridor".
5. Click "Pick Home" then select the location in CAD you wish to launch the drone from. You may be asked to select a home position after selecting the scan polyline.
6. The flight plan should automatically be plotted. The number of images, estimated flight time and the area covered will be displayed at the top of the drone mission plan dialog. Use the parameter sliders to adjust the flight plan in real time. Yellow points are waypoints without images and white points are waypoints with images.
7. When the plan is satisfactory, press the "Save Plan" button to save the drone mission plan as a .plan file. These files can be read by the DJI Companion App, Drone Deploy and qGroundControl. For example, the .plan file can be loaded into Drone Deploy where the mission can be edited further and uploaded to the drone.



Features

Icons: The Mission Home position is represented by the Yellow target. The flight plan is drawn in solid White. Each location where a photo is taken will be shown as a circle with an arrowhead inside. The direction of this arrow head is the heading of the camera while taking this image.

Height Above Ground: This setting controls the height above ground level the aircraft will fly for the mission. The highest allowed height is 400 ft and the lowest allowed height is 40ft. Higher heights result in fewer images taken.

Flight Speed: This setting controls the flight speed of the aircraft. Faster speeds result in lower mission times. The lowest speed is 1ft/s and the fastest speed allowed is 50ft/s

Frontlap: This setting controls the amount of overlap in the forward direction of each image. A front lap of 60% minimum is recommended. Larger overlap results in more images.

Sidelap: This setting controls the amount of side overlap of each image. A side lap of 70% minimum is recommended. Larger overlap results in more images.

Gimbal Angle: This setting controls the Gimbal angle for the flight. 90 degrees points the camera straight down while smaller angles orient the camera closer to horizontal. The recommended Gimbal angle is 75 degrees.

Azimuth: This setting controls the angle flight lines for the Area and Structure scan will be drawn at. Adjust this setting if necessary if the generated plan does not completely cover the scan area or the drone would fly over areas that are not desired. 0 degrees points north and the degrees increase in the clockwise direction

Corridor Width: This setting only becomes active while in Corridor Scan Mode. This controls the amount of back

and forth passes the drone will take along the corridor. The more passes the wider the corridor becomes.

FOV: This setting should be sent to the FOV of the camera on the drone

Aspect Ratio X: This setting should be sent to the X portion of the aspect ratio of the camera on the drone

Aspect Ratio Y: This setting should be sent to the Y portion of the aspect ratio of the camera on the drone

Delete Flight Plan: This button will delete everything in the flight plan

Reverse Flight Plan: This button will reverse the direction of the flight plan

Constant Heading: Checking this box has the heading of the drone remain constant throughout the mission (except when not possible with a corridor scan and outer perimeter of structure scan). Unchecking the box will have the drone fly more like an airplane.

Show Image Coverage: Pressing the "Show Image Coverage" button will draw the area that the camera will capture during the mission. To see the area of each individual image, enable the "Show Individual Images" option.

Simulator: Pressing the "Run Simulation" button after a drone mission plan has been made, will run a simulation with the drone represented by a red target. It will move along the path with the path traversed highlighted in red. Use the Simulation Speed drop box to speed up or slow down the simulation. The "Run Simulation" button will become the "Stop Simulation" button while the simulation is running. Pressing this button stops the simulation. The current image count will appear next to this button and update as the drone flies over each of the image points.

Automatic Checks: While generating the drone mission plan, the routine automatically checks to ensure the entire scan area is covered by the images. If it is not, the user will be shown a warning. Use the "Show Image Coverage" features to view what area is not covered. Use the parameter sliders to adjust the drone mission plan until the full area is covered. While generating the drone mission plan, the routine automatically checks to ensure that the drone mission plan is within 400 meters of the home position so that a line of sight from the operator to the drone is possible. If the drone mission plan extends beyond 400m, the user will receive a warning.

Area Scan Single Pass: This Path Mode creates a flight plan that covers a selected perimeter, only flying the drone over the area once. The flight paths are drawn in the direction of the Flight Angle and with offsets that are a function of the Height Above Ground, Sidelap and camera parameters.

Area Scan Double Pass: This Path Mode is the same as the Single Pass mode except the drone will not fly over the area two times. The second pass will have the flight lines draw perpendicular to the previous pass.

Corridor Scan: This Path Mode will create a drone mission plan along an open polyline with each image evenly spaced out along the polyline. The spacing is a function of the Height Above Ground, Sidelap and camera parameters. Adjusting the Corridor Width slider to be larger than 1 pass will result in the scanned area being wider as each pass is made at an offset from the original.

Structure Scan: This Path Mode will create the same path as a Area Scan: Single Pass with the addition of a perimeter scan at the end. The Perimeter Scans sets the heading of the drone camera to be towards the inside of the closed polyline. This feature allows structures to be scanned with more images from better 3D models of the exterior vertical surfaces. The scan polyline can only include line segments.

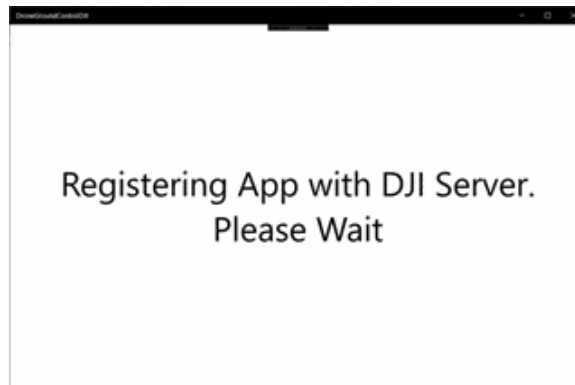
Elevation Compensation: For use on highly uneven terrain. The drone is only able to sense the height above ground level from where it was launched. Terrain that has large elevation differences can result in the camera captures too large or too small of an area, or in the presence of a large hill, cause the drone to crash. This setting retrieves the elevation at each photo point and applies an offset to the Height Above Ground level so that the drone's actual height above ground level at each photo point is the desired value.

Stop For Pictures: Enabling this setting will have the drone stop at each Geolocation to take each picture. This may result in more accurate image overlap and stability but will increase flight time and use more of the drone's battery.

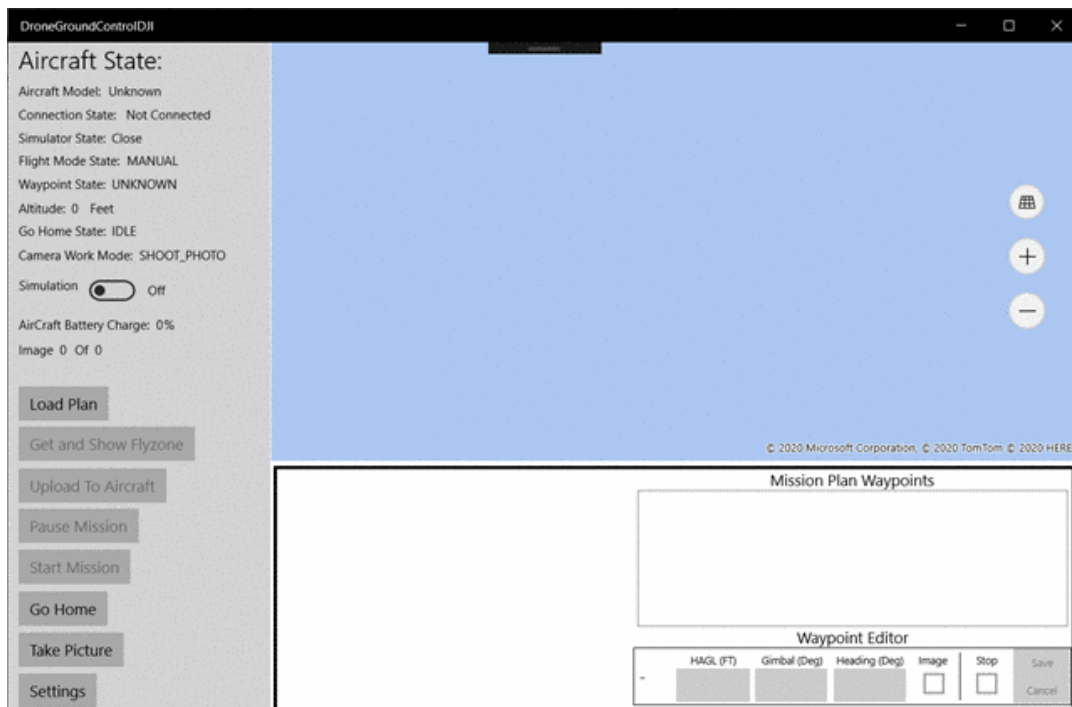
DJI Companion APP

Basic Workflow

1. After launching the app there will be a screen that say it is registering with DJI. This screen will hold itself until a response is received. If this screen gets stuck on for a while then it is something wrong with DJI's servers.



2. Upon success the main window will be shown.



3. Press load plan and select the drone mission plan created by pointcloud. The screen should look similar to this image.



4. Press upload to aircraft. Once the mission is done uploading the Waypoint State should say "READY_TO_EXECUTE" and the Start Mission button will be enabled. Sometimes the drone won't report this right away. Try uploading the plan again or restarting the app if this happens. There are a lot of background threads running within DJIs SDK and some of these errors are a result of not knowing exactly how they work.
5. Press Start Mission when ready. The mission will be executed, and the drone will auto land when completed. If the mission fails to start, an error message will be shown to the user. The Aircraft position will be shown on the map and the aircraft Altitude will be updated in real time. There is other aircraft state information displayed in the app that is mainly used for testing purposes.
6. You can pause or stop the mission at any time while it is running.

This process should result in a lot of pop-ups to be displayed. If everything is running as expected they should all report "NO_ERROR".

Features

Load Plan: This button will open the Windows File Selection Tool. Select the .plan file for the current mission. If there are no problems the application will report "NO_ERROR".

Get and Show Flyzone: This button will retrieve the nearby flyzones and display them on the map.

Upload to Aircraft: This button will upload the selected plan to the aircraft. The program should report "No Error" if the plan is possible for the drone to execute and will report errors otherwise. In some rare cases, this may return "NO_ERROR" but the Waypoint State will revert back to "READY_TO_UPLOAD" instead of "READY_TO_EXECUTE". This is an internal error with the drone firmware that the DJI SDK will not report. Making a slight modification to the drone mission plan in the Pointcloud drone mission planner should solve this issue.

Simulation: It may be useful to simulate the drone mission plan beforehand. Enabling the Simulation toggle switch will put the drone in simulation mode. The application will run as normal, but the drone will not take off. The drone must be turned off and on again to exit Simulation Mode.

Pause/Resume Mission: This button can be used while a mission is running to pause or resume the mission. There is an option to swap batteries when pausing the mission. As of writing this guide the feature does not work. Disconnecting the drone and reconnecting it causes an error with DJIs software preventing the mission from being continued. This needs to be fixed before using this feature

Start/Stop Mission: This button is used to start the drone mission plan, or if the mission is running it is used to

stop the mission.

Go Home: This button will have the drone return home and land.

Take Picture: This button will have the drone camera take a picture and save it to the SD card if enough space is available.

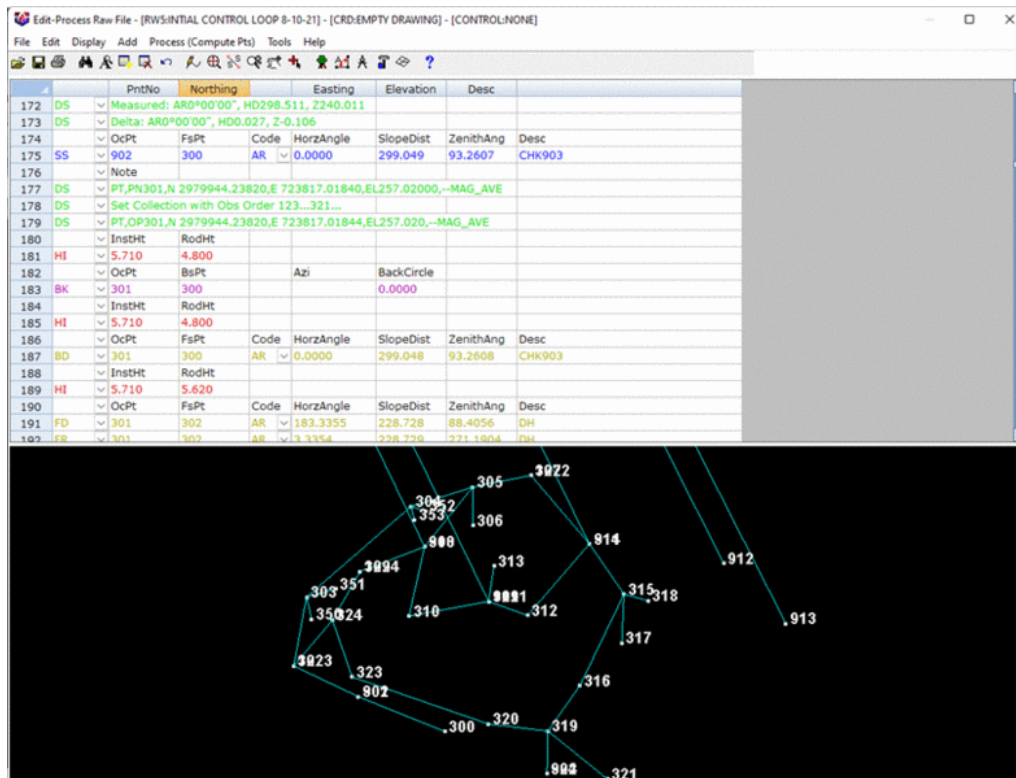
Pulldown Menu Location: Survey

Keyboard Command: mission_planning

Prerequisite: Set Coordinate System in Drawing Setup

Edit-Process Raw Data File

This program reads or creates a raw data (.RW5) file that contains various lines of data (records) that can be likened to a surveyor's field book. The RW5 file can contain both Traverse/Side Shot data from an EDM total station as well as GPS position coordinates and vector data from a GNSS receiver.



The Edit Process Raw Editor contains 7 separate pull down menus:

File

Edit

Display

Add

Process (Compute Pts)

Tools

Help

Within the Editor itself, you can specify point coordinates, job information, notes, and the angles and distances that make up traverse or sideshots records. Once the raw data is created or read it can be processed and reduced to coordinates that are stored in the current coordinate (.crd; .cgc; .mdb; .zak) file.

The raw file can also be created or appended using the *Locate Point*, *Traverse*, *Sideshot*, and *Inverse* commands in the COGO menu. To store the data inputs from these commands into a raw file, first toggle on the *Raw File ON/OFF*

command on the COGO menu. It is possible to always have the raw data file open to store data inputs. To enable this option, choose Configure from the Settings menu, then choose Survey Module, then choose General Settings. Turn on the Automatic Raw File toggle in this dialog.

Legacy raw files created by TDS data collector programs are also compatible without conversion. The command *Data Collectors* on the Tools menu has options for reading other data collectors native file formats and converting them to raw data (.RW5) format. Within the raw data editor, the File menu includes an import menu for converting raw data from other formats.

When you select the *Edit-Process Raw Data File* command you are prompted to specify the name of the raw data (.RW5) file. The current coordinate file is used automatically. To change the current coordinate file, use the Set Coordinate File command in the Points menu before starting this command. If no coordinate file is current, the program will prompt you to set the current coordinate (.CRD) file.

Edit-Process Raw Data File uses a spreadsheet for editing the raw data as shown.

174		OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	Desc
175	SS	902	300	AR	0.0000	299.049	93.2607	CHK903
176	TR	Note						
177	SS	PT,PN301,N 2979944.23820,E 723817.01840,EL257.02000,--MAG_AVE						
178	HI	Set Collection with Obs Order 123...321...						
179	PT	PT,OP301,N 2979944.23820,E 723817.01844,EL257.020,--MAG_AVE						
180	DS	InstHt	RodHt					
181	HI	5.710	4.800					
182		OcPt	BsPt		Azi		BackCircle	

Each row of the spreadsheet is represented by a number located at the far left side of the editor. Various messages and reports often reference possible problems with the data by this row number. Each row of the spreadsheet represents one record of data. There are 14 types of **data records**. The type of data record is shown in the first column. Different record types use different numbers of columns. Whenever the data record type changes between rows, a record header is added to the spreadsheet that describes each column of data in the following row. To edit the raw data, simply highlight the cell and type in the new value. To change the type of record, pick on the down arrow in the first column and choose a new data type from the list. To delete a row, highlight any cell in the row and hit the Delete key or choose Delete Row from the Edit menu. Records can be added pressing the Insert key, pressing the down arrow key from the last line in the spreadsheet, or by choosing one of the add records from the Add menu.

The different data record types are described below.

TR (Traverse)

The traverse record contains the occupied point number, foresight point number, angle mode, horizontal angle, distance, vertical angle and description. When processed, this record will calculate and store the coordinates for the foresight point. Traversing also moves the setup by making the traverse foresight point the next occupied point and the traverse occupied point becomes the next backsight point. The different angle codes are NE for northeast bearing, SE for southeast, SW for southwest, NW for northwest, AZ for azimuth, AL for angle left, AR for angle right, DL for deflection angle left and DR for deflection angle right. To set the angle code, pick on the Code down arrow and choose from the list. The horizontal and vertical angles should be entered as dd.mmss. For example, 45.2305 is 45 degrees, 23 minutes and 5 seconds. The vertical angle can be shown as vertical angle (0 degrees level), zenith angle (90 degrees level) or elevation difference. The vertical angle mode is set in the Display menu. The distance mode is also set in the Display menu as either slope or horizontal distance. The description field is used as the foresight point description.

SS (SideShot)

The sideshot record is the same as the traverse record except that sideshot does not move the setup.

HI (Instrument and Rod Height)

This record sets the instrument and rod heights used in elevation calculations. This record should precede any

traverse and sideshot records that you want the heights applied to.

BK (BackSight)

The backsight record contains the occupied point number, backsight point number, backsight azimuth and the set azimuth. This record should precede any traverse and sideshot records that use this setup. If no backsight point is entered, the program uses the backsight azimuth to turn angles from. The Set Azimuth is the circle reading of the instrument when sighting the backsight. A Set Azimuth of zero is the default.

PT (Store Point)

The store point record consists of a point number, northing, easting, elevation and description. When processing, this data will be stored as a point in the coordinate file. If the first Occupied point and/or the initial Backsight point are not defined in the coordinate file set for processing to, both points will need to be added to the rw5 file as PT (Store Point) records.

DS (Description)

The description record is an additional note that appears in the spreadsheet editor and printouts. This record can contain various information that is recorded in data collectors during field operations. This data can vary from user, temperature and general data to each line of data associated with "Set Collection". When "Sets" of data collected using various brands of data collection software is converted/imported into the raw editor, the actual measurements made during the spinning of the angles and distances are recorded as DS records and the mean value of the angle and distance is recorded as a SS record. The OF records are recorded by SurvCE for offset left/right, in/out and depth. DS records are not used in processing, but are a record of the values entered in the data collector for reference. SurvCE also stores names of images in the description record using AT,TN coding followed by the name of the image file. The program will create links to these images for the points. For example,
-AT,TNPHOTO1,TV.\Pictures\PN1_2208.jpg

CL (Closing Shot)

The closing shot record is the traverse record where the foresight point is the closing point for the traverse. This record is used by the adjustment commands in the Process menu. There should be only one CL record in each Traverse loop (Name Record) in the raw file. If there is no CL record, the process adjustment routines will prompt for which shot is the closing shot. The closing shot can also be define in the field by using special codes defined in the Open Settings found under the File pulldown within the editor. Please refer to the "Open Settings" documentation below for more information on these codes.

AB (Angle Balance)

The Angle Balance record is the measurement data observed that closes the angles of the traverse. Typically this record is the measurement data recorded from the closing shot to the initial backsight point. The backsight could be either external or internal to the traverse. Angle Balance routine in the Process menu uses this record and compares the angle between the occupied point and foresight point of this record with a user-specified reference angle. There should be only one AB record in the raw file. If there is no AB record, then the Angle Balance routine will prompt for which shot to use as the angle balance.

CL + AB (Closing Shot and Angle Balance)

This record is used as both the closing shot and angle balance records.

FD (Foresight Direct)

The foresight direct is a traverse record used in a direct and reverse set. When the program finds one the of direct-reverse measurement records, it will look for the other three records to complete the set.

FR (Foresight Reverse)

The foresight reverse is a traverse record used in a direct and reverse set.

BD (Backsight Direct)

The backsight direct is a traverse record used in a direct and reverse set.

BR (Backsight Reverse)

The backsight reverse is a traverse record used in a direct and reverse set.

EL (Elevation Only)

This record sets the elevation in the CRD file for the specified point number. Often used when an existing point with good vertical control is being traversed through. Using this record type for the point would keep the elevation from changing on the existing point regardless of the measurement data.

AZ (Azimuth Only)

Applies to SurvNET, the optional Network Least Squares analysis and adjustment routine.

CSE (Control Standard Error)

Applies to SurvNET, the optional Network Least Squares analysis and adjustment routine.

SSE (Set-up Standard Error)

Applies to SurvNET, the optional Network Least Squares analysis and adjustment routine.

MSE (Measurement Standard Error)

Applies to SurvNET, the optional Network Least Squares analysis and adjustment routine.

NAME (Traverse Name)

This record acts as an identifier for the group of records that make up a traverse. All the records after the NAME record belong to that traverse up to the next NAME record or the end of the file. This record allows you to have multiple traverses in one raw file. When running one of the Process commands, the program will bring up a list of all the traverse names. Simply choose which traverse to process. If you have only one traverse in the raw file, then you don't need the NAME record.

GPS (GPS Measurement)

This record contains the Latitude and Longitude of a point as measured by GPS surveying equipment using Carlson SurvCE data collection software. This record has additional information tied to it such as localization files, geoid files, coordinate projection systems etc. This record has its own processing routine in the Process pulldown within the editor. Processing procedures are discussed in the Process (Compute Pts) pulldown documentation.

BP (Base Point)

This record contains the Point Name, Latitude, Longitude, Elevation and Description of a GPS base receiver point.

G0 (GPS Vector Comment)

This record is the description for the GPS vector.

G1 (GPS Vector Deltas)

This record contains the delta x,y,z for the GPS vector.

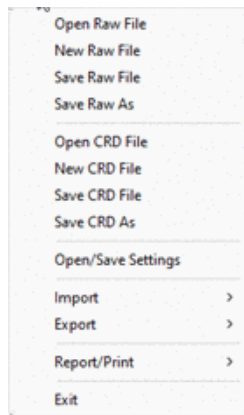
G2 (GPS Vector Variances)

This record contains the x,y,z variances for the GPS vector.

G3 (GPS Vector Covariances)

This record contains the x,y,z covariances for the GPS vector.

File Menu



Open Raw File

This command prompts for a rw5 file to load into the editor.

New Raw File

This command clears the editor spreadsheet.

Save Raw File

This saves the rw5 file. If the file hasn't been named you will be prompted for the file name and the location to save the file. After you perform the first save, this command acts as a quick save and saves the file to the name and location specified during the initial saving of the file.

Save Raw As

This command saves the raw editor data in the spreadsheet to a rw5 file and always prompts for file name and location to save.

Open CRD File

This command prompts for an existing coordinate file to set as the active coordinate file for the raw editor.

New CRD File

This command prompts for a new coordinate file to set as the active coordinate file for the raw editor. The coordinate data will be initialized as empty.

Save CRD File

This command saves the current coordinate data in the raw editor to the current coordinate file.

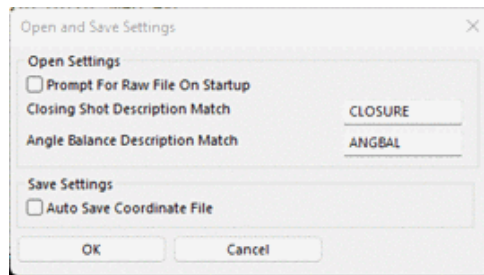
Save CRD As

This command saves the current coordinate data to a specified coordinate file name.

Note: The file format for a Coordinate File includes crd, crdb, cgc, mdb and zak

Open/Save Settings

This option allows for defining codes that represent the closing shot and angle balance shot of a traverse. These codes can be entered in the description of a point while in the field. When the rw5 is opened in the raw file editor, the measurement data containing the closing shot code will be set to a CL record and the measurement data containing the angle balance code will be set to an AB record. This allows for quick processing of the survey data and saves the time spent setting up the file for processing.



Import

These routines convert raw data from other formats into the current Carlson RW5 format. The converted raw data will be added to the end of any existing data in the editor. In many cases, the raw data file to import can be downloaded directly from the data collector or instrument using the *Data Collectors* command. The following supported formats (along with their standard file extension) are listed here.



Some Sample File Formats are listed at the end of this section.

C&G (.CGR; .RAW; .TXT; *)

CalTrans (.DMP)

Carlson (.RW5)

EFB (.RAW; .OBS) Electronic Field Book

Fieldbook (.FBK): From Softdesk, Land Development Desktop or Civil 3D. The import handles the following record types:

AD

AZ / AZM / AZIMUTH

B / BRG / BEARING

BEG / BEGIN

BS / BACKSITE / BACKSIGHT

C3

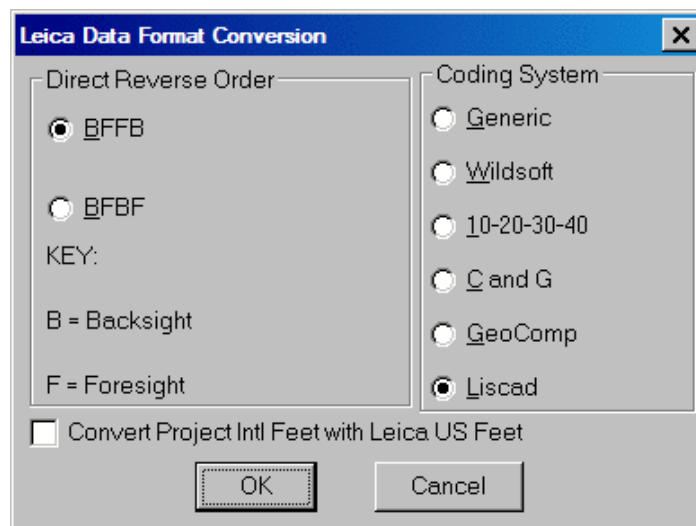
END
F1
FC1
NE / NEZ
PRISM
STN / STA / STATION
ZD

Geodimeter (.OBS; .RAW; job;*)

Horizon (.RAW)

LandXML (.XML): LandXML is the industry standard data format for exchanging project data. It can contain any number of different data types including surfaces besides raw measurements.

Leica (.GSI; .RAW; GRE): This reads the Leica raw file in Wildsoft, Liscad, 10-20-30-40, C&G, or GeoComp format. There are options to specify direct-reverse shot order if any and to convert from International Feet to Leica US Feet.



Maptech (.FLD)

MDL Laser (.CDS)

Nikon (.TRN; .RAW)

PC Cogo (.BAT)

SDMS (.prj;*)

SMI (.RAW)

Sokkia SDR (.SDR; .RAW;*)

StarNET (.DAT) The import handles the following StarNET record types: E - Elevation record

C - Coordinate record

B - Bearing / Azimuth record

M - Measurement record

SS - SideShot record

TB - Begin Traverse record

T - Traverse record

TE - End Traverse record

DV - 3D Distance Record (creates a slope distance/zenith angle record)

D - 2D Distance Record (creates a horizontal distance)

A - Horizontal Angle Record (creates an angle-only record)

V - Zenith Angle record (creates a zenith angle-only record)

When parsing these records, if a measurement, coordinate or azimuth has standard errors assigned to it, then standard error records are created in the RW5 file so none of that information is lost. The import also handles the following DOT commands:

.ORDER - Specifies point order (AtFromTo or FromAtTo) in the measurement record, and/or the order of NORTH/EAST or EAST/NORTH in control records.

.DELTA - Specifies whether the data is SlopeDist/Zenith or HDist/VDist. The default is SD/ZE.

.2D / .3D - Specifies the data format. Without this information the fields can be confused while parsing.

SurvCOGO (.RAW or .TXT)

SurvCE Archive (.SC5) When downloading a rw5 file from SurvCE using SurvCOM, there's an option to copy the rw5 file to a sc5 file as a read-only backup.

Survis (.RAW)

TDS (.RW5; RAW)

Topcon (raw;*)

Trimble (.dc)

3TA5 (.TXT)

Zeiss (.DAT)

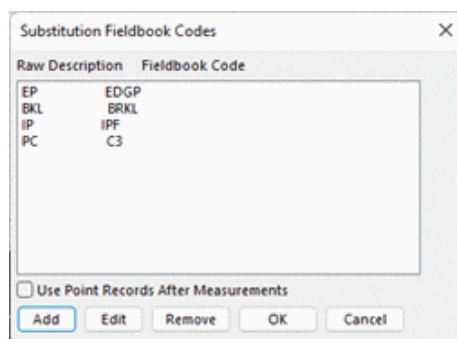
Export

These routines convert the Carlson raw data (.RW5) file to other formats. The following file formats are supported.

CalTrans (DMP)

dgDialog (.DGD)

Fieldbook (.FBK): This export routine provides an option to "Setup Fieldbook Codes". This allows the user to substitute the raw description contained in the rw5 file with the fieldbook code used in AutoDesk Land Desktop or Civil 3D.



FL DOT (.OBS)

GPS Data (.TXT;*)

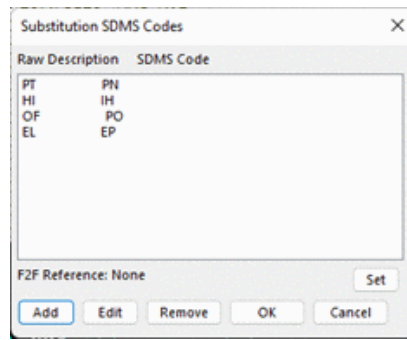
LandXML (.XML)

Leica (.GSI)

MOSS (.MOS)

RMGeo (.txt)

SDMS (.PRJ) This export routine provides an option to "Setup SDMS Codes". This allows the user to substitute the raw description contained in the rw5 file with the SDMS codes used in SDMS program.



SFN: This format is used in the Netherlands.

Sokkia (.SDR)

TDS (RW5;RAW)

Topobase (PO)

Trimble (JXL): JobXML format.

VA Dot (TOP)

WinCMM (.LSA)

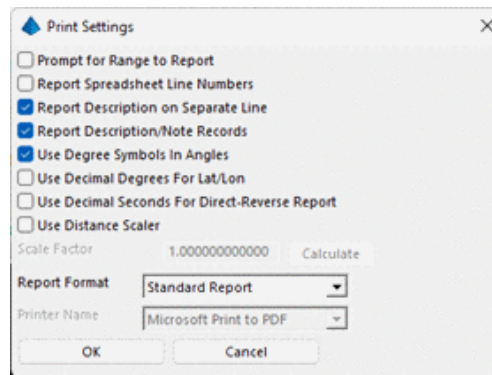
Report/Print

There are three types of reports: Raw Data, Coordinates and Summary. A sample of the raw data report is shown below. This report shows the data from the raw editor spreadsheet. The Coordinates report lists the point data (point number, northing, easting, elevation, description) from the current coordinate file. The summary report groups the traverse, sideshot and store point numbers along with a list of the setups and the shots from each setup. The summary report also groups the range of point numbers by date using the dates in the note records.

```
Raw File> c:\data\survey.rw5
CRD File> c:\data\survey.crd
Note
Survey Example
PntNo  Northing Easting  Elevation Desc
1      5000    5000    100      START
OcPt   BsPt     SetAzi
1
InstHgt RodHgt
5.32    6.0
OcPt   FsPt     HorzAngle SlopeDist ZenithAng Desc
TR 1   2      AR 268.5330 711.420  89.4050  P2
InstHgt RodHgt
5.43    6.0
OcPt   FsPt     HorzAngle SlopeDist ZenithAng Desc
TR 2   3      AR 262.5448 457.760  89.3236  P3
InstHgt RodHgt
5.4     6.0
OcPt   FsPt     HorzAngle SlopeDist ZenithAng Desc
TR 3   4      AR 208.5710 201.310  89.1803  P4
TR 4   5      AR 247.1657 497.120  88.5235  P5
TR 5   6      AR 277.4835 223.980  90.2926  P6
TR 6   7      AR 92.4113 233.880  90.2746  P7
InstHgt RodHgt
5.42    6.0
OcPt   FsPt     HorzAngle SlopeDist ZenithAng Desc
TR 7   8      AR 261.2756 387.250  91.4405  CLOSE
SS 7   19     AR 289.3456 112.450  91.3423  SS1
```

Report/Print Settings

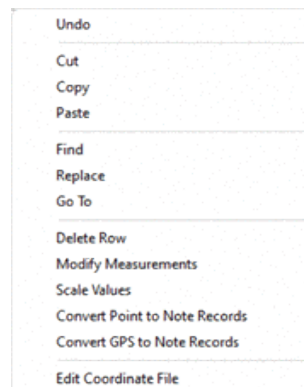
This dialog has settings for the report functions. The Use Distance Scaler allows for reporting distances in different units. For example, the survey distances could be in US Feet and then use a scale factor to report the distances in chains. For Report Format, the Report Formatter option allows for customized reports and exporting to Excel. The Spreadsheet Report option creates a report using the spreadsheet layout including the spreadsheet colors. The spreadsheet report goes directly to the specified printer.



Exit

Exits the raw file editor.

Edit Menu



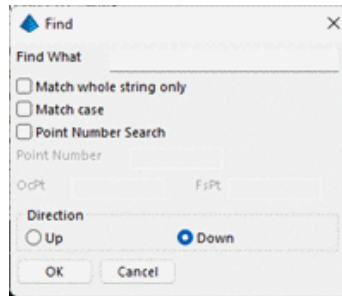
Undo: This command undoes the last data entry or the last copy, cut or delete command performed on keyboard entered data only. This will not undo a change to the Type or Code columns, nor a cut or copy command to a row.

Cut: Standard windows cut command. Removes data from editor and places it in the windows clipboard.

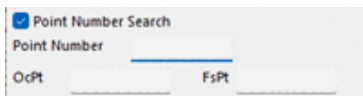
Copy: Standard windows copy command. Copies selected data to windows clipboard.

Paste: Pastes copied data to the row location selected

Find: Tool to search and find a particular word, letter, numeric value or a point number.

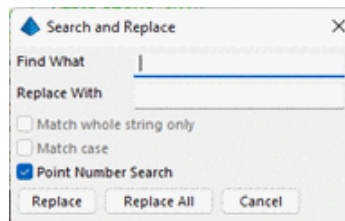


- **Match Whole String** filters search to the entire entered text string. For example, if you enter the Find What field searching for TREE, it will only find fields that match exactly TREE as opposed to just TR which will find any field beginning with TR ie. Tree, Transformer etc.
- **Match Case** only finds fields with the same capitalization setting
- **Point Number Search** searches specifically for a point record.



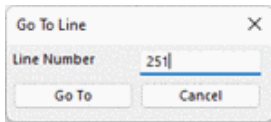
- The **Point Number Search** can be for a Point in the form of a Point (PT) Record, Occupied point or Foresight point.
- **Direction** allows for a Up or Down directional search from the active cell in the editor.

Replace: Tool to search and replace a particular word, letter, numeric value or point number.



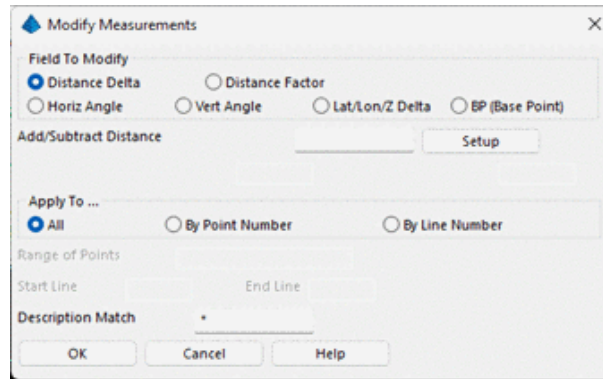
- **Match Whole String** filters search to the entire entered text string. For example, if you enter the Find What field searching for TREE, it will only find fields that match exactly TREE as opposed to just TR which will find any field beginning with TR ie. Tree, Transformer etc.
- **Match Case** only finds fields with the same capitalization setting
- **Point Number Search** searches specifically for a point record and replaces it with the one entered. This is useful when redundant measurements have been made to the same point and are to be part of a least squares network or in the case of mistaken point identification.

Go To: Tool to advance the focus of the active cell to a specified line number.



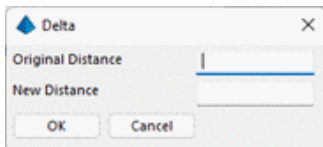
Delete Row: This command deletes the row containing the active cursor or cell. You can delete a row by placing the cursor in any of the cells in the row that you wish to delete, or by picking on the row number at the far left of the editor.

Modify Measurements: This option allows for a change in distance, horizontal angle, vertical angle or lat/lon by a specified amount for the entire file or for a specified point number or line number range.

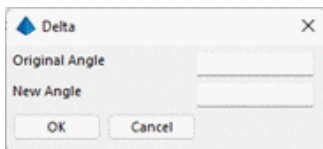


To modify a measurement, choose which field to modify, enter the change in distance, angle (dd.mmss format), Lat/Lon, or Coordinate values followed by how to apply the change(s).

- **Distance Delta** applies a distance to add or subtract to all measurements included in the "Apply to..." This option is useful in instances where a consistent distance value was mistakenly introduced, such as the wrong prism constant. There is a helpful Setup that allows the input of Original Distance and New Distance for calculating the delta.

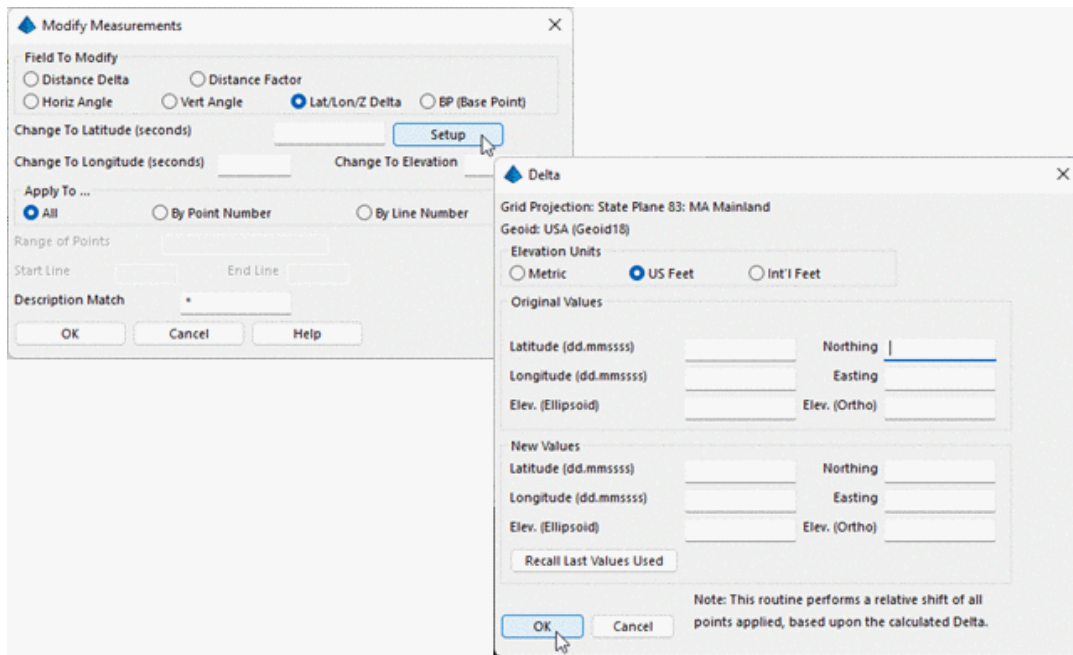


- **Distance Factor** method multiplies the distances by the specified value which can be used to convert distance units between feet and meters or to apply a scale factor. There is a helpful Setup that allows the input of Original Distance and New Distance for calculating the delta.
- **Horiz Angle** applies a horizontal angle to adjust all measurements included in the "Apply to...". This option is useful in instances where a mistaken angle was applied to multiple measurements such as an incorrect backsight. There is a helpful Setup that allows the input of Original Angle and New Angle for calculating the delta.



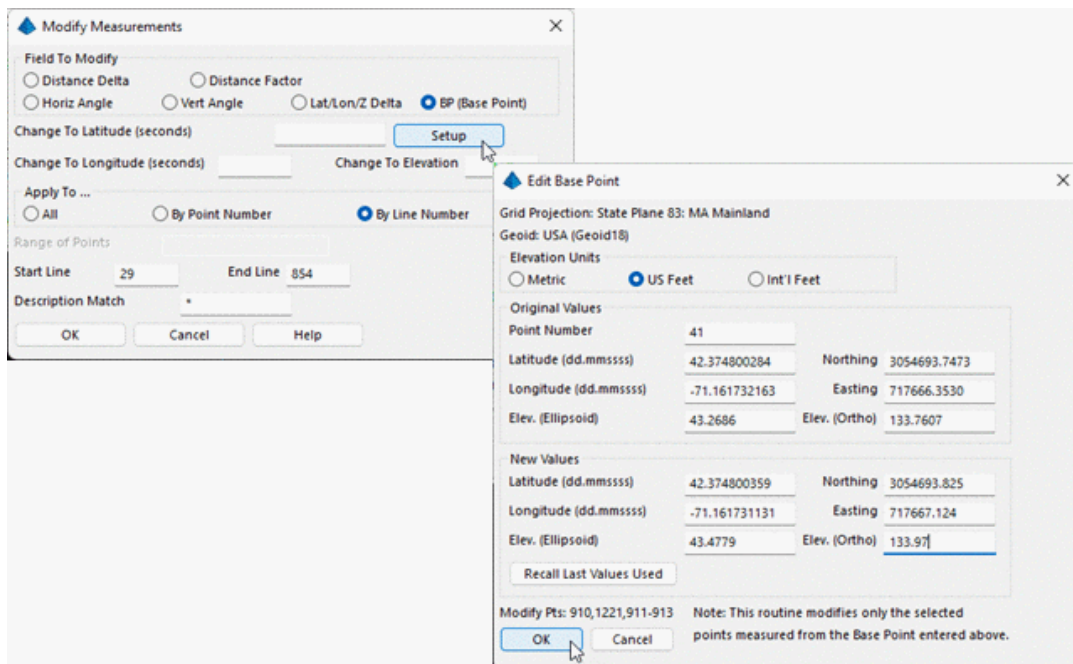
- **Vert Angle** applies a vertical angle to adjust all measurements included in the "Apply to...". Similar to Horiz. Angle, There is a helpful Setup that allows the input of Original Angle and New Angle for calculating the delta.

- **Lat/Lon/Z Delta** is used to adjust GPS records in cases where Delta "shift" is desired for all or a group of points.



The points specified in the "Apply to..." selection are moved a delta latitude, longitude, elevation or a combination of all three. The **Setup** option supplies an easy to use optional entry of direct coordinate values; Northing, Easting and Orthometric Elevation. The delta is calculated in the Setup dialog box and populates those values in the Delta change values seen in the first dialog box. This routine shifts all selected points the same value regardless of a specific base point. It is useful in cases where the entire GPS survey needs to be altered in the same direction.

- **BP (Base Point)** is used to adjust GPS records in cases where Delta "shift" is desired for a group of points measured from a specific base point.

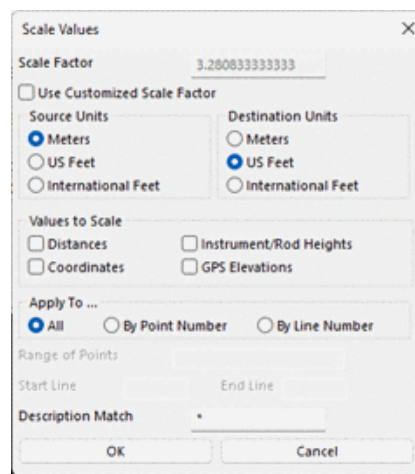


The points specified in the "Apply to..." selection are moved a delta latitude, longitude, elevation or a combination of all three. The **Setup** option supplies an easy to use entry where you can specify a base point that was used to measure points from. The routine recalls the coordinates of the base point and identifies all points measured from that base point. New values can be entered into the appropriate spaces, either Latitude, Longitude, Ellipsoid height or Northing, Easting, Orthometric height. The delta is calculated in the Setup dialog box and populates those values in the Delta change values seen in the first dialog box. This routine shifts all points measured from the specified base point but that selection can be further modified through the "Apply to..." selection.

The **Recall Last Values** Used is convenient if the base point is used multiple times within the RW5 file and the same shift is to be applied.

Apply to... controls how to apply the modification. If **All** is selected, the change will be applied to all records in the specified field. If **By Point Number** is chosen, enter the point number or range of numbers in the Range of Points field. If by **Line Number** is chosen, define the area for the change by specifying the Starting and Ending line.

Scale Values: This function scales selected values which by a specified scale factor.



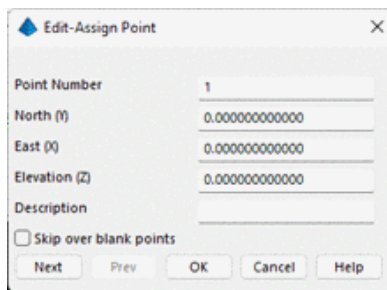
A common use of this feature is to scale measured values from meters to feet or visa versa. The Values to Scale options allow filtering for particular values to be scaled. The Apply To... section controls which records within the RW5 are to be scaled.

Convert Points To Notes Records: This function converts point (PT) records to note (DS) records. This leaves the information of the point coordinates in the rw5 file as display only. The points marked as notes (DS) records will not be used when processing the data and will not be stored in the Coordinate File upon completion. When this option is selected, you will be prompted to enter all or a range of point numbers to be included in the conversion.

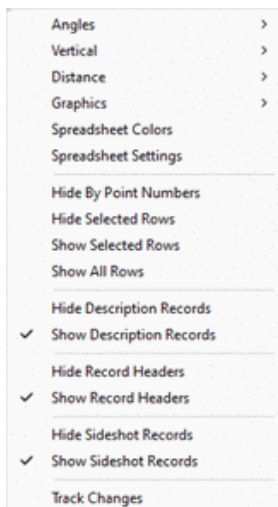
The point data in the DS records can be converted back to PT records by picking the Code field in the spreadsheet and switching DS to PT.

Convert GPS to Note Records: This function converts GPS point (GPS) records to note (DS) records. This leaves the coordinates of the GPS point, *Latitude, Longitude, Ellipsoid* in the rw5 file as display only. The GPS points marked as notes (DS) records will not be used when processing the data and will not be stored in the Coordinate File upon completion. When this option is selected, you will be prompted to enter all or a range of point numbers to be included in the conversion.

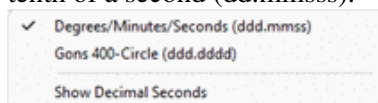
Edit Coordinate File: This option allows for editing and/or listing of the coordinate data in the active coordinate file. The active coordinate file is displayed in the Header of the raw data editor. This routine brings up the edit point dialog and allows editing of the points one at a time.



Display Menu



Angles: This option chooses the angle format between degrees/minutes/seconds (dd.mmss) and Gons-400 decimal degree circle (dd.dddd). This setting applies to the angles in the spreadsheet editor as well as the angle format for reports. There is also a separate setting to Show Decimal Seconds which use edit and process angles to the nearest tenth of a second (dd.mmsss).

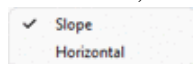


Vertical: The options contained in this menu allow for specifying the type of vertical measurement information you will input or is contained in the rw5 file. The Vertical Angle selection assumes the barrel or scope of the instrument is level when reading 0 (zero). With this setting, the vertical component of a measurement record will have a header of VertAng. The Zenith Angle selection, most commonly used, assumes the barrel/scope to be level when reading 90. Using this setting results in a header of ZenithAng. Elevation difference displays the elevation difference between the occupied and foresight points. If the Distance option is specified as Slope, this elevation difference will be used to calculate the horizontal distance of the measurement. The header for this record is ElevDiff. The None selection assumes all distances are horizontal distances and removes the vertical component for a measurement from the editor. Switching modes can be performed at any time.

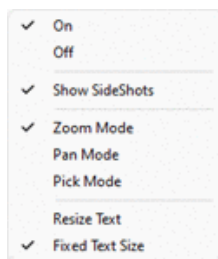


Distance: This option controls the display of either Slope or Horizontal Distances. Changing the display results in the distance data adjusting to reflect the correct value for the selection made. The Vertical data, VertAng, ZenithAng

or VertDiff, is used to convert the distance value when changing this display option.



Graphics: The Raw Data Editor uses an optional graphics window to display the points and traverse lines in real time. As data is entered or edited, the graphics window will be updated to show the configuration or new configuration of the traverse, sideshots or vectors.



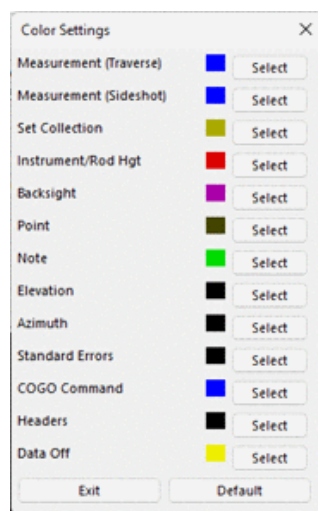
The overall graphics window is toggled On or Off and is displayed below the raw data records.

- **Show Sidehots** toggles the display of sideshots On or Off. When a cell is selected, the traverse or sideshot line in the display window will change to the color yellow for a graphical reference.

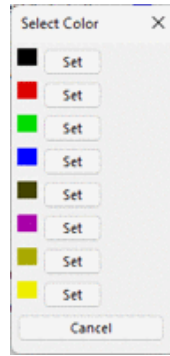
Zoom, Pan or Pick Mode sets the behavior of the left click mouse button. It should be noted that if your mouse is configured to use the middle button for Pan and it Scrolls for Zoom, the Zoom and Pan options are redundant.

- **Zoom Mode:** Within the graphics window, real time zoom is available. To zoom in press and hold the left mouse button and drag in the direction of the + symbol. To zoom out, press and hold the left button and drag in the direction of the - symbol.
- **Pan Mode:** Real time pan is available within the graphics window. To pan, set the graphics window to pan mode, then press and hold the left mouse button and then drag to desired position.
- **Pick Mode,** you can select a point from the graphics window below the data fields and the cursor will go to that point or measurement record in the data field.
- **Resize Text:** With this option on the text becomes smaller/larger in the view when you zoom in/out.
- **Fixed Text Size:** With this option on, the text stays a fixed size while zooming in and out.

Spreadsheet Colors: This option allows for the assignment of colors to record types.



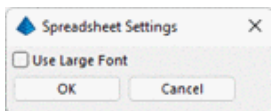
From the Color Settings dialog select the record to edit by clicking on the **Select** button next to the desired record. The color slide beside the select button shows the current setting for the record.



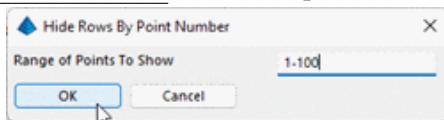
To change the color for a data record, select the **Set** button next to the desired color.

The Default button will return all colors back to the installed defaults.

Spreadsheet Settings: This command has an option to use a larger font for the spreadsheet for easier viewing.



Hide by Point Numbers: This option hides point records by selecting the points that are to remain displayed.



Hide/Show Selected Rows: This option allows for hiding single or multiple rows. This could be used to prevent crucial information from being accidentally altered during editing of data or data entry. Hiding a record does not exclude it from processing. To hide records or rows, select the column at the far left of the editor and select **Hide Selected Rows**. Multiple rows of data can be selected by selecting the first row of data to hide then while holding down the shift key on the keyboard, select the last row to hide. When a row or rows of data are hidden, the row numbers will remain as originally assigned. Hidden rows will not display a line number. To show previously hidden rows, select the row above and below the hidden rows and select **Show Selected Row**.

Show All Rows: turns on all rows in the raw file. This resets the raw file to its' original display.

Hide/Show Description Records: This option hides all description (DS) records. Show Description Records will turn them back on.

Hide/Show Record Headers: This option controls the display of the headers just above each record, such as InstHT, RodHT, Latitude, Longitude etc.

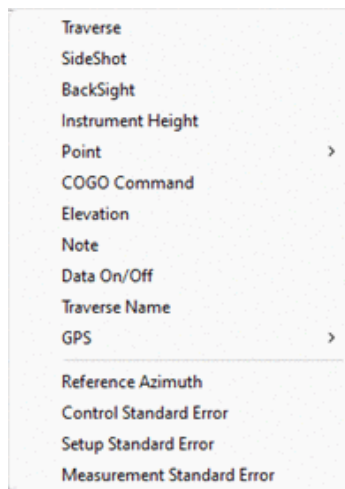
Hide/Show Sideshot Records: This option controls the display of the sideshot records themselves. This is useful in isolating traverse measurements from sideshot measurements.

Track Changes: When toggled on, this option tracks changes in the raw file by adding a note (DS) record just below the edited line. For example if a rod height was altered, a new record would be displayed in the RW5. Example:

185		InstHt	RodHt				
186	HI	5.150	6.0				
187	DS	Old RodHt: 5.100					
188		OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng
189	SS	5	56	AR	225.2329	19.871	92.5323
190	SS		57	AR	118.713	45.05	37.11

Add Menu

The add menu contains options for adding specific records and functions to the raw data editor.



Each element will be added in the raw file editor at the location of the cursor. This is important especially in situations such as adding an Instrument or Rod Height.

Traverse: Adds a traverse record (TR) to the spreadsheet editor. The new record will be insert above the row that contains the active cell unless this row is the last row in the file. If so, you will be prompted to insert above or below the current row.

SideShot: Adds a sideshot record (SS) to the spreadsheet editor. The new record will be insert above the row that contains the active cell unless this row is the last row in the file. If so, you will be prompted to insert above or below the current row.

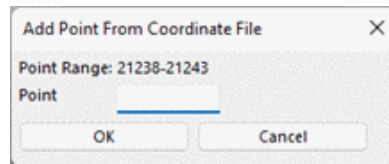
Backsight: Adds a backsight (BK) to the spreadsheet editor. The new record will be insert above the row that contains the active cell unless this row is the last row in the file. If so, you will be prompted to insert above or below the current row.

Instrument Height: Adds an instrument height (HI) record to the editor. This record contains both the instrument and rod height setting.

Point: Adds a point (PT) record to the editor. Provides options to either add a Blank Point Record or Import From Coordinate File.



Inserting a blank record allows for manual input to define the coordinates for the point. Import From Coordinate File imports the coordinate values from an existing point or range of points contained in the coordinate file. Enter the point number or range of points and select OK. The points will be read into the rw5 file at the top of the file.



COGO Command: Adds COGO Command (CC) record with a field to specify the command (Translate, Rotate, Scale or Align) and a field for entering the parameters. The COGO commands are executed in sequence as the rw5 file is processed from top to bottom by any of the process methods in the Process menu. The COGO commands are all transformation commands that are applied to the points in the current coordinate file. The following list is the syntax of the COGO commands:

- Translate: Range Dx Dy Dz Process_Zero_Z
- Rotate: Range Angle Base_Y Base_X
- Scale: Range Scale Base_Y Base_X Use_Z
- Align: Range From1 To1 From2 To2

All the parameters are entered into one spreadsheet cell next to the COGO function. The parameters use space separators. The following list is the parameter definitions:

Range: point numbers

Dx: delta easting (X)

Dy: delta northing (Y)

Dz: delta elevation (Z)

Process_Zero_Z: toggle for whether to process points with elevation of zero (0=No, 1=Yes)

Angle: rotation angle in dd.mmss format

Base_Y: base point northing

Base_X: base point easting

Scale: scale factor

Use_Z: toggle for whether to scale the elevations (0=No, 1=Yes)

From1: point number of first source point

To1: point number of first destination point

From2: point number of second source point

To2: point number of second destination point

For example, to translate points 1-10 by a delta Z of 6.0 while filtering out zero elevation points, set the parameters for the COGO Translate record as "1-10 0 0 6.0 0".

Elevation: Adds an elevation (EL) record to the editor. The new record will be insert above the row that contains the active cell unless this row is the last row in the file. If so, you will be prompted to insert above or below the current row.

Note: Adds a note (DS) record to the editor. Note records are for information display and do not affect processing except for three particular notes which are:

- Elevation: 2D
- Elevation: 3D
- Tilt Sensor Offset

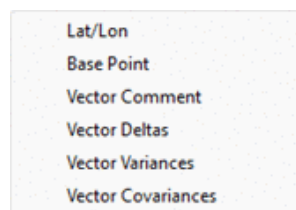
- **Elevations 2D and 3D** are special notes that set the elevation mode for processing for the records that follow the note. The raw editor starts in 3D mode. The "Elevation: 2D" note will switch processing to 2D mode and the "Elevation: 3D" note will switch the mode back to 3D. In 2D mode, the processing will not set the elevations in the coordinate file.
- **Tilt Sensor Offsets** are notes placed on GNSS receiver locations when the Tilt Sensor option is on, usually in cases where the rod is not plumb. The note record indicates the offset values between the phase center of the receiver and the bottom of the pole. These values are applied during processing.

1436		▼	Note					
1437	DS	▼	GSRD,TXD4.127,TYD-5.302,THD1.776,TDTCarlson_BRx7					
1438	DS	▼	GSAL,TEL0.025,TNL0.794,TZL0.047,T2D0.794,TTL0.049					
1439	DS	▼	GT,PN21114,SW2218,ST312264800,EW2218,ET312264800					
1440	DS	▼	HRMS:0.055, VRMS:0.082, STATUS:FIXED, SATS:26, AGE:2.0, PDOP:0.893, HDOP:0.465,					

Data On/Off: Adds a data on/off (DO) record to the editor. This record toggles the raw data between processing on and off modes. The raw data starts in processing on mode. Working from top to down, when a DO record is reached, the processing mode is turned off. Then next DO record will turn processing back on, and so on. Data records that are in processing off mode and skipped when running the routines in the Process menu.

Traverse Name: Adds a traverse name (Name) to the editor. The new record will be insert above the row that contains the active cell unless this row is the last row in the file. If so, you will be prompted to insert above or below the current row.

GPS: Adds a GPS record to the editor. The new record will be insert above the row that contains the active cell unless this row is the last row in the file. If so, you will be prompted to insert above or below the current row.



Reference Azimuth: Applies to SurvNET, the optional Network Least Squares analysis and adjustment routine.

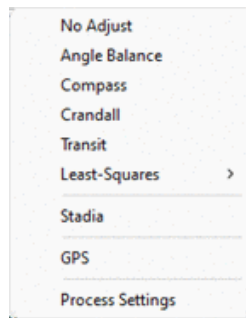
Control Standard Error: Applies to SurvNET, the optional Network Least Squares analysis and adjustment routine.

Setup Standard Error: Applies to SurvNET, the optional Network Least Squares analysis and adjustment routine.

Measurement Standard Error: Applies to SurvNET, the optional Network Least Squares analysis and adjustment routine.

Process (Compute Pts) Menu

This menu contains tools to process raw data by various methods. The calculated coordinates, and notes if specified, are stored to the active specified coordinate file. The coordinate file can be specified using *Set Coordinate File*, under the Points pulldown within the drawing screen, or from the Tools menu of the editor, discussed later in this section.



There are several adjustment options:

No Adjust

Angle Balance

Compass Rule

Crandall Rule

Transit Rule

Least-Squares - *Note, this is a legacy adjustment routine that has been superseded by SurvNET, a comprehensive least squares adjustment program that processes EDM traverse measurements, GNSS vectors and Differential Leveling.*

Stadia

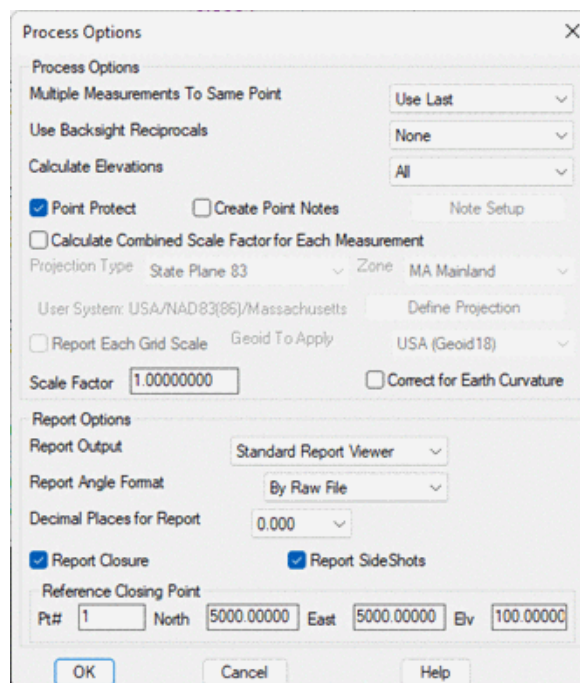
GPS

Process Settings

No Adjust, Angle Balance, Compass, Crandall, and Transit processing contain options that are accessed through a separate dialog box when each option is first selected. Many of these options are identical. The options in common are shown below:

Common Process Options

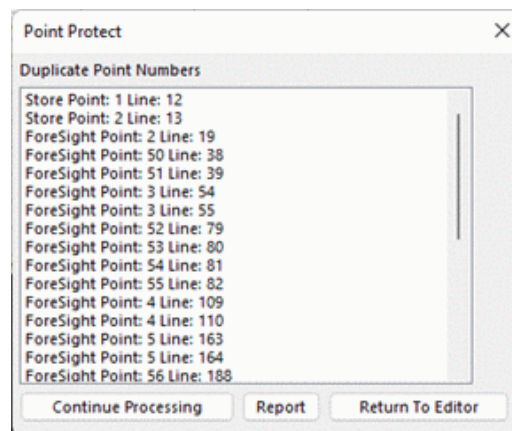
These are processing options which are common to No Adjust, Angle Balance, Compass Rule, Crandall Rule, and Transit rule.



- **Multiple Measurements To Same Point:** This option sets the method of how to handle multiple measurements to the same point (identical point name). There are three available options, Use Last, Average or Use First. **Use First** uses the first measurement to calculate the position of the point. **Average** uses the average of all the measurements for the position, and **Use Last** takes the last measurement to the to calculate the position.
- **Use Backsight Reciprocals:**This setting applies to backsight measurements that are not part of a complete direct/reverse set. Backsight measurements that are part of a direct/reverse set use the direct/reverse settings under Process Settings to reduce the measurements.

A foresight to point 15 from a setup on 14, followed by a backsight from 15 to 14, makes a pair of "reciprocal" measurements. The backsight "reciprocal" measurement can be ignored for its impact on recalculating the occupied point (None Option), or the elevation of component of the reciprocal measurements can be averaged (Average Elevation option), or both the elevation and distance can be averaged (Average Elev & Dist) to recalculate the setup (occupied point) coordinates.

- **Calculate Elevations:** This option determines whether the elevations of the points will be calculated and written to the coordinate file. Options of whether to calculate All elevations or just the Sideshots Only are provided.
- **Point Protect:** This option will check the coordinate (.CRD) file for existing point data before processing. If the foresight point number for any traverse or sideshot record already is a stored coordinate in the coordinate (.CRD) file, then the program shows a list of conflicting point numbers. You can either continue processing and overwrite the coordinate (.CRD) file coordinates with the calculated raw file coordinates or cancel the processing to go back to the editor to change foresight numbers.

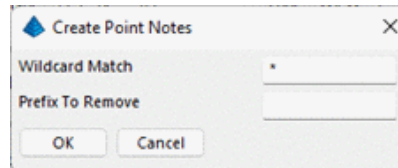


A report of the conflicting point numbers can be generated to the standard report viewer in Carlson by selecting the Report option on the Point Protect dialog box. From the report viewer, the report can then be printed, sent to the screen or saved to a file.

Raw File: C:\Carlson Work Files\Workshops and Webinars\Workshop Data Files\SurvNET\Ground Traverse\Loop 1.RW5
 CRD File: C:\Carlson Work Files\Product Development\Test Files\2023 Alpha\DWG\Empty Drawing.crd

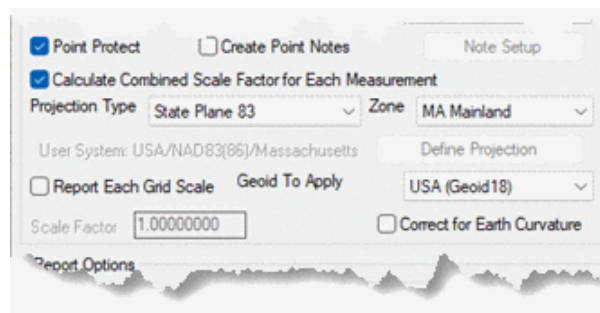
Store Point: 1 Line: 12
 Store Point: 2 Line: 13
 ForeSight Point: 2 Line: 19
 ForeSight Point: 50 Line: 38
 ForeSight Point: 51 Line: 39
 ForeSight Point: 3 Line: 54
 ForeSight Point: 3 Line: 55
 ForeSight Point: 52 Line: 79
 ForeSight Point: 53 Line: 80
 ForeSight Point: 54 Line: 81
 ForeSight Point: 55 Line: 82

Create Point Notes: This option will generate a note (.NOT) file named after the coordinate file. The note file contains additional descriptions for points. With this option active, the text from all note records (DS records) will be stored to the note file for the foresight point number preceding the note records. When this option is active, the Note Setup button brings up a dialog to set a Wildcard Match which filters the notes in the raw file



and only stores the notes that match to the note file.

- The **Prefix To Remove** option removes the specified string from the start of the note in the raw file when storing to the note file.
- **Calculate Combined Scale Factor at Each Setup:** This option will calculate a scale factor for each TR and SS record.



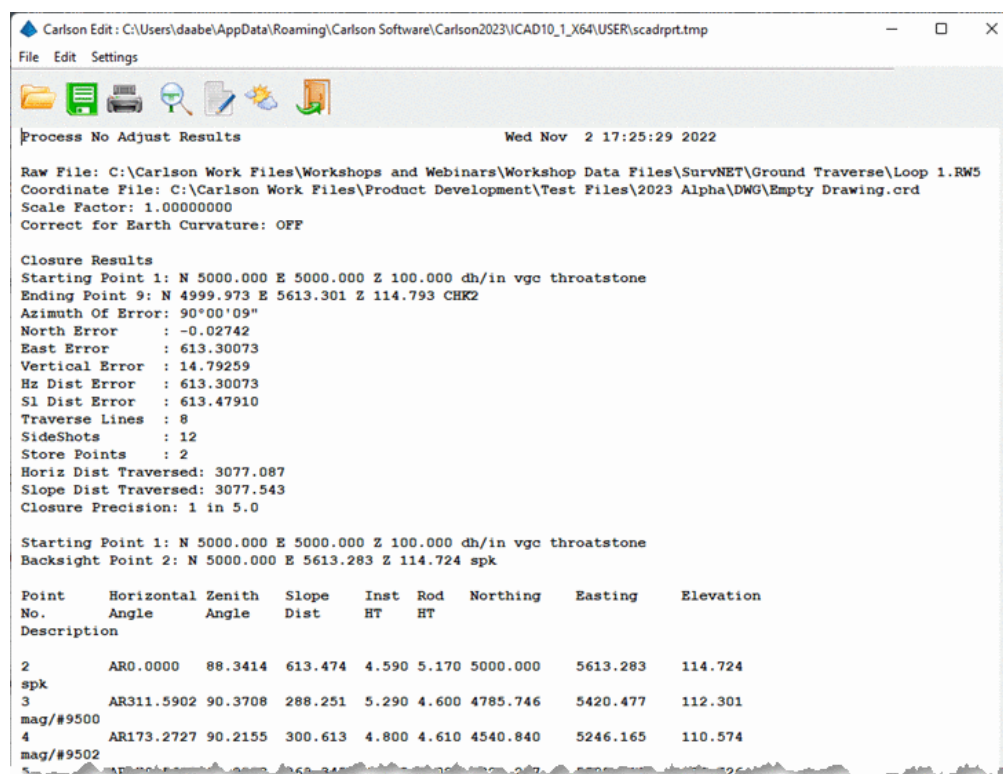
This scale factor is calculated as the average of the scale factors at the occupied and foresights points. At these points the scale factor is calculated as the projection grid factor multiplied by the elevation factor which is the earth radius divided by the elevation plus the earth radius [$SF = \text{Grid Factor} * (\text{Earth Radius} / (\text{Elevation} + \text{Earth Radius}))$]. In order to calculate these projection grid factors, the traverse coordinates must be in grid coordinates. When this option is desired, set the projection and zone to use. The elevation for the scale factor can be adjusted by the geoid height using the geoid specified in the **Geoid To Apply** list. The geoid height is added to the point elevation to adjust the elevation value used in the scale factor equation. The geoid surface files are not installed by default due to the large size of these files but will be loaded on demand as they are selected. To install the geoids manually, go to the Carlson Software web page www.carlsonsw.com and download the Geoid Grid Files from the Support->Downloads section.

- **Report Each Grid Scale:** This option becomes available if the Calculate State Plane Factor at Each Setup has been selected. With this option on, the scale factor at each point will be shown in the process results report.
- **Scale Factor:** displays the calculated combined scale factor. This can be entered directly if known.
- **Correct for Earth Curvature:** This option adjusts the calculated points for the effect of the Earth's curvature. Typically this adjustment is small and adjusts the elevation more than the horizontal values.

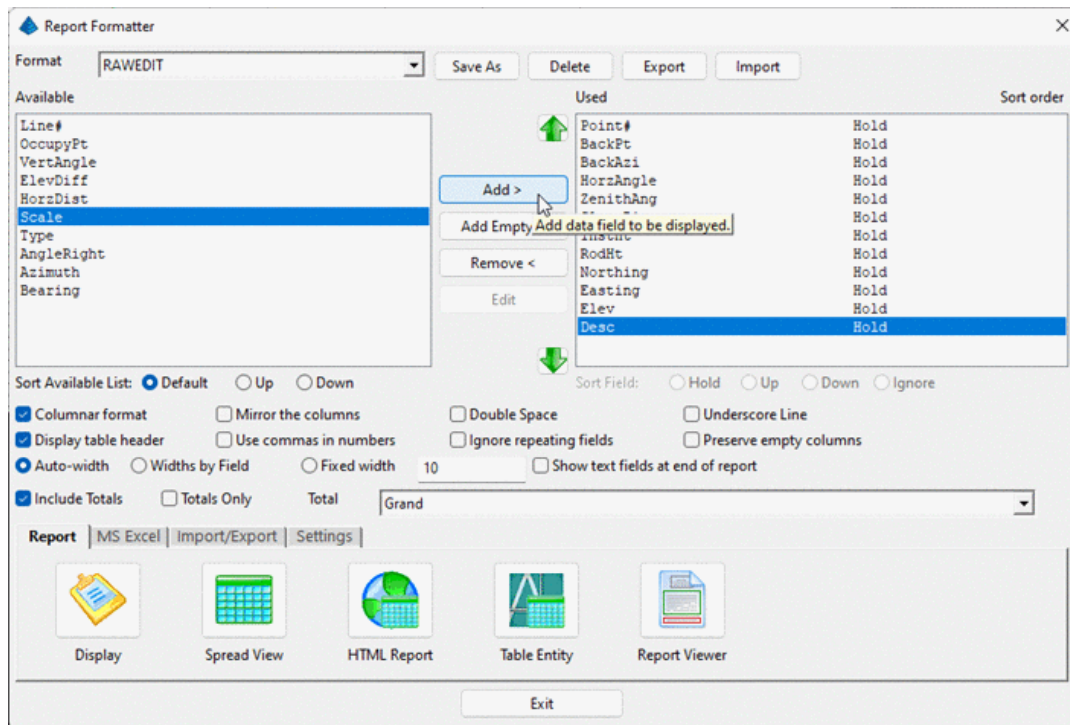
Report Options

- **Report Output:** There are three report output options contained in the raw editor, the *Standard Report Viewer*, the *Custom Report Formatter* and the *Tabular Report Viewer*. Each is documented below.

- **Report Angle Format:** This option controls the angle format displayed on the process results report. The option of **By Raw File** will display the angles in the format that is contained in the raw file. The **Bearing** option will display the angle in a bearing format. The **Azimuth** option will display the azimuth of the measurement and the **Angle Right** option will display the angle right measurement of the observation.
- **Decimal Places for Report:** This option controls the number of decimal places for the reported data.
- **Report Closure:** This option determines whether the closure report will be displayed after processing. If processing a survey where the traverse has not been closed, turn this toggle off for quick processing. To use this option, you will need to supply a **Reference Closing Point**. This is an optional field for entering the coordinates to compare the ending traverse point with. This reference closing point is used to calculate the closure. Without using this option the program will by default use the starting coordinate as the reference closing point.
- **Report Sideshots:** Controls whether the sideshot data is shown in the process report.
- The **Standard Report Viewer** is the default report viewer throughout the program. Any routine that generates a report has this option and the data contained in the report depends upon the routine executed. The report viewer is also a text editor. It allows for addition and deletion of text in order to customize the report for printing or for saving to a particular format for a file. Options to print, send to the screen in the drawing window as text or save to a file are available.



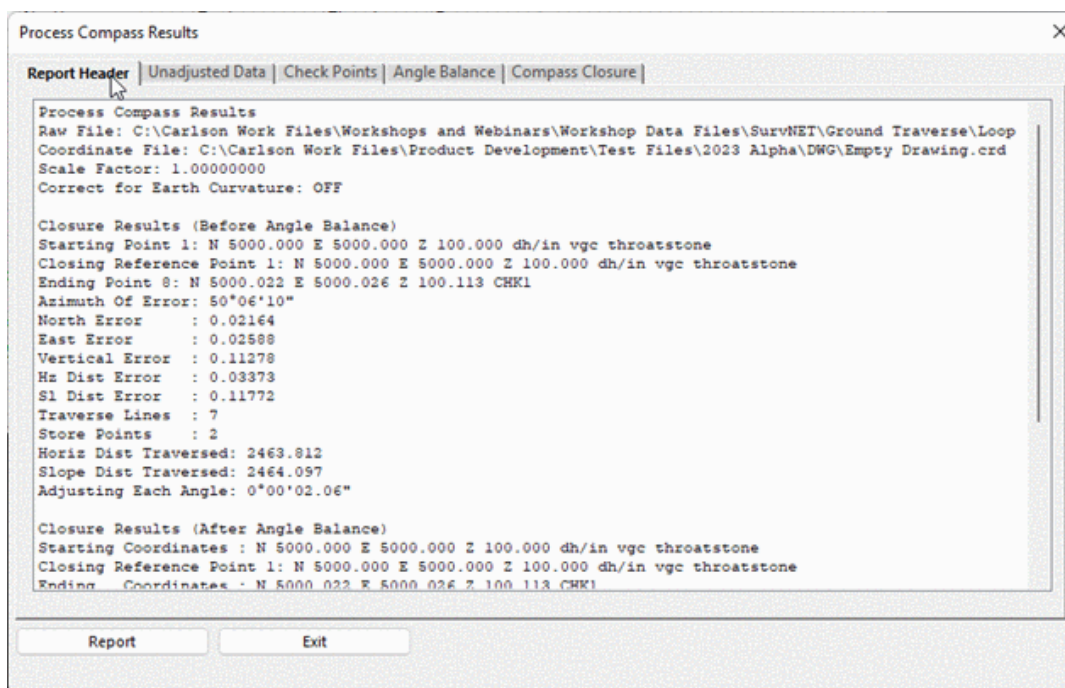
- The **Custom Report Formatter** allows for customization of the process results by selecting the fields and the layout of the fields to display. The settings can be saved to a format name and recalled when needed. Options to Delete, Export and Import saved Formats are also available.



To create a report, select data from the Available list and then select the Add button. This will populate the Used field with the selected data. Standard window selection methods can be used when selecting the data to report. Holding the ctrl key while selecting data allows for making random selections. Holding the shift key while selecting data will select the first item picked, last item picked and all items between.

For more information on the Report Formatter Dialog box, see the Help menu in the Introduction portion of the User Manual.

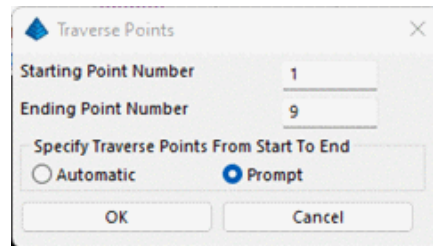
- The **Tabular Report Viewer** displays a report viewer consisting of tabs. Each tab organizes and displays different data depending upon the process options chosen. The process results using the No Adjust method results in three tabs the Report Header, Unadjusted Data and the Store Points tabs. Each of these tabs display different information which corresponds to the tab title. Using an adjustment method results in five tabs. In addition to the three listed above, an Angle Balance and Compass Closure tab is added.



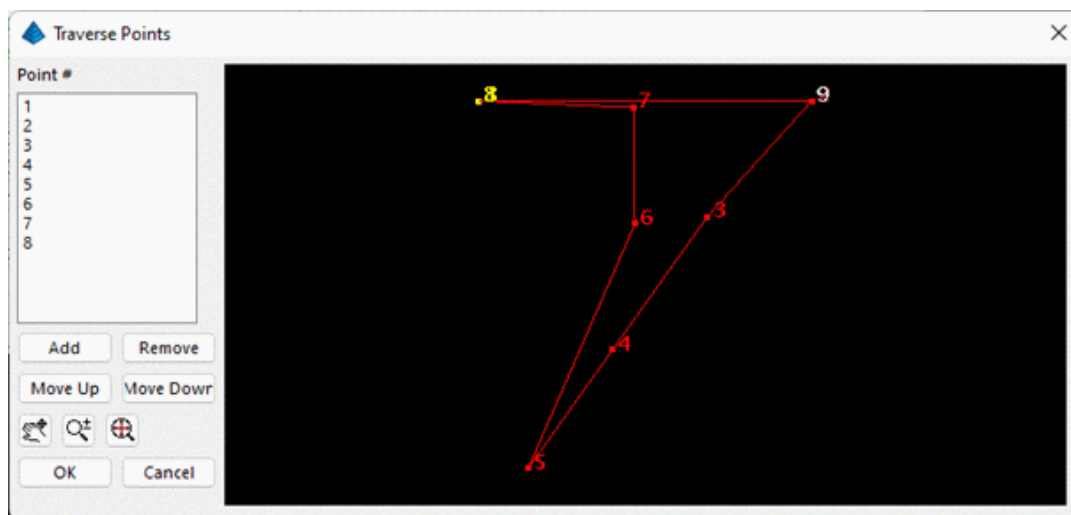
From the Tabular Report Viewer, the Standard Report Viewer can be switched to by pressing the **Report** option at the bottom of the dialog. This is useful when wishing to combine all tabs into one report for printing or saving to a file. An example of a Tabular Report for a compass rule adjustment is shown above.

Processing Methods

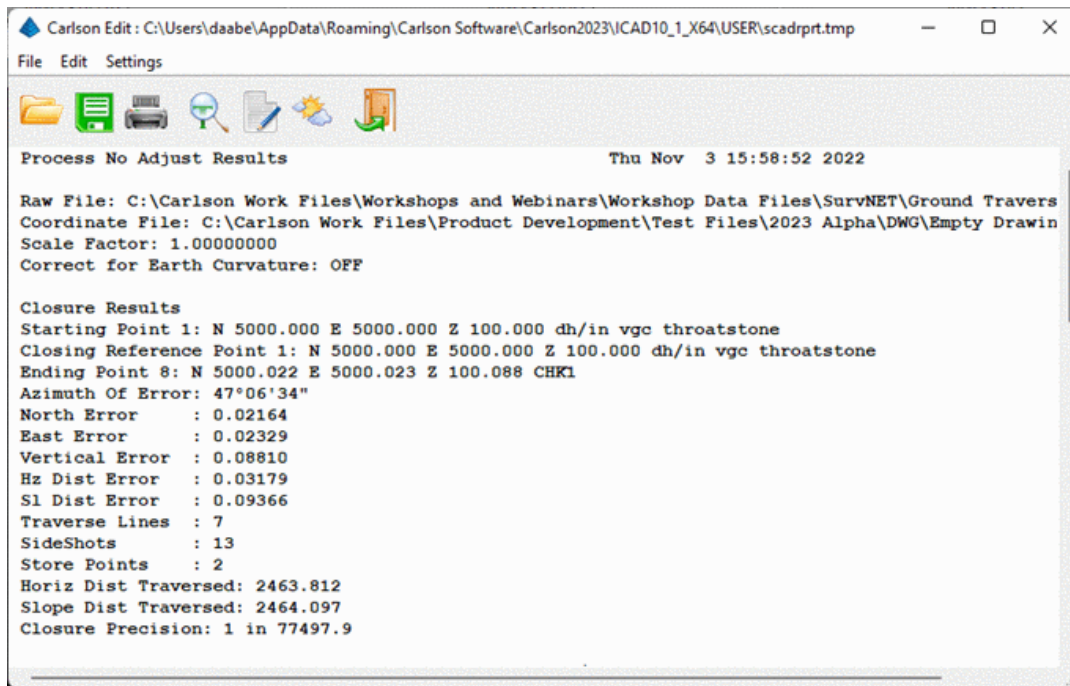
No Adjust: No Adjust means that no angle balance or traverse adjustment will be applied. Options are specified in the Process Options dialog. After picking OK, and Report Closure was NOT selected, for a process option, the program process all data records. If the Report Closure option was selected, a Traverse Points dialog box appears for entering the starting and ending point numbers.



The program reads the raw file to set the defaults for these point numbers which are used to calculate the closure. If you set the option to **Automatic**, the routine will continue to process using the numbers seen in the dialog box. If you choose the option Prompt, you will see the following graphic dialog box:



This dialog box depicts the traverse with both point numbers and a graphic display. You can examine the traverse, rearrange the order, add or remove points with the buttons on the left side of the dialog box. Once you are satisfied with the order of the traverse, click OK and the routine will calculate the results. The program starts processing the raw file from the top record down. The result is displayed in the Standard Report Viewer which can be saved, printed or drawn in the Drawing editor.

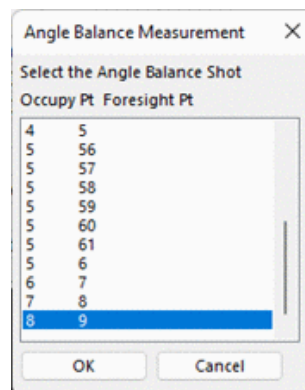


Note: The difference between the ending point and the reference closing point is the closure error and the sum of the traverse distances from the starting to the ending point is used as the total distance traversed.

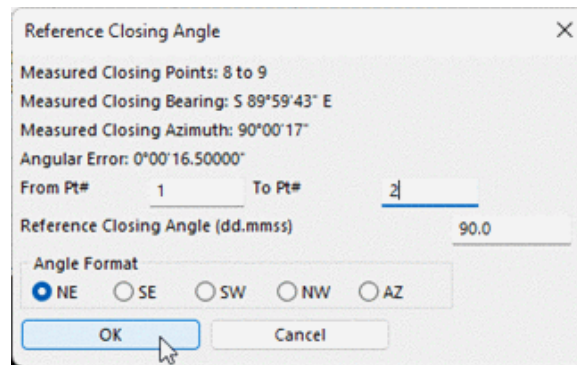
Angle Balance: This process method applies an angle balance to the traverse lines when calculating the coordinates. The angle balance takes the angular error divided by the number of traverse lines and adjusts the angle of each traverse line by this amount. The angular error is the difference between the angle balance shot and a reference angle. The angle balance shot is specified as a type AB or CL+AB record in the raw file. If no AB record is found in the raw file, then the program will prompt for which traverse shot to use as the angle balance shot. The angle from the angle balance shot is calculated as the angle from the occupied point to the foresight point. The reference angle can be specified as a bearing, azimuth or by two point numbers in the dialog shown.

The process is identical to the No Adjust method but with one or two additional prompts needed for selecting the angle balance shots.

Note: If the raw file has the Closing Point and Angle Balance Fields selected for the appropriate measurement records, these additional prompts will not be necessary.



Select the correct pair of point numbers for the Angle Balance Measurement. This is often the last leg of the traverse relocating the initial backsight and setup points.



Enter the Reference Closing Angle. This can be entered by point numbers or entering the angle. This is often the original backsight and setup points.

```

9          1123 88.3848  613.446  5.000  4.810  4999.973  5013.298  114.768
CHK2

Angle Balance
Angular Error: 0°00'16.50" for 8 traverse sides
Adjusting Each Angle: 0°00'02.06"
Closure Results (After Angle Balance)
Starting Coordinates : N 5000.000 E 5000.000 Z 100.000 dh/in vgc throatstone
Closing Reference Point 1: N 5000.000 E 5000.000 Z 100.000 dh/in vgc throatstone
Ending Coordinates : N 5000.002 E 5000.010 Z 100.113 CHK1

Azimuth Of Error: 77°26'25"
North Error      : 0.00222
East Error       : 0.00997
Vertical Error   : 0.11278
Hz Dist Error    : 0.01021
Sl Dist Error    : 0.11324
Traverse Lines  : 7
SideShots
Total Hz Dist Traversed: 2463.81248
Total Sl Dist Traversed: 2464.09650

Closure Precision: 1 in 241320.1

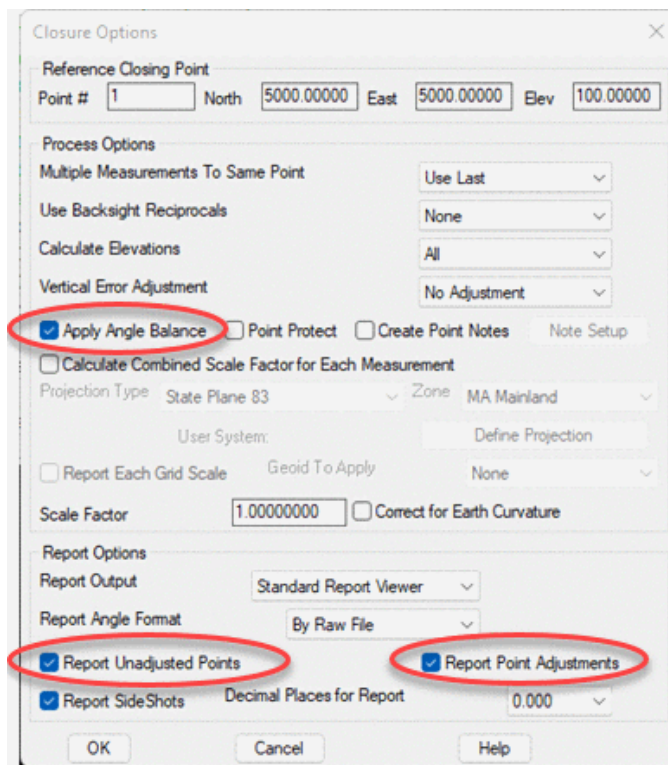
Point      Horizontal Zenith Slope  Inst Rod  Northing  Easting  Elevation
No.        Angle  Dist

```

The angle balance report shows the unadjusted points, the unadjusted closure, the angular error, the adjusted points and then the adjusted closure. Typically but not always, applying the angle balance correction will improve the traverse closure.

Compass, Crandall, Transit: These processing methods apply the selected rule to the traverse lines when calculating the coordinates. After adjusting the traverse, the sideshots are also recalculated. The closure error is calculated as the difference between the closing shot and a reference point. The closing shot is specified as a type CL or CL+AB record in the raw file. If no CL record is found in the raw file, then the program will prompt for which traverse shot to use as the closing shot. The foresight point is used as the closing coordinate. The reference point can be specified by point number or by entering the northing, easting and elevation. The process results report shows the unadjusted points, closure error, adjustments to each traverse point and adjusted point.

Compass Rule: This common closure method performs a typical linear adjustment of the traverse proportional to the length of each leg, weighting both distances and angles. Beyond the *common options*, the Compass Rule Adjustment dialog box presents a few additional options:



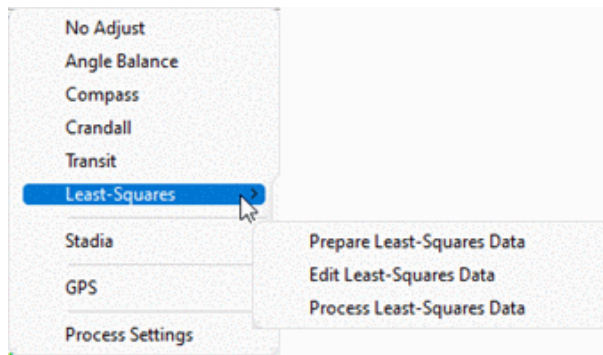
- **Apply Angle Balance** matches the Angle Balance method described above. This option performs an angle balance as part of the adjustment which negates the need to perform a separate angle balance.
- **Report Unadjusted Points** provides the option to report the original unadjusted points along with the adjusted positions.
- **Report Point Adjustments** provides the option to report the amount each individual point was adjusted.

Once these options have been selected, proceed with the traverse adjustment as shown above in the *Angle Balance* method.

Crandall Rule adjustment method performs an adjustment to a closed traverse applying error predominately on distances while heavily weighting the directions (angles). The procedure for performing this adjustment is identical to the *Compass Rule*.

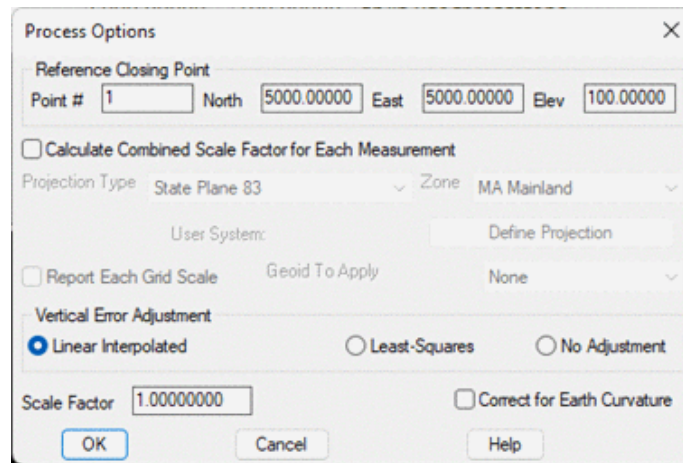
Transit Rule: This method of adjustment is similar to the *Compass Rule*. The main difference is the distribution of error is proportional to the length of the latitude and departure rather than the length of the traverse leg. The procedure for performing this adjustment is identical to the *Compass Rule*.

Least Squares: *Note, this is a legacy adjustment routine that has been superseded by SurvNET, a comprehensive least squares adjustment program that processes EDM traverse measurements, GNSS vectors and Differential Levelling.*



Within the Least Squares option, there are 3 steps to processing the data and adjusting the network.

- **Prepare Least Squares Data:** From the raw file data, this routine makes initial calculations for the coordinate points in the traverse.



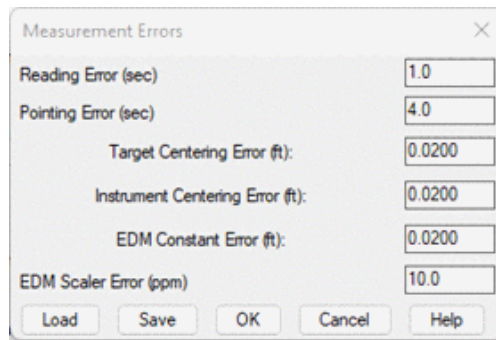
This data, along with the control point coordinates and the angle and distance measurements, is stored to a data file with the same name as the current RW5 file except with a .LSQ extension (ie: survey.lsq goes with survey.rw5). The constraints of the routine are:

- All angle readings must be in angle right mode.
- The coordinates of the starting and the ending points must be known.

The routine begins with a dialog for specifying the reference closing coordinates and any scale factors to apply to the distance measurements. The **Reference Closing Point** is the last point in the traverse, whose coordinates must be known. If an angle balance shot is used in the traverse, the Reference Angle Balance Angle must also be specified, either as a value or as the angle between known points.

When you click <OK>, a **Measurement Error** dialog box will appear.

Since angles and distances have errors of different magnitudes, they are normalized using weights, based on the accuracy and confidence with which these quantities have been measured. There is a dialog for specifying the estimated measurement errors. The Reading Error is the horizontal angular error in the instrument. For example, for a "5-second" instrument this error would be 5. The Pointing Error accounts for several factors in the horizontal angle reading including accuracy lining up the crosshairs on the target, the target size and the optical quality of the instrument. The Target and Instrument Centering Errors are the distance off the point due to faulty centering. The EDM Constant Error is the accuracy of the instrument distance measurements. The EDM Scaler Error is entered in parts per million for the increased error in longer measurements. These settings can be saved and loaded as a way to store settings for different equipment.



The program will calculate the weights for each distance and angle measurement using these measurement errors. The control points, points to adjust, distance and angle measurements with weights are reported. You can edit these measurements and weights using the Edit Least-Squares Data routine or go directly to the Process Least-Squares Data routine. You can also **Save** these settings as a .LQE file and **Load** them for future surveys. When you click <OK>, the routine calculates and reports the initial measurements.

Typical results of the initial measurement report.

Control Points

```
Point# Northing Easting
2 5000.000 5613.280
1 5000.000 5000.000
8 5000.000 5000.000
```

Adjust Points

```
Point# Northing Easting
2 5000.000 5613.280
3 4785.746 5420.474
4 4540.840 5246.162
5 4321.237 5090.216
6 4772.921 5287.377
7 4987.693 5284.473
```

Distance Observations

```
Occupy FSight Distance StdErr
1 2 613.283 0.035
2 3 288.234 0.023
3 4 300.606 0.023
4 5 269.341 0.022
5 6 492.839 0.030
6 7 214.792 0.020
7 8 284.717 0.022
```

Angle Observations

```
BSight Occupy FSight Angle StdErr
1 2 3 311°59'02" 10.133"
2 3 4 173°27'27" 14.874"
3 4 5 179°56'18" 15.367"
4 5 6 348°12'06" 10.386"
5 6 7 155°38'38" 15.040"
6 7 8 93°15'22" 15.684"
```

- **Edit Least Squares Data:** Once the initial data is processed with *Prepare Lest Squares Data*, you can view

and edit the data. This routine edits the points, measurements and weights stored in the .LSQ file associated with the current RW5 file. The editor works through the dialog shown. You can edit, add or remove the control points, adjust points, angle measurements or distance measurements. The program does not check that the editing is valid. So you need to make sure that your changes keep a good set of least-squares data (i.e. don't delete a needed control point). The Distance Error button allows you to set the distance standard error weights for all the distance measurements to the same value. Likewise the Angle Error button sets the standard error weights for all the angle measurements.

Edit Least-Squares Data

Control Points

Point#	North	East
2	5000.00000	5613.28050
1	5000.00000	5000.00000
8	5000.00000	5000.00000

Adjust Points

Point#	North	East
3	4785.74603	5420.47442
4	4540.83967	5246.16213
5	4321.23722	5090.21574
6	4772.92100	5287.37683

Distance Observations

From	To	Distance	Std-Error
1	2	613.28309	0.03522
2	3	288.23419	0.02253
3	4	300.60599	0.02293
4	5	269.34088	0.02194

Angle Observations

From	At	To	Angle	Std-Error
1	2	3	311.5902	10.13330"
2	3	4	173.2727	14.87353"
3	4	5	179.5618	15.36687"
4	5	6	348.1206	10.38605"

Distance Error Angle Error OK Cancel Help

- **Process Least Squares Data:** This routine applies a least-squares adjustment to the data stored in the .LSQ associated with the current raw data (.RW5) file. The closing errors are distributed among the other points, using the "Method of Least Squares" (Ref : Wolf, P.R. and Ghilani, C.D., 1996, "Adjustment Computations", John Wiley and Sons, NY, Third Edition). After the adjustment, the rest of the raw file is processed to recalculate the sideshots. There is an option to draw standard error ellipses around the adjusted points. The ellipse axes are multiplied by Ellipse Scale Factor to make the ellipse larger for easier viewing.

Process Options

Report Angle Format

Bearing Azimuth Angle Right By File

Calculate Elevations

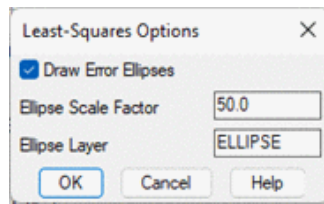
All SideShots Only None

Report SideShots

Decimal Places for Report 0.000 Use Report Formatter

OK Cancel Help

Clicking <OK> will bring up the Error Ellipse options.



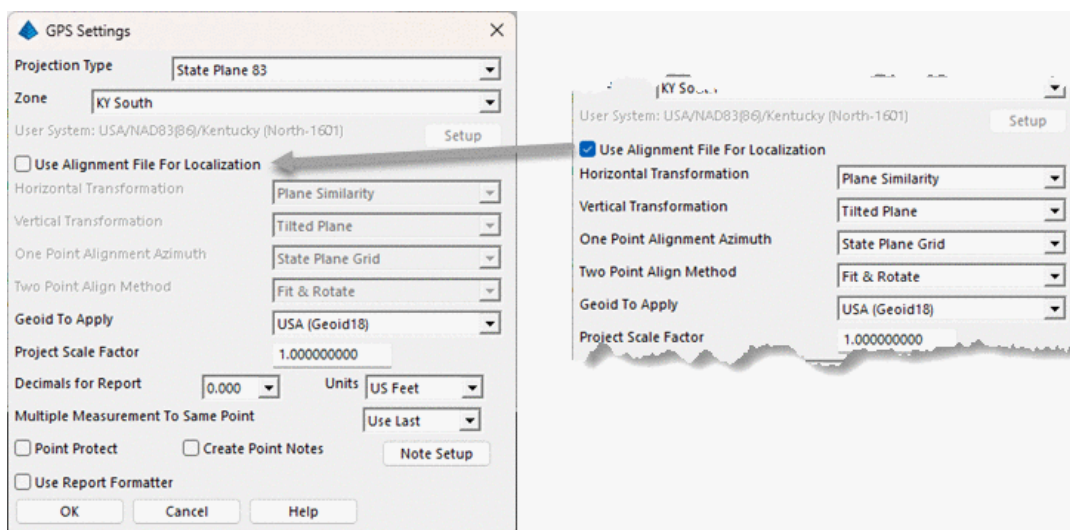
Error Ellipses are a graphic depiction of the network error for each point. The error is usually very small, so the scale factor is applied so you can visibly see the ellipses.

The least-squares process report shows the input data and the results. For each point, the amount adjusted and the standard error in X and Y are reported. The Reference Standard Deviation is based on the sum of the residuals and the initial estimated standard errors. The Chi-Squares test is a goodness-of-fit test that checks the reference standard deviation with the least-squares model. If this test fails, there may be a blunder in the measurement data or the initial estimated standard errors were too low or too high.

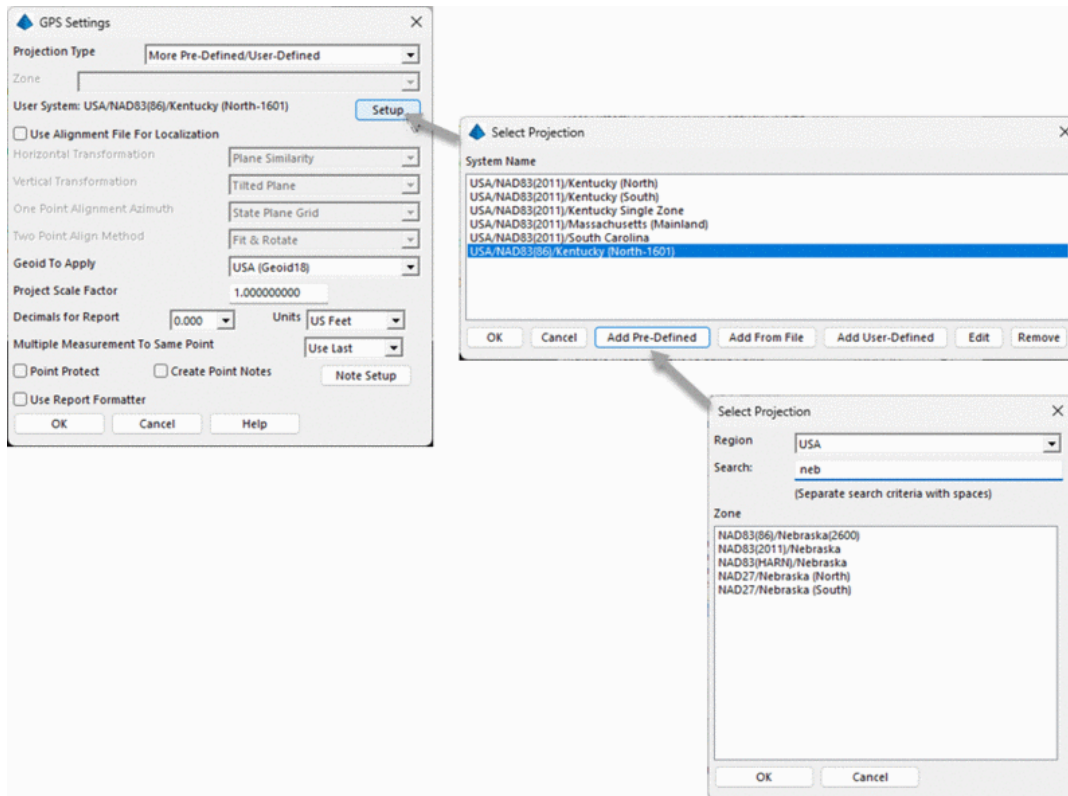
Stadia Processing Method: Provides functionality to process Stadia surveying notes. Stadia sighting depends on two horizontal cross-hairs, known as stadia hairs, within the telescope. These hairs are parallel to the horizontal cross-hair and are equally spaced above and below it. The distance between the two stadia hairs is known as the intercept. The distance from the instrument to the rod is 100 times the intercept. For example, an intercept of 3.10 would represent a distance of 310 (3.10 X 100). For entering in stadia notes, you would enter the horizontal angle, the distance (entered as the intercept X 100) and the vertical angle. The processing options for Stadia reduction are similar to the Common Options shown above.

GPS: The process GPS routine allows for reduction of GPS records that reside in a raw (*.RW5) file from latitude, longitude and WGS84 Ellipsoid Height to State Plane or local coordinates. When a DZ (Depth) record follows a GPS record, the depth value is used to adjust the GPS point elevation. When a GSAL record follows a GPS record, the position is adjusted by the record delta tilt values for the rod tilt. When using a CRDB coordinate file format, the process GPS routine stores the lat/lon and RMS values from the raw data to point attributes in the CRDB.

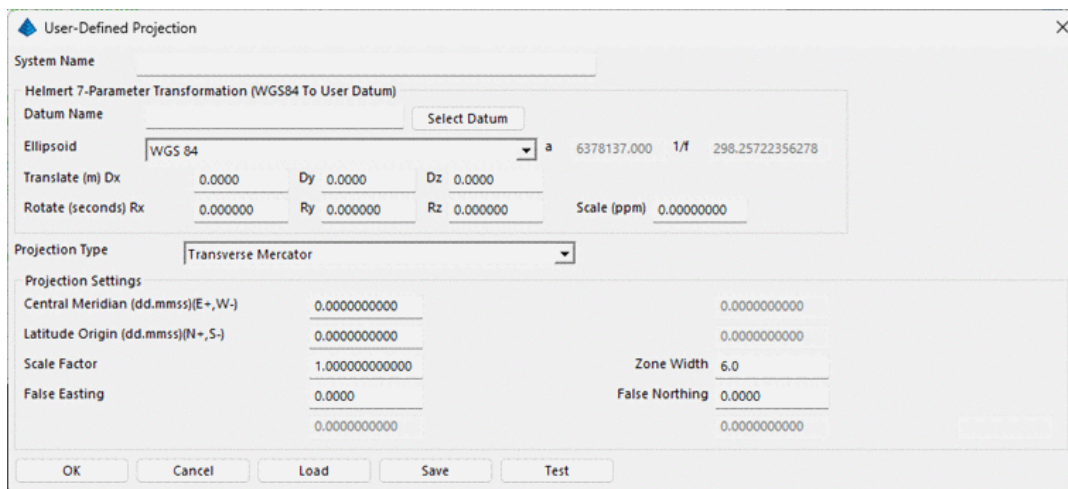
The Process GPS settings dialog is shown below.



- **Projection Type:** Defines the datum coordinate system to be used for converting the latitude, Longitude and WGS84 Ellipsoid height collected from the GPS receiver into Cartesian coordinates. The supported projection types are State Plane 83, State Plane 27, UTM, Lat/Long, Great Britain-OSGB36, Australia, New Zealand-NZGD2000, New Zealand-NZGD49, France NTF-GR3DF97A, and Barbados National Grid. A User-Defined option is also available for defining a user projection or loading More Predefined projections.



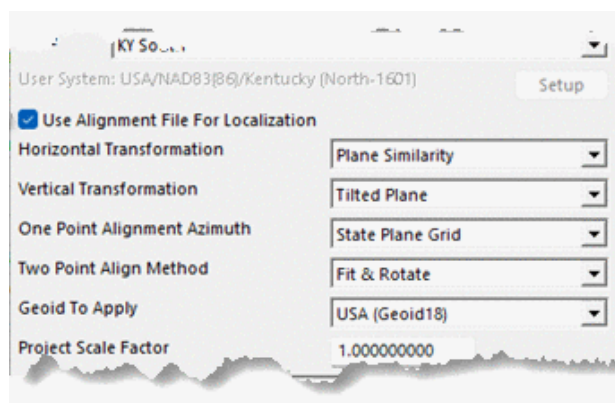
- **Zone:** for State Plane projections, you must select the correct state zone that you are working in. For UTM, the Automatic Zone option will have the program automatically use the program automatically use the correct UTM zone for your location. Otherwise for UTM, you can manually set a specific UTM zone. This manual option applies to working on the border between zones and you want to force the program to always use one of those zones.
- **Add User-Defined** is an option to define a new projection using a dialog box to populate all appropriate data.



Enter a name for your system (e.g. PRVI for Puerto Rico/Virgin Islands), then select a Projection type and enter the appropriate parameters. Note that all latitude and longitude values are in Degrees Minutes and Seconds (dd.mmss) and False Northing and False Eastings are always presented in meters. Define a Datum shift by selecting the Select Datum radial button. You may select a predefined Ellipsoid or set your own parameters by typing in a new ellipsoid name in the Ellipsoid field and entering values for a and 1/f. When you enter in a new Ellipsoid name, the Datum name field will be blank. The values for Dx, Dy, Dz, Rx, Ry, and Rz and scale are "to WGS84". If the values you have are "from WGS84", simply reverse the sign of each value (positive becomes negative and vice versa).

You may save your system to a ".udp" file. To Load a user defined coordinate system from a file, select the Load radial button. A list of user defined systems will be displayed. Select the desired system and press OK.

- **Use Alignment File For Localization:** With this option toggle on, a prompt for the Alignment File to Process will be displayed. This file is typically created by SurvCE (Carlson's Data Collection System) using the Localization routine or by Carlson Field Using the Align to Local Coordinates routine. This file (.DAT, LOC, GC3) contains the parameters to transform the derived State Plane coordinates to the defined local coordinates.



At the end of the process, the coordinates will be written to the current coordinate (*.crd) file and a report will be presented in the Carlson editor for saving or printing purposes.

- **Horizontal Transformation:** The transformation in the align Local Coordinates command can either be by plane similarity or rigid body methods. The difference is that the rigid body method does a transformation with a translation and rotation and without a scale. The plane similarity does a rotation, translation and scale. This option only applies when two or more points are used in Align Local Coordinates or the Localization routine in SurvCE.
- **Vertical Transformation:** This transformation has two options, Level Plane and Tilted Plane.
- **One Point Alignment Azimuth:** This option applies to the rotation when using one point in Align Local Coordinates or the Localization routine in SurvCE. For this alignment method, the state plane coordinate is translated to the local coordinate. Then the rotation can use either the state plane grid or the geodetic as north. No scale is applied in this transformation. The state plane and geodetic true north diverge slightly in the east and west edges of the state plane zone. This option allows you to choose which north to use.
- **Two Point Alignment Method:** There are two options when using this method, Fit & Rotate and Rotate Only. Fit & Rotate will use the second point in the localization file for direction and scaling. The Rotate Only option allows you to use the second point in the localization file for direction but not for scaling. When using the Rotate Only option, any scale factor entered in the Project Scale Factor will be used.

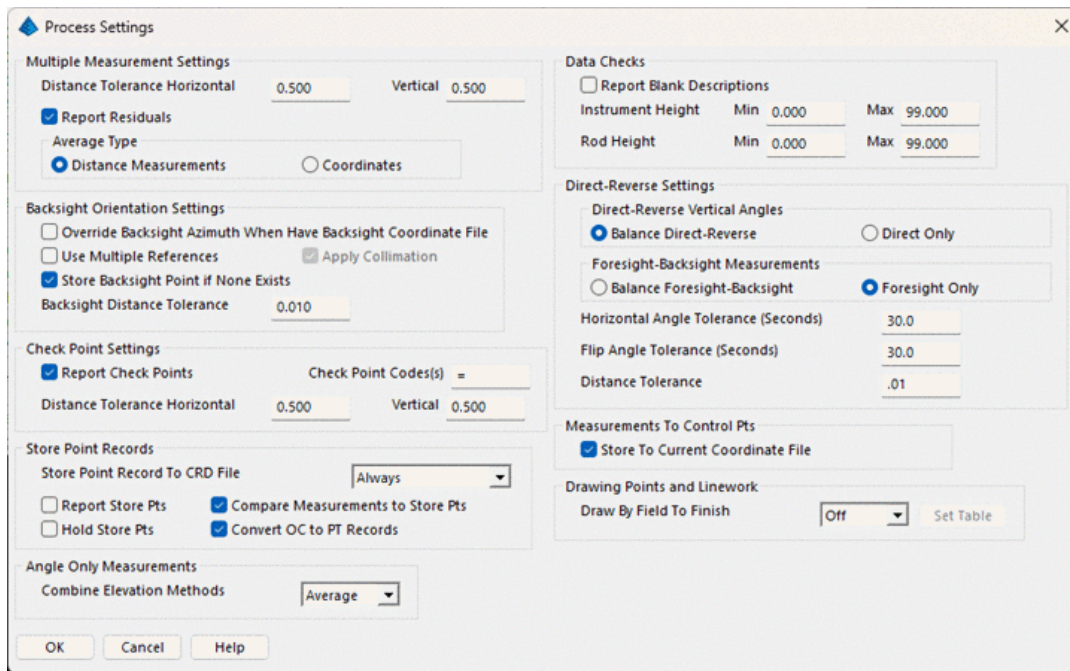
- **Geoid to Apply:** There are a vast number of supported Geoids which are available for selection using the drop down menu.

This option will account for the geoid undulation in determining the orthometric elevation of the measurement. The definition of the geoid model as currently adopted by the national Geodetic survey is the equipotential surface of the Earth's gravity field which best fits, in a least squares sense, global mean sea level. Orthometric elevation measurements are used in survey calculations. In order to convert ellipsoid heights (H_e) as measured by GPS into orthometric elevations (E_0), you must provide for a correction between the GPS-measured ellipsoid (reference ellipsoid) and a constant level gravitational surface, the geoid. This correction is the geoid undulation (U_g). The formula is $H_e = E_0 + U_g$.

Carlson applies the Geoid model by subtracting the Geoid undulation from the GPS elevation. The resulting elevation is then used and displayed. In practice, the Geoid model is most applicable to two types of alignment scenarios. One of these types is when setting up the base over a known point and having no alignment control points. The other is when there is one alignment control point. When using multiple alignment control points, the Geoid model is not as important because Carlson can model the elevation difference which can generally pick up the local Geoid undulation.

- **Project Scale Factor:** For most applications, the Scale Factor should be set to 1.0. The scale factor represents the "combined" grid/elevation factor that reduces ground distances to grid. After converting the LAT/LONG from the GPS records to state plane coordinates and applying the coordinate alignment (Localization) file, the Project Scale Factor is applied as the final adjustment to the coordinates. This adjustment is used on the X, Y, and not the Z. The Project Scale Factor is applied by dividing the distance between the coordinate and a base point by the Project Scale Factor. The coordinate is then set by starting from the base point and moving in the direction to the coordinate for the adjusted distance. The base point is the first point in the alignment (Localization) file. If there are no points specified in the alignment file, then 0,0 is used as the base point. If using an alignment file (Localization File) this value will be automatically calculated and displayed. Manual entry of a scale factor is also permitted and is often used with the Two Point Alignment Method when a scale factor is known.
- **Units:** Coordinates can be reduced into one of three available units, Metric, US Feet or International Feet.
- **Multiple Measurements to Same Point:** Specify what you would like to program to do when 2 or more points are located with the same point name. The options are Use Last, Average and Use First. the options Use First or Last will overwrite previous point locations while the Average option will perform an average of all same named points.
- **Point Protect** and **Create Point Notes** are described in the Common Process Options above.

Process Settings: This dialog box contains a large variety of settings that affect the way data will be processed. It is limited to traditional traverse methods such as EDM total stations. Processing GPS locations has a separate dialog box contained in the processing option itself.



- **Multiple Measurement Settings:** These options provide control for managing how multiple measurements to the same point are handled and reported.
- **Distance Tolerance Horizontal and Vertical:** Allows for user input of desired tolerance values for multiple measurements. Exceeded tolerances will be displayed on the process results report. With the Report Residuals option ON, the residual values of the measurements will be shown on the process results report.
- **Report Residuals:** These options control how the residual differences between multiple measurements will be displayed. The options are Distance Measurements or Coordinates.
- **Backsight Orientation Settings:** These options are specific to the back sight record.
 - **Use Multiple References** option will take multiple backsight measurements for an occupation and computes a least squares orientation for the instrument. There is also an option to compute and correct for the instrumental collimation error from the available measurements if both direct and reverse readings to one or more stations in the same set have been recorded. The program uses the BD (backsight direct) and BR (backsight reverse) records to identify the measurements to process. You can backsight different targets. The targets do need to have known coordinates either as points in the current coordinate file or as SP records in the raw file. The measurements can be complete with angles and distance, and they can be partial with only angles or only distance. When this option is active, the calculated backsight orientation will override the SetAzi field in the BK (backsight) record for the setup. The process report will include all the measurements used, the residuals and the resulting backsight orientation. The least-squares routine will also calculate the occupied station coordinate by resection if possible from the measurements and the report includes this calculated position along with the reference position and residuals. This calculation of the occupy point is used only for a check for the report and does not affect the occupy coordinate for processing. Note that if the occupied station position is unknown, there must be sufficient measurements to at least three known reference stations to support the resection and orientation solution. Here is an example of the raw data and the report.

Raw Data

PntNo	Northing	Easting	Elevation	Desc
71	4998.1900	5199.8200	125.0000	PK
22	4770.1200	5192.5000	90.0000	PK
2	4900.2700	5007.3100	75.0000	PK
53	5345.8600	4799.0400	150.0000	PK
1	5000.0000	5000.0000	100.0000	PK
InstHt	RodHt			

```

5.000    1.000
OcPt     BsPt           Azi           SetAzi
1        1              0°00'00''    0°00'00''
OcPt     FsPt           HorzAngle     SlopeDist  ZenithAng   Desc
BD 1     71          AR 0°31'09''    200.930    84°00'06''  PK
InstHt   RodHt
5.000    2.000
OcPt     FsPt           HorzAngle     SlopeDist  ZenithAng   Desc
BD 1     22          AR 50°03'22''    300.120    92°29'02''  PK
InstHt   RodHt
5.000    3.000
OcPt     FsPt           HorzAngle     SlopeDist  ZenithAng   Desc
BD 1     2          AR 85°48'25''    103.580    105°06'37'' PK
InstHt   RodHt
5.000    4.000
OcPt     FsPt           HorzAngle     SlopeDist  ZenithAng   Desc
BD 1     53          AR 239°50'24''    403.000    83°01'04''  PK
BR 1     53          AR 59°50'31''    403.001    276°58'56'' PK
InstHt   RodHt
5.000    3.000
OcPt     FsPt           HorzAngle     SlopeDist  ZenithAng   Desc
BR 1     2          AR 265°48'38''    103.581    254°53'23'' PK
InstHt   RodHt
5.000    2.000
OcPt     FsPt           HorzAngle     SlopeDist  ZenithAng   Desc
BR 1     22          AR 230°03'39''    300.121    267°30'58'' PK
InstHt   RodHt
5.000    1.000
OcPt     FsPt           HorzAn
gle SlopeDist ZenithAng Desc
BR 1     71          AR 180°31'16''    200.931    275°59'54'' PK

```

Multiple Backsight Orientation

```

OcPt     BsPt           Azimuth       SetAzi
1        1              00°00'00''    269°59'57''
Orientation Standard Deviation: 0.000
Instrument Height: 5.000

```

Occupy Point	Northing	Easting	Elevation
Reference	5000.000	5000.000	100.000
Calculated	5000.000	4999.999	100.000
Delta	0.000	0.001	-0.000
Standard Dev	0.003	0.003	0.000

Target Points	Northing	Easting	Elevation	Rod Height
71	4998.190	5199.820	125.000	1.000
22	4770.120	5192.500	90.000	2.000
2	4900.270	5007.310	75.000	3.000
53	5345.860	4799.040	150.000	4.000

Measurement	Hz Angle	Residual	Zenith	Hz Dist	Residual
71	0°31'09''	-0°00'04''	84°00'06''	199.830	-0.001
22	50°03'22''	0°00'01''	92°29'02''	299.838	-0.003
2	85°48'25''	-0°00'02''	105°06'37''	99.999	-0.002
53	239°50'24''	0°00'03''	83°01'04''	400.011	-0.006
53	59°50'31''	-0°00'04''	276°58'56''	400.012	-0.007
2	265°48'38''	-0°00'15''	254°53'23''	100.000	-0.003
22	230°03'39''	-0°00'16''	267°30'58''	299.839	-0.004
71	180°31'16''	-0°00'11''	275°59'54''	199.831	-0.002

- **Store Backsight Point if None Exists:** This option allows for the creation of a backsight point based on the the initial backsight (BD) record. The point will be created based on the measurement data from the Occupied Point which needs starting coordinates, a back sight azimuth and distance to the backsight point.

- **Backsight Distance Tolerance:** This value is a tolerance for reporting the difference between a stored point used for a backsight and the field measurement to that point.
- **Check Point Settings:** These options provide user controls for survey check points. Essentially, points located more than once intended for precision checks.
 - **Report Check Points:** When toggled ON, any point coded as a check point in the raw data file, will be reported. When selected the *Check Point Code* and Distance Tolerance fields become active and allow for editing.
 - **Check Point Code** is a user specified code entered in during the survey that tells the program to check the coordinates of a particular point with the coordinates of another point. This code is configurable by the user. An example of a point description coded as a Check Point would be as such, "trav =8". This description tells the program that the description of the point is "trav" and to check the coordinates of the this point with that of point #8.
 - **Distance Tolerance Horizontal** and **Vertical** are user specified tolerances for the check point. If either of these tolerances is exceeded it will be reported on the process results report.
- **Store Point Records:** These options control how any store point (PT) record is handled during processing of the raw data file. There are three options for storing Store Point (PT) records, *Never*, *Always*, and *When CRD Empty*. *Never* prevents any Store Point (PT) Record Report in the raw file from being written to the crd file. With this option on no existing point in the crd file would be overwritten. *Always* will write to the coordinate file and will overwrite any existing point with the same number of the Store Point (PT) records. The *When CRD Empty* option will only write Store Point (PT) records to the coordinate file when it is empty.
 - **Report Store Points** displays all store points in the process results report.
 - **Hold Store Points** option will hold the coordinate values for the store point record when measurements are taken to the store points. This will prevent the coordinates of the point from changing if measurements to the point dictate a change in coordinate position.
 - **Compare Measurements to Store Pts:** This option will add residuals to the report for field measurements to already stored PT records or points contained in the current CRD file.
 - **Convert OC to PT Records** option will convert description (DS) records to store point (PT) records when the description record contains contains an OC tag with an occupy point number and coordinates for a point number that isn't already used in another PT or SS record.
- **Angle Only Measurements:** The **Combine Elevations Method** applies to points calculated from Angle Only measurements. Angle Only points are calculated as part of the processing for the No Adjust, Compass, Crandall, Transit and Angle Balance process methods. To calculate points from Angle Only measurements, there needs to be multiple SS records with horizontal and vertical angles and no distance with the same target foresight point number from setups at different occupy points. The elevations can be set as the average from the multiple measurements, using the highest measured elevation, or using the lowest measured elevation. For example, to survey the top of a tree, you could have a SS to foresight point 99 from occupy point 1 with a horizontal and vertical angle and another SS to foresight point 99 from occupy point 2 with a horizontal and vertical angle. Then point 99 can be calculated by angle-angle intersect which determines the horizontal distances from 99 back to occupy points 1 and 2. These distances are then used with the vertical angles and occupy point elevation to calculate the elevation at point 99.
- **Data Checks:**
 - **Report Blank Descriptions** option reports any raw data records with empty descriptions. The
 - **Instrument Height** and **Rod Height** settings are used to check the instrument and rod heights when the raw file is processed. The program will report warnings if there are any heights that exceed the specified min/max ranges.
- **Direct Reverse Settings:**
 - **Direct-Reverse Vertical Angles:** This option determines how to handle direct-reverse vertical angle measurements when processing. *Balance Direct-Reverse* will take the mean of the direct-reverse

measurements and use this value when processing the file. **Direct Only** will only use the direct measurement to the point for processing.

- **Foresight-Backsight Measurements:**

- **Balance Foresight-Backsight** allows for averaging in the Foresight and backsight measurements when using direct-reverse sets. This option applies the mean angle of the Foresight and backsight to the backsight azimuth if entered.
- **Foresight Only** option will average the foresight measurements only of a direct-reverse set. This is the default setting

- **Horizontal Angle Tolerance (Seconds):** This is the tolerance that the angle measured by the direct measurements and the angle measured by the reverse measurements in a direct-reverse set must fall within.

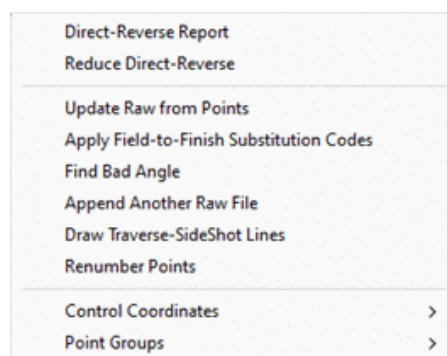
- **Flip Angle Tolerance (Seconds):** User specified value for the acceptable difference in measured horizontal angles determined from the direct (BD-FD) and reverse (BR-FR) observations.

- **Distance Tolerance:** User specified tolerance for the difference in distance measurements to the same points. When this value is exceeded on a measurement, it will be displayed on the process results report.

- **Measurements To Control Points:** The **Store To Current Coordinate File** option applies when a control coordinate file is used in addition to the active coordinate file. When processing the raw file, measurements to point numbers that are in the Control Coordinate File will not be stored into the active coordinate file when this option is disabled (off). When this option enabled (on), all control points that have a foresight measurement to them, will be stored in the active coordinate file.

- **Drawing Points and Linework:** This option controls the drawing of points and linework using Field to Finish. It differs from the draw traverse and sideshot lines under the Tools Menu of the Raw Editor by using a field to finish code table (*.fld) to define how the points and linework are to be drawn and layerized. There are three settings for this option, Manual, Auto and Prompt. Manual means that the file will not be processed using the field to finish codes and no points or linework will be drawn upon exiting the raw editor. The Auto option will use the current or last used field to finish file (*.fld) to draw the points and lines on the drawing screen when the raw editor is existed. The option of Prompt will give the option to draw the points and lines to the screen. With this setting specified, a prompt will be displayed when exiting the editor.

Tools Menu



Direct-Reverse Report: This routine creates a report of direct and reverse shots along with the resulting averaged shots. Any tolerance specified in the Process Settings>Direct-Reverse Settings section, that is exceeded will be displayed in this report. The residuals are the difference between the measurement and the final average. If the current spreadsheet display mode for distances is set to horizontal, then the report will show horizontal distances. Otherwise, the report uses slope distances.

Reduce Direct-Reverse: This routine processes the direct and reverse shots and simplifies the raw file by replacing the sets of direct and reverse shots with the resulting average traverse record. It is not necessary to perform this operation to ensure mean angles are used during the processing.

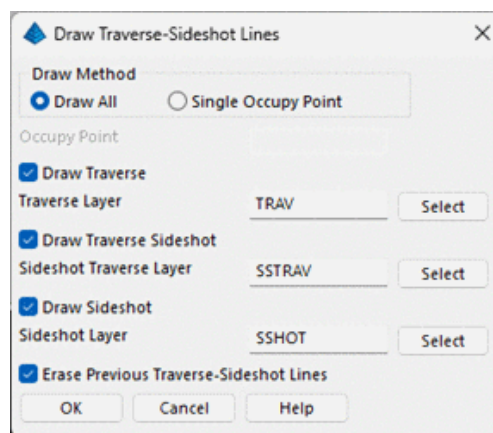
Update Raw from Points: This routine is used to update the raw data based upon the coordinates of the points contained in the coordinate file. There is a choice to update only the point descriptions in the raw data or update both the descriptions and the measurement values. An example for updating the measurements is when the raw data has been processed using the compass rule adjustment method, the points in the crd file are now adjusted. However the raw data remains unchanged. If a record of the rw5 file reflecting the angles and distances between the points after an adjustment has been ran is desired, this routine can be run thus updating the raw data to reflect the adjusted angles and distances. Another application for this routine is that of building a rw5 file for future processing and adjustment. For example if a point file or text file has been received from another engineering firm or fellow surveyor and you would like to build a rw5 file for future reference and processing this option can also be used to accomplish this. The rw5 file would be set up with the occupied points, foresight points and the desired angle type to use specified for the traverse. This would be all the manual entry of the data necessary. After creating the "shell" of the traverse then run the update raw from points routine and the raw data, as contained in the coordinate file, will be imported into the rw5 file thus filling out the horizontal angle, distance and vertical components specified.

Apply Field-to-Finish Substitution Codes: This routine updates the point descriptions in the raw data using substitutions defined in the Field-to-Finish command.

Find Bad Angle: This routine applies the angular error to each traverse record one at a time. The adjusted traverse record that improves the closure the most is reported as the Bad Angle. The angular error is the difference between the angle balance shot and a reference angle.

Append Another Raw File: This routine prompts for another raw data (.RW5) file which is read and the data added to the end of the existing raw data (.RW5) file. For example, if you are editing the raw file from the first days work and have a separate raw file with a second days work, you can use this routine to add the second raw data to the first raw file.

Draw Traverse-Sideshot Lines: This routine draws lines for all the traverse and sideshot records.



Each option has a layer selection for that particular entity. If a layer does not exist, it will be created as it is drawn.

- Draw All: draws all traverse and/or sideshot lines contained in the raw file.
- Single Occupy Point: draws traverse legs and sideshots from a specific occupied point.
- Draw Traverse: draws all traverse lines
- Draw Traverse Sideshot: draws traverse sideshot lines. Sideshot Traverses are traverses that do not lead to the closing or ending point.
- Draw Sideshot: draws sideshot lines from the occupied point to the foresight point.

This command does not process the raw file. Instead it reads the raw file and for each traverse and sideshot record, the program looks up the coordinates for the occupied and foresight points in the CRD file. So it may be necessary to run a Process Option before running this routine. With the Erase Previous Traverse-Sideshot Lines toggled on, any previous linework drawn using this method will be erased from the drawing screen before drawing the lines again.

Renumber Points: This routine renumbers points in the raw file. This applies to all point numbers including: TR, SS, and PT records. The points can be renumbered by either incrementing the number or by adding a suffix. The Renumber Duplicates Only option only renumbers points that are duplicated in the raw file.

The screenshot shows a dialog box titled "Renumber Points". It has a close button (X) in the top right corner. The dialog contains the following fields and options:

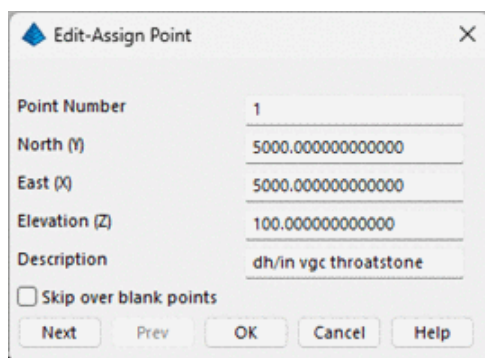
- Range of Points to Renumber:** A text box containing "100-TR100".
- Line Number To Begin Renumbering:** A text box containing "1".
- Line Number To End Renumbering:** A text box containing "670".
- Renumbering Method:** Two radio buttons are present. The first is "Increment Point Number By" (unselected). The second is "Add Point Number Suffix" (selected). To the right of the selected radio button is a text box containing "_B".
- Renumber Duplicates Only:** A checked checkbox.
- At the bottom are two buttons: "OK" and "Cancel".

- **Range of Points to Renumber:** Enter in the range of points to change, ie 1-4.
- **Line Number to Begin Renumbering:** This corresponds to the line number located at the far left of the raw data editor. Enter the line number to begin the renumbering.
- **Line Number To End Renumbering:** This also corresponds to the line number located at the far left on the raw data editor. Enter the line number to end the renumbering. If the range of numbers specified does not occur between the beginning line number and the ending line number, no changes will be made.
- **Renumbering Method**
 - **Increment Point Number By:** This number will be added to the existing point number to create the new point number. For example, if the number to add is 10 and the existing point numbers 1 and 6, the new renumber points will be 11 and 16.
 - **Add Point Number suffix:** This method adds a suffix to all points within the specified range. For example, point 1 could be renamed as 1.B

Control Coordinates: This option allows for the use of a supplemental control file for use in processing the raw file. The points in the control file will not be stored in the current (active) coordinate file unless specified to do so in the Process Settings.

Set Control Coordinate File
List Control Coordinates
Edit Control Coordinates

- **Set Control Coordinate File:** browse and select a coordinate file (CRD, CRDB, CGC, MDB, OR ZAK) to use as a Control File
- **List Control Coordinates:** Lists all points contained in the Control File
- **Edit Control Coordinates:** Opens a dialog box for displaying and editing points in the Control File:



Point Groups: This option can be used to organize the survey data into point groups.

Create All Point Group
Create Traverse Point Group
Create Sideshot Point Group

There are three options for the creation of point groups:

- **Create All Point Group:** creates a user specified group containing all of the points defined in the rw5 file.
- **Create Traverse Point Group:** creates a user specified group containing only the points defined in the traverse records (TR) of the rw5 file
- **Create Sideshot Point Group:** creates a user specified group that contains only the points defined in the sideshot records (SS) of the rw5 file

More about the format of the raw data (.RW5) file

The Carlson raw data format is a comma delimited ASCII file containing record types, headers, recorded data and comments. The format is based on the RW5 raw data specification, with the exception of angle sets. Angle sets are recorded as BD, BR, FD and FR records to allow reduction of all possible combinations. Essentially, these records are identical to a sideshot record.

Backsight Record

Record type: BK

Field headers:

OP Occupy Point

BP Back Point

BS Backsight

BC Back Circle

Sample:

BK,OP1,BP2,BS315.0000,BC0.0044

Line of Sight Record

Record type: LS

Field headers:

HI Height of Instrument

HR Height of Rod*

*GPS heights may be recorded to phase center or ARP depending on GPS make.

Samples:

LS,HI5.000000,HR6.000000

LS,HR4.000000

Occupy Record

Record type: OC

Field headers:

OP Point Name

N Northing (the header is N space)

E Easting (the header is E space)

EL Elevation

– Note

Sample:

OC,OP1,N 5000.00000,E 5000.00000,EL100.000,–CP

Store Point Record

Record type: SP

Field headers:

PN Point Name

N Northing

E Easting

EL Elevation

– Note

Sample:

SP,PN100,N 5002.0000,E 5000.0000,EL100.0000,–PP

Traverse / Sideshot Record / Backsight Direct / Backsight Reverse / Foresight Direct / Foresight Reverse

Record type: TR / SS / BD / BR / FD / FR

Field headers:

OP Occupy Point

FP Foresight Point

(one of the following)

AZ Azimuth

BR Bearing

AR Angle-Right

AL Angle-Left

DR Deflection-Right

DL Deflection-Left

(one of the following)

ZE Zenith

VA Vertical angle

CE Change Elevation

(one of the following)

SD Slope Distance

HD Horizontal Distance

– Note

Samples:

TR,OP1,FP4,AR90.3333,ZE90.3333,SD25.550000,-CP
SS,OP1,FP2,AR0.0044,ZE86.0133,SD10.313750,-CP
BD,OP1,FP2,AR0.0055,ZE86.0126,SD10.320000,-CP
BR,OP1,FP2,AR180.0037,ZE273.5826,SD10.315000,-CP
FD,OP1,FP3,AR57.1630,ZE89.4305,SD7.393000,-CP
FR,OP1,FP3,AR237.1612,ZE270.1548,SD7.395000,-CP

GPS

Record type: GPS

Field headers:

PN Point Name

LA Latitude (WGS84)

LN Longitude (WGS84, negative for West)

EL Ellipsoid elevation in meters*

- Note

*GPS heights may be recorded to phase center or ARP depending on GPS make.

Sample:

GPS,PN701,LA42.214630920,LN-71.081409184,EL-21.8459,-C

DZ

Record type: Depth

PN Point Name

DZ Depth

Sample:

DZ,PN10002,DZ0.3962

Alphabetical listing of Field Headers

AD Azimuth Direction (0 for North, 1 for South)

AL Angle-Left

AR Angle-Right

AZ Azimuth

BC Back Circle

BP Back Point

BR Bearing (this field will be recorded as N123.4500W)

BS Backsight (when back point is not defined)

CE Change Elevation

DL Deflection-Left

DR Deflection-Right

DT Local Date (MM-DD-YYYY)

DZ Depth

E Easting (the header is E space)

EC Earth Curvature (0 for off, 1 for on)

EL Elevation (GPS value is ellipsoid elevation in meters)

EO EDM Offset

FE Foresight Elevation

FP Foresight Point

HD Horizontal Distance

HI Height of Instrument

HR Height of Rod

LA Latitude

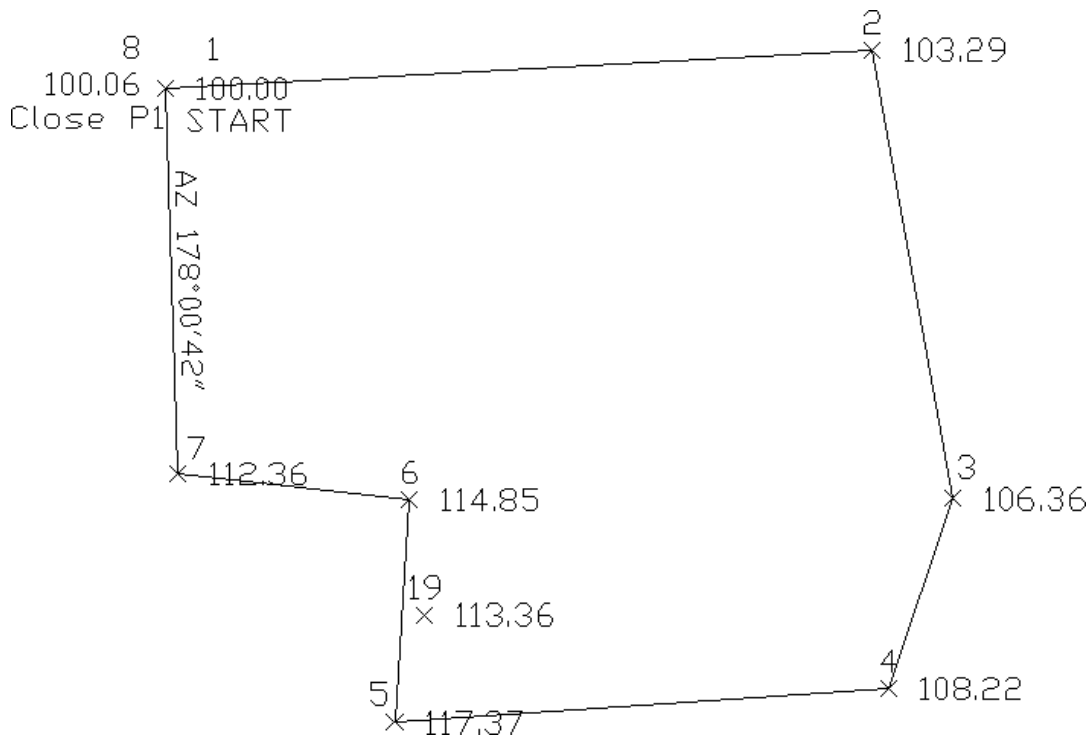
LN Longitude

N Northing (the header is N space)

OC Occupy Point Coordinates

OP Occupy Point
 PN Point Name
 SD Slope Distance
 SF Scale Factor
 TM Local Time (HH:MM:SS)
 UN Distance Unit (0 for feet, 1 for meter, 2 for
 US feet)
 VA Vertical Angle
 ZE Zenith
 – Note

Traverse Examples



This first example is a closed traverse with an internal backsight of azimuth 178d0'42"
 Use the functions under the Add menu to create and fill out the raw file as shown here.

Type	PntNo	Northing	Easting	Elevation	Description		
PT	1	5000.0000	5000.0000	100.0000	START		
	OcPt	BsPt	Azi	SetAzi			
BK	1		178.0042	0.0000			
	InstHt	RodHt					
HI	5.320	6.000					
	OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	Description
TR	1	2	AR	268.5330	711.320	89.4050	P2
	InstHt	RodHt					
HI	5.430	6.000					
	OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	Description
TR	2	3	AR	262.5448	457.760	89.3236	P3
	InstHt	RodHt					
HI	5.400	6.000					
	OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	Description
TR	3	4	AR	208.5710	201.310	89.1803	P4
TR	4	5	AR	247.1657	497.120	88.5235	P5
SS	5	19	AR	289.3456	112.45	91.4405	SS1
TR	5	6	AR	277.4835	223.980	90.2926	P6
TR	6	7	AR	92.4143	233.880	90.2746	P7
	InstHt	RodHt					
HI	5.420	6.000					
	OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	Desc
CL+AB	7	8	AR	261.2756	387.250	91.4405	CLOSE

Notice that the record from point 7 to 8 is set as a CL+AB record. This tells the program that point 8 is the closing point and that the angle from 7 to 8 is the closing angle. For traverse adjustment, the closing reference point is 1 and the closure error is the difference between point 1 and point 8. For angle balance, the reference closing angle is 358d0'42" (178d0'42" + 180). The angle balance error is the difference between this reference angle and the angle from points 7 to 8.

Now let's process using Compass adjustment with Angle Balance. Choose Compass under the Process menu and fill out the dialogs as shown.

Closure Options

Reference Closing Point

Point #: North: East: Elev:

Report Angle Format

Bearing Azimuth Angle Right By File

Calculate Elevations

All SideShots Only None

Vertical Error Adjustment

Linear Interpolated Least-Squares No Adjustment

Create Point Notes Report Unadjusted Points

Report Point Adjustments Point Protect Report SideShots

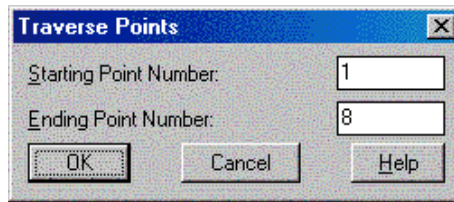
Apply Angle Balance Use Report Formatter

Decimal Places for Report:

Calculate State Plane Scale Factor at Each Setup Zone: 27 83

Scale Factor: Correct for Earth Curvature

OK Cancel Help



First half of process report:

Process Results 05/23/2002 10:06
 Raw file> c:/scadxml/data/example.rw5
 CRD file> C:/scadxml/DATA/example.crd

Scale Factor: 1.00000000
 Correct for Earth Curvature: OFF
 Starting Point 1: N 5000.00 E 5000.00 Z 100.00
 BackSight Azimuth: 178°00'42''

Point No.	Horizontal Angle	Zenith Angle	Slope Dist	Inst HT	Rod HT	Northing	Easting	Elev
2 P2	AR268.5330	89.4050	711.32	5.32	6.00	5038.43	5710.27	103.29
3 P3	AR262.5448	89.3236	457.76	5.43	6.00	4587.89	5791.20	106.36
4 P4	AR208.5710	89.1803	201.31	5.40	6.00	4397.30	5726.43	108.22
5 P5	AR247.1657	88.5235	497.12	5.40	6.00	4363.08	5230.59	117.37
19 SS1	AR289.3456	91.4405	112.45	5.40	6.00	4471.32	5260.88	113.36
6 P6	AR277.4835	90.2926	223.98	5.40	6.00	4586.54	5245.67	114.85
7 P7	AR92.4143	90.2746	233.88	5.40	6.00	4613.25	5013.33	112.36
8 CLOSE	AR261.2756	91.4405	387.25	5.42	6.00	5000.09	4999.97	100.06

Closure Results (Before Angle Balance)

Starting Point 1: N 5000.00 E 5000.00 Z 100.00
 Closing Reference Point 1: N 5000.00 E 5000.00 Z 100.00
 Ending Point 8: N 5000.09 E 4999.97 Z 100.06
 Azimuth Error : 341°38'22''
 North Error : 0.09061
 East Error : -0.03007
 Vertical Error: 0.05953
 Hz Dist Error : 0.09547
 S1 Dist Error : 0.11251
 Traverse Lines> 7
 SideShots> 1
 Horiz Dist Traversed: 2712.29
 Slope Dist Traversed: 2712.62
 Closure Precision: 1 in 28409

Remainder of process report:

Compass Closure

Adjusted Point Comparison

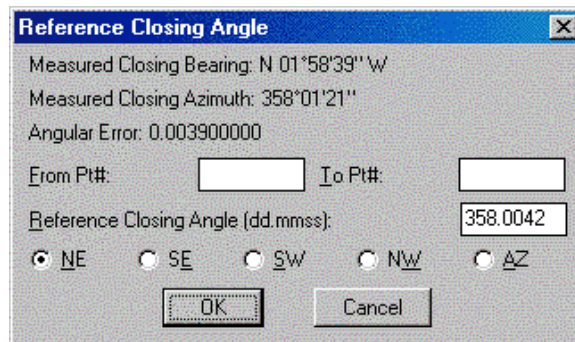
Point#	Original		Adjusted		Dist	Bearing
	Northing	Easting	Northing	Easting		
2	5038.445	5710.269	5038.440	5710.294	0.025	S 79°46'08" E
3	4587.914	5791.222	4587.907	5791.263	0.042	S 79°46'08" E
4	4397.319	5726.469	4397.310	5726.517	0.049	S 79°46'08" E
5	4363.044	5230.628	4363.032	5230.693	0.067	S 79°46'08" E
6	4586.509	5245.681	4586.496	5245.755	0.075	S 79°46'08" E
7	4613.178	5013.335	4613.163	5013.416	0.083	S 79°46'08" E
8	5000.017	4999.905	5000.000	5000.000	0.097	S 79°46'08" E

Max adjustment: 0.097

Starting Point 1: N 5000.00 E 5000.00 Z 100.00

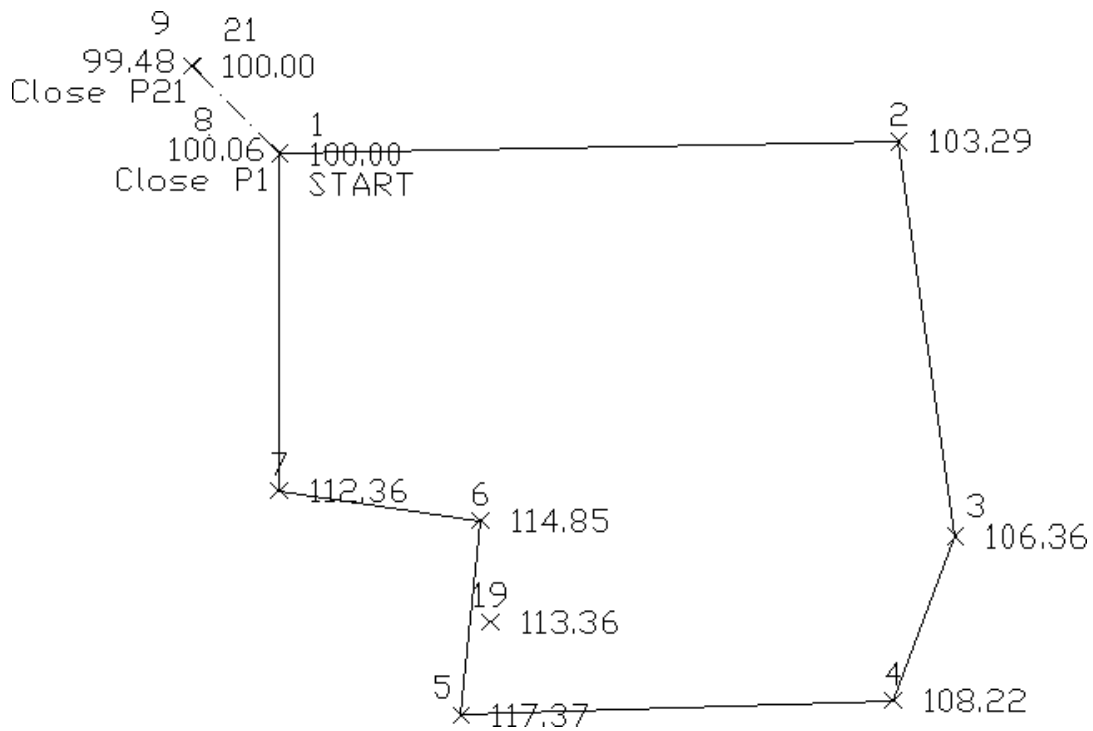
BackSight Azimuth: 178°00'42"

Point No.	Horizontal Angle	Zenith Angle	Slope Dist	Inst HT	Rod HT	Northing	Easting	Elev
2	AR268.5326	89.4050	711.34	5.32	6.00	5038.44	5710.29	103.29
P2								
3	AR262.5434	89.3236	457.76	5.43	6.00	4587.91	5791.26	106.36
P3								
4	AR208.5704	89.1803	201.30	5.40	6.00	4397.31	5726.52	108.22
P4								
5	AR247.1657	88.5235	497.09	5.40	6.00	4363.03	5230.69	117.37
P5								
19	AR289.3456	91.4405	112.47	5.40	6.00	4471.28	5260.97	113.36
SS1								
6	AR277.4839	90.2926	223.99	5.40	6.00	4586.50	5245.75	114.85
P6								
7	AR92.4130	90.2746	233.88	5.40	6.00	4613.16	5013.42	112.36
P7								
8	AR261.2758	91.4405	387.27	5.42	6.00	5000.00	5000.00	100.06
CLOSE								

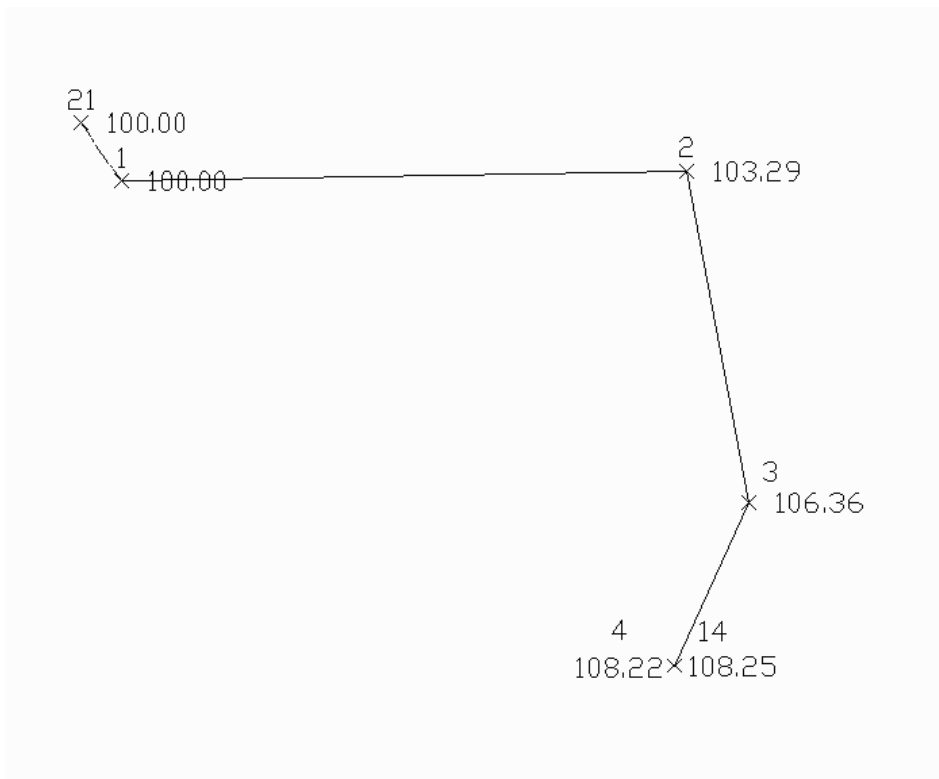


Shown above is the resulting process report. The angle balance had an error of 39 seconds which was divided among the 7 traverse sides. The Compass Closure shows how each traverse point was adjusted and then the resulting adjusted angles and distances.

Here is another layout of the last example that shows an external backsight setup. In this case there are two known points. Point 1 is the starting point and point 21 is the initial backsight. The setup could also use a backsight azimuth (ie north azimuth for example) instead of a backsight point number.



The closing record setup has changed from the last example. In this example, the shot from 7 to 8 is the closing shot with point 8 as the closing point. The closing reference point is still point 1. The angle balance shot is from 8 to 9 and the reference angle is from 1 to 21.



Example of an open traverse

The traverse starts from the known point 1 and ends at the known point 14. In this case there is no angle balance shot. The closing shot is from 3 to 4 with point 4 being the closing point. Point 14 is the closing reference point.

The closing record setup has changed from the last example. In this example, the shot from 7 to 8 is the closing shot

with point 8 as the closing point. The closing reference point is still point 1. The angle balance shot is from 8 to 9 and the reference angle is from 1 to 21.

Here is an example of an open traverse.

Compass Report from Open Traverse example:

Process Results

Raw file> d:/scdev/data/tsurvey.rw5

CRD file> d:/scdev/data/tsurvey.crd

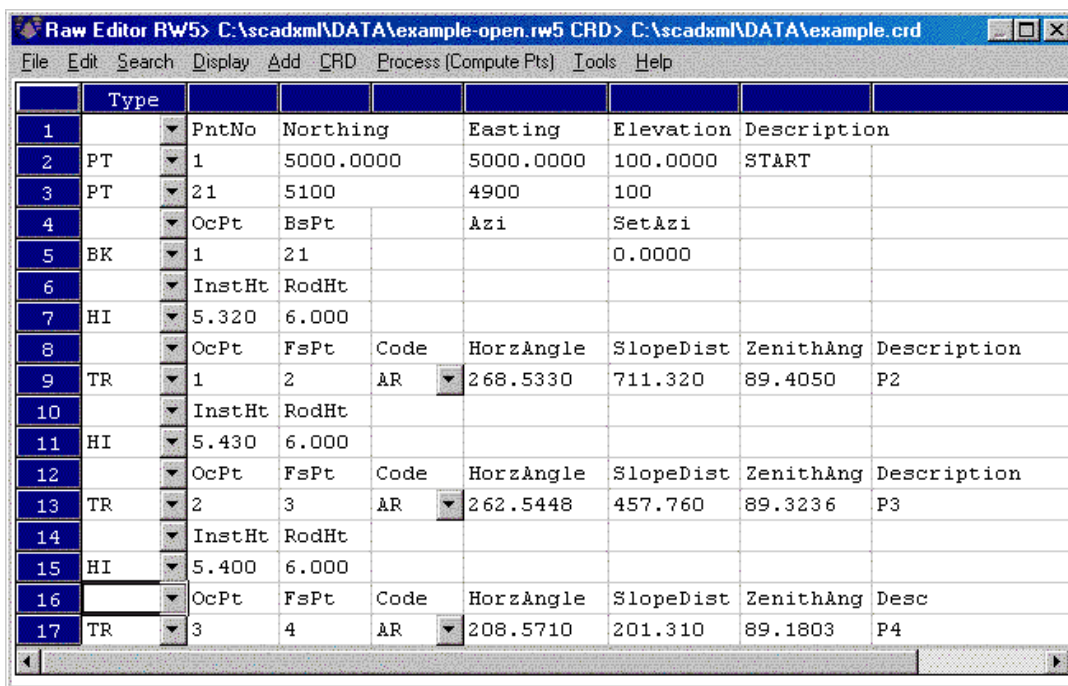
Compass Closure

Adjusted Point Comparison

	Original		Adjusted			
Point#	Northing	Easting	Northing	Easting	Distance	Bearing
2	5013.76	5711.18	5013.78	5711.13	0.047	N 63d21'19" W
3	4560.69	5776.42	4560.72	5776.35	0.078	N 63d21'19" W
4	4372.46	5705.08	4372.50	5705.00	0.091	N 63d21'19" W

Point No.	Horizontal Angle	Vertical Angle	Slope Dist	Inst HT	Rod HT	Northing	Easting	Elev
2	AR133.5324	89.4050	711.27	5.32	6.00	5013.78	5711.13	103.29
3	AR262.5506	89.3236	457.74	5.43	6.00	4560.72	5776.35	106.36
4	AR208.5712	89.1803	201.30	5.40	6.00	4372.50	5705.00	108.22

The traverse starts from the known point 1 and ends at the known point 14. In this case there is no angle balance shot. The closing shot is from 3 to 4 with point 4 being the closing point. Point 14 is the closing reference point.



Portion of typical Sokkia/SDR raw data file:

00NMSDR20 V03-05 Jan-22-98 19:14 122211
 10NMW970709A
 13CPSea level crn: N
 02TP00015000.000005000.0000085.63500005.22000000PK-FD
 08KI00035000.000005192.9200081.7450000MN-SET

07TP0001000390.00000000.00000000
09F100010003193.10000092.40416660.00000000MN-SET
09F100010100193.00000091.31388880.00000000SN-REC

Portion of typical Wild/Leica raw data file:

410001+000000SB 42....+00000000 43....+00000000 44....+00000000 45....+00000000 110002+00000002
21.124+35959590 22.104+08748240 31...1+00000000 51..0.+0012+000 110003+00000003
21.124+00000000 22.104+08748240 31...1+00267075 51..0.+0012+000 110004+00000004
21.124+00420390 22.104+08702570 31...1+00168234 51..0.+0012+000 110005+00000005
21.124+26029130 22.104+09311370 31...1+00206133 51..0.+0012+000 410006+000000IP 42....+00000000
43....+00000000 44....+00000000 45....+00000000 110007+00000006 21.124+25827090 22.104+09504550
31...1+00106228 51..0.+0012+000 110008+00000007 21.124+27151500 22.104+09312240 31...1+00106066
51..0.+0012+000

Portion of typical SMI raw data file:

CM Definitions: SS: Side Shot; TR: Traverse; OC: Occupied Coordinates;
PC: Point Coordinates; CM: Comment; OS: Occupied Station;
TS = time stamp; e = electronic; m = manual; CM TS TUE 04/09/91 09:41:25P
PC 1 5000.00000 5000.00000 0.00000
SS e HI:4.000 HR:5.000 PIPE/F
0 1 2 BAZ:0.00000 AR:0.00040 ZA:91.24330 SD:92.020
SS e HI:0.000 HR:0.000 BC/BR FRAME 1ST
0 1 3 BAZ:0.00000 AR:28.47220 ZA:91.20250 SD:65.240

Portion of typical PC COGO raw data file:

* NEW SET UP INST. AT 1 359 59 59 ON 4
L ANG 1000 4 1 77 18 52 4.44 * 1000 WALL# 283.22
L ANG 1001 4 1 55 44 28 9.8 * 1001 WALL# 283.28
L ANG 1002 4 1 38 37 8 15.89 * 1002 WALL# 283.48
L ANG 1008 4 1 27 18 34 123.82 * 1008 WALL# 287.75

Portion of typical Nikon raw data file:

MP,NOR,,5000.0000,5000.0000,100.0000,1
CO,Temp:111F Press:29.9inHg Prism:666 23-May-2000 10:30:36
ST,NOR,,1,,5.0000,0.0000,0.0000
SS,1,5.0000,131.0605,91.3744,88.4935,10:36:15,CL1
SS,2,5.0000,137.6770,90.2923,88.5236,10:36:50,CL1

Portion of typical MDL/Laser raw data file:

D052097F04P52I494P01P02
H32473V-0639R016202P03
H06687V-0706R014936P91
H03840V-0483R017380

Portion of typical Geodimeter raw data file:

50=HAWTHORN
54=19398
23=3222
2=1
37=1000.00
38=5000.00
39=700.000

Portion of typical Survis raw data file:

_OCCUPY_PNT_
621 616 5.140
148.36076

10255015.7245 3790987.2398 87.6695 ir
10255535.8009 3790669.8100 100.3900 ir
COMMENT
Thu Apr 08 08:14:14 1999
BACKSIGHT
0.00000 90.33400 609.4200 11.900 ir
SIDESHOT
1 0 0
18.47550 90.55000 17.4200 5.300 TP:gps1

Portion of typical Fieldbook raw data file:

NE 32 10696.4141 10043.5613 "SN-SET"
AZ 32 27 0
STN 32
BS 27
AD 27 0.00000 NULL "SN-SET"
AD 33 183.23250 183.660 "SN-SET"

Portion of typical SurvCOGO raw data file:

19100 , 0 , 19101 , 5 , 5.25 , 4.7 , 35.15 , 550 , 91.23 ,START
19101 , 19100 , 19102 , 5 , 5.15 , 4.7 , 35.15 , 120.23 , 88.34 ,
19102 , 19101 , 19103 , 5 , 5.2 , 4.7 , 125.1444 , 180.41 , 90 ,
19103 , 19102 , 19104 , 5 , 5.2 , 4.7 , 125.15 , 240.03 , 90 ,
19104 , 19103 , 19105 , 5 , 5.3 , 4.7 , 315.15 , 305.5 , 90 ,IRON PIN
19105 , 19104 , 19106 , 5 , 5.4 , 4.7 , 215.15 , 140.35 , 90 ,IRON PIN
19106 , 19105 , 19107 , 5 , 5.05 , 4.7 , 215.15 , 200 , 90 ,TACK IN FENCE
19107 , 19106 , 19108 , 5 , 5.2 , 4.7 , 300.23 , 400 , 90 ,

Pulldown Menu Location: Survey

Keyboard Command: rawedit

Prerequisite: None

Edit-Process Level Data

This command is for entering and calculating level data. It has a spreadsheet editor for entering the level measurements, and the level calculations are updated as the data is entered. There is also a processing and reporting feature.

Carlson Software supports two level file formats:

LEV Files: The .LEV file is the old format. You can still edit and process files in this format. The LEV format only supports differential levels, single and three-wire. The LEV file has 5 record types:

- 1) SR - Start Record. Contains the starting benchmark measurement.
- 2) TP - Turning point record, contains the backsight and foresight to the turning point.
- 3) LV - Side Shot (or level) record. Contains the foresight measurement to the point.
- 4) ER - End Record, contains the measurement to the ending benchmark.
- 5) Note/Comment - starts with two dashes

TLV Files: The TLV file format can contain Differential and/or Trig-Level data. This is Carlson's new format and is supported by SurvCE (Carlson data collection program). The TLV file has the following record types:

- 1) H1 - First header record contains project information
- 2) H2 - Second header record contains date, time, temperature and pressure information
- 3) BM - Benchmark record, contains the point number, elevation and description of the benchmark.
- 4) LS - Rod height, only used with TRIG-LEVEL data.

- 5) BS- Backsight measurement. This record contains the backsight point number and measurement:
 - a) Differential data: VD and HD - Vertical Difference (rod reading) and Horizontal Distance
 - b) Trig data: SD/ZE - Slope Distance and Zenith Angle
- 6) FS - Foresight measurement. This record contains the foresight point number and measurement.
 - a) Differential data: VD and HD - Vertical Difference (rod reading) and Horizontal Distance
 - b) Trig data: SD/ZE - Slope Distance and Zenith Angle
- 7) Note/Comment - starts with two dashes

This routine runs the *.TLV / *.LEV file editor and file report functions.

LEV File Editor:

If you are creating a new .LEV file, you must choose either single-wire or three-wire for your level format data entry preference.

Three Wire leveling, or precise leveling, is a process of direct leveling wherein three cross hairs, or threads, are read and recorded rather than the single horizontal cross hair. Note below, in the sample three-wire editor graphic, the additional columns representing top and bottom readings.

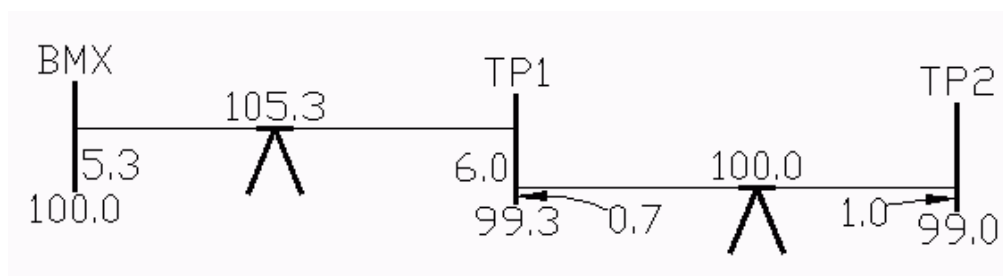
The commands starts by asking you, with a dialog box, to select an existing level file (.LEV) to process or to select a name for a new level file. The below examples are using existing files. Once this choice is made the small, Level Format dialog appears.



Format choice box

Regardless of whether you choose Single or Three Wire, the Level Editor appears in its own window. Below we see the editor displaying the contents of two existing files of level information. One is single wire and the other is three wire. The pulldown menus are the same for both, as described below in detail.

In the spreadsheet, the background color of the cells indicate the data type. White cells are for user-specified values. Blue cells are program calculated values. Black cells are data fields that aren't used by the level record for that row.



	Type	Point#	BS	HI	FS	Elevation	Code	AdjustElv	Description
1									
2	SR	BMX	5.300	105.300		100.000	EL	100.000	100.0
3	TP		0.700	100.000	6.000	99.300	EL		TP1
4	TP		15.000	114.000	1.000	99.000	EL		TP2
5	TP	A	6.800	111.900	8.900	105.100	EL		TBM1
6	TP		2.300	109.100	5.100	106.800	EL		TP3
7	TP		16.300	119.200	6.200	102.900	EL		TP4
8	ER	BMX	6.500		11.760	107.440	EL	107.500	
9	SR	BMX	6.500	114.000		107.500	EL	107.500	BM
10	TP		10.000	118.200	5.800	108.200	EL		TP4
11	TP	C	10.500	116.750	11.950	106.250	EL		TBM2
12	TP		4.800	116.250	5.300	111.450	EL		TP5
13	ER	BMX			16.300	99.950	EL	100.000	
14	SR	BMX	8.400	115.900		107.500	EL	107.500	BM
15	TP		1.500	113.600	3.800	112.100	EL		TP
16	TP	B	6.900	111.400	9.100	104.500	EL		TBM3
17	TP		5.000	111.190	5.210	106.190	EL		TP7
18	ER	C			4.990	106.200	EL	106.250	
19	SR	A	6.300	111.400		105.100	EL	105.100	TBM1
20	ER	B			6.980	104.420	EL	104.525	

Level File Editor - sample Single Wire data

	Type	Point#	BS-Top	BS-Mid	BS-Bot	HI	FS-Top	FS-Mid	FS-Bot	Elevation	Code	AdjustElv	Description
1													
2	SR	BMX	6.300	5.300	4.300	105.300				100.000	EL	100.000	
3	TP		1.300	0.700	0.100	100.000	7.000	6.000	5.000	99.300	EL		
4	TP		16.000	15.000	14.000	114.000	1.900	1.000	0.100	99.000	EL		
5	TP	A	7.800	6.800	5.800	111.900	9.900	8.900	7.900	105.100	EL		
6	TP		3.300	2.300	1.300	109.100	6.100	5.100	4.100	106.800	EL		
7	TP		17.300	16.300	15.300	119.200	7.200	6.200	5.200	102.900	EL		
8	ER	BMX					12.760	11.760	10.760	107.440	EL	107.440	
9	SR	BMX	8.500	6.500	4.500	113.900				107.400	EL	107.400	
10	TP		11.000	10.000	9.000	118.100	6.800	5.800	4.800	108.100	EL		
11	TP	C	11.500	10.500	9.500	116.650	12.950	11.950	10.950	106.150	EL		
12	TP		5.800	4.800	3.800	116.150	6.300	5.300	4.300	111.350	EL		
13	ER	BMX					17.130	16.130	15.130	100.020	EL	100.120	
14	SR	BMX	9.400	8.400	7.400	115.800				107.400	EL	107.400	
15	TP		2.500	1.500	0.500	113.500	4.800	3.800	2.800	112.000	EL		
16	TP	B	7.900	6.900	5.900	111.300	10.100	9.100	8.100	104.400	EL		
17	TP		6.000	5.000	4.000	111.090	6.210	5.210	4.210	106.090	EL		
18	ER	C					5.990	4.990	3.990	106.100	EL	106.200	
19	SR	A	7.300	6.300	5.300						EL		
20	ER	B					7.980	6.980	5.980	104.110	EL	104.420	

Level File Editor - sample Three Wire data

Level Settings ✖

3-Wire Tolerance:

3-Wire Distance:

File->Settings

File: Standard File routines - Open, Save, Save As, Settings, Print and Exit. Settings brings up a dialog where you can adjust the 3-wire tolerance and distance values. Open will allow you to open up another existing .LEV file.

File->Import: This routine imports Carlson .LEV format, Leica level data in .GSI or .XML format, TDS .RAW format, Topcon or Trimble .DAT format into the level editor.

File->Settings: This function has a setting for whether to use the Report Formatter for the Print function. The Report Formatter allows for customized reports and output to Excel. Without the Report Formatter, the Print function creates an automatic report.

Edit: Cut, Copy, Paste and Go To. Go To will take you to the row of your choosing.

Add: These options provide the standard level run routines. Details on each and a graphic of the pull-down follow.

Tools: This pull-down is for adjusting and storing elevations.

The Add and Tools pull-downs at the top of the editor provide the following features:



Level Editor - Add pull-down menu

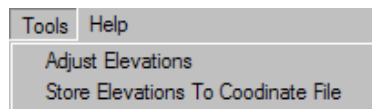
Level Start (SR): Starts the level run, usually with a known starting elevation or benchmark.

Level Turning Point (TP): Turning point procedure for leveling.

Level Side Shot (LV): For entering leveling side shots.

Level End (ER): Enter your value.

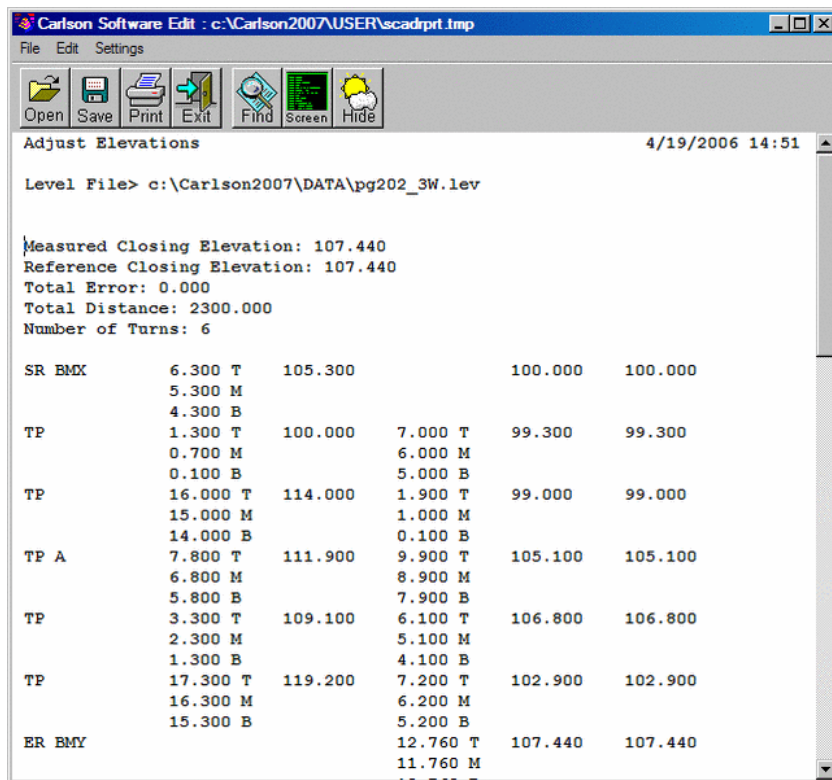
Note: You can add a note, or comments, into the editor as you move through the level run.



Level Editor - Tools pull-down menu

Adjust Elevations: This function will do a simple adjustment of your level data and place the adjusted elevations in the Adjusted Elevation column. If you are running 3 wire level loop the corrections will be inversely proportionate to the distance between the measurements. If you are running a single wire level loop, the corrections will be averaged by the number of turns.

Store Elevations to Coordinate File: If you have an active coordinate file passed to the level editor, this function will be available to store the elevations calculated in the level file to the active coordinate file by matching point numbers. The point number in the level file must match the point number in the coordinate file for an elevation to be stored. If adjusted elevations have been calculated, they will be stored. If not, the unadjusted elevations will be stored. This function shows a report of the original elevations from the coordinate file and the level elevations and reports matching or missing points numbers. After the report, there is a prompt to confirm whether to store the elevations to the coordinate file.



Selecting Print (editor File menu) provided this Level File Report sample

Editor Columns:

Type: These are small pulldown menus with two-letter level procedure choices. The two letters are abbreviations as indicated in the next dialog. These steps may be made with the Add pulldown or with this method. The options are SR, TP, ER, LV and DS. DS stands for description shot.

Point # - Point number of measurement.

BS - Backsight rod reading

HI - Calculated height of instrument

FS - Foresight rod reading

Elevation - Elevation of point

Code: The code is used by SurvNet for network least-squares processing of networked level loops. The code can be either EL or FE where EL is for calculated elevations and FE is for fixed elevations. FE should only be assigned to a START or END record (where you can enter the value for the adjusted elevation). If FE is assigned to an intermediate record it is ignored. Here is how the FE records are used. Say you run from one benchmark to another (point 1 to point 10). Point 1 and point 10 are the START and END records of the first loop and both are FE records. Then you start another loop at point 5 (halfway between 1 and 10). This is not a benchmark and can be adjusted so it should be assigned an EL code. Point 5 is the START record for the second loop. You run from point 5 to point 20 which is a benchmark. Point 20 is the END record and is assigned an FE code. When SurvNET processes the file, it will hold points 1, 10 and 20, allowing all others to be adjusted, including point 5 (even though it is a START record).

Adjusted Elevation - Adjusted elevation of point

Description - description of point

TLV File Editor:

TLV files can contain trig-level and/or differential level data. The editor will allow both type records in the same file.

Below is a sample Trig-Level TLV file:

Trig-Level Editor TLV> C:\Data\Triglev\Loop.tlv CRD>									
File Edit Add Tools Help									
		Point#	RodHt	Zenith	S.Dist	HI/Elev	AdjustElv	Description	
1		Note							
2	DS	H1,PRTest Project,OPTest Operator,LPTest Loop							
3	DS	H2,DT6/10/05,TM2:45PM,PRpressure 112 MB,TP101 F							
4		Point#				Elev	AdjustElv	Description	
5	BM	2				1000.010	1000.010	pt2 desc	
6		Point#	RodHt	Zenith	S.Dist	HI/Elev	AdjustElv	Description	
7	BT	2	4.500	89.31000	1500.540	991.806		pt2 desc	
8	BT	2	4.500	270.29000	1500.530	991.806		pt2 desc	
9		Note							
10	DS	test							
11		Point#	RodHt	Zenith	S.Dist	HI/Elev	AdjustElv	Description	
12	BT	2	4.500	89.31000	1500.560	991.805		pt2 desc	
13	BT	2	4.500	270.29000	1500.510	991.806		pt2 desc	
14	FT	11	4.500	89.31000	1200.540	997.463		SS Minus	
15	FT	11	4.500	270.29000	1200.530	997.463		SS Minus	
16	FT	11	4.500	89.31000	1200.560	997.463		SS Minus	
17	FT	11	4.500	270.29000	1200.510	997.462		SS Minus	
18	FT	12	4.500	89.31000	1300.540	998.311		100+00.00 RS	

Below is a sample Differential-Level TLV file:

Trig-Level Editor TLV> C:\Data\Level formats\Trimble\Benchloop.tlv CRD>								
File Edit Add Tools Help								
		Point#		V.Diff	H.Dist	HI/Elev	AdjustElv	Description
1		Point#				Elev	AdjustElv	Description
2	BM	1				100.000	100.000	86
3		Point#		V.Diff	H.Dist	HI/Elev	AdjustElv	Description
4	BS	1		3.953	26.870	103.953		BM
5	FS	1		4.055	15.410	99.898		BM
6	BS	1		4.053	15.110	103.951		BM
7	FS	2		3.949	27.620	100.002		BM
8		Point#				Elev	AdjustElv	Description
9	BM	3				100.000	100.000	BM

Menu Options:

File Menu:

Open - Open an existing .TLV file.

New - Creates a new TLV level file.

Save - Save changes

Save As - Save as different file name

Settings - Not used with TLV files.

Import - You can import the following level file formats: Leica GSI format, Leica XML format, and Trimble DAT format.

Print - get hard copy printout of data.

Exit - Exit Level Editor Program

Edit Menu:

Clipboard: Cut, Copy, Paste

Go To - "Go To" will take you to the row of your choosing. Enter the row number.

Add Menu:

Add: These options allow you to add or insert a new record into the level editor.

Benchmark Record (BM): Point with known elevation.

Backsight Record (BS): Differential-level measurement to the backsight point.

Foresight Record (FS): Differential-level measurement to foresight point.

Backsight Record (BT): Trig-level measurement to the backsight point.

Foresight Record (FT): Trig-level measurement to foresight point.

Note: You can add a note, or comments, into the editor as you move through the level run.

Tools Menu:

Adjust Elevations: This function will do a simple adjustment of your level data and place the adjusted elevations in the Adjusted Elevation column. If you have distances, either HD or SD for all your measurements, the corrections will be inversely proportionate to the distance between the measurements. If you are running a single wire level loop (VD but no HD), the corrections will be averaged by the number of turns.

Store Elevations to Coordinate File: It is important that the point numbers in the level file match the point numbers in the coordinate file. If you have an active coordinate file passed to the level editor, this option will be available to you. The elevations calculated in the level file will be stored in the active coordinate file by matching point numbers. The point must exist in the coordinate file before an elevation will be stored. After the elevations have been stored, a report will show which points were stored and which ones were not. If adjusted elevations have been calculated, they will be stored. If not, the unadjusted elevations will be stored.

Editor columns

Measurement records will have the following columns:

Trig Level Record:

Type - Two character abbreviation that shows the record type:

BM - Benchmark

BS - Differential-level backsight record

BT - Trig-Level backsight record

FS - Differential-level foresight record

FT - Trig-Level foresight record

DS - Note or Comment

Point # - Point number of measurement.

RodHt - Rod reading

Zenith - Zenith angle

S.Dist - Slope Distance

HI/Elev - Elevation of HI if a backsight record, or the foresight point if a foresight record

Adjusted Elevation - Adjusted elevation of foresight point

Description - description of point

Differential Level Record:

Type - Two character abbreviation that shows the record type, same as above.

Point # - Point number of measurement.

V.Diff - Rod Reading

H.Dist - Horizontal Distance

HI/Elev - Elevation of HI if a backsight record, or the foresight point if a foresight record

Adjusted Elevation - Adjusted elevation of foresight point

Description - description of point

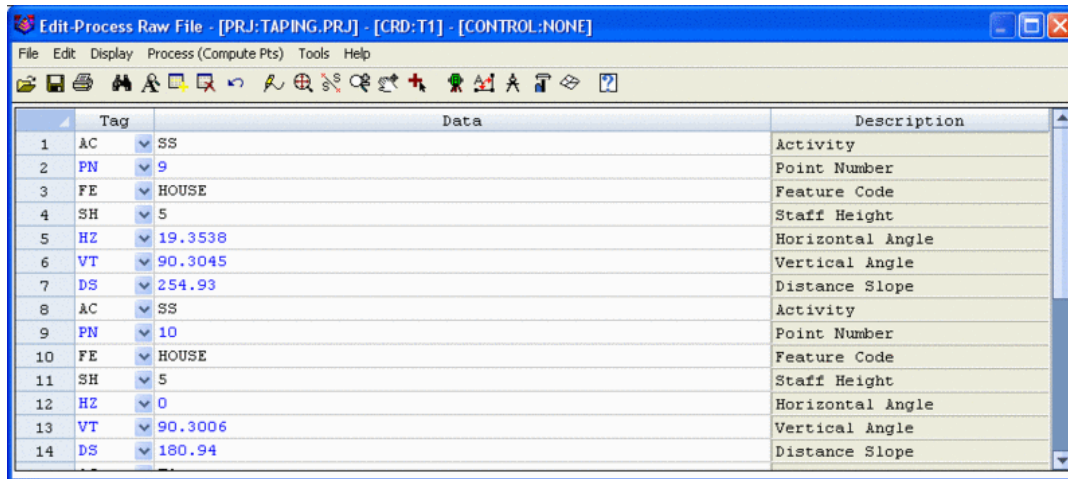
Pulldown Menu Location: Survey

Keyboard Command: diglevel

Prerequisite: .LEV (level) file to process

Edit Process SDMS File

This command processes SDMS format raw data from PRJ files. There is a spreadsheet editor with the data tag, value and description for each of the records. The processing functions are the same as the Edit Process Raw Data command. See that section of the manual for a description of the processing functions. The Edit Process SDMS command allows you to work with the SDMS raw data in its native format. Alternatively, you can run Edit Process Raw Data and convert the SDMS PRJ file into a Carlson RW5 file.



	Tag	Data	Description
1	AC	SS	Activity
2	PN	9	Point Number
3	FE	HOUSE	Feature Code
4	SH	5	Staff Height
5	HZ	19.3538	Horizontal Angle
6	VT	90.3045	Vertical Angle
7	DS	254.93	Distance Slope
8	AC	SS	Activity
9	PN	10	Point Number
10	FE	HOUSE	Feature Code
11	SH	5	Staff Height
12	HZ	0	Horizontal Angle
13	VT	90.3006	Vertical Angle
14	DS	180.94	Distance Slope

Pulldown Menu Location: Survey

Keyboard Command: sdmseedit

Prerequisite: None

SurvNET

SurvNET performs a least squares adjustment and statistical analysis of a network of raw survey field data, including total station measurements, differential level data and GPS vectors. SurvNET simultaneously adjusts a network of interconnected traverses with any amount of redundancy. The raw data can contain any combination of angle and distance measurements, and GPS vectors. SurvNET can adjust any combination of trilaterations, traverses, triangulations, networks and resections. The raw data does not need to be in a linear format, and individual traverses do not have to be defined using any special codes. **All measurements are used in the adjustment.**

SurvNET implements the standard parametric observation equation method with independent weighting for azimuths, directions, angles, distances, GPS baselines, coordinates, elevations and level data to compute least squares estimates of all unknowns in accordance with well-established reference texts: Ghilani, Charles D. & Wolf, Paul R., (2006) *Adjustment Computations: Spatial Data Analysis*. Hoboken, NJ: Wiley.

SurvNET produces a wealth of statistical information that allows an effective way to evaluate the quality of survey measurements. In addition to the least squares statistical information there is an option to compute traverse closures during the preprocessing of the raw data. Traverse closures can be computed for both GPS and total station traverses. This option has no effect on the computation of final least squares adjusted coordinates. This option is useful for surveyors who (due to statutory requirements) are still required to compute traverse closures and for those surveyors who still like to view traverse closures prior to the least squares adjustment.

For additional information, feel free to explore the introduction to SurvNET or let's get started!

Pulldown Menu Location: Survey

Keyboard Command: survnet

Introduction

- SurvNET reduces survey field measurements to coordinates in assumed, UTM, SPC83 SPC27, and a variety of other coordinate systems. SurvNET calculates the minimum necessary corrections to measured horizontal angles, slope distances and vertical angles in order to fit the desired control. **NOTE:** SurvNET can only process raw field measurements, it is not designed to process bearing or azimuth traverses. If you wish to use SurvNET to process your traverses, you must collect the angles and distances.
- In the 2D/1D model in a state plane coordinate system, a grid factor is computed for each individual line during the reduction. The elevation factor is computed for each individual line if there is sufficient elevation data. If the raw data has only 2D data, the user has the option of defining a project elevation to be used to compute the elevation factor.
- SurvNET supports a variety of map projections and coordinate systems including the New Brunswick Survey Control coordinate system, UTM, and user-defined systems consisting of either a pre-defined ellipsoid or a user defined ellipsoid and one of the following projections, Transverse Mercator, 1-Standard Parallel Lambert Conformal, 2-Standard Parallel Lambert Conformal, Oblique Mercator, and the Double Stereographic projection.
- A full statistical report containing the results of the least squares adjustment is produced and written to the report (.RPT) file. An error report (.ERR) file is created and contains any error messages that are generated during the adjustment.
- Coordinates can be stored into a variety of file types including the Carlson (.CRD) file, C&G (.CRD) file, Simplicity file (.ZAK), Carlson SQLITE (.CRDB), or an Autodesk Land Desktop (.MDB) file types. An ASCII coordinate (.NEZ) file is always created that can be imported into most any mapping/surveying/GIS program. The user has the option to compute unadjusted preliminary coordinates.
- There is an option to compute Traverse Closures during the preprocessing of the raw data. Closures can be computed for both GPS and total station traverses. Closure for multiple traverse loops in the same raw file can be computed.
- When processing Angle-Only records for triangulation, if there is a zenith angle and rod height (zero is a valid rod height), a 3D triangulation will be performed, calculating an elevation of the triangulation point. This is true in both the 3D and 2D/1D models.
- SurvNET can combine GPS vectors and total station data in a single adjustment. GPS Vector files from Leica, Thales, Topcon and Trimble can be input, as well as GPS files in the StarNet format. Additionally GPS vectors can be read from NGS G-files. There is also an option to read the G-file section of an OPUS report.
- SurvNET includes a variety of blunder detection routines. One blunder detection method is effective in detecting if the same point number has been used for two different points. Additionally this blunder detection method is effective in detecting if two different point numbers have been used for the same physical position. This method also flags other raw data problems. Another blunder detection method included in SurvNET is effective in isolating a single blunder, distance or angle in a network. This method does not require that there be a lot of redundancy, but is effective if there is only one blunder in the data set. Additionally, SurvNET includes a blunder detection method that can isolate multiple blunders, distances or angles in a network. This method does require that there be a lot of redundancy in the network to effectively isolate the multiple blunders.
- Other key features include: Differential and Trig level networks and loops can be adjusted using the network least squares program. Geoid modeling is used in SurvNET, allowing the users to choose between a variety of geoid models. The user can alternately enter the project geoid separation. There are description codes to identify duplicate points with different point numbers. The user can specify the confidence interval from 50 to 99 percent.

Least squares is very flexible in terms of how the survey data needs to be collected. Generally speaking, any combination of angles, and distances combined with a minimal amount of control points and/or azimuths are needed. This data can be collected in any order. There needs to be at least some redundancy in the measurements. Redundant measurements are measurements that are in excess of the minimum number of measurements required to determine the unknown coordinates. Redundancy can be created by including multiple GPS and other control points within a

network or traverse. Measuring angles and distances to points in the network that have been located from another setup in the survey creates redundancy. Running additional cut-off traverses or additional traverses to existing control points creates redundancy. Following are some general rules and tips in collecting data for least squares reduction:

- Backsights should be to point numbers. Some data collectors allow the user to backsight an azimuth not associated with a point number. SurvNET requires that all backsights be associated with a point number.
- There has to be at least a minimum amount of control. Control is defined as known points or azimuths. They can be held FIXED or allowed to move a designated amount. The minimum amount of control is either two points or one point and a reference azimuth. You can however have as many control points as you wish. Control points can be entered in either the raw data file or there can be a supplemental control point file containing the control point. Reference azimuths are entered in the raw data file. The control points and reference azimuths do not need to be for the first points in the raw data file. The control points and azimuths can be associated with any point in the network or traverse. The control points do not need to be adjacent to each other. It is permissible, though unusual, to have one control point on one side of the project and a reference azimuth on the other side of the project.
- Some data collectors do not allow the surveyor to shoot the same point twice using the same point number. SurvNET requires that all measurements to the same point use a single point number. The raw data may need to be edited after it has been downloaded to the office computer to insure that points are numbered correctly. An alternative to renumbering the points in the raw data file is to use the 'Pt Number substitution string' feature in the project 'Settings' screen. See the 'Redundant Measurement' section for more details on this feature.
- The majority of all problems in processing raw data are related to point numbering issues. Using the same point number twice for different points, not using the same point number when measuring the same point, misnumbering backsights or foresights, and misnumbering control points are all common problems.
- It is always best to explicitly define the control for the project. A good method is to put all the control for a project into a separate raw (RW5) file. A big source of problems with new users is a misunderstanding in defining their control for a project.
- Some data collectors may have preliminary unadjusted coordinates included with the raw data. These coordinate records should be removed from the raw file. The only coordinate values that should be in the raw file are the control points. Since there is no concept of 'starting coordinates' in least squares there is no way for SurvNET to determine which points are considered control and which points are preliminary unadjusted points. So all coordinates found in a raw data file will be considered control points.
- When a large project is not processing correctly, it is often useful to divide the project into several raw data files and debug and process each file separately as it is easier to debug small projects. Once the smaller projects are processing separately they can be combined for a final combined adjustment.

SurvNET gives the user the option to choose one of two mathematical model options when adjusting raw data, the **3D model** and the **2D/1D model**.

In the process of developing SurvNET, numerous projects have been adjusted using both the 2D/1D model and the 3D model. There are slight differences in final adjusted coordinates when comparing the results from the same network using the two models. But in all cases, the differences in the results are typically less than the accuracy of measurements used in the project. The main difference in terms of collecting raw data for the two different models is that the 3D model requires that rod heights and instrument heights need to be measured, and there needs to be sufficient elevation control to compute elevations for all points in the survey. When collecting data for the 2D/1D model, the field crews do not need to collect rod heights and instrument heights.

2D/1D Model

In the 2D/1D model, raw distance measurements are first reduced to horizontal distances and then optionally to grid distances. Then, a two dimensional horizontal least squares adjustment is performed on these reduced horizontal distance measurements and horizontal angles. After the horizontal adjustment is performed, an optional one-dimensional vertical least squares adjustment is performed in order to adjust the elevations if there is sufficient data to compute elevations. The 2D/1D model is the model that has been traditionally been used in the past by non-geodetic surveyors in the reduction of field data. There are several advantages to SurvNET 's implementation of the 2D/1D model. One advantage is that an assumed coordinate system can be used. It is not necessary to know

geodetic positions for control points. Another advantage is that 3D raw data is not required. It is not necessary to record rod heights and heights of instruments. Elevations are not required for the control points. The primary disadvantage of SurvNET 's implementation of the 2D/1D model is that GPS vector data cannot be used in 2D/1D projects.

In the 2D/1D model, it is allowed to mix 2D and 3D measurements. Elevations will be calculated only if there is enough information in the raw data file to do so. Least squares adjustment is used for elevation adjustment as well as the horizontal adjustment. To compute an elevation for the point, the instrument record must have an HI, and the foresight record must have a rod height, slope distance and vertical angle. If working with .CGR raw data, a 0.0 (zero) HI or rod height is valid. It is only when the field is blank that the record will be considered a 2D measurement. Carlson SurvCE/SurvPC 2.0 or higher allows you to mix 2D and 3D data by inserting a 2D or 3D comment record into the .RW5 file. A 3D traverse must also have adequate elevation control in order to process the elevations. Elevation control can be obtained from the supplemental control file, coordinate records in the raw data file, or elevation records in the raw data file. The "Adjust Elevations" box in the project settings must be checked to adjust the calculated elevations. If it is unchecked, elevations will still be calculated if the 3D data is available, but they will not be adjusted.

3D Model

In the 3D model, raw data is not reduced to a horizontal plane prior to the least squares adjustment. The 3-dimensional data is adjusted in a single least squares process. In SurvNET 's implementation of the 3D model, XYZ geodetic positions are required for control. The raw data must contain full 3D data including rod heights and measured heights of the instrument(s). The user must designate a supported geodetic coordinate system. The main advantage of using the 3D model is that GPS vectors can be incorporated into the adjustment.

SurvNET can also automatically reduce field measurements to State Plane coordinates in either the NAD 83 or NAD 27 coordinate systems. If a grid coordinate system is selected, the grid scale factor is computed for each individual line during the reduction. The elevation factor is also computed for each individual line if there is sufficient elevation data. If the raw data has only 2D data, the user has the option of defining a project elevation to be used to compute the elevation factor.

A full statistical report containing the results of the least squares adjustment is produced and written to the report (.RPT) file. An error report (.ERR) file is created and contains any error messages that are generated during the adjustment.

Coordinates can be written to the following formats, including:

- C&G numeric (*.crd)
- C&G alphanumeric (*.cgc)
- Carlson numeric (*.crd)
- Carlson alphanumeric (*.crd)
- Carlson SQLite (*.crdb)
- MS Access Database (Autodesk Land Desktop) (*.mdb)
- Simplicity (*.zak)
- ASCII P,N,E,Z,D,C (*.nez)

A file with the extension .OUT is always created and contains an ASCII formatted coordinate list of the final adjusted coordinates formatted suitable for printing. Additionally, an ASCII file with an extension of .NEZ containing the final adjusted coordinates in a format suitable for input into 3rd party software that is capable of inputting an ASCII coordinate file.

All brands, product names and trademarks cited herein not licensed to Carlson Software are the property of their respective license holders.

Starting Survnet

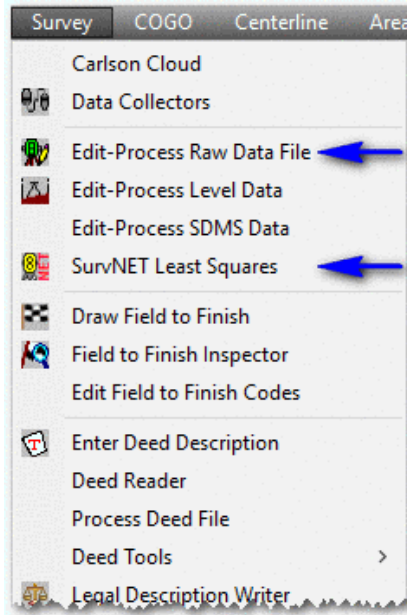
Using SurvNET Dialog Box

The dialog box version of SurvNET offers an interactive graphics user interface which utilizes the CAD drawing editor for graphics while mainlining data entry in a docked dialog box.

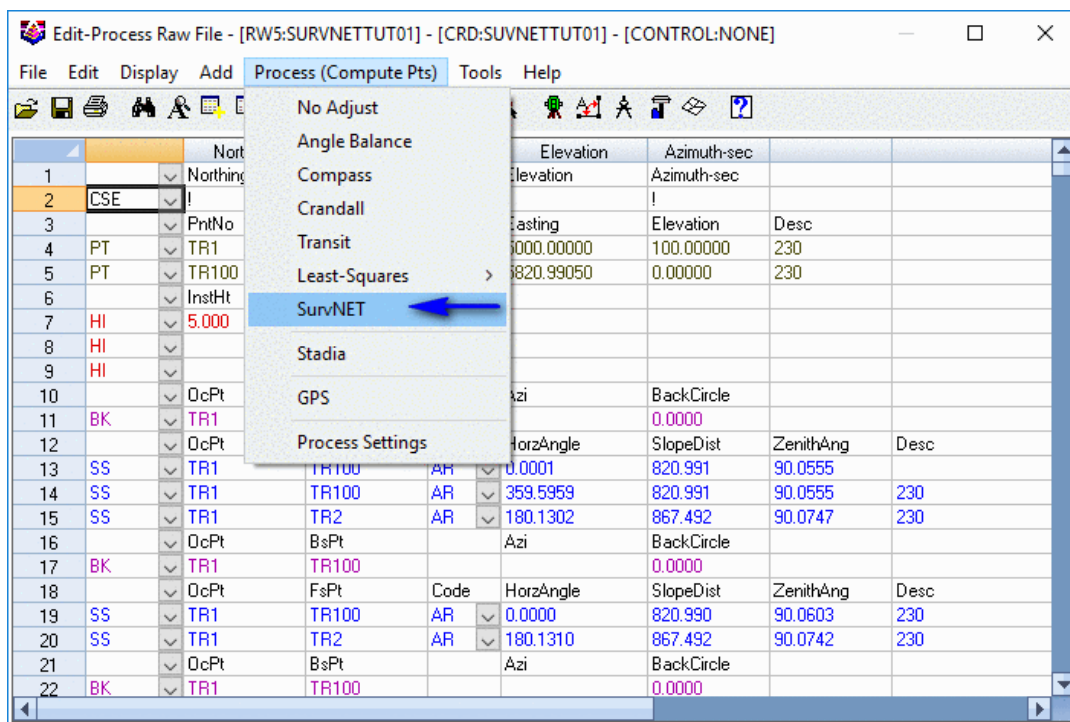
Double-click on the SurvNET icon on the desktop or use the Start menu – Programs (or All Programs) – SurvNET

Running From Carlson

Entry into the SurvNET program is easy. It can be accessed in two different ways. The easiest way to start the program is to select **SurvNET** from the **Survey** menu. The other method is to start SurvNET from within the **Raw Data File** editor. To bring up the Raw Data File editor, select **Edit - Process Raw Data File** from the **Survey** menu (see below).

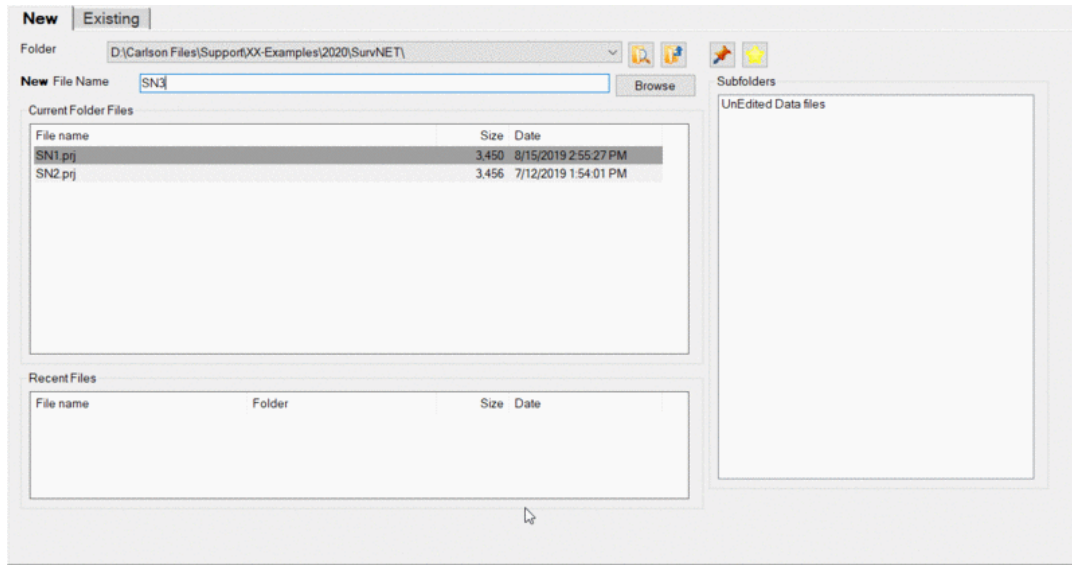


To access SurvNET from within the Carlson Raw Data Editor, choose the **Process (Compute Pts)** menu then the **SurvNET** menu item (see below).

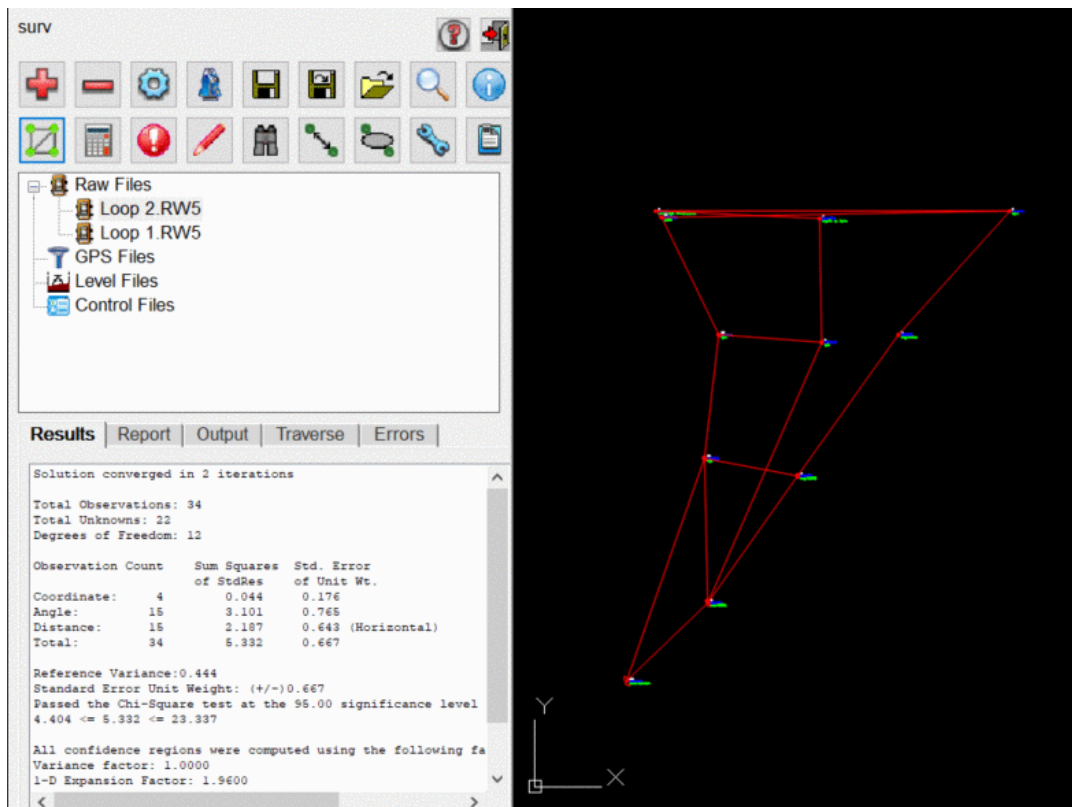


SurvNET Start-up Dialog

The **SurvNET Start-up** dialog is displayed when SurvNET is first started (see below). SurvNET is a project based program. Before performing a least squares adjustment an existing project must be opened or a new project created. This opening dialog box allows the user to open or create a project on start-up.



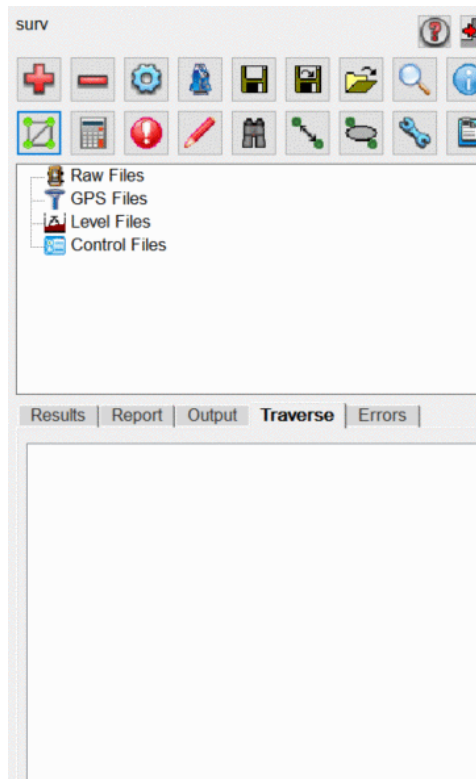
The following is a view of the SurvNET main window after an existing project has been opened and processed:



Note: The various report sub-windows within SurvNET and the SurvNET application itself can all be resized to suit specific screen "real-estate" preferences.

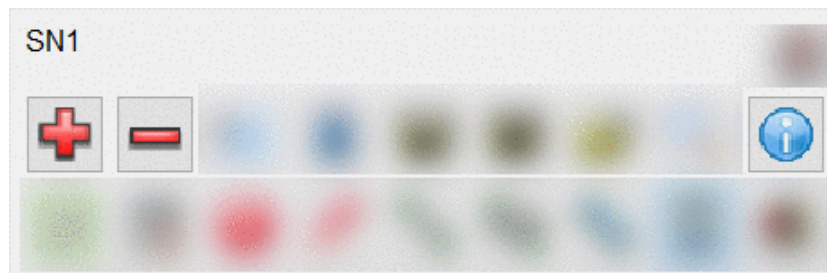
Menu System Overview

The following graphic shows the main SurvNET least squares interface.



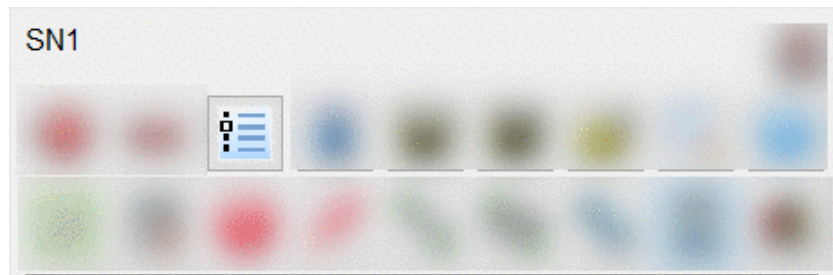
Least squares operations are initiated using the menus summarized below:

Entering and Editing Raw Files



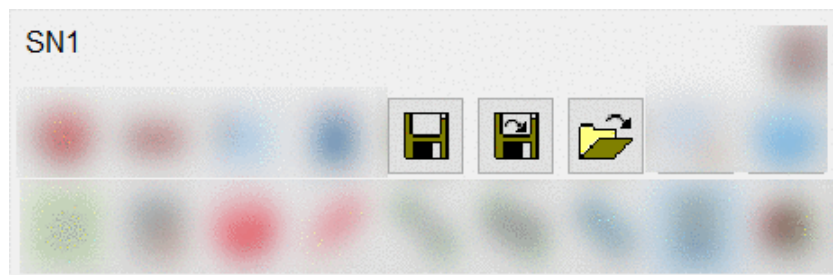
- Add and Remove Raw Files
- Review and Edit Control Points and Measurements
- Editing Raw data
- Assigning Standard Errors

Project Settings



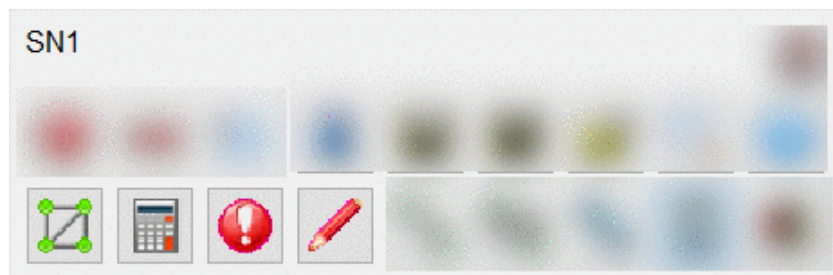
- Coordinate System
- Input File defaults
- Pre-processing
- Default Standard Errors
- Adjustment Standards
- Output Options
- Drawing Settings
- Global Settings

Working with Project Files



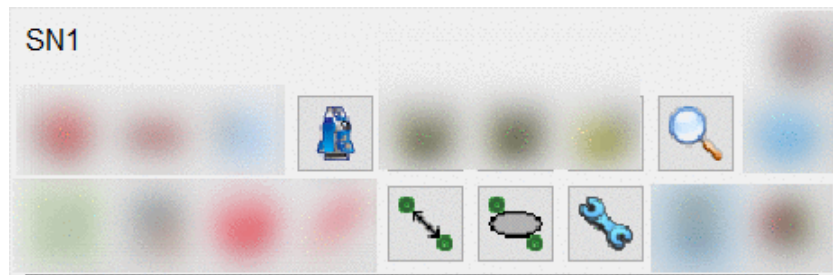
- Open and Create Project Files
- Save Current
- Saveas - copy and create new file

Analyzing and Processing



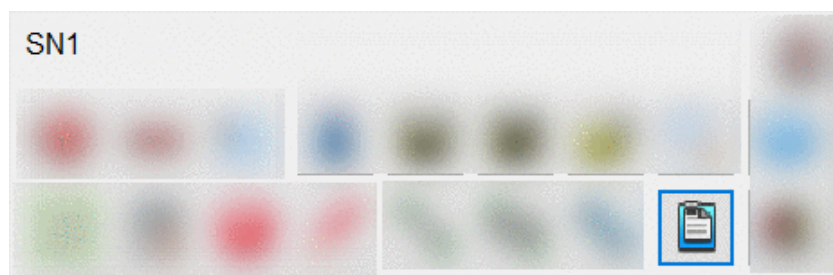
- Process Network
- Pre-process Network
- Blunder Detection
- Draw Network

Tools



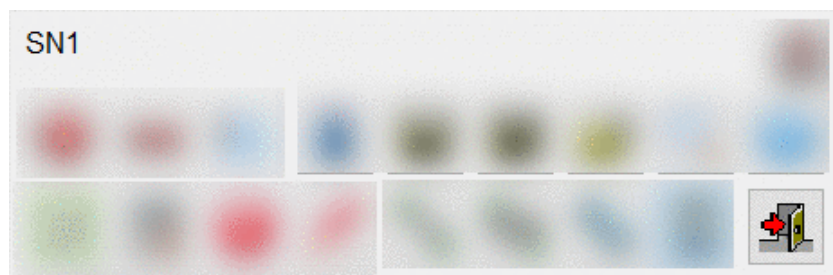
- EDM Calibration
- Point Search
- Inverse
- Error Ellipse
- Conversion Tools

Reports

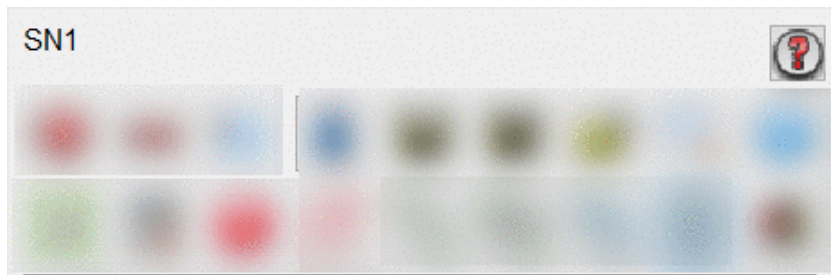


- Main
- Output
- Errors
- Results

Exit - Exits the program



Help - launches the Carlson Help Menu



File Menu

The **File** menu consists of three main buttons.

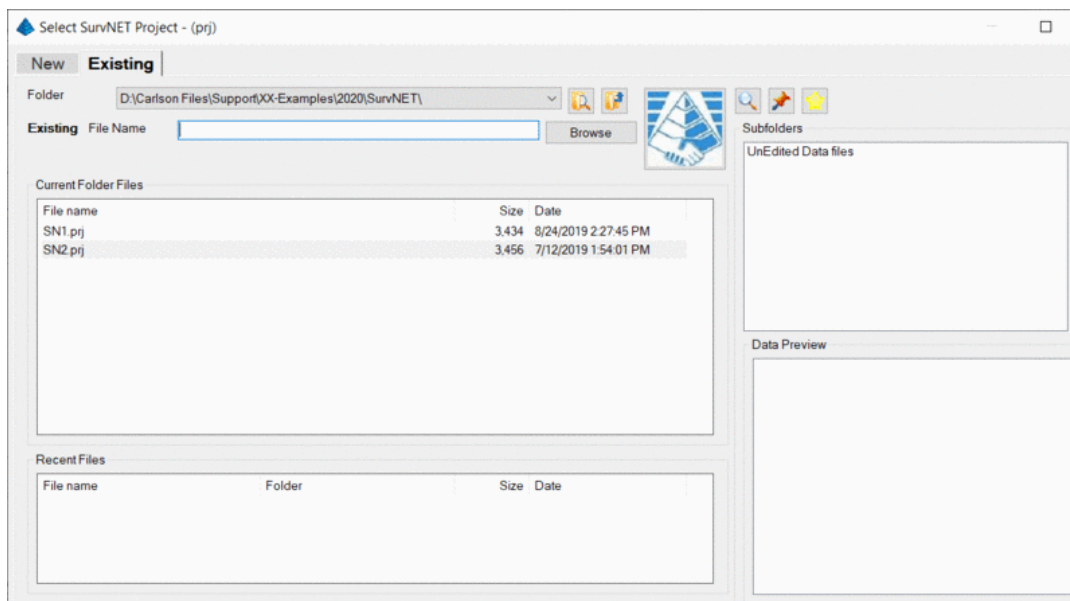


From left to right the are:

Save - saves the current file

Saveas - saves the current file as a new one, essentially making a copy.

Load/New - opens a new dialog box that allows users to select and existing project file or create a new one.

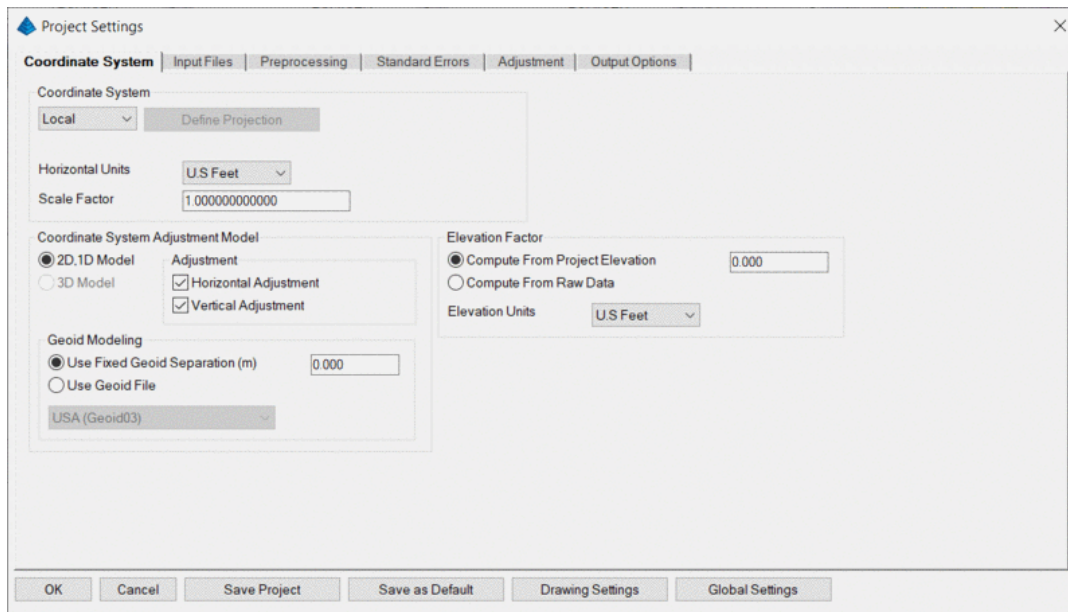


Settings Menu

Selecting the **Settings** menu opens the following menu:

Project Settings

The Project Settings dialog box has six tabbed windows:

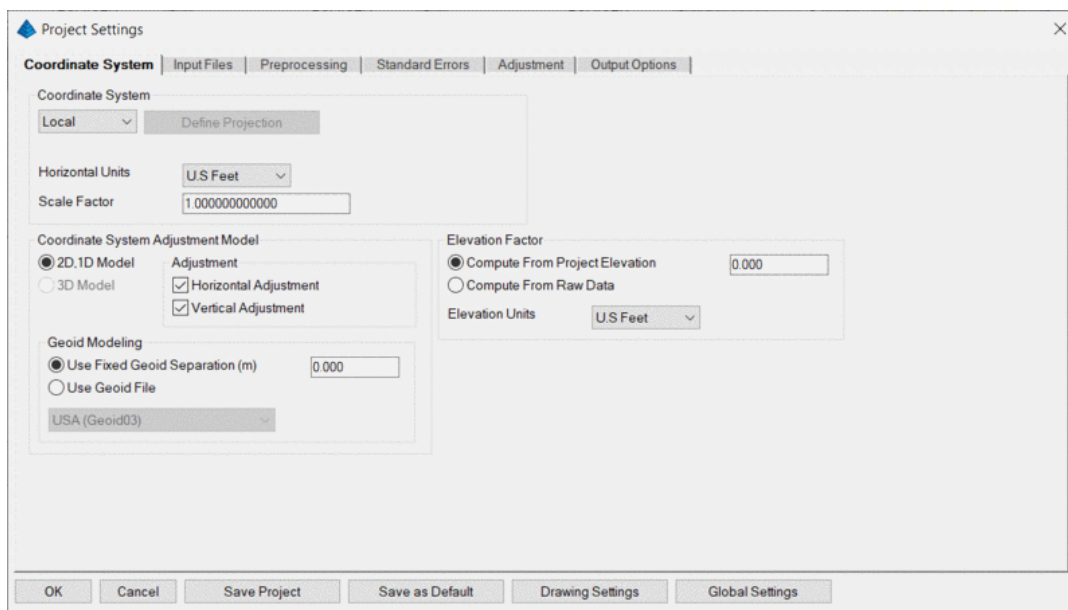


1. Coordinate System
2. Input Files
3. Preprocessing
4. Adjustment
5. Standard Errors

6. Output Options

In addition to the above tabs, there are options to

- Save Project - saves the current project
- Save as Default - saves the current settings as defaults for future projects
- Drawing Settings - opens the Drawing Settings dialog box
- Global Settings - opens the Global Settings dialog box



The Coordinate System tab contains settings that relate to the project coordinate system, the adjustment model and other geodetic settings.

You can select either the 3D model or the 2D/1D mathematical model. If you choose 2D/1D mathematical model, you can choose to only perform a horizontal adjustment, a vertical adjustment or both. In the 3D model, both horizontal and vertical are adjusted simultaneously. The 3D model requires that you choose a geodetic coordinate system. Local, assumed coordinate systems cannot be used with the 3D model. GPS vectors can only be used when using the 3D model.

If using the 2D/1D mathematical model, you can select Local (assumed coordinate system), or a geodetic coordinate system such as State Plane NAD83, State Plane NAD27, UTM, or a user-defined coordinate system as the coordinate system. When using the 3D model you cannot use a local system.

Select the "Horizontal Units" for output of coordinate values (Meters, US Feet, or International Feet). In the 3D model, both horizontal and vertical units are assumed to be the same. In the 2D/1D model, horizontal and vertical units can differ if you select "Compute from Project Elevation". The "Horizontal Unit" setting in this screen refers to the output units. It is permissible to have input units in feet and output units in meters. Input units are set in the Input Files tab.

If you choose SPC 1983, SPC 1927, or UTM, the appropriate zone will need to be indicated. The grid scale factor is computed for each measured line using the method described in Section 4.2 in the publication by: Stem, James E., (1990) *State Plane Coordinate System of 1983*. Rockville, MD: NOAA.

If using the 2D/1D model and you select a geodetic coordinate system, you have a choice as to how the elevation factor is computed. You can choose to either enter a project elevation or you can choose to have elevations factors computed for each distance based on computed elevations. In order to use the "Compute Elevation from Raw Data", all HI's and foresight rod heights must be collected for all points.

If you choose a geodetic coordinate system and are using the 2D/1D model, you will want to select "Project Elevation" if any of your raw data measurements are missing any rod heights or instrument heights. There must be enough information to compute elevations for all points in order to compute elevation factors. For most survey projects, it is sufficient to use an approximate elevation, such as can be obtained from a Quad Sheet for the project elevation.

Geoid Modeling

If you are using either the 3D or the 2D/1D adjustment model using non-local coordinate system, you must choose a geoid modeling method. A project geoid separation can be entered or the grid model can be selected from a list of models. The project must fall within the geographic range of the geoid grid files.

Geoid modeling is used as follows. Entering a 0.0 value for the separation is the method to use if you wish to ignore the geoid separation. In the 2D/1D model, it is assumed that elevations entered as control are entered as orthometric heights. Since grid reduction requires the data be reduced to the ellipsoid, the geoid separation is used to compute ellipsoid heights. The difference between using geoid modeling and not using geoid modeling or using a project geoid separation is insignificant for most surveys of limited extents.

In the 3D model, it is also assumed that elevations entered as control are orthometric heights. Since the adjustment is performed on the ellipsoid, the geoid separation is used to compute ellipsoid elevations prior to adjustment. After the adjustment is completed, the adjusted orthometric elevations will be computed from the adjusted ellipsoid elevations and the computed geoid separation for each point.

If you choose one of the geoid models from the list, the geoid separations are computed by interpolation with data points retrieved from geoid separation files. The geoid separation files should be found in the primary the installation directory. Grid files have an extension of .grd. These files can be downloaded from the Carlson website (www.carlsonsw.com). If you pick a geoid model that is not on your computer, SurvNET will automatically connect to the Carlson website and download the specific geoid file.

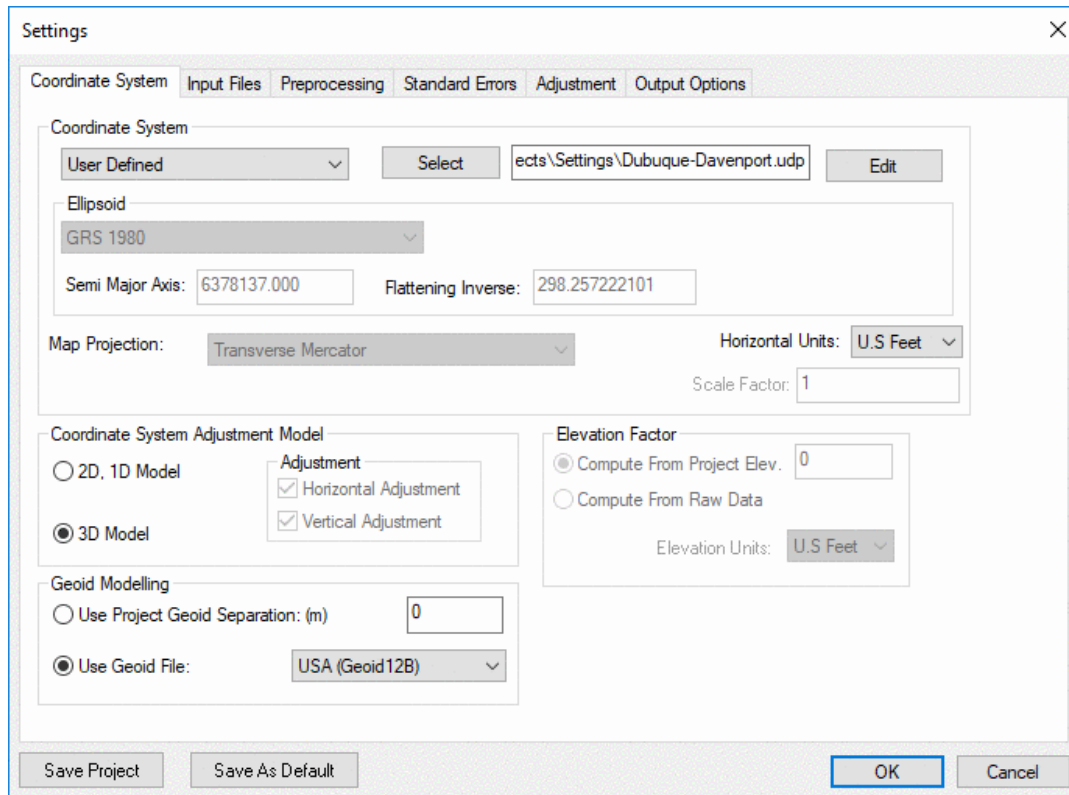
If you choose to enter a project geoid separation, the best way to determine a project geoid separation is by using the NGS on-line Geodetic Toolkit. Enter a latitude and longitude of the project midpoint and the program will output a project separation.

Working With User-defined Coordinate Systems

SurvNET allows the creation of user-defined geodetic coordinate systems (UDP). The ability to create user-defined coordinate system allows the user to create geodetic coordinate systems based on the supported projections that are not explicitly supported by SurvNET. A SurvNET user-defined coordinate system consists of an ellipsoid, and a map projection. The ellipsoid can be one of the explicitly supported ellipsoids or a user-defined ellipsoid. The supported map projections are:

- Transverse Mercator
- Lambert Conformal Conic with 1 standard parallel
- Lambert Conformal Conic with 2 standard parallels
- Oblique Mercator
- Double Stereographic projection

User-defined coordinate systems are created, edited, and attached to a project from the Project Settings 'Coordinate System' dialog box. To attach an existing UDP file, *.udp, to a project use the 'Select' button. To edit an existing UDP file or create a new UDP file use the 'Edit' button.



The following dialog box is used to create the user-defined coordinate system. The ellipsoid needs to be defined and the appropriate map projection and projection parameters need to be entered. The appropriate parameter fields will be displayed depending on the projection type chosen.

Note: *Grid Origin* values for both the Northing and Easting values need to be input in Metric values!

Test - Use the 'Test' button to enter a known latitude and longitude position to check that the UDP is computing correct grid coordinates. Following is the test UDP dialog box. Enter the known lat/long in the top portion of the dialog box then press 'Calculate' and the computed grid coordinates will be displayed in the 'Results' list box:

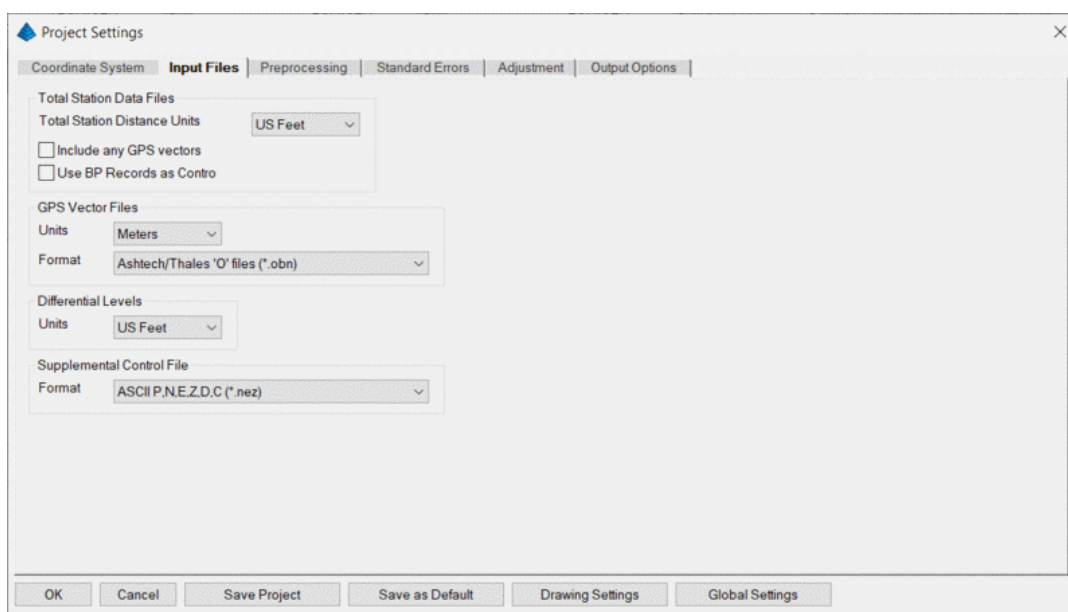
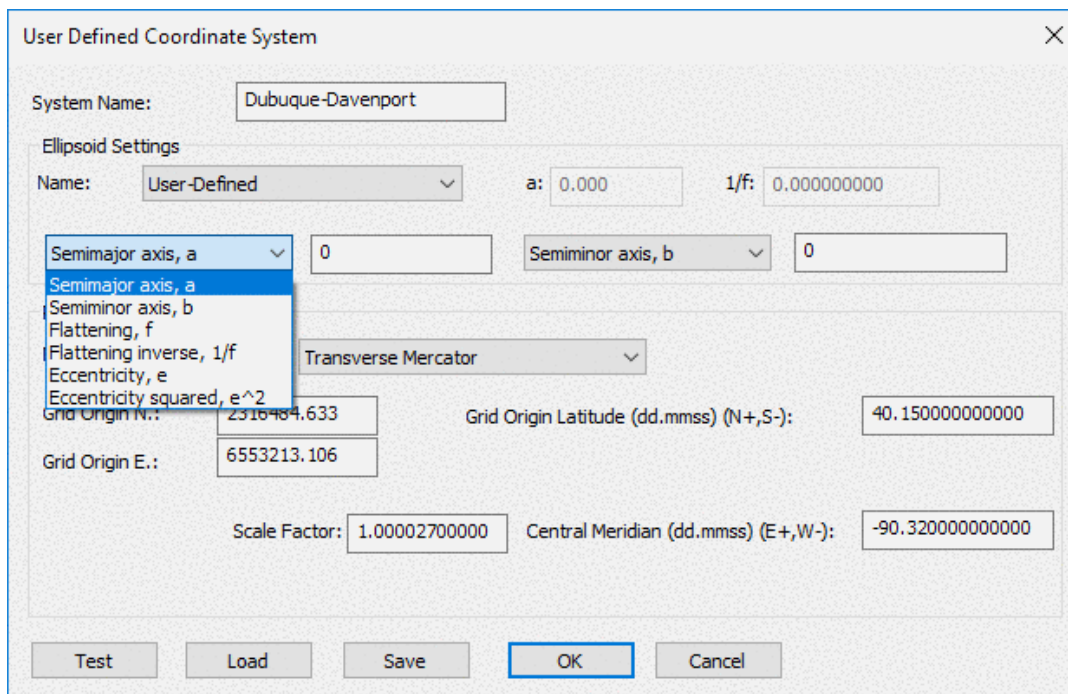
Load - Use the 'Load' to load the coordinate system parameters from an existing .UDP file.

Save - Use the 'Save' button to save the displayed UDP. The 'Save' button prompts the user to enter the .UDP file name.

OK - Use the 'OK' button to save the UDP using the existing file name and return to the 'Coordinate System' dialog box.

Cancel - Use the 'Cancel' button to return to the 'Coordinate System' dialog box without saving any changes to the .UDP file.

If you need to define an ellipsoid, choose the 'User-Defined' ellipsoid option. With the user-defined ellipsoid you will then have the option to enter two of the ellipsoid parameters.



This dialog box is used to set the default parameters for the **Input Raw** files that will be used for processing the data.

Note that if you are processing Carlson RW5 total station files, you have the option of including GPS vectors that can be collected by SurvCE/SurvPC and stored in the RW5 file. Alternatively, you can select the RW5 file in the GPS vector section. **If the "Include any GPS vectors" option in the Total Station Data Files section is checked, you will have the option to use the Base Point records in the RW5 file as control.** If "Use BP records as control" is NOT checked, or the RW5 file is selected in the GPS vector section, you must define the control in either the RW5 file as POINT records or in a Supplemental Control File.

GPS Vector Files: GPS vector files can be entered and processed. Both GPS vector files and total station raw files can be combined and processed together. **You must have chosen the 3D mathematical model in the Coordinate System tab in order to include GPS vectors in the adjustment.**

A variety of popular GPS vector file formats are supported, including (but not necessarily limited to):

- **ASCII (StarNet):** See below for more information on StarNet format. These files typically have .GPS exten-

sions.

- **Ashtech/Thales 'O' files:** Typically have .obn extensions and are binary files.
- **Carlson .RW5:** (containing GPS vectors)
- **GeoLab IOB**
- **LandXML (*.xml)**
- **Leica:** Leica files are ASCII files.
- **NGS G-File**
- **NGS G-File from OPUS report**
- **Topcon (.tvf):** Topcon .tvf files are ASCII files.
- **Topcon (.xml):** Topcon also can output their GPS vectors in XML format which is in ASCII format.
- **TDS (.raw):** TDS raw data file with vector information
- **Trimble Data Exchange Format (.asc):** These files are in ASCII format
- **Trimble data collection (.dc):** These files are ASCII.
- **Trimble LandXML (*.jxl)**

The following is a typical vector record in the StarNet ASCII format. GPS vectors typically consist of the 'from' and 'to' point number, the delta X, delta Y, delta Z values from the 'from' and 'to' point, with the XYZ deltas being in the geocentric coordinate system. Additionally the variance/covariance values of the delta XYZ's are included in the vector file.

```
.GPS WEIGHT COVARIANCE
C PRS34452 1305780.345005 -4667085.299019 4132689.544939 0.005000 0.005000 0.005000 --MON
C UCNJ 1305780.345005 -4667085.299019 4132689.544939 0.00000100 0.00000100 0.00000100 --MON
G1 UCNJ-1000 8399.71318061 -4742.15645068 -8036.07224424 --MNS
G2 3.939428e-006 2.474560e-005 1.160301e-005
G3 3.924536e-006 -3.360765e-006 -1.028503e-006
G1 UCNJ-1001 8328.15569476 -4796.59445569 -8072.25948922 --MNS
G2 9.596618e-007 1.687749e-005 1.936038e-006
G3 1.176891e-007 -8.668020e-009 -4.798408e-006
```

The first line defines what values are in the G2 and G3 records. It can be either GPS WEIGHT COVARIANCE (G2 is Variance, G3 is Covariance), or GPS WEIGHT STDERRCORRELATION (G2 is standard error, G3 is standard error correlation). This line is optional; the default is COVARIANCE.

The next two lines are coordinate control records. These records are also optional. If used, they must be Geocentric Coordinates (XYZ) in metric units. The format is as follows:

```
C PointName X Y Z StdErrX StdErrY StdErrZ --PointDescription
```

The standard errors and point description are optional.

The GO record is a comment.

The G1 record includes the 'from' and 'to' points and the delta X, delta Y, and delta Z in the geocentric coordinate system and an optional description of the rover point.

The G2 record is the variance (or standard error) of X, Y, and Z. The G3 record contains the covariance (or standard error correlation) of XY, ZX, and ZY. Most all GPS vector files contain the same data fields in different formats.

Use the 'Add' button to insert GPS vector files into the list. Use the 'Remove' button to remove GPS vector files from the list. All the files in this list will be used in the least squares adjustments. All the GPS files in the list must be in the same format. If the GPS file format is ASCII you have the option to edit the GPS vector files. The Edit option allows the editing of any of the ASCII GPS files using Notepad. Typically, only point numbers or point descriptions would be edited. The variance/covariance values are used to determine the weights that the GPS vectors will receive during the adjustment and are not typically edited.

Level Raw Files: Differential and Trig level files can be entered and processed. Differential or Trig Level raw files have a .TLV extension and are created using the Carlson Level Editor. The Carlson Level Editor has several Import options that allow you to bring in levels from different formats (*e.g.* Leica, Trimble, Topcon, *etc.*). You can process level data in the same project with the traverse and GPS vector data but the vertical measurements in the

traverse/vectors will be part of the adjustment. If you want to use the elevations calculated ONLY from the level data, you must process it in a separate project after your traverse/vector data has been adjusted. The elevations will be written to the Output Coordinate file. Make sure your point numbers in the level data match the points in the traverse/vector data.

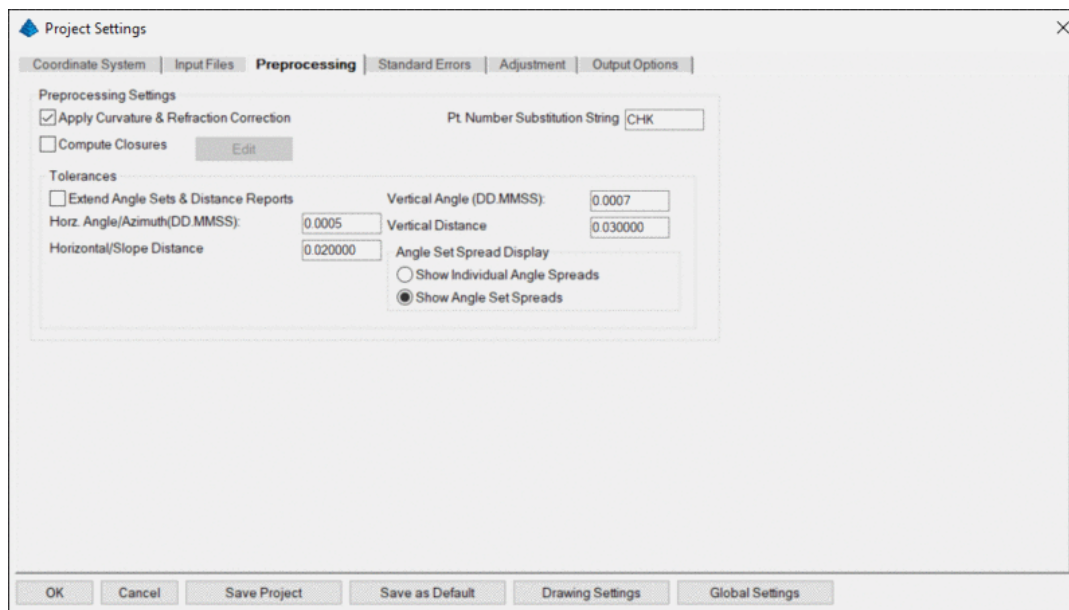
Supplemental Control File: The supplemental control file option allows the user to designate an additional coordinate file to be used as Control. The supplemental control file can be from a variety of different file types, including (but not necessarily limited to):

- **C&G or Carlson numeric (.CRD) files**
- **C&G Alphanumeric coordinate files (*.cgc)**
- **Carlson Alphanumeric coordinate files (*.crd)**
- **Autodesk Land Desktop (*.mdb)**
- **Simplicity coordinate files (*.zak)**
- **ASCII (.NEZ) file**
- **ASCII latitude and longitude (3D model only)**
- **CSV ASCII NEZ with std. errors** (only external control file that allows you to assign standard errors to specific points)
- **SDMS (.ctl) control file**
- **ASCII Geocentric (.xyz)** (Geocentric coordinates XYZ in metric units)

Note: For additional information on file format pertaining to the file formats above, refer to the Supplemental Control Files section of the documentation.

Note: You will not be allowed to use the same file for supplemental control points and for final output. Least Squares considers all points to be measurements. If the output file is also used as a supplemental control file, then after the project has been processed all the points in the project would now be in the control file and all the points in the file would now be considered control points if the project was processed again. The simplest and most straight-forward method to define control for a project is to include the control coordinates in a raw data file.

The Preprocessing tab contains settings that are used in the preprocessing of the raw data.



Apply Curvature and Refraction Corrections: Check this box if you wish to have the curvature refraction correction applied in the 2D/1D model when reducing the slope distance/vertical angle to horizontal distance and vertical distance. Curvature/refraction primarily impacts vertical distances.

Pt. Number Substitution String: This option is used to automatically renumber point names. Some data collectors do not allow the user to use the same point number twice during data collection. In Least Squares, it is common to collect measurements to the same point from different locations. If the data collector does not allow you to use

the correct point number, this option can be used to automatically renumber these points during processing. For example you could enter the string '=' in the Pt. Number Substitution String. Then if you shot point 1 but had to call it something else such as 101, you could enter '=1' in the description field and during preprocessing point 101 would be renumbered as point '1'.

Compute Closures: Traditional traverse closures can be computed for both GPS and total station traverses. This option has no effect on the computation of final least squares adjusted coordinates. This option is useful for surveyors who, due to statutory requirements, are still required to compute traditional traverse closures and for those surveyors who still like to view traverse closures prior to the least squares adjustment. This option is used to specify a previously created Traverse Closure file.

Tolerances: When sets of angles and/or distances are measured to a point, a single averaged value is calculated for use in the least squares adjustment. You may set the tolerances so that a warning is generated if any differences between the angle sets or distances exceed these tolerances. Tolerance warnings will be shown in the report (.RPT) and the (.ERR) file after processing the data.

If you check the **Extended Angle Sets & Distance Report** option, all the sets will be shown with the difference between the high angle and low angle, and the difference between the high distance and low distance. If this option is not checked ONLY the sets that exceed the tolerances will be shown in the report.

Horz./Slope Dist Tolerance: This value sets the tolerance threshold for the display of warnings if the difference between highest and lowest horizontal distance exceeds this value. In the 2D model it is the horizontal distances that are being compared. In the 3D model it is the slope distances that are being compared.

Vert. Dist Tolerance: This value sets the tolerance threshold for the display of a warning if the difference between highest and lowest vertical difference component exceeds this value (used in 2D model only).

Horz. Angle Tolerance: This value sets the tolerance threshold for the display of a warning if the difference between the highest and lowest horizontal angle exceeds this value.

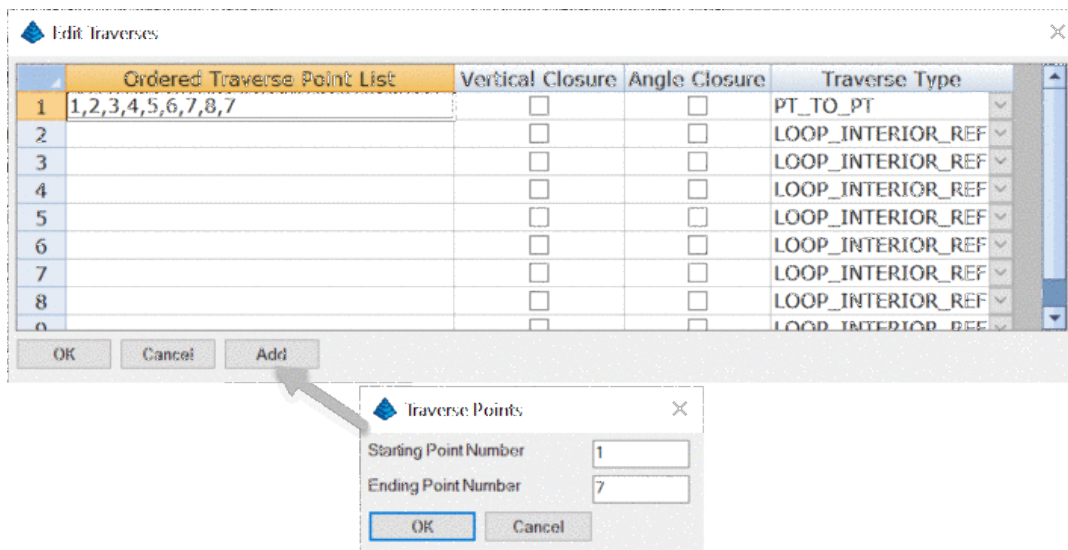
Vert. Angle Tolerance: This value sets the tolerance threshold for the display a warning if the difference between the highest and lowest vertical angle exceeds this value (used in 3D model only).

Angle Set Spread Display: You can choose to see the splits of individual angles in each set (2 per set), or just the splits of all the sets combined (high / low angle of all the sets combined).

Traverse Closure Files

Some statutes and jurisdictions still require the computation of traditional **Traverse Closures**. SurvNET gives the surveyor the ability to compute the closures of multiple traverses within a project as part of the preprocessing of the project raw data. Closures for single or multiple traverses can be computed for a single project. Additionally, GPS closures can be computed for GPS loops. To compute closures you must first create a "Closure File" (.CLS). Closure Files define the type of traverse loops that are to be computed and an ordered list of the point numbers comprising the traverse. Since the raw data for SurvNET is not expected to be in any particular order, it is required that the user must specify the points and the correct order of the points in the traverse loop. Both GPS and angle/distance traverses can be defined in a single traverse closure file.

To add a traverse closure file, select Edit from the dialog box.



1. **Add** - This command permits you to automatically add a traverse from the row file, or manually enter the order of the traverse points.

First, enter the point sequence which defines the traverse in the "edit control" portion of the dialog box. Toggle the check boxes to set whether Vertical Closure and/or Angle Closures need to be computed. Then choose what traverse type is being entered. When the values are correct, press the **Save** button and the traverse will be entered into the Traverse Closure file. Add additional traverse information via the "Record" button icon located at the upper left (you may also remove a recorded traverse via the Delete button icon also located in the upper left portion of the dialog box).

If you need to edit one of the traverses, mouse-click the traverse to be edited. Make the appropriate edit(s), then click the **OK** button to save the changes.

As an example of traverse point entry, points can be entered in the form:

10, 23, 30-35, 45, 23, 10

A comma separates the point numbers. You can select a range (e.g. 30-35) when the points are sequential.

NOTE: You must start with the first backsight point number and end with the last foresight point number for a loop traverse.

For example, if you have a simple loop traverse with angle closure using points 1, 2, 3 and 4, it will be entered as "4,1,2,3,4,1" (or 4,1-4,1) where 1 is the first occupied point and 4 is the initial backsight.

You can turn the "Angle Closure" ON or OFF. If the angle closure is ON, you will be shown the total angular error and error per angle point. If the final closing angle was not collected, you can turn "Angle Closure" OFF and only the linear closure will be computed.

You can turn the "Vertical Closure" ON or OFF. If the vertical closure is ON, you will be shown the total vertical distance closure.

In order to calculate the traverse closure, you must select the **Traverse Type**. It can be:

- **Pt. to Pt. Trav.** - A Point-to-Point Traverse is a traverse that starts at a set of known coordinates and ends at another known coordinate. This option assumes you start from **two** control points and:
 1. tie into one control point if only a linear closure is desired, or,
 2. tie into two control points if an angle closure is desired.

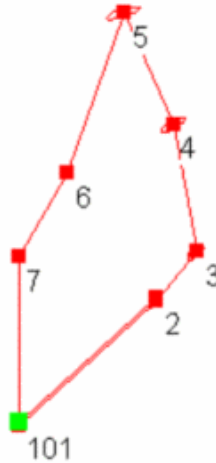
The first backsight distance and last foresight distance are not used in computing the linear closure. Following is an example:

100,101,2-5 (angle closure example collected)

In the above Point-to-Point list, Pt. 100 is the starting backsight point, Pt. 101 is the starting instrument point. Pt. 4 is (would be) the ending instrument point and the foresight to the angle closure point is point 5. If a closing angle was not collected, the list would look as follows:

100,101,2-4

- **Loop Trav., Int. Az. Ref.** - A closed loop traverse that begins by backsighting the last interior point on the traverse. Following is an example:

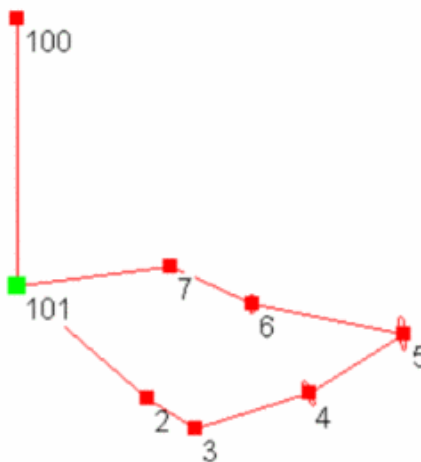


7,101,2-7,101

In the above example, closed loop with angle balance list, point 7 is the backsight point and point 101 is the first occupied point. If the closing angle 6-7-101 was not collected, the list would be entered as follows:

7,101,2-7

- **Loop Trav., Ext. Az. Ref.** - A closed loop traverse that begins by backsighting an exterior point (point not on the traverse):

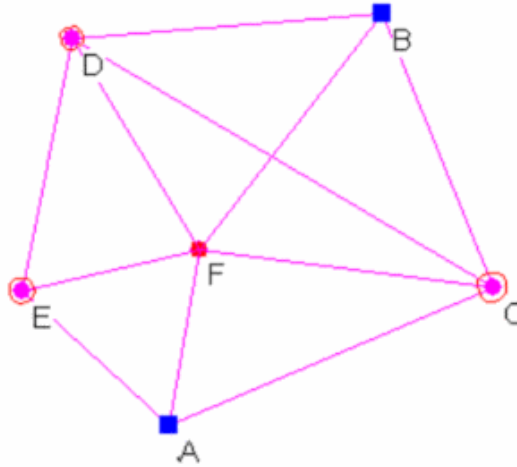


100,101,2-7,101,100

In the above example loop with exterior reference and angle balance list, point 100 is the backsight point and point 101 is the first occupied point. If the closing angle 7-101-101 was not collected, the list would be entered as follows:

100,101,2-7,101

- **GPS Loop Closure** - GPS loop closures can be computed using this option



A, E, F, A

In the above example GPS loop, closure will be computed from the GPS loop going from A-E-F-A.

- **GPS Point to Point Closure** - GPS Point to Point closures can be computed using this option:

A, E, D, B

In the above example the closure will be computed from the GPS traverse going from A-E-D-B. The starting and ending points **MUST** be control points.

When the data is processed, the Closure reports will appear in the RPT and ERR files. You will notice that two Closures are shown, one with no angle balance and one with angle balance.

Following is an example of a closed loop traverse report:

Traverse Closures

=====

Traverse points:

103-118,43-44

Traverse starting and ending on different points;

Compute angle closure.

Compute vertical closure.

BS	IP	FS	Angle	FS H. Dist.	FS V. Dist.
103	104	105	173-07'48.5''	310.4921	-7.7483
104	105	106	167-48'21.5''	253.4875	5.6291
105	106	107	200-52'46.0''	381.4923	8.4877
106	107	108	149-09'05.5''	410.5476	-16.6830
107	108	109	080-42'36.5''	245.5731	9.4221
108	109	110	174-21'17.5''	175.3848	-5.6971
109	110	111	201-42'21.5''	367.0019	-11.8161
110	111	112	171-52'54.5''	237.7809	7.5346
111	112	113	192-32'53.5''	368.8402	-7.0329
112	113	114	171-30'59.0''	338.0028	-19.1945
113	114	115	184-54'03.5''	344.5010	16.3157
114	115	116	149-20'19.5''	353.8460	7.5562

115	116	117	202-19'01.5''	390.1123	-9.9180
116	117	118	112-36'32.0''	293.9935	2.0060
117	118	43	146-06'36.5''	411.3680	-7.7112
118	43	44	270-04'01.5''		

Closing Az: S 47-39'47.8''W
 Computed Closing Az: S 47-39'51.3''W
 Total angular error: 000-00'03.5''
 Angular error per point: 000-00'00.2''
 Correct Ending Coordinates, North: 1400952.0140 East: 2241884.7010
 Ending Coordinates, North: 1400951.7936 East: 2241884.8160
 Error, N: -0.2204 E: 0.1150 Total: 0.2486 Brg: N 27-33'06.7''W
 Distance Traversed: 4882.4241 Closure: 1: 19643
 Correct Ending Elevation: 948.1710
 Ending Elevation: 948.1203
 Elevation Error: -0.0507

Closure After Angle Adjustment

103	104	105	173-07'48.3''	310.4921	-7.7483
104	105	106	167-48'21.3''	253.4875	5.6291
105	106	107	200-52'45.8''	381.4923	8.4877
106	107	108	149-09'05.3''	410.5476	-16.6830
107	108	109	080-42'36.3''	245.5731	9.4221
108	109	110	174-21'17.3''	175.3848	-5.6971
109	110	111	201-42'21.3''	367.0019	-11.8161
110	111	112	171-52'54.3''	237.7809	7.5346
111	112	113	192-32'53.3''	368.8402	-7.0329
112	113	114	171-30'58.8''	338.0028	-19.1945
113	114	115	184-54'03.3''	344.5010	16.3157
114	115	116	149-20'19.3''	353.8460	7.5562
115	116	117	202-19'01.3''	390.1123	-9.9180
116	117	118	112-36'31.8''	293.9935	2.0060
117	118	43	146-06'36.3''	411.3680	-7.7112
118	43	44	270-04'01.3''		

Closing Az: S 47-39'47.8''W
 Computed Closing Az: S 47-39'47.8''W
 Total angular error: 000-00'00.0''
 Angular error per point: 000-00'00.0''
 Correct Ending Coordinates, North: 1400952.0140 East: 2241884.7010
 Ending Coordinates, North: 1400951.7739 East: 2241884.8363
 Error, N: -0.2401 E: 0.1353 Total: 0.2756 Brg: N 29-24'26.1''W
 Distance Traversed: 4882.4241 Closure: 1: 17715

Following is an example of a GPS loop closure report:

Traverse Closures

=====

GPS Loop Points:

A,E,F,A

GPS Loop Closure;

Misclosure, X: -0.0323 Y: -0.0162 Z: -0.0105
 Closure error: 0.0376 Perimeter: 20229.3858
 Precision: 1:537594

GPS Loop Points:

C,F,D,B,C

GPS Loop Closure;

Misclosure, X: -0.0121 Y: -0.0101 Z: 0.0002
 Closure error: 0.0158 Perimeter: 41332.9807
 Precision: 1:2622216

GPS Loop Points:

F,D,B,F

GPS Loop Closure;

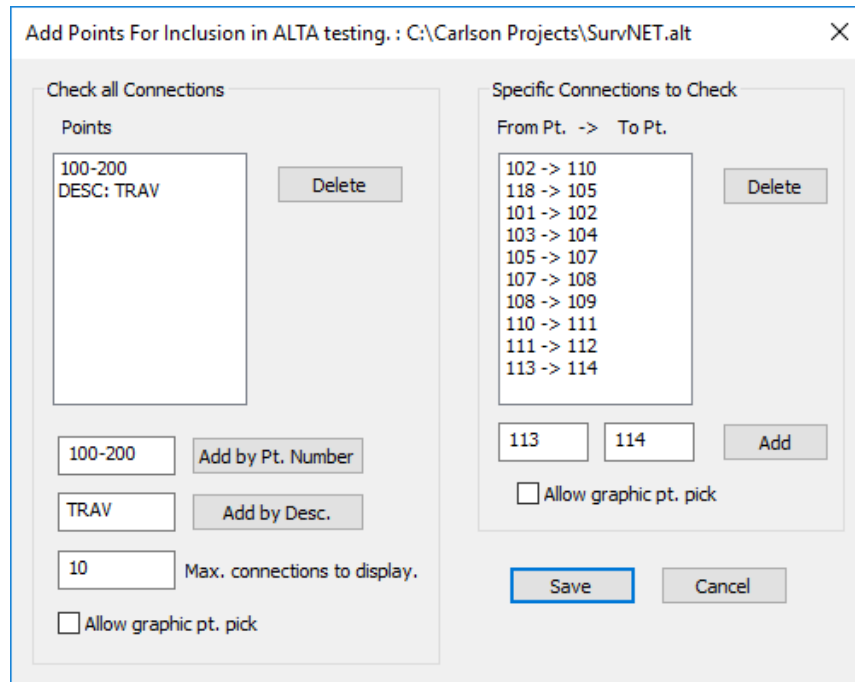
Misclosure, X: -0.0022 Y: -0.0044 Z: 0.0097
 Closure error: 0.0109 Perimeter: 30814.5047
 Precision: 1:2833226

ALTA Tolerance Reports

SurvNET provides the ability to generate reports that give the surveyor the information needed to determine if the survey is within ALTA positional tolerances. It is required that the user define which points are to be included in the ALTA testing. The points to be included for ALTA testing are defined in an .ALT file.

There are two options in the **File** menu that are used to create and edit the ALTA files:

1. **Open ALTA, Rel. Err. Ellipse File** - This command permits you to browse for and open a previously created ALTA File for editing. Once opened, it displays the dialog box below.
2. **New ALTA, Rel. Err. Ellipse File** - This command provides you the ability to create a named ALTA File in a folder of your choosing. After supplying a file name to create, the following dialog box is displayed.



The above dialog box allows the user to define the points to be included in the ALTA report processing.

Notice that you can enter points based on Descriptions in the left-hand list box. For example, if you wished to check connections between all points with TP, EIP, MON descriptions, enter the descriptions in the edit field and press the **Add by Desc.** button. If TP, EIP, and MON represented Traverse Points, Existing Iron Pipes and Monuments (respectively), then ALTA testing would be performed on those point types.

After you have created the .ALT point file, you need to set some additional **Project Settings**. The Adjustment tab within the Project Settings, has a **Relative Error Ellipses/ALTA Report** section where the ALTA report settings are located. These settings define the ALTA Tolerances, specify the .ALT file to be used, and define the type of reporting to be generated.

Note: The **Rel. Err. Points File** check box must be checked, and an .ALT file must be chosen to get an ALTA report. The .ALT file defines which points will be included in the ALTA reporting. Also note typical ALTA specifications define their positional standard as 0.07 plus 50 PPM.

Sample ALTA Report

There are two sections in the .RPT file created through the ALTA reporting. The following report shows the sections of the ALTA report generated by the data in the dialog box.

The first section of the report displays the *Relative Error Ellipses* between points in the **Specific Connection** list. All the connections will be displayed whether they pass or fail the ALTA certification. The point sequences used in this section come from the list on the right-hand side of the above dialog box.

The second section of the report performs an ALTA tolerance test on the points in the "Check all Connections" list. Every possible connection between the points listed will be checked. In this section, only the worst connections will be shown. The number of connections shown is determined by the "Max. connections to display" value in the above dialog box. The point sequences used in this section come from the list on the left-hand side of the above dialog box.

An asterisk (*) will be placed beside each connection shown in the report that **does not pass** based on the confidence interval, tolerance and PPM (parts per million) settings in the Adjustment section of the Project Settings.

The following is a sample ALTA report:

Relative Error and ALTA Tolerances

=====

SPECIFIC CONNECTIONS: Tolerance of 0.070 + 50 PPM. at the 95% CI.

Sta.	Sta.	Dist.	Actual Semi Major	Allowable Semi Major	Ratio Actual/Allowable	Semi Minor	Max. Err.
27	500	204.5030	0.0793	0.0802	0.9890	0.0588	S 85-06' 3"
500	502	66.8572	0.1132	0.0733	1.5432	0.0842	S 86-05' 0"
34	36	237.9748	0.0731	0.0819	0.8920	0.0731	N 00-00' 0"

ALL CONNECTIONS: Tolerance of 0.070 + 50 PPM. at the 95% CI.

All possible connections between the following points were checked:

505, 506, 507, 508, 509, 510, 511, 512, 513, 514, 515, 516, 517, 518, 519, 520, 521, 522, 523, 524, 525, 550, 551, 552, 553, 554, 557, 558, 559, 560

1225 connections tested, the 10 largest relative error ellipses will be shown:

Sta.	Sta.	Dist.	Actual Semi Major	Allowable Semi Major	Ratio Actual/Allowable	Semi Minor	Max. Err.
506	556	806.5402	1.0818	0.1103	9.8054	0.2586	S 86-37' 4"
507	556	827.2364	1.0832	0.1114	9.7268	0.2446	S 86-37' 0"
505	556	818.7994	1.0779	0.1109	9.7158	0.2386	S 86-48' 2"
508	556	854.9436	1.0836	0.1127	9.6108	0.2477	S 86-38' 3"
509	556	880.6338	1.0848	0.1140	9.5129	0.2489	S 86-45' 3"
521	556	798.2729	1.0387	0.1099	9.4500	0.2318	S 87-22' 5"
512	556	793.0518	1.0334	0.1097	9.4245	0.2127	S 87-21' 3"
517	556	907.2084	1.0856	0.1154	9.4106	0.2379	S 87-07' 3"
510	556	918.6572	1.0861	0.1159	9.3682	0.2525	S 86-52' 4"
516	556	935.1194	1.0885	0.1168	9.3228	0.2426	S 87-22' 2"

*** All connection combinations passed ***

The first part of the report labeled "Specific Connections" will show all selected connections whether they passed or failed. If a connection failed, an asterisk (*) will be placed at the end of the line.

The second part of the report, labeled "All Connections" will only show the connections that "failed" (we chose to see the worst 10).

If all the connections pass in the **Specific Connection** section, you will also see this message in the report:

***** All Specific Connections passed *****

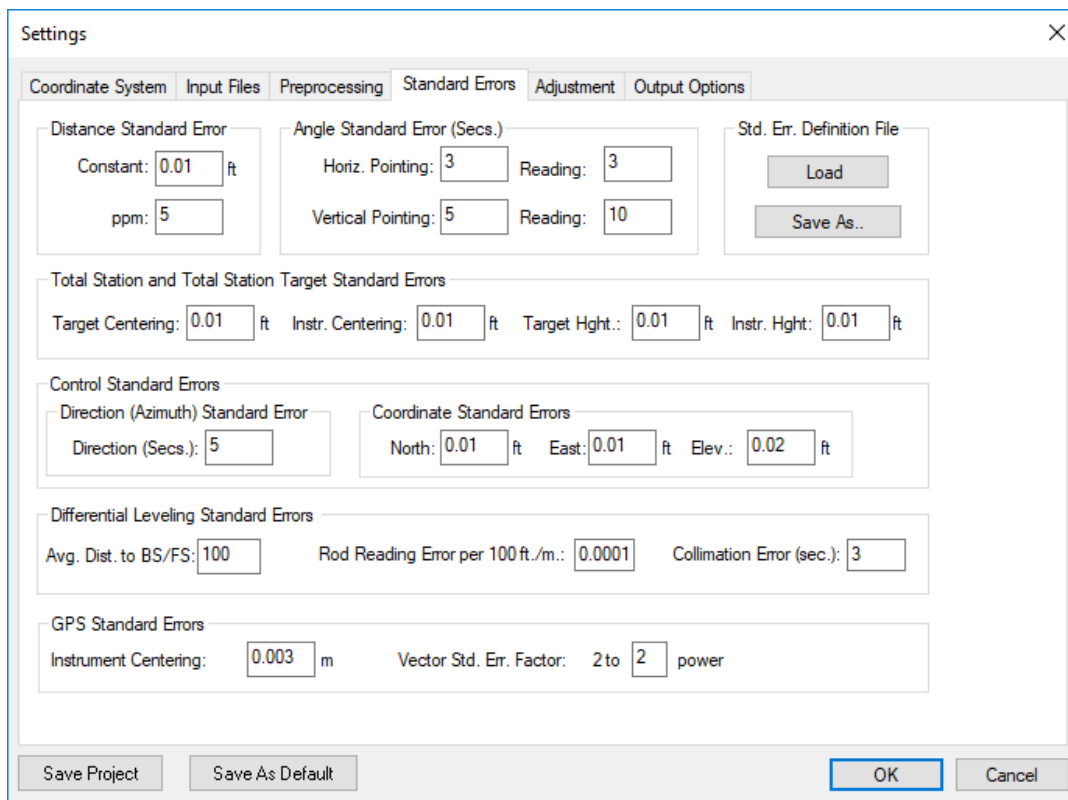
If all the connections pass in the **All Connections** section, you will not see any Error Ellipses. You will see the following message:

***** All connection combinations passed *****

Standard errors are the expected measurement errors based on the type equipment and field procedures being used. For example, if you are using a 5 second total station, you would expect the angles to be measured within +/- 5 seconds (Reading error).

The Distance Constant, PPM settings, and Angle Reading should be based on the equipment and field procedures being used. These values can be obtained from the published specifications for the total station. Or, the distance PPM and constant can be computed for a specific EDM by performing an EDM calibration using an EDM calibration baseline.

Survey methods should also be taken into account when setting standard errors. For example, you might set the target centering standard error higher when you are sighting a held prism pole than you would if you were sighting a prism set on a tripod.



The settings from this dialog box will be used for the Project Default settings. These default Standard Errors can be overridden for specific measurements by placing SE records directly into the Raw Data File.

If the report generated when you process the data shows that generally you have consistently high standard residuals for a particular measurement value (angles, distances, etc.), then there is the chance that you have selected standard errors that are better than your instrument and methods can obtain. (See explanation of report file). Failing the chi-square test consistently is also an indication that the selected standard errors are not consistent with the field measurements.

You can set the standard errors for the following:

Distance and Angle Standard Errors

Distance Constant: Constant portion of the distance error. This value can be obtained from published EDM specifications, or from an EDM calibration.

Distance PPM: Parts per million component of the distance error. This value can be obtained from published EDM specification, or from an EDM calibration.

Horizontal Angle Pointing: The horizontal angle pointing error is influenced by atmospheric conditions, optics, experience and care taken by instrument operator.

Horizontal Angle Reading: Precision of horizontal angle measurements, obtain from theodolite specifications.

Vertical Angle Pointing: The vertical angle pointing error is influenced by atmospheric conditions, optics, experience and care taken by instrument operator.

Vertical Angle Reading: Precision of vertical angle measurements, obtain from theodolite specifications.

Instrument and Target Standard Errors

Target Centering: This value is the expected amount of error in setting the target or prism over the point.

Instrument Centering: The expected amount of error in setting the total station over the point.

Target Height: The expected amount of error in measuring the height of the target.

Instrument Height: The expected amount of error in measuring the height of the total station.

Control Standard Errors

Direction (Bearing / Azimuth): The estimated amount of error in the bearing / azimuth (direction) found in the azimuth records of the raw data.

North, East, Elev: The estimated amount of error in the control north, east and elevation values. You may want to have different coordinate standard errors for different methods of obtaining control. For example, standard errors of control derived from RTK GPS would be higher than control derived from GPS static measurements.

GPS Standard Errors

Instrument Centering: This option is used to specify the error associated with centering a GPS receiver over a point.

Vector Standard Error Factor: This option is used as a factor to increase GPS vector standard errors as found in the input GPS vector file. Some people think that the GPS vector variances/covariances as found in GPS vector files tend to be overly optimistic. This factor allows the user to globally increase the GPS vector standard errors without having to edit the GPS vector file. A factor of 0 is the default value and results in no change to the GPS vector standard errors as found in the GPS vector file. Acceptable values are 0 through 9. **It is not a linear progression.** The actual multiplication factor is the number 2 raised to the power entered in the factor box. For example, if the value of 4 is entered, the multiplication factor will be 2 to the 4th power or 16.

The Vector Standard Error is 2 to the Power of x as shown below:

0 = original value from GPS

1 = x2

2 = x4

3 = x8

4 = x16

A way to determine the proper factor to use for your specific make and model of GPS equipment is to perform a GPS only survey without mixing in Total Station. Then process with a different factors and check the Chi-Square result that falls closest to the middle of the range.

Differential Leveling Standard Errors

These setting only effect level data and are not used when processing total station or GPS vector files.

Avg. Dist. To BS/FS: This option is used to define the average distance to the backsight and foresight during leveling.

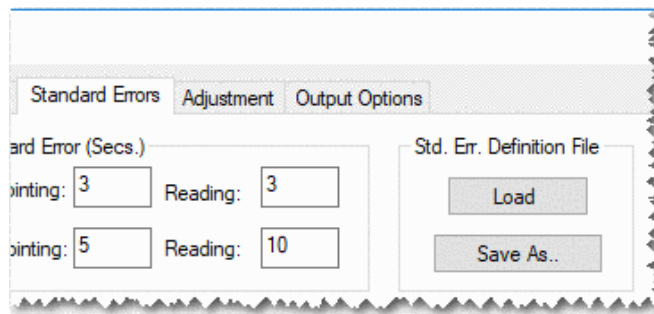
Rod Reading Error per 100 ft./m: This option is used to define the expected level reading error.

Collimation Error: This is the expected differential leveling collimation error in seconds.

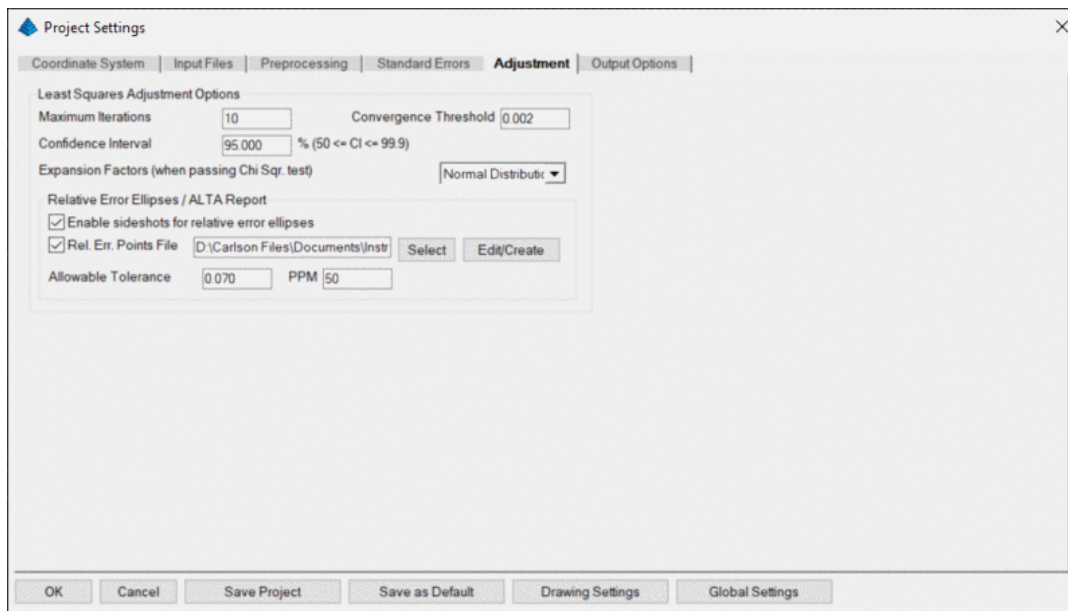
Standard Error Definition Files

The Standard error settings can be saved and then later reloaded into an existing or new project. Creating libraries of standard errors for different types of survey equipment or survey procedures is convenient method of creating standards within a survey department that uses a variety of equipment and performs different types of surveys.

Standard error library files (.sef) can be created two ways. From the 'Settings – Standard Errors' tab, the 'Load' button can be used to import an existing .SEF file into the current project. A .SEF file can also be created from the existing project standard errors by using the 'Save As..' button.



Adjustment



Maximum Iterations: Non-linear least squares is an iterative process. The user must define the maximum number of iterations to make before the program quits trying to find a converging solution. Typically if there are no blunders in the data, the solution will converge in less than 5 iterations.

Convergence Threshold: During each iteration, corrections are computed. When the corrections are less than the threshold value, the solution has converged. This value should be somewhat less than the accuracy of the measurements. For example, if you can only measure distances to the nearest 0.01' then a reasonable convergence threshold value would be 0.005'.

Confidence Interval: This setting is used when calculating the size of error ellipses, and in the Chi²; (Chi-square) testing. For example, a 95% confidence interval means that there is a 95% chance that the error is within the tolerances shown.

Expansion Factors (when passing Chi²; test): This setting allows users to choose between a normal distribution or F distribution for error ellipses on points that pass the Chi Square Test.

Enable sideshots for relative error ellipses: Check this box if you want to see the error ellipses and relative error ellipses of sideshots. This check-box must be set if you want to use the "relative error ellipse inverse" function with sideshots. When unchecked, this toggle filters out sideshots during the Least Squares processing. Since the sideshots are excluded from the Least Squares processing, error ellipses cannot be computed for these points. When this toggle is off, the sideshots are computed after the network has been adjusted. The final coordinate values of the sideshots will be the same regardless of this setting.

- **Note:** Large numbers of sideshots slow down least squares processing. It is best to uncheck this box while de-

bugging your project to avoid having to wait for the computer to finish processing. After the project processes correctly you may turn on the option for the final processing.

- **Note: Any sideshots that are selected for the ALTA report will automatically be included in the adjustment process in order to calculate the error ellipses - even if this option is turned OFF.** Even if you do not need ALTA report, this is an easy way to get statistics on only selected sideshots which can greatly speed up the adjustment process.

Relative Err. Points File: Any certification specifying "Positional Tolerance", "Positional Accuracy" or "Positional Precision" deal with the relative error between specified points. The ALTA standards require that surveyors certify to the relative positional tolerance between points. Relative error ellipses are an accepted method of determining the relative positional error required by the ALTA standards. The points that are to be included in the relative error checking are specified by the user. These points are defined in an ASCII file with an extension of .ALT. To select an .ALT file for relative error checking use the 'Select' button and then browse to the location of the file. Refer to the Creating ALTA Tolerance Reports for a discussion on creating the .ALT file.

If the Relative Err. Points File box is checked, an ALTA section will be created in the report.

Allowable Tolerance, PPM: These fields allow the user to set the allowable error for computations. Typically the user would enter the current ALTA error standards. Many States are adopting similar certifications to the ALTA standards. These certifications may have different tolerances depending on the type of survey (for example rural vs. urban). Make sure the appropriate tolerance and PPM (parts per million) values have been defined. The ALTA standards define their positional standard as 0.07' plus 50 PPM and typically require that the computations be performed to a 95% confidence. The confidence interval is set in the *Confidence Interval* edit field.

Refer to the discussion on creating and interpreting the ALTA report.

Output Options

These settings apply to the output of data to the report and coordinate files.

The screenshot shows the 'Settings' dialog box with the 'Output Options' tab selected. The 'Display Precision' section has dropdowns for North/East (4), Elev. (4), Distances (4), and Directions (0.00 sec). The 'Format' section has dropdowns for Direction (Azimuth), Angles (Degrees), and Coordinate (North, East), and a text field for Null Elevation (-9999999.0). The 'ASCII NEZ Output' section has a dropdown for Format (P, N, E, Z, CD, DESC (fixed columns)), and dropdowns for N/E Precision (8) and Elevation Precision (8). The 'Coordinate File Output' section has a checked 'Write to coordinate file' checkbox, a file path 'C:\Carlson Projects\SurvNET.crd', and 'Do not overwrite elevations' and 'Do not overwrite descriptions' checkboxes. The 'Scaled Coordinate File' section has a 'Create Scaled/Ground NEZ File' checkbox, a file path field, and a 'Scaled NEZ File Format' dropdown set to 'Coordinate File'. The 'Scale' section has a 'Scale Factor' field, a 'Pt. to Scale Around' field, a 'Use Current NE values' dropdown, and 'N:' and 'E:' text fields. At the bottom are 'Save Project', 'Save As Default', 'OK', and 'Cancel' buttons.

Display Precision

These settings determine the number of decimal places to display in the reports for the following types of data. **The**

display precision has no effect on any computations, only the display of the reports.

- Coordinates (North, East, Elevation) - Chose 0-4 decimal places.
- Distances - Chose 0-4 decimal places
- Directions (Azimuths or Bearings) - nearest second, tenth of second, or hundredth of second.

Format

These settings determine the format for the following types of data.

Direction - Choose either bearings or azimuth for direction display. If the angle units are degrees, bearings are entered as QDD.MMSSss and azimuths are entered as DDD.MMSSss. If the angle units are grads, bearings are input as QGGG.ggggg and azimuths are input as GGG.ggggg.

Angles - Choose the units you are working in: degrees or gradians.

Coordinate - Choose the order of coordinate display, either north-east or east-north.

Null Elevation - Choose the value for null elevations in the output ASCII coordinate NEZ file. The Null Elevation field defaults to SurvNET's value for NO ELEVATION. This is used to differentiate between NO ELEVATION (a true 2D point) and ZERO ELEVATION (which is a valid elevation).

Coordinate File Output

These settings determine the type and format of the output NEZ file. An ASCII .NEZ and .OUT files are always created after processing the raw data. The .OUT file will be a nicely formatted version of the .NEZ file. The .NEZ file will be an ASCII file suitable to be input into other programs. There are a variety of options for the format of the .NEZ file. Following are the different ASCII file output options.

- P,N,E,Z,CD,DESC (fixed columns); - Point,north,east,elev.,code,desc in fixed columns separated by commas.
- P,N,E,Z,CD,DESC; Point,north,east,elev.,code,desc separated by commas.
- P N E Z CD DESC (fixed columns); Point,north,east,elev.,code,desc in fixed columns with no commas.
- P N E Z CD DESC; Point,north,east,elev.,code,desc in fixed columns with no commas.
- P,N,E,Z,DESC (fixed columns); Point,north,east,elev., desc in fixed columns separated by commas.
- P,N,E,Z,DESC; Point,north,east,elev., desc separated by commas.
- P N E Z DESC (fixed columns); Point,north,east,elev., desc in fixed columns with no commas.
- P N E Z DESC; Point,north,east,elev.,code,desc separated by spaces.
- P,E,N,Z,CD,DESC (fixed columns); - Point,east,north,elev.,code,desc in fixed columns separated by commas.
- P,E,N,Z,CD,DESC; Point,east,north,elev.,code,desc separated by commas.
- P E N Z CD DESC (fixed columns); Point,east,north,elev.,code,desc in fixed columns with no commas.
- P E N Z CD DESC; Point,east,north,elev.,code,desc in fixed columns with no commas.
- P,E,N,Z,DESC (fixed columns); Point,east,north,elev., desc in fixed columns separated by commas.
- P,E,N,Z,DESC; Point,east,north,elev., desc separated by commas.
- P E N Z DESC (fixed columns); Point,east,north,elev., desc in fixed columns with no commas.
- P E N Z DESC; Point,east,north,elev.,code,desc separated by spaces.
- CSV ASCII with Standard Errors

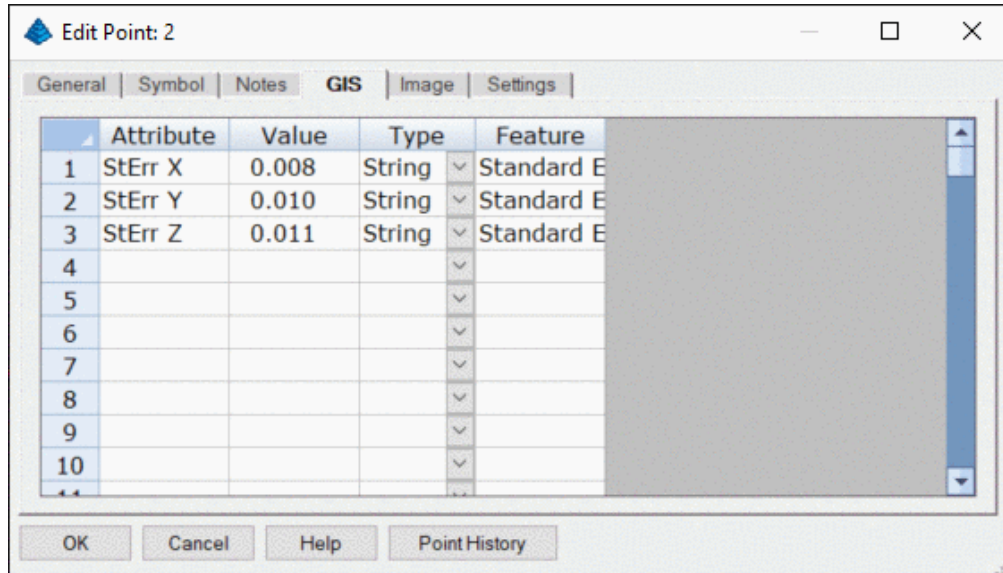
You can also set the output precision of the coordinates for the ASCII output file. This setting only applies to ASCII files, not to the C&G or Carlson binary coordinate files which are stored to full double precision.

* N/E Precision: number of places after the decimal to use for North and East values (0 – 8) in the output NEZ ASCII file.

* Elevation Precision: number of places after the decimal to use for Elevation values (0 – 8) in the output NEZ ASCII file.

Write to Coordinate File: If you want to write the calculated coordinates directly to a Carlson-supported coordinate file, check the "Write to Coordinate File" box and select the file. You can choose the type of Carlson .CRD or CRDB/C&G file to be created when you 'select' the file to be created. You may wish to leave this box unchecked until you are satisfied with the adjustment. Specifying the File Extension will govern the type of file being created.

Note: if you use the Carlson **CRDB** file format, you are allowed to store attributes for each point as well as the standard Point Number, Northing, Easting, Elevation and Description. If this option is used when processing a SurvNET least squares adjustment, the standard error results will be stored for each point.

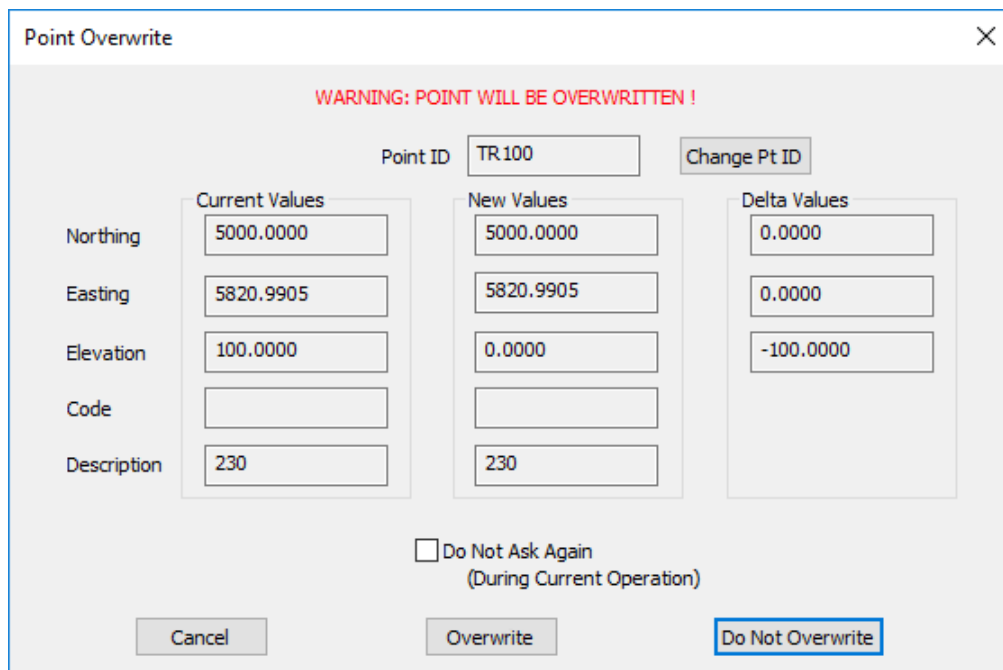


There are two check-boxes below the Output File edit box:

1. **Do not overwrite elevations** - If this is checked the existing elevations in the coordinate file will not be changed.
2. **Do not overwrite descriptions** - If this is checked the existing point descriptions in the coordinate file will not be changed.

Overwriting existing points:

* NOTE: After the adjustment the coordinates will be written to the selected output file. If coordinate points already exist in the coordinate file, before a point is written, you will see the following "Point Protection" dialog box:

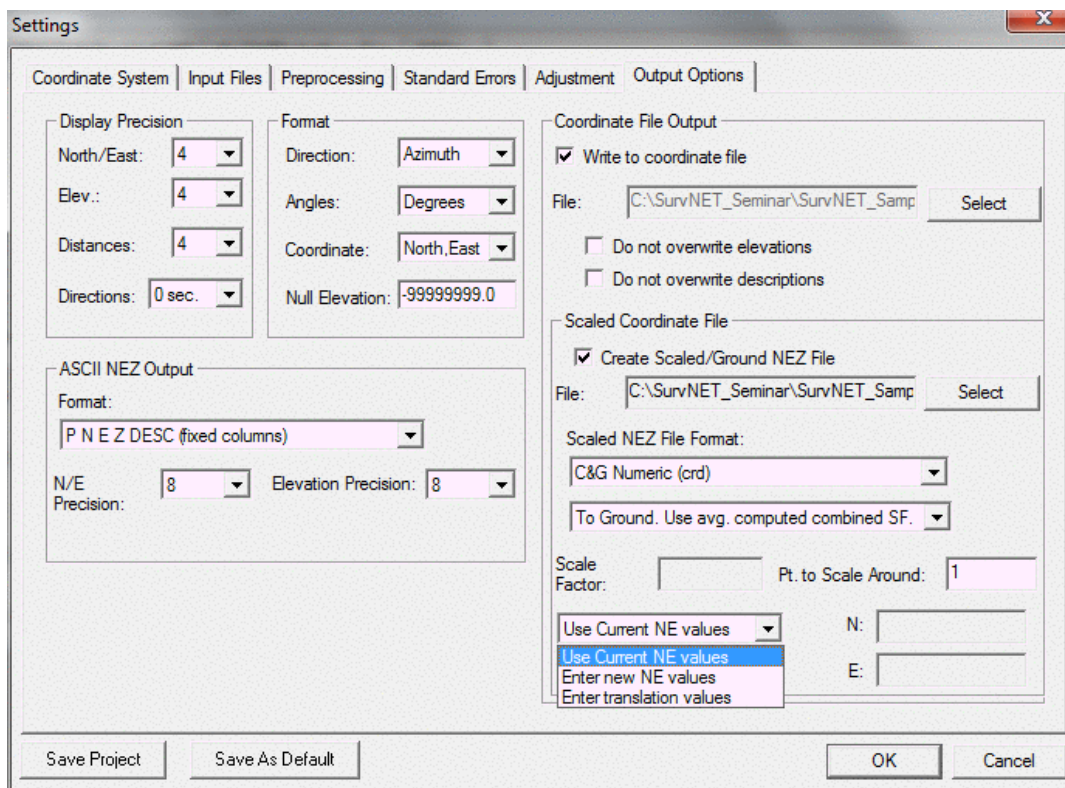


Cancel: Cancel the present operation. No more points will be written to the coordinate file.

Overwrite: Overwrite the existing point. Notice that if you check the 'Do Not Ask Again' box, all further duplicate points will be overwritten without prompting.

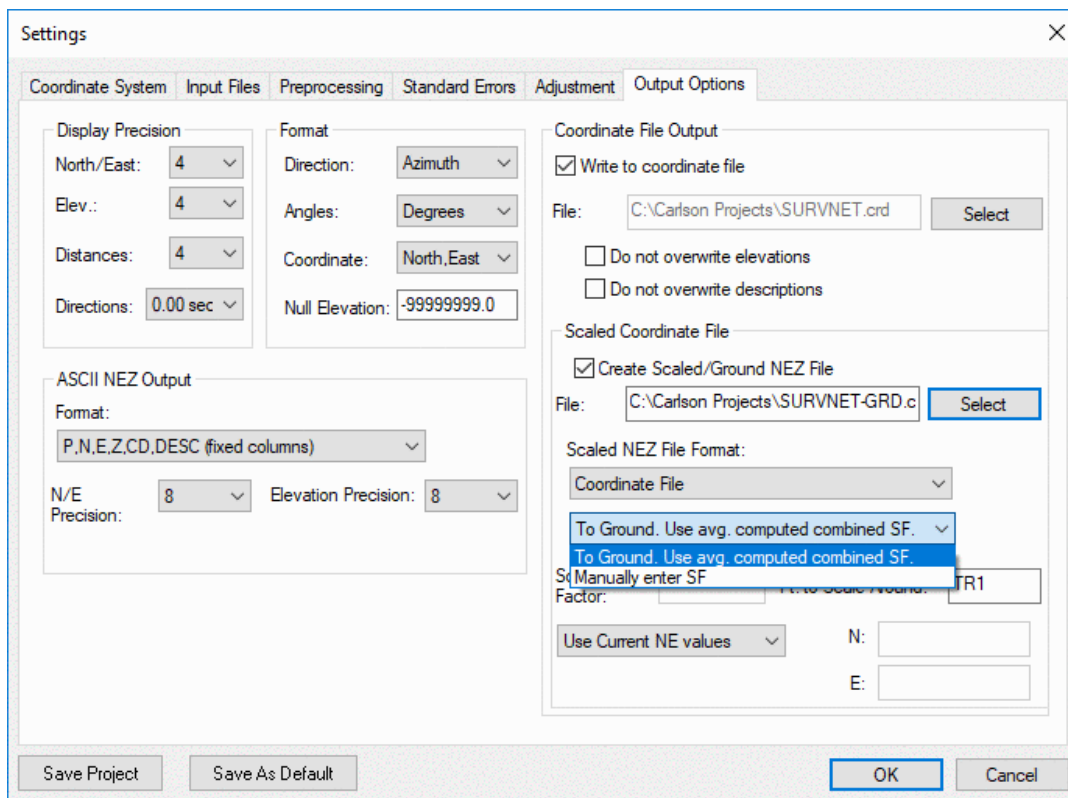
Do not Overwrite: The existing point will not be overwritten. Notice that if you check the 'Do Not Ask Again' box, all further duplicate points will automatically not be overwritten - only new points will be written.

Scaled Coordinate File: You are also allowed to output coordinates to a second, scaled or ground, coordinate file. Check the "Create Scaled/Ground NEZ file" box (see below):



If you are working with a **Local** coordinate system as defined in the Coordinate System tab, you will only be allowed to manually enter a scale factor. You can select the output file format and the output file.

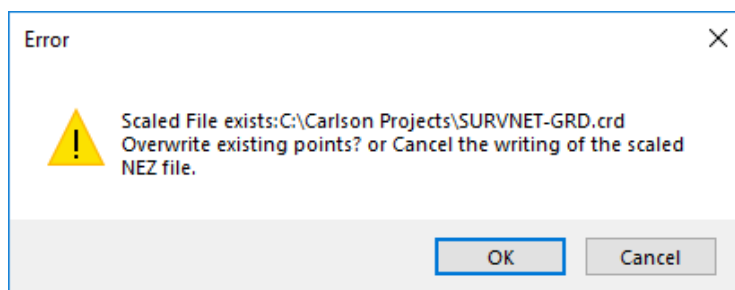
If you are working in a non-local coordinate system (*i.e.* 1983 State Plane), you will have the following scaling options:



You can either enter a scale factor or use the computed grid to ground scale factor. If you select to use the computed scale factor, SurvNET will use the "average combined scale factor" as shown in the project report file.

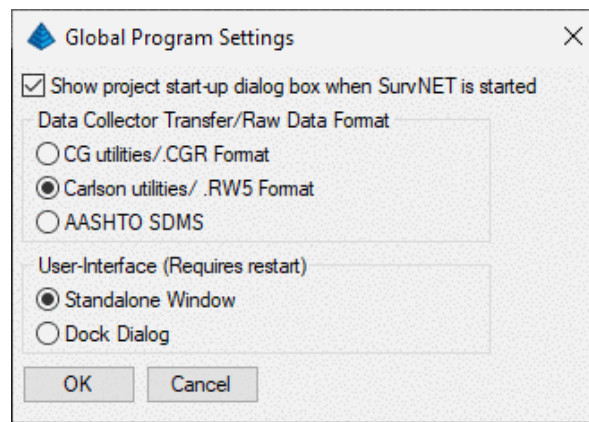
You will also be required to enter the point number of the point you will be scaling around. This function can also be used as a translation tool. You can either "Use the Current NE values" (no translation), "Enter the new NE values" (desired N and E values for the scaling point), or "Enter Translation Values" (enter the actual delta-north and delta-east values).

If the Scaled Coordinate file exists when you process the project, you will see the following warning dialog box:



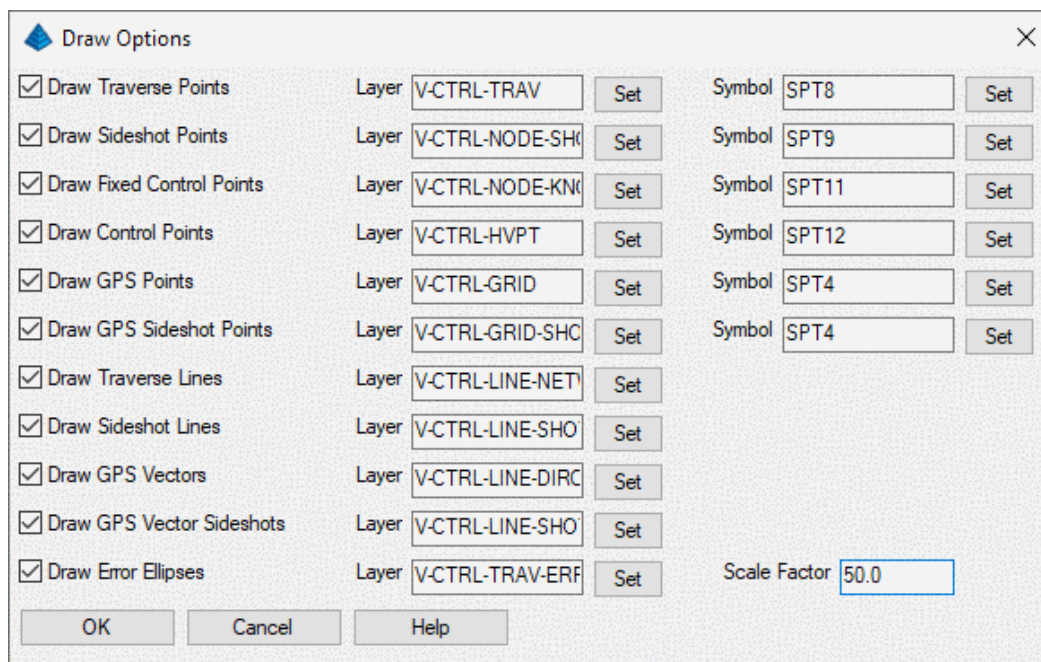
If you pick OK the points in the Scaled File will be overwritten. If you Cancel no point will be written to the Scaled File.

The menu option "Settings – Global Settings" displays the following dialog box. If the 'Use Carlson Utilities' is chosen then the .RW5 editor will be the default raw editor and Carlson SurvCom will be the default data collection transfer program. If the 'Use C&G Utilities' is chosen then the C&G .CGR editor will be the default raw editor and C&G's data collection transfer program will be the default data collection transfer program. For additional information, refer to the Raw Data Files and Raw Data Editor Functions section of this manual.



User-Interface (for CAD versions of SurvNET) - Indicate the desired graphical user interface (*gui*, "goosey") you wish to use.

If you are running SurvNET from inside Carlson Survey, you can draw the network into the active drawing file. These are the settings for the Draw Network function.



NOTE: As a reference, the following is the suggested *National CAD Standards* layer list for the Drawing Settings dialog box:

- V-CTRL-TRAV
- V-CTRL-NODE-SHOT
- V-CTRL-NODE-KNOW
- V-CTRL-HVPT
- V-CTRL-GRID
- V-CTRL-GRID-SHOT
- V-CTRL-LINE-NETW
- V-CTRL-LINE-SHOT
- V-CTRL-LINE-DIRC
- V-CTRL-LINE-SHOT
- V-CTRL-TRAV-ERRO

Check the type points and lines you want to draw. If you want to draw the error ellipse at each point check the "Draw Error Ellipses" box. You can choose a layer for all the points and lines and a symbol for all the points.

Process Menu



The Processing buttons from left to right are:

Network Adjustment - adjusts the network and creates a results report

Preprocess, compute unadjusted coordinates - processes the network, calculates positions and creates an error and coordinate report

Blunder Detection - processes the network based on three user selected parameters and creates an error report

Draw Network - draws the network in an open CAD drawing based on the Drawing Settings set in the Project Settings



When you select Process – *Network Adjustment* from the menu, the raw data will be processed and adjusted using least squares based on the Project Settings. If there is a problem with the reduction, you will be shown error messages that will help you track down the problem. Additionally a .ERR file is created that will log and display error and warning messages.

The data is first preprocessed to calculate averaged angles and distances for sets of angles and multiple distances. For a given setup, all multiple angles and distances to a point will be averaged prior to the adjustment. The standard error as set in the Project Settings dialog box is the standard error for a single measurement. Since the average of multiple measurements is more precise than a single measurement, the standard error for the averaged measurement is computed using the standard deviation of the mean formula.

Non-linear network least squares solutions require that initial approximations of all the coordinates be known before the least squares processing can be performed. So, during the preprocessing approximate coordinate values for each point are calculated using basic coordinate geometry functions. If there is inadequate control or an odd geometric situation, SurvNET may generate a message indicating that the initial coordinate approximations could not be computed. The most common cause of this problem is that control has not been adequately defined or there are point numbering issues.

Side-shots are separated from the raw data and computed after the adjustment (unless the "Enable sideshots for relative error ellipses" toggle is checked in the Adjustment dialog box). If side shots are filtered out of the least squares process and processed after the network is adjusted, processing is greatly sped up, especially for a large project with a lot of side shots.

If the raw data processes completely, a report file, .RPT, a .NEZ file, an .OUT file, and an .ERR file will be created in the project directory. The file names will consist of the project name plus the above file extensions. These different files are shown in separate windows after processing. Additionally, a graphic display of the network can be generated.

.RPT file: This is an ASCII file that contains the statistical and computational results of the least squares processing.

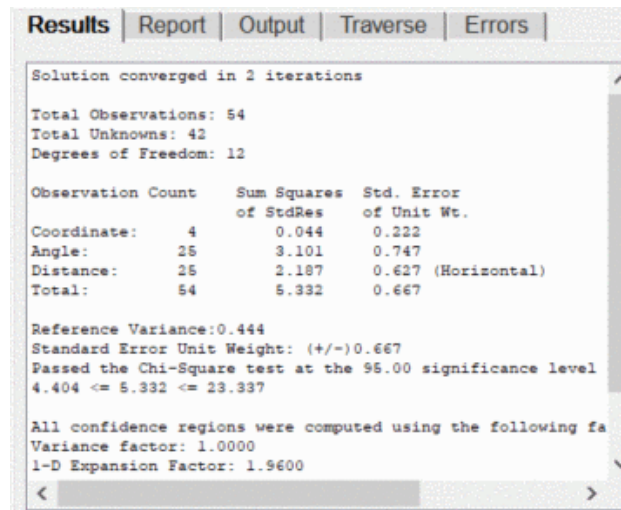
.NEZ file: This file is an ASCII file containing the final adjusted coordinates. This file can be imported into any program that can read ASCII coordinate files. The format of the file is determined by the ASCII NEZ setting in the Project Settings dialog box.

.OUT file: The .OUT file is a formatted ASCII file of the final adjusted coordinates suitable for display or printing.

.ERR file: The .ERR file contains any warning or error messages that were generated during processing. Though some warning messages may be innocuous it is always prudent to review and understand the meaning of the messages.

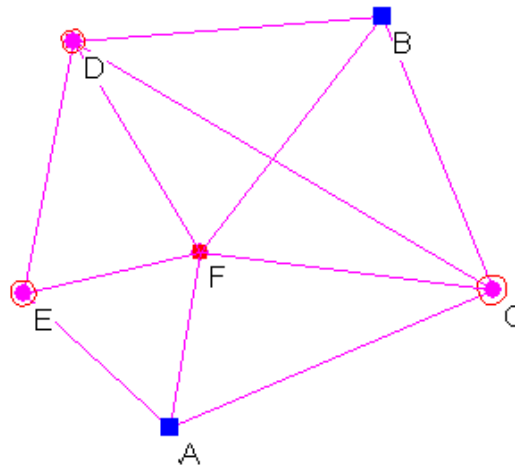
If you have "Write to Coordinate File" checked in the Output options dialog, the coordinates will also be written to a coordinate file (.CRD by default).

The following is an example of the initial report displayed in the dialog box showing the calculation results.

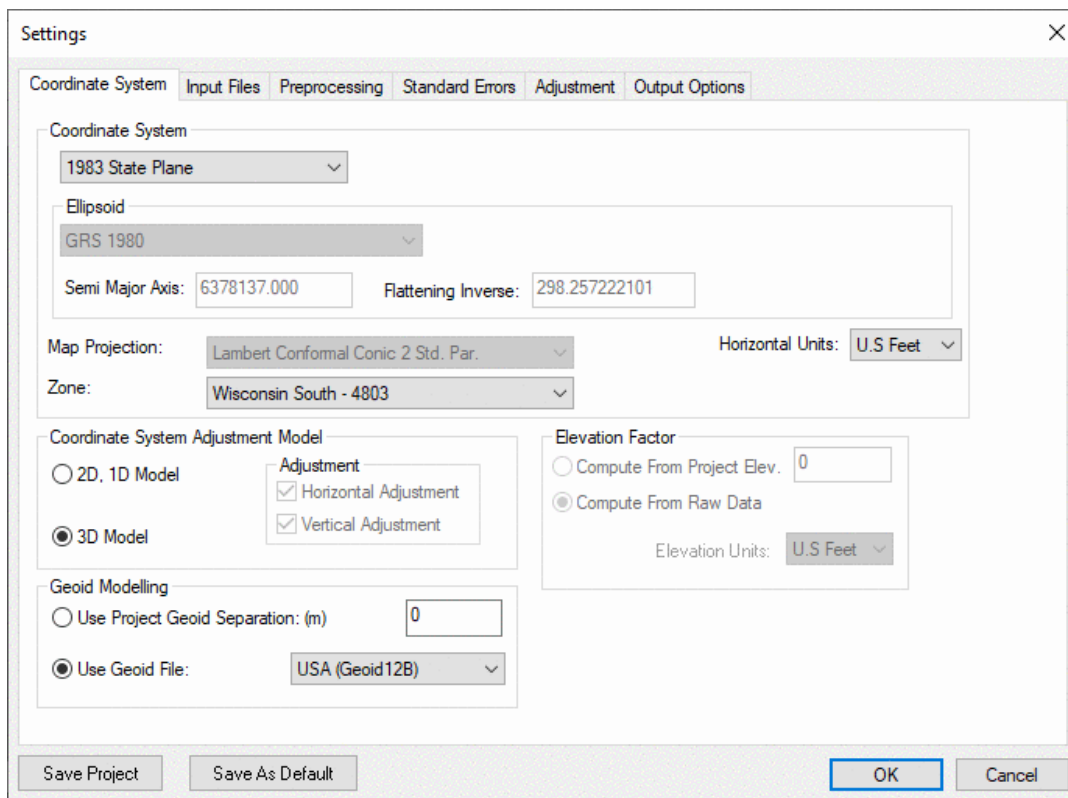


Users can see immediately the results of the Chi Square test along with other pertinent information. you can also toggle each tab to see additional information. To review and print a complete report, use the Report button.

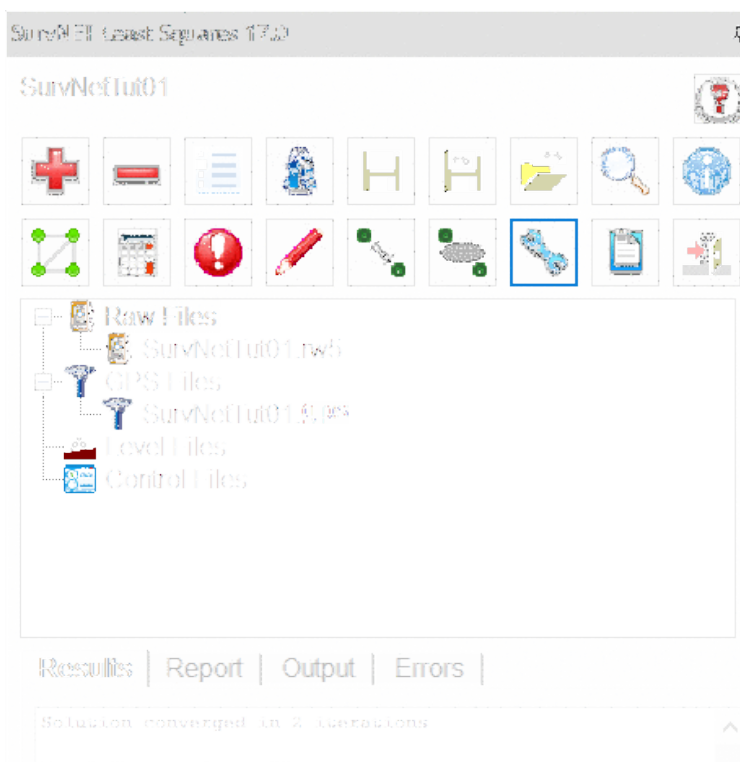
GPS vector networks can be adjusted with SurvNET. This chapter will describe the processing of a simple GPS network. Following is a graphic view of the GPS network that is to be adjusted. Points A and B are control points. The magenta lines represent measured GPS vectors. Most GPS vendor's software can output GPS vectors to a file as part of the post-processing of GPS data.



When processing GPS vectors, certain project settings are important. In the following settings dialog box, notice that the 3D-model has been chosen and SPC 1983 with the appropriate zone has been chosen. The 3-D model and a geodetic coordinate are required when processing GPS vectors. Though it is not required for GPS processing, it is in most cases appropriate to choose to do geoid modeling.



The following settings dialog box shows the raw files used in processing GPS files. A GPS vector file must be chosen. GPS vector files from various GPS vendors are supported. Select the vector files to be processed:



Coordinate control for the network can be in one of several files. The control can be located in the GPS vector file itself. More typically, the control points can be regular coordinate records in the .RW5 or the .CGR file. They also

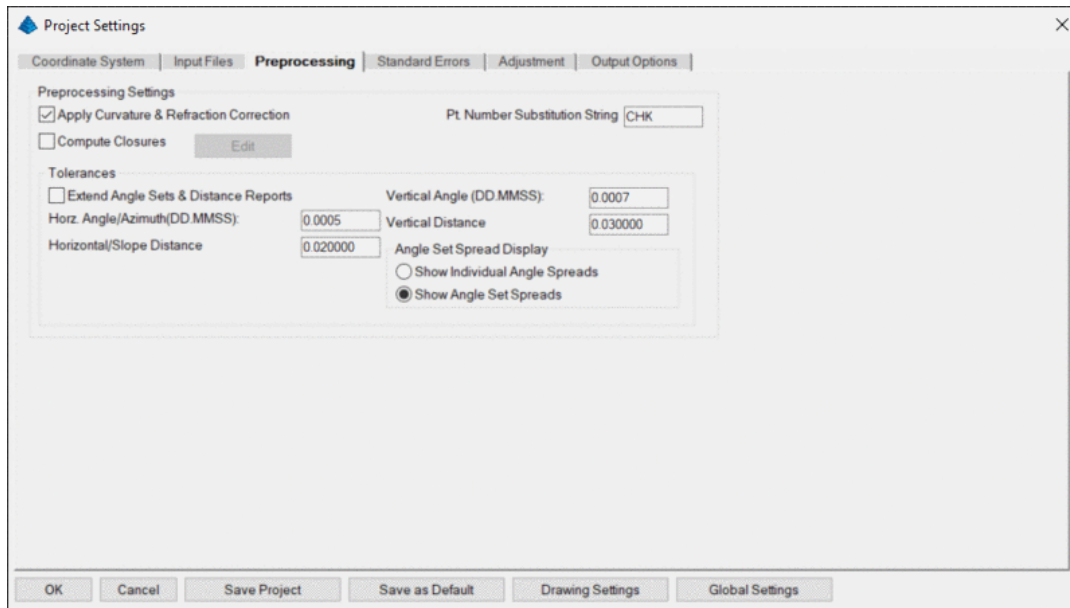
can be entered as 'Supplemental Control' in one of the available formats.

When the control coordinates are in the raw data file they are expected to be grid coordinates with orthometric heights.

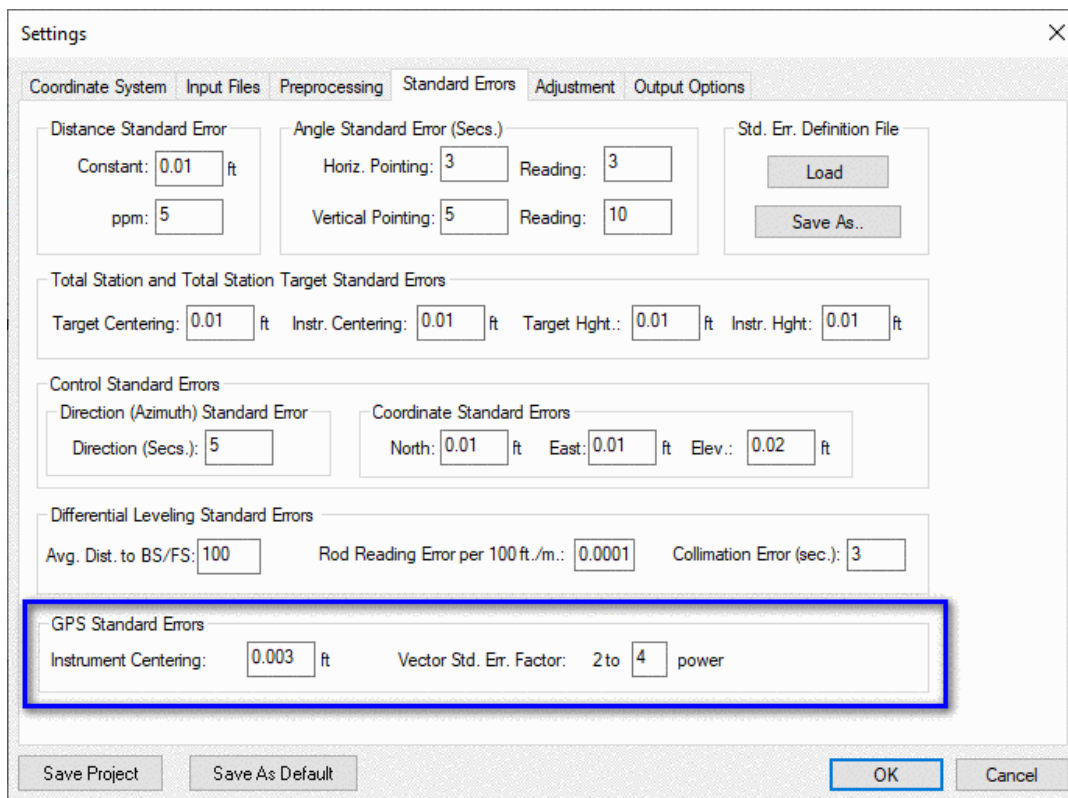
The supplemental control file formats support grid coordinates with orthometric heights, geographic coordinates with orthometric heights, or geocentric coordinates with ellipsoid heights.

If the control coordinates are found in the GPS vector file, they are assumed to be Geocentric ECEF (earth-centered-earth-fixed) XYZ coordinates. As shown in the dialog above, it is not unusual to have different distance units for GPS, total station data, and control data. GPS vector data is usually in metric units but the total station raw file can be in US Feet. So, the distance units must be specified for the different raw data types.

In the Preprocessing Settings dialog box, the only important setting is the **Compute Closures** option. If GPS loop or point-to-point closures need to be computed, the point numbers defining the loops need to be entered into a Closure File. See the discussion on traverse closures to see how to create Closure Files.



There are two GPS standard errors fields in the Standard Errors Settings dialog box. The GPS vector XYZ standard errors and covariances do not need to be defined as project settings since they are found in the GPS vector data files.



For more information, see the discussion on GPS Standard Errors.

Processing a GPS vector network together with conventional total station data is similar to processing a GPS network by itself. The only difference in regards to project settings is that a raw data file containing the total station data needs to be chosen as well as a GPS vector file. The project must be set up for the 3D model and a geodetic coordinate system needs to be chosen. **The total station data must be 3D, all rod heights and instrument heights must be measured.**

Following is a view of the Input Files Settings dialog box showing both a GPS vector file and a total station raw data file chosen in a single project. It is not uncommon to have different distance units for GPS data and total station data, so make sure the correct units are set for data types.



One of the most common problems for new users in combining GPS and total station data is not collecting HI's and rod heights when collecting the total station data. Since the 3D model is being used complete 3D data needs to be collected. If you only have 2D traverse data, you can adjust the GPS vectors first and then use the adjusted coordinates as control for the Traverse data project.



The 'Preprocess, Compute Unadjusted Coordinates' option allows the computation of unadjusted coordinates. If there are redundant measurements in the raw data, the first angle and distance found in the raw data is used to compute the coordinates. If a State Plane grid system has been designated the measurements are reduced to grid prior to the computation of the unadjusted coordinates. If the point is located from two different points the initial computation of the point will be the value stored.

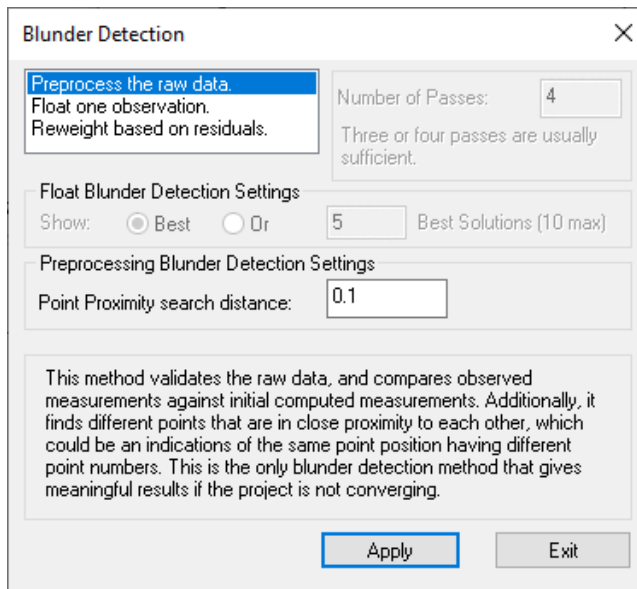


A variety of blunder detection tools are available that gives the user additional tools in analyzing the survey data and detecting blunders. The standard least squares adjustment processing and its resulting report can often be used to determine blunders. No blunder detection method can be guaranteed to find all blunders. So much depends on the nature of the network geometry, the nature of the measurements, and the intuition of the analyst. Generally, the more redundancy there is in a network the easier it is to detect blunders.

There are three different methods that can be used to track down blunders in a network or traverse:

1. Preprocess the raw data
2. Float one observation
3. Reweight based on residuals

Option 1) Preprocess the raw data:



The **Preprocess the raw data** option validates the raw data. It displays angle and distance spreads as well as checks the validity of the raw data. Traverse Closures are computed if specified. It also performs a "K-Matrix" analysis. The "K-Matrix" analysis compares the unadjusted, averaged measurements with the computed preliminary measurements (measurements calculated from the preliminary computed coordinates). This method will catch blunders such as using the same point number twice for two different points. The report will be sent to the .ERR file. The .ERR file will contain the tolerance checks, closures and the K-Matrix analysis. Following is an example of the report created using the 'Preprocess the raw data' option. Notice that the first section of the report shows the angle and distance spreads from the multiple angle and distance measurements. The second part of the report shows the 'K-matrix' analyses.

Additionally there is a 'Point Proximity Report' section that reports pairs of different points that are in close proximity to each other which may indicate where the same point was collected multiple times using different point numbers.

The 'Preprocess the raw data' option is one of the simplest and effective tools in finding blunders. Time spent learning how this function works will be well spent. If the project is not converging due to an unknown blunder in the raw data, this tool is one of the most effective tools in finding the blunder. Many blunders are due to point numbering errors during data collections, and the 'K-matrix' analysis and 'Point Proximity' search are great tools for finding this type blunders.

```

=====
LEAST SQUARES ADJUSTMENT ERROR REPORT
=====
Tue Mar 21 16:04:00
Input Raw Files:
  C:\Carlson Projects\cgstar\CGSTAR.CGR
Output File: C:\Carlson Projects\cgstar\cgstar.RPT
Checking raw data syntax and angle & distance spreads.
Warning: Missing Vert. Angle. Assumption made as to whether it is direct or reverse.
  1                5.00  180.00050                4
Warning: Missing Vert. Angle. Assumption made as to whether it is direct or reverse.
  1                5.00  180.00070                5
Warning: Missing Vert. Angle. Assumption made as to whether it is direct or reverse.
  1                5.00  180.00100               10
Warning: Missing Vert. Angle. Assumption made as to whether it is direct or reverse.
  1                5.00  180.00020               11
Warning: Missing Vert. Angle. Assumption made as to whether it is direct or reverse.
  1                5.00  325.54320                2      H&T
Warning: Missing Vert. Angle. Assumption made as to whether it is direct or reverse.
  1                5.01  145.54300                2      H&T
Warning: Missing Vert. Angle. Assumption made as to whether it is direct or reverse.
  1                5.01  180.00020               12
Horizontal Angle spread exceeds tolerance:
  IP: 1,  BS: 5,  FS: 2

```

Low: 109-19'10.0'' , High: 109-19'17.0'' , Diff: 000-00'07.0''
 Horizontal Angle spread exceeds tolerance:
 IP: 2, BS: 1, FS: 6
 Low: 190-32'02.0'' , High: 190-32'10.0'' , Diff: 000-00'08.0''
 Horizontal Angle spread exceeds tolerance:
 IP: 2, BS: 1, FS: 3
 Low: 096-03'48.0'' , High: 096-03'56.0'' , Diff: 000-00'08.0''
 Horizontal Angle spread exceeds tolerance:
 IP: 3, BS: 2, FS: 4
 Low: 124-03'50.0'' , High: 124-03'56.0'' , Diff: 000-00'06.0''
 Horizontal Angle spread exceeds tolerance:
 IP: 5, BS: 4, FS: 10
 Low: 039-26'35.0'' , High: 039-26'45.0'' , Diff: 000-00'10.0''
 Horizontal Angle spread exceeds tolerance:
 IP: 10, BS: 5, FS: 11
 Low: 241-56'23.0'' , High: 241-56'35.0'' , Diff: 000-00'12.0''
 Horizontal Angle spread exceeds tolerance:
 IP: 11, BS: 10, FS: 12
 Low: 114-56'20.0'' , High: 114-56'34.0'' , Diff: 000-00'14.0''
 Horizontal Angle spread exceeds tolerance:
 IP: 12, BS: 11, FS: 3
 Low: 140-39'18.0'' , High: 140-39'31.0'' , Diff: 000-00'13.0''
 Horizontal Angle spread exceeds tolerance:
 IP: 5, BS: 4, FS: 1
 Low: 117-30'35.0'' , High: 117-30'50.0'' , Diff: 000-00'15.0''
 Horizontal Distance from 2 to 3 exceeds tolerance:
 Low: 324.15, High: 324.20, Diff: 0.04
 Vertical Distance from 2 to 3 exceeds tolerance:
 Low: 6.62, High: 8.36, Diff: 1.74
 Vertical Distance from 3 to 4 exceeds tolerance:
 Low: 11.46, High: 11.51, Diff: 0.05
 Horizontal Distance from 12 to 3 exceeds tolerance:
 Low: 144.64, High: 144.66, Diff: 0.02
 K-Matrix Analysis.
 Distance: From pt.: 4 To pt.: 5
 Measured distance: 309.61 Initial computed distance: 309.65
 Difference: -0.04
 Distance: From pt.: 12 To pt.: 3
 Measured distance: 144.63 Initial computed distance: 144.66
 Difference: -0.03
 Distance: From pt.: 5 To pt.: 6
 Measured distance: 348.51 Initial computed distance: 523.29
 Difference: -174.79
 Angle: IP: 4 BS: 3 FS: 5
 Measured angle: 093-02'11.5''
 Initial computed angle: 093-01'45.1''
 Difference: 000-00'26.4''
 Angle: IP: 12 BS: 11 FS: 3
 Measured angle: 140-39'24.5''
 Initial computed angle: 140-40'32.6''
 Difference: -000-01'08.1''
 Angle: IP: 5 BS: 4 FS: 1
 Measured angle: 117-30'42.5''
 Initial computed angle: 117-31'16.4''
 Difference: -000-00'33.9''
 Angle: IP: 5 BS: 4 FS: 6
 Measured angle: 145-30'34.0''
 Initial computed angle: 079-39'46.4''
 Difference: 065-50'47.6''
 Point Proximity Report:
 Points 3 and 30 are within 0.05 of each other.

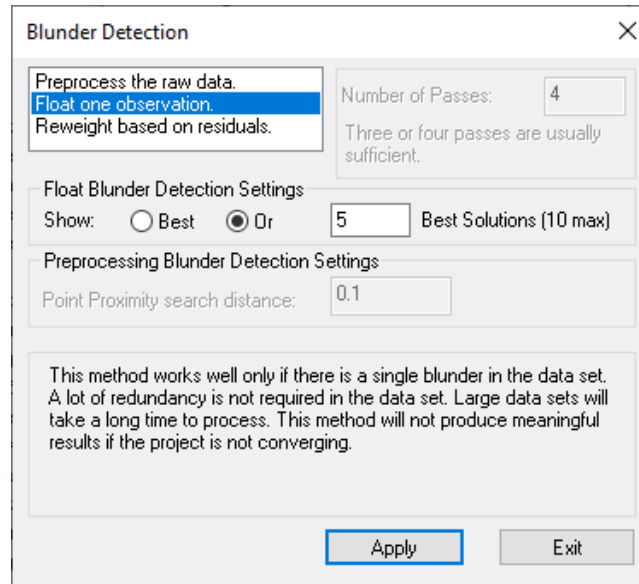
The problem with the above project was that point 6 was accidentally used twice for two separate side shots. Because of the point numbering problem the project would not converge, using the regular least squares processing. The 'Preprocess the raw data' option was then used. Notice in the K-matrix section the distance from 5 to 6 shows

a difference of 174.79' and the angle 4-5-6 shows a difference of 065-50'47.6". Then notice that the other listed differences are in the range of .02' for the distances and less than a minute for the angles. This report is clearly pointing out a problem to point 6.

Note the point proximity report section. During data collection, point number 30 was used as the point number when the point was previously collected as point 3.

In the first section of the report, notice that there are several warnings concerning whether a horizontal angle reading was collected in direct or reverse reading. The preprocessing software uses the vertical angle reading to determine the angle face of the horizontal angle reading. If the vertical angle is missing, the program makes its best guess as to whether the angle was collected in direct or reverse face. Since all horizontal angle spreads in the report are reasonable, the preprocessing software must have made the correct determination.

Option 2) Float one observation:



This option is useful in finding a single blunder, either an angle or distance, within a network or traverse. If there is more than a single blunder in the network then it is less likely that this method will be able to isolate the blunders. **If the standard least squares processing results in a network that will not converge, then this blunder detection method will not work.** Use the Preprocess the raw data blunder detection method if the solution is not converging. Also this method will only work on small and moderately sized networks. This method performs a least squares adjustment once for every non-trivial measurement in the network. So for large networks this method may take so long to process that it is not feasible to use this method.

With this method, an adjustment is computed for each non-trivial individual angle and distance measurement. Consecutively, a single angle or distance is allowed to float during each adjustment. The selected angle or distance does not "constrain" the adjustment in any way. If there is a single bad angle or distance, one of the adjustment possibilities will place most of the error in the "float" measurement, and the other measurements should have small residuals. The potentially bad angle or distance is flagged with a double asterisk (**). Since an adjustment is computed for each measurement, this method may take a long time when analyzing large data files.

The adjustments with the lowest reference variances are selected as the most likely adjustments that have isolated the blunder. You have the choice to view the best adjustment, or the top adjustments with a maximum of ten. In the above example, we asked to see the top 5 choices for potential blunders. The results are shown in the .ERR file. Following is a section of the report generated where an angular blunder was introduced into a small traverse. Notice the '**' characters beside the angle measurements. In this report the two most likely adjustments were displayed. The blunder was introduced to angle 101-2-3. Angle 101-2-3 was chosen as the 2nd most likely source of the blunder, showing that these blunder detection methods though not perfect, can be a useful tool in the analysis of survey measurements. Notice how much higher the standard residuals are on the suspected blunders than the standard

residuals of the other measurements.

Adjusted Observations
 =====

Adjusted Distances

From Sta.	To Sta.	Distance	Residual	StdRes.	StdDev
101	2	68.780	-0.006	0.608	0.008
2	3	22.592	-0.006	0.573	0.008
3	4	47.694	-0.002	0.213	0.008
4	5	44.954	-0.001	0.069	0.008
5	6	62.604	0.005	0.472	0.009
6	7	35.512	0.006	0.539	0.008
7	101	61.704	0.003	0.314	0.009
Root Mean Square (RMS)			0.005		

Adjusted Angles

BS Sta.	Occ. Sta.	FS Sta.	Angle	Residual	StdRes	StdDev(Sec.)
7	101	2	048-05'06''	-5	0	21
101	2	3	172-14'33''	-2	0	27
2	3	4	129-27'44''	-222 *	7	56 **
3	4	5	166-09'59''	11	0	25
4	5	6	043-12'26''	22	1	21
5	6	7	192-11'52''	12	0	25
6	7	101	148-38'19''	8	0	25
Root Mean Square (RMS)					85	

Adjusted Azimuths

Occ. Sta.	FS Sta.	Bearing	Residual	StdRes	StdDev(Sec.)
101	7	N 00-00'00''			
E	0	0	4		
Root Mean Square (RMS)			0		

Statistics
 =====

Solution converged in 2 iterations
 Degrees of freedom:3
 Error Factors... (not shown)
 Standard error unit Weight: +/-0.88
 Reference variance:0.78
 Passed the Chi-Square test at the 95.00 significance level
 0.216 <= 2.347 <= 9.348

Adjusted Observations
 =====

Adjusted Distances

From Sta.	To Sta.	Distance	Residual	StdRes.	StdDev
101	2	68.781	-0.005	0.473	0.009
2	3	22.592	-0.005	0.512	0.009
3	4	47.690	-0.006	0.586	0.009
4	5	44.950	-0.005	0.523	0.009
5	6	62.605	0.006	0.607	0.009
6	7	35.512	0.006	0.560	0.009
7	101	61.708	0.006	0.614	0.009
Root Mean Square (RMS)			0.006		

Adjusted Angles

BS Sta.	Occ. Sta.	FS Sta.	Angle	Residual	StdRes	StdDev(Sec.)
7	101	2	048-05'22''	11	0	24
101	2	3	172-11'03''	-213 *	7	58 **
2	3	4	129-31'23''	-3	0	29
3	4	5	166-09'48''	1	0	26
4	5	6	043-12'11''	6	0	21
5	6	7	192-11'50''	10	0	27
6	7	101	148-38'24''	13	0	27
Root Mean Square (RMS)					81	

Adjusted Azimuths

Occ. Sta.	FS Sta.	Bearing	Residual	StdRes	StdDev(Sec.)
101	7	N-00-00'00''E	-0	0	5
Root Mean Square (RMS)			0		

Statistics
 =====

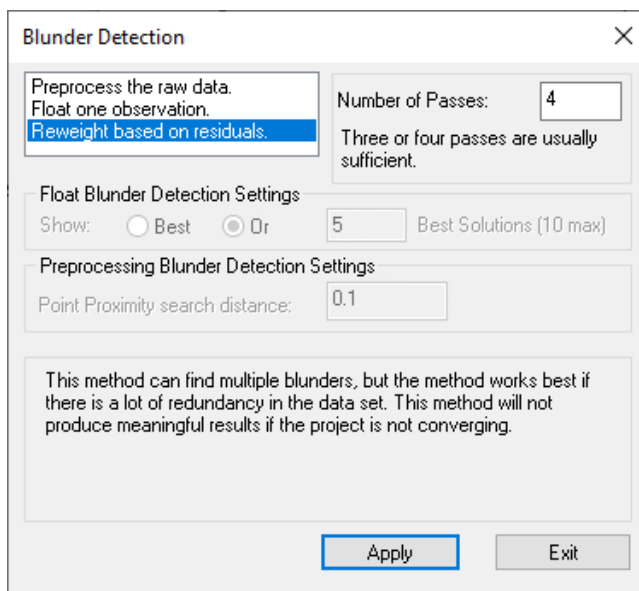
```

Solution converged in 2 iterations
Degrees of freedom:3
Error Factors... (not shown)
Standard error unit Weight: +/-0.94
Reference variance:0.89
Passed the Chi-Square test at the 95.00 significance level
0.216 <= 2.675 <= 9.348

```

The blunder is most likely in the measurement containing the largest residual and standard residual. The observation marked with ** is the observation that floated. It is also most likely the measurement containing the blunder.

Option 3) Reweight based on residuals:



This method is capable of detecting multiple blunders but one is more likely to find the blunders if there is a high degree of redundancy (network of interconnected traverses). The higher the degree of freedom the more likely this method will find the blunders. **This method will not work if the standard least squares processing will not converge.** Use the Preprocess the raw data blunder detection method if the network is not converging.

First, select the number of adjustments or passes you wish to make. Each time an adjustment is completed, the measurements will be re-weighted based on the residuals and standard errors. Hopefully, after three or four passes, the blunders will become obvious. The results are shown in the .ERR file, look for the measurements with the highest standard residuals. These measurements are more likely to contain blunders.

The theory behind this method is that after processing, the measurements with blunders are more likely to have higher residuals and computed standard errors. So, in the next pass the measurements are re-weighted based on the computed residuals, with less weight being assigned to the measurements with high residuals. After several passes it is likely that the measurements with the blunders have been reweighed such that they have little effect on the network.

As a rule of thumb, three or four passes are usually sufficient. Following is a section of the report showing the results of the 'Reweight based on residuals'. This report was generated using the same data used in the earlier example. Notice that it has flagged the same two angle measurements.

The 'Reweight based on residuals' method performs a new adjustment for each pass. So, this method will take longer than the standard least squares adjustment, but does not take near as long to complete processing as the Float one Observation method for larger networks.

```

Adjusted Observations
=====
Adjusted Distances

```


From Sta.	To Sta.	Distance	Residual	StdRes.	StdDev
101	2	68.778	-0.009	0.827	0.014
2	3	22.588	-0.010	0.942	0.015
3	4	47.694	-0.002	0.208	0.009
4	5	44.954	-0.001	0.077	0.006
5	6	62.608	0.010	0.919	0.016
6	7	35.517	0.011	1.040	0.016
7	101	61.705	0.004	0.398	0.011
Root Mean Square (RMS)			0.008		

Adjusted Angles

BS Sta.	Occ. Sta.	FS Sta.	Angle	Residual	StdRes	StdDev (Sec.)
7	101	2	048-05'07''	-4	0	21
101	2	3	172-13'19''	-77 *	2	65
2	3	4	129-29'56''	-91 *	3	64
3	4	5	166-09'44''	-3	0	24
4	5	6	043-12'05''	0	0	9
5	6	7	192-11'40''	-0	0	19
6	7	101	148-38'10''	-1	0	20
Root Mean Square (RMS)					45	

Adjusted Azimuths

Occ. Sta.	FS Sta.	Bearing	Residual	StdRes	StdDev (Sec.)
101	7	N 00-00'00''E	0	0	2
Root Mean Square (RMS)			0		

Statistics

=====

Solution converged in 1 iterations
 Degrees of freedom:3
 Error Factors... (not shown)
 Standard error unit Weight: +/-1.33
 Reference variance:1.77
 Passed the Chi-Square test at the 95.00 significance level
 0.216 <= 5.322 <= 9.348

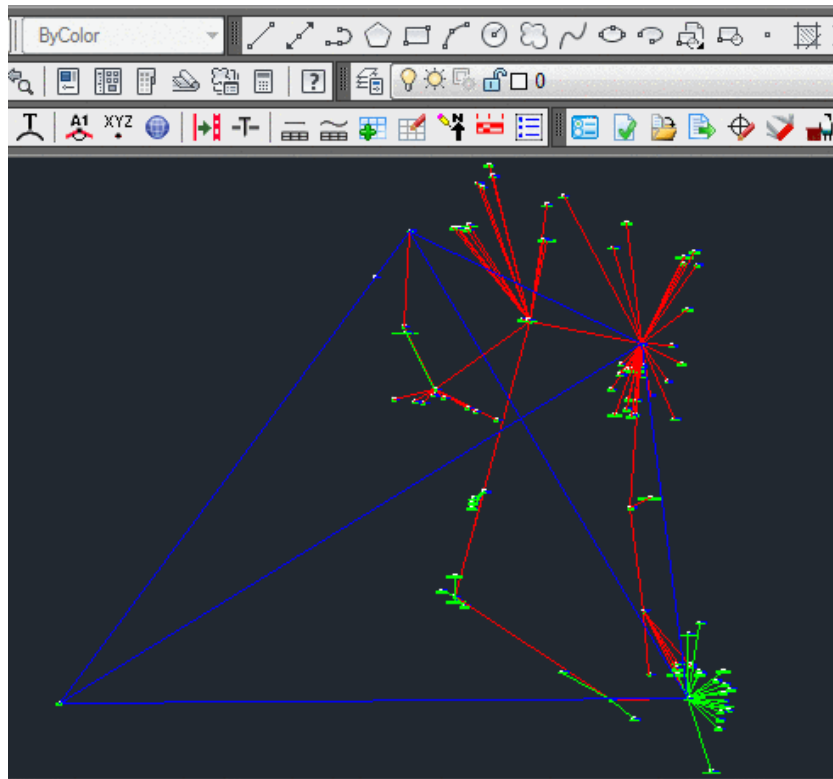
The blunders are most likely in the measurements containing the largest residuals and standard residuals.

If you are running SurvNET from inside Carlson Survey with AutoCAD or IntelliCAD, you can draw the network in the DWG file. The Drawing Settings option located in the Settings Menu will determine the layers and symbols used for each point and line entity.

The Draw Network option will not be available until the network has been adjusted. After the adjustment, you can select Draw Network.



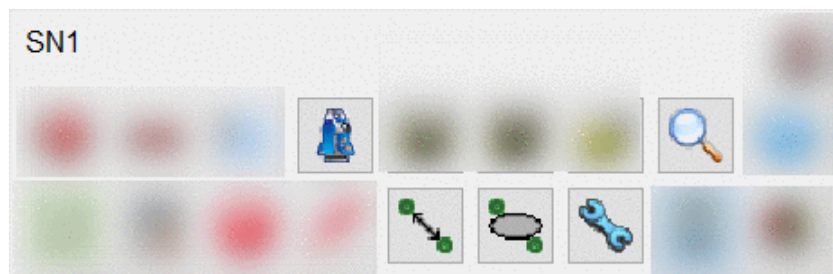
Your project will be drawn in the drawing (.DWG) file. Put focus back in the CAD program to view the network.



Once the network is drawn in CAD, you are able to use the CAD entities themselves interactively with the raw files. This allows you to make edits to data such as adding specific standard errors to points and measurements with greater ease. See Raw Data Files and Raw Data Editor Functions for more on editing raw data.

Tools Menu

There are a variety of **Tools** for working within the SurvNET project manager.



From left to right/top to bottom, the tools are:

EDM Calibration - offers an option to enter field measurements and calibrate to a published base line.

Point Search - Searches for points by name

Inverse - calculates the direction and horizontal distance between two points

Relative Error Ellipses - calculates the error ellipse between two points

Conversion Tools - converts data formats of raw input files

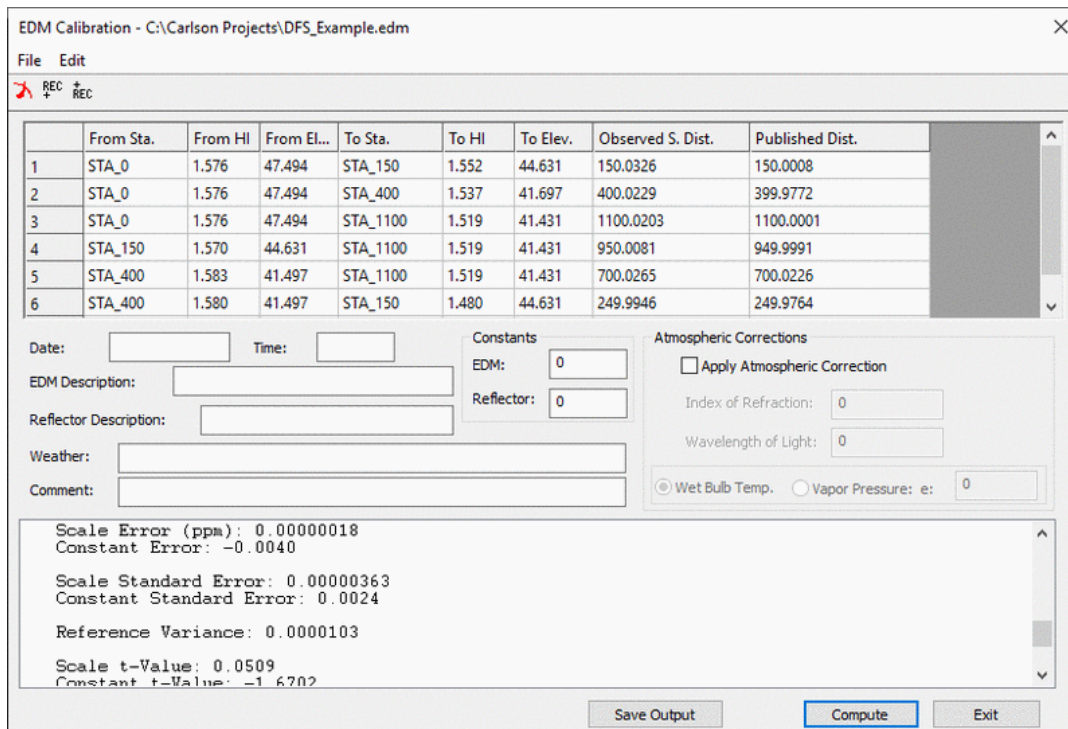
- Convert GPS/Total Station File
- Convert Level Files

The EDM Calibration program allows a surveyor to enter and process the raw data collected on an EDM calibration

baseline. The purpose of an EDM calibration is to determine if the EDM is measuring within standards. The program performs a statistical analysis of that data as outlined in "Use of Calibration Base Lines", by Charles J. Fronczek, NOAA Technical Memorandum NOS NGS-10. The NGS document can be downloaded from the NGS website. NGS maintains a webpage on EDM Calibration Base Lines. The manual and other information on EDM calibrations can be found at <https://www.ngs.noaa.gov/CBLINES/calibration.shtml>. Following is the main EDM Calibration dialog box. NGS publishes the EDM calibration data in metric units. SurvNET's EDM calibration program currently expects the data to be collected in meters.

	From Sta.	From HI	From EL...	To Sta.	To HI	To Elev.	Observed S. Dist.	Published Dist.
1	STA_0	1.576	47.494	STA_150	1.552	44.631	150.0326	150.0008
2	STA_0	1.576	47.494	STA_400	1.537	41.697	400.0229	399.9772
3	STA_0	1.576	47.494	STA_1100	1.519	41.431	1100.0203	1100.0001
4	STA_150	1.570	44.631	STA_1100	1.519	41.431	950.0081	949.9991
5	STA_400	1.583	41.497	STA_1100	1.519	41.431	700.0265	700.0226
6	STA_400	1.580	41.497	STA_150	1.480	44.631	249.9946	249.9764

The basic flow of this program is to first fill out the lower portion of the dialog box which contains different text fields, EDM constant values, and the optional Atmospheric Corrections settings. Next, fill out the grid in the upper portion of the dialog box. This grid contains the field data collected and also the published distances between monuments of the baseline. After this information has been filled out use the 'Compute' button. The program will then display the result of the calibration in the window in the lower portion of the dialog box as follows.



After the file is processed, the results can be stored as an ASCII text file. Use the **Save Output** or the menu option "File/Save Results File As..." to save the results. First, you will be prompted for an output file name. The input data can also be stored. Once stored it can be opened and processed again.

Following is the entire output with a brief explanation of the results. Comments about the results are inserted in **bold**:

```
EDM Calibration Report
  Observed Data
EDM Type:
Date:      Time:
Prism description:
Weather description:
Comment:
Atmosphere Correction: OFF
Constants: Refractor: 0.000    EDM: 0.000
From      From      From      To      .      To      To      Observed  Published
Sta.      Elev.      HI      Sta.      Elev.      HI      Temp.  Pressure  Slope Dist. Dist.
STA_0     47.494    1.576    STA_150  44.631    1.552    0.0    0.0      150.0326  150.0008
STA_0     47.494    1.576    STA_400  41.497    1.537    0.0    0.0      400.0229  399.9772
STA_0     47.494    1.576    STA_1100 41.431    1.519    0.0    0.0      1100.0203 1100.0001
STA_150   44.631    1.570    STA_1100 41.431    1.519    0.0    0.0      950.0081  949.9991
STA_400   41.497    1.583    STA_1100 41.431    1.519    0.0    0.0      700.0265  700.0226
STA_400   41.497    1.580    STA_150  44.631    1.480    0.0    0.0      249.9946  249.9764
STA_400   41.497    1.580    STA_0     47.494    1.526    0.0    0.0      400.0260  399.9722
```

The above section shows the input. The input consists of the observed slope distances and the measured HI's. The from and To elevations are published data from the data sheet from NGS on the particular baseline being observed. The published distances are also published data from the data sheet from NGS. In this example atmospheric pressure was turned off so the Temperature and Pressure fields are irrelevant.

Results

Null Hypothesis, H_0 : EDM scale error and EDM constant error = 0.0

If the scale error and the EDM constant are 0.0, then the EDM is without error. So the purpose of the statistical test is to test how close to 0.0 are the results.

Scale Error (ppm): -0.00000044

Constant Error: -0.0032

The two above lines show the values for the computed scale error and constant error.

Scale Standard Error: 0.00000403
Constant Standard Error: 0.0026

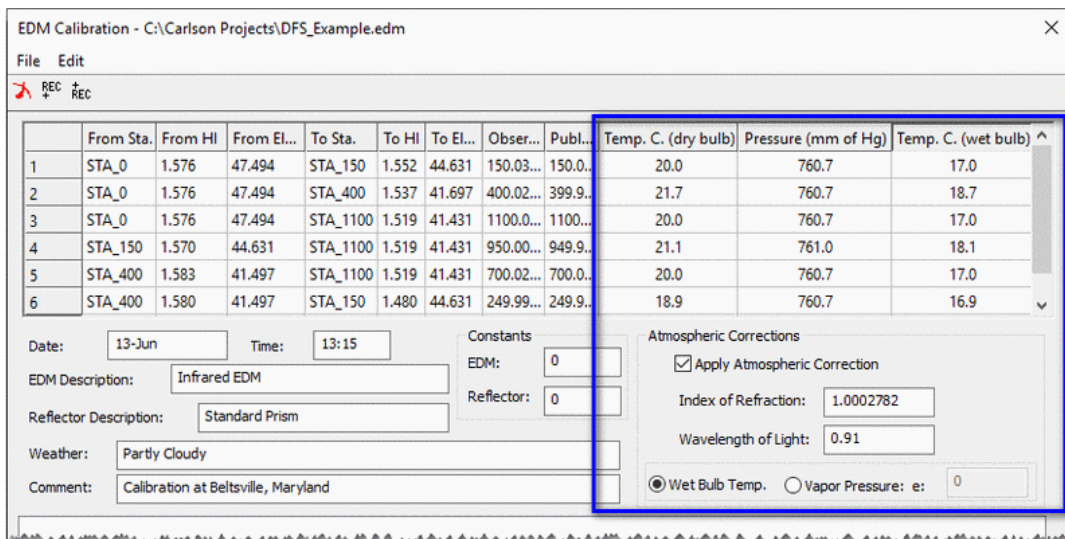
The two above lines show the values for the computed standard errors of the scale error and constant error.

Reference Variance: 0.0000126
Scale t-Value: -0.1096
Constant t-Value: -1.2110
Degrees of Freedom: 5
Critical t-Value at the 1 percent confidence level: 4.0320
Cannot reject the H0 for the scale error. (The scale factor is 0.0)
Cannot reject the H0 for the constant error. (The constant is 0.0)

The above lines show the final results of the statistical test. Since the test determined that we cannot reject the null hypothesis, this EDM is in good working order.

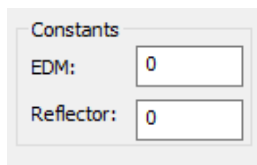
EDM Calibrations and Atmospheric Corrections

The atmospheric correction algorithms used in the EDM calibration are from the NGS manual. To use this method both dry-bulb and wet-bulb temperature needs to be measured, or the vapor pressure, e, and the dry bulb temperature needs to be measured. Refer to the NGS documentation for a detailed explanation of the atmospheric corrections that they use.

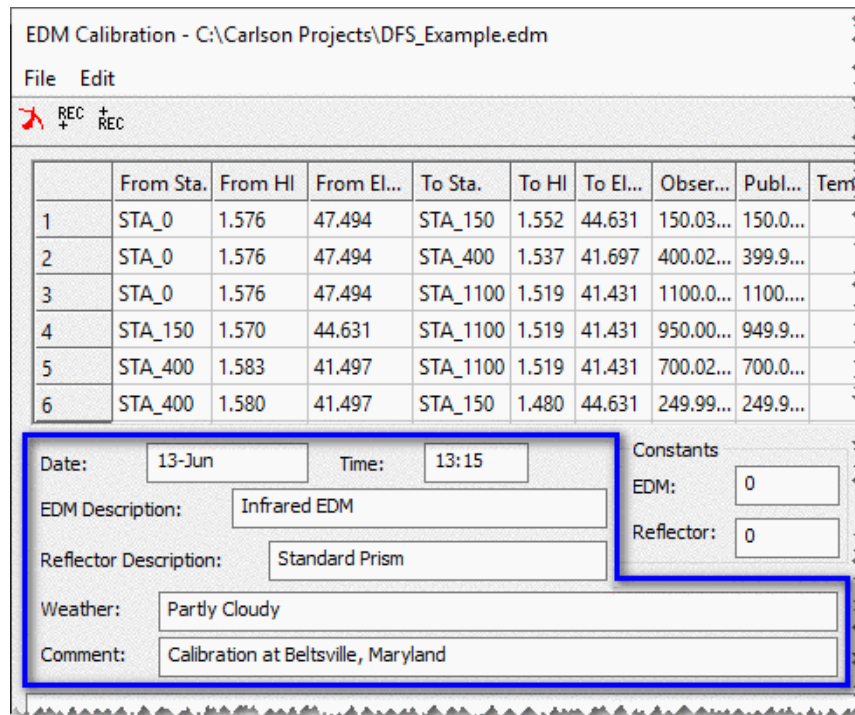


It is probably most common to turn atmospheric correction off in the calibration program, and turn atmospheric correction ON on the EDM (total station). When atmospheric correction is turned off in the calibration program, the user does not need to enter the temperature into the grid or any of the other atmospheric values. If atmospheric corrections are turned OFF then the grid input columns 'Temp. (dry bulb)', 'Pressure, (mm of Hg)', and 'Temp. (wet bulb)' will not be displayed since they are not needed.

Constants can be entered for both the EDM and the reflector. These values are added to the observed distances during processing. Typically they are set to 0.0.

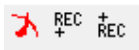


The following text fields have no effect on any computations and are simply comments that can be used to document the calibration.

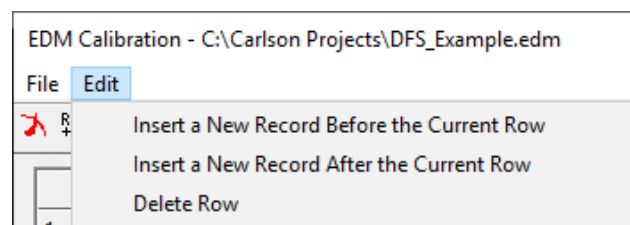


Entering Data Into the EDM Calibration Grid

Blank data records are inserted into or deleted from the grid using the following tool bar.



The first button deletes the current highlighted record. The second button inserts a new blank record before the current highlighted record. The third button inserts a new blank record after the current highlighted record. Alternately the 'Edit' menu options could be used to delete and insert new data records.



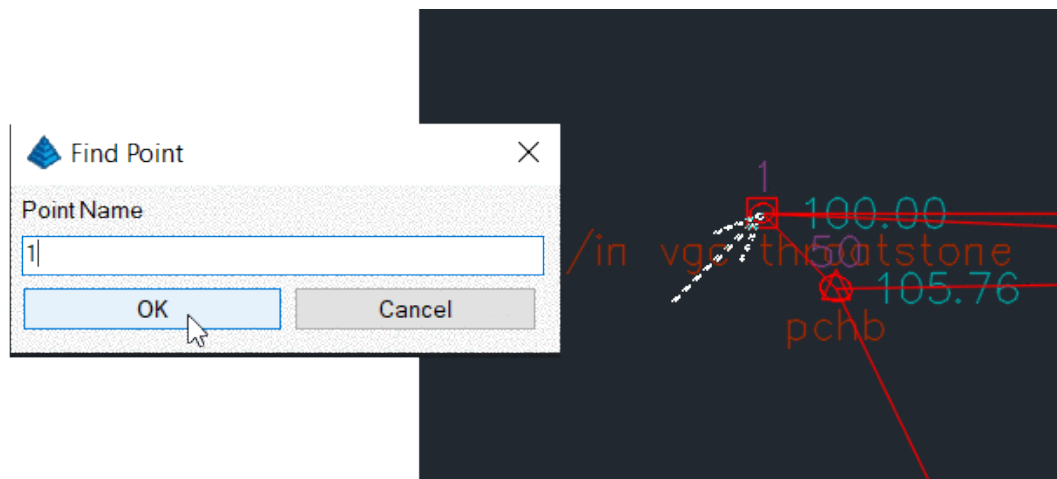
Following is a brief explanation of the fields that make up the grid.

	From Sta.	From HI	From El...	To Sta.	To HI	To El...	Obser...	Publ...	Temp. C. (dry bulb)	Pressure (mm of Hg)	Temp. C. (wet bulb)
1	STA_0	1.576	47.494	STA_150	1.552	44.631	150.03...	150.0...	20.0	760.7	17.0
2	STA_0	1.576	47.494	STA_400	1.537	41.697	400.02...	399.9...	21.7	760.7	18.7
3	STA_0	1.576	47.494	STA_1100	1.519	41.431	1100.0...	1100....	20.0	760.7	17.0
4	STA_150	1.570	44.631	STA_1100	1.519	41.431	950.00...	949.9...	21.1	761.0	18.1
5	STA_400	1.583	41.497	STA_1100	1.519	41.431	700.02...	700.0...	20.0	760.7	17.0
6	STA_400	1.580	41.497	STA_150	1.480	44.631	249.99...	249.9...	18.9	760.7	16.9

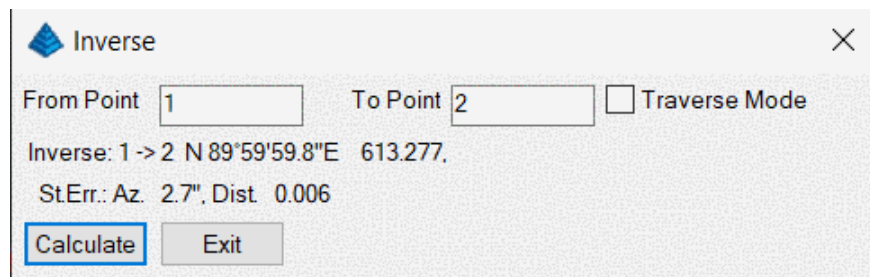
- **From Sta.** - This field represents the station name where the EDM is located. Any name can be used, but you must be consistent and used the same name whenever you occupy or measure a distance to the station.

- **From HI.** - This field represents the height of instrument of the from station. It should be in the same units as the measurements. If horizontal distances are being entered into the grid then all the HI fields should be set to a constant value such as 0.0.
- **From Elev.** - This field represents the elevation of the station. This value is published as part of the baseline calibration sheets obtained from NGS. If horizontal distances, then all the Elevation fields should be set to a constant.
- **To Sta.** - This field represents the station name where the prism is located. Any name can be used, but you must be consistent and used the same name whenever you occupy or measure a distance to the station.
- **To HI.** - This field represents the height of instrument of the to station. It should be in the same units as the distance measurements. If horizontal distances are being entered into the grid, then all the HI fields can be set to a constant value such as 0.0.
- **To Elev.** - This field represents the elevation of the station where the prism is located. This value is published as part of the baseline calibration sheets obtained from NGS. If horizontal distances, then all the Elevation fields should be set to a constant.
- **Observed S. Dist.** - This is the measured slope distance. This can be a measured horizontal distance. If it is a horizontal distance then all the HI's and elevations should be set to a constant value.
- **Published Dist.** - This field represents the published distance between the From and To station. This value is published as part of the baseline calibration data obtained from NGS for the particular baseline being observed.
- **Temp. C. (dry bulb)** - This field is only present if atmospheric corrections are turned on. Provide the dry bulb temperature in degrees Centigrade.
- **Pressure. (mm of Hg)** - This field is only present if atmospheric corrections are turned on. Provide the atmospheric pressure in millimeters of Mercury.
- **Temp. (wet bulb)** - This field is only present if atmospheric corrections are turned on. Provide the wet bulb temperature in degrees Centigrade.

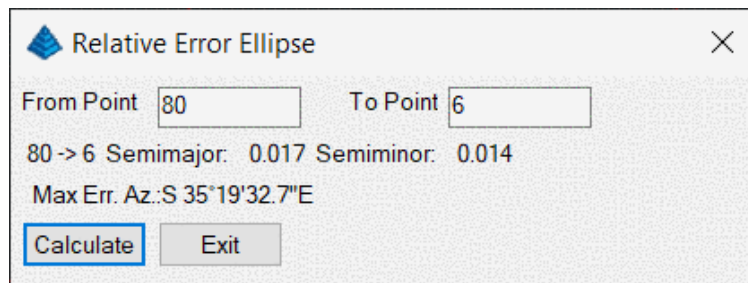
Searches and indicates the location of a point in the drawing file.



The 'Inverse' command is only active after a network has been processed successfully. This option can be used to obtain the bearing and distance between any two points in the network. Additionally the standard deviation of the bearing and distance between the two points is displayed.

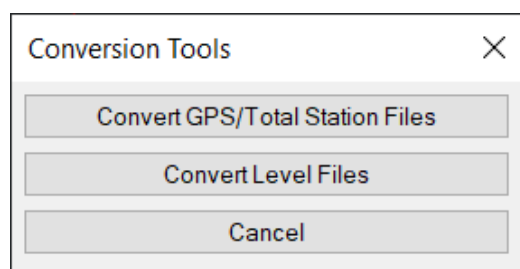


The 'Relative Error Ellipse' command is only active after a network has been processed successfully. This option can be used to obtain the relative error ellipse between two points. It shows the semi-major and semi-minor axis and the azimuth of the error ellipse, computed to a user-define confidence interval. This information can also be used to determine the relative precision between any two points in the network. It is the relative error ellipse calculation that is the basis for the ALTA tolerance reporting. If the Enable sideshots for relative error ellipses toggle is enabled, then all points in the project can be used to compute relative error ellipses. The trade-off is that with large projects, processing time will be increased.



If you need to certify as to the "Positional Tolerances" of your monuments, as per the ALTA Standards, use the **Relative Error Ellipse** inverse routine to determine these values, or use the specific ALTA tolerance reporting as you desire.

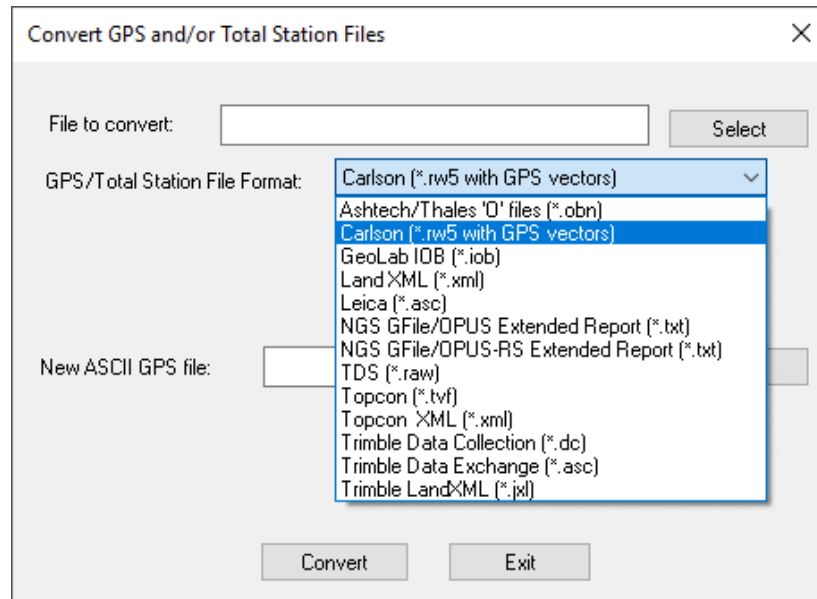
For example, if you must certify that all monuments have a positional tolerance of no more than 0.07 feet with 50 PPM at a 95 percent confidence interval. First set the confidence interval to 95 percent in the Settings/Adjustment screen. Then process the raw data. Then you may inverse between points in as many combinations as you deem necessary and make note of the semi-major axis error values. If none of them are larger than 0.07 feet + (50PPM*distance), you have met the standards. It is however more convenient to create a Relative Error Points File containing the points you wish to check and include the ALTA tolerance report. This report takes into account the PPM and directly tells you if the positional tolerance between the selected points meets the ALTA standards.



Convert GPS/Total Station Files

The purpose of this option is to convert GPS vector files that are in the manufacturers' binary or ASCII format into the StarNet ASCII file format. The advantage of creating an ASCII file is that the ASCII file can be edited using a standard text editor. Being able to edit the vector file may be necessary in order to edit point numbers so that the point numbers in the GPS file match the point numbers in the total station file.

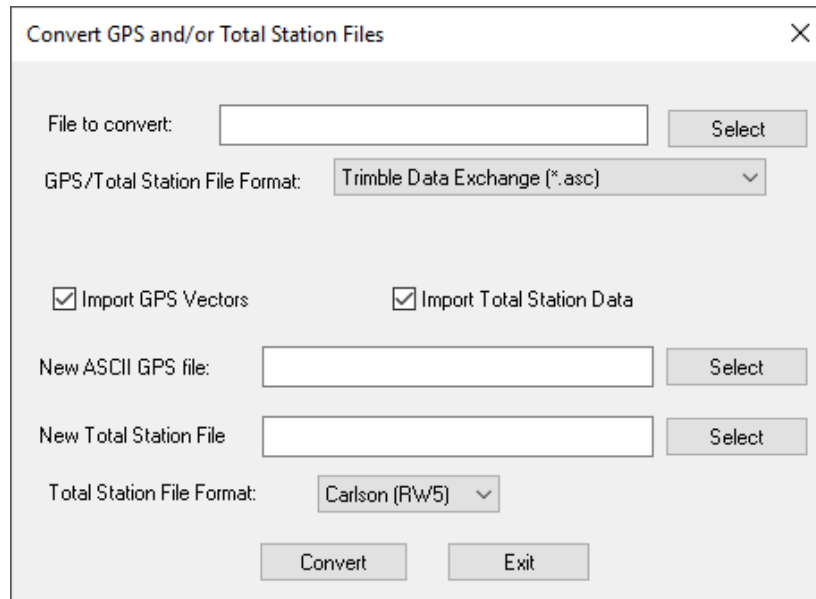
There is also a tool to convert Trimble Data Exchange total station data to either the Carlson RW5 format or the C&G CGR format. The following dialog box is displayed after choosing this option.



First choose the file format of the GPS vector file to be converted. Next use the 'Select' button to navigate to the vector file to be converted. If you are converting a Thales file you have the option to remove the leading 0's from Thales point numbers. Next, use the second 'Select' button to select the name of the new ASCII GPS vector file to be created. Choose the 'Convert' button to initiate the file conversion. Press the 'Cancel' button when you have completed the conversions. The file created will have an extension of .GPS. Following are examples of different GPS formats that can be converted to ASCII:

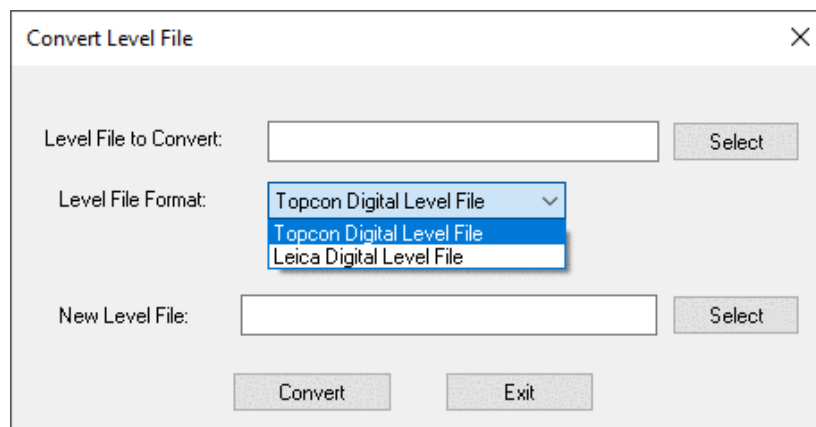
- **Ashtech/Thales 'O' files:** - Typically have .obn extensions and are binary files. Notice that you have the option to remove the leading 0's from Thales point numbers, by checking the "Remove leading 0's from Thales point numbers" check box.
- **Carlson .RW5** - (containing GPS vectors)
- **GeoLab IOB**
- **LandXML (.XML):** - The LandXML format is an industry standard format. Currently SurvNET will only import LandXML survey point records. The conversion will not import LandXML vectors.
- **Leica:** - The Leica vector file is an ASCII format typically created with the Leica SKI software. This format is created by Leica when baseline vectors are required for input into 3rd party adjustment software such as SurvNET. The SKI ASCII Baseline Vector format is an extension of the SKI ASCII Point Coordinate format.
- **NGS G-File:** - National Geodetic Survey format.
- **NGS G-File from OPUS report:** - National Geodetic Survey format as used in the OPUS report
- **TDS (.RAW):** - Import vector records from a TDS raw data file.
- **Topcon (.TVF):** - The Topcon Vector File is in ASCII format and typically has an extension of .TVF
- **Topcon (.XML):** - The Topcon XML file is an ASCII file. It contains the GPS vectors in an XML format. This format is not equivalent to LandXML format.
- **Trimble Data Exchange Format (.ASC):** - The Trimble TDEF format is an ASCII file. It is typically output by Trimble's office software as a means to output GPS vectors for use by 3rd party software.
- **Trimble Data Collection (.dc):** - The Trimble .dc format is an ASCII file. It is typically output by Trimble's data collector. It contains a variety of measurements including GPS vectors. This option only converts GPS vectors found in the .DC file.
- **Trimble LandXML (*.jxl):** This is Trimble's Land XML format.

NOTE:The Trimble Data Collection (.dc) and Trimble LandXML (.jxl) formats allow you to bring in both GPS vectors (creating a .GPS file) and Total Station data (creating a .RW5 file) at the same time.



Convert Level Files

The purpose of this option is to convert differential level files from digital levels into C&G/Carlson differential level file format. At present the only level file format that can be converted are the level files downloaded from the Topcon digital levels.

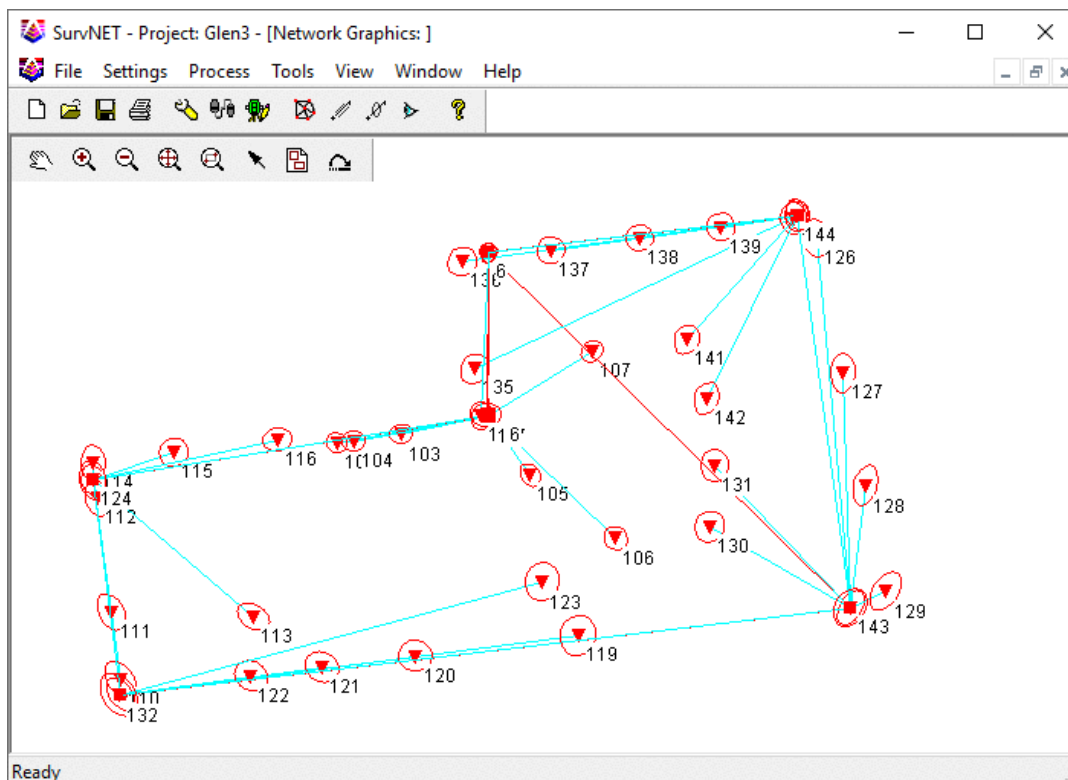


View Menu








Graphics

SurvNET provides a window that graphically displays the survey network. Additionally the user is able to display Error Ellipses, and GPS vectors. The user has much control over how the network is displayed. The graphic tool is a useful tool in debugging networks since the raw data can be displayed prior to adjustment. If there are problems with the raw data the graphics often reflect the problem. The actual graphics cannot be output or saved. The graphics can be shown independent of whether the project has been processed.

The following snapshot shows a view of the graphic window. The graphic window can be accessed using the "eye" icon on the main tool bar. A project must be opened before the graphic window can be displayed. The graphics window will only display Error Ellipses after the project has been processed.



The tool bar in the graphics window contains buttons that allow the user to pan, zoom in, zoom out, zoom extents, and zoom to a window. Additionally there is a button that allows the user to navigate to points in the .CGR raw data editor. Also, there are buttons that will refresh the graphic, and change the graphic settings.

Icon	Friendly Name	Action
	Pan	Use this button to position the graphics to a different position on the display.
	Zoom In	Use this button to increase the magnification of the graphics.
	Zoom Out	Use this button to decrease the magnification of the graphics.
	Zoom Extents	Use this button to fit the content of the entire project into the limits of the available display.
	Zoom Window	Use this button to display the content of the project within the rectangular confines defined by two corner "pick-points".
	Pick Point	This button allows the user to navigate within the (.CGR) Raw File Editor from the graphics window. It also allows you to pick points when creating a Traverse Closure file.
	Graphics Settings	This buttons is used to change the graphic display settings.

This button will refresh the graphic view. Graphics are generated from the saved raw data file. If you make changes to the raw file in the raw editor you must save the project before the changes will be reflected in the refreshed graphic screen.

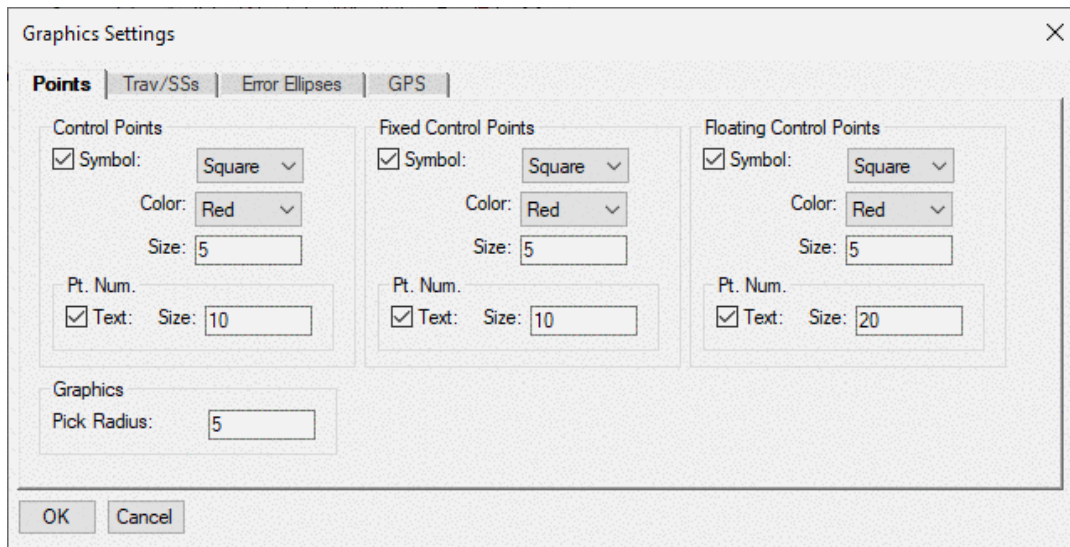
Display Controls

Following is a description of the options in the Graphics Setting dialog box.

Points Options

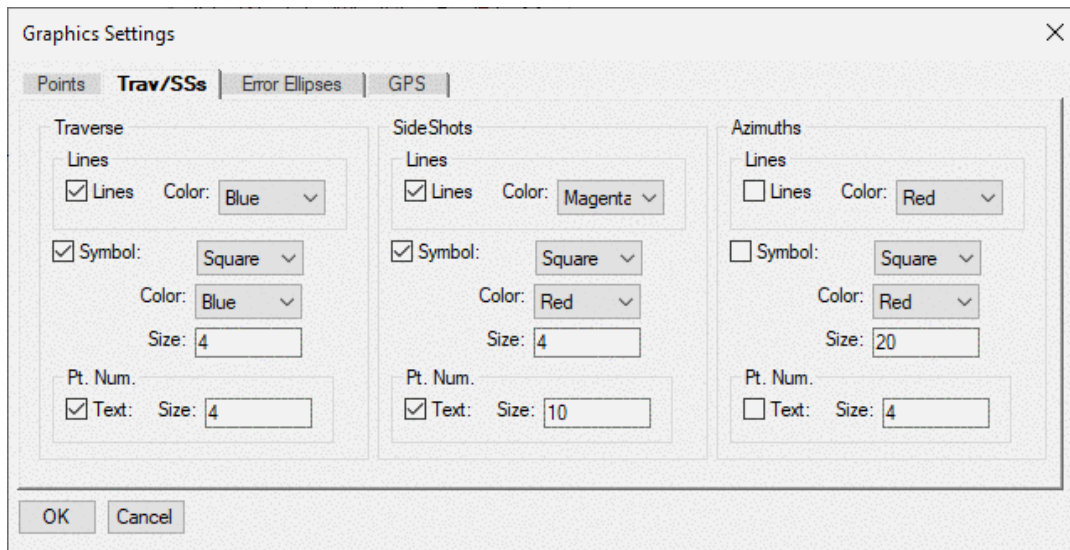
These settings determine how the different type control points are displayed in the graphics window. Different graphic settings can be applied to standard control points, fixed control points and floating control points. The symbol node display can be controlled as to symbol type, symbol color, symbol size. The control point name can be displayed and its size set from this setting dialog box.

The graphic pick radius defines a search radius. This radius is used when navigating the .CGR editor using the graphic window. You can pick a point graphically and the cursor in the editor will go to the next field containing that point number. The radius is defined in terms of the distance units of the raw data file.



Trav/SS's Options

These settings determine how the network line work will be displayed for total station raw data. There are settings for traverse data, side shot data, and azimuth control. The program considers any point that has only a single angle and distance to it a side shot. The user can control the color of the traverse lines. The symbol node display can be controlled as to symbol type, symbol color, symbol size. The point name can be displayed and its size set from this setting dialog box.



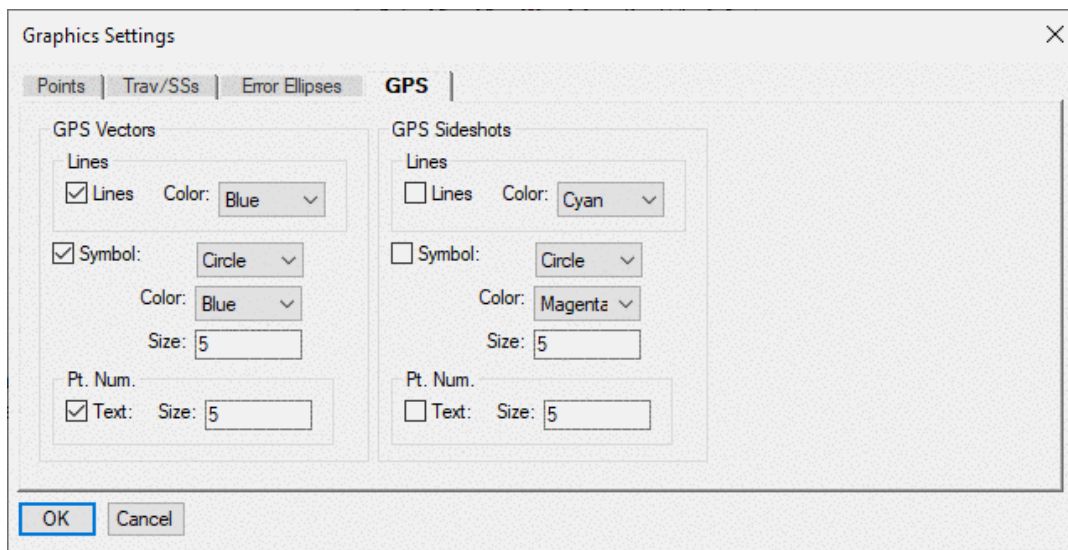
Error Ellipses Options

These settings determine how the error ellipses will be displayed in the graphic window. Error ellipses will only be displayed if there is a successful least squares adjustment. The display of the error ellipses is relative. The program automatically determines a default relative error ellipse size. The user can modify the visual size of the error ellipses using the track bar in the following dialog box. The user can also control the color of the error ellipse from the following dialog box.



GPS Options

The settings in the following dialog box determine how GPS vectors will be displayed in the graphic window. The user can control the color of the GPS vector lines. The symbol node display can be controlled as to symbol type, symbol color, symbol size. The GPS point names can be displayed and their size set from this setting dialog box. GPS side-shots (points defined by a single vector) can be controlled separately.



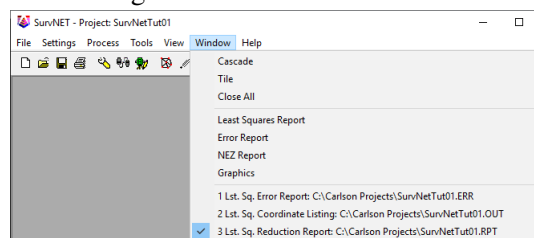
SurvNET Window Menu

Selecting the **Window** menu opens the following menu:

Menu Command(s)

Menu Image

- Cascade
- Tile
- Close All
- Least Squares Report
- Error Report
- NEZ Report
- Graphics



Window Menu

This command option will cause the open "reports" to cascade in sequence so that their dialog box title bars are most likely visible... permitting you to select the desired window to review the material.

This command option will cause the open "reports" to fit into a tile pattern that fills the SurVNET Window application so that all open windows are simultaneously visible at the same time.

This command option will cause all of the open "reports" close but will leave the SurVNET application (and the SurVNET project) open. To re-open the report(s), either select the desired report from the Window menu or perform a Network Adjustment.

This option opens/displays the Least Squares report.

This option opens/displays the Error report.

This option opens/displays the NEZ (Northing, Easting, Elevation) report.

This option opens/displays the Graphic result of the network.

Toolbars

Many of the more commonly used functions can be accessed using the toolbar:



Icon	Menu Location	Action
	File – New Project	Creates a New Project.
	File – Open Project	Opens an Existing Project.
	File – Save Project	Saves the Current Project.
	File – Print	Prints the Specified Output.
	Settings – Project Settings – Standard Errors	This icon initiates the Project Settings – Standard Errors tab.
	Tools – Data Collector Transfer	[Legacy Application] - This icon will initiate either the specified "handshaking" program designed to communicate with numerous field collection data collection devices thus permitting the user to directly transfer data from the field device into SurvNET (and vice versa).
	Tools – Edit Raw Files	This icon can be used to review or edit your raw data file(s). If your project has multiple raw data files, you will be shown a list and asked to select the file you wish to edit.
	Process – Network Adjustment	This icon will process the available data against the specified Project Settings and displays any pertinent results or alerts.
	Tools – Inverse	This icon will provide the ability to inverse between a pair of specified points (once the network has been adjusted) to get the angle and distance information between the specified points.
	Tools – Relative Error Ellipse	This icon will provide the ability to generate the Relative Error Ellipse information between a pair of specified points (once the network has been adjusted).
	Window – Graphics	This icon is active once a project has been opened and displays the graphical results (if any) of the traverse, sideshot(s) and GPS points.
	Help – SurvNET Help	This icon will take you to the SurvNET Help/on-line documentation.

SurvNET Toolbar Short-cuts

Raw Traverse Data

SurvNET works equally well for both Carlson users and C&G users. The primary difference between the two users is that a Carlson user will typically be using an .RW5 file for raw data and a C&G user will typically be using a .CGR file.

SurvNET is capable of processing either C&G (.CGR) raw data files, Carlson (.RW5) raw data files or SDMS (.PRJ) raw data files. If the raw data is in another format, you can use our conversion tools to create one of the supported formats. Select the desired type of raw file editor as described under Settings – Global Settings.

Measurement, coordinate, elevation and direction (Brg/Az) records are all recognized. Scale factor records in the .CGR file are not processed since SurvNET calculates the State Plane scale factors automatically.

Once raw data is entered, the network can be processed. Often it is desired to apply a standard error for a particular measurement or control point that is different than those set in the Project Settings. This can be accomplished by editing the Standard Errors of specific or groups of points or measurements.

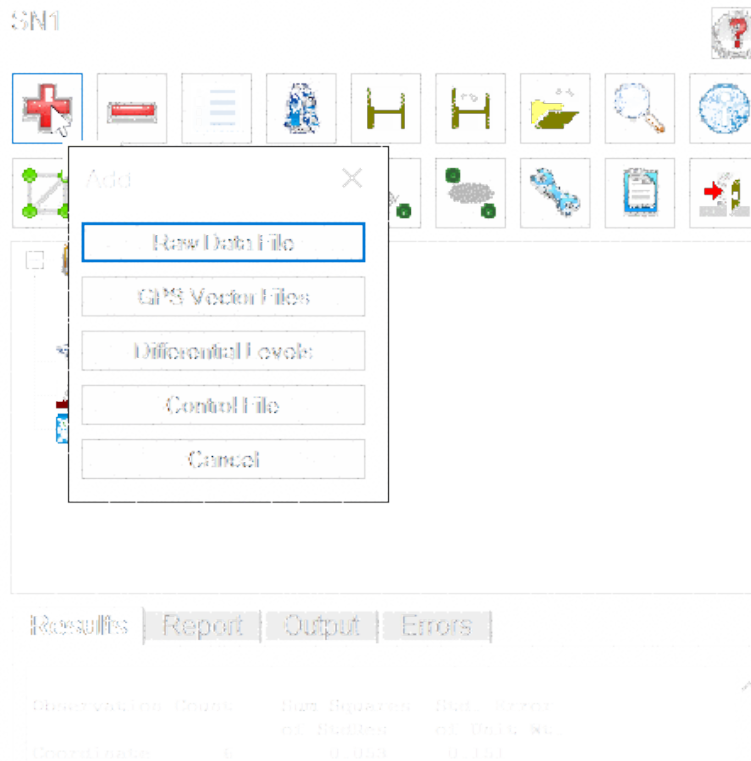
There are four types of Raw Data Files files you can add and process as part of a single network:

Raw Data File - traditional total station files (usually ground measurements)

GPS Vector File - GPS observations in the form of vector data

Differential Levels - standard differential level or Trig observations for vertical control

Control File - point records to be used as control for the network.



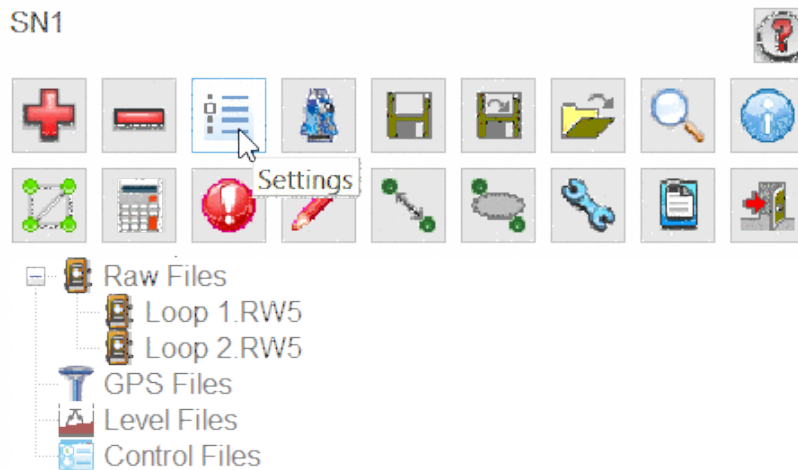
Raw Data File

To process traverse raw data, click the **Add** (red + button) and select the **Raw Data File** to insert raw total station files into the list. Use **Remove** (red - button) to remove raw files from the list. All the files in this list are included in the Least Squares adjustments. Having the ability to choose multiple files allows one to keep Control in one file and measurements in another file. Or, different files collected at different times can be processed all at one time. If you have multiple crews working on the same project using different equipment, you can have "crew-specific" raw data files with standard error settings for their particular equipment. Having separate data files is also a convenient method of working with large projects. It is often easier to debug and process individual raw files. Once the individual files are processing correctly, all the files can be included for a final adjustment. You can either enter C&G (.CGR) raw files or Carlson (.RW5) files into the list for processing. You cannot have both .CGR and .RW5 files in the same project to be processed at the same time. Notice that you have the ability to highlight multiple files when removing files from the project.

Note that if you are processing Carlson RW5 total station files, you have the option of including GPS vectors that

can be collected by SurvCE/SurvPC and stored in the RW5 file. Alternatively, you can select the RW5 file in the GPS vector section. **If the "Include any GPS vectors" option in the Total Station Data Files section is checked, you will have the option to use the Base Point records in the RW5 file as control.** If "Use BP records as control" is NOT checked, or the RW5 file is selected in the GPS vector section, you must define the control in either the RW5 file as POINT records or in a Supplemental Control File.

To set these options, select the **Settings** option from the main menu.



GPS Vector Files

GPS vector files can be entered and processed. Both GPS vector files and total station raw files can be combined and processed together. **You must have chosen the 3D mathematical model in the Coordinate System tab in order to include GPS vectors in the adjustment.**

A variety of popular GPS vector file formats are supported, including (but not necessarily limited to):

- **ASCII (StarNet):** See below for more information on *StarNet* format. These files typically have .GPS extensions.
- **Ashtech/Thales 'O' files:** Typically have .obn extensions and are binary files.
- **Carlson .RW5:** (containing GPS vectors)
- **GeoLab IOB**
- **LandXML (*.xml)**
- **Leica:** Leica files are ASCII files.
- **NGS G-File**
- **NGS G-File from OPUS report**
- **Topcon (.tvf):** Topcon .tvf files are ASCII files.
- **Topcon (.xml):** Topcon also can output their GPS vectors in XML format which is in ASCII format.
- **TDS (.raw):** TDS raw data file with vector information
- **Trimble Data Exchange Format (.asc):** These files are in ASCII format
- **Trimble data collection (.dc):** These files are ASCII.
- **Trimble LandXML (*.jxl)**

The following is a typical vector record in the StarNet ASCII format. GPS vectors typically consist of the 'from' and 'to' point number, the delta X, delta Y, delta Z values from the 'from' and 'to' point, with the XYZ deltas being

in the geocentric coordinate system. Additionally the variance/covariance values of the delta XYZ's are included in the vector file.

```
.GPS WEIGHT COVARIANCE
C PRS34452 1305780.345005 -4667085.299019 4132689.544939 0.005000 0.005000 0.005000 --MON
C UCNJ 1305780.345005 -4667085.299019 4132689.544939 0.00000100 0.00000100 0.00000100 --MON
G1 UCNJ-1000 8399.71318061 -4742.15645068 -8036.07224424 --MNS
G2 3.939428e-006 2.474560e-005 1.160301e-005
G3 3.924536e-006 -3.360765e-006 -1.028503e-006
G1 UCNJ-1001 8328.15569476 -4796.59445569 -8072.25948922 --MNS
G2 9.596618e-007 1.687749e-005 1.936038e-006
G3 1.176891e-007 -8.668020e-009 -4.798408e-006
```

The first line defines what values are in the G2 and G3 records. It can be either GPS WEIGHT COVARIANCE (G2 is Variance, G3 is Covariance), or GPS WEIGHT STDERRCORRELATION (G2 is standard error, G3 is standard error correlation). This line is optional; the default is COVARIANCE.

The next two lines are coordinate control records. These records are also optional. If used, they must be Geocentric Coordinates (XYZ) in metric units. The format is as follows:

```
C PointName X Y Z StdErrX StdErrY StdErrZ --PointDescription
```

The standard errors and point description are optional.

The GO record is a comment.

The G1 record includes the 'from' and 'to' points and the delta X, delta Y, and delta Z in the geocentric coordinate system and an optional description of the rover point.

The G2 record is the variance (or standard error) of X,Y, and Z. The G3 record contains the covariance (or standard error correlation) of XY, ZX, and ZY. Most all GPS vector files contain the same data fields in different formats.

Use the 'Add' button to insert GPS vector files into the list. Use the 'Remove' button to remove GPS vector files from the list. All the files in this list will be used in the least squares adjustments. All the GPS files in the list must be in the same format. If the GPS file format is ASCII you have the option to edit the GPS vector files. The Edit option allows the editing of any of the ASCII GPS files using Notepad. Typically, only point numbers or point descriptions would be edited. The variance/covariance values are used to determine the weights that the GPS vectors will receive during the adjustment and are not typically edited.

Differential Levels

Differential and Trig level files can be entered and processed. Differential or Trig Level raw files have a .TLV extension and are created using the Carlson Level Editor (**See Edit Process Level Data in the Survey Menu**). The Carlson Level Editor has several Import options that allow you to bring in levels from different formats (*e.g.* Leica, Trimble, Topcon, *etc.*). You can process level data in the same project with the traverse and GPS vector data but the vertical measurements in the traverse/vectors will be part of the adjustment. If you want to use the elevations calculated ONLY from the level data, you must process it in a separate project after your traverse/vector data has been adjusted. The elevations will be written to the Output Coordinate file. Make sure your point numbers in the level data match the points in the traverse/vector data.

Supplemental Control File: The supplemental control file option allows the user to designate an additional coordinate file to be used as Control. The supplemental control file can be from a variety of different file types, including (but not necessarily limited to):

- **C&G or Carlson numeric (.CRD) files**
- **C&G Alphanumeric coordinate files (*.cgc)**
- **Carlson Alphanumeric coordinate files (*.crd)**
- **Autodesk Land Desktop (*.mdb)**
- **Simplicity coordinate files (*.zak)**
- **ASCII (.NEZ) file**
- **ASCII latitude and longitude (3D model only)**

- **CSV ASCII NEZ with std. errors** (only external control file that allows you to assign standard errors to specific points)
- **SDMS (.ctl) control file**
- **ASCII Geocentric (.xyz)** (Geocentric coordinates XYZ in metric units)

Note: For additional information on file format pertaining to the file formats above, refer to the Supplemental Control Files section of the documentation.

Note: You will not be allowed to use the same file for supplemental control points and for final output. Least Squares considers all points to be measurements. If the output file is also used as a supplemental control file, then after the project has been processed all the points in the project would now be in the control file and all the points in the file would now be considered control points if the project was processed again. The simplest and most straight-forward method to define control for a project is to include the control coordinates in a raw data file.

Supplemental Control Files

In order to process a raw data file, you must have (as a minimum) a control point and a control azimuth, or two control points. Control points can be inserted into the raw data file or alternately control points can be read from a coordinate file as linked via the Input Files portion of the program.

The major advantage of putting coordinate control points in the actual raw data file is that specific standard errors can be assigned to each control point (as described in the Control Standard Errors section above). If you do not include an SE record, the standard error will be assigned from the NORTH, EAST, and ELEVATION standard errors from the Control Standard Error project default values found in the Project Settings dialog box.

The standard errors for the control points from a supplemental control file will be assigned the NORTH and EAST standard errors from the project settings dialog box.

In the ASCII .NEZ file, the coordinate records need to be in the following format:

```
Pt. No., Northing, Easting, Elevation, Description<cr><lf>
103, 123233.23491, 238477.28654, 923.456, Mon 56-7B<CR><LF>
```

Each line is terminated with the "non-printing" carriage-return <CR> and line-feed <LF> characters. When viewed in a simple ASCII file editor such as Windows Notepad, the file snippet above would resemble:

```
Pt. No., Northing, Easting, Elevation, Description
103, 123233.23491, 238477.28654, 923.456, Mon 56-7B
```

In the ASCII latitude and longitude file, the records need to be in the following format:

```
Pt. No., Latitude (NDDD.mmsssss), Longitude (WDDD.mmsssss), Elevation (Orthometric), Description<cr><lf>
FRKN,N35.113068642,W083.234174724,649.27<CR><LF>
```

Each line is terminated with the "non-printing" carriage-return <CR> and line-feed <LF> characters. When viewed in a simple ASCII file editor such as Windows Notepad, the file snippet above would resemble:

```
Pt. No., Latitude (NDDD.mmsssss), Longitude (WDDD.mmsssss), Elevation (Orthometric), Description
FRKN,N35.113068642,W083.234174724,649.27
```

In the ASCII XYZ Geocentric file, the records need to be in the following format:

```
Pt. No. X Y Z Descriptions<cr><lf>
105 1413426.6020 -4537671.2000 4239299.9360 MON<CR><LF>
```

Each line is terminated with the "non-printing" carriage-return <CR> and line-feed <LF> characters. When viewed in a simple ASCII file editor such as Windows Notepad, the file snippet above would resemble:

```
Pt. No. X Y Z Descriptions
105 1413426.6020 -4537671.2000 4239299.9360 MON
```

NOTE: It is not allowed for the supplemental control file and the final output file to be the same file. Since Least Squares considers all points to be control points, only control points should be in a supplemental control file.

Once raw data files are added to the network it can be adjusted using the parameters as set in the Project Settings. In many instances, there will be a need to assign specific standard errors for particular measurements or control points. This can be accomplished by editing individual points or measurements or by assigning a standard error to a particular raw data file.

Standard errors are estimated errors that are assigned to measurements or coordinates. A standard error is an estimate of the standard deviation of a sample. A higher standard error indicates a less accurate measurement. The higher the standard error of a measurement, the less weight it will have in the adjustment process.

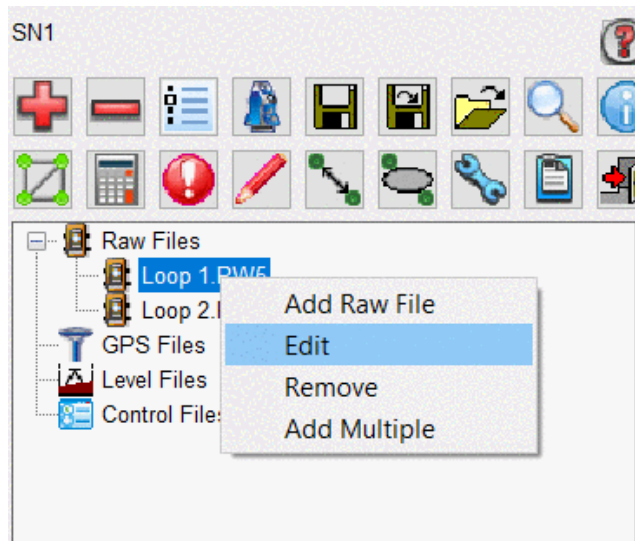
Although you can set default standard errors for the various types of measurements in the Project Settings of SurvNET, standard errors can also be placed directly into the raw data file. A standard error record inserted into a raw data file applies to all the measurements following the **SE** record. The standard error does not change until another SE record is inserted that either changes the specific standard error, or sets the standard errors back to the project defaults. The advantage of entering standard errors into the raw file is that you can have different standard errors for the same type measurement in the same job.

For example, if you used a 1" total station with fixed backsights and foresights for a portion of a traverse and a 10" total station with backsights and foresights to hand-held prisms on the other portion of the traverse, you would want to assign different standard errors to reflect the different methods used to collect the data.

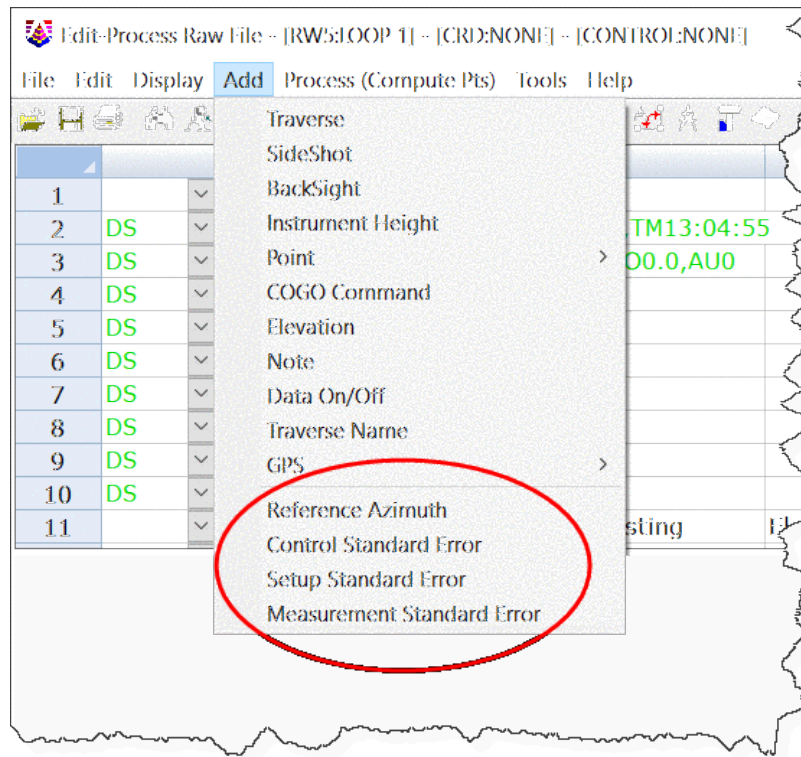
Make sure the SE record is placed before the measurements for which it applies. If you do not have standard errors defined in the raw data file, the default standard errors in the project settings will be applied to the entire file.

Editing Standard Errors

The dialog box option allows users to edit raw field data directly through a the typical Carlson data Editor by selecting a raw file and *right-clicking*:



Carlson Raw Data Editor



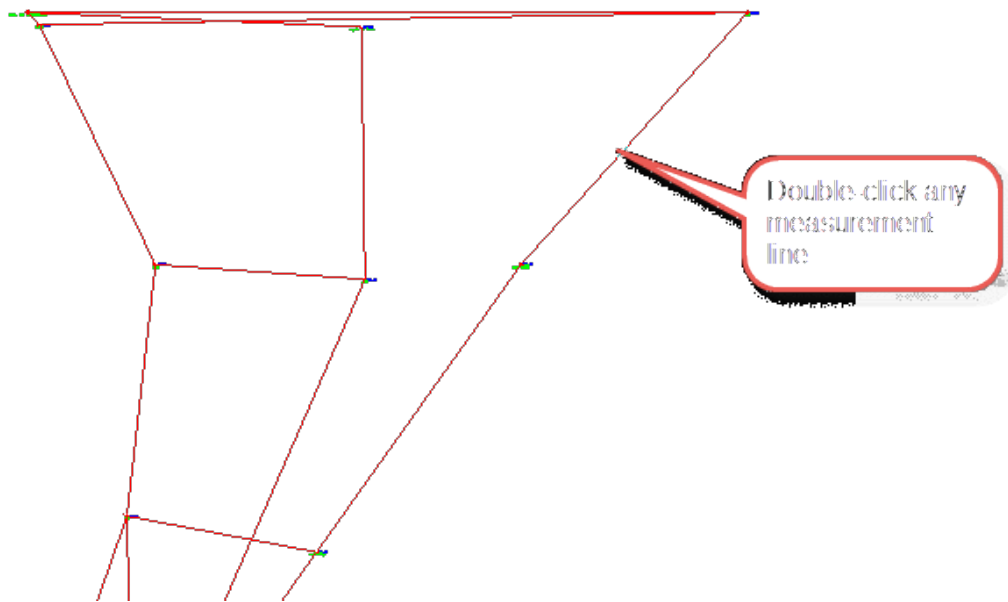
Note, editing standard errors in the Carlson Data Editor is described more fully in the **Standalone** version of this help menu.

Graphic CAD Editor

With the dialog box editor, users can review and edit information for points, vectors and measurement lines by

selecting the information button  or by double-clicking any entity.

Select any a graphic entity such as a traverse line or control point and *double-click* on it.



Line: 2 to 3

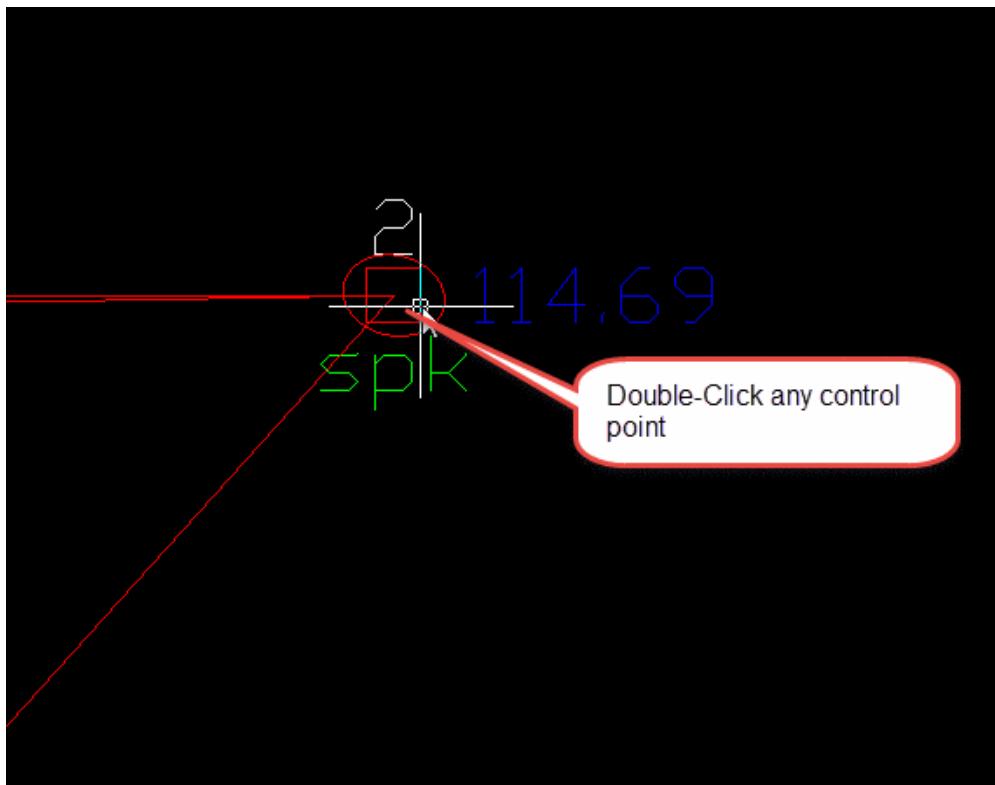
Measurement Data														
	BsPt	OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	InstHt	RodHt	Description	MeasureErr	SetupErr	File	Line
1	1	2	3	AR	311.5908	288.251	90.3704	5.290	4.600	mag/#9500	Default	Default	D:\Carlson f56	
2	1	2	3	AR	131.5902	288.251	269.2249	5.290	4.600	mag/#9500	Default	Default	D:\Carlson f58	
3	2	3	2	AR	179.5954	288.244	270.3406	4.800	5.170	spk	Default	Default	D:\Carlson f117	

Adjusted Data				
Distance	Residual	StdRes	StdDev	
288.234	0.003	0.288	0.004	
Angle	Residual	StdRes	StdDev	BsPt
311°59'02"	-0.252	0.041	2.626	1

OK Cancel Edit Measurement Errors Edit Setup Errors Edit Raw File

CSE - Control Standard Errors

You can insert or add Control Standard Error records for a specific point by double-clicking on a control point and selecting **Control Standard Error**



Control Point

Point 1 at N: 5000.000, E: 5000.002

Pt	Northing	Easting	Elevation	Description	Control Error	File	Line
1	5000.00000	5000.00000	100.00000	dh/in vgc thro:Default		D:\Carlson Files\Support\10	

OK Cancel Edit Control Errors Edit Raw File

You can set standard errors for Northing, Easting, Elevation, and Azimuth selecting the **Edit Control Errors** option. Azimuth standard errors are entered in seconds. The North, East and Elevation standard errors affect the PT (coordinate) and EL (elevation) records.

Control Standard Errors

*=Project Settings Default
 !=Fixed
 #=Float

Northing

Easting

Elevation

Azimuth(sec)

OK Cancel

You can hold the North, East and Elevation fixed by entering an exclamation "!" symbol. You can allow the North, East and Elevation to FLOAT by entering a hash (or "pound") "#" symbol. You can also assign the North, East and Elevation actual values. If you use an asterix "*" symbol, the current standard error value will return to the control standard error project defaults.

North	East	Elevation	Azim	
!	!	!		(Fix all values)
#	#	#	30.0	(Allow the N., E. & Elevation to Float)
0.01	0.01	0.03	5.0	(assign values)
*	*	*	*	(return the control standard errors back to project defaults)

When you fix a coordinate point, the original value does not change during the adjustment and all measurements will be adjusted to fit the fixed point. If you allow a value to float, it will not be used in the actual adjustment, it will just be used to help calculate the initial coordinate values required for the adjustment process. Placing a very high or low standard error on a coordinate point accomplishes almost the same thing as setting the standard error as float or fixed. The primary purpose of using a float point is if SurvNET cannot compute preliminary values, a preliminary float value can be computed and entered for the point.

Direction records cannot be FIXED or FLOAT. You can assign a low standard error (or zero to fix) if you want to weight it heavily, or a high standard error to allow it to float.

In the example below, the first CSE record containing the '!' character and sets points 103, 204, and 306 to be fixed. The last CSE record contains the '*' character. It sets the standard errors for point 478 and any other points that follow to the project settings. The Azimuth standard error was left blank.

MSE - Measurement Standard Errors

You can set the standard errors for various types of measurements for a specific measurement by *double-clicking* on a drawing entity (line or point) and selecting **Edit Measurement Errors**

	BsPt	OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	InstHt	RodHt	Description	MeasureErr	SetupErr	File	Line
1	4	5	6	AR	348.1209	492.842	90.0829	5.150	4.770	spk	Default	Default	D:\Carlson f214	
2	4	5	6	AR	168.1154	492.840	269.5115	5.150	4.770	spk	Default	Default	D:\Carlson f216	
3	5	6	7	AR	155.3838	214.811	90.4458	4.900	5.060	mag/in cc b	Default	Default	D:\Carlson f243	
4	5	6	7	AR	335.3823	214.810	269.1500	4.900	5.060	mag/in cc b	Default	Default	D:\Carlson f245	
5	5	6	5	AR	179.5944	492.841	270.0633	4.900	5.020	mag/#9501	Default	Default	D:\Carlson f248	
6	6	7	8	AR	93.1517	284.763	91.0127	5.180	4.870	dh/chk1	Default	Default	D:\Carlson f267	

Measurement Standard E... X

*=Project Settings Default

Distance Constant

Distance PPM

Horizontal Pointing

Horizontal Reading

Vertical Pointing

Vertical Reading

OK Cancel

Example:

	Distance	Point	Read	V.Point	V.Read	PPM
MSE	0.01	3	3	3	3	5

You can enter any combination of the above values. If you do not want to change the standard error for a particular measurement type, leave it blank.

If you use an "*" symbol, the standard error for that measurement type will return to the standard measurement error project defaults.

SSE - Setup Standard Errors

These standard errors are a measure of how accurately the instrument and target can be set up over the points as discussed in the Instrument and Target Standard Errors section. You can edit a particular setup by double-clicking on that point or measurement line and selecting **Edit Setup Errors**

	BsPt	OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	InstHt	RodHt	Description	MeasureErr	SetupErr	File	Line
1	4	5	6	AR	348.1209	492.842	90.0829	5.150	4.770	spk	Default	Default	D:\Carlson f214	
2	4	5	6	AR	168.1154	492.840	269.5115	5.150	4.770	spk	Default	Default	D:\Carlson f216	
3	5	6	7	AR	155.3838	214.811	90.4458	4.900	5.060	mag/in cc b	Default	Default	D:\Carlson f243	
4	5	6	7	AR	335.3823	214.810	269.1500	4.900	5.060	mag/in cc b	Default	Default	D:\Carlson f245	
5	5	6	5	AR	179.5944	492.841	270.0633	4.900	5.020	mag/#9501	Default	Default	D:\Carlson f248	
6	6	7	8	AR	93.1517	284.763	91.0127	5.180	4.870	dh/chk1	Default	Default	D:\Carlson f267	

Setup Standard Errors

*=Project Settings Default

Target Centering

Target Height

Instrument Centering

Instrument Height

OK Cancel

Example:

	Rod Ctr	Inst Ctr	Inst Ht	Rod Hgt
SSE	0.005	0.005	0.01	0.01

You can enter any combination of the above values. If you do not want to change the standard error for a particular measurement type, leave it blank.

If you use an "*" symbol, it will return the standard error to the Instrument and Target Standard Errors project default values.

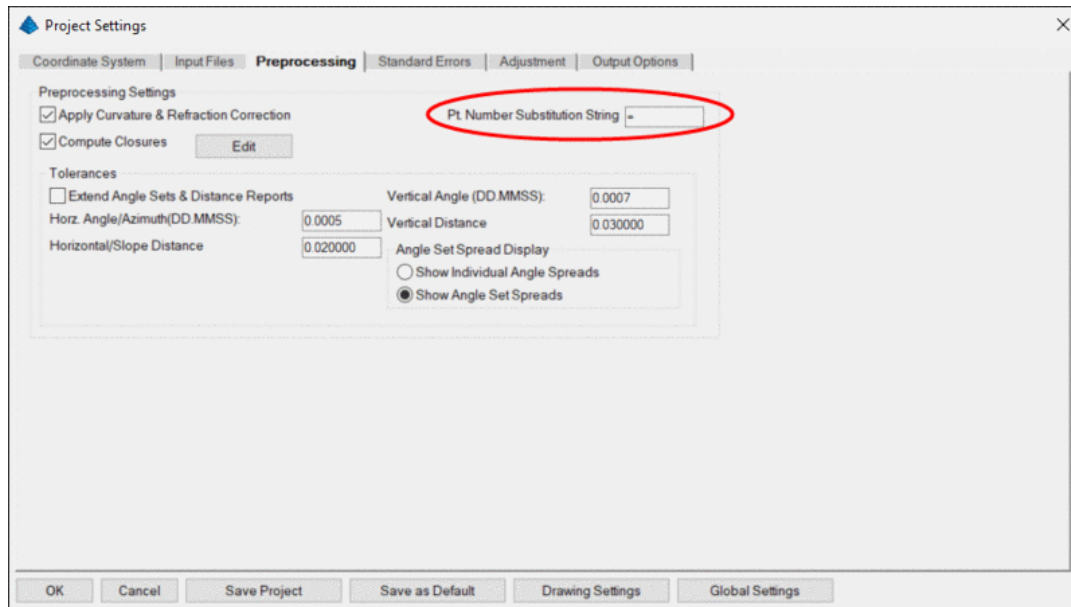
Redundant Measurements

One of the benefits of SurvNET is the ability to process redundant measurements. In terms of total station data, redundant measurement is defined as measuring angles and/or distances to the same point from two or more different

setups.

It is required that the **same point number** be used when locating a point that was previously recorded. However, since some data collectors will not allow you to use the same point number if the point already exists, SurvNET also uses the following convention for collecting redundant points while collecting the data in the field:

If the point description contains a user-defined string, for example an "=" (equal sign) followed by the original point number, SurvNET will treat that measurement as a redundant measurement to the point defined in the description field. The user-defined character or string is set in the Project Settings dialog. For example, if point number 56 has the description "=12", we will treat it as a measurement to point 12 and point 56 will not exist. Make sure the *Preprocessing Settings* dialog box has the **Pt. Number Substitution String** set to the appropriate value.



Alternately, the point numbers can be edited after the raw data has been downloaded from the data collector.

SurvNET Editor

Please refer to topic on Carlson or CGEditor raw editor.

Data Collector Transfer

Please refer to the Carlson or CG data collector transfer topic.

Example Projects

On the installation disk there are a variety of different least squares projects one can use to become familiar with least squares and SurvNET. These projects are located in the SurvNET Examples folder under the Carlson Projects folder (e.g. C:\Carlson Projects\SurvNET Examples).

NOTE: When you open a project for review, you will need to check the project settings, input data files to see if the paths are correct. If they do not, you will have to re-select them.

Simple Traverse with Traverse Closure

This project is located in \Carlson Projects\SurvNET Examples\2DTraverse. The name of the project is Traverse. This project illustrated a basic loop traverse with two control points and a known azimuth for control. This project also illustrates how to obtain traditional closure information as part of the least squares report. The program uses the 2D/1D model and uses a local coordinate system.

Traverse using State Plane Coordinates

This project is located in \Carlson Projects\SurvNET Examples\SPCTraverse. The name of the project is Trav01SPCUSft. This project illustrated a basic network with three GPS control points for control. This project is computed using the **SPC83 NC Grid** coordinate system. The project is set up to generate traditional loop closure data. The program uses the 2D/1D model. No elevations are computed or adjusted as there were no HI's or rod readings collected. Notice, that the project uses two raw data files. One file contains the raw angle & distance data. The other raw data file contains the control for the project.

Network with ALTA Reporting

The ALTA reporting project is located in \Carlson Projects\SurvNET Examples\ALTARpt. The name of the project is ALTARpt. This project illustrates how to perform ALTA tolerance testing on points within a network.

GPS Network with GPS Loop Closures

The GPS network project is located in \Carlson Projects\SurvNET Examples\GPSNetwork. The name of the project is GPSOnly. This project is a simple GPS network. In addition to the least squares computation and report, GPS loop closures were generated for various GPS loops for this project.

Level Network

The differential leveling project is located in \Carlson Projects\SurvNET Examples\LevelNetwork. The name of the project is network1. This project is a simple differential leveling network.

Basic 3D Project

The basic 3D adjustment project is located in \Carlson Projects\SurvNET Examples\3DNetwork. The name of the project is pg08. This project is a simple four point example network. Notice in the raw data that all set up records have an HI and all FS readings have valid rod heights. Also note that there are valid vertical angles for every slope distance. Since the 3D model is a true one process 3-dimensional adjustment, you must enter all valid slope distances and vertical angles. Be aware that you cannot just enter a horizontal distance and a vertical angle of 90 from reduced field notes when adjusting using the 3D model.

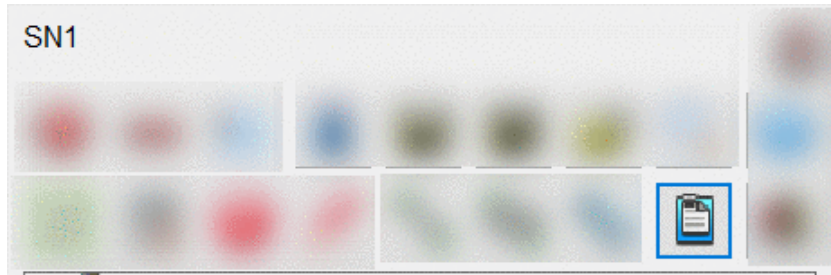
3D Project Combining Total Station and GPS Vectors

The total station raw data combined with GPS vectors example is located in \Carlson Projects\SurvNET Examples\GPSandTtlSta. The name of the project is GPSandTS. This project illustrates a 3D model adjustment that combines both GPS vectors and data from a total station. Since there is GPS data the 3D model must be used. Notice that the GPS vectors are in meters but the total station data is in US feet and the output coordinates are in US feet. Always make sure your units are correct for each data type, especially when using the 3D model.

Resection

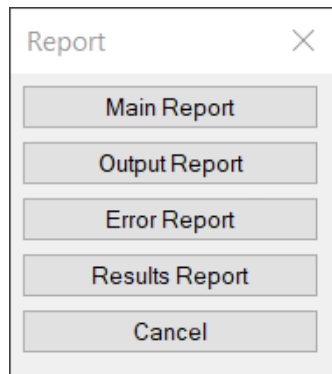
The total station raw data combined with GPS vectors example is located in \Carlson Projects\SurvNET Examples\Resection. The name of the project is ResectRaw. This project illustrates an angle and distance resection. There is no real difference in a resection project than any other angle and distance network in terms of how the data is collected or how the project is set up.

Network Processing Reports

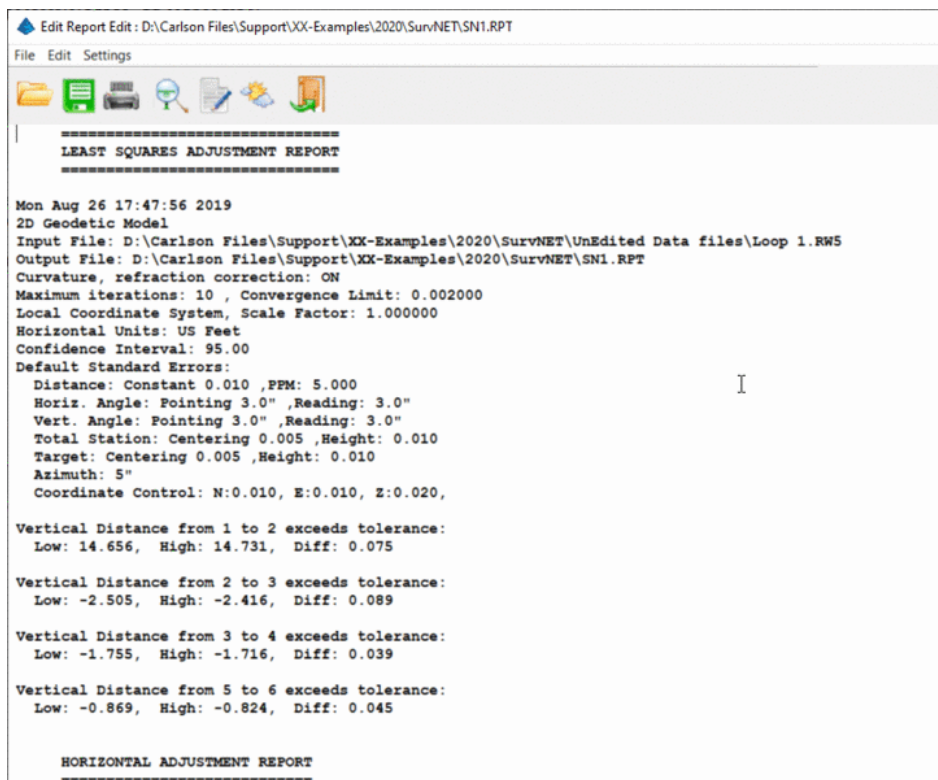


Report File: A report file consisting of the project name with an .RPT extension is generated after successfully processing the raw data. The report file will be shown in a text window so you can analyze the data.

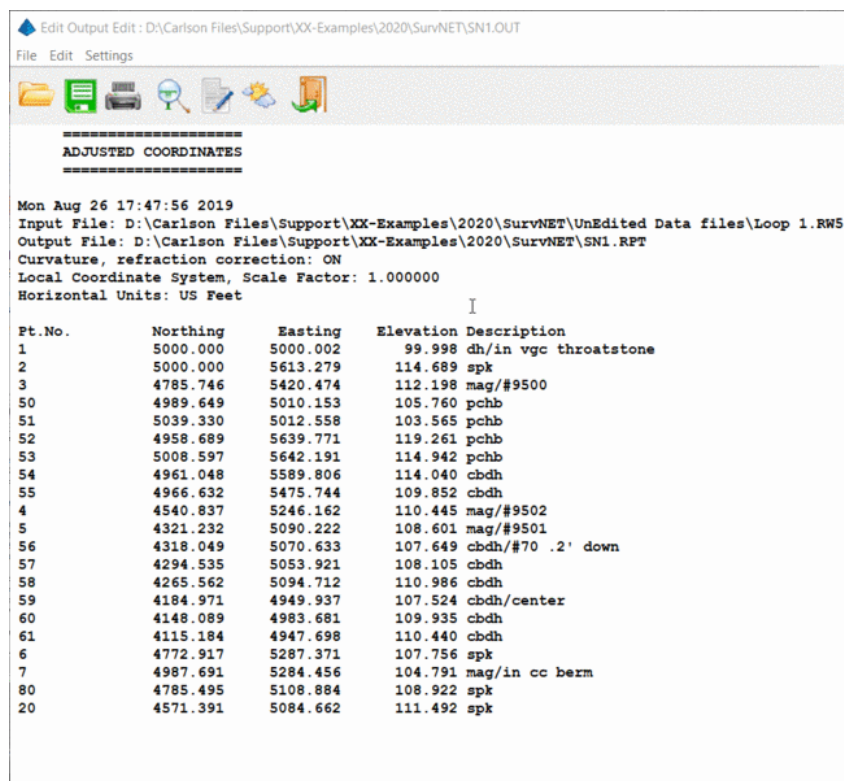
By clicking on the **Report** button, you will see several options for available reports.



This report contains essentially all of the data analysis of the network, errors, horizontal and vertical adjustments, data statistics, the optional ALTA report and the results of the chi square test are all included.



This report contains final adjusted coordinates from the network based on the options made in the Project Settings.



This report contains essential information about measurements that exceed the tolerances of standard errors set in

the Project Settings. It also contains information such as Point Aliases and traverse closure, if selected.

```

Edit Report Edit : D:\Carlson Files\Documents\Instructional Material\Help Documents 2021\Surv.ERR
File Edit Settings

LEAST SQUARES ADJUSTMENT ERROR REPORT

Sun Jul 19 11:30:46 2020
Input File: D:\Carlson Files\Documents\Instructional Material\Help Documents 2021\Loop 2.RMS
Input File: D:\Carlson Files\Documents\Instructional Material\Help Documents 2021\Loop 1.RMS
Output File: D:\Carlson Files\Documents\Instructional Material\Help Documents 2021\Surv.RPT

Point Aliases:
Pt. 6B = Pt. 6
Pt. 5B = Pt. 5B
Pt. 4B = Pt. 4
Pt. 5B = Pt. 5
Pt. 9 = Pt. 1
Pt. 9 = Pt. 2

Horizontal Angle spread exceeds tolerance:
IP: 7, BS: 6, FS: 1
Low: 093°15'17.0" , High: 093°15'28.0" , Diff: 000°00'11.0"

Horizontal Angle spread exceeds tolerance:
IP: 1, BS: 7, FS: 2
Low: 357°31'13.0" , High: 357°31'32.0" , Diff: 000°00'19.0"

Vertical Distance from 1 to 2 exceeds tolerance:
Low: 14.656, High: 14.731, Diff: 0.075

Vertical Distance from 2 to 3 exceeds tolerance:
Low: -2.505, High: -2.416, Diff: 0.089

Vertical Distance from 3 to 4 exceeds tolerance:
Low: -1.755, High: -1.716, Diff: 0.039

Vertical Distance from 5 to 6 exceeds tolerance:
Low: -0.869, High: -0.824, Diff: 0.045

Traverse Closures
=====
Traverse points:
1,2,3,4,5,6,7,1

Loop Traverse: Interior direction reference;
Do not compute angle closure.
Do not compute vertical closure.

BS IP FS Angle FS H. Dist.
1 2 3 311-59'02.0" 288.232
2 3 4 173-27'27.0" 300.605
3 4 5 179-56'18.0" 269.341
4 5 6 348-12'06.0" 492.840
5 6 7 155-38'38.5" 214.792
6 7 1 093-15'22.5" 284.718

Current Ending Coordinates: North: 5000.000 East: 5000.000
Ending Coordinates: North: 5000.024 East: 5000.024
Error, N: 0.024 E: 0.024 Total: 0.034 Brg: S 45°18'58.6"W
Distance Traversed: 1850.527 Closure: 1: 54438

```

This report displays just the essential pass/fail Chi square test along with the contributing data statistics.

```

Edit Report Edit : D:\Carlson Files\Support\XX-Examples\2020\SurvNET\SN1.RES
File Edit Settings

Solution converged in 2 iterations

Total Observations: 36
Total Unknowns: 22
Degrees of Freedom: 14

Observation Count    Sum Squares    Std. Error
                    of StdRes      of Unit Wt.
Coordinate            6            0.053      0.151
Angles:              15            3.149      0.735
Distances:          15            2.405      0.642 (Horizontal)
Total:               36            5.607      0.633

Reference Variance:0.401
Standard Error Unit Weight: (+/-)0.633
Failed the Chi-Square test at the 95.00 significance level
5.629 <= 5.607 <= 26.119

All confidence regions were computed using the following factors:
Variance factor: 0.6329
1-D Expansion Factor: 3.0332
2-D Expansion Factor: 2.7346
Expansion factors for 95.00 confidence regions taken from F-distribution table

```

2D-1D Local Coordinate System

The following explanations should be used in conjunction with the report at the end of the explanatory text. The report is shown in a tabbed window.

Project Settings

The first section of the report displays the project settings at the time the project was processed.

Tolerances

The second section of the report displays warning and error messages generated during the preprocessing of the raw data. The primary messages displayed will be warnings when multiple angles, horizontal distances, and vertical differences exceed the tolerance settings as set in the project settings. The low and high measurement and the difference are displayed. It is prudent to pay attention to any messages generated in this section of the report. Some warnings may be innocuous but it is prudent to check and understand all warning messages.

The next four sections list the reduced and averaged, but unadjusted measurements that make up the network. Multiple measurements of the same angle or distance are averaged to a single measurement. The standard error of multiple averaged measurements is less than the standard error of a single measurement. When multiple measurements are used, the standard error for the averaged measurement will be computed using the average of the mean formula.

The first of the four sections lists the control coordinates used in the network adjustment. These coordinates could have been read from the .CGR raw data file, or from the .CRD or .NEZ supplemental coordinate file. Notice that the standard errors for the control points are displayed.

The second of the four measurement sections shows the control reference azimuths and azimuth standard errors used in the adjustment. Azimuths are defined in the .CGR or RW5 files (Add->Reference Azimuth).

The third of the four measurement sections shows the distances and distance standard errors used in the adjustment. These distances are horizontal distances computed from all slope distance and vertical angles for that distance, including all foresight and backsight distances. The standard error settings used to calculate the final distance standard error include the distance standard error, the PPM standard error, the target centering standard error and the instrument centering standard errors. The techniques and formulas used to calculate the final distance standard error are found in section 6.12 of the textbook "Adjustment Computations, Statistics and Least Squares in Surveying and GIS", by Paul Wolf and Charles Ghilani (Wolf, Paul, and Charles Ghilani. *Adjustment Computations, Statistics and Least Squares in Surveying and GIS*. John Wiley & Sons, 1997).

The fourth of the four measurement sections shows the angles and angle standard errors used in the adjustment. These angles are the averaged angle value for all the multiple angles collected. The standard error settings used to calculate the final angle standard error include the pointing standard error, the reading standard error, the target centering standard error and the instrument centering standard errors. The techniques and formulas used to calculate the final angle standard error are found in section 6.2 of the textbook "Adjustment Computations, Statistics and Least Squares in Surveying and GIS", by Paul Wolf and Charles Ghilani (Wolf, Paul, and Charles Ghilani. *Adjustment Computations, Statistics and Least Squares in Surveying and GIS*. John Wiley & Sons, 1997).

If you have a Traverse Closure file selected, there will be a fifth section in the Unadjusted Observations section which shows the error of closure report.

If the adjustment of the network converges the next section displays a list of the final adjusted coordinates and the computed standard X, Y standard error. An interpretation of the meaning of the X, Y standard error, is that there is a 68% probability that the adjusted X, Y is within plus or minus the standard error of the X, Y of its true value.

Also shown are the Delta N, Delta E and Delta EL values of the non-fixed control points (how much the control points moved).

The next section displays the error ellipses for the adjusted coordinates. The error ellipse is a truer representation of the error of the point than the X, Y standard error. The error ellipses are calculated to the confidence interval as defined in the settings screen. In this report the error ellipse axis is larger than the X, Y standard errors since the error ellipses in this report are calculated at a 95% probability level as set in the Settings screens. The maximum error axis direction is along the axis of the semi-major axis. The direction of the minimum error axis direction is

along the semi-minor axis and is perpendicular to the semi-major axis. If a point is located from a variety of stations, you will most likely see that the error ellipse will approach a circle, which is the strongest geometric shape.

The next three sections list the adjusted horizontal distance, horizontal angle, and azimuth measurements. In addition to the adjusted measurement the, residual, the standard residual and the standard deviation of the adjusted measurement is displayed.

The **residual** is defined as the difference between the unadjusted measurement and the adjusted measurement. The residual is one of the most useful and intuitive measures displayed in the report. Large residuals in relation to the standards of the survey are indications of problems with the data.

The standard residual is the *a priori* standard error divided by the residual of a measurement. The *a priori* standard errors are the standard errors of the measurements as displayed in the unadjusted measurement section. A standard residual of 1 indicates that the adjustment applied to the measurement is consistent with the expected adjustment to the measurement. One or a few measurements having high standard residuals, in relation to the rest of the standard residuals, may be an indication of a blunder in the survey. When all standard residuals are consistently large there is likely an inconsistency in the a priori standard errors and the adjustments being made to the measurements. In other words the standard errors defined for the project are too small, in relation to the survey methods used.

The standard deviation of the measurement means that there is a 68% probability that the adjusted measurement is within plus or minus the standard deviation of the measurement's true value.

Additionally, the root mean square of each measurement type is displayed. The root mean square is defined as the square root of the average of the squares of a set of numbers. Loosely defined, it is as an average residual for that measurement type.

Statistics

The next section displays some statistical measures of the adjustment including the number of iterations needed for the solution to converge, the degrees of freedom of the network, the Error Factor for each type of measurement, the standard error of unit weight, the reference variance and the results of a Chi²; (Chi-square) test.

- **Degrees of Freedom** - The degree of freedom is an indication of how many redundant measurements are in the survey. Degree of freedom is defined as the number of measurements in excess of the number of measurements necessary to solve the network. The higher the DOF the more meaningful the statistics are.
- **Standard Error** - The possible measurement types are:
 - Coordinates (control points - in the 2D/1D model each point counts as 2 measurements (N & E); in the 3D model each point counts as 3 measurements (N, E, & EL))
 - Elevations (elevation control - from elevation records in the raw data file)
 - Azimuths (azimuth control - from reference azimuth records in the raw data file)
 - Angles (horizontal angles)
 - Distances (horizontal in 2D/1D model and mark-to-mark in 3D model)
 - Vertical Angles (mark-to-mark, 3D model only)
 - GPS Vectors (3D model only)

The total standard error of unit weight relates to the overall adjustment and not an individual measurement. A value of one indicates that the results of the adjustment are consistent with *a priori* standard errors. The total standard error of unit weight equals the sum of all the measurement standard errors of unit weight.

- **Reference Variance** - The reference variance is the standard error of unit weight squared.
- **Chi²; Test** - The Chi²; (Chi-square) test is a test of the "goodness" of fit of the adjustment. It is not an absolute test of the accuracy of the survey. The a priori standard errors which are defined in the project settings dialog box or with the SE record in the raw data (.RW5 or .CGR) file are used to determine the weights of the measurements. These standard errors can also be looked at as an estimate of how accurately the measurements were made. The chi-square test merely tests whether the results of the adjusted measurements are consistent with the a priori standard errors. **Notice that if you change the project standard errors and then reprocess the survey, the results of the chi-square test change... even though the final adjusted coordinates may change very little.**

If the project passes the Chi Square test, the expansion factor used to calculate the confidence regions (error ellipses at say 95% confidence) will be taken from the normal distribution table. If the project fails the Chi Square test, the expansion factors will be taken from the F-Distribution table:

All confidence regions were computed using the following factors:

Variance factor: 1.0000

1-D Expansion Factor: 1.9600

2-D Expansion Factor: 2.4477

Expansion factors for 95.00 confidence regions taken from normal distribution table

The next section displays the computed sideshots of the network. Sideshots are filtered out of the network adjustment as part of the preprocessing process if the Enable Sideshots for Relative Error Ellipses toggle is off. Least squares adjustment requires a lot of computer resources. Sideshots are filtered out to minimize the computer resources needed in a large network adjustment. The sideshots are computed from the final adjusted network points. The results of the side shot computations are the same whether they are reduced as part of the least squares adjustment or from the final adjusted coordinates.

The next part of the report displays the results of the vertical adjustment. In the 2D/1D model the horizontal and the vertical adjustments are separate least squares adjustment processes. As long as there are redundant vertical measurements the vertical component of the network will also be reduced and adjusted using least squares.

The first section displays the vertical benchmarks used in the vertical adjustment. Next, is listed the points that will be adjusted as part of the vertical adjustment. The following section displays the measurements used in the adjustment. The measurements consist of the vertical elevation difference between points in vertical adjustment. The lengths between these points are used to determine the weights in the vertical adjustment. Longer length lines are weighted less in the vertical adjustment than shorter length lines.

The next section displays some statistics about the vertical control: Number of unknown elevations, number of routes, number of fixed and non-fixed benchmarks, and degrees of freedom.

The next section displays the adjusted elevations and the computed standard deviations of the computed elevations. Following the adjusted elevation section is a section displaying the final adjusted elevation difference measurements and their residuals. Finally, the computed side shot elevations are displayed.

```
=====
LEAST SQUARES ADJUSTMENT REPORT
=====
```

Wed Oct 12 09:54:00

2D Geodetic Model.

Input Raw Files:

C:\Carlson Projects\cgstar.cgr

Output File: C:\Carlson Projects\cgstar.RPT

Traverse File: C:\Carlson Projects\cgstar.cls

Curvature, refraction correction: ON

Maximum iterations: 10 , Convergence Limit: 0.002000

Local Coordinate System, Scale Factor: 1.000000

Horizontal Units: US Feet

Confidence Interval: 95.00

Default Standard Errors:

Distance: Constant 0.010 ,PPM: 5.000

Horiz. Angle: Pointing 0.0'' ,Reading: 5.0''

Vert. Angle: Pointing 0.0'' ,Reading: 20.0''

Total Station: Centering 0.010 ,Height: 0.010

Target: Centering 0.010 ,Height: 0.010

Azimuth: 5''

Coordinate Control: N:0.010, E:0.010, Z:0.020,

Horizontal Distance from 2 to 3 exceeds tolerance:

Low: 324.153, High: 324.196, Diff: 0.042

Vertical Distance from 2 to 3 exceeds tolerance:

Low: 6.617, High: 8.362, Diff: 1.745

Vertical Distance from 3 to 4 exceeds tolerance:

Low: 11.463, High: 11.512, Diff: 0.050

Horizontal Distance from 12 to 3 exceeds tolerance:

Low: 144.641, High: 144.661, Diff: 0.020

HORIZONTAL ADJUSTMENT REPORT

=====

Unadjusted Observations

=====

Control Coordinates: 2 Observed Points, 0 Fixed Points, 0 Approx. Points

Sta.	N:	E:	StErr N:	StErr E:
1	658428.2600	2150182.7000	0.0200	0.0200
4	658863.5500	2149911.0300	0.0200	0.0200

Azimuths: 1 Observations

Occ. Sta.	FS Sta.	Bearing	StErr (Sec.)
1	2	N 45-00'00.0''E	05.0

Distances: 14 Observations

From Sta.	To Sta.	Dist.	StErr
1	5	290.450	0.015
1	2	292.213	0.015
2	6	52.388	0.016
2	3	324.186	0.015
3	4	275.603	0.015
3	20	134.663	0.018
20	21	116.073	0.018
21	22	50.115	0.017
4	5	309.647	0.015
5	10	129.985	0.016
10	11	126.010	0.016
10	15	10.000	0.017
11	12	129.426	0.016
12	3	144.651	0.016

Angles: 17 Observations

BS Sta.	Occ. Sta.	FS Sta.	Angle	StErr (Sec.)
5	1	2	109-19'13.0''	13.9
1	2	6	190-32'06.0''	51.9
1	2	3	096-03'52.0''	12.9
2	3	30	350-57'34.0''	07.1
2	3	4	124-03'53.0''	14.0
2	3	20	185-23'56.0''	23.7
3	20	21	180-15'26.0''	33.9
20	21	22	183-26'45.0''	61.6
3	4	5	093-02'11.5''	13.3
4	5	10	039-26'40.0''	19.7
5	10	11	241-56'29.0''	30.4
5	10	15	056-23'10.0''	249.9
10	11	12	114-56'27.0''	30.3
11	12	3	140-39'24.5''	29.8
12	3	2	325-54'30.0''	17.8
4	5	30	079-39'33.0''	07.1
4	5	1	117-30'42.5''	13.8

Traverse Closures

=====

Traverse points:

5,1-5,1

Loop Traverse; Interior direction reference;

Compute angle closure.

Compute vertical closure.

BS	IP	FS	Angle	FS H. Dist.	FS V. Dist.
5	1	2	109-19'13.0''	292.213	7.566
1	2	3	096-03'52.0''	324.186	6.984
2	3	4	124-03'53.0''	275.603	-11.491
3	4	5	093-02'11.5''	309.647	4.356
4	5	1	117-30'42.5''	290.450	-7.504

Closing Az: S 64-19'13.0''E

Computed Closing Az: S 64-19'21.0''E

Total angular error: 000-00'08.0''

Angular error per point: 000-00'01.6''

Correct Ending Coordinates, North: 658428.260 East: 2150182.700

Ending Coordinates, North: 658428.312 East: 2150182.664

Error, N: 0.052 E: -0.036 Total: 0.064 Brg: S 34-49'32.1''E

Distance Traversed: 1492.100 Closure: 1: 23369

Correct Ending Elevation: 569.850

Ending Elevation: 569.761

Elevation Error: -0.089

Closure After Angle Adjustment

5	1	2	109-19'14.6''	292.213	7.566
1	2	3	096-03'53.6''	324.186	6.984
2	3	4	124-03'54.6''	275.603	-11.491
3	4	5	093-02'13.1''	309.647	4.356
4	5	1	117-30'44.1''	290.450	-7.504

Closing Az: S 64-19'13.0''E

Computed Closing Az: S 64-19'13.0''E

Total angular error: 000-00'00.0''

Angular error per point: 000-00'00.0''

Correct Ending Coordinates, North: 658428.260 East: 2150182.700

Ending Coordinates, North: 658428.310 East: 2150182.654

Error, N: 0.050 E: -0.046 Total: 0.068 Brg: S 42-39'20.9''E

Distance Traversed: 1492.100 Closure: 1: 21990

Traverse points:

5,1-5

Loop Traverse; Interior direction reference;

Do not compute angle closure.

Compute vertical closure.

BS	IP	FS	Angle	FS H. Dist.	FS V. Dist.
5	1	2	109-19'13.0''	292.213	7.566
1	2	3	096-03'52.0''	324.186	6.984
2	3	4	124-03'53.0''	275.603	-11.491
3	4	5	093-02'11.5''	309.647	4.356

Correct Ending Coordinates, North: 658554.124 East: 2149920.937

Ending Coordinates, North: 658554.166 East: 2149920.896

Error, N: 0.042 E: -0.041 Total: 0.059 Brg: S 44-22'19.4''E

Distance Traversed: 1492.100 Closure: 1: 25238

Correct Ending Elevation: 577.354

Ending Elevation: 577.265

Elevation Error: -0.089

Adjusted Coordinates

=====

Adjusted Local Coordinates

Sta.	N:	E:	StErr N:	StErr E:	DN	DE
1	658428.2359	2150182.7169	0.0163	0.0164	-0.0241	0.0169
4	658863.5741	2149911.0131	0.0163	0.0164	0.0241	-0.0169
2	658634.8584	2150389.3399	0.0192	0.0178		
5	658554.0894	2149920.9457	0.0178	0.0177		
3	658887.0003	2150185.6059	0.0177	0.0189		
20	658999.2418	2150111.2028	0.0246	0.0268		
21	659096.2758	2150047.5057	0.0336	0.0401		
10	658657.0743	2150000.2697	0.0203	0.0194		
11	658636.1774	2150124.5368	0.0221	0.0217		
12	658742.8596	2150197.8311	0.0224	0.0191		

Adjusted Coordinates Error Ellipses, 95% CI

Sta.	Semi Major Axis	Semi Minor Axis	Max. Error Az.
1	0.0420	0.0379	N 47-22'19.9''E
4	0.0420	0.0379	N 47-22'19.9''E
2	0.0502	0.0399	N 35-00'21.0''E
5	0.0458	0.0408	S 44-05'12.1''E
3	0.0466	0.0430	S 70-27'40.9''E
20	0.0658	0.0600	S 80-49'45.3''E
21	0.1032	0.0758	N 62-59'30.6''E
10	0.0512	0.0459	N 32-30'51.3''E
11	0.0560	0.0512	N 39-41'54.3''E
12	0.0549	0.0467	S 01-01'01.0''E

Adjusted Observations

=====

Adjusted Distances

From Sta.	To Sta.	Distance	Residual	StdRes.	StdDev
-----------	---------	----------	----------	---------	--------

1	5	290.454	0.003	0.2	0.013
1	2	292.209	-0.005	0.3	0.013
2	3	324.165	-0.021	1.4	0.013
3	4	275.590	-0.013	0.9	0.013
3	20	134.663	0.000	0.0	0.018
20	21	116.073	0.000	0.0	0.018
4	5	309.644	-0.003	0.2	0.012
5	10	129.993	0.008	0.5	0.014
10	11	126.012	0.002	0.1	0.014
11	12	129.434	0.008	0.5	0.014
12	3	144.658	0.007	0.5	0.015
Root Mean Square (RMS)			0.009		

Adjusted Angles

BS Sta.	Occ. Sta.	FS Sta.	Angle	Residual	StdRes	StdDev (Sec.)
5	1	2	109-19'22.5''	09.5	0.7	09.1
1	2	3	096-03'40.6''	-11.4	0.9	08.8
2	3	4	124-03'44.8''	-08.2	0.6	10.5
2	3	20	185-23'56.0''	-00.0	0.0	23.7
3	20	21	180-15'26.0''	-00.0	0.0	33.9
3	4	5	093-02'16.8''	05.3	0.4	09.5
4	5	10	039-26'36.3''	-03.7	0.2	15.8
5	10	11	241-56'25.5''	-03.5	0.1	24.0
10	11	12	114-56'41.1''	14.1	0.5	24.8
11	12	3	140-39'42.2''	17.7	0.6	22.8
12	3	2	325-54'33.2''	03.2	0.2	14.3
4	5	1	117-30'55.4''	12.9	0.9	10.3
Root Mean Square (RMS)				09.3		

Adjusted Azimuths

Occ. Sta.	FS Sta.	Bearing	Residual	StdRes	StdDev (Sec.)
1	2	N 45-00'00.2''E	00.2	0.0	04.7
Root Mean Square (RMS)			00.2		

Statistics

=====

Solution converged in 2 iterations

Total Observations:28

Total Unknowns:20

Degrees of Freedom:8

Observation	Count	Sum Squares of StdRes	Std. Error of Unit Wt.
Coordinate	4	4.335	1.948
Azimuths:	1	0.002	0.091
Angles:	12	3.291	0.980
Distances:	11	3.567	1.065 (Horizontal)
Total:	28	11.194	1.183

Reference Variance:1.399

Standard Error Unit Weight: (+/-)1.183

Passed the Chi-Square test at the 95.00 significance level

2.180 <= 11.194 <= 17.535

All confidence regions were computed using the following factors:

Variance factor: 1.0000

1-D Expansion Factor: 1.9600

2-D Expansion Factor: 2.4477

Expansion factors for 95.00 confidence regions taken from normal distribution table

Sideshots

=====

From	To	Bearing	Dist.	N	E	StDev. N	StDev. E
2	6	N 55-32'06.2''E	52.388	658664.5046	2150432.5320	0.0239	0.0233
21	22	N 29-50'12.2''W	50.115	659139.7480	2150022.5719	0.0376	0.0430
10	15	N 86-00'31.2''W	10.000	658657.7704	2149990.2940	0.0237	0.0260
From	Bearing	From	Bearing	To	N	E	StDev. N
5	N 77-49'15.4''E	3	S 47-58'45.2''E	30	658664.5029	2150432.5341	0.0398

LEAST SQUARES VERTICAL ADJUSTMENT REPORT

Wed Oct 12 09:54:00

2D Geodetic Model.

Input Raw Files:

C:\Carlson Projects\cgstar.cgr

Output File: C:\Carlson Projects\cgstar.RPT
 Traverse File: C:\Carlson Projects\cgstar.cls
 Curvature, refraction correction: ON

VERTICAL BENCHMARKS

Station	Elevation	Std. Error
1	569.8500	0.040
4	572.9500	0.040

POINTS TO BE ADJUSTED

Station
 2,5,3,10,11,12,30

MEASUREMENT SUMMARY

From	To	Elev. Diff. (unadjusted)	StdErr
1	5	7.5037	0.0200
1	2	7.5659	0.0201
2	3	6.9843	0.0200
3	4	-11.4907	0.0196
4	5	4.3557	0.0206
5	10	2.2639	0.0168
10	11	1.0931	0.0166
11	12	0.3828	0.0167
12	3	3.3590	0.0174
3	30	-7.3186	0.0354
5	30	-0.0334	0.0527

STATISTICAL SUMMARY

Total Unknown Elevations:7
 Total Elev. Routes:11
 Total Fixed BM's:0
 Total non-fixed BM's:2
 Degrees of freedom:4

ADJUSTED ELEVATIONS

Station	Adjusted Elev	Standard Dev.	Error Ellipse at 95% CI
1	569.8432	0.05007	0.09814
4	572.9568	0.05007	0.09814
2	577.4359	0.05819	0.11405
5	577.3168	0.05389	0.10562
3	584.4468	0.05591	0.10958
10	579.5885	0.06295	0.12339
11	580.6892	0.06601	0.12937
12	581.0797	0.06400	0.12544
30	577.1905	0.06953	0.13628

ADJUSTED MEASUREMENT SUMMARY

From	To	Elev. Diff. (adjusted)	Residuals	Std. Dev.
1	5	7.4736	-0.0301	0.037
1	2	7.5926	0.0268	0.038
2	3	7.0109	0.0267	0.038
3	4	-11.4900	0.0007	0.036
4	5	4.3600	0.0043	0.036
5	10	2.2717	0.0078	0.037
10	11	1.1008	0.0077	0.037
11	12	0.3905	0.0077	0.037
12	3	3.3670	0.0081	0.038
3	30	-7.2563	0.0624	0.049
5	30	-0.1262	-0.0928	0.052

Vertical Sideshots

Station	Elevation
6	577.135
20	571.777
21	581.262
22	580.151
15	579.588

2D-1D State Plane Coordinate System

Note: Highlighted explanatory text is found within the report text.

=====
LEAST SQUARES ADJUSTMENT REPORT
=====

Wed Oct 12 10:15:00
2D Geodetic Model.
Input Raw Files:
 C:\Carlson Projects\cgstar.cgr
Output File: C:\Carlson Projects\cgstar.RPT
Traverse File: C:\Carlson Projects\cgstar.cls
Curvature, refraction correction: ON
Maximum iterations: 10 , Convergence Limit: 0.002000
1983 State Plane Coordinates, zone:Georgia West - 1002
Elevation factor computed from raw data elevations.
Elevation Units: US Feet
Horizontal Units: US Feet
Confidence Interval: 95.00
Project Geoid Height: 0.000
Default Standard Errors:
 Distance: Constant 0.010 ,PPM: 5.000
 Horiz. Angle: Pointing 0.0'' ,Reading: 5.0''
 Vert. Angle: Pointing 0.0'' ,Reading: 20.0''
 Total Station: Centering 0.010 ,Height: 0.010
 Target: Centering 0.010 ,Height: 0.010
 Azimuth: 5''
 Coordinate Control: N:0.010, E:0.010, Z:0.020,
Horizontal Distance from 2 to 3 exceeds tolerance:
 Low: 324.153, High: 324.196, Diff: 0.042
Vertical Distance from 2 to 3 exceeds tolerance:
 Low: 6.617, High: 8.362, Diff: 1.745
Vertical Distance from 3 to 4 exceeds tolerance:
 Low: 11.463, High: 11.512, Diff: 0.050
Horizontal Distance from 12 to 3 exceeds tolerance:
 Low: 144.641, High: 144.661, Diff: 0.020

HORIZONTAL ADJUSTMENT REPORT
=====

Unadjusted Observations
=====

The control coordinates and azimuths are shown:

Control Coordinates:	2 Observed Points,	0 Fixed Points,	0 Approx. Points
Sta.	N:	E:	StErr N: StErr E:
1	658428.2600	2150182.7000	0.0200 0.0200
4	658863.5500	2149911.0300	0.0200 0.0200

Grid Azimuths:	1 Observations			
Occ. Sta.	FS	Sta.	Bearing	StErr (Sec.)
1	2	N	45-00'00.0''E	05.0

The first distance listing in the Unadjusted Observation section of the report shows the unadjusted horizontal ground distances

Distances:	14 Observations		
From Sta.	To Sta.	Ground Dist.	StErr
1	5	290.450	0.015
1	2	292.213	0.015
2	6	52.388	0.016
2	3	324.186	0.015
3	4	275.603	0.015
3	20	134.663	0.018
20	21	116.073	0.018
21	22	50.115	0.017
4	5	309.647	0.015

5	10	129.985	0.016
10	11	126.010	0.016
10	15	10.000	0.017
11	12	129.426	0.016
12	3	144.651	0.016

Angles: 17 Observations

BS Sta.	Occ. Sta.	FS Sta.	Angle	StErr (Sec.)
5	1	2	109-19'13.0''	13.9
1	2	6	190-32'06.0''	51.9
1	2	3	096-03'52.0''	12.9
2	3	30	350-57'34.0''	07.1
2	3	4	124-03'53.0''	14.0
2	3	20	185-23'56.0''	23.7
3	20	21	180-15'26.0''	33.9
20	21	22	183-26'45.0''	61.6
3	4	5	093-02'11.5''	13.3
4	5	10	039-26'40.0''	19.7
5	10	11	241-56'29.0''	30.4
5	10	15	056-23'10.0''	249.9
10	11	12	114-56'27.0''	30.3
11	12	3	140-39'24.5''	29.8
12	3	2	325-54'30.0''	17.8
4	5	30	079-39'33.0''	07.1
4	5	1	117-30'42.5''	13.8

Closure Report if a project closure file has been created and selected:

Traverse Closures

=====

Traverse points:

5,1-5,1

Loop Traverse; Interior direction reference;

Compute angle closure.

Compute vertical closure.

BS	IP	FS	Angle	FS H. Dist.	FS V. Dist.
5	1	2	109-19'13.0''	292.183	7.566
1	2	3	096-03'52.0''	324.153	6.984
2	3	4	124-03'53.0''	275.575	-11.491
3	4	5	093-02'11.5''	309.615	4.356
4	5	1	117-30'42.5''	290.420	-7.504

Closing Az: S 64-19'13.0''E

Computed Closing Az: S 64-19'21.0''E

Total angular error: 000-00'08.0''

Angular error per point: 000-00'01.6''

Correct Ending Coordinates, North: 658428.260 East: 2150182.700

Ending Coordinates, North: 658428.312 East: 2150182.664

Error, N: 0.052 E: -0.036 Total: 0.064 Brg: S 34-48'25.3''E

Distance Traversed: 1491.946 Closure: 1: 23446

Correct Ending Elevation: 569.850

Ending Elevation: 569.761

Elevation Error: -0.089

Closure After Angle Adjustment

5	1	2	109-19'14.6''	292.183	7.566
1	2	3	096-03'53.6''	324.153	6.984
2	3	4	124-03'54.6''	275.575	-11.491
3	4	5	093-02'13.1''	309.615	4.356
4	5	1	117-30'44.1''	290.420	-7.504

Closing Az: S 64-19'13.0''E

Computed Closing Az: S 64-19'13.0''E

Total angular error: 000-00'00.0''

Angular error per point: 000-00'00.0''

Correct Ending Coordinates, North: 658428.260 East: 2150182.700

Ending Coordinates, North: 658428.310 East: 2150182.654

Error, N: 0.050 E: -0.046 Total: 0.068 Brg: S 42-39'45.7''E

Distance Traversed: 1491.946 Closure: 1: 22059

The reduced, unadjusted grid distances are shown. The grid factor, the elevation factor, and the combined factor used to reduce the ground distance to a grid distance are included in the listing:

Grid Distances: 14 Observations

From Sta.	To Sta.	Grid Dist.	Grid Factor	Z Factor	Combined Factor
1	5	290.420	0.99992459	0.99997256	0.99989716
1	2	292.183	0.99992452	0.99997256	0.99989708
2	6	52.382	0.99992447	0.99997239	0.99989686
2	3	324.153	0.99992451	0.99997221	0.99989673
3	4	275.575	0.99992459	0.99997232	0.99989692
3	20	134.649	0.99992456	0.99997235	0.99989691
20	21	116.061	0.99992458	0.99997243	0.99989701
21	22	50.110	0.99992460	0.99997223	0.99989683
4	5	309.615	0.99992464	0.99997249	0.99989713
5	10	129.972	0.99992462	0.99997233	0.99989696
10	11	125.997	0.99992459	0.99997225	0.99989684
10	15	9.999	0.99992461	0.99997228	0.99989689
11	12	129.413	0.99992456	0.99997221	0.99989677
12	3	144.636	0.99992455	0.99997213	0.99989667

The reduced, unadjusted horizontal angles with the t-T correction applied are shown. The t-T correction is generally a small correction. For most surveys of limited size the correction is negligible. The t-T correction is displayed in seconds.

Grid Horizontal Angles: 17 Observations

BS Sta.	Occ. Sta.	FS Sta.	Angle	StErr (Sec.)	t-T
5	1	2	109-19'13.0''	13.9	0.0
1	2	6	190-32'06.0''	51.9	0.0
1	2	3	096-03'52.0''	12.9	0.0
2	3	30	350-57'34.0''	7.1	0.0
2	3	4	124-03'53.0''	14.0	0.0
2	3	20	185-23'56.0''	23.7	0.0
3	20	21	180-15'26.0''	33.9	0.0
20	21	22	183-26'45.0''	61.6	0.0
3	4	5	093-02'11.5''	13.3	-0.0
4	5	10	039-26'40.0''	19.7	-0.0
5	10	11	241-56'29.0''	30.4	0.0
5	10	15	056-23'10.0''	249.9	0.0
10	11	12	114-56'27.0''	30.3	0.0
11	12	3	140-39'24.5''	29.8	0.0
12	3	2	325-54'30.0''	17.8	-0.0
4	5	30	079-39'33.0''	7.1	-0.0
4	5	1	117-30'42.5''	13.8	-0.0

Adjusted Coordinates
=====

The adjusted GRID coordinates are show along with the Delta N and Delta E values of the non-fixed control points:

Adjusted Grid Coordinates

Sta.	N:	E:	StErr N:	StErr E:	DN	DE
1	658428.2529	2150182.7047	0.0163	0.0164	-0.0071	0.0047
4	658863.5571	2149911.0253	0.0163	0.0164	0.0071	-0.0047
2	658634.8548	2150389.3067	0.0192	0.0178		
5	658554.0988	2149920.9564	0.0178	0.0177		
3	658886.9763	2150185.5926	0.0177	0.0189		
20	658999.2069	2150111.1980	0.0246	0.0268		
21	659096.2313	2150047.5082	0.0336	0.0401		
10	658657.0735	2150000.2729	0.0203	0.0194		
11	658636.1786	2150124.5275	0.0221	0.0217		
12	658742.8499	2150197.8152	0.0224	0.0191		

In the Adjusted Coordinates section of the report there is a section displaying the latitude and longitude of the final adjusted points. Additionally the convergence angle, the grid factor, the elevation factor, and the

combined factor are displayed for each point. Also calculated is the Average Combined Scale Factor for the project which is useful if you want to scale the grid coordinates to ground.

Adjusted Geographic Coordinates

Sta.	Latitude	Longitude	Conv. Ang.	Grid Factor	Z Factor	Combined Factor
1	31-48'34.11031''N	84-38'16.70951''W	-000-14'54.3''	0.99992455	0.99997274	0.99989729
4	31-48'38.40662''N	84-38'19.87996''W	-000-14'56.0''	0.99992464	0.99997259	0.99989724
2	31-48'36.16380''N	84-38'14.32554''W	-000-14'53.1''	0.99992448	0.99997238	0.99989686
5	31-48'35.34450''N	84-38'19.74929''W	-000-14'56.0''	0.99992464	0.99997238	0.99989702
3	31-48'38.65018''N	84-38'16.69910''W	-000-14'54.4''	0.99992455	0.99997205	0.99989660
20	31-48'39.75767''N	84-38'17.56692''W	-000-14'54.8''	0.99992457	0.99997265	0.99989723
21	31-48'40.71513''N	84-38'18.30993''W	-000-14'55.2''	0.99992459	0.99997220	0.99989680
10	31-48'36.36700''N	84-38'18.83526''W	-000-14'55.5''	0.99992461	0.99997228	0.99989689
11	31-48'36.16555''N	84-38'17.39419''W	-000-14'54.7''	0.99992457	0.99997222	0.99989679
12	31-48'37.22436''N	84-38'16.55020''W	-000-14'54.3''	0.99992454	0.99997221	0.99989675

Average Combined Scale Factor: 0.99989695

Error ellipses shown at 95% confidence interval:

Adjusted Coordinates Error Ellipses, 95% CI

Sta.	Semi Major Axis	Semi Minor Axis	Max. Error Az.
1	0.0420	0.0379	N 47-22'21.7''E
4	0.0420	0.0379	N 47-22'21.7''E
2	0.0502	0.0399	N 35-00'26.0''E
5	0.0458	0.0408	S 44-05'10.1''E
3	0.0466	0.0430	S 70-27'45.8''E
20	0.0658	0.0600	S 80-48'51.3''E
21	0.1031	0.0758	N 62-59'38.9''E
10	0.0512	0.0459	N 32-30'53.2''E
11	0.0560	0.0512	N 39-41'57.3''E
12	0.0549	0.0467	S 01-00'56.8''E

The adjusted distances, angles and reference azimuths are shown:

Adjusted Observations

Adjusted Distances

From Sta.	To Sta.	Distance	Residual	StdRes.	StdDev
1	5	290.430	0.009	0.6	0.013
1	2	292.179	-0.004	0.3	0.013
2	3	324.137	-0.016	1.0	0.013
3	4	275.564	-0.011	0.7	0.013
3	20	134.649	-0.000	0.0	0.018
20	21	116.061	0.000	0.0	0.018
4	5	309.618	0.003	0.2	0.012
5	10	129.980	0.009	0.5	0.014
10	11	125.999	0.002	0.1	0.014
11	12	129.421	0.009	0.5	0.014
12	3	144.644	0.008	0.5	0.015

Root Mean Square (RMS) 0.008

Adjusted Angles

BS Sta.	Occ. Sta.	FS Sta.	Angle	Residual	StdRes	StdDev (Sec.)
5	1	2	109-19'20.1''	07.1	0.5	09.1
1	2	3	096-03'42.4''	-09.6	0.7	08.8
2	3	4	124-03'46.7''	-06.3	0.4	10.5
2	3	20	185-23'56.0''	00.0	0.0	23.7
3	20	21	180-15'26.0''	-00.0	0.0	33.9
3	4	5	093-02'13.7''	02.2	0.2	09.5
4	5	10	039-26'36.3''	-03.7	0.2	15.8
5	10	11	241-56'25.3''	-03.7	0.1	24.0
10	11	12	114-56'42.1''	15.1	0.5	24.8
11	12	3	140-39'42.9''	18.4	0.6	22.8
12	3	2	325-54'33.0''	03.0	0.2	14.3
4	5	1	117-30'57.1''	14.6	1.1	10.3

Root Mean Square (RMS) 09.1

Adjusted Azimuths

Occ. Sta.	FS Sta.	Bearing	Residual	StdRes	StdDev (Sec.)
1	2	N 45-00'00.0''E	00.0	0.0	04.7
Root Mean Square (RMS)			00.0		

Project Statistics are shown:

Statistics

=====

Solution converged in 2 iterations

Total Observations:28

Total Unknowns:20

Degrees of Freedom:8

Observation	Count	Sum Squares of StdRes	Std. Error of Unit Wt.
Coordinate	4	0.364	0.564
Azimuths:	1	0.000	0.014
Angles:	12	2.878	0.916
Distances:	11	2.923	0.964 (Horizontal)
Total:	28	6.165	0.878

Reference Variance:0.771

Standard Error Unit Weight: (+/-)0.878

Passed the Chi-Square test at the 95.00 significance level

2.180 <= 6.165 <= 17.535

All confidence regions were computed using the following factors:

Variance factor: 1.0000

1-D Expansion Factor: 1.9600

2-D Expansion Factor: 2.4477

Expansion factors for 95.00 confidence regions taken from normal distribution table

Sideshots are shown:

Sideshots

=====

From	To	Bearing	Dist.	N	E	StDev. N	StDev. E
2	6	N 55-32'06.0''E	52.382	658664.4980	2150432.4944	0.0239	0.0233
21	22	N 29-50'10.6''W	50.110	659139.6992	2150022.5773	0.0376	0.0430
10	15	N 86-00'30.8''W	9.999	658657.7695	2149990.2981	0.0237	0.0260

Elevation adjustment report is shown:

LEAST SQUARES VERTICAL ADJUSTMENT REPORT

Wed Oct 12 10:15:00

2D Geodetic Model.

Input Raw Files:

C:\Carlson Projects\cgstar.cgr

Output File: C:\Carlson Projects\cgstar.RPT

Traverse File: C:\Carlson Projects\cgstar.cls

Curvature, refraction correction: ON

VERTICAL BENCHMARKS

Station	Elevation	Std. Error
1	569.8500	0.040
4	572.9500	0.040

POINTS TO BE ADJUSTED

Station
2,5,3,10,11,12,30

MEASUREMENT SUMMARY

From	To	Elev. Diff. (unadjusted)	StdErr
1	5	7.5037	0.0200
1	2	7.5659	0.0201
2	3	6.9843	0.0200
3	4	-11.4907	0.0196
4	5	4.3557	0.0206
5	10	2.2639	0.0168

10	11	1.0931	0.0166
11	12	0.3828	0.0167
12	3	3.3590	0.0174
3	30	-7.3173	0.0354
5	30	-0.0329	0.0527

STATISTICAL SUMMARY

Total Unknown Elevations:7
 Total Elev. Routes:11
 Total Fixed BM's:0
 Total non-fixed BM's:2
 Degrees of freedom:4

ADJUSTED ELEVATIONS

Station	Adjusted Elev	Standard Dev.	Error Ellipse at 95% CI
1	569.8432	0.04991	0.09782
4	572.9568	0.04991	0.09782
2	577.4358	0.05800	0.11368
5	577.3168	0.05372	0.10528
3	584.4467	0.05573	0.10923
10	579.5885	0.06275	0.12299
11	580.6892	0.06579	0.12895
12	581.0797	0.06379	0.12503
30	577.1915	0.06931	0.13584

ADJUSTED MEASUREMENT SUMMARY

From	To	Elev. Diff. (adjusted)	Residuals	Std. Dev.
1	5	7.4736	-0.0301	0.037
1	2	7.5926	0.0267	0.038
2	3	7.0109	0.0266	0.038
3	4	-11.4899	0.0008	0.036
4	5	4.3600	0.0043	0.036
5	10	2.2717	0.0077	0.037
10	11	1.1007	0.0077	0.037
11	12	0.3905	0.0077	0.037
12	3	3.3670	0.0080	0.037
3	30	-7.2552	0.0621	0.049
5	30	-0.1253	-0.0924	0.052

Vertical Sideshots

Station	Elevation
6	577.135
20	571.777
21	581.262
22	580.151
15	579.588

GPS Network

Note: The following section shows the report generated by the least squares adjustment of the GPS network. The 3D adjustment model **MUST** be used when processing GPS vectors. **Explanations of the report are included in the report section and are in bold text.**

```
=====
LEAST SQUARES ADJUSTMENT REPORT
=====
```

```
Wed Oct 12 10:59:00
3D Geodetic Model.
Input Raw Files:
  C:\Carlson Projects\gpsonly\control.cgr
  C:\Carlson Projects\gpsonly\chapt16.gps
Output File: C:\Carlson Projects\gpsonly\gpsOnly.RPT
Traverse File: C:\Carlson Projects\gpsonly\gpsLoops.cls
Curvature, refraction correction: OFF
Maximum iterations: 10 , Convergence Limit: 0.002000
1983 State Plane Coordinates, zone:Wisconsin South - 4803
Horizontal Units: Meters
Confidence Interval: 95.00
```

Geoid Height computed from geoid grid files using the GEOID99 model.

Default Standard Errors:

Distance: Constant 0.010 ,PPM: 5.000
 Horiz. Angle: Pointing 10.0'' ,Reading: 3.0''
 Vert. Angle: Pointing 3.0'' ,Reading: 3.0''
 Total Station: Centering 0.005 ,Height: 0.010
 Target: Centering 0.010 ,Height: 0.010
 Azimuth: 5''
 Coordinate Control: N:0.001, E:0.001, Z:0.030,
 GPS: Centering:0.003, Vector Err. Factor:7.0
 3-DIMENSIONAL ADJUSTMENT REPORT
 =====

The following section shows the unadjusted measurements that make up the network. The control coordinates are displayed first followed by the GPS vectors. The control coordinates are displayed as latitude/longitude, SPC Grid XYZ, and geocentric XYZ. If geoid modeling is set both ellipsoid and orthometric elevations are displayed, ellipsoid elevation in the latitude/longitude section and orthometric elevation in the SPC section. The GPS vector section shows the unadjusted delta XYZ, variances and covariances of the vectors.

Unadjusted Observations

=====

Control Coordinates: 0 Observed Points, 2 Fixed Points, 0 Approx. Points

Sta.	Latitude	Longitude	Z (Ellip.)	StErr N:	StErr E:	StErr Z:NE
A	43-15'46.28901''N	89-59'42.16399''W	1348.23	FIXED	FIXED	FIXED
B	43-23'46.36261''N	89-54'00.75701''W	1200.63	FIXED	FIXED	FIXED

Grid XYZ

Sta.	N:	E:	Z (Geoid):	StErr N:	StErr E:	StErr Z:
A	140291.2060	600402.2380	1382.62	FIXED	FIXED	FIXED
B	155110.5390	608083.9250	1235.46	FIXED	FIXED	FIXED

Geocentric XYZ

Sta.	X:	Y:	Z:	StErr X(m):	StErr Y(m):	StErr Z(m):
A	402.3489	-4652970.2600	4349737.2110	FIXED	FIXED	FIXED
B	8085.9876	-4642687.5392	4360415.1538	FIXED	FIXED	FIXED

The GPS vectors are shown (Delta X, Delta Y and Delta Z values). Also shown are the bearing, horizontal distance and vertical difference of each vector:

GPS Vectors: 13 Observations (Meters)

Multiplication factor applied to Variance and Covariance values: 128 (2 to 7 power)

From Sta.	Delta X	Variance Delta X	Covariance XY	Bearing
To Sta.	Delta Y	Variance Delta Y	Covariance XZ	H.Distance
	Delta Z	Variance Delta Z	Covariance YZ	V.Distance
A	11644.223	0.126	-0.001226	N 67-00'00.4''E
C	3601.217	0.12	0.001219	12650.1512
	3399.255	0.1258	-0.001219	-292.0672
A	-5321.716	0.02764	-0.0002688	N 47-56'12.9''W
E	3634.075	0.02458	0.0002765	7167.7644
	3173.665	0.02568	-0.0002688	-471.6711
B	3960.544	0.02952	-0.0002854	S 21-46'39.9''E
C	-6681.247	0.03261	0.000265	10643.7341
	-7279.015	0.02884	-0.0002854	-141.2376
B	-11167.608	0.03458	-0.000352	S 85-18'14.1''W
D	-394.520	0.03485	0.0003648	11205.8960
	-907.959	0.03419	-0.0003482	-351.2707
D	15128.165	0.01872	-0.000183	S 59-25'16.9''E
C	-6286.705	0.02068	0.0001715	17576.6958
	-6371.058	0.01676	-0.0001843	184.8828
D	-1837.746	0.01577	-0.0001523	S 11-24'12.6''W
E	-6253.853	0.01636	0.0001562	9273.8256
	-6596.670	0.01644	-0.0001549	14.2116
F	-1116.452	0.009586	-0.0001011	S 10-01'41.1''W
A	-4596.161	0.008457	0.0001126	6420.1982
	-4355.906	0.009766	-0.0001037	355.1552
F	10527.785	0.03288	-0.000288	S 82-31'36.9''E

C	-994.938	0.0277	0.0003072	10617.6451
	-956.625	0.0307	-0.0002906	70.0452
F	-6438.136	0.0121	-0.0001178	S 76-43'39.1''W
E	-962.069	0.01277	0.0001331	6615.1455
	-1182.230	0.01132	-0.0001139	-112.6820
F	-4600.379	0.01196	-0.0001267	N 31-16'31.7''W
D	5291.779	0.01266	0.0001152	8857.9833
	5414.431	0.01543	-0.0001267	-136.3798
F	6567.231	0.008521	-8.32E-005	N 37-41'59.1''E
B	5686.293	0.009573	8.832E-005	10742.1697
	6322.392	0.007759	-8.192E-005	202.1754
B	-6567.231	0.007073	-8.064E-005	S 37-45'19.5''W
F	-5686.303	0.009582	7.808E-005	10741.8133
	-6322.381	0.008503	-8.064E-005	-220.2601
A	1116.458	0.00849	-0.0001024	N 10-01'07.3''E
F	4596.155	0.0104	0.0001152	6419.8396
	4355.914	0.01202	-0.000105	-361.6188

The optional Traverse Closure shows the GPS loop closures for the GPS loops defined in the closure file.

Traverse Closures

=====

GPS Loop Points:

A,E,F,A

GPS Loop Closure;

Misclosure, X: -0.0323 Y: -0.0162 Z: -0.0105

Closure error: 0.0376 Perimeter: 20229.3858

Precision: 1: 537594

GPS Loop Points:

C,F,D,B,C

GPS Loop Closure;

Misclosure, X: -0.0121 Y: -0.0101 Z: 0.0002

Closure error: 0.0158 Perimeter: 41332.9807

Precision: 1: 2622216

GPS Loop Points:

F,D,B,F

GPS Loop Closure;

Misclosure, X: -0.0022 Y: -0.0044 Z: 0.0097

Closure error: 0.0109 Perimeter: 30814.5047

Precision: 1: 2833226

Following are the final adjusted coordinates. Included in the report are point grid factor, elev. factor and the combined factor. Following the adjusted coordinates are the error ellipses, followed by the adjusted measurements section.

Adjusted Geographic Coordinates

Sta.	Latitude	Longitude	Z (Ellip.)	Conv. Ang.	Grid Factor	Z Factor
C	43-18'26.10221''N	89-51'05.56799''W	1068.43	000-06'07.2''	0.99993389	0.99983235
E	43-18'21.80297''N	90-03'38.24477''W	880.40	-000-02'30.0''	0.99993392	0.99986185
D	43-23'16.34105''N	90-02'16.89840''W	859.36	-000-01'34.1''	0.99993257	0.99986515
F	43-19'11.10720''N	89-58'52.60718''W	989.61	000-00'46.3''	0.99993355	0.99984472

Average Combined Scale Factor: 0.99978451

Adjusted Grid Coordinates, (Meters)

Sta.	N:	E:	Z (Geoid):	StErr N:	StErr E:	StErr Z:	DN	DE
C	145233.5291	612043.7357	1103.09	0.0962	0.0972	0.0973		
E	145091.9273	595081.6287	914.80	0.0831	0.0837	0.0840		
D	154179.9652	596918.9980	894.01	0.0812	0.0791	0.0819		
F	146611.7765	601518.4253	1024.16	0.0447	0.0427	0.0452		

Adjusted Geocentric Coordinates, (Metric)

Sta.	X:	Y:	Z:	StErr X(m):	StErr Y(m):	StErr Z(m):
C	12046.5393	-4649368.8693	4353136.26	0.0962	0.0966	0.0979
E	-4919.3737	-4649336.0614	4352910.73	0.0831	0.0833	0.0844
D	-3081.6241	-4643082.1683	4359507.34	0.0812	0.0801	0.0810
F	1518.7615	-4648373.9615	4354092.93	0.0447	0.0437	0.0442

Adjusted Grid Coordinates (N,E) Error Ellipses, 95% CI

Sta.	Semi Major Axis	Semi Minor Axis	Max. Error Az.	Elev.
C	0.2380	0.2355	N 89-13'58.4''E	0.1919
E	0.2050	0.2033	N 86-52'01.5''E	0.1655
D	0.1989	0.1937	N 00-18'30.1''E	0.1588
F	0.1094	0.1046	N 00-33'33.6''E	0.0867

The adjusted observations are shown:

Adjusted Observations
 =====

GPS Vectors: 13 Observations (Meters)

From Sta.	Delta X	Residual	StdRes	StdDev
To Sta.	Delta Y	Residual	StdRes	StdDev
	Delta Z	Residual	StdRes	StdDev
A	11644.3130	0.0898	0.3	0.0972
C	3601.4692	0.2527	0.7	0.0962
	3399.2263	-0.0287	0.1	0.0973
A	-5321.6362	0.0802	0.5	0.0837
E	3634.2495	0.1741	1.1	0.0831
	3173.6413	-0.0239	0.1	0.0840
B	3960.4857	-0.0585	0.3	0.0972
C	-6681.3803	-0.1336	0.7	0.0962
	-7278.9822	0.0326	0.2	0.0973
B	-11167.6912	-0.0836	0.4	0.0791
D	-394.6781	-0.1577	0.8	0.0812
	-907.9287	0.0306	0.2	0.0819
D	15128.1635	-0.0012	0.0	0.0989
C	-6286.7011	0.0043	0.0	0.0981
	-6371.0766	-0.0183	0.1	0.0993
D	-1837.7496	-0.0037	0.0	0.0883
E	-6253.8932	-0.0398	0.3	0.0892
	-6596.6118	0.0579	0.5	0.0900
F	-1116.4126	0.0397	0.4	0.0427
A	-4596.2985	-0.1375	1.5	0.0447
	-4355.7192	0.1870	1.9	0.0452
F	10527.7779	-0.0073	0.0	0.0966
C	-994.9079	0.0298	0.2	0.0954
	-956.6675	-0.0429	0.2	0.0964
F	-6438.1351	0.0013	0.0	0.0795
E	-962.1000	-0.0306	0.3	0.0789
	-1182.2028	0.0277	0.3	0.0797
F	-4600.3856	-0.0069	0.1	0.0749
D	5291.7932	0.0147	0.1	0.0773
	5414.4090	-0.0221	0.2	0.0779
F	6567.2261	-0.0050	0.1	0.0427
B	5686.4222	0.1296	1.3	0.0447
	6322.2236	-0.1681	1.9	0.0452
B	-6567.2263	0.0048	0.1	0.0427
F	-5686.4224	-0.1191	1.2	0.0447
	-6322.2237	0.1570	1.7	0.0452
A	1116.4126	-0.0451	0.5	0.0427
F	4596.2983	0.1430	1.4	0.0447
	4355.7191	-0.1950	1.8	0.0452

The final section displays a variety statistical measures, followed by sideshots if there are any. Side shots would be a point that has only a single GPS vector going to or from the point.

Statistics
 =====

Solution converged in 2 iterations
 Total Observations:39
 Total Unknowns:12
 Degrees of Freedom:27
 Observation Count Sum Squares Std. Error

		of StdRes	of Unit Wt.
Vectors:	39	25.231	0.967
Total:	39	25.231	0.967

Reference Variance:0.934
Standard Error Unit Weight: (+/-)0.967
Passed the Chi-Square test at the 95.00 significance level
14.573 <= 24.955 <= 43.195
All confidence regions were computed using the following factors:
Variance factor: 1.0000
1-D Expansion Factor: 1.9600
2-D Expansion Factor: 2.4477
Expansion factors for 95.00 confidence regions taken from normal distribution table
Sideshots
=====

GPS Vectors and Total Station

The following is a report generated from a project that combined GPS vectors and total station data. The 3D adjustment model **MUST** be used when processing GPS vectors. Notice that the report is very similar to the GPS vector only project report. Explanations of the report are included in the report and are in **bold, normal text**.

```

=====
LEAST SQUARES ADJUSTMENT REPORT
=====
Wed Oct 12 11:11:00
3D Geodetic Model.
Input Raw Files:
  C:\Carlson Projects\GPSandTS\Orginal.M07052.RW5
  C:\Carlson Projects\GPSandTS\OM07052A.GPS
Output File: C:\Carlson Projects\GPSandTS\m07052.RPT
Traverse File: C:\Carlson Projects\GPSandTS\m07052.cls
Curvature, refraction correction: ON
Maximum iterations: 10 , Convergence Limit: 1.000000
1983 State Plane Coordinates, zone:Texas North Central - 4202
Horizontal Units: US Feet
Confidence Interval: 95.00
Geoid Height computed from geoid grid files using the GEOID03 model.
Default Standard Errors:
  Distance: Constant 0.003 ,PPM: 3.000
  Horiz. Angle: Pointing 3.0'' ,Reading: 3.0''
  Vert. Angle: Pointing 3.0'' ,Reading: 3.0''
  Total Station: Centering 0.003 ,Height: 0.005
  Target: Centering 0.003 ,Height: 0.005
  Azimuth: 10''
  Coordinate Control: N:0.010, E:0.010, Z:0.020,
  GPS: Centering:0.005, Vector Err. Factor:1.0
Horizontal Distance from 2004 to 2002 exceeds tolerance:
  Low: 425.153, High: 425.184, Diff: 0.031
Vertical Distance from 2001 to 2028 exceeds tolerance:
  Low: 2.227, High: 2.292, Diff: 0.065
Mark to Mark Vertical Angle from 2003 to 2004 exceeds tolerance:
  Low: 088-39'31.4'' , High: 088-39'41.1'' , Diff: 000-00'09.7''
Mark to Mark distance from 2004 to 2002 exceeds tolerance:
  Low: 425.172, High: 425.203, Diff: 0.031
Mark to Mark Vertical Angle from 2001 to 2028 exceeds tolerance:
  Low: 090-15'26.8'' , High: 090-15'58.8'' , Diff: 000-00'32.0''
Mark to Mark Vertical Angle from 2028 to 2046 exceeds tolerance:
  Low: 089-57'54.2'' , High: 089-58'07.9'' , Diff: 000-00'13.7''
Mark to Mark Vertical Angle from 2028 to 2061 exceeds tolerance:
  Low: 091-00'27.8'' , High: 091-00'41.1'' , Diff: 000-00'13.3''
Mark to Mark Vertical Angle from 2046 to 2083 exceeds tolerance:
  Low: 089-21'42.6'' , High: 089-21'49.0'' , Diff: 000-00'06.4''
Mark to Mark Vertical Angle from 2083 to 2002 exceeds tolerance:
  Low: 088-48'02.7'' , High: 088-48'14.1'' , Diff: 000-00'11.3''
3-DIMENSIONAL ADJUSTMENT REPORT

```

=====
The control coordinates are shown in Geodetic, Grid and Geocentric coordinate systems. Notice that in this example, geoid modeling was used. Notice that the ellipsoid elevation is displayed with the latitudes and longitudes. Orthometric elevations are displayed with the SPC83 grid coordinates.

Unadjusted Observations

=====

Control Coordinates:	1 Observed Points,	2 Fixed Points,	0 Approx. Points			
Sta.	Latitude	Longitude	Z (Ellip.)	StErr N:	StErr E:	StErr Z:NE
5	32-24'35.39394''N	94-49'52.08763''W	253.817	FIXED	FIXED	FIXED
1	32-24'55.19534''N	94-49'33.44142''W	248.060	FIXED	FIXED	FIXED
2003	32-24'42.93765''N	94-49'22.76059''W	255.704	FLOAT	FLOAT	0.000

Grid XYZ

Sta.	N:	E:	Z (Geoid):	StErr N:	StErr E:	StErr Z:
5	6851811.008	3100515.417	340.442	FIXED	FIXED	FIXED
1	6853866.683	3102042.915	334.692	FIXED	FIXED	FIXED
2003	6852660.757	3103001.258	342.330	FLOAT	FLOAT	0.000

Geocentric XYZ

Sta.	X:	Y:	Z:	StErr X(m):	StErr Y(m):	StErr Z(m):
5	-453934.078	-5370753.946	3398927.169	FIXED	FIXED	FIXED
1	-453420.933	-5370467.692	3399441.169	FIXED	FIXED	FIXED
2003	-453160.021	-5370694.806	3399123.657	99747.776	54068.184	84421.637

Notice that in the 3-D model, distances are not reduced to horizontal or grid. Slope distances are reduced to mark to mark distances. A Mark to mark distance is the computed slope distance from the monument to monument.

Mark to Mark Slope Distances: 101 Observations

From Sta.	To Sta.	Dist.	StErr
2003	2001	713.020	0.008
2003	2004	454.830	0.007
2003	2005	95.942	0.006
2004	2002	425.188	0.007
2004	2007	276.272	0.007
2004	2008	291.021	0.007
2004	2009	272.204	0.007
2004	2010	281.020	0.007
2004	2011	307.650	0.007
2004	2012	318.715	0.007
2001	2014	427.579	0.008
2001	2015	426.947	0.008
2001	2017	396.702	0.008
2001	2018	422.118	0.008
2001	2019	236.372	0.007
2001	2020	127.451	0.006
2001	2021	120.691	0.006
2001	2022	116.666	0.006
2001	2023	114.515	0.006
2001	2024	94.819	0.006
2001	2025	124.650	0.006
2001	2026	161.794	0.006
2001	2027	140.828	0.006
2001	2028	494.273	0.007
2001	2029	307.380	0.007
2001	2030	245.443	0.007
2001	2031	247.078	0.007
2001	2032	288.191	0.007
2001	2033	132.091	0.006
2001	2034	327.524	0.008
2001	2035	316.236	0.007
2001	2036	355.847	0.008
2001	2037	194.246	0.006
2001	2038	254.679	0.007
2001	2039	467.332	0.009

2001	2040	533.896	0.009
2001	2041	729.607	0.011
2001	2042	196.157	0.007
2001	2043	225.061	0.007
2001	2044	439.031	0.008
2028	2046	1243.938	0.012
2028	2047	361.901	0.008
2028	2048	513.032	0.009
2028	2049	361.805	0.008
2028	2050	361.250	0.008
2028	2051	703.162	0.011
2028	2052	652.990	0.010
2028	2053	618.889	0.010
2028	2054	643.429	0.010
2028	2055	498.669	0.009
2028	2056	483.619	0.009
2028	2057	511.066	0.009
2028	2058	513.548	0.009
2028	2059	525.440	0.009
2028	2060	11.009	0.005
2028	2061	509.379	0.007
2028	2062	23.319	0.005
2028	2063	35.409	0.005
2028	2064	9.151	0.005
2061	2066	297.764	0.007
2061	2067	200.923	0.007
2061	2068	163.679	0.006
2061	2069	51.856	0.006
2061	2070	26.490	0.005
2061	2071	79.182	0.006
2061	2072	103.125	0.006
2061	2073	186.018	0.006
2061	2074	304.536	0.006
2061	2075	279.157	0.007
2061	2076	277.273	0.007
2046	2078	475.616	0.007
2046	2079	89.991	0.006
2046	2080	20.232	0.005
2046	2081	61.170	0.006
2046	2082	58.911	0.006
2046	2083	814.805	0.009
2078	2085	53.677	0.006
2078	2086	70.701	0.006
2078	2087	90.911	0.006
2083	2002	334.150	0.006
2083	2090	241.764	0.007
2083	2091	133.283	0.006
2002	2094	334.831	0.008
2002	2095	196.403	0.007
2002	2096	192.984	0.006
2002	2097	334.947	0.008
2002	2098	188.679	0.006
2002	2099	114.068	0.006
2002	2100	185.073	0.006
2002	2101	131.057	0.006
2002	2102	120.385	0.006
2002	2103	175.168	0.006
2002	2104	148.159	0.006
2002	2105	119.767	0.006
2002	2106	280.762	0.007
2002	2107	158.649	0.006
2002	2108	153.766	0.006
2002	2109	158.381	0.006
2002	2110	155.574	0.006
2002	2111	167.254	0.006
2074	1	418.772	0.008

Notice that in the 3-D model, vertical angles are considered as separate measurements. Vertical angles have also been converted to mark to mark vertical angles.

Mark to Mark Vertical Angles: 101 Observations

From Sta.	To Sta.	Vertical Ang.	StErr (Sec.)
2003	2001	089-59'14.5''	03.0
2003	2004	088-39'36.2''	03.0
2003	2005	090-52'55.3''	04.2
2004	2002	089-27'12.5''	03.0
2004	2007	089-20'18.1''	04.2
2004	2008	088-24'41.6''	04.2
2004	2009	088-20'22.6''	04.2
2004	2010	089-56'09.3''	04.2
2004	2011	085-01'15.4''	04.2
2004	2012	085-07'02.5''	04.2
2001	2014	089-45'31.0''	04.2
2001	2015	089-46'23.6''	04.2
2001	2017	089-43'33.3''	04.2
2001	2018	090-08'56.5''	04.2
2001	2019	090-40'55.0''	04.2
2001	2020	088-57'10.4''	04.2
2001	2021	088-47'57.4''	04.2
2001	2022	088-42'22.7''	04.2
2001	2023	088-39'17.3''	04.2
2001	2024	092-32'27.5''	04.2
2001	2025	093-33'19.2''	04.2
2001	2026	092-05'05.6''	04.2
2001	2027	091-40'35.1''	04.2
2001	2028	090-15'42.8''	03.0
2001	2029	090-56'04.2''	04.2
2001	2030	091-09'38.1''	04.2
2001	2031	091-09'32.2''	04.2
2001	2032	090-54'51.6''	04.2
2001	2033	089-45'41.3''	04.2
2001	2034	090-44'21.7''	04.2
2001	2035	085-04'16.4''	04.2
2001	2036	088-48'17.8''	04.2
2001	2037	084-26'58.6''	04.2
2001	2038	086-11'06.8''	04.2
2001	2039	087-27'14.1''	04.2
2001	2040	088-19'39.1''	04.2
2001	2041	089-03'57.7''	04.2
2001	2042	090-47'01.9''	04.2
2001	2043	090-49'09.2''	04.2
2001	2044	089-17'51.9''	04.2
2028	2046	089-58'01.1''	03.0
2028	2047	089-48'39.4''	04.2
2028	2048	089-44'04.4''	04.2
2028	2049	089-18'31.3''	04.2
2028	2050	089-19'58.9''	04.2
2028	2051	089-01'11.1''	04.2
2028	2052	089-37'02.7''	04.2
2028	2053	089-38'14.5''	04.2
2028	2054	089-52'42.1''	04.2
2028	2055	089-37'17.0''	04.2
2028	2056	089-36'08.6''	04.2
2028	2057	089-38'13.9''	04.2
2028	2058	089-47'46.7''	04.2
2028	2059	089-48'39.4''	04.2
2028	2060	089-30'16.3''	04.2
2028	2061	091-00'34.4''	03.0
2028	2062	091-58'59.1''	04.2
2028	2063	091-30'44.0''	04.2
2028	2064	093-54'55.9''	04.2
2061	2066	089-28'19.4''	04.2

2061	2067	089-33'11.5''	04.2
2061	2068	090-01'30.6''	04.2
2061	2069	092-49'51.7''	04.2
2061	2070	090-48'22.6''	04.2
2061	2071	088-59'08.0''	04.2
2061	2072	088-34'19.3''	04.2
2061	2073	088-53'32.3''	04.2
2061	2074	089-19'34.5''	03.0
2061	2075	089-02'34.1''	04.2
2061	2076	088-52'41.9''	04.2
2046	2078	091-41'56.5''	03.0
2046	2079	092-56'27.2''	04.2
2046	2080	092-31'47.5''	04.2
2046	2081	090-34'48.1''	04.2
2046	2082	090-25'54.8''	04.2
2046	2083	089-21'45.8''	03.0
2078	2085	091-59'10.6''	04.2
2078	2086	094-33'42.0''	04.2
2078	2087	092-14'47.8''	04.2
2083	2002	088-48'08.4''	03.0
2083	2090	091-45'04.2''	04.2
2083	2091	089-25'09.0''	04.2
2002	2094	089-35'07.6''	04.2
2002	2095	087-20'52.8''	04.2
2002	2096	087-45'48.7''	04.2
2002	2097	086-49'13.5''	04.2
2002	2098	090-03'17.3''	04.2
2002	2099	090-19'24.7''	04.2
2002	2100	087-28'27.3''	04.2
2002	2101	090-26'33.7''	04.2
2002	2102	090-00'15.2''	04.2
2002	2103	089-16'44.7''	04.2
2002	2104	089-08'12.4''	04.2
2002	2105	089-33'50.7''	04.2
2002	2106	090-44'21.2''	04.2
2002	2107	089-04'40.1''	04.2
2002	2108	088-43'52.2''	04.2
2002	2109	088-43'43.5''	04.2
2002	2110	089-04'44.7''	04.2
2002	2111	089-07'46.5''	04.2
2074	1	090-01'27.6''	04.2

Horizontal Angles: 101 Observations

BS Sta.	Occ. Sta.	FS Sta.	Angle	StErr (Sec.)
2001	2003	2004	168-34'02.0''	06.4
2001	2003	2005	059-32'00.0''	09.8
2003	2004	2002	161-02'16.0''	06.6
2003	2004	2007	182-26'35.0''	07.0
2003	2004	2008	157-56'12.0''	06.9
2003	2004	2009	154-18'53.0''	07.0
2003	2004	2010	157-59'26.0''	07.0
2003	2004	2011	146-45'42.0''	06.9
2003	2004	2012	146-45'43.0''	06.8
2003	2001	2014	201-59'12.0''	06.4
2003	2001	2015	202-08'04.0''	06.4
2003	2001	2017	203-41'20.0''	06.5
2003	2001	2018	211-05'58.0''	06.4
2003	2001	2019	030-08'11.0''	06.7
2003	2001	2020	026-31'26.0''	08.3
2003	2001	2021	019-53'03.0''	08.5
2003	2001	2022	012-46'09.0''	08.7
2003	2001	2023	005-13'38.0''	08.7
2003	2001	2024	354-47'10.0''	09.8
2003	2001	2025	042-03'50.0''	08.4
2003	2001	2026	030-58'40.0''	07.5
2003	2001	2027	356-54'03.0''	07.9
2003	2001	2028	097-51'17.0''	06.3

2003	2001	2029	001-21'06.0''	06.4
2003	2001	2030	001-19'07.0''	06.7
2003	2001	2031	009-34'48.0''	06.7
2003	2001	2032	008-21'35.0''	06.5
2003	2001	2033	269-04'45.0''	08.4
2003	2001	2034	016-18'26.0''	06.4
2003	2001	2035	004-12'03.0''	06.4
2003	2001	2036	332-05'53.0''	06.4
2003	2001	2037	291-53'36.0''	07.2
2003	2001	2038	228-39'25.0''	06.9
2003	2001	2039	205-53'14.0''	06.4
2003	2001	2040	169-11'06.0''	06.3
2003	2001	2041	148-15'59.0''	06.2
2003	2001	2042	321-52'43.0''	07.1
2003	2001	2043	346-51'27.0''	06.8
2003	2001	2044	159-49'11.0''	06.4
2001	2028	2046	093-37'03.0''	06.2
2001	2028	2047	270-14'38.0''	06.5
2001	2028	2048	266-14'44.0''	06.4
2001	2028	2049	269-37'27.0''	06.5
2001	2028	2050	267-24'53.0''	06.5
2001	2028	2051	243-22'42.0''	06.3
2001	2028	2052	243-42'14.0''	06.3
2001	2028	2053	238-35'14.0''	06.4
2001	2028	2054	237-25'22.0''	06.4
2001	2028	2055	225-23'07.0''	06.5
2001	2028	2056	223-50'19.0''	06.5
2001	2028	2057	220-23'28.0''	06.5
2001	2028	2058	219-49'11.0''	06.5
2001	2028	2059	218-34'59.0''	06.4
2001	2028	2060	195-53'28.0''	69.6
2001	2028	2061	132-38'59.0''	06.4
2001	2028	2062	185-30'42.0''	33.6
2001	2028	2063	184-05'46.0''	22.8
2001	2028	2064	202-20'41.0''	83.7
2028	2061	2066	062-05'00.0''	06.6
2028	2061	2067	064-50'44.0''	07.1
2028	2061	2068	065-32'29.0''	07.6
2028	2061	2069	081-34'37.0''	15.8
2028	2061	2070	140-10'23.0''	29.6
2028	2061	2071	167-45'04.0''	11.8
2028	2061	2072	182-53'28.0''	10.0
2028	2061	2073	201-26'16.0''	07.7
2028	2061	2074	278-54'20.0''	06.6
2028	2061	2075	277-42'49.0''	06.7
2028	2061	2076	278-40'33.0''	06.7
2028	2046	2078	000-42'06.0''	06.2
2028	2046	2079	344-41'17.0''	10.2
2028	2046	2080	145-10'41.0''	38.1
2028	2046	2081	278-14'36.0''	13.8
2028	2046	2082	119-21'31.0''	14.3
2028	2046	2083	108-00'05.0''	06.1
2046	2078	2085	040-56'15.0''	15.1
2046	2078	2086	030-52'34.0''	12.0
2046	2078	2087	022-46'29.0''	10.0
2046	2083	2002	145-53'36.0''	06.6
2046	2083	2090	356-22'59.0''	06.7
2046	2083	2091	186-44'57.0''	08.5
2083	2002	2004	064-14'35.0''	06.6
2083	2002	2094	252-05'26.0''	06.9
2083	2002	2095	142-56'46.0''	07.8
2083	2002	2096	168-34'25.0''	07.9
2083	2002	2097	100-03'00.0''	06.8
2083	2002	2098	224-51'23.0''	07.8
2083	2002	2099	069-34'02.0''	09.0
2083	2002	2100	196-45'31.0''	08.0

2083	2002	2101	113-49'02.0''	08.8
2083	2002	2102	118-25'07.0''	09.2
2083	2002	2103	159-27'09.0''	08.1
2083	2002	2104	148-55'00.0''	08.6
2083	2002	2105	118-24'58.0''	09.3
2083	2002	2106	091-49'18.0''	07.0
2083	2002	2107	172-11'19.0''	08.4
2083	2002	2108	184-15'44.0''	08.5
2083	2002	2109	192-14'20.0''	08.4
2083	2002	2110	202-13'25.0''	08.5
2083	2002	2111	213-52'09.0''	08.2
2061	2074	1	210-25'35.0''	06.9

Unadjusted vector data is shown:

GPS Vectors: 6 Observations (Meters)

Multiplication factor applied to Variance and Covariance values: 2 (2 to 1 power)

From Sta.	Delta X	Variance Delta X	Covariance XY	Bearing
To Sta.	Delta Y	Variance Delta Y	Covariance XZ	H.Distance

Delta Z	Variance Delta Z	Covariance YZ	V.Distance
1			
315.769	5.28E-005	8.497E-007	S 28-38'43.2''E
2002			
-373.192	5.485E-005	9.31E-009	721.852
-531.162	5.601E-005	-3.512E-006	6.769
2001			
22.748	5.131E-005	4.072E-007	S 05-23'03.0''E
2002			
-259.604	5.233E-005	-4.792E-008	474.172
-396.166	5.287E-005	-1.689E-006	4.409
1			
293.021	5.091E-005	2.949E-007	S 61-50'19.5''E
2001			
-113.588	5.179E-005	2.886E-009	342.026
-134.997	5.23E-005	-1.32E-006	2.373
5			
806.171	5.339E-005	1.024E-006	N 60-22'21.5''E
2001			
172.665	5.533E-005	-5.993E-009	907.399
379.011	5.618E-005	-3.755E-006	0.568
5			
828.920	5.363E-005	1.144E-006	S 88-23'05.3''E
2002			
-86.940	5.632E-005	-1.107E-009	833.628
-17.154	5.752E-005	-4.498E-006	5.006
1			
-513.151	5.357E-005	1.147E-006	S 38-37'06.7''W
5			
-286.253	5.649E-005	-3.598E-009	780.683
-514.008	5.783E-005	-4.666E-006	1.702

Option traverse and GPS closures are shown if a Closure File is specified:

Traverse Closures

=====

Traverse points:

2003,2004,2002,2083,2046,2028,2001,2003,2004

Loop Traverse; Interior direction reference;

Compute angle closure.

Compute vertical closure.

BS	IP	FS	Angle	FS H. Dist.	FS V. Dist.
2003	2004	2002	161-02'16.0''	425.135	4.056
2004	2002	2083	295-45'25.0''	334.052	-6.984
2002	2083	2046	214-06'24.0''	814.691	-9.063
2083	2046	2028	251-59'55.0''	1243.841	-0.717
2046	2028	2001	266-22'57.0''	494.229	2.259
2028	2001	2003	262-08'43.0''	712.964	-0.157
2001	2003	2004	168-34'02.0''	454.670	10.636

Closing Az: S 07-30'32.0''E
 Computed Closing Az: S 07-30'50.0''E
 Total angular error: 000-00'18.0''
 Angular error per point: 000-00'02.6''
 Correct Ending Coordinates, North: 6852209.986 East: 3103060.674
 Ending Coordinates, North: 6852209.966 East: 3103060.633
 Error, N: -0.020 E: -0.041 Total: 0.046 Brg: N 63-58'04.2''E
 Distance Traversed: 4479.583 Closure: 1: 97122
 Correct Ending Elevation: 352.964
 Ending Elevation: 352.993
 Elevation Error: 0.029

Closure After Angle Adjustment					
2003	2004	2002	161-02'18.6''	425.135	4.056
2004	2002	2083	295-45'27.6''	334.052	-6.984
2002	2083	2046	214-06'26.6''	814.691	-9.063
2083	2046	2028	251-59'57.6''	1243.841	-0.717
2046	2028	2001	266-22'59.6''	494.229	2.259
2028	2001	2003	262-08'45.6''	712.964	-0.157
2001	2003	2004	168-34'04.6''	454.670	10.636

Closing Az: S 07-30'32.0''E
 Computed Closing Az: S 07-30'32.0''E
 Total angular error: 000-00'00.0''
 Angular error per point: 000-00'00.0''
 Correct Ending Coordinates, North: 6852209.986 East: 3103060.674
 Ending Coordinates, North: 6852209.949 East: 3103060.606
 Error, N: -0.037 E: -0.069 Total: 0.078 Brg: N 61-38'31.8''E
 Distance Traversed: 4479.583 Closure: 1: 57506

Adjusted coordinates are shown in Geographic, Grid and Geocentric formats, as well as Grid Factor, Elevation Factor and Combined Factors.

Adjusted Geographic Coordinates

Sta.	Latitude	Longitude	Z (Ellip.)	Conv. Ang.	Grid Factor	Z Factor	C
2003	32-24'42.93768''N	94-49'22.76059''W	255.704	002-00'19.5''	0.99993450	0.99998777	
2001	32-24'49.95503''N	94-49'21.90114''W	255.874	002-00'20.0''	0.99993412	0.99998776	
2004	32-24'38.45922''N	94-49'22.25214''W	266.347	002-00'19.8''	0.99993474	0.99998728	
2002	32-24'34.62991''N	94-49'20.19858''W	270.409	002-00'20.9''	0.99993494	0.99998707	
2028	32-24'51.12045''N	94-49'27.50058''W	253.618	002-00'16.9''	0.99993406	0.99998787	
2046	32-24'39.37540''N	94-49'31.84052''W	254.353	002-00'14.6''	0.99993469	0.99998783	
2061	32-24'48.33434''N	94-49'32.45093''W	244.650	002-00'14.2''	0.99993421	0.99998830	
2074	32-24'51.07272''N	94-49'33.93275''W	248.233	002-00'13.4''	0.99993406	0.99998813	
2078	32-24'43.84653''N	94-49'30.11725''W	240.257	002-00'15.5''	0.99993445	0.99998851	
2083	32-24'34.70477''N	94-49'24.09435''W	263.423	002-00'18.8''	0.99993494	0.99998740	

Average Combined Scale Factor: 0.99992226

Adjusted Grid Coordinates, (US Feet)

Sta.	N:	E:	Z (Geoid):	StErr N:	StErr E:	StErr Z:	DN	DE
2003	6852660.757	3103001.258	342.330	0.015	0.012	0.000	0.000	-0.000
2001	6853372.046	3103050.070	342.503	0.013	0.012	0.007		
2004	6852209.985	3103060.655	352.971	0.015	0.012	0.006		
2002	6851829.410	3103250.131	357.031	0.014	0.012	0.007		
2028	6853472.952	3102566.255	340.248	0.014	0.011	0.008		
2046	6852273.760	3102235.966	340.978	0.016	0.016	0.012		
2061	6853176.730	3102152.013	331.279	0.017	0.009	0.008		
2074	6853448.848	3102015.391	334.863	0.012	0.008	0.007		
2078	6852730.484	3102367.798	326.884	0.019	0.017	0.014		
2083	6851825.283	3102916.106	350.046	0.015	0.013	0.009		

Adjusted Geocentric Coordinates, (Metric)

Sta.	X:	Y:	Z:	StErr X(m):	StErr Y(m):	StErr Z(m):
------	----	----	----	-------------	-------------	-------------

2003	-453160.021	-5370694.805	3399123.658	0.005	0.002	0.003
2001	-453127.904	-5370581.279	3399306.173	0.004	0.003	0.003
2004	-453153.229	-5370772.333	3399008.931	0.005	0.003	0.003
2002	-453105.160	-5370840.885	3398910.010	0.004	0.003	0.003
2028	-453272.031	-5370549.223	3399336.111	0.004	0.003	0.003
2046	-453401.361	-5370733.113	3399030.798	0.005	0.004	0.005
2061	-453404.600	-5370581.884	3399262.193	0.005	0.002	0.003
2074	-453439.457	-5370534.490	3399333.990	0.004	0.002	0.002
2078	-453349.975	-5370659.726	3399144.769	0.006	0.005	0.005
2083	-453206.345	-5370829.304	3398910.815	0.005	0.003	0.004

Error ellipses are shown:

Adjusted Grid Coordinates (N,E) Error Ellipses, 95% CI

Sta.	Semi Major Axis	Semi Minor Axis	Max. Error Az.	Elev.
2003	0.038	0.030	N 16-10'14.4''E	0.021
2001	0.032	0.028	N 20-45'50.0''E	0.021
2004	0.037	0.030	N 14-44'50.6''E	0.021
2002	0.035	0.028	N 21-39'49.4''E	0.021
2028	0.035	0.025	N 18-55'31.6''E	0.020
2046	0.039	0.038	S 53-41'52.7''E	0.029
2061	0.041	0.022	S 00-10'19.3''E	0.017
2074	0.030	0.019	S 12-01'00.7''E	0.015
2078	0.047	0.042	S 05-10'11.3''E	0.032
2083	0.037	0.031	N 01-23'07.4''E	0.023

Adjusted observations are shown:

Adjusted Observations

=====

Adjusted Mark to Mark Distances

From Sta.	To Sta.	Distance	Residual	StdRes.	StdDev
2003	2001	713.017	-0.003	0.3	0.007
2003	2004	454.828	-0.001	0.2	0.006
2004	2002	425.187	-0.001	0.1	0.006
2001	2028	494.269	-0.003	0.5	0.007
2028	2046	1243.943	0.005	0.4	0.011
2028	2061	509.377	-0.003	0.4	0.007
2061	2074	304.534	-0.002	0.3	0.006
2046	2078	475.616	0.000	0.0	0.007
2046	2083	814.805	0.000	0.0	0.009
2083	2002	334.150	-0.001	0.1	0.006
2074	1	418.771	-0.001	0.1	0.008
Root Mean Square (RMS)			0.002		

Adjusted Horizontal Angles

BS Sta.	Occ. Sta.	FS Sta.	Angle	Residual	StdRes	StdDev (Sec.)
2001	2003	2004	168-34'03.9''	01.9	0.3	05.7
2003	2004	2002	161-02'21.1''	05.1	0.8	05.8
2003	2001	2028	097-51'18.5''	01.5	0.2	04.1
2001	2028	2046	093-37'04.9''	01.9	0.3	03.6
2001	2028	2061	132-39'02.9''	03.9	0.6	05.6
2028	2061	2074	278-54'30.8''	10.8	1.6	05.1
2028	2046	2078	000-42'06.0''	-00.0	0.0	06.2
2028	2046	2083	108-00'05.4''	00.4	0.1	03.8
2046	2083	2002	145-53'29.9''	-06.1	0.9	05.4
2083	2002	2004	064-14'26.4''	-08.6	1.3	04.9
2061	2074	1	210-25'42.6''	07.6	1.1	06.1
Root Mean Square (RMS)				05.5		

Adjusted Mark to Mark Vertical Angles

From Sta.	To Sta.	Vertical Ang.	Residual	StdRes	StdDev (Sec.)
2003	2001	089-59'14.3''	00.2	0.1	02.1
2003	2004	088-39'35.2''	01.0	0.3	02.7
2004	2002	089-27'11.5''	00.9	0.3	02.8
2001	2028	090-15'43.9''	-01.1	0.4	02.7
2028	2046	089-58'04.3''	-03.2	1.1	02.0

2028	2061	091-00'34.2''	00.2	0.1	02.7
2061	2074	089-19'34.3''	00.1	0.0	02.9
2046	2078	091-41'56.5''	00.0	0.0	03.0
2046	2083	089-21'47.9''	-02.1	0.7	02.6
2083	2002	088-48'09.3''	-00.9	0.3	02.9
2074	1	090-01'27.2''	00.4	0.1	03.6

Root Mean Square (RMS) 01.3

GPS Vectors: 6 Observations (Meters)

From Sta.	Delta X	Residual	StdRes	StdDev
To Sta.	Delta Y	Residual	StdRes	StdDev
	Delta Z	Residual	StdRes	StdDev
1	315.7728	0.0038	0.5	0.0037
2002	-373.1927	-0.0003	0.0	0.0043
	-531.1593	0.0028	0.4	0.0022
2001	22.7442	-0.0036	0.5	0.0031
2002	-259.6057	-0.0018	0.3	0.0046
	-396.1633	0.0025	0.3	0.0029
1	293.0285	0.0078	1.1	0.0036
2001	-113.5870	0.0009	0.1	0.0039
	-134.9961	0.0006	0.1	0.0022
5	806.1738	0.0033	0.5	0.0036
2001	172.6667	0.0022	0.3	0.0039
	379.0044	-0.0066	0.9	0.0022
5	828.9181	-0.0021	0.3	0.0037
2002	-86.9390	0.0009	0.1	0.0043
	-17.1589	-0.0048	0.6	0.0022
1	-513.1453	0.0057	0.8	0.0000
5	-286.2538	-0.0010	0.1	0.0000
	-514.0005	0.0075	1.0	0.0000

Statistics are shown:

Statistics

=====

Solution converged in 1 iterations

Total Observations:54

Total Unknowns:30

Degrees of Freedom:24

Observation	Count	Sum Squares of StdRes	Std. Error of Unit Wt.
Coordinate	3	0.000	0.005
Angles:	11	7.641	1.250
Distances:	11	0.874	0.423 (Mark-to-Mark)
VertAngles	11	2.079	0.652 (Mark-to-Mark)
Vectors:	18	5.298	0.814
Total:	54	15.892	0.814

Reference Variance:0.662

Standard Error Unit Weight: (+/-)0.814

Passed the Chi-Square test at the 95.00 significance level

12.401 <= 15.873 <= 39.364

All confidence regions were computed using the following factors:

Variance factor: 1.0000

1-D Expansion Factor: 1.9600

2-D Expansion Factor: 2.4477

Expansion factors for 95.00 confidence regions taken from normal distribution table

Sideshots are shown:

Sideshots

=====

From	To	Bearing	Dist.	N	E	StDev. N	StDev. E
2003	2005	N 63-27'32.8''E	95.923	6852703.619	3103087.072	0.002	0.005
2004	2007	S 05-03'48.2''E	276.232	6851934.831	3103085.034	0.003	0.005
2004	2008	S 29-34'11.2''E	290.887	6851956.985	3103204.202	0.003	0.005
2004	2009	S 33-11'30.2''E	272.068	6851982.307	3103209.596	0.003	0.005
2004	2010	S 29-30'57.2''E	280.998	6851965.455	3103199.093	0.003	0.005
2004	2011	S 40-44'41.2''E	306.465	6851977.800	3103260.681	0.004	0.005
2004	2012	S 40-44'40.2''E	317.533	6851969.414	3103267.904	0.004	0.005

2001	2014	N 25-54'44.8''E	427.542	6853756.604	3103236.905	0.004	0.005
2001	2015	N 26-03'36.8''E	426.910	6853755.553	3103237.618	0.004	0.005
2001	2017	N 27-36'52.8''E	396.667	6853723.526	3103233.934	0.004	0.005
2001	2018	N 35-01'30.8''E	422.084	6853717.690	3103292.320	0.004	0.005
2001	2019	S 34-03'43.8''W	236.337	6853176.257	3102917.700	0.003	0.004
2001	2020	S 30-26'58.8''W	127.420	6853262.200	3102985.496	0.003	0.004
2001	2021	S 23-48'35.8''W	120.655	6853261.660	3103001.361	0.003	0.004
2001	2022	S 16-41'41.8''W	116.627	6853260.335	3103016.566	0.003	0.004
2001	2023	S 09-09'10.8''W	114.475	6853259.029	3103031.860	0.003	0.004
2001	2024	S 01-17'17.2''E	94.718	6853277.351	3103052.199	0.003	0.004
2001	2025	S 45-59'22.8''W	124.400	6853285.614	3102960.600	0.003	0.004
2001	2026	S 34-54'12.8''W	161.674	6853239.454	3102957.561	0.003	0.004
2001	2027	S 00-49'35.8''W	140.757	6853231.304	3103048.039	0.003	0.004
2001	2029	S 05-16'38.8''W	307.316	6853066.033	3103021.804	0.003	0.005
2001	2030	S 05-14'39.8''W	245.374	6853127.699	3103027.642	0.003	0.005
2001	2031	S 13-30'20.8''W	247.009	6853131.868	3102992.383	0.003	0.005
2001	2032	S 12-17'07.8''W	288.132	6853090.512	3102988.761	0.003	0.005
2001	2033	S 86-59'42.2''E	132.080	6853365.122	3103181.968	0.003	0.004
2001	2034	S 20-13'58.8''W	327.472	6853064.781	3102936.818	0.004	0.005
2001	2035	S 08-07'35.8''W	315.040	6853060.169	3103005.536	0.003	0.005
2001	2036	S 23-58'34.2''E	355.742	6853046.999	3103194.628	0.004	0.005
2001	2037	S 64-10'51.2''E	193.319	6853287.849	3103224.091	0.003	0.004
2001	2038	N 52-34'57.8''E	254.094	6853526.437	3103251.880	0.004	0.005
2001	2039	N 29-48'46.8''E	466.833	6853777.095	3103282.166	0.004	0.006
2001	2040	N 06-53'21.2''W	533.626	6853901.819	3102986.062	0.004	0.006
2001	2041	N 27-48'28.2''W	729.453	6854017.259	3102709.775	0.005	0.007
2001	2042	S 34-11'44.2''E	196.123	6853209.828	3103160.295	0.003	0.004
2001	2043	S 09-13'00.2''E	225.021	6853149.930	3103086.112	0.003	0.004
2001	2044	N 16-15'16.2''W	438.964	6853793.463	3102927.202	0.004	0.006
2028	2047	N 12-01'29.3''E	361.871	6853826.882	3102641.646	0.004	0.006
2028	2048	N 08-01'35.3''E	512.986	6853980.913	3102637.884	0.004	0.006
2028	2049	N 11-24'18.3''E	361.751	6853827.559	3102637.789	0.004	0.006
2028	2050	N 09-11'44.3''E	361.197	6853829.507	3102623.977	0.004	0.006
2028	2051	N 14-50'26.7''W	703.004	6854152.505	3102386.192	0.004	0.008
2028	2052	N 14-30'54.7''W	652.924	6854105.035	3102402.608	0.004	0.007
2028	2053	N 19-37'54.7''W	618.829	6854055.808	3102358.344	0.004	0.007
2028	2054	N 20-47'46.7''W	643.378	6854074.413	3102337.826	0.004	0.007
2028	2055	N 32-50'01.7''W	498.619	6853891.915	3102295.902	0.004	0.006
2028	2056	N 34-22'49.7''W	483.570	6853872.045	3102293.190	0.004	0.006
2028	2057	N 37-49'40.7''W	511.016	6853876.581	3102252.853	0.005	0.006
2028	2058	N 38-23'57.7''W	513.505	6853875.386	3102247.297	0.005	0.006
2028	2059	N 39-38'09.7''W	525.396	6853877.566	3102231.101	0.005	0.006
2028	2060	N 62-19'40.7''W	11.008	6853478.064	3102556.506	0.003	0.005
2028	2062	N 72-42'26.7''W	23.303	6853479.879	3102544.005	0.003	0.005
2028	2063	N 74-07'22.7''W	35.394	6853482.635	3102532.211	0.003	0.005
2028	2064	N 55-52'27.7''W	9.129	6853478.073	3102558.698	0.003	0.005
2061	2066	S 63-29'05.8''E	297.728	6853043.815	3102418.425	0.004	0.006
2061	2067	S 60-43'21.8''E	200.901	6853078.482	3102327.251	0.003	0.006
2061	2068	S 60-01'36.8''E	163.667	6853094.964	3102293.790	0.003	0.005
2061	2069	S 43-59'28.8''E	51.789	6853139.471	3102187.983	0.003	0.005
2061	2070	S 14-36'17.2''W	26.485	6853151.101	3102145.334	0.003	0.005
2061	2071	S 42-10'58.2''W	79.163	6853118.070	3102098.855	0.003	0.005
2061	2072	S 57-19'22.2''W	103.085	6853121.074	3102065.243	0.003	0.005
2061	2073	S 75-52'10.2''W	185.969	6853131.330	3101971.671	0.003	0.005
2061	2075	N 27-51'16.8''W	279.096	6853423.489	3102021.610	0.003	0.006
2061	2076	N 26-53'32.8''W	277.199	6853423.952	3102026.631	0.003	0.006
2046	2079	N 00-05'13.2''E	89.866	6852363.626	3102236.103	0.004	0.005
2046	2080	S 19-25'22.8''E	20.210	6852254.700	3102242.687	0.004	0.005
2046	2081	N 66-21'27.8''W	61.163	6852298.288	3102179.937	0.004	0.005
2046	2082	S 45-14'32.8''E	58.905	6852232.284	3102277.794	0.004	0.005
2078	2085	S 57-02'17.3''W	53.640	6852701.299	3102322.792	0.005	0.006
2078	2086	S 46-58'36.3''W	70.472	6852682.401	3102316.278	0.005	0.006
2078	2087	S 38-52'31.3''W	90.834	6852659.769	3102310.788	0.005	0.006
2083	2090	N 60-12'59.4''W	241.632	6851945.307	3102706.391	0.004	0.005
2083	2091	S 49-51'01.4''E	133.266	6851739.355	3103017.970	0.004	0.005
2002	2094	S 18-37'02.6''E	334.796	6851512.133	3103357.014	0.004	0.005

2002	2095	N 52-14'17.4''E	196.177	6851949.545	3103405.221	0.003	0.005
2002	2096	N 77-51'56.4''E	192.822	6851869.942	3103438.645	0.004	0.005
2002	2097	N 09-20'31.4''E	334.404	6852159.379	3103304.414	0.004	0.005
2002	2098	S 45-51'05.6''E	188.665	6851698.001	3103385.505	0.003	0.005
2002	2099	N 21-08'26.6''W	114.057	6851935.791	3103208.995	0.003	0.005
2002	2100	S 73-56'57.6''E	184.878	6851778.293	3103427.802	0.003	0.005
2002	2101	N 23-06'33.4''E	131.043	6851949.938	3103301.564	0.003	0.005
2002	2102	N 27-42'38.4''E	120.375	6851935.979	3103306.106	0.003	0.005
2002	2103	N 68-44'40.4''E	175.141	6851892.903	3103413.358	0.003	0.005
2002	2104	N 58-12'31.4''E	148.131	6851907.449	3103376.038	0.003	0.005
2002	2105	N 27-42'29.4''E	119.754	6851935.431	3103305.813	0.003	0.005
2002	2106	N 01-06'49.4''E	280.717	6852110.074	3103255.587	0.003	0.005
2002	2107	N 81-28'50.4''E	158.616	6851852.908	3103406.997	0.003	0.005
2002	2108	S 86-26'44.6''E	153.716	6851819.880	3103403.552	0.003	0.005
2002	2109	S 78-28'08.6''E	158.330	6851797.760	3103405.265	0.003	0.005
2002	2110	S 68-29'03.6''E	155.542	6851772.364	3103394.835	0.003	0.005
2002	2111	S 56-50'19.6''E	167.222	6851737.940	3103390.118	0.003	0.005

Vertical Adjustment

This is the vertical adjustment report for the 2D/1D model adjustment:

LEAST SQUARES VERTICAL ADJUSTMENT REPORT

Fri Apr 5 12:00:00

2D Geodetic Model.

Input Raw Files:

C:\Carlson Projects\SurvNetTut01.rw5

Output File: C:\Carlson Projects\SurvNet\SurvNetTut01.RPT

Output CRD File: C:\Carlson Projects\SurvNet\SurvNetTut01.crd

Curvature, refraction correction: ON

VERTICAL BENCHMARKS

Station	Elevation	Std. Error
TR1	100.0000	FIXED
TR100	0.0000	FIXED

POINTS TO BE ADJUSTED

Station

TR2, TR3, TR4, TR5, TR6, TR7, TR7B, TR7C

MEASUREMENT SUMMARY

From	To	Elev. Diff. (unadjusted)	StdErr
TR1	TR100	-2.4904	0.0147
TR1	TR2	-2.2258	0.0147
TR2	TR3	-4.0634	0.0147
TR3	TR4	-1.5208	0.0143
TR4	TR5	7.2363	0.0152
TR5	TR6	-2.1809	0.0151
TR6	TR7	0.1132	0.0145
TR7	TR7B	5.1236	0.0151
TR7	TR100	1.6154	0.0151
TR7	TR7C	3.3496	0.0163
TR7B	TR1	-2.3590	0.0157
TR7B	TR3	-8.5867	0.0228
TR7B	TR5	-3.0523	0.0174
TR7C	TR100	2.5981	0.0145

STATISTICAL SUMMARY

Total Unknown Elevations:8

Total Elev. Routes:14

Total Fixed BM's:2

Total non-fixed BM's:0

Degrees of freedom:6

ADJUSTED ELEVATIONS

Station	Adjusted Elev	Standard Dev.	Error Ellipse at 95% CI
TR1	100.0000	FIXED	0.00000
TR100	0.0000	FIXED	0.00000
TR2	84.5686	41.14026	80.63343

TR3	67.3188	45.22845	88.64613
TR4	58.3139	52.40847	102.71870
TR5	57.5924	46.54994	91.23621
TR6	41.6824	48.31011	94.68608
TR7	28.6393	33.33185	65.32922
TR7B	67.3621	35.07818	68.75197
TR7C	13.6980	38.41413	75.29031

ADJUSTED MEASUREMENT SUMMARY

From	To	Elev. Diff. (adjusted)	Residuals	Std. Dev.
TR1	TR100	-100.0000	-97.5096	0.000
TR1	TR2	-15.4314	-13.2056	41.140
TR2	TR3	-17.2498	-13.1864	41.122
TR3	TR4	-9.0049	-7.4841	41.178
TR4	TR5	-0.7215	-7.9578	41.980
TR5	TR6	-15.9099	-13.7290	41.104
TR6	TR7	-13.0431	-13.1563	40.602
TR7	TR7B	38.7229	33.5992	34.757
TR7	TR100	-28.6393	-30.2547	33.332
TR7	TR7C	-14.9413	-18.2909	39.239
TR7B	TR1	32.6379	34.9968	35.078
TR7B	TR3	-0.0433	8.5433	41.824
TR7B	TR5	-9.7698	-6.7175	39.342
TR7C	TR100	-13.6980	-16.2961	38.414

Vertical Sideshots

Station	Elevation
TR1B	98.0007
100	105.4404
101	105.4607
102	108.9285
103	104.9080
104	105.7794
105	105.3716
106	106.9132
TR2C	80.7241
CC1	81.0316
TR2B	82.4713
125	91.8345
126	91.1025
127	91.4001
128	88.7827
129	89.2359
130	89.7661
131	89.1042
132	89.2006
133	88.9114
150	73.7072
151	75.1551
152	75.3588
153	75.9565
154	76.0468
155	75.3200
156	74.6387
157	75.2392
158	74.5438
159	73.2178
160	67.3246
161	67.9606
162	69.2969
200	64.3422
201	62.2583
202	63.0329
203	63.3425
204	65.8137
205	62.8474
MN2	37.7423

CC200	39.9471
CC201	40.0756
225	49.5442
226	47.5001
227	47.9010
228	47.3816
229	47.5252
230	45.7544
231	45.6291
232	44.9244
233	45.0296
234	49.3004
235	48.8310
236	51.1671
237	50.9847
238	48.0367
239	50.4217
240	47.6531
241	48.1279
242	48.5686
300	30.4839
301	33.9766
302	33.5054
303	33.9313
304	33.1538
305	33.6198
306	27.7555
307	28.5585
308	29.1737
CC300	34.4190
CC301	33.9808
GPS1	31.1822
CC400	64.5606
CC500	59.4460
TR1BCK	63.1440
CC600	67.3962
500	64.7735
501	67.3561
502	71.5196
503	71.5325
504	72.8553
505	72.7991
506	72.5789
507	72.5883
508	74.6496
509	74.5704
510	74.6176
511	76.0482
512	72.7430
513	73.4538
1000	116.6556
1001	117.9243
1002	120.3538
1003	116.8152
1004	104.9979
1005	105.7780
1006	105.4189
1007	106.1852
1008	103.5893
1009	103.5550
1010	105.2201
1011	119.9053
1012	116.3988
1013	105.0764
1014	106.2287
1015	106.4953

1016	105.6196
1017	106.2795
1018	105.5693
1019	105.3773
1020	104.2048
1021	108.0350
1022	106.9444
1023	107.2728
1024	107.4679
1025	109.0913
1026	109.3273
1100	34.7236
1101	33.1876
1102	23.4532
1103	23.6922
1104	20.4123
1105	30.4324
1106	29.3272
1107	30.9160
1108	20.5319
1109	22.5550
1110	22.4347
1111	21.7911
1112	21.2860
1113	21.7111
1114	34.2394
1115	33.5384
1116	34.0657
1117	34.0688
1118	40.6604
1200	6.1629
1201	5.1502
1202	6.9316
1203	16.0098
1204	16.4797
1205	8.7831
1206	8.9042
1207	6.3770
1208	8.4799
1209	7.3919
1210	19.2004
1211	6.7228
1212	7.0911
1213	7.0724

This is a sample report of a differential level network adjustment:

```
=====
LEAST SQUARES ADJUSTMENT REPORT
=====
```

Fri Apr 5 12:00:00

Output File: C:\SurvNET.Seminar\SurvNetExamples1\LevelNetwork\network1.RPT

Curvature, refraction correction: ON

Maximum iterations: 10 , Convergence Limit: 0.002000

Local Coordinate System, Scale Factor: 1.000000

Horizontal Units: US Feet

Confidence Interval: 95.00

Default Standard Errors:

Distance: Constant 0.003 ,PPM: 5.000

Horiz. Angle: Pointing 1.0'' ,Reading: 1.0''

Vert. Angle: Pointing 3.0'' ,Reading: 3.0''

Total Station: Centering 0.003 ,Height: 0.010

Target: Centering 0.003 ,Height: 0.010

Azimuth: 5''

Coordinate Control: N:0.010, E:0.010, Z:0.010,

```
HORIZONTAL ADJUSTMENT REPORT
```

Unadjusted Observations

```

Control Coordinates:      0 Observed Points,      2 Fixed Points,      0 Approx. Points
Sta.      N:      E:      StErr N:      StErr E:
BMX      1000.000      1000.000      FIXED      FIXED
BMX      1000.000      1000.000      FIXED      FIXED
LEAST SQUARES VERTICAL ADJUSTMENT REPORT

```

Fri Apr 5 12:00:00

Level File: DiffLevel.tlv

Output File: C:\SurvNET.Seminar\SurvNetExamples1\LevelNetwork\network1.RPT

Curvature, refraction correction: ON

Differential Leveling Standard Errors

Avg. Dist. to BS/FS:50.0

Rod Reading Err. per 100'/m:0.000

Collimation Err. (sec.) 3.0:

Unadjusted Trigonometric Level Report

```

Header1:  Operator: loop Name:  Project Name:
Header2:  date: pressure: 1630170 temperature: time:
Benchmark: Name: BMX      El:  100.000 Desc: BMX

```

Differential Level Loop:

```

Rod:      Rod Ht.      0.000
Backsight: Name: BMX      VD:  5.300 HD:  100.000 HI:  105.300 Desc: BMX
Average HI:  105.300
Foresight: Name: 1      VD:  6.000 HD:  100.000 EL:  99.300
Average EL:  99.300
Backsight: Name: 1      VD:  0.700 HD:  100.000 HI:  100.000
Average HI:  100.000
Foresight: Name: SS1      VD:  3.500 HD:  100.000 EL:  96.500
Average EL:  96.500
Foresight: Name: SS2      VD:  4.500 HD:  100.000 EL:  95.500
Average EL:  95.500
Foresight: Name: 2      VD:  1.000 HD:  100.000 EL:  99.000
Average EL:  99.000
Backsight: Name: 2      VD:  15.000 HD:  100.000 HI:  114.000
Average HI:  114.000
Foresight: Name: A      VD:  8.900 HD:  100.000 EL:  105.100
Average EL:  105.100
Backsight: Name: A      VD:  6.800 HD:  100.000 HI:  111.900
Average HI:  111.900
Foresight: Name: 3      VD:  5.100 HD:  100.000 EL:  106.800
Average EL:  106.800
Backsight: Name: 3      VD:  2.300 HD:  100.000 HI:  109.100
Average HI:  109.100
Foresight: Name: 4      VD:  6.200 HD:  100.000 EL:  102.900
Average EL:  102.900
Backsight: Name: 4      VD:  16.300 HD:  100.000 HI:  119.200
Average HI:  119.200
Foresight: Name: BMX      VD:  11.760 HD:  100.000 EL:  107.440
Average EL:  107.440
Benchmark: Name: BMX      El:  107.500 Desc: BMX
Backsight: Name: BMX      VD:  6.500 HD:  100.000 HI:  114.000 Desc: BMX
Average HI:  114.000
Foresight: Name: 5      VD:  5.800 HD:  100.000 EL:  108.200
Average EL:  108.200
Backsight: Name: 5      VD:  10.000 HD:  100.000 HI:  118.200
Average HI:  118.200
Foresight: Name: C      VD:  11.950 HD:  100.000 EL:  106.250
Average EL:  106.250
Backsight: Name: C      VD:  10.500 HD:  100.000 HI:  116.750
Average HI:  116.750
Foresight: Name: 6      VD:  5.300 HD:  100.000 EL:  111.450
Average EL:  111.450
Backsight: Name: 6      VD:  4.800 HD:  100.000 HI:  116.250
Average HI:  116.250
Foresight: Name: BMX      VD:  16.130 HD:  100.000 EL:  100.120

```

Benchmark: Name: BMX	El: 100.000	Average EL: 100.120
Benchmark: Name: BMY	El: 107.500	
Backsight: Name: BMY	VD: 8.400 HD: 100.000 HI: 115.900	Average HI: 115.900
Foresight: Name: 7	VD: 3.800 HD: 100.000 EL: 112.100	Average EL: 112.100
Backsight: Name: 7	VD: 1.500 HD: 100.000 HI: 113.600	Average HI: 113.600
Foresight: Name: B	VD: 9.100 HD: 100.000 EL: 104.500	Average EL: 104.500
Backsight: Name: B	VD: 6.900 HD: 100.000 HI: 111.400	Average HI: 111.400
Foresight: Name: 8	VD: 5.210 HD: 100.000 EL: 106.190	Average EL: 106.190
Backsight: Name: 8	VD: 5.000 HD: 100.000 HI: 111.190	Average HI: 111.190
Foresight: Name: C	VD: 4.990 HD: 100.000 EL: 106.200 Desc: TBM	Average EL: 106.200
Backsight: Name: A	VD: 6.300 HD: 100.000 HI: 111.400 Desc: TBM1	Average HI: 111.400
Foresight: Name: B	VD: 6.980 HD: 100.000 EL: 104.420 Desc: TBM3	

VERTICAL BENCHMARKS

Station	Elevation	Std. Error
BMY	107.5000	FIXED
BMX	100.0000	FIXED

POINTS TO BE ADJUSTED

Station
1, 2, A, 3, 4, 5, C, 6, 7, B, 8

MEASUREMENT SUMMARY

From	To	Elev. Diff. (unadjusted)	StdErr
BMX	1	-0.7000	0.0015
1	2	-0.3000	0.0029
2	A	6.1000	0.0033
A	3	1.7000	0.0036
3	4	-3.9000	0.0039
4	BMY	4.5400	0.0041
BMY	5	0.7000	0.0015
5	C	-1.9500	0.0021
C	6	5.2000	0.0025
6	BMX	-11.3300	0.0029
BMY	7	4.6000	0.0015
7	B	-7.6000	0.0021
B	8	1.6900	0.0025
8	C	0.0100	0.0029
A	B	-0.6800	0.0033

STATISTICAL SUMMARY

Total Unknown Elevations:11
Total Elev. Routes:15
Total Fixed BM's:4
Total non-fixed BM's:0
Degrees of freedom:4

ADJUSTED ELEVATIONS

Station	Adjusted Elev	Standard Dev.	Error Ellipse at 95% CI
BMY	107.5000	FIXED	0.00000
BMX	100.0000	FIXED	0.00000
1	99.3094	0.02592	0.09659
2	99.0283	0.03777	0.14074
A	105.1494	0.03661	0.13642
3	106.8527	0.04407	0.16425
4	102.9562	0.03964	0.14772
5	108.1787	0.02456	0.09154
C	106.1985	0.02979	0.11103
6	111.3667	0.03114	0.11604

7	112.0948	0.02486	0.09263
B	104.4875	0.03118	0.11618
8	106.1827	0.03635	0.13547

ADJUSTED MEASUREMENT SUMMARY

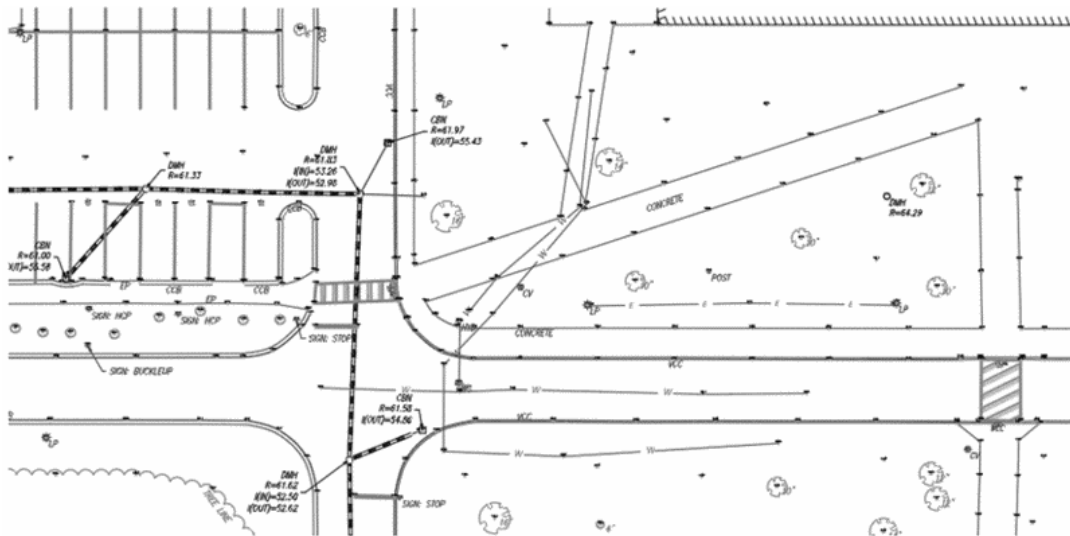
From	To	Elev. Diff. (adjusted)	Residuals	Std. Dev.
BMX	1	-0.6906	0.0094	0.026
1	2	-0.2811	0.0189	0.034
2	A	6.1211	0.0211	0.035
A	3	1.7033	0.0033	0.038
3	4	-3.8965	0.0035	0.039
4	BMY	4.5438	0.0038	0.040
BMY	5	0.6787	-0.0213	0.025
5	C	-1.9801	-0.0301	0.027
C	6	5.1682	-0.0318	0.030
6	BMX	-11.3667	-0.0367	0.031
BMX	7	4.5948	-0.0052	0.025
7	B	-7.6073	-0.0073	0.028
B	8	1.6951	0.0051	0.032
8	C	0.0159	0.0059	0.033
A	B	-0.6619	0.0181	0.034

Vertical Sideshots

Station	Elevation
SS1	96.509
SS2	95.509

Draw Field to Finish

This command turns data collector field notes into a final drawing by matching the descriptions of the field points with user-defined codes. The points are brought into the drawing with attributes defined by the code, including the layer, symbol, size and linetype. Essentially, Field to Finish drafts a plan as you survey.



Example of a survey processed with Draw Field to Finish

Draw Field to Finish	Create/Edit a Field Code	See Some Examples
Create a Code/Point List	Use The Tree Feature	
Work with Code Tables	Use the Pipe Feature	
Use Special Codes	Use the Utility Feature	

Two main files are used in Draw Field to Finish - a coordinate file (**CRD/CRDB**) and a field code definition file (**FLD**). The coordinate file consists of point#, N,E,Z coordinates and text description fields. The description field contains the codes for the Draw Field to Finish processing. An ASCII data file can be converted into a coordinate file using the *Import Text/ASCII File* command. The field code definition file defines the layer, symbol, size and other actions to apply with each code. These file names are displayed at the top line of the Draw Field to Finish dialog box.

Field to Finish is an extremely versatile and robust program. The concept however is straight forward. Each point located in the field is coded with a description. That point description is related to a matching code in a Code Table (**FLD**) that instructs the Field to Finish program on how to process and draw it. Points can be placed in a drawing using specific symbols, on particular layers and annotated with text or block attributes. A series of points can be used to draw 2D and/or 3D polylines for use in drafting or creating surface models. Beyond a large variety of settings for each defined code in the code table, there are also Special Codes that modify the behavior of the code, such as PC (point of curvature) and PT (point of tangent).

Field to Finish is a very powerful tool but it is not required to place points from a coordinate file into a drawing. *Draw-Locate Points* provides a simpler method for drawing points but it is much less versatile.

Using a drawing template: Field-to-Finish will "layerize" the points and linework according to the code definitions. If the layers to use are not already defined in the drawing, Field-to-Finish will create any layers defined in the code table when it processes that code. It will however, randomly assign a color, linetype etc. to that layer. To provide consistency and have the same properties for these layers in all your drawings, it is recommended you use a prototype drawing, or Template (DWT). Each new drawing created can then be started using this template which contains all layers needed matching those in the Code Table (FLD) file. See Carlson Configure -> Startup Settings to select a prototype Template.

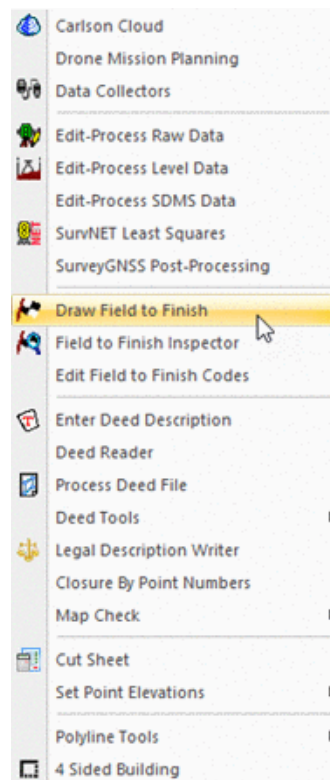
Connecting Line Work: If a code is created as a Line, 2D Polyline or 3D Polyline, all codes with that same description will be connected with no extra coding needed. For example if the code EP is defined as a 3D Polyline and points 100-110 all have EP as a description, Field to Finish will draw a 3d Polyline connecting all points 100 through 110. Stopping and starting line work is where additional coding is needed. There are basically two different methods for connecting linework and controlling the start/stop or gaps between line segments.

1. **Adding group numbers:** As described above the linetype is defined by the code as either lines, 2D polylines, 3D polylines (breaklines), or both 2D and 3D polylines. To control the connection of the linework, distinct lines with the same code are defined by adding a group number to the end of the code name in the data file. With this method, all points with the description CODE1 will be one line while points with CODE2 will be another line. Both CODE1 and CODE2 use the definition for CODE. For example, the code EP could be a code for edge of pavement that is to be connected as 3D polylines. If there are two separate edge of pavement lines on the left and right sides of a road, all the points for the left side could have the description EP1 and the points on the right side could be EP2. Besides having the number after the code, the number can be used as a prefix by defining the code with a # special character. For example, when the code is defined as #CODE, then the points with descriptions 10CODE and 20CODE get matched to this code.
2. **Adding Special Codes:** The second method is to add a special code to the end of a line segment or the beginning of a new line segment. For this method, you use the same code for all similar line segments and

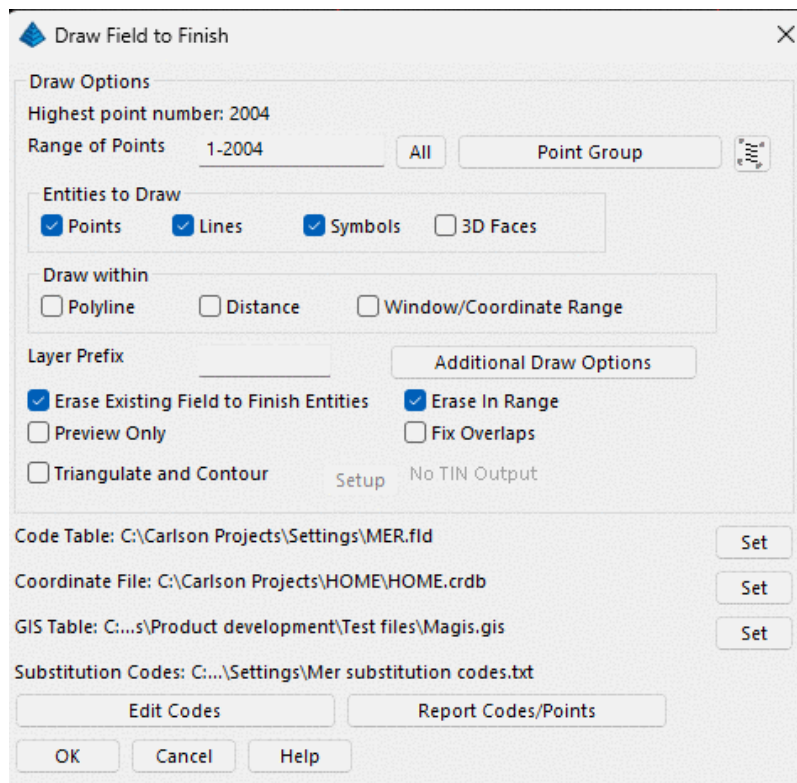
just instruct Field to Finish when to stop drawing. For example the end of an edge of pavement line would be coded EP END. (The code END is the default end line special code). Alternatively, you can instruct Field to Finish to start a new line segment by adding a beginning special code IE. EP BEG. The program will realize that a new line segment should not be connected to a previous one and will create an appropriate gap.

Legacy PointCAD format: This method also connects points with the same code. The difference is that instead of using a number after the code for distinct lines, you use the same code with an additional code for starting and ending the line. For example, +0 is used to start a line and -0 to end. So the coding for a segment of edge of pavement could be EP+0, EP, EP, EP-0. Another special code that has been added to Field to Finish is +7, -7. This 7 code will use the linetype definition of line, 2D polyline or 3D polyline defined by the Draw Field to Finish code. For example, if EP is defined as a 3D polyline, then the coding EP+7, EP, EP, EP-7 will create a 3D polyline. Otherwise codes like +0, -0, which is defined as start and end line, will draw EP as a line. Other PointCAD special codes are: +4 starts a curved 2D polyline, *4 starts a closed curved 2D polyline, +1 begins a 3-point arc, +5 starts a 3D polyline, *5 starts a closed 3D polyline, +6 starts a 2D polyline, *6 starts a closed 2D polyline, +7 starts a line whose type is specified by the field code definition, -05 starts a curved 3D polyline section, -50 ends that section, +8 starts a 2D and 3D polyline combination, *8 starts a closed 2D and 3D polyline combination, -08 starts a 2D and 3D polyline combination curved section, -80 ends that section. //, followed by a field code, concatenates that field code's description on to the point's description. For example, OAK//04 might become LIVE OAK TREE 4" if the field code OAK translates to LIVE OAK TREE and the field code 04 translates to 4".

Draw Field to Finish is found in the **Survey** Pull-down Menu



Draw Field to Finish



Range of Points: Specify the range of points to draw.

Point Group: Specify the point group(s) to process.

Point List: Select points from a point list dialog box.

Entities To Draw: The Points option draws only the points and point attributes. The Lines option draws only the linework and the Symbols draws only the symbols. Any combination of these options can be processed as well as individual processing of each entity.

Draw Within: These options are methods to filter the points to draw. The Polyline method prompts for a closed polyline and only draws points inside this polyline. The Distance method uses a specified center point and distance to only draw points within this circle. The Window/Coordinate Range prompts for lower left and upper right points to define the rectangular area to draw points.

Layer Prefix: Optional layer prefix added to all entities drawn with Draw Field to Finish.

Erase Existing Draw Field to Finish Entities: When checked, this option will erase from the drawing any old entities created by previous Field-To-Finish runs before drawing the new entities.

Erase In Range: This option only erases and redraws those Draw Field to Finish entities that are within the specified range of points to process.

Preview Only: When checked, this option will temporarily draw the points and linework and allow you to review it with zoom and pan.

Fix Overlaps: This option checks the points drawn by Field-to-Finish for any point labels that overlap with other points or linework. For any overlaps, the point labels can be automatically moved or you can step through each overlap to decide how to handle it. See the command called Fix Point Attribute Overlaps in the Points chapter for more information on this feature.

Triangulate and Contour: This option uses the points and 3D polylines created by Field-to-Finish as input for

building a surface in Triangulate and Contour. Pick the Setup button for the options like contour interval and TIN file to create.

Code Table: Sets the FLD file to process which contains the code definitions.

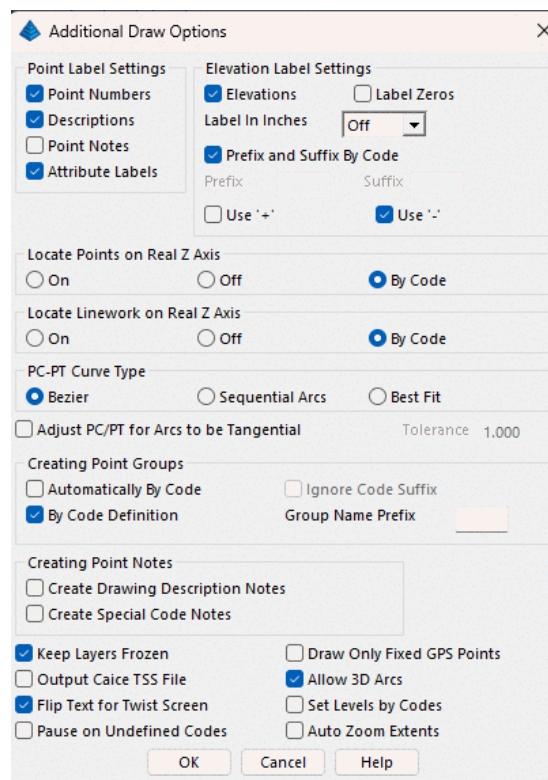
Coordinate File: Sets the CRD file to process which contains the point numbers, coordinates and descriptions.

GIS Table: Sets the GIS file which defines the GIS feature and attribute names. This file is optional and is used in GIS processing.

Edit Codes: Opens the **Field to Finish** Edit Code dialog box.

Report Codes: Opens the Report Code dialog box.

Additional Draw Options:



Point Label Settings: Specify whether you want Draw Field to Finish to label the **Point Numbers**, **Descriptions**, or **Points Notes** which are contained in the note (.NOT) file that is associated with the coordinate (.CRD) file. The **Attribute Labels** option controls whether or not to draw attribute labels. These attributes can be the description and/or additional comments in the form of block attributes or text and may or may not be associated with a GIS feature.

Elevation Label Settings: Specify the elevation labeling options. The Elevations toggle controls whether to draw the elevation attribute label for the points. The Label Zeros option will label the elevations of points with z=0. The Prefix and Suffix By Code sets whether to use the general Prefix and Suffix for all point elevation labels from this dialog or use the Prefix and Suffix defined in each code definition. Use '+' and Use '-' will place the appropriate symbol in front of the elevation. The Label In Inches option allows for elevation label in feet and inches instead of in decimals.

Locate Points on Real Z Axis: Choose between locating all the points at real Z elevation, all at zero elevation or to use the real Z setting as defined in the individual codes (**By Code**). These options can be used to override the setting contained in the FLD as part of the code and is global in nature. It is usually recommended to use the By Code option.

Locate Linework on Real Z Axis: Choose between drawing all the linework at real Z elevation, all at zero elevation or to use the 2D/3D polyline setting as defined in the individual codes (**By Code**). These options can be used to override the setting contained in the FLD as part of the code and is global in nature. It is usually recommended to use the By Code option.

PC-PT Curve Type: Sets the method for drawing curves with more than 3 points. The **Bezier** option draws a smooth polyline through all the curve points. The **Sequential Arcs** method draws multiple arcs with arc end points at each of the curve points. These arcs are tangent to the preceding line segment. The **Best Fit** method creates a single best-fit curve for all the curve points between the PC and PT.

Adjust PC/PT for Arcs to be Tangential: This option will adjust the PC and PT polyline vertices to make the curve tangential. The program will only adjust these points if the adjustment distance is less than the specified tolerance. This option applies to cases where the tangents are well defined and the PC/PT are harder to survey exactly.

Creating Point Groups: Point Groups can be created in one or two different ways. Each field code definition can specify Point Group(s) that all point numbers that use that code will be added to. Multiple field codes can use the same Point Group name. Check the By Code Definition checkbox for that option. The second method is to automatically create Point Groups for each code that is processed. Check the Automatically By Code checkbox for that option. Ignore Code Suffix, if checked, will cause the codes to be considered after removing the numeric suffix. For example, points with the EP10 and EP11 codes will both be automatically added to the Point Group named EP. No matter how the Point Group is created, the Group Name Prefix can be used to add a prefix to the group name. Note: if the Point Group already exists, it will be erased first before being created again by either of these two methods.

Creating Point Notes: These options append point notes to the coordinate file data for some of the data fields processed by Field-to-Finish. These notes can then be used by other commands like List Points to report these fields. For example, this enables List Points to report both the point coordinate file description as well as the point drawing description as generated by Field-to-Finish.

Keep Layers Frozen: This option will not thaw layers that Field-to-Finish uses for drawing entities. So when you have this option on and some layers are frozen, then you won't see the entities that Field-to-Finish creates until you thaw those layers.

Draw Only Fixed GPS Points: This option applies to coordinate files created by GPS measurements. The program checks for GPS status records and only uses points that have GPS solution status of Fixed.

Output Caice TSS File: This option creates a TSS file from the processing results.

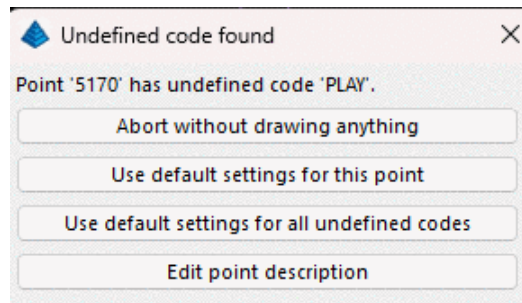
Allow 3D Arcs: When creating polyline arcs with the PC special code, this option checks whether to create the arc in 3D for tilted or vertical arcs such as a doorway or bridge arch. Otherwise, only the 2D coordinates are used to define the curvature.

Flip Text for Twist Screen: This option will rotate the point labels and symbol by 180 degrees when needed to make them right-side up readable relative to the current twist screen drawing view. This option applies to the Rotate To Line and Rotate special code (ROT).

Set Levels By Codes: This option assigns the optional Level Names for use by selection filters.

Auto Zoom Extents: When checked, this will force a zoom extents after Draw Field to Finish is done.

Pause on Undefined Codes: When checked, Draw Field to Finish will pause if it encounters a description that is not defined in the code table.

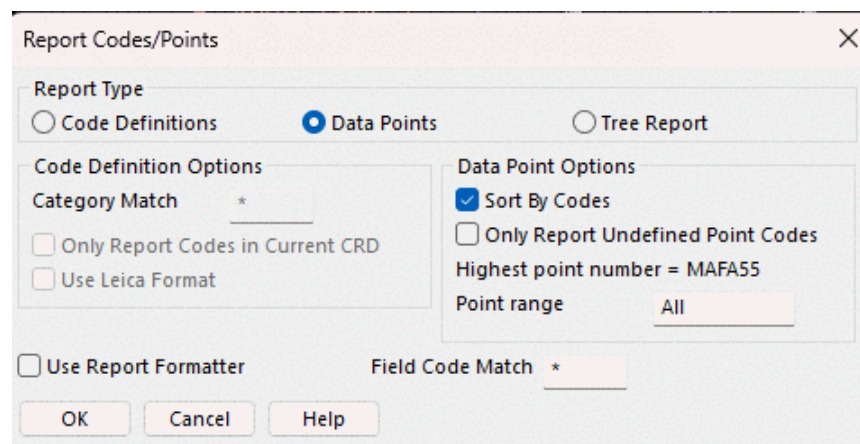


The options for handling undefended codes are:

- **Abort without drawing anything:** This stops the command. Run Draw Field to Finish again to correct the code table.
- **Use the default settings for this point:** This option draws a point using the current Point Default settings with no linework. To set your own default, define a code called "SC_DFLT" or "DEFAULT." This default code will instruct Field to Finish on how to process undefined codes.
- **Use default settings for all undefined codes:** This option will draw all undefined codes using the current Point Default settings with no linework. To set your own default, define a code called "SC_DFLT" or "DEFAULT." This default code will instruct Field to Finish on how to process undefined codes.
- **Edit Point Description:** Use this option to edit the point description or select a different code.

Report Codes/Points

This routine prints the code table or the data file to the screen, file, or printer. A useful option here is to print the data file (CRD Points) and choose Sort by Codes which will group the data points by distinct codes.



A useful option here is to print the data file (CRD Points) and choose Sort by Codes which will group the data points by distinct codes.

Report Type:

- Code Definitions: Creates a list of code definitions from the currently selected FLD file.
- Data Points: Creates a list of points grouped by codes of the current CRD (coordinate file)
- Tree Report: Creates a list of all points coded as a Tree Feature.

Code Definition Options

- Only Report Codes in Current CRD: Reports only the codes from the current FLD that are contained in the current CRD (coordinate file)
- Use Leica Format: Creates a list of code definitions using the Leica comma delineated format.

Data Point Options

- Sort by Codes: Sorts the list of points by code name
- Only Report Undefined Codes: Restricts the list to report only those points that have descriptions not contained in the code table. This is useful for reviewing a data set before processing.
- Point Range: Enter the range of points to be included in the list.

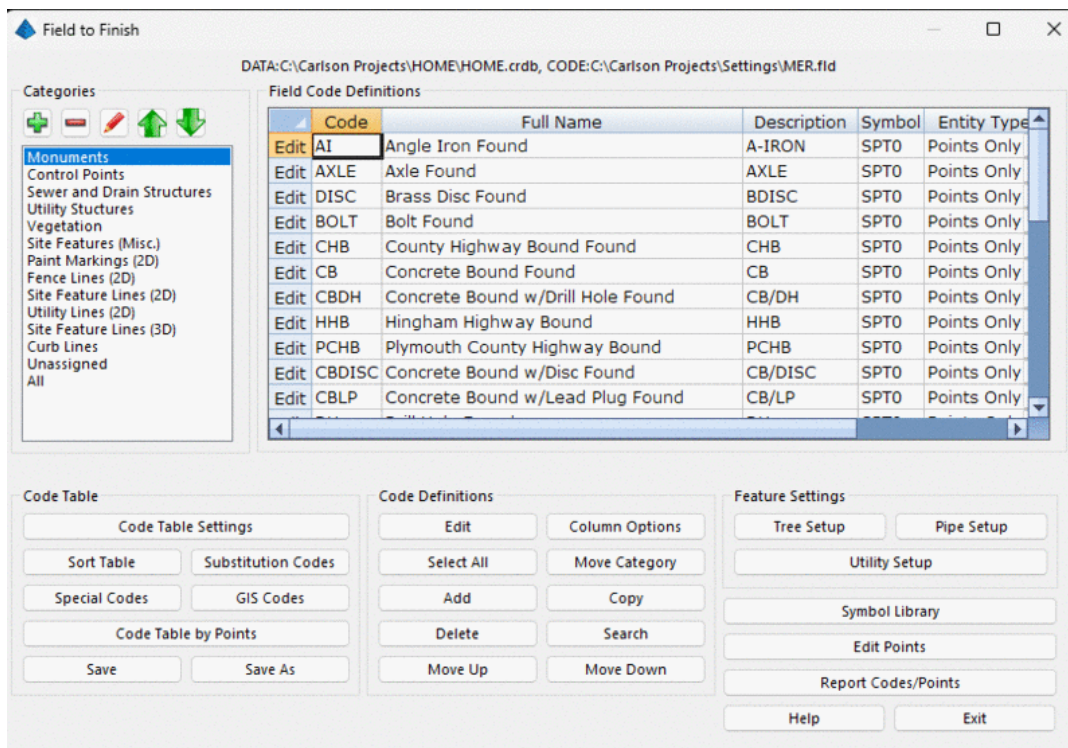
Use Report Formatter: Enables the option to use the Carlson Report Formatter for more control of the creation of the point list.

Field Code Match: Option to filter and limit the list to only select codes.

Edit Codes

Selecting Edit Codes takes you to the Field to Finish Code Table dialog box. This dialog box contains access to all of the tools and options for Field to Finish. It is the heart of this program.

The **Field to Finish** dialog box allows you to load the coordinate and field code definition file. Here you can add, view and edit the code definitions, view and edit the coordinate file, create reports, and then return to the Draw Field to Finish dialog box to process the files. The top section displays the code definitions. The bottom section has three columns of functions each pertaining to controls for different elements of the command. The **Code Table** section provides controls for settings, sorting and reporting of codes. The **Code Definitions** section provides tools for the creation and editing of codes. The **Feature Settings** section provides controls for the special tree, utility and pipe feature types.



The code table editor has a list of categories and a spreadsheet of codes with optional displays of related definitions. The spreadsheet shows the codes for the currently highlighted category. The category toolbar buttons allow you to add, remove, edit the names and change the order of the categories. There are two fixed categories. The **Unassigned** category shows any codes with blank categories. The **All** category shows all the codes. You can control which fields are visible in the spreadsheet by using the **Column Options** button. You can make edits to the fields in the spreadsheet or highlight a row and pick the **Edit** button to bring up a dialog to edit the code.

This section of the manual is separated into three categories:

Code Table

Code Definitions

Feature Settings **Quick Links for this section:**

Code Table Settings

Sort Table

Substitution Codes

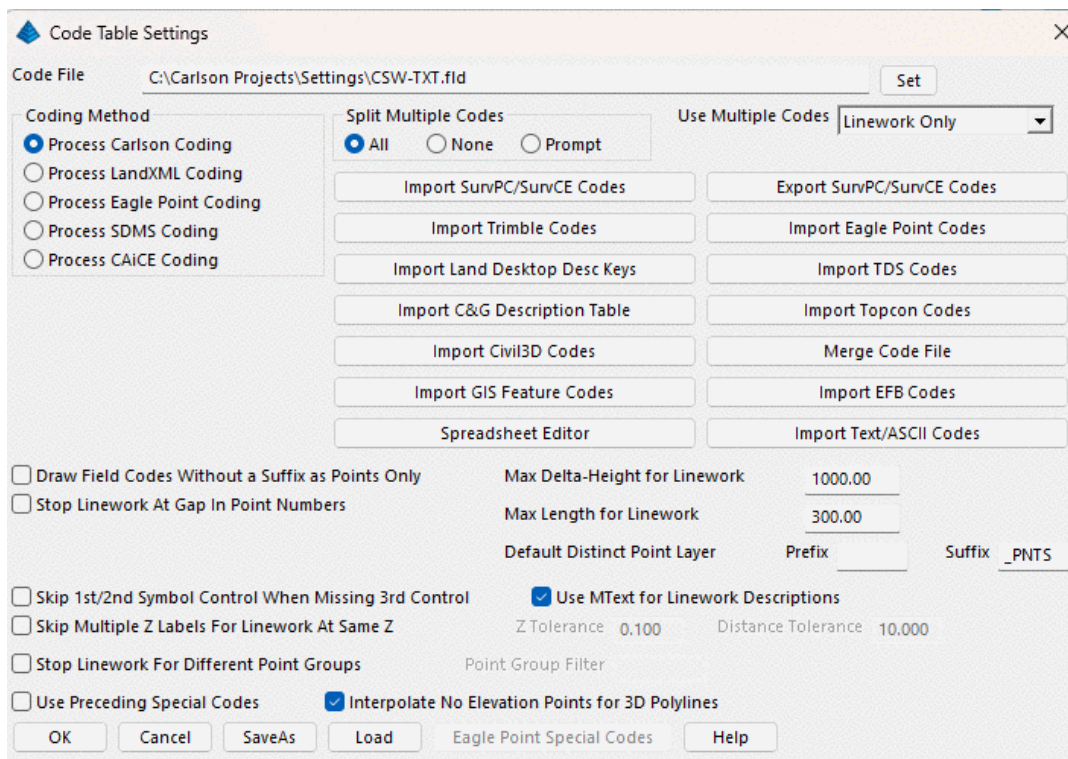
Special Codes

GIS Special Codes

Code Table by Points

Code Table Settings:

These options provide tools for defining the coding method to be used for processing of the point data. Various import tools allow for the importing of codes from different software packages. Controls for handling multiple codes are located on this dialog. All special codes can be replaced to other characters defined by the user. The special codes are listed and edited on this dialog.



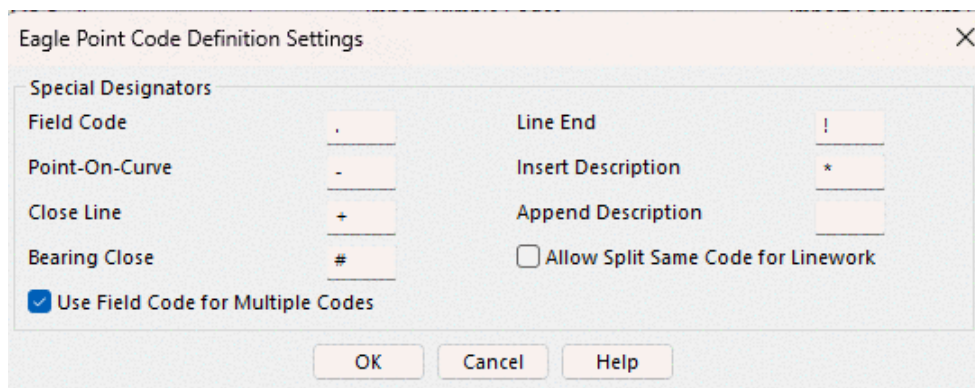
Set: Choose this button to select an existing or create a new code table. The name of the current table is shown in the field to the left of this button.

Coding Method:

Carlson Coding: When checked, this option interprets and processes coordinate files based upon the Carlson Coding method and data collection method.

LandXML Coding: This method prompts for a LandXML or Leica/Hexagon HeXML file to use with the processing. This method applies when the point descriptions do not have linework coding and the LandXML file has PlanFeatures that define how points are connected to create linework. This method reads the LandXML file to find the points that begin and end lines and curves.

Eagle Point Coding: When checked, coordinate files are processed based on the Eagle Point Data Collection method. When selected the *Eagle Point Codes* button at the bottom of the dialog box becomes available for selection and displays the following dialog. This dialog allows for customization of the eagle point special designators.



Currently the supported designators include, "Field Code", "Point-On-Curve", "Close Line", "Bearing Close", "Line End", "Insert Description" and "Append Description". Also supported is the ability to recognize overwriting of

descriptions just as Eagle Point does by using the space separator instead of the "Insert Description" designator. Examples of supported coding are as follows:

.TC Places a node and or line per the field code library.

TC Places a node and or line per the field code library.

-TC Specifies a point on a curve.

TC- Specifies a point on a curve.

..TC Stops the line.

TC! Stops the line.

.TC+ Closes the line back to the starting point.

TC+ Closes the line back to the starting point.

.TC# Typically coded on the third corner of a rectangle to close the figure with having to locate the fourth corner.

TC# Typically coded on the third corner of a rectangle to close the figure with having to locate the fourth corner.

WV.WI Places a node as specified by the code "WV" in the field code library and then begins a line as specified by code "W" in the field code library.

.TC.EP.FL Results in three lines coming together.

TC1.TC2.TC3 Results in three lines coming together. All three lines are specified by the definition of the single code "TC" in the field code library.

TC.TC1 When used in conjunction with the "Draw Field Codes Without a Suffix as Points Only" toggle, "TC" will be recognized as the node and "TC1" will be recognized as the line so that if the code "TC" in the field code library is defined as a polyline, line or 3D polyline, duplicate lines will not be unintentionally placed when this shot only pertains to a single element. Keep in mind that all line work must have a numeric suffix when using this toggle.

TREE * OAK Result on screen would be: TREE OAK

TREE OAK * Result on screen would be: OAK TREE

TREE OAK Result on screen would be: OAK

TC1!.TC2-.VLT6# Stops "TC1", continues "TC2" as a point on a curve and closes VLT6 as a rectangle using the "Bearing Close" code.

Note: The use of the "Use Multiple Codes for Linework Only" toggle is recommended when using Eagle Point Coding.

SDMS Coding: This option processes coordinate files based upon SDMS coding method. When active, the program will prompt for an SDMS .PRJ file to process.

CAiCE Coding: When checked, coordinate files are processed based on the CAiCE Data Collection method. Examples of supported coding are as follows:

169 is just the code 169.

145C10 is the code 145 and line #10.

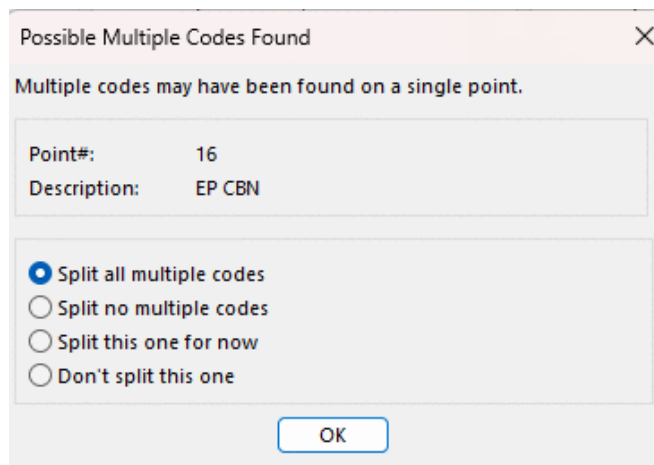
169C25C is the code 169, line #25, and the point is on a curve.

172C12B is the code 172, line #12, and this point closes the line.

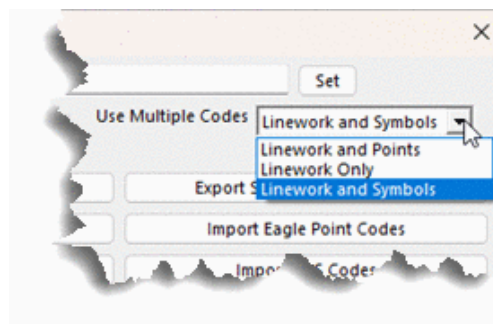
Split Multiple Codes:

Multiple codes are defined by including each code in the point description field separated by a space. A single data point can be used in different lines by assigning it multiple codes. For instance, a point might be part of both a curb line and a driveway line with a description of "CURB DRW". Field-to-Finish uses spaces as the delimiter for multiple codes. You should avoid spaces in the descriptions except for where multiple codes are intended or after the "/" character. For example, a code for light post should not be "LGT POST" but instead should be "LGTPOST".

There are three options for the handling of multiple codes when encountered. The **All** option will split all multiple codes and process each code based upon its code definition. When **None** is select the first code in the description string will be processed and the remaining characters left a part of the description (this is exclusive of any Special Codes used). If the **Prompt** option is checked on, when Field-to-Finish detects multiple codes on a point the following dialog will be displayed with options for handling the codes.



Use Multiple Codes:



Multiple Codes are simply more than one code separated by a space (default setting). Field to Finish has options to allow more control over the use of these multiple codes. This setting has three options:

- Linework and Points - Uses the space to connect multiple line codes as well as creating multiple point blocks at each location.
- Linework Only - Limits the multiple coding to be used for connecting line work only.
- Linework and Symbols - Uses the space to connect multiple line codes as well as creating multiple symbols at each location.

Import/Export Options:

Import SurvCE Codes: This option imports a SurvCE Feature Code List (fcl) into a Carlson Field to Finish (fld) code definition file.

Import Trimble Codes: This option imports Trimble .FXL file codes into the Carlson Field to Finish (fld) code definition file.

Import Land Desktop Desc Key: This option imports and converts a Land Desktop Description Key into a Carlson Draw Field to Finish (fld) code definition file. The Land Desktop Description Key file is a mdb file and is found in the Land Desktop Project file path. It is located in the under the COGO/DescKey directory.

Import C&G Description Table: This option imports C&G code tables (tbl) into the Carlson Field to Finish (fld) code definition file.

Import Civil 3D Codes: This option imports Civil 3D .fdb_xdef file codes into the Carlson Field to Finish (fld) code definition file.

Import GIS Feature Codes: This option imports features in a .GIS file from Define GIS Features into F2F codes.

Export SurvCE Codes: This option creates a SurvCE Feature Code List (fcl) from the current a Carlson Field to Finish (fld) code definition file.**Import TDS Codes:** This option imports TDS codes into the Carlson Field to Finish (fld) code definition file.

Import TDS Codes: This option imports TDS codes into the FLD code definition file.

Import Eagle Point Codes: This option imports Eagle Point codes into the Carlson Field to Finish (fld) code definition file.

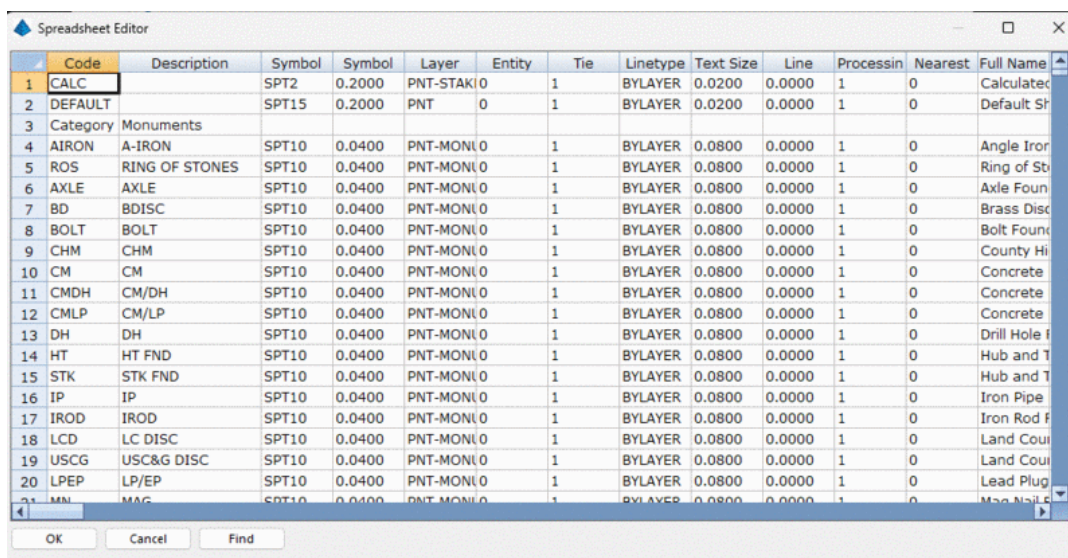
Import Topcon Codes: This function imports Topcon codes and layers from an XML file.

Merge Code File: This function adds code definitions from another (fld) code definition file into the current code table. The program shows a list of the codes from the other file that are different than the current code table. You can choose which codes to import. Any codes that conflict with an existing code definition are defaulted to not import.

Import EFB: This option imports the SDMS Electronic Field Book codes (.xml) into the Carlson Field to Finish (fld) code definition file.

Import Text/ASCII Codes: This option imports code definitions from a user-defined format. Each row in the text file should represent one code. The program will prompt for the delimiter (ie. comma separated) that is used in the text file and then for the field type for each of the columns (ie. "Layer" or "Description").

Spreadsheet Editor: This function lets you edit all the codes with all their settings in a single large spreadsheet. This method can be a handy way to use copy and paste to edit the codes. This method can also be quicker to make many changes than navigating the Edit Code dialog. This method is for expert users and you need to take care to make valid inputs.



Code	Description	Symbol	Symbol	Layer	Entity	Tie	Linetype	Text Size	Line	Processin	Nearest	Full Name	
1	CALC	SPT2	0.2000	PNT-STAK	0	1	BYLAYER	0.0200	0.0000	1	0	Calculated	
2	DEFAULT	SPT15	0.2000	PNT	0	1	BYLAYER	0.0200	0.0000	1	0	Default Sh	
3	Category	Monuments											
4	AIRON	A-IRON	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Angle Iron
5	ROS	RING OF STONES	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Ring of St
6	AXLE	AXLE	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Axle Foun
7	BD	BDISC	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Brass Disc
8	BOLT	BOLT	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Bolt Foun
9	CHM	CHM	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	County Hi
10	CM	CM	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Concrete
11	CMDH	CM/DH	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Concrete
12	CMLP	CM/LP	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Concrete
13	DH	DH	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Drill Hole f
14	HT	HT FND	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Hub and T
15	STK	STK FND	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Hub and T
16	IP	IP	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Iron Pipe
17	IROD	IROD	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Iron Rod f
18	LCD	LC DISC	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Land Cou
19	USCG	USC&G DISC	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Land Cou
20	LPEP	LP/EP	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Lead Plug
21	MN	MAG	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Mag Nail f

Additional Options:

Draw Field Codes Without a Suffix as Points Only: This option is useful for when wanting to use a field code sometimes for linework and sometimes for just points but it is preferred to number the lines rather than using start

and stop codes. For example, if the field code EP is defined to use the Line Entity type, then EP25 will be drawn as a Line, however if just EP is used, no linework will connect to that COGO point. When using this option, it is required to add a numerical value after each line code.

Stop Linework At Gap In Point Numbers: This option is a method for controlling the start and stop of drawing linework. This method will automatically stop linework where there is a gap in the point numbers for the linework code. For example, if there are points with code EP then points with code CL then more points with code EP, the EP linework for the first set of EP points will stop at the last EP before the CL points and then new EP linework will start after the CL points.**Max Delta-Height for Linework:** Use this option to specify the maximum elevation difference that Draw Field to Finish should draw any section of linework. This option is for use with 3d polylines and lines.

Max Length for Linework: Specify the maximum length that Draw Field to Finish should draw any section of linework. If two codes are distant beyond this limit, no linework will be drawn between them.

Default Distinct Point Layer: These settings control the default layer for when the Distinct Point Layer option is turned on for each code definition.

Skip 1st/2nd Symbol Control When Missing 3rd Control: For multi-point symbols when the 3rd control point is missing, this option centers the symbol on the 1st point instead of using the 1st point as a control point.

Use MText for Linework Descriptions: Controls whether to create MText or regular Text entities for linework description labels.

Skip Multiple Z Labels For Linework At Same Z: This option labels only the first point elevation for points on polyline or rectangle from the RECT code when the elevation difference between the points is less than the specified **Z Tolerance** and the horizontal distance is less than the **Distance Tolerance**. For example, for points on the corners of a level utility pad, this option will label the pad elevation just once.

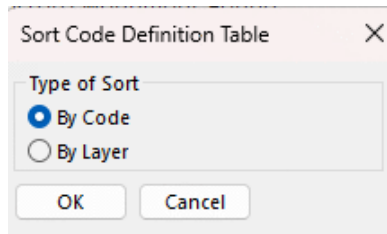
Stop Linework For Different Point Groups: This option applies to when you have points for the same job collected at different times or with different crews and you want to prevent linework connecting points between them. For example, this applies when you survey a job with two crews and combine their points into a single CRD and both crews used "EP1". In this case, to keep each set of "EP1" separate, you can put each set of points into a separate point group with Point Group Manager and use this Field-to-Finish option. The **Point Group Filter** is optional and is used to make the program only check groups that match the filter. A point can be in multiple groups such as "CONTROL" and "DAY1". Some groups could be used for different days such as "CONTROL". So the Point Group Filter could be set to "DAY" to filter out the common groups and only process the time related groups.

Use Preceding Special Codes: This setting tells the program to expect the special codes before the main code. For example, if "BEG" is the special code for begin linework and "EP" is a main code, then the program looks for "BEG EP". The default sequence is for the special code to come after the main code as in "EP BEG".

Interpolate No Elevation Points for 3D Polyline: For points tagged as "No Elevation" that are part of a 3D polyline, this option will interpolate an elevation for this point from the other points in the 3D polyline. Otherwise, this point will be skipped in the 3D polyline.

Sort Table

This sorts the code table by either code name or layer.



Substitution Codes:

This option defines a lookup table for translations of the raw point descriptions. This translation is done as a pre-processing step before the regular Field-to-Finish processing. For example, if you had a substitution setup for "25" = "EP", then a point description of "25" would get translated to "EP" and then this would be processed with Field-to-Finish.

This works for both numerical and alpha raw descriptions.



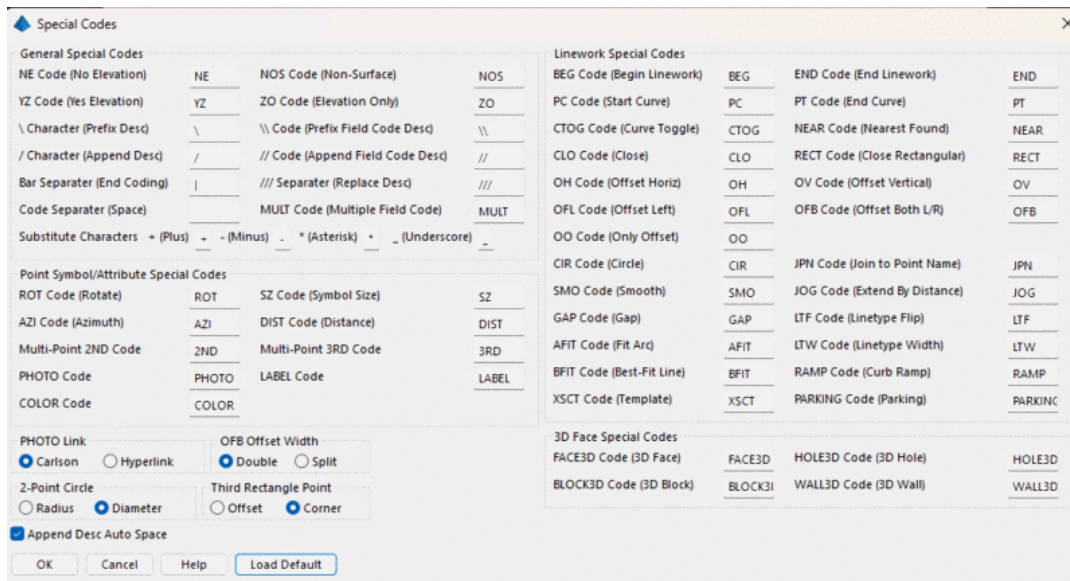
- The **Match With Number Suffix** option will match with a number following the string. For example, "CL" will match with "CL2". Otherwise, the strings have to be an exact match. The substitution codes are stored in a file named f2f_sub.dta in the %appdata%\Carlson...\USER folder. You can also use the **Import** and **Export** functions to load and save substitution codes to a comma separated text file.

Special Codes:

Special Codes are essentially modifiers of an original code. They perform additional operations and/or offer addi-

tional control over the original code. These special codes work with line work, point data and symbols. There are also options to control the placement of labels and add additional text to attributes.

Below shows the default dialog box containing the Special Codes. These can all be modified at the users discretion but the must be unique and not conflict with any codes defined in the code table.



A special code is placed after (or before if that option is enabled) the regular code with a space separating the code and special code. For example if an edge of pavement shot was taken at the beginning of a curve, it would be coded EP PC, whereas the PC tells Field to Finish to begin drawing an arc. Special codes may be strung together to modify a single code or added to multiple codes within the same string. For example, if a ling string ended at the PT of a curve it could be coded EP PT END whereas both PT and END are the special codes. An example of multiple codes combined with special codes could be a driveway intersecting the edge of pavement at the beginning of a curve: EP PC DRV SMO. The PC starts drawing an arc on the edge of pavement and the driveway begins with a smoothing code. It is important that any special code modifying another code must be placed immediately after the code it is modifying.

Quick Links for this Section

- General Special Codes
- Linework Special Codes
- Point Symbol/Attribute Special Codes
- 3D Face Special Codes

General Special Codes

NE: No Elevation - This special code represents no elevation. A point with this special code is located at zero elevation and the elevation attribute is set to None.

NOS: Non-Surface - This special code indicates that the point should be "non-surface"; that is, that it should be ignored when contouring or creating surfaces. This can also be controlled per-field code by turning on the Non-Surface toggle in the Edit Field Code Definition dialog box. When used, this special code tags the point to be non-surface but maintains the elevation attribute.

YZ: Yes Elevation: - This special code locates overrides the code setting for non-surface and locates the point and

maintains the elevation attribute.

ZO: Elevation Only - This special code represents elevation only (Z-Only). A point with this special code is used as part of a 3D polyline for elevating the 3D polyline without effecting the horizontal position of the polyline. For example, this code could be used on a grade break point along a curb where only the elevation should change and not the horizontal alignment.

"\": Prefix Description - This special code takes the part of the description after the "\" and puts it as the prefix before the point description. For example, a data point with the description "TR \ 24ft" and a "TR" code definition with a description of "Tree" would be drawn with a description of "24ft Tree".

"\\": Prefix Field Code Description - This special code causes text after the "\\" to be interpreted as a field code. That field code's description is then prefixed to the first field code's description. For example, if the field code 02 has the description 2" and the field code OAK has the description *oak tree*, then OAK\\02 will result in the point having the description of 2" *oak tree*.

"/": Append Description - A special character "/" (the forward slash or divide key) can be used for an unprocessed description to append. Everything after the "/" is added directly to the point description and is not considered a code and no further substitution is done on it. For example, a data point with the description "UP/#150" for a code UP that has UP for its description would be drawn "UP #150".

"/": Append Field Code Description - This special code causes text after the "/" to be interpreted as a field code. That field code's description is then appended to the first field code's description. For example, if the field code 02 has the description 2" and the field code OAK has the description *oak tree*, then 02//OAK will result in the point having the description of 2" *oak tree*. If the "/" character has been replaced with a different character, for example with a & character, then the "/" code would become "&&".

"—": End Coding - The bar separator indicates the end of coding. Everything after the bar is ignored for Field-to-Finish processing. This is useful if additional information is needed for a point but no action should be taken and no description should be added.

"/": Replace Description - This special code takes the part of the description after the "/" and uses it as the point description label.

" ": Code Separator - The default for separating multiple codes is a space. This can be modified to another character if desired. Caution should be taken when modifying special codes to be sure there are no conflicts with other codes or special codes.

MULT: Multiple Field Code - This code applies when the Split Multiple Codes under Code Table Settings is set to None and you want to override this setting and explicitly split selected codes. Multiple codes apply to points with dual code definitions for drawing two different style points or for connecting different linework to the same point. For example, if a point is both a sidewalk and driveway corner, then the point description could be "SW MULTDR".

Substitute Characters - The characters (*, -, +, /, and _) can be used and substituted in Draw Field to Finish. The way these characters are used is that when the file is processed the description field is searched for these characters. If the "+" symbol was changed to "-" then the program would look for "-" and change it to "+". This is useful when a particular data collector may not have all the symbols available. With these substitutions you can make a character that is provided on the data collector generate the symbol needed. Multiple characters can also be used. For example "-" can be used to in order to produce a "/" character or any of the characters listed above.

Linework Special Codes

BEG: Begin Linework - This special code denotes the beginning of a new line string. If a code is already designated to be a line, 2D or 3D polyline, then Field to Finish will connect like codes without the need for the BEG code. However, BEG can be used to designate the start of a new line string or can be used to force linework

between codes designated as points only.

END: End Linework - This special code denotes the end of a line string. This is used when line numbers are not being used. See Connecting Line Work for description.

PC: Start Curve - This code begins a three point arc or a curved line when used with the "PT" code (see below). The point with this special code is the first point on the arc. The next point with the code is considered a point on the arc, and third point with the code is the arc endpoint. The PT code is not necessary when coding a 3 point arc.

Example of 3 point curve:

- 10, 500, 500, 0, EP PC - start curve
- 11, 525, 527, 0, EP - second point on curve
- 12, 531, 533, 0, EP - end point of curve

For curves beyond 3 points, the **PT** code needs to be used for the last point on the curve. The routine then looks back for the PC and "fits" a curve between all intermediate points.

PT: End Curve - This code can be used with "PC" to define a curve with more than three points or a tangent two-point curve. Starting at the point with the "PC", the program will look for a "PT". If the "PT" is found, all the points between the "PC" and "PT" are used for the curve which is drawn as a smoothed polyline that passes through all points and only curves the polyline between points. If no "PT" is found, then the regular three point arc is applied as explained above. If no points are found between the "PC" and "PT", then the point prior to the "PC" and the point after the "PT" are used to create tangents for the resulting curve.

Both the PC and PT codes used for drawing arcs are subject to the PC-PT Curve Type found in the Additional Draw Options.

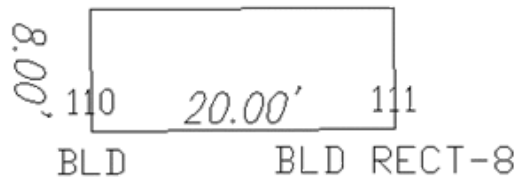
CTOG: Curve Toggle - This special code toggles curve mode on and off. Instead of using PC to start a curve, you can use CTOG. Likewise, instead of using PT to end a curve, you can use CTOG.

NEAR: Nearest Found - This special code sets the current polyline to Nearest Found connection order. This applies to codes that have the Connection Order set to Sequential and you want to override this setting to Nearest Found for the current polyline. This is useful in situations when a point on a polyline was taken out of sequential order.

CLO: Close - This code forces the lines drawn between a series of points with the same code to close back to the first point with the same code. For example, shots 1-4 all have the BLD description with the exception of point 4. Its description is BLD CLO. This will force the linework drawn for the BLD code to close back to point 1 which is the first point with the description of BLD.

RECT: Close Rectangular - The "RECT" special code creates a rectangle as a 2D or 3D polyline using one of two different methods.

- *2 Point Method*: Locate 2 points for one side of a structure and add the code RECT plus a dimension for the remaining dimension of the rectangle. For example, you are locating a shed with the dimensions 20x8. Take a shot at each corner on the 20' side and code the second shot with the 8' dimension. A positive dimension offsets to the right and a negative dimension offsets to the left. First code: **BLD** Second Code **BLD RECT-8**



- 3 Point Method: Locate 3 corners of the structure and code the 3rd shot as RECT. This tells Field to Finish to form a parallelogram out of the 3 shots taken by copying the 2 sides located respectively.



Offsets: OH, OV, OFL, OFB, OO

The codes "OH" and "OV" stand for offset horizontal and offset vertical. These offset codes apply to 2D and 3D polylines. A single set of offset codes can be used to offset the polyline a set amount. For example,

10, 500, 500, 100, EP OH2.5 OV-.5

11, 525, 527, 101, EP

12, 531, 533, 103, EP

This would create a polyline connecting points 10,11 and 12 and an offset polyline with a 2.5 horizontal and -0.5 vertical offset. The direction of the horizontal offset is determined by the direction of the polyline. A positive horizontal offset goes right from the polyline direction and a negative goes left. The horizontal and vertical offset amounts apply starting at the point with the offset codes until a new offset code or the end of the polyline. Only one horizontal and vertical offset can be applied to 2D polylines. For 3D polylines, multiple offset codes can be used to make a variable offset. For example,

10, 500, 500, 100, EP OH2.5 OV-.5

11, 525, 527, 101, EP OH5.5 OV-.75

12, 531, 533, 103, EP OH7.5

This would offset the first point horizontal 2.5 and vertical -0.5, the second point horizontal 5.5 and vertical -0.75 and the third point horizontal 7.5 and vertical -0.75.

Multiple Offsets: When there are multiple "OH" codes for the same point, the polyline is offset multiple times, one for each OH code.

Alternatively and in conjunction with the "OH" code, a second number after the offset amount repeats the offset by that number. For example "OH2 5" means offset by 2 and repeat 5 times. This multiple offset option is useful drawing stair steps.

The "OFL" code stands for offset left horizontal. The only difference with the "OH" code is that you don't have to enter the "-" to go left.

The "OFB" code stands for offset both left and right horizontal. For example, if the points follow the center of a ROW, the OFB code can be used to create the left and right edges of the ROW. There is a setting in Special

Codes for Offset Both for whether the offset value is for the full or half width between the two offset lines. Also, there is a setting in each code definition under the Linetype tab for whether to create a closed polyline from the left and right offset polylines. If 3D polylines are offset, they maintain the elevations of the original located 3D polyline.

The "OO" code stands for offset only. This option only draws the offset polylines and not the polyline connecting the points. The "OO" code is a companion code that can be combined with any of the other offset codes. For example, "EP OH2 OO" will offset the EP line horizontally by 2 but draw only the offset polyline.

CIR: Circle - The "CIR" special code causes the point to create a circle in one of three different ways.

- *Single Point and Radius method:* The first way uses just the current point as the center with the CIR special code followed immediately by the radius. For example "CIR7.5" will create a circle centered on this point with radius 7.5 and at the elevation of the current point.
- *Two Point method:* The second method uses two points, the first point specifying the center and the elevation, and the second point specifying the radius. Only the first point has the "CIR" special code and the second point is the next point with a matching field code. Another variation of method 2 is using 2 points that are on the perimeter and define the diameter. For the 2 point method, whether the points define the radius or diameter is defined on the Code Table Settings > Special Codes > 2-Point Circle.
- *3 Point method:* The third method uses 3 or more points that specify the perimeter of the circle in 2D with the first point specifying the elevation. For this method, the "CIR" special code is only on the first point and the rest of the points are the next points with matching field codes.

The "CIR" code can be used with all of the linetypes including "points only". The circles are always parallel to the X-Y plane unless the code linetype is set to "3D Polyline". Then the circle is drawn as a 3D polyline. Any active linework for the code is ended before processing the "CIR" special code.

Examples of drawing circles:

PointNo. Description

Method 1 (Single point at center with radius value)

82 PP CIR7.5

Method 2a (Point at center plus point at perimeter)

83 PP CIR

84 PP

Method 2b (2 Points on perimeter that define the diameter)

83 PP CIR

84 PP

See 2-Point Circle for options to designate methods 2a or 2b.

Method 3 (Points on perimeter)

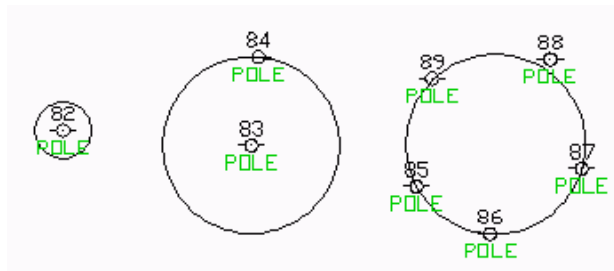
85 PP CIR

86 PP

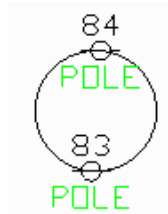
87 PP

88 PP

89 PP



Methods 1, 2a, 3



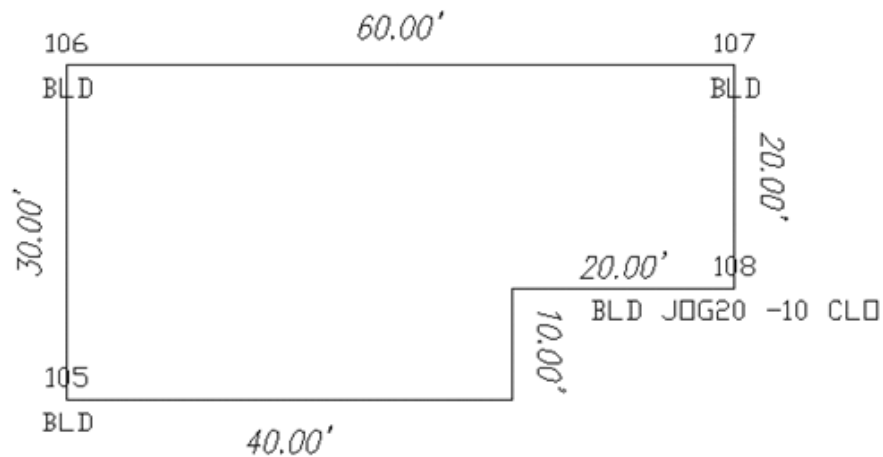
Method 2b

JPN: Join to Point Name - The "JPN" (Join to Point Name) special code joins to the point named immediately after the code. For example, "JPN205" causes a line to be drawn from the current point to the point "205". JPN is designed to work for adding a segment at the start of linework, but can be used at any point during the survey.

SMO: Smooth - This code is used to smooth the polyline. It is ideal for long winding roads or walkways that are generally curved as it prevents the need of adding PCs and PTs to define the arc segments. Tree lines and landscape areas are also good examples of utilizing this special code.

JOG: Extend By Distance - The "JOG" special code allows for additional points to be inserted into the line work at perpendicular or straight offsets. Only offsets should follow the JOG code. Positive numbers indicate a jog to the right and negative numbers indicate a jog to the left. Alternatively, "R#" and "L#" can be used where # is the distance to either the right or the left. Finally, "S#" can be used to make an offset straight ahead by using a positive # or behind by using a negative #. For example, "BLDG JOG S10.1 R5 L12.2 L5 L12.2" or equivalently "BLDG JOG S10.1 5 -12.2 -5 -12.2" advances 10.1 units and then draws a closed rectangle on the right hand side of an existing line. The offsets are always done in the X-Y plane. If the current line is vertical, an offset to the right is along the positive X-axis.

Example of building location using jog special. The command adds both a jog of 20 to the right and 10 to the left and then closes the building at point 108. **BLD JOG20 -10 CLO**



GAP: -This special code makes a single segment break in the current linework. For example, if you have a curb polyline that you want to break to skip over a driveway, then you could add the GAP code at the start of the driveway and continue the curb as normal on the other side.

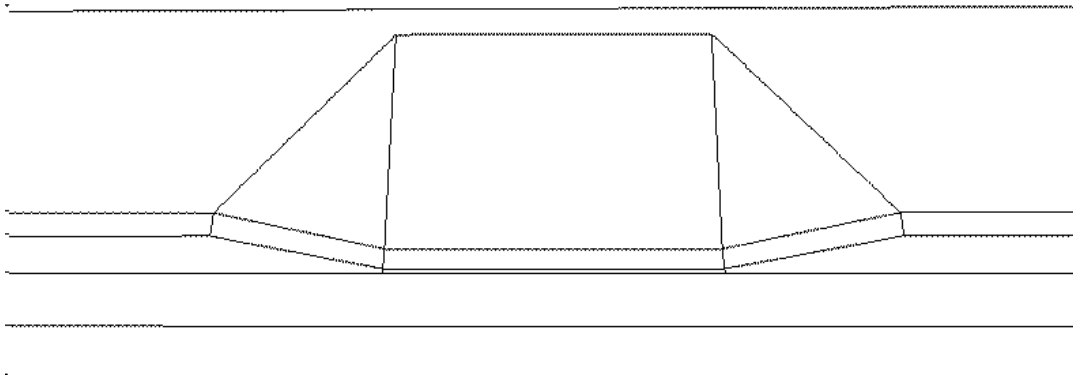
LTF: Linetype Flip - This special code switches the side for the linetype. This option applies to non-symmetrical linetypes like the tree line or guard rail for when you want the linetype to face the other way.

AFIT: Fit Arc - This special code adjusts the PC/PT points for the current arc to make the arc tangential. This special code is a way to individually control this tangential arc adjustment. To adjust all arcs within a specified adjustment tolerance, use the Adjust PC/PT setting under the Additional Draw Options from the first Field-To-Finish dialog.

LTW: Line Width - This special code sets the line width. The width value is entered after the LTW code. This width is applied to lines and 2D polylines.

BFIT: Best-Fit Line - This code creates a best-fit line using the points for the linework. This feature can be used when you have multiple points on a feature that you know is a straight line such as a sidewalk and you want a single line to be drawn. Each of the points that you want to include in the best-fit need to have the BFIT code.

RAMP: Curb Ramp - This special code adjusts the 3D polylines to create a curb ramp. The routine looks for parallel 3D polylines for the bottom of curb, top of curb and back of curb. These 3D polylines can be created in Field-To-Finish by using a Template under the Code > Linetype definition, or using the offset special codes, or by having three separate 3D polylines. The ramp is centered at the point with the RAMP code. The dimensions of the ramp follow the code in order of width, depth and taper. For example, a description of "BC RAMP 3 2 1" uses "BC" for a bottom of curb code then "RAMP" for the special code and 3 for the width, 2 for the depth and 1 for the side taper.



XSCT: Template - This special code defines a template from a series of points with this special code. This template is then applied like a template defined under Linetype for a code. This XSCT code is a way to define a template in the field instead of having a template with fixed dimensions defined in the code table. For example, you could have a code for BC for back-of-curb. Then store three points with a description of "BC XSCT" for three points to cross section the curb: back-of-curb, top-of-curb and flow line. After these three "BC XSCT" points, you could have single "BC" points along the curb and the program will apply the template along these curb points and draw three parallel lines.

PARKING: Parking - This special code draws parking stall lines using three points. Points one and two are used to draw the first parking line and define the length and angle of the lines. The third point defines that position of the last parking line. On the first point, a number needs to follow the PARKING code for the number of parking lines to draw. For example, PARKING8.



Point Symbol/Attribute Special Codes

ROT: Rotate - This code is used to set the rotation of the point symbol. If a point number follows the ROT code, then angle from the current point to this point number is used for the rotation. For example, "ROT45" would rotate the symbol towards point number 45. If there is no point number after the ROT code, then the rotation point is the next point number with the same code as the current point or a companion code for the current code. ROT can also be used to rotate towards an angle clockwise from north by using '+' or '-' in front of the number. For example ROT+45 rotates the point symbol to the northeast and ROT-90 rotates the point symbol to the west.

SZ: Symbol Size - This code is used to set a different symbol size. There are several ways to use this code. It can take multiple scale factors for different dimensions by putting an ID character after the factor.

- **SZ:** If nothing follows the SZ code, then the next point with the same field code as the current point will be used to determine the size.
- **SZ#:** The value of the new symbol size is specified after the SZ. This value is the actual size in drawing units. For example, SZ2.
- **SZ#X:** The value after the SZ is used to scale the symbol in the X dimension. For example, SZ2X.
- **SZ#Y:** The value after the SZ is used to scale the symbol in the Y dimension. For example, SZ2Y.
- **SZ#Z or SZ#V:** The value after the SZ is used to scale the symbol in the Z (Vertical) dimension. For example, SZ2Z.
- **SZ#H:** The value after the SZ is used to scale the symbol in the X,Y (Horizontal) dimensions. For example, SZ2H.
- **SZ#S:** The value after the SZ is a symbol size scaler that get multiplied by the drawing horizontal scale to determine the actual drawing units. For example, SZ0.2S.

The X, Y, Z, V and H can be combined. For example, to scale a symbol by 10 horizontally and 25 vertically, use SZ10H25Z. Or to scale a symbol by 2 in the X direction and 4 in the Y direction, use SZ2X4Y.

When multiple SZ codes are used in the same point description, the symbol is drawn multiple times at the different sizes. For example, a point description of "TREE SZ5 SZ10" will draw the tree symbol twice. One symbol will be size 5 and the other size 10.

AZI & DIST

The AZI and DIST codes are used together to offset the point. The AZI sets the offset azimuth and DIST sets the distance. The values should directly follow the code. For example, AZI25 DIST4.2 would draw the point offset 4.2 at an azimuth of 25 degrees.

2ND: Multi-Point Code - When used on the first point of a multi-point symbol, the "2ND" code indicates that the second point of the sequence (i.e., the next point after the current one) should be used as the second symbol insertion point for a multi-point symbol. Please refer to Symbol Points in the Edit Field Code Definition section below.

3RD: Multi-Point Code - When used on the first point of a multi-point symbol, the "3RD" code indicates that the third point of the sequence should be used as the third symbol insertion point. The "3RD" code should be used with the "2ND" code. Please refer to Symbol Points in the Edit Field Code Definition section below.

Note: The 2nd and 3rd codes are not necessary if the Automatic Multi-Point Special Codes option is enabled. Please refer to Symbol Points in the Edit Field Code Definition section below.

PHOTO - This code attaches a photo file to the point. The name of the photo file should be right after the PHOTO code. The PHOTO Link setting controls whether the photo is attached using a Carlson-format link or a CAD Hyperlink. Use the Image Inspector command to view photos attached to points by either link method. To use the Hyperlink, you can Ctrl-click or right-click on the point entity.

In addition to the PHOTO code, Field-to-Finish will also automatically create the photo links for SurvCE photos. The program looks for the photo database from SurvCE which should have the same name as the coordinate file with an extension of .phdb. This photo database file should be in a sub-folder of the coordinate file folder and called Pictures_X where X is the name of the coordinate file. For example, if your coordinate file is C:\Projects\Job1\Job1.crd, then the program looks for C:\Projects\Job1\Pictures_Job1\Job1.phdb.

When SurvCE stores photos, it creates this photo database using this naming. So to process with Field-to-Finish, copy the coordinate file and photo files from the data collector to your computer.

LABEL - This code controls the point attribute format using a number after the code. This number uses 0=attribute block, 1=text, 2=both, 3=none. For example, LABEL1 means draw that point using text attributes.

COLOR - This code sets the entity color. The color is specified after the "COLOR" code by either a CAD color number (1 to 256) or color letter. The color letters are R=red, Y=yellow, G=green, C=cyan, B=blue, M=magenta, W=white. For example, COLOR1 means use CAD color 1 (red) and COLORB means use color blue.

PHOTO Link - Sets the option for linking the photos. Carlson format links using the process described above. Hyperlink is a CAD link that is enabled through the CAD properties dialog box. You can also double click on any point and use the image tab to add an image file.

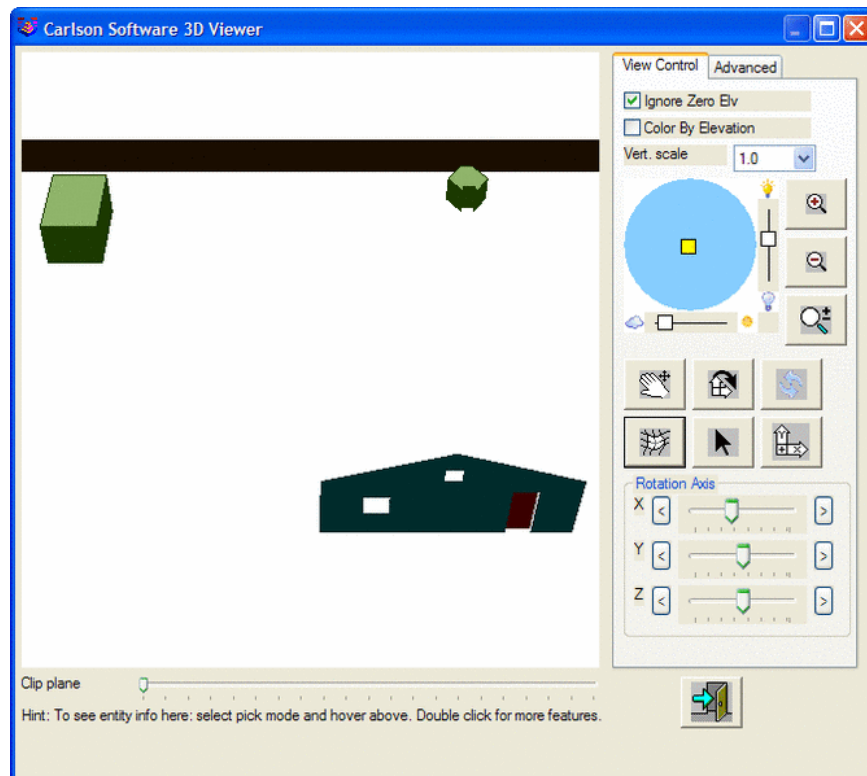
OFB Offset Width -This option works with the **OFB** special code.

- The Double option expects the user to enter half the total offset distance and the routine will double it. For example, if the total width of a pathway is 10, the OFB code will be 5.
- The Split option expects that the user will enter the total distance and the routine will split it in half. For example, if the total width of a pathway is 10, the OFB code will be 10.

2-Point Circle - This option works with the CIR special code. The radius option will use the first point with the CIR code as the radius point and the second located point as the outside of the circle. The Diameter option expects that the first point with the CIR code is one point identifying the outside of the circle and the second point another. Together they create a 2 point circle diameter.

Append Desc Auto Space -applies to the special codes that control the point description label. This option sets whether to insert a space in the description label when appending to the description.

3D Face Special Codes Below are the special codes that can be used for the easy creation of 3D surfaces. The resulting 3D face entities can be viewed in the Carlson 3D viewer by entering "cube" on the command line.



FACE3D - Makes a triangle mesh of 3D face entities by triangulating points starting with the current point and continuing until the line ends or another 3D special code is found. The points must be ordered along the perimeter. Although the mesh will be built if the points are clockwise or counterclockwise along the perimeter, the visible side in the Carlson 3D viewer, "cube", is the clockwise side by default. On the Advanced tab, the shading mode may be set to *Shade both* or *Shade back* if you would prefer to see both sides or just the counter-clockwise side.

HOLE3D - Makes an exclusion area within the triangle mesh identified by the point number following this code (e.g., "HOLE3D101" will start a hole in point # 101). If no point number is given ("HOLE3D"), the exclusion area is applied to the last mesh or if there is a mesh in the process of being constructed by the current sequence of points, it is ended and the hole is applied to it. Note that a hole can only be applied to a mesh that was created by FACE3D (not BLOCK3D or WALL3D). Note also that it can be difficult to predict what the "last mesh" was if it used a different field code since the points of the coordinate file are processed by order of field code first and then point number. There is no limit to how many holes can be applied to a FACE3D mesh. The points of the hole itself are not added to the FACE3D mesh; they are projected on to the best plane that contains the FACE3D mesh and then the hole is cut-out.

Example 1:

2500 HOUSE1 FACE3D */front of house*

2501 HOUSE1

2502 HOUSE1

2503 HOUSE1

2504 HOUSE1

2505 VENT1 HOLE3D2500 */applies 2505-2508 as a hole to last mesh that uses point #2500. So any point in the range 2500-2504 would have the same effect.*

2506 VENT1

2507 VENT1

2508 VENT1

Example 2:

2500 HOUSE1 FACE3D */front of house*

2501 HOUSE1

2502 HOUSE1

2503 HOUSE1

2504 HOUSE1

2505 HOUSE1 HOLE3D */stops the above mesh and applies 2505-2508 as a hole*

2506 HOUSE1

2507 HOUSE1

2508 HOUSE1

Example 3:

2500 HOUSE1 FACE3D */front of house*

2501 HOUSE1

2502 HOUSE1

2503 HOUSE1

2504 HOUSE1

2505 WINDOW1 FACE3D HOLE3D2503 */applies 2505-2508 as a hole to above mesh 2500-2504 and starts a new mesh using the WINDOW field code.*

2506 WINDOW1

2507 WINDOW1

2508 WINDOW1

Example 4 (same result as Example 3):

2500 HOUSE1 FACE3D */front of house*

2501 HOUSE1

2502 HOUSE1

2503 HOUSE1
2504 HOUSE1
2505 WINDOW1 FACE3D /starts a new mesh using the WINDOW field code.
2506 WINDOW1
2507 WINDOW1
2508 WINDOW1 HOLE3D2504 /makes the mesh 2505-2508 also be a hole in the mesh 2500-2504.

BLOCK3D - Makes a set of 3D faces to make a 3d block using the height value entered after the code (e.g., "BLOCK3D2.3" with height 2.3). Heights can be positive or negative. With 3 points, makes a parallelogram base that is extruded up (or down if height is negative) to form a 6-sided block, including top and bottom. With 4 or more points, makes a closed polygon for the base that is then extruded by the height. The points can be laid out in clockwise or counterclockwise order around the perimeter. The perimeter or base does not have to be a convex polygon.

WALL3D - Makes a set of 3D faces above the polyline using a height value entered after the code (e.g., "WALL3D2.3" with height 2.3). This height is a signed value so you can shoot either the top of the wall or the bottom of the wall. The height can be negative if the points on the top of the wall have been shot. If no height parameter exists, then the height is determined by the distance from the current point to the next point.

After the height value, you can have a width/thickness value. If no width value is entered, then a width of zero is used. When the width value starts with a minus "-", then the width is put on the left side of the points. When the width value starts with a plus "+", then the width is put on the right side of the points. When the width value has no plus or minus, then the points go through the center of the wall width.

Both sides of the wall will have triangles and so both sides will always be visible in the Carlson 3D viewer.

Example – 6' high wall shot along the bottom:

```
2000 1000.000 1060.000 100.000 WALL1 WALL3D6.0 /wall 6'  
2001 1100.000 1060.000 100.000 WALL1  
2002 1100.000 1160.000 100.000 WALL1
```

Example – 6' high and 1.5' wide wall shot along the bottom:

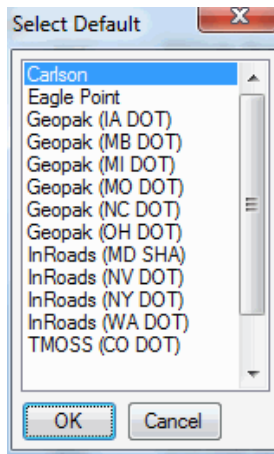
```
2000 1000.000 1060.000 100.000 WALL1 WALL3D6.0 1.5 /wall 6' high and 1.5' thick  
2001 1100.000 1060.000 100.000 WALL1  
2002 1100.000 1160.000 100.000 WALL1
```

Example – 6' high wall, height specified by 1st to 2nd point, shot along the bottom:

```
2020 1100.000 1160.000 100.000 WALL2 WALL3D /height by 2nd pt  
2021 1100.000 1160.000 106.000 WALL2  
2022 1000.000 1160.000 106.000 WALL2
```

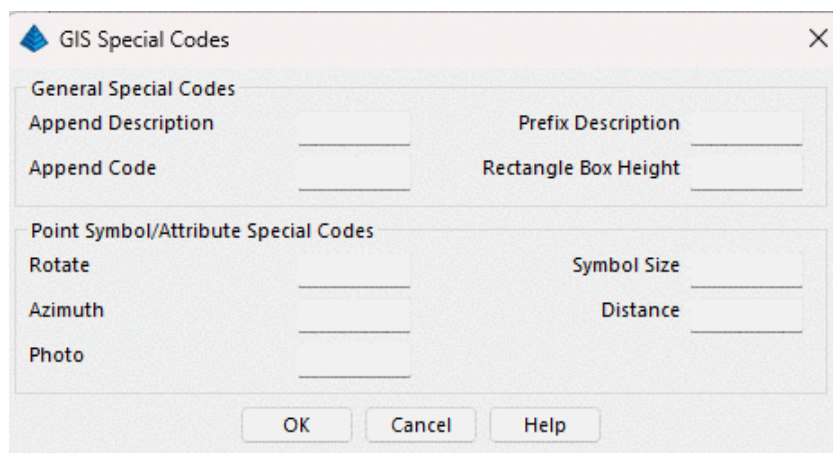
Load Default

This button sets the special codes to Carlson, Eagle Point, Geopak, InRoads or TMOSS defaults.

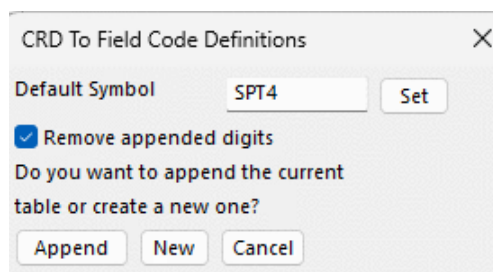


GIS Special Codes: This option allows you to use GIS attribute for Field-to-Finish special coding. For a select group of special codes, a GIS attribute can be assigned. When processing the points, if a point has GIS data for the specified attribute, then that attribute value is used for the special coding. For example, you can have a GIS attribute of COMMENT set to the Append Description special code. Then if a point has a GIS attribute for COMMENT, the value of that COMMENT will be added to the description label for that point.

The Append Code option adds the GIS code to the description code to make the code for processing. For example, for a point with description "ROAD" and a GIS attribute named "TYPE" with a value of "DIRT", then set the GIS Special Code for Append Code to "TYPE" and the program will process this point using the combined code of "ROADDIRT".



Code Table by Points - This command will create code table definitions based on the coordinate file field descriptions. This is useful when creating a code table from scratch



The options available when creating a code table from existing descriptions are:

- Default Symbol: Use the Set button to select a symbol to use for all of the codes created
- Remove appended digits: Removes the numeric group number from any code descriptions. For example EP1 would be created as EP. This avoids unnecessary duplication of code names.
- Append or New: allows you to append an existing code table or create a new one.

Save: Saves the Draw Field to Finish field code definition (.FLD) file.

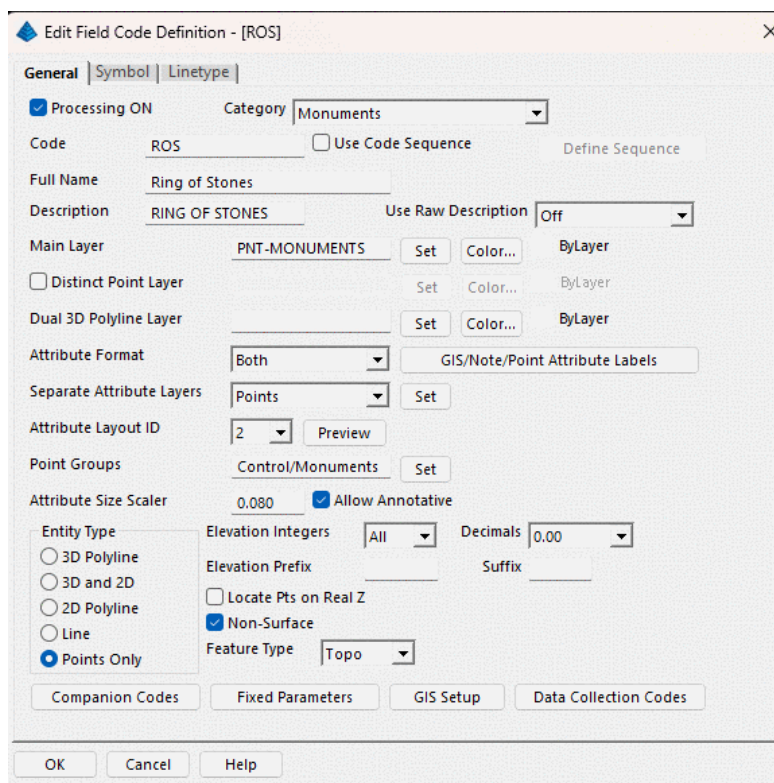
Save As: Reacts the same as Save but allows for specification of file name and location to save to.

Code Definitions

Edit: the Edit button reacts differently depending on whether a Single code is selected or Multiple codes are selected.

Edit (Single Code)

If only one field code is selected, the Edit Field Code Definition dialog box is opened.



The code definition dialog has three tabs: **General**, **Symbol** and **Linetype**. Here are the settings under General:

General Tab

Processing ON: This toggle controls whether this code will be processed. This allows you to process only specific codes if desired.

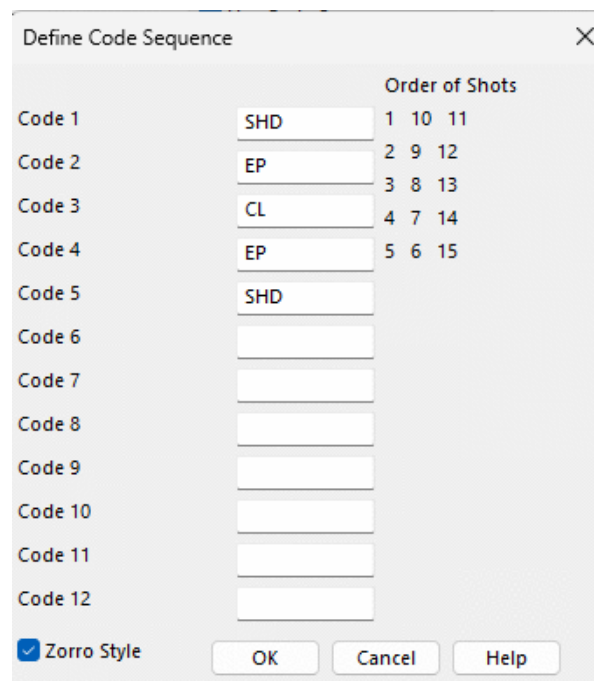
Category: Contains a "drop down" menu that displays the names of the Categories Created in the Draw Field to Finish dialog box for selection.

Code: This is the key name that identifies the code and is matched with the field data descriptions. It is important to note that the * character, used in this field, is regarded as a wildcard or "match anything" code. For example, a field code definition with the code defined as TREE* will be used for any raw description of TREE. Raw descriptions of TREEA, TREE12, TREE, etc. will match the TREE code definition. This will always be the case unless there is a more specific code is found. For example is there was a code TREEA in the code definition file, then that code would be used instead of the TREE code.

It is also important to note that the wild card is not necessary for line number grouping such as EP1, EP2 EP3 etc. The code for this example is simply EP. The program is designed to treat all codes with a numerical suffix as the code itself but separates each one into a separate line string.

Use Code Sequence: This specifies a sequence type code. Sequences are a way to simplify field entry of a sequence of codes. For example, a road cross-section could be SHD1 EP1 CL EP2 SHD2. Instead of entering these different descriptions, one sequence definition can store these descriptions in order. Then just the sequence code (such as RD) is used in the field. The cross-section can be shot in left to right then left right order, right to left then right to left order, or alternating left to right then right to left order. The alternating method is known as the Zorro style. The one restriction is that the shots always start from a right or left edge. To set up a sequence, choose the Sequence toggle in the Edit Code dialog. Then pick the Define Code Sequence button. This brings up a dialog for entering the sequence codes in order. These sequence codes should be defined as normal codes somewhere else in the Draw Field to Finish code table (ie SHD as a 3D polyline). In the field, the one template code is used for all the cross-sections shots (ie RD for all the points). Then Draw Field to Finish will substitute this template code with the sequence codes (ie substitute RD with SHD).

Define Code Sequence: This sets the code names that make up the sequence.

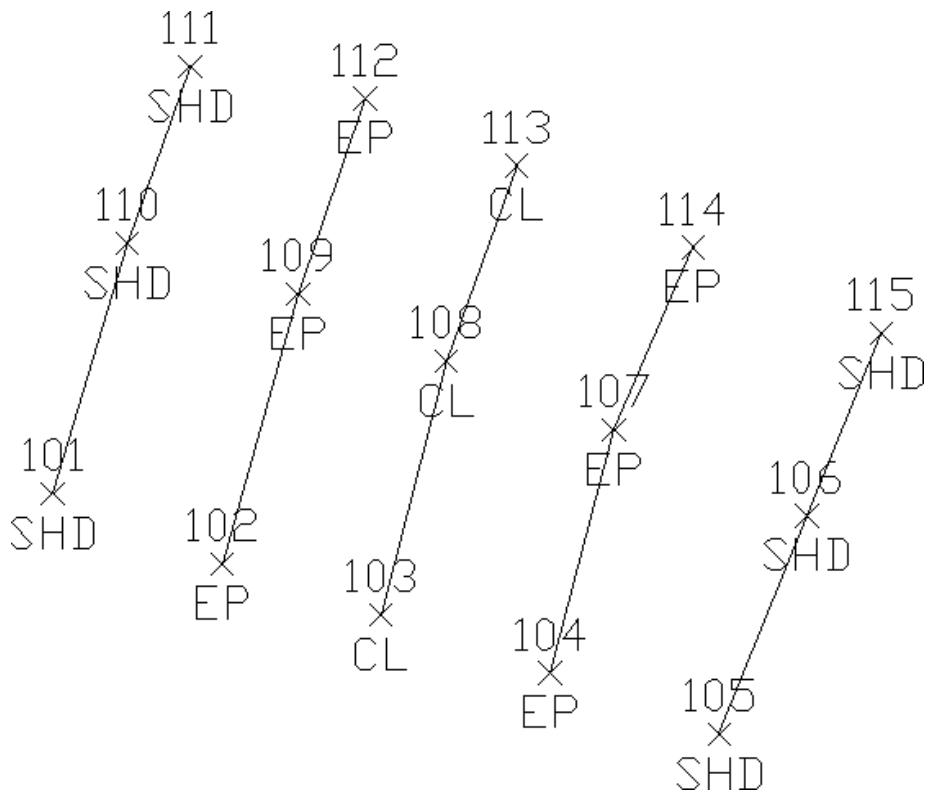


The image shows a dialog box titled "Define Code Sequence" with a close button (X) in the top right corner. The dialog contains a table with 12 rows, each representing a code. The first column is labeled "Code" and the second column is labeled "Order of Shots". The "Order of Shots" column contains three numbers for each code, representing the sequence order. The codes are: Code 1 (SHD), Code 2 (EP), Code 3 (CL), Code 4 (EP), Code 5 (SHD), Code 6 (empty), Code 7 (empty), Code 8 (empty), Code 9 (empty), Code 10 (empty), Code 11 (empty), and Code 12 (empty). At the bottom left, there is a checked checkbox labeled "Zorro Style". At the bottom right, there are three buttons: "OK", "Cancel", and "Help".

Code	Order of Shots
Code 1	1 10 11
Code 2	2 9 12
Code 3	3 8 13
Code 4	4 7 14
Code 5	5 6 15
Code 6	
Code 7	
Code 8	
Code 9	
Code 10	
Code 11	
Code 12	

Zorro Style

OK Cancel Help



Resulting points and linework showing Zorro style template

Full Name: This is an optional field that describes the code for viewing and printing but also works with the **Legend** command.

Description: This value is assigned to the point description attribute when the point is drawn. This description can be different than the field code. For example, the code **IP** could have a description of **IRON PIPE** which will be applied to the attribute of the point.

Note: An additional description can be added to a point by entering it after a forward slash "/" for suffix or backwards slash "\" for prefix in the data description field.

Example **IP\3"** would be **3" IRON PIPE**.

Use Raw Description: This option turns off the Description field described above. Instead the points will be drawn with their original unprocessed descriptions.

- Attribute Block: The raw description will populate the *Description Attribute* of the point block.
- Text Attribute: The raw description will add text to the drawing in accordance with the **Attribute Format** options and the respective **GIS/NOTE/Point Attribute Labels** options.
- Both: will perform both of the above options.

Main Layer: The point and line work for the code will be created and placed on this layer.

- The **Set** button allows the selection of an existing layer contained in the current drawing.
- The **Color** button allows the selection of a color from the CAD color selection dialog box.

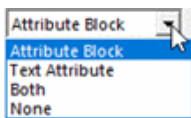
Distinct Point Layer: When this toggle is selected, the line work is created on the layer defined in the Main Layer field and the points are created in the specified distinct point layer. For example, you could have DRIVEWAY for linework and DRIVEWAY_PNT for the points. This option allows for greater control over the placement of points and linework.

- The **Set** button allows the selection of an existing layer contained in the current drawing.
- The **Color** button allows the selection of a color from the CAD color selection dialog box.

Dual 3D Polyline Layer: Displays the layer that the 3d polyline will be drawn on when using an Entity Type of 3D and 2D. The layer name can be typed in this field. The result is the 2D and 3D polylines being drawn on separate layers. For Example, you could have the 2D polyline drawn on the layer DRIVEWAY and the 3D polyline drawn on the layer BREAKLINES.

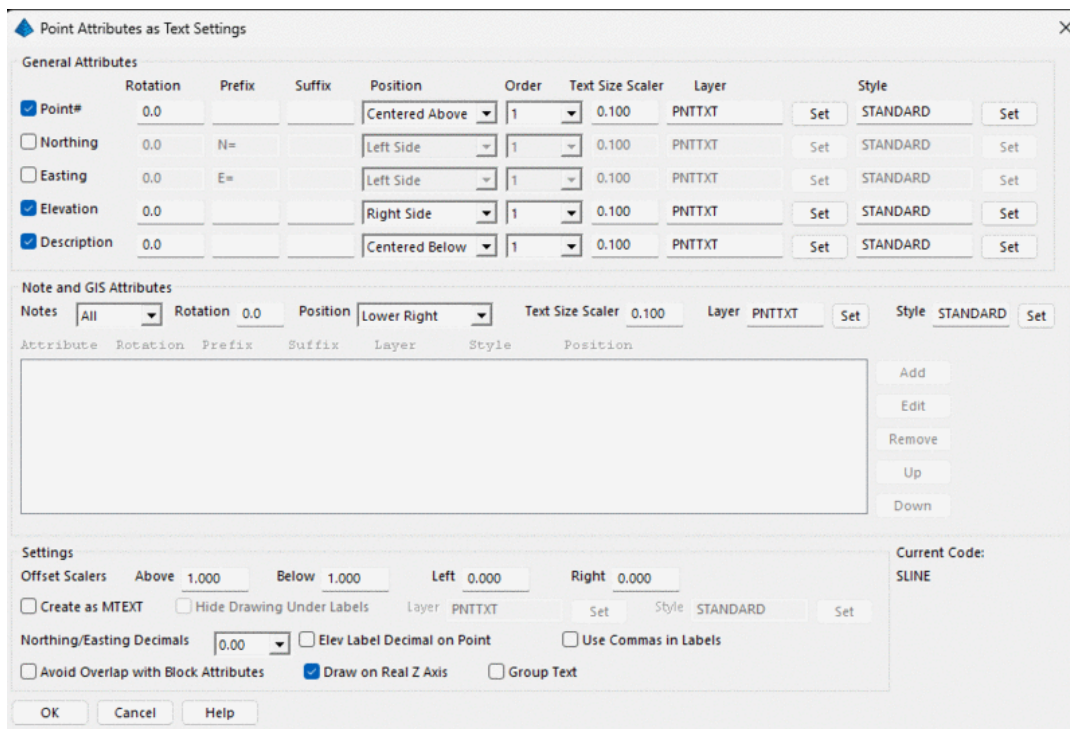
- The **Set** button allows the selection of an existing layer contained in the current drawing.
- The **Color** button allows the selection of a color from the CAD color selection dialog box.

Attribute Format:



This chooses the type of point entities to create. The **Attribute Block** format creates a Carlson point entity which is a block with attributes for point#, elevation and description. The layer of each attribute as well as the point block can be controlled by setting the Separate Attribute Layers.

The **Text Attribute** format creates text entities for each of the point attributes.



When the **Text Attribute** or **Both** format is selected, the placement, appearance, and function of each is controlled by selecting the GIS/Note/Point Attribute Labels button.

For the **General Attributes** control, there are toggles to control whether or not each attribute is displayed for the Point# (number), Northing, Easting Elevation and Description. In addition, there are controls for for each attribute that include:

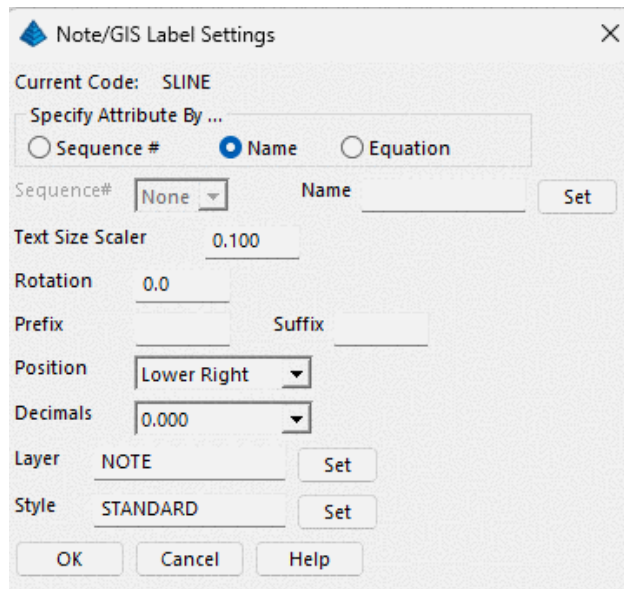
- Rotation - rotates the attribute display
- Prefix - adds a prefix to the attribute
- Suffix - adds a suffix to the attribute
- Position - controls the placement of the attribute in relationship to the point (node)
- Order - controls the order of each attributes.
- Text Size Scaler - sets the size of the attributes in relation to the drawing scale. Example 0.10 at a drawing scale of 1" = 20' will create a text size of 2.0
- Layer - sets the layer for the attribute to be placed
- Style - sets the text style for the attribute.

The **Note and GIS Attributes** section of the dialog box adds additional control over both Notes contained in the .NOT file and GIS attributes defined either by the GIS module, SurvCE/PC or other data collectors or created via Field to Finish fixed parameters. These notes/attributes are in addition to the point description but are directly related to each individual point. Making use of these additional attributes can add a great deal of power and flexibility to the Field to Finish program.

The **Notes** drop-down box allows 3 options.

- None - will not place any additional attributes into the drawing
- All (default) - will place all attributes contained in the .NOT file or created during the Field to Finish process into the drawing.
- Selected - will only place the specifically selected attributes into the drawing.

For **selected**, use the Add, Edit and Remove buttons to build the list of fields to label.



To specify the field to label, the *Sequence#* method sets the field by its order position. For example, a sequence of 3 would use the third attribute for the point. 5 Wires in the below example

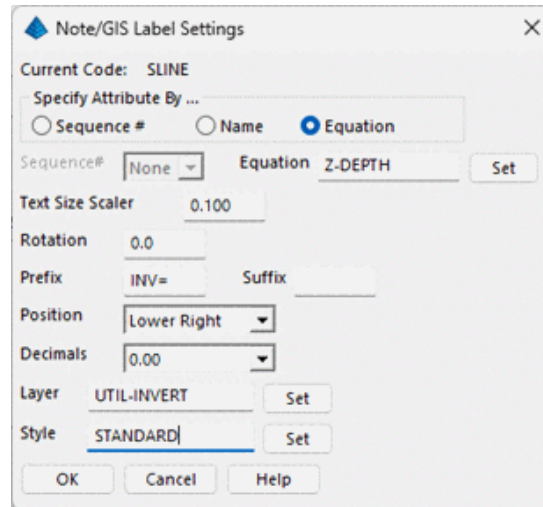
Note	
1	#43
2	WOOD
3	5 WIRES
4	
5	
6	
7	
8	
9	
10	

The **Name** method sets the field to label by field name. The field name can be predefined through the GIS feature file, by SurvCE/PC in the field or can be created via the Fixed Parameters in the Edit Field Code Definition dialog box. Example of Named Attributes:

Note	
1	NUMBER=#43
2	MATERIAL=WOOD
3	WIRES=5
4	
5	
6	
7	
8	
9	
10	

The **Equation** method sets the value by the specified equation of attribute names and numbers. Besides the attribute

names, there are keywords of "X", "Y" and "Z" for the values of the current point coordinates. For example, if there is an attribute named DEPTH, then you could define a label for the invert as an equation of "Z - DEPTH".



Another example is to make a label that is 1.5 higher than the point elevation using an equation of "Z + 1.5". Besides attribute names, you can make an equation by attribute sequence using "NOTE" plus the sequence number. For example, if the depth value is the 3rd attribute, then to define a label for the invert elevation, use the equation "Z - NOTE3".

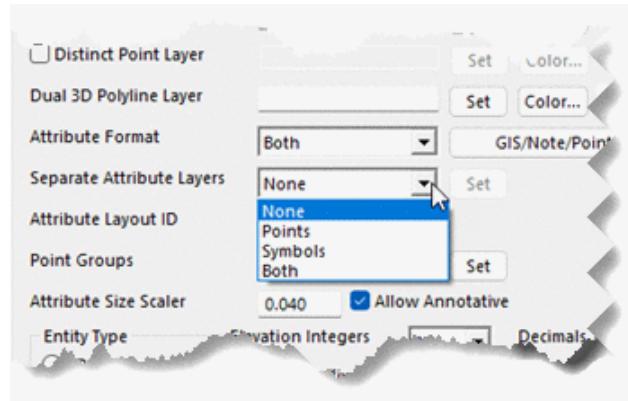
Note: Besides labeling attributes as text with this method, the Symbol > Custom Attributes feature is a way to label attributes as block attributes.

The **Settings** portion of the dialog box adds additional control of the placement and appearance of the attributes as they are placed in the drawing.

- Offset Scalers - control the distance for the text from the point for the different positions. These offset distances are calculated by multiplying the scaler by the horizontal scale for the drawing.
- Create as MTEXT - creates a Mtext entity containing all of the attributes.
 - Hide Drawing Under Labels - creates a background mask for the Mtext entity
 - Layer - with the *Create as Mtext* option enabled, all attributes will be placed on the same layer as set in this portion of the dialog box.
 - Style - with the *Create as Mtext* option enabled, all attributes will be use the same text style as set in this portion of the dialog box.
- Northing/Easting Decimals - sets the precision for the horizontal coordinates displayed
- The Elevation Label Decimal On Point option will place the elevation label so that the decimal point of the label is on top of the point location.
- Use Commas In Labels option will add commas for the thousandths place for the northing, easting and elevation labels. ie. N=1,000,000.00
- Avoid Overlap With Block Attributes option will attempt to resolve conflicts between the point block attributes and those set in this dialog box by shifting the horizontal location of the latter.
- Draw On Real Z Axis option controls whether to the text entities at the point elevation or at zero elevation.

- Group Text option creates a CAD Group for all the point labels.

Separate Attribute Layers:



This controls the layers of the point block, point attributes and/or point symbols.

- None - the point attribute layers are the default layers of, "PNTNO", "PNTELEV" and "PNTDESC", the parent layer for the point attribute block and symbol is the Main Layer designated in the Field Code definition.
- Points - allows the designation of each point block and associated attributes layers. For example if the code DWL has a layer name "DRIVEWAY", the point attributes could then be "DRIVEWAYNO", "DRIVEWAYELEV" and "DRIVEWAYDESC".
- Symbols allows the designation of the symbol associated with the point
- Both allows both of the above options.

Attribute Layout ID: Controls the location of the point number, elevation and description. These attribute layouts are defined in the drawings that are stored in the Carlson SUP directory with the file name of SRVPNO plus the ID number (i.e. SRVPNO1.DWG, SRVPNO2.DWG, etc.). If you want to change the attribute positions for a layout ID, then open and edit the associated SRVPNO drawing. Layout Ids of 1-9 are available.

Point Groups: This field is for the name of the point group that all points with this code will be added to. If the points for this code belong to multiple point groups, you can specify multiple point group names in this field separated by commas. Under Draw in Additional Draw Options, there is an option whether to automatically use the code name as the point group name or to use the name defined in the code definition. A point group does not have to exist for this option to work. It will create the designated point groups when processed.

Attribute Size Scaler: This is a scaler value that is multiplied by the horizontal scale to obtain the actual size for the point number, elevation and description attributes. This control is independent of the symbol size scaler and other text/attribute scalers allowing complete control over each element associated with the code.

Allow Annotative: This option will create the points as annotative when the annotative setting is on in the Point Defaults command.

Entity Type: This defines the line entity to be created.

- Points Only - does not connect points with any line work, however does create an option symbol is selected.
- Line - draws a CAD line entity
- 2D Polyline - creates a 2 dimensional Polyline drawn at elevation 0.

- 3D and 2D - creates both a 2D Polyline at elevation 0 and a 3D Polyline with each vertex matching the point elevation. The 3d polyline is placed on the layer specified in the **Dual 3d polyline Layer** setting and a 2d polyline in the layer identified in the **Main Layer** setting. Since 3d polylines do not display linetypes and are often difficult to draft with, this option is useful. 3D Polylines can be used for adding breaklines for surface modeling while 2d Polylines will maintain proper linetypes and can be edited through general CAD commands such as offset and fillet.

Note: The generation of Points, Symbols 2D and 3D Polylines can also be controlled through the Draw Field to Finish and Additional Draw Options.

Elevation Integers: This controls the number of digits to display to the left of the decimal point for the elevation label. The All setting will show the full elevation digits. The other settings allow you to limit the number of digits to display for the purpose of reducing the amount of space the elevation labels take up in the drawing. For example, if a site is in the 4000 foot elevation range, then this setting could be set to three digits (000) and an elevation of 4321 would be labeled as 321.

Elevation Decimals: This controls the display precision for the elevation label.

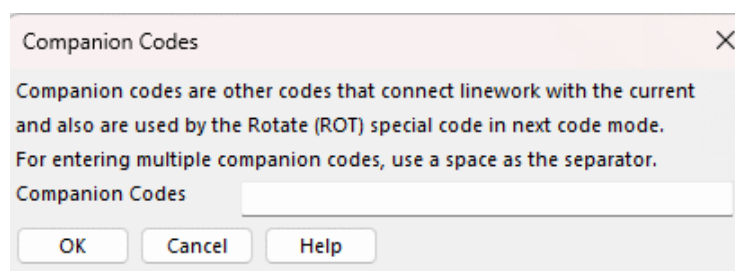
Elevation Prefix/Suffix: These set the prefix and suffix for the elevation label per code. In the Draw function under Additional Draw Settings, there is an override to set the elevation prefix/suffix for all the codes.

Locate Pts on Real Z Axis: This option will draw the points at the actual point elevation. Otherwise the points are drawn at zero elevation. For example, you could turn this option off for the FH for fire hydrant code to draw them at zero. Then the GND code could have this option on to draw the ground shots at their elevations. It is important to remember that Carlson's Triangulate and Contour routine will utilize points for creating a surface model regardless of whether they are set at Real Z or not. If the point is placed in the drawing at Real Z, Triangulate and Contour will use the point block elevation for the Z value whereas if they are drawn at zero (0), it will use the elevation from the CRD file. To ensure that a point is not used for surface modeling in Triangulate and Contour, use the Non-Surface option or utilize the NE or NOS special codes.

Non-Surface: Entities created with this flag are ignored when contouring or creating surfaces regardless of their elevation.

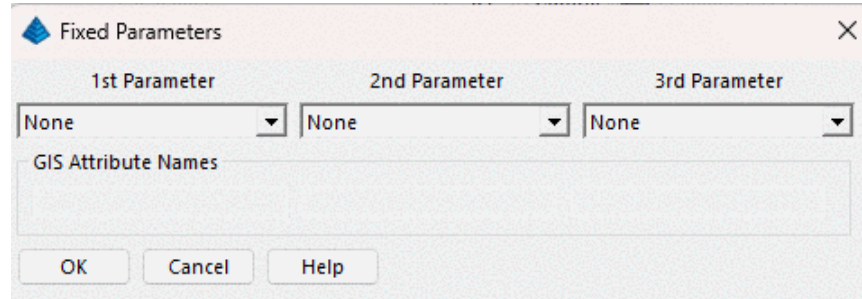
Feature Type: Controls how to process this code. The Default Feature Type is Topo. This is used for most survey work and places points using all of the above assigned settings. The Tree, Utility and Pipe Features are processed more specifically and are controlled via individual setups.

Companion Codes: This option allows different codes to connect when defined as line, polyline or 3d polyline. For example, a main line power pole code may be defined as PP while a service utility pole may be defined as UP. When processing Draw Field to Finish, it may be desired to connect all PP and UP codes together. This could be accomplished by defining a companion for UP as PP and a companion code for PP as UP. Each code needs to reference the other as a companion code.

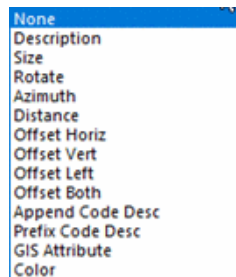


Companion Codes can also be used in conjunction with special codes such as the Rotate (RO) code.

Fixed Parameters: This option is a coding method where you specify a sequence of parameters that follow the main code.



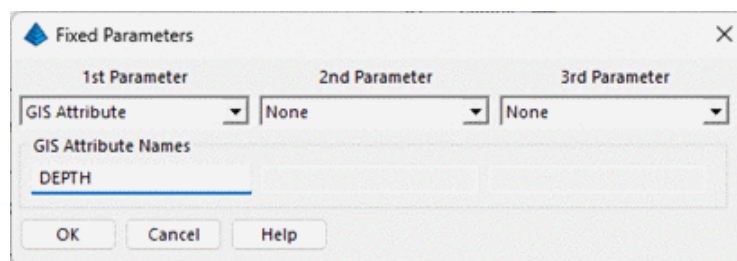
There can be up to three parameters each one having the options shown below:



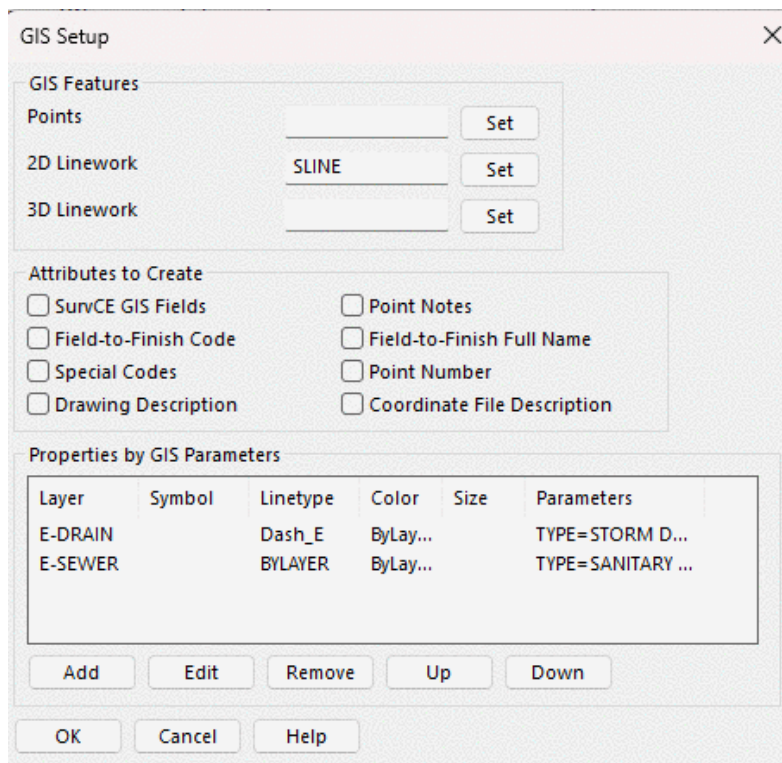
Each of these options take the place of keying in special codes in the field. Instead, the parameters can be entered directly following the main code followed by spaces.

For example: a code called CURB was used to create a consistent front and back of curb lines using the Horizontal and Vertical offset special codes. The first parameter is set as **Offset Horiz** and the second as **Offset Vert**. It would be coded in the field as **CURB 1 .5** for a horizontal offset of 1 and a vertical offset of 0.5 eliminating the need for the OH and OV special codes.

GIS attributes can also be created using Fixed Parameters by choosing the GIS Attribute parameter. This way the GIS attributes can follow the main code in the description. For example, you could have a main code of MH for manhole followed by the depth to store as a GIS attribute. The description would be "MH 4.4" which results in the creation of a named attribute, DEPTH with a value of 4.4.



GIS Setup:



With GIS processing activated, the entities created by Field-to-Finish are linked to a GIS feature name and attributes. The Features and respective attributes are contained in the .GIS file created in the GIS pull-down menu in the Survey module or within the full GIS module. There are two basic required files for using GIS, (beyond the use of Field to Finish), a .GIS file that contains the features and attributes and a database file to contain that data associated with those features and attributes. *See GIS module help for more information*

GIS processing in Field-to-Finish starts with the GIS Table setting in the initial Draw Field To Finish dialog.

Setting the GIS Table is optional but useful. The GIS Table is used as the reference in the Set functions for selecting a GIS feature name to assign to Field-to-Finish codes. Additionally, when processing the Field-to-Finish codes, any associated attributes from the GIS Table will be attached to the entities. Also, attributes generated from Field-to-Finish can be added to the GIS Table. Essentially, there is a "read/write" functionality between the FLD file and the GIS file.

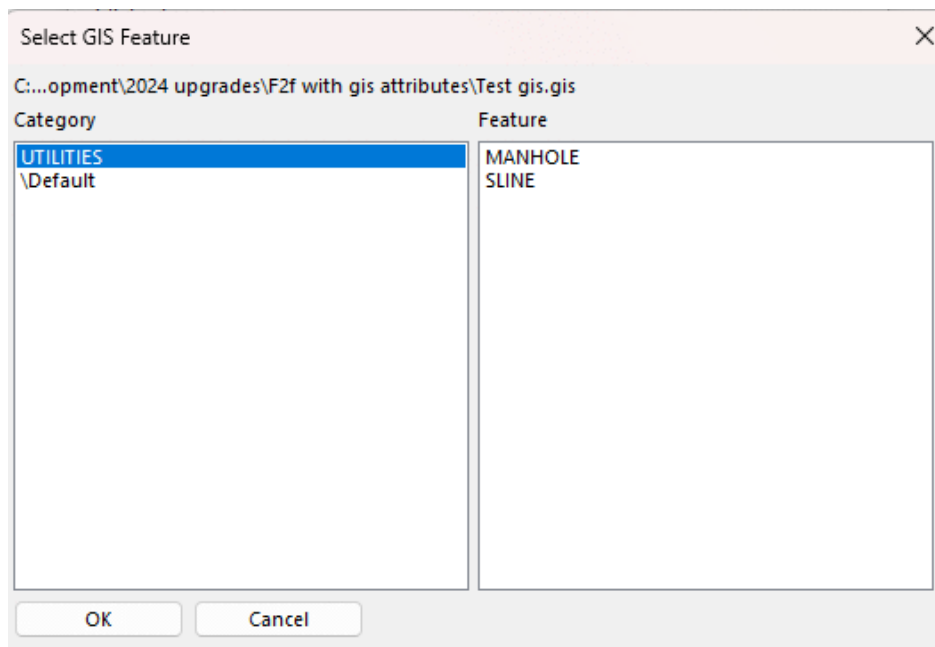
GIS Features: Each Field-to-Finish code has settings to assign GIS feature names. In the Edit Field Code Definition dialog, the GIS Setup button brings up a dialog for setting the GIS feature names and attribute options for the current code.

Points - selects a GIS feature defined as a Point

2D Linework - selects a GIS feature defined as a 2D polyline

3d Linework - selects a GIS feature defined as a 3D polyline

Set - in all 3 of the above options, the Set button will open a dialog box listing the available features in the active GIS table

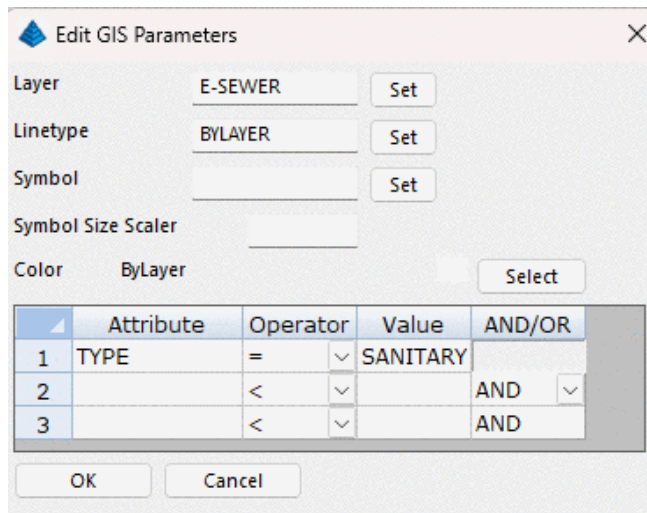


Since Field-to-Finish codes are capable of drawing both points and linework and GIS can have different features for points and linework, there are separate settings for the GIS feature names for points and linework. For example, a Field-to-Finish code UP for utility pole could be setup to draw both points with symbols at the poles and polylines between these points. Then you could have different GIS feature names for the pole points and linework with separate GIS attributes for each.

Attributes to Create: each option toggled on will create a GIS feature and add it to the .GIS file.

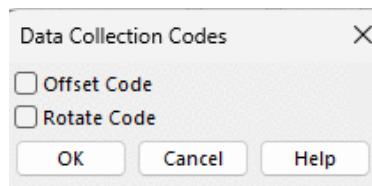
- **SurvCE GIS Fields:** This option uses the attribute data generated by SurvCE which is stored in a .vtt file with the same file name as the current coordinate file except with the .vtt extension.
- **Point Notes:** This option creates GIS attribute data generated by SurvCE from the .not file with the same file name as the coordinate file except with the .not extension.
- **Field-to-Finish Code:** This option creates an attribute named CODE with a value of the Field-to-Finish code name (ie. UP).
- **Field-to-Finish Full Name:** This option creates an attribute named FULL_NAME with a value of the Field-to-Finish Full Name (ie. Utility Pole).
- **Special Codes:** This option creates attributes for Field-to-Finish special codes including OH (Offset Horizontal), OV (Offset Vertical), SZ (Size), ROT (Rotation), AZI (Azimuth) and DIST (Distance).
- **Point Number:** This option creates an attribute named POINT_NAME with a value of the point number from the coordinate file.
- **Drawing Description:** This option creates an attribute named POINT_DWG_DESC with a value of the point description for the point block created in the drawing.
- **Coordinate File Description:** This option creates an attribute named POINT_RAW_DESC with a value of the point description from the coordinate file.

The Point Properties By GIS Parameters lets you control the layer, symbol, linetype, color or size of the points based on GIS attributes.

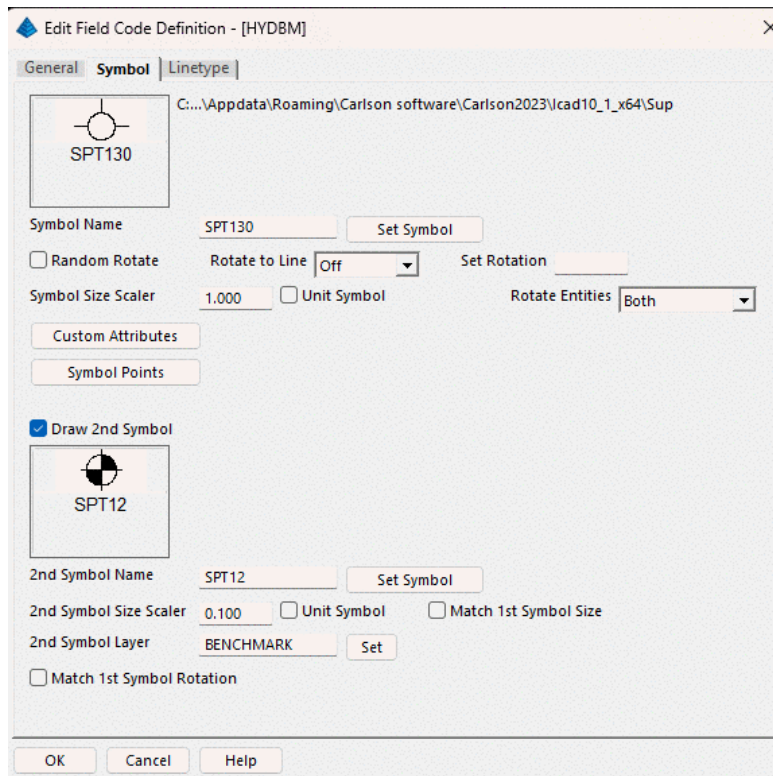


In the above example, a feature with the name SLINE (meaning sewer line) was selected. When Field to Finish finds 2 or more points with that code name and the attribute Type is set to Sanitary, it will draw a 2D polyline on the layer E-Sewer with a bylayer line type. Each code and respective feature can contain as many attributes as desired. Each attribute can then trigger different options for the placement of lines or symbols on different layers at different sizes.

Data Collection Codes: These settings apply to Carlson Field for turning on the Offset mode and Rotate mode automatically by F2F code.



Symbol Tab



Set Symbol: This is the point symbol for the code. The dialog allows you to select from the symbols defined in the Symbol Library which is setup with the Settings->Symbol Library command. Besides the symbols from the symbol library, you can also use any symbols that are defined as blocks in the current drawing by entering the block name in the symbol edit box. To have a point without a symbol, use the Carlson symbol named SPT0 which represents "no symbol". If you create a custom symbol (DWG file) and you want to process Field-To-Finish on other computers, be sure to load this symbol (DWG file) in the Settings > Symbol Library on each computer which will place it into the appropriate sub-folder. Or you can upload the symbol onto Carlson Community for the other computer to find. Field-To-Finish automatically checks the Carlson Community for any symbols that aren't found on a local computer.

Note: a great way to distribute custom symbols as well as all other custom settings and files is to use the Settings Server contained in the Project/Data Folders setup.

Random Rotate: This option will randomly rotate the symbol. For example, this option could be used for tree symbols to have the trees drawn in various orientations.

Rotate To Line: This option applies to points that are part of Field-to-Finish linework. This option will align the point attributes and symbol to the associated linework. The Middle mode uses the average angle for the line segments coming to and from the point. The Forward mode uses the line angle going from the point. The Backward mode uses the line angle coming into the point. The Ends mode uses the perpendicular angle for the line end segment and applies for culverts.

Rotate Entities: This setting controls whether to apply rotation to the symbol, the point attribute block or both.

Set Rotation: Sets a specific rotation for a symbol. This rotation is subject to the current view as set in the Twist Screen command.

Symbol Size Scaler: This is a scaler value that is multiplied by the horizontal scale to obtain the actual size in the drawing. The horizontal scale can be set in *Drawing Setup*.

Unit Symbol: This option, when enabled, will draw the point symbol at unit (1:1) scale and will ignore the horizontal drawing scale. For example, this option could be used for a symbol that is already drawn to actual dimensions such as a car symbol.

Custom Attributes: This feature allows you to use customized blocks that have customized attributes (the tag/value pairs). This feature works for both point attribute blocks and symbols. For attribute blocks, Field-to-Finish looks for attributes with the tags "PT#", "ELEV2", and "DESC2". The custom attributes feature allows you to define additional attributes in their custom blocks on a per-field code basis. The dialog shows five attributes at a time. The number of attributes is unlimited. Use the Next and Back buttons to show more attributes.

Custom Attributes

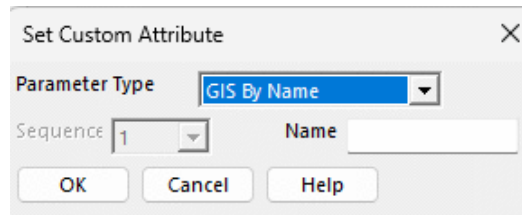
Attributes must not be Constant or Preset.

Find Custom Attributes

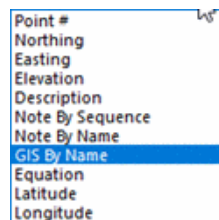
Tag #	Value	Set	Decimals	Prefix	Suffix
Tag #1 TYPE	Value GIS[TYPE]	Set	Decimals	Prefix	Suffix
Tag #2 SIZE	Value GIS[SIZE]	Set	Decimals	Prefix	Suffix
Tag #3	Value	Set	Decimals	Prefix	Suffix
Tag #4	Value	Set	Decimals	Prefix	Suffix
Tag #5	Value	Set	Decimals	Prefix	Suffix

OK Cancel Help Back Next Clear

For an example, the custom block could have an attribute with the tag "TREE_SPECIES" and there is a separate field code for each species of tree. Each of those field codes can specify the value that should be assigned to the attribute that has the TREE_SPECIES tag. Then when the points are drawn, the tree species is shown. Note that the custom attributes must have their Constant and Preset properties set to "no". The custom attributes settings in F2F should not use those tags that the software already handles (PT#, ELEV2, and DESC2), or the setting will be ignored.

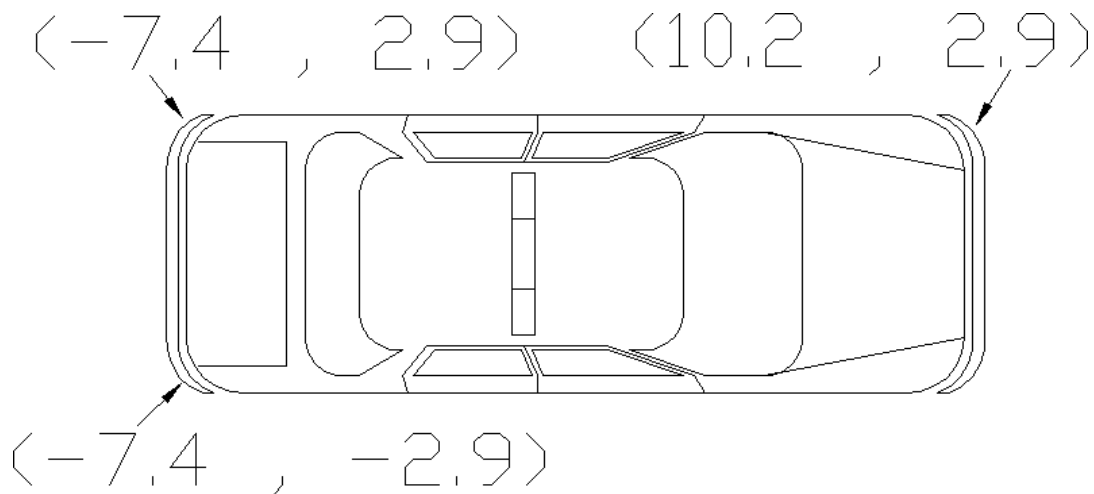
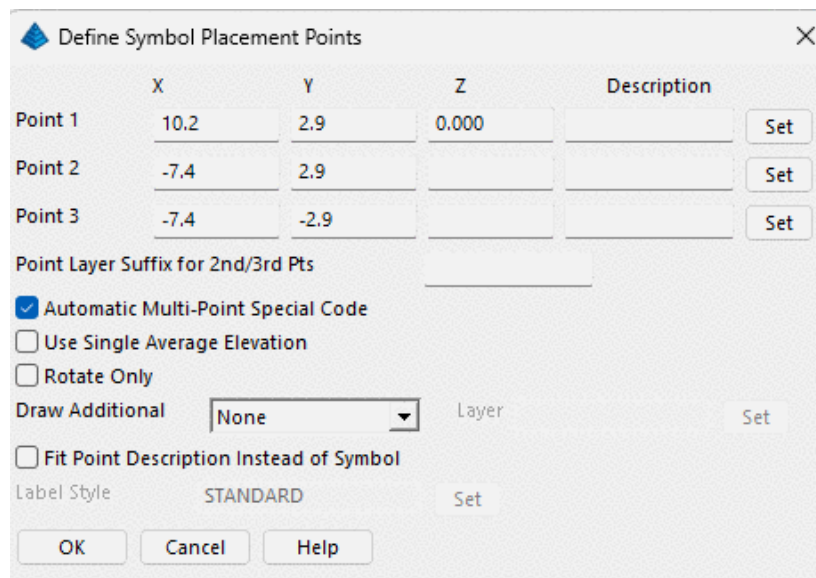


The Values for the attributes can be fixed strings that you enter in the dialog shown here. They can also be dynamic parameters including point number, northing, easting, elevation or description for the current point as well as a point note, GIS attribute or equation. The Equation type sets the value by the specified equation of attribute names and numbers. Besides the attribute names, there are keywords of "X", "Y" and "Z" for the values of the current point coordinates. For example, if there is an attribute named DEPTH, then you could define a value for the invert as an equation of "Z - DEPTH". To setup a parameter value, pick the Set button and then select the attribute. The Decimals setting applies to fields that are real values. The Prefix and Suffix fields are added to the Value for the attribute string. Available Parameter Types:



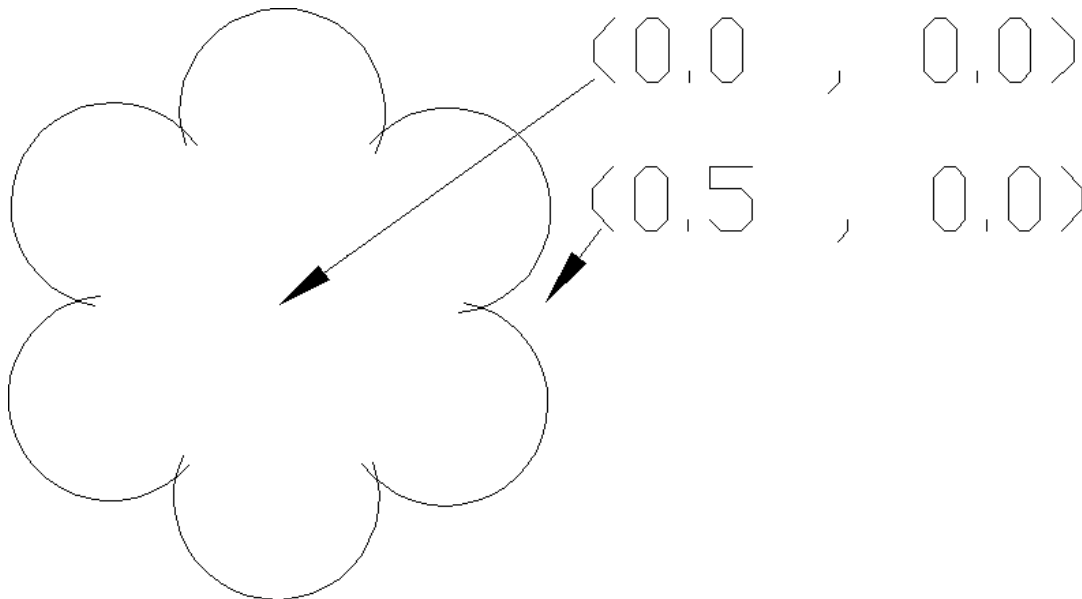
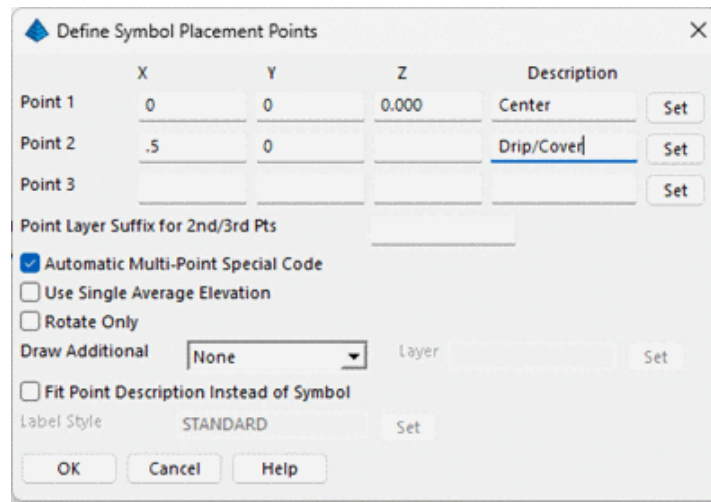
Besides labeling as block attributes, the Attribute Format method of Text mode is a way to label the attributes as text entities.

Symbol Points: For each code definition, the symbol insertion points can be defined with up to three points. To define the symbol insertion points, choose the Symbol Pts button in the Edit Code Definition dialog box. By default, the symbol insertion is defined by one point at the symbol center (0,0). A one point insertion definition can be used to insert a symbol offset from the center. With a two insertion point definitions, the program will rotate and scale the symbol. For example, two insertion points can be used to insert a tree symbol to size the tree, where the first point is for the tree center and the second is for the drip line. With three insertion point definitions, the program will rotate and scale the symbol in both X and Y. For example, three points can be used to insert a car symbol with the first point being the front drivers side, the second point as the back driver side (to rotate and scale the length) and the third as the back passenger side (to scale the width). Besides the insertion point coordinates, you can define a description for each point which is used for the drawn point description and is used for prompting in the Insert Multi-Point Symbol command and in Carlson Field data collection.



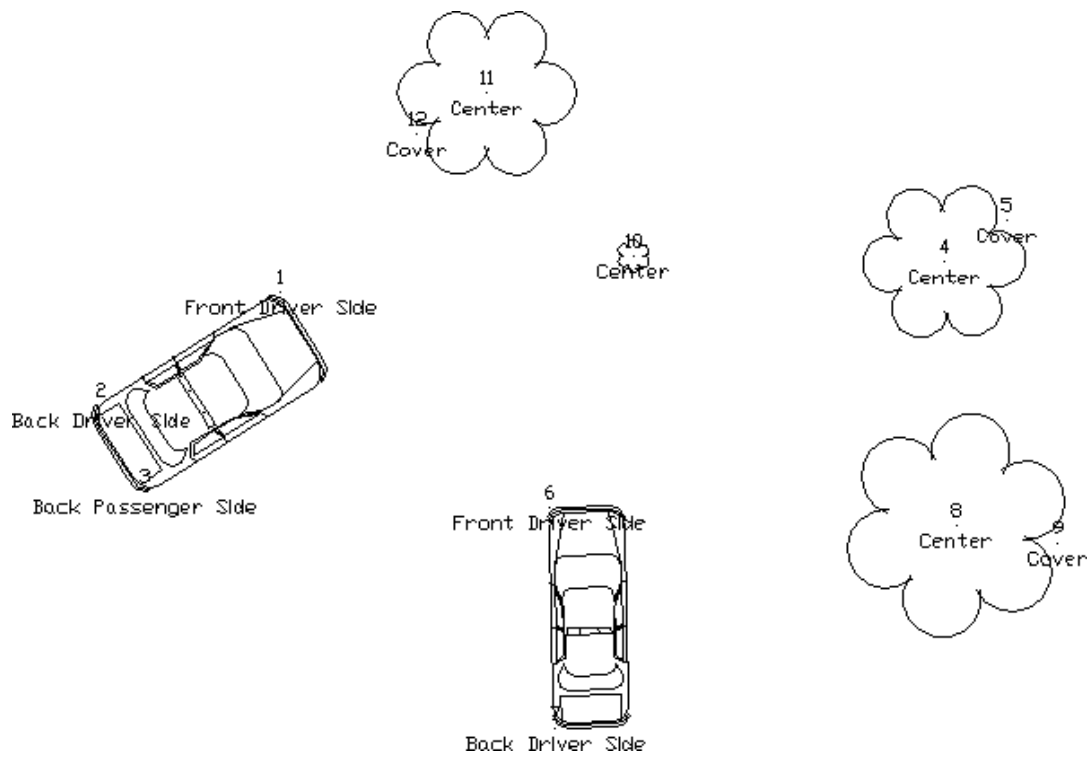
Three Point Symbol Drawing

The coordinates for the insertion point definitions are for the symbol at unit size. To figure these coordinates, you will need to open the symbol drawing (.DWG) file. By default, the symbols are located in the Carlson SUP directory. For example to make an insertion point for the tree drip line, open the tree symbol drawing and find the coordinate at the edge of the tree symbol (in this case 0.5,0.0).



Two Point Symbol Example

Not all of the symbol insertion points need to be used when drawing the points. If a code definition has a three insertion points, it is possible to use just the first two or first one. There are special codes to associate multiple points to the same symbol. The first code point is used as the first symbol insertion point. The "2ND" code is used to specify the second symbol insertion point. A point number can follow the "2ND" to identify a specific point. Otherwise without the point number, the program will use the next point with the current code. The "3RD" code is used to specify the third symbol insertion point and similar to the "2ND" code, a point number after the "3RD" is optional. The "2ND" and "3RD" codes should be assigned to the first point. For example, consider a code of "CAR" with a three point symbol insertion definition. If point #1 has a description of "CAR 2ND 3RD", then point #1 will be used as the first symbol insertion point and the next two points with the "CAR" description will be used as the second and third symbol insertion points.



Multi Point Symbol Drawing

The **Point Layer Suffix for 2nd/3rd Points** option allows for having different layers for the 2nd and 3rd points used with the multi-point symbol.

The **Automatic Multi-Point Special Code** option saves from having to enter the 2ND or 3RD special codes. This option applies when the code is always used from multi-point symbols.

The **Use Single Average Elevation** option averages the elevations of the symbol points for the point elevation label.

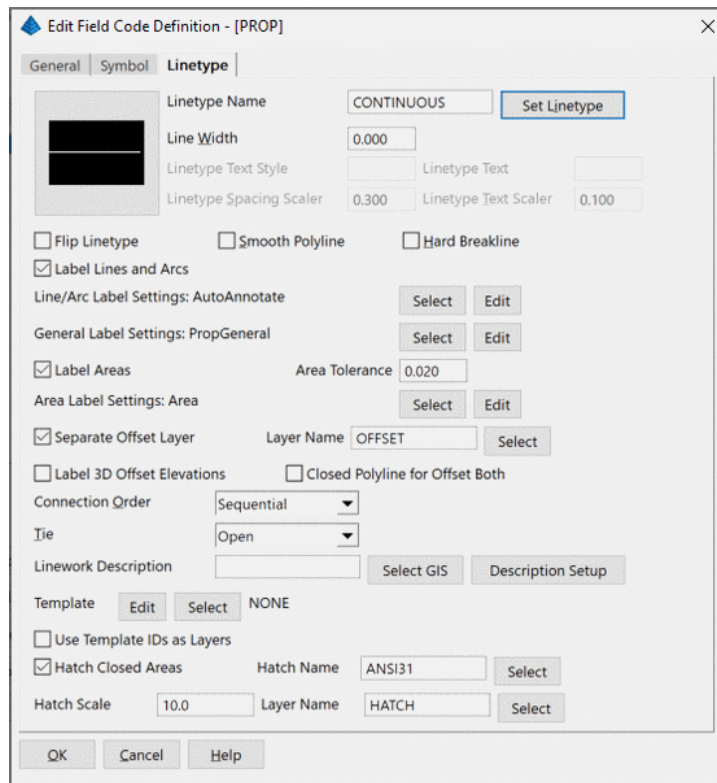
The **Rotate Only** option uses the symbol points only to rotate the symbol and does not scale.

The **Draw Additional** has options for drawing a 4th point or rectangle. The **Create 4th Corner Point** option draws another point when using a three multi-point symbol. This option applies when the symbol is rectangular and you shoot three corner points and want the program to draw a point at the 4th corner. The **Rectangle** option creates a rectangle to fit the 3 symbol points and applies to creating a breakline for surface modeling around the symbol.

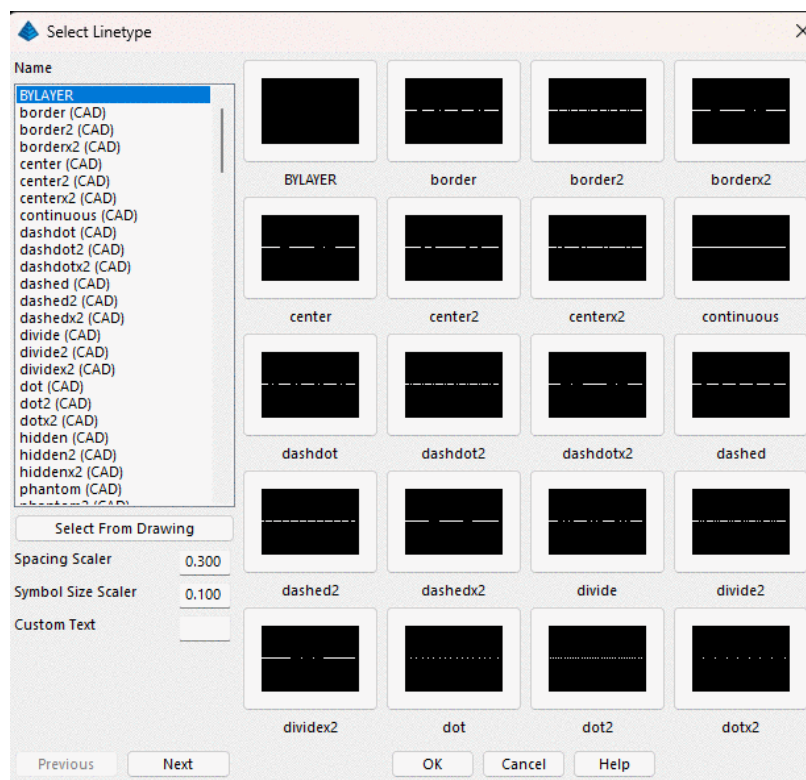
The **Fit Point Description Instead Of Symbol** option is a method to drawing labels that are positioned by these points such as road markings. To set a custom text to label, you can use the `///` (Replace Description) special code. For example with a code called MARK, the description could be "MARK 3RD `///`BUS LANE". Or turn on the Automatic Multi-Point Special Code and then the "3RD" isn't needed and the description could be "MARK`///`BUS LANE". For the text control points, lower-left = 0,0, upper-right = 1,1, upper-left=0,1, and lower-right = 1,0.

Draw 2nd Symbol: This option creates a second symbol on each point. For example, this additional symbol can be used to add a 3D symbol to a 2D symbol used as the first symbol or simply add a second symbol to the point. Besides selecting the symbol name, there are settings for the Symbol Size Scaler Layer and Rotation which mimics the settings described for the first symbol with the addition of matching the 1st symbol size and rotation.

Linetype Tab



Set Linetype: Line work can be drawn in any of the special linetypes or with the linetype for the layer ("BY-LAYER"). There are three types of pre-defined linetypes: **CAD**, **Entity** and **Continuous**.



The type is shown as part of the linetype names in the list. The **CAD** linetypes are the default linetypes available

in AutoCAD and IntelliCAD. The **Entity** linetypes insert text or symbol entities at the linetype interval. These linetypes are the same as used with the Annotate->Polyline To Special Line command. The **Continuous** linetypes define a special linetype in CAD and create continuous polylines with that special linetype. These linetypes are the same as with the Annotate->Change Polyline Linetype command. Besides these pre-defined linetypes within Field-to-Finish, you can also use any linetype that is defined in the drawing by entering that linetype name in the linetype edit box or by picking the **Select From Drawing** button within the Set Linetype dialog. The spacing and size of the special linetypes is determined by the CAD LTSCALE system variable and by the 3 controls shown above or in the field code settings:

- Line Type Spacing Scaler - controls the spacing between the text and/or symbols inserted in the linetype for either the Entity or Continuous linetypes.
- Line Type Text Scaler - controls the size of the text and/or symbols inserted in the linetype for either the Entity or Continuous linetypes.
- Custom Text - allows text entry for the "User Defined" linetype.

Note: The special linetype "**Hedge**" is drawn with a width set by Line Width. The offset special codes of OH or OFB can also be used to control the width of the "hedge" linetype. The special linetype "**userdash**" is drawn with user specified distances for the length of the dash and the length of the gap between dashes.

Line Width: This controls the width for the linework. Only applies to 2D polylines. The LTW special code can also be used to set the line width for a specific line.

Linetype Text Style: Sets the text style created in the *Style* command in CAD

Linetype Text: This is the text that is used for the user-defined linetype. To use, Set Linetype to either Other_E or UserDef_C. Then this text will be used for the linetype. For example, if you have a code for a 8" PVC pipeline, then you could set this text to 8" PVC. This linetype text will use the font set in the **Linetype Text Style**. If this style is blank, the text will use the current font.

Linetype Spacing Scaler: This is a scaler value that is multiplied by the CAD LTSCALE system variable to give the distance between text or symbols in the line.

Linetype Text Scaler: This is a scaler value that is multiplied by the CAD LTSCALE system variable to give the size of the text or symbol in a line.

Flip Linetype: This option switches the side for the linetype which applies to non-symmetrical linetypes like the treeline or guard rail. The Special Code LTF can be added to the first code in the line sequence to accomplish this as well.

Smooth Polyline: This option applies a modified Bezier smoothing to the polyline. The smoothed polyline will pass through all the original points.

Hard Breakline: This option will tag the 3D polylines created with this code as hard breaklines. In *Triangulate & Contour*, contours are not smoothed as they cross hard barriers.

Label Lines and Arcs: This option will label the lines and arcs drawn using the specified Line/Arc and General Label Settings.

- **Line/Arc Label Settings:** *Selected Setting* Select an existing Label Settings file (AAN) by clicking the **Select** button or modify the file by clicking the **Edit** button.
- **General Label Settings:** *Selected Setting* Select an existing Label Settings file (ADF) by clicking the **Select** button or modify the file by clicking the **Edit** button.

Label Areas: This option will label closed areas within the specific tolerance. The areas will be labeled using the specified Area Label Settings.

- **Area Tolerance:** This is the tolerance used to determine if an area is closed and should be labeled. If a polyline closure is greater than this tolerance, the area will not be labeled.
- **Area Label Settings:** *Selected Setting* Select an existing Label Settings file (ARS) by clicking the **Select** button or modify the file by clicking the **Edit** button.

Separate Offset Layer: This setting uses another layer name for polylines created using an offset special code.

Label 3D Offset Elevations: This option labels the elevations and symbols of the vertices for polylines created using an offset special code. The Attribute Format must be set to Text Attribute or both and the Elevation attribute enabled for this option to take effect.

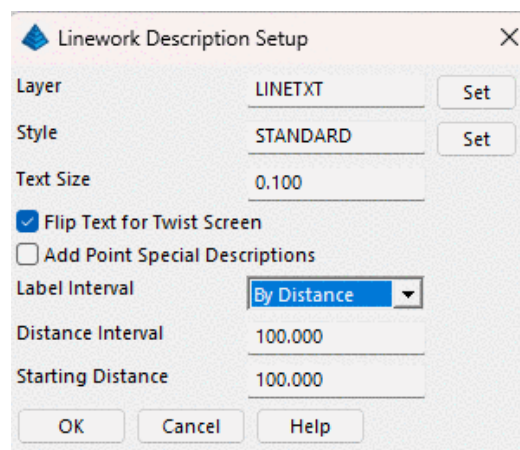
Closed Polyline for Offset Both: This option will create a closed polyline from the left and right offset polylines when the OFB (Offset Both) special code is used.

Connection Order: The points of a distinct code can be connected in their point number order, sequential, or by nearest found which creates the line by connecting each point to the next closest point.

Tie: The options for this is **Open** or **Closed**. When set to Open, the polyline created will end at the last point in the line string. When set to Closed, the last point in the line string will close back to the first. For example if you have points 1, 2, 3, and 4 with the code BLDG and Tie is set to Closed, the linework will be drawn from point 1 to 2 to 3 to 4 and then back to point 1, closing the figure.

Linework Description: This allows for an optional description which is labeled along linework created by the code.

- The **Select GIS** button chooses a GIS attribute to use for the linework description from the GIS Table. The program checks all the points used for the linework for the GIS attribute for this description. To setup the GIS attributes, the Linework Description string is formatted with "[GIS:attribute]". For example, if you have GIS attributes for "Client" and "Weather", then the string to label both attributes with prefixes could be "Client=[GIS:Client], Forecast=[GIS:Weather]" or without prefixes could be "[GIS:Client] [GIS:Weather]".
- **Description Setup** button displays a dialog to control the layer, style and size for these labels.



Layer - Sets the layer for the line text

Style - sets the text style for the line text

Text Size - sets the text size scaler for the line text

Flip Text for Twist Screen - orients the linework labels to be right side up for the drawing current twist view

Add Point Special Descriptions - adds a special description created using the prefix or suffix special codes and adds that to the description. For example if a sewer line were drawn with the Linework Description of Sewer and a prefix was added to the code of /8", the linework would be labeled as 8" Sewer

Label Interval - sets the option for the frequency of labeling the linework description. The options are from 1 to 4 labels per line string or by a Distance Interval.

- Distance Interval - sets the distance between labels in terms of drawing units.
- Starting Distance - sets the distance from the beginning of the line for the first description label.

Edit and Set Template: For 3D polyline codes, this option allows you to assign a template (.TPL) file to the code. The code points act as the base line for the template and the program will draw parallel 3D polylines for each break point (grade ID) in the template. The template file is defined in the Civil Design module or by using the Edit button. In the Edit Template dialog, there's a special setting for Field-to-Finish for the Offset Target Grade. This Offset Target Grade controls which grade in the template to use as the center line for the code points. If this field is blank, then the start of the template is used.

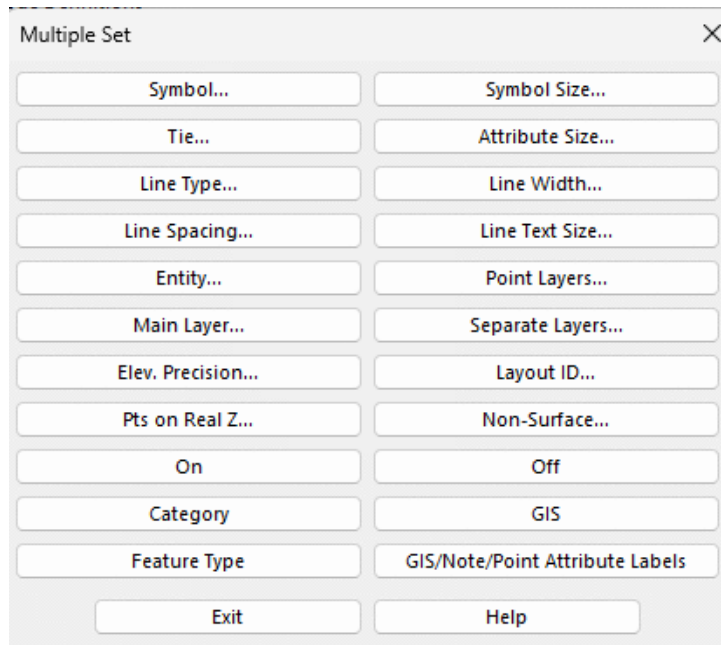
The templates are dynamic and can be adjusted along the 3D polyline by using the OH (offset horizontal) and OV (offset vertical) special codes. For example, when there is a curb template that flattens when it comes to a driveway, then an OV0 code could be used at the driveway start point and then use OV.5 at the driveway end point to restore the curb to 0.5 feet high.

Use Template IDs as Layer option will use the grade IDs from the template for the layers of the polylines created.

Hatch Closed Areas: This option applies for codes used to draw closed linework which is used as the perimeter for a hatch area. The hatch settings include the hatch pattern name, scale and layer.

Edit (Multiple Codes)

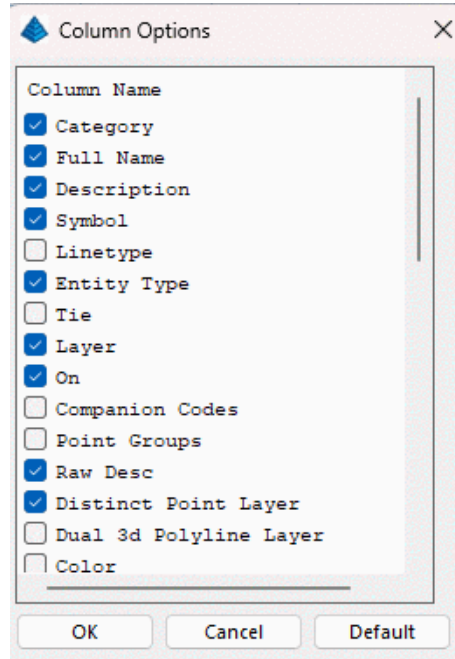
If more than a single code is selected, the Edit button will allow a limited number of "Global Edits" to the codes.



Each of these options will effect all codes selected and is a great way to make changes to a large number of codes at the same time. For example, if you wanted to set all the codes to a particular attribute size, use the Select All option and then Edit. You can then set all codes in the library to the same size attribute.

Code Definitions (continued)

Column Options: Controls the displayed columns in the display window.



Each item can be toggled on or off and the display window will update accordingly. It is important to remember that each field in each column can be modified directly from the display window. This helps greatly in the ease of editing codes in the table. Turn the columns on of interest and use copy/paste or a drop down selection as is available.

Select All: This option selects all the codes within a selected category. This can be used when only wanting to process a couple of codes. For example, use the select all option to select all the codes and then turn them off. Now select the codes for processing and turn them on. Also it can be used to make a global change to all the codes.

Move Category: Moves the selected code(s) to a predefined category.

Add: Adds a new empty code definition which is inserted in the list at the position after the currently selected one. If none are selected for positioning, the new code is placed at the top. Only one code definition may be highlighted before running this routine.

Copy: This option copies the definition of a selected code. It opens the Edit Field Code Definition dialog and copies the definition of the selected code to the appropriate settings. It does not copy the name of the code. It is a time saving tool to use when creating codes that are similar with only a couple of differences.

Delete: This command will remove the highlighted code definitions from the list.

Search: Allows you to search for a specific code in the list.

Move Up/Move Down: allows for the manual resorting of the code list by moving the selected code either up or down.

Feature Settings

Tree Setup:

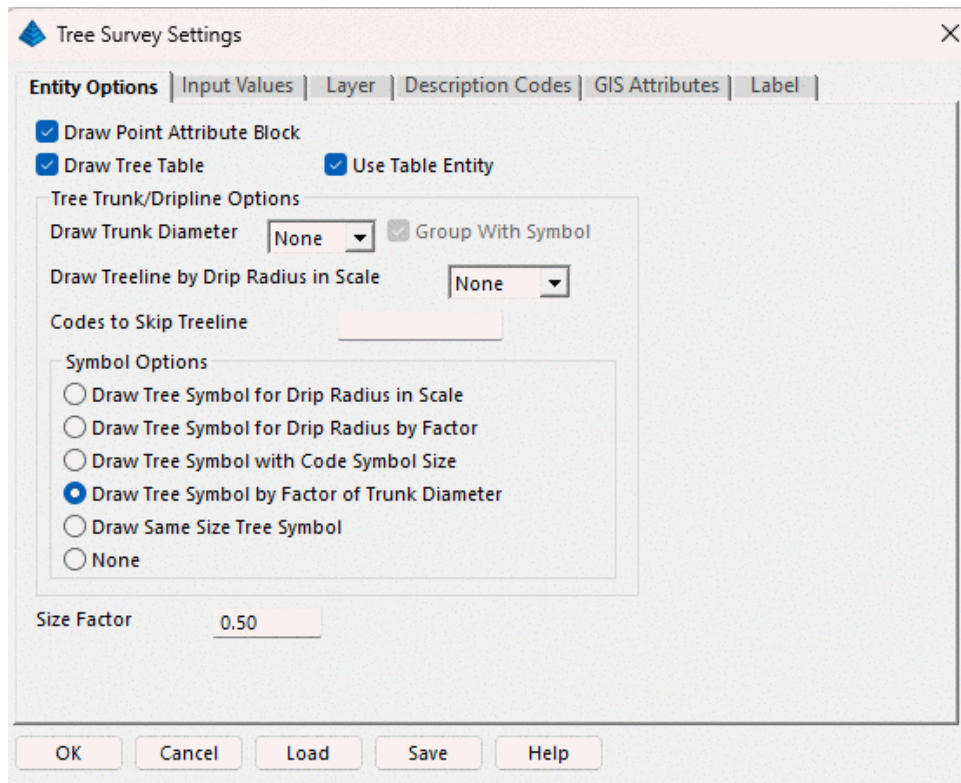
Tree surveys can be coded simply by using general Field-to-Finish coding methods such as defining a code for a tree ("OAK") with a tree symbol and using the SZ special code for sizing the symbol. The **Tree Feature** provides more power and flexibility in cases where a simple symbol is not sufficient. The tree survey works with three optional attributes for each tree: trunk, drip and tag. Trunk is the diameter of the tree trunk. Drip is the radius of the tree canopy. Tag is an id for the tree for reporting.



Tag	Trunk	Drip	Comment
T100	12"	15	MAPLE
T101	10"	10	CEDAR

Important: The Tree Survey Settings apply to codes that are set to a Feature Type of Tree. To set the Feature Type, go to Edit Codes and then the General tab of the Edit Field Code Definition dialog.

Entity Options Tab:



Draw Point Attribute Block -controls whether to draw the point block with the point #, elevation and description attributes.

Draw Tree Table - This option makes the program prompt for whether to draw a tree table when the program finds a couple points with tree codes. Otherwise, there must be many tree codes to have the program prompt for creating a tree table. The Use Table Entity option draws the table as a Carlson Table Entity. Otherwise the table is drawn with regular CAD lines and text.

Draw Trunk Diameter - Sets whether to create a circle or solid with the trunk diameter. The Group With Symbol option creates a CAD group to combine the tree and trunk symbols.

Draw Treeline by Drip Radius in Scale - shrink wraps the tree drip lines to get the overall treeline perimeter. The perimeter polyline can be drawn either as Bubbles or Smooth. The Bubbles creates a treeline style polyline with a series of arcs. The Smooth creates a regular polyline.

Codes to Skip Treeline - allows you to skip specific Field-to-Finish codes from using with the treeline. If you have more than a single code to skip, use either a space or comma to separate the codes.

Draw Tree Symbol for Drip Radius in Scale - draws individual symbols for each tree using the symbols defined in the code table and scaled by the drip size attribute.

Draw Tree Symbol for Drip Radius by Factor - draws individual symbols for each tree using the symbols defined in the code table and scaled by the drip size attribute and the Size Factor from this dialog.

Draw Tree Symbol with Code Symbol Size - draws individual symbols for each tree using the symbol name and size defined in the code table.

Draw Tree Symbol by Factor of Trunk Diameter - draws individual symbols for each tree using the symbols defined in the code table and scaled by the trunk size attribute multiplied by 12. For example, a 10" trunk size is drawn as a 10ft symbol.

Draw Same Size Tree Symbol - draws individual symbols for each tree using the symbols defined in the code table and at size of 6.

None - simply does not draw a tree symbol

Input Values Tab:

The screenshot shows the 'Tree Survey Settings' dialog box with the 'Input Values' tab selected. The dialog has a title bar with a close button (X) and a tabbed interface with the following tabs: 'Entity Options', 'Input Values', 'Layer', 'Description Codes', 'GIS Attributes', and 'Label'. The 'Input Values' tab contains the following settings:

- Default Tree ID to Point ID
- Begin Tree ID from: 100
- Input Trunk Value: Diameter
- Input Drip Value: Radius
- Canopy Companion Code: DRIP

At the bottom of the dialog are five buttons: 'OK', 'Cancel', 'Load', 'Save', and 'Help'.

Default Tree ID to Point ID - option uses the point number for the tree tag unless the point description contains a tree tag.

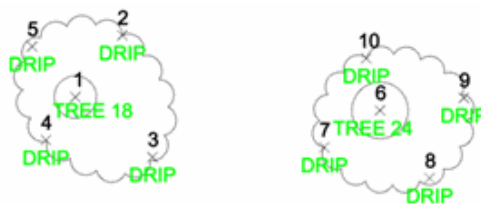
Begin Tree ID From - this is the number to start the incremental numbering of tree tags from. Each point with a tree feature code will be numbered consecutively thereafter.

Input Trunk Value - controls whether the trunk size is entered as a radius or diameter.

Input Drip Value - controls whether the canopy drip size is entered as a radius or diameter.

Canopy Companion Code - this tree coding method handles a point at the trunk and then a series of 3 or more points for the canopy. The point description coding uses separate codes for the trunk and the canopy points.

The program draws a Bezier smooth polyline thru the points for the canopy. In this example, points 1 and 6 have the tree feature code from the Field-to-Finish code table, and the rest of the points have the Canopy Companion Code.



Layer Tab:

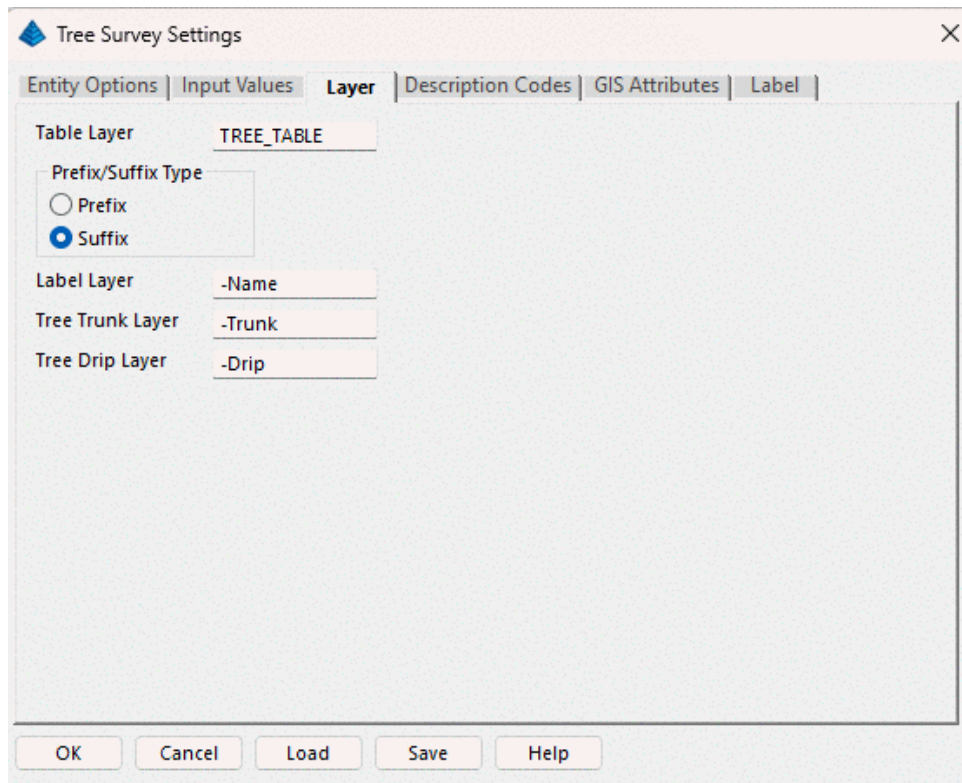


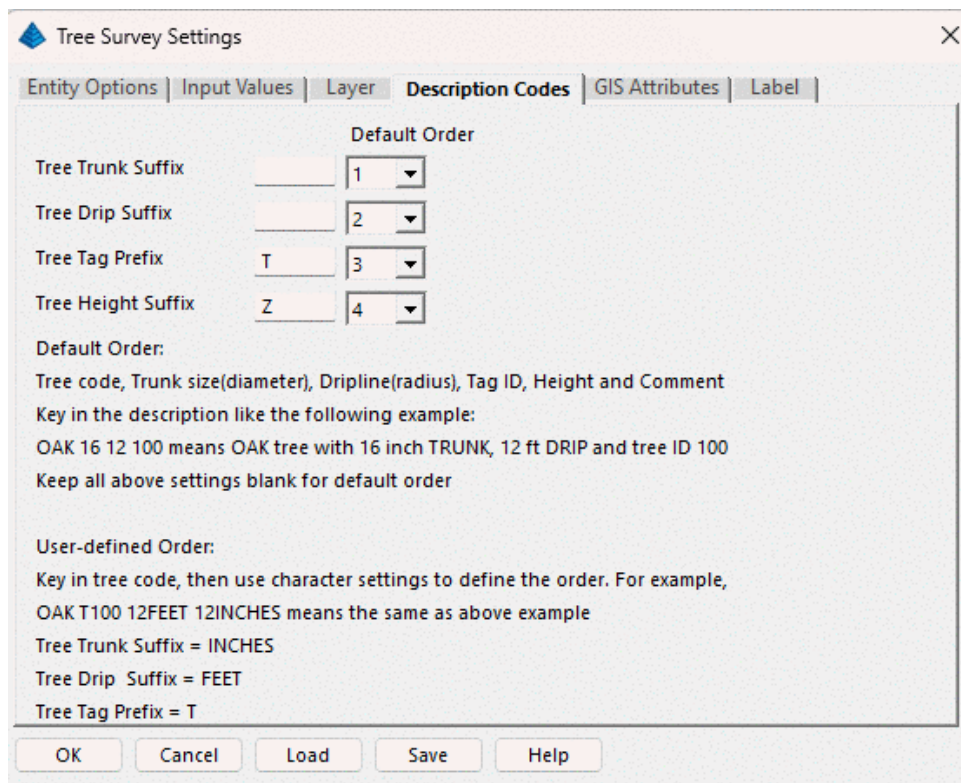
Table Layer - sets the layer for the optional tree table to be drawn

Prefix/Suffix Type - designates whether the layer names will be added as a suffix or prefix to the Main Layer as designated in the code itself.

- Label Layer - added to the layer name for the tree label ie. Maple, Oak etc.
- Tree Trunk Layer - added to the layer name for the label of the trunk size
- Tree Drip Layer - added to the layer name for the label of the drip line size

Example: the main layer for the code DT is set to Tree. The tree labels would be placed on the layers Tree-Name, Tree-Trunk and Tree-Drip

Description Codes Tab:



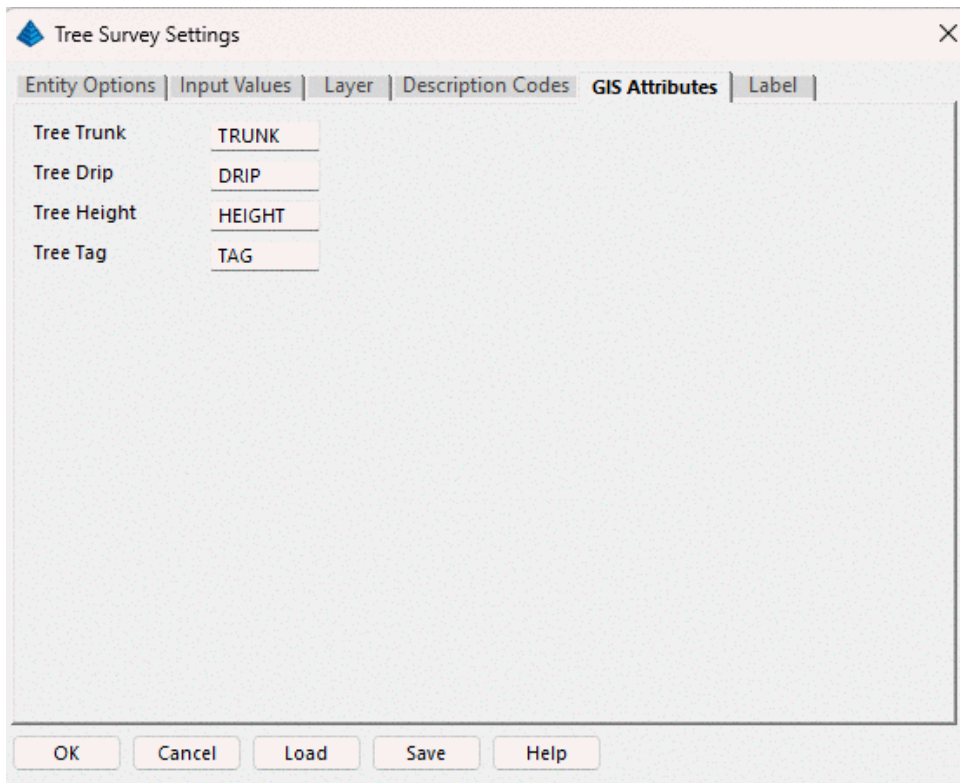
There are settings to help identify the tree attributes in the point description. The program looks for the trunk size, drip size, tag ID and height in the point description after the tree code. By default, the program expects the attributes to be in the order of trunk size, drip size, tag ID and height. Here's an example default order:

OAK 16 12 100 28

where OAK is the tree code from the code table, 16 is the trunk diameter, 12 is the drip radius, 100 is the tag ID and height is 28.

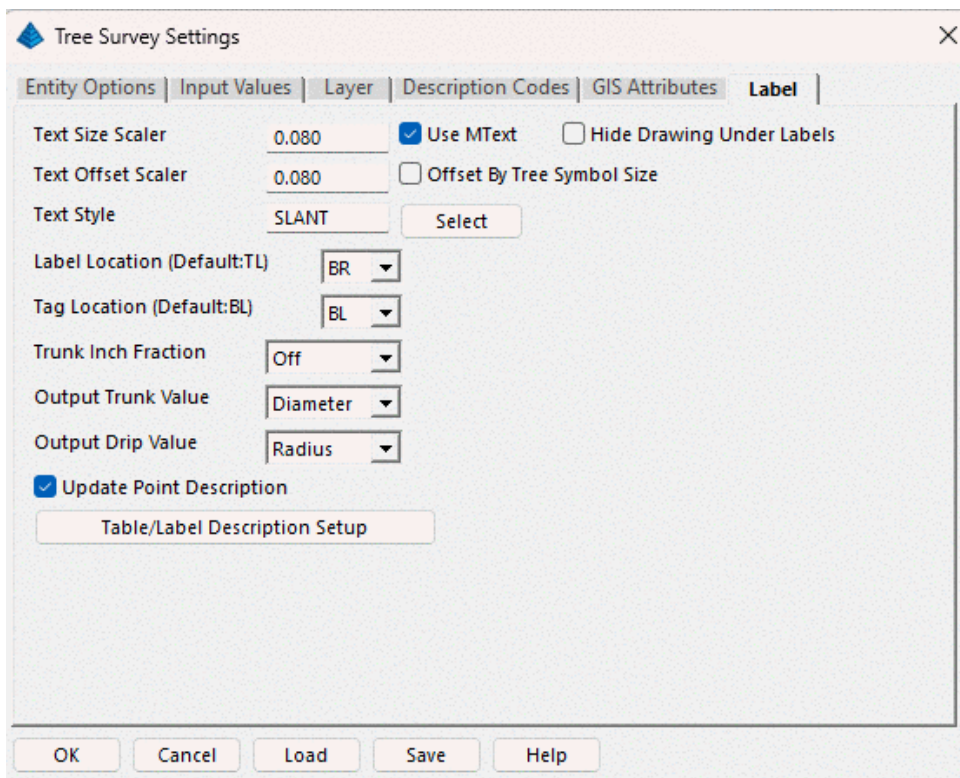
You can change the order that the program looks for the attributes by setting the Default Order numbers. Another way to change the attribute order is to use the suffix/prefix settings to identify the attributes. When the program finds a specified prefix or suffix, it tells the program which attribute to use. For example, if the Trunk Suffix is "in" and the Drip Suffix is "ft" and the Tag Prefix is "T", then OAK T100 16in 12ft means tag ID of 100, trunk diameter 16 and drip radius 12 feet.

GIS Attributes Tab:



In addition to looking for the tree attributes in the point description, the program can also read these attributes from GIS fields. On the **GIS Attributes** dialog tab, you can set the GIS field names for the tree attributes. The data collected in the field would be stored in the .vtt file which would contain the values for each of the defined attributes. Normally, this option would be used in place of adding the tree data directly to the code description.

Label Tab:



There are settings for the tree text labels for the size, offset from trunk center, style and location. When creating a tree table, only the tag text is labeled. Otherwise, the label is drawn.

Text Size Scaler - sets the scale size of the text the value of which is multiplied by the horizontal scale as set in the drawing setup. Options to **Use Mtext** and **Hide Drawing Under Labels** (applies a wipeout or text mask) are available.

Offset Scaler - applies a scale factor to control the placement of the text as it pertains to the point. **Offset By Tree Symbol Size** moves the labels beyond the tree symbol to avoid overlap between the labels and the symbol. The amount to move is factored by the Text Offset Scaler.

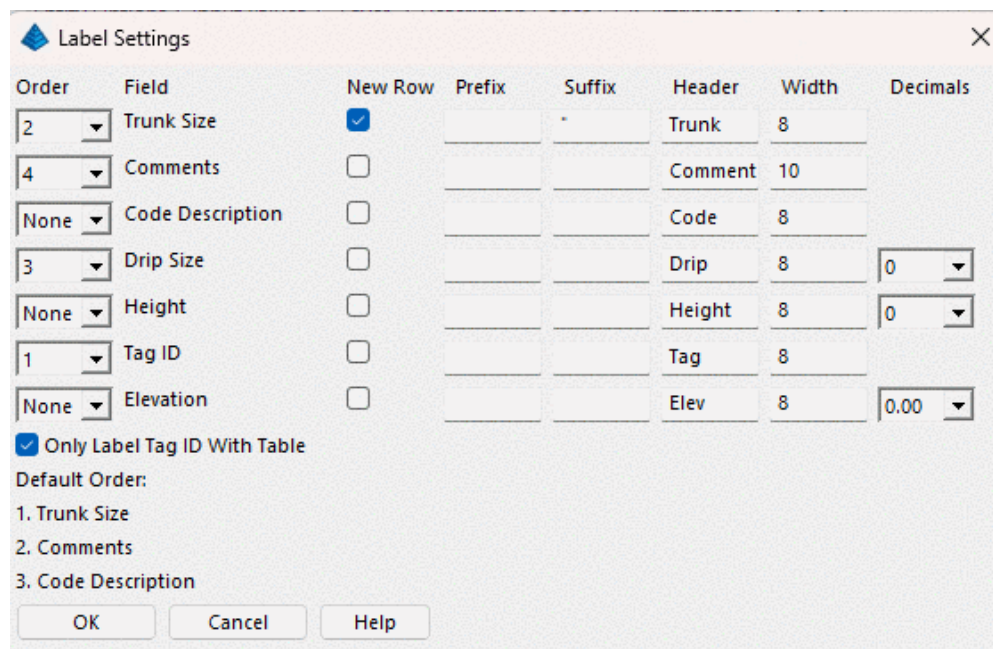
Trunk Inch Fraction - sets the precision in inches to use for the trunk size labels.

Output Trunk Value - sets whether to label the trunk size as a radius or diameter.

Output Drip Value - sets whether to label the canopy (drip line) size as a radius or diameter.

Update Point Description - sets whether to only use the label setup for the table and not the point drawing description. With this toggled on, the description attribute in the drawing will be updated to reflect the values to be used in the table as opposed to the coding as input in the field. The code description in the coordinate file is not affected by this setting.

Table/Label Description Setup - opens a Label Settings dialog box that controls the fields to be labeled in the drawing as well as included in the tree table.



For each field, there are settings to control the placement, prefix, suffix and the caption for the table columns. These settings effect both the labeling in the drawing and the table.

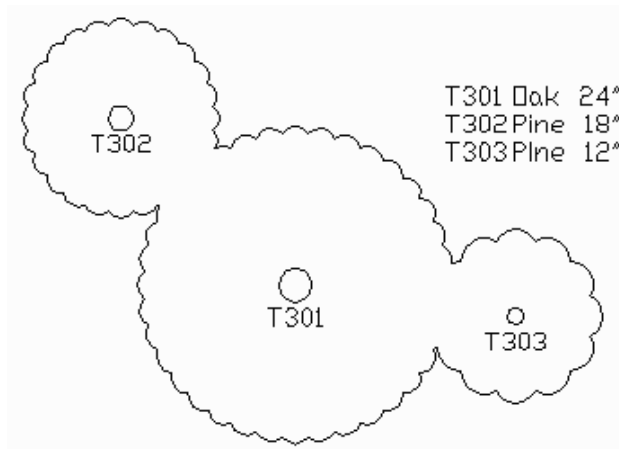
- Order - sets the order for the field
- New Row - when toggled on, places the field on a new row for the description label. This does not affect the tree table.
- Prefix - adds an optional prefix for the label

- Suffix - adds and optional suffix for the label
- Header - designates the caption for the top of the column in the tree table
- Width - sets the width of the field for the tree table column
- Only Label Tag ID With Table - when enabled, tree tags will only be used when the option for a tree table is enabled.

When Field-to-Finish draws entities, the program checks for codes set as tree features and applies the settings from the Tree Survey dialog. When tree features are found, the number of trees are reported along with a prompt for whether to draw a tree table. The tree table has the tag ID, code description and trunk diameter.

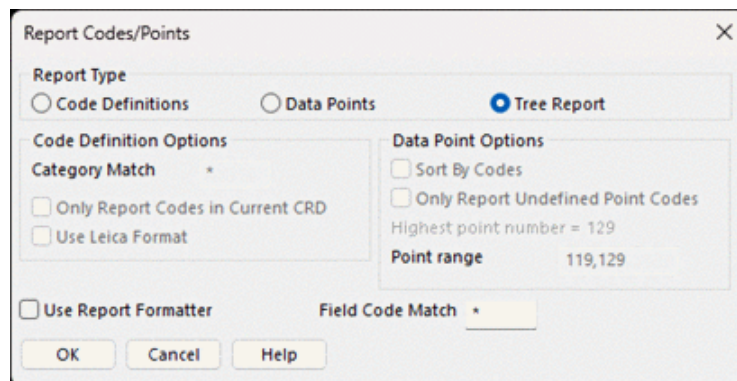
Here is an example with the following three points:

Point#	Northing	Easting	Description
1	4994.73	4923.15	OAK 24 38 301
2	5034.59	4881.40	PINE 18 24 302
3	4987.32	4975.79	PINE 12 20 303

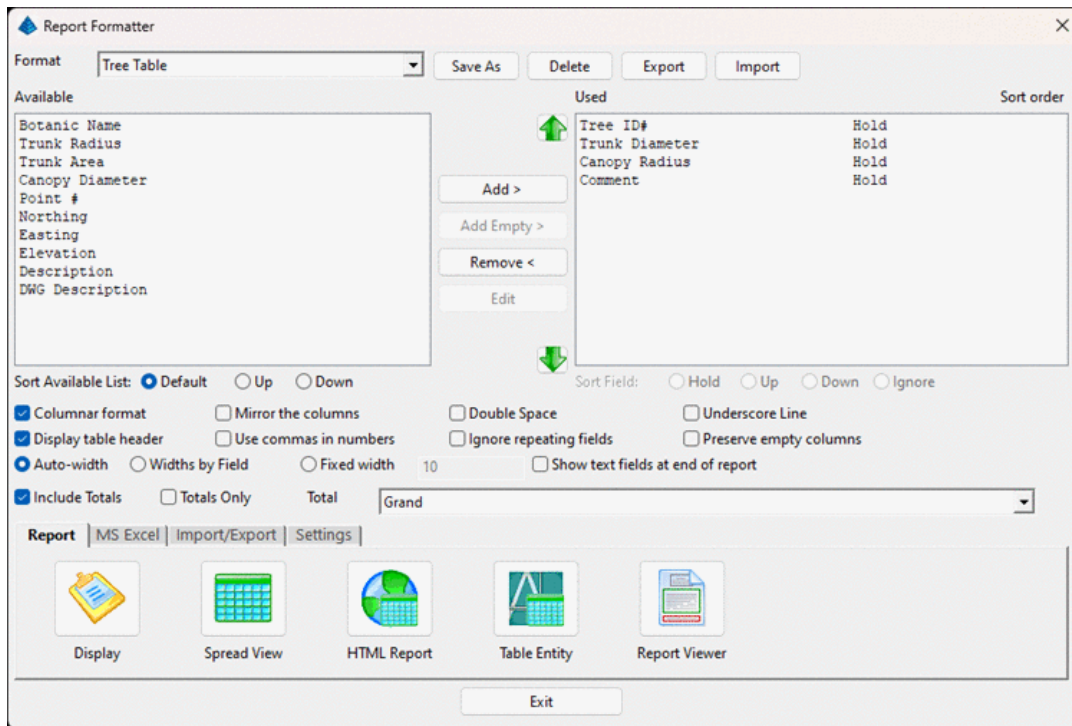


Drip line drawn as Treeline method along with a tree table.

Another feature of Tree Survey is the **Tree Report** under the Report Codes/Points function. The Report Formatter option can be used to make a custom report and output to Excel or create a custom table in the drawing. To use this feature, use the Report Codes/Points option found in the Code Table or Draw Field to Finish dialog box.



Select the Tree Report option and click **<OK>**. Alternatively, enable the Use Report Formatter option and click **<OK>** to reveal additional options for the report.

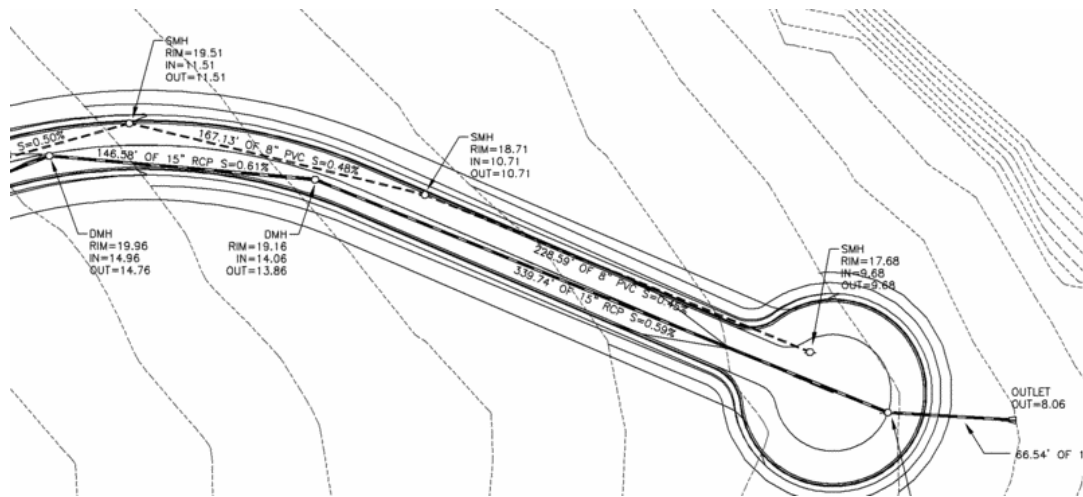


Example Tree Table Report

Tree ID# Trunk Diameter Canopy Radius Comment
 T100 12" 15' MAPLE
 T101 10" 10' CEDAR

Pipe Setup:

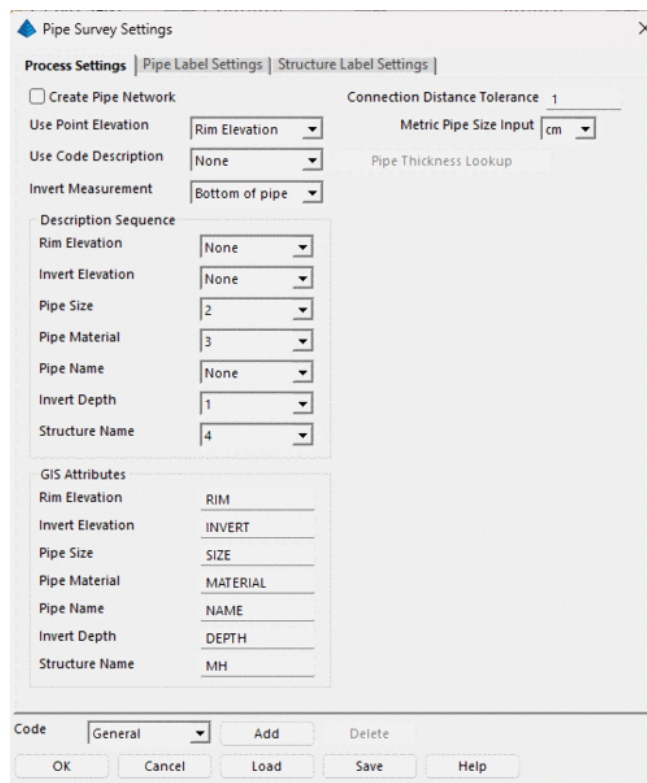
The **Pipe Feature** adds additional properties and options to the drawing a pipe line or network. Size, material and name are some of those options. in addition, the Pipe Feature allows more control over labeling of the pipe parameters and ability to draw the pipe in 3D.



Important: The Pipe Settings apply to codes that are set to a Feature Type of **Pipe**. To set the Feature Type, go to Edit Codes and then the General tab of the Edit Field Code Definition dialog.

When a code is tagged as a Pipe Feature, the program looks for additional parameters after the code. The parameters are separated by spaces in the point description and in the order specified in the Description Sequence in the settings dialog. For example, if code 'P' is defined as a Pipe Feature and the order of size, material and name, then a description of 'P 8 PVC 31' would define a pipe point for an 8 inch pipe of material PVC and name of 31. These pipe parameters are optional. The program will use the provided parameters and leave the rest blank. Besides using the point description, the pipe attributes can alternatively be set from GIS attributes using the attribute names setup in the dialog.

Process Settings Tab:



Create Pipe Network - when enabled, a SewerNET pipe network (.SEW) file will be created from the points connected with a pipe feature type. The **Connection Tolerance** limits the distance between points to be considered pipes belonging to the same structure. This is particularly useful when locating the actual invert for each pipe within a structure. The .SEW file can be utilized in the Sewer Network (**SewerNET**) contained in the **Hydrology** module. From there, the network can be edited, analyzed, and used for plan and profile creation.

Use Point Elevation - sets the control parameter for the point elevation. The field located point can be used as the **Rim Elevation**, the **Invert Elevation** or can be set to **None**.

Use Code Description - the code description can be used as the **Pipe Material**, **Pipe Name** or set to **None**.

Invert Measurement - when a pipe is field located directly, this control if the elevation will be used as the **Top** of Pipe or **Bottom** of Pipe.

Description Sequence - sets the order for the code descriptions as they are input into the code. The order of these setting must match the method in which the code is created. The fields that can be used and ordered are:

- Rim Elevation
- Invert Elevation
- Pipe Size
- Pipe Material
- Pipe Name
- Invert Depth
- Structure Name

For a structure with multiple inverts, pipe sizes and/or materials, the values should be in the description separated by commas. For example, for a structure with two incoming PVC pipes with sizes of 15" and 24", and inverts at depths 5.9 and 5.5, the description would be 'P 15,24 PVC 5.9,5.5'. If a value, such as the material does not change between structures, it need not be reentered.

GIS Attributes - using GIS attributes mimics the process described above. The difference is how the values are input into the pipe feature. Using the GIS attribute option requires that the data be entered as an attribute that is carried with the point coordinate file. The advantage to this method is that it does not necessitate long code description. Instead, the point attribute data can be entered in in a tabular format.

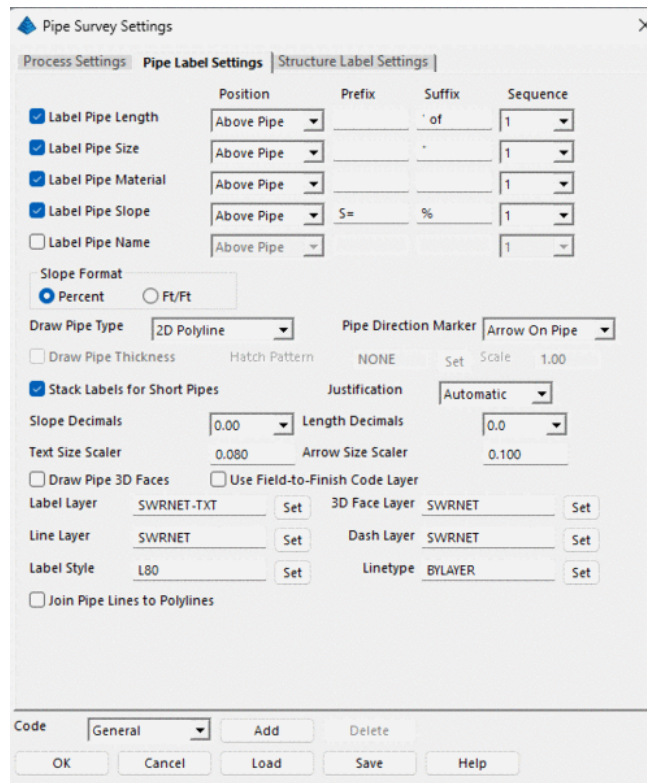
Example:

```
DEPTH=5.0  
SIZE=12  
MATERIAL=CONCRETE
```

The attributes can be created in advanced or in the field using SurvCE/PC and can be set up for prompts during the collection of the data.

Code - the Code defaults as General which would apply to all pipe feature codes. You can also Add a code that is contained in the Code Table as a Pipe Feature which can then have individual settings. For example, you could have a code called STORM for storm sewers and one called SANITARY for sanitary sewers. Each pipe network can have unique settings for layers and labels.

Pipe Label Settings Tab:



Label Pipe Length, Size, Material and Slope - all have controls over the position and content of the labels:

- Position - sets the position of the label as above or below the pipe
- Prefix - adds a prefix to the label
- Suffix - adds a suffix to the label
- Sequence - sets the sequence of the pipe label

Slope Format - can be expressed as a percentage of slope or foot per foot (ft/ft)

Draw Pipe type - sets the type of pipe to be drawn.

- 2D Polyline - draws a consistent 2D polyline at elevation 0
- 3D Polyline - draws a 3D polyline with each vertex matching the point elevation
- Double/Width - draws 2 dashed lines offset from each other the size of the pipe with no scale exaggeration.
- Dashed/Width - draws a solid and dashed line whose width is the size of the pipe with no scale



- None - does not draw line work between points
- Field-to-Finish - uses the linetype as selected in the Field to Finish Code Table

Pipe Direction Marker - draws an arrow in the direction of flow for each pipe segment. The choices are an **Arrow** on the Pipe, **Parallel** to the pipe or **None**

Draw Pipe Thickness - a pipe thickness (if available) can be drawn outside the pipe with the exception of the 2D or 3D polyline options.

- **Hatch Pattern** - an optional Hatch Pattern can be used to depict the pipe thickness. The selection and scale of the hatch pattern is also available.

Stack Labels for Short Pipes - automatically stacks the selected labels to make them look more readable.

Justification - the text justification for the labels can be set as Automatic, Left/Right or Centered.

Slope Decimals - sets the precision for the pipe slope labels

Length Decimals - sets the precision for the pipe distance labels

Text Size Scaler - sets the text size in terms of a scale factor which is applied to the horizontal scale as set in the Drawing Setup. (text scaler x horizontal scale = text size)

Arrow Size Scaler - sets the Arrow size in terms of a scale factor which is applied to the horizontal scale as set in the Drawing Setup. (text scaler x horizontal scale = text size)

Draw Pipe 3D Faces - when enabled, a 3 dimensional pipe as 3D Faces is drawn separately from the other pipe lines which can be viewed in Orbit view or Carlson's Drawing Viewer'

- **Use Field-to-Finish Code Layer** - draws the 3D pipes on the layer established in the Code Tablee

Layers for the Labels, Lines, 3D Faces and the Dashes (when using the Dashed/Width option) can be typed in or selected using the Set button.

Label Style - sets the text style for the pipe labels

Linetype - sets the linetype for the line work. This will override the linetype defined in the layer unless bylayer is selected and does not apply to the Dashed/Width option.

Join Pipe Lines to Polylines - joins all connected pipes into a single polyline. This does not work for the Double/Width or Dashed/Width pipe types

Structure Label Settings Tab:



Available/Used Labels - to setup the Pipe Structure Labels, highlight any available pipe structure labels on the left side of the dialog box and either double click or hit the Add button to move it to the Used Labels portion of the dialog box on the right.

Label Format - select the desired label format from the drop down list.

- Individual Labels - labels text at each structure location
- Data Table - creates a Data Table at each structure location. When this option is used, you are able to control the **Justification** of the table as well as the **Width** of the Labels and Values. These options only become available when the Data Table option has been selected.
- Attribute Block - adds the data for each structure to a predefined block containing the relative attributes. When this option is used, there is a Select button to select the block to be used.
- Along Pipe - labels the structure information along each respective pipe segment.

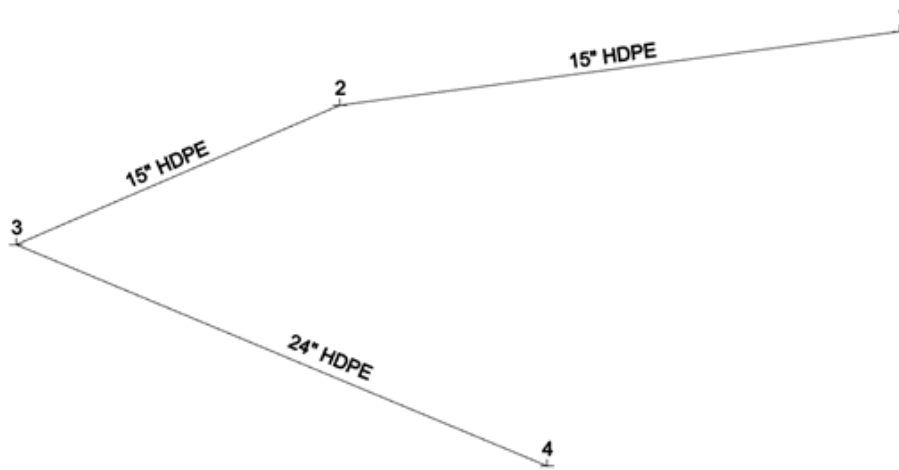
Station/Offset Decimals - sets the precision for the CL Station and Offset Labels

Elevation/Depth Decimals - sets the precision for Elevation labels of Rim, Inverts, and Depth.

Example coding of a pipe network. PIPE code is defined as a Pipe Feature. The 1st description is the pipe size. The 2nd description is the pipe material. The 3rd description is the pipe depth. Point 3 has two sizes and two depths separated by commas because two pipe connect at this point.

Point	Description
1	PIPE1 15 HDPE 4.75
2	PIPE1 15 HDPE 6.15

- 3 PIPE1 15,24 HDPE 5.96,5.5
- 4 PIPE1 24 HDPE 9.9



Example coding for 2 separate pipes using END code to separate the pipes. CUL code is defined as a Pipe Feature. 1st description is the Pipe Material and 2nd description is the pipe size. In this case, turn OFF the Create Pipe Network because the separate pipes are using the same code of CUL.

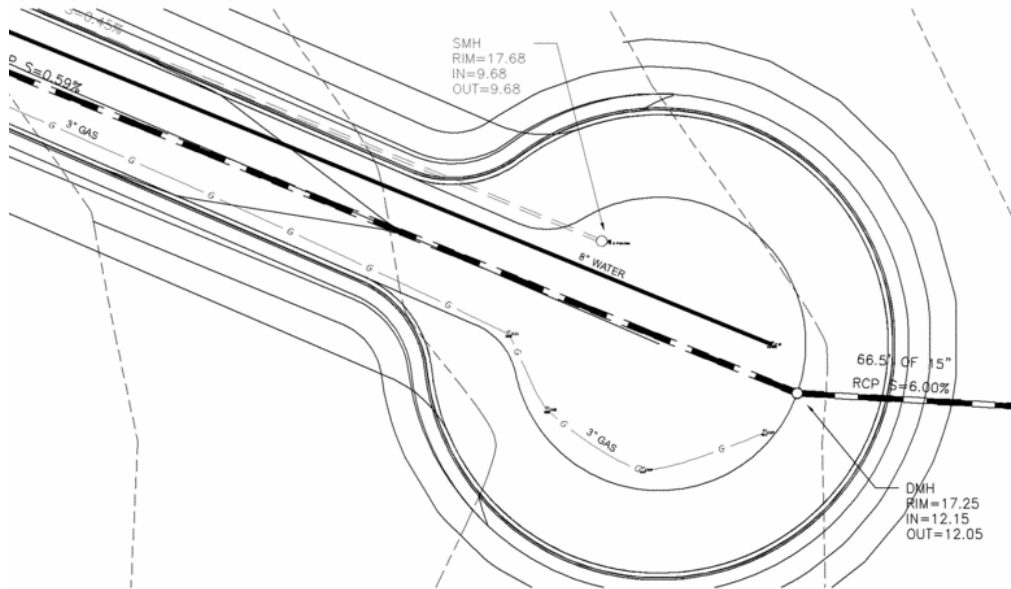
Point	Description
7	CUL RCP 18
8	CUL END RCP 18
14	CUL RCP 18
15	CUL END RCP 18

Example coding for 2 separate pipes using number suffix on codes to separate the pipes. CUL code is defined as a Pipe Feature. 1st description is the Pipe Material and 2nd description is the pipe size. In this case, the Create Pipe Network can be applied. The separate pipes are using separate codes of CUL1 and CUL2.

Point	Description
7	CUL1 RCP 18
8	CUL1 RCP 18
14	CUL2 RCP 18
15	CUL2 RCP 18

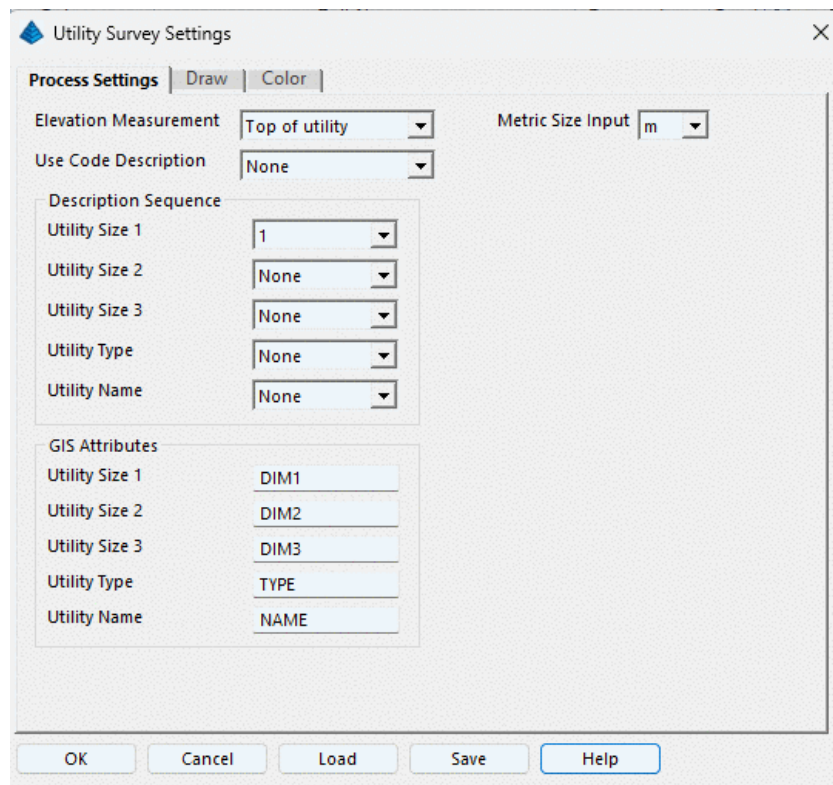
Utility Setup:

The Utility Feature works similar to the Pipe feature. The differences with utility are supporting different pipe shapes (circle, rectangle, etc), different types of utilities (gas, electric, etc), and not having manhole structures. For each utility type, there are settings for pipe shape, color, labels and drawing 2D Polylines, 3D Polylines and 3D Faces.



Important: The Utility Settings apply to codes that are set to a Utility Feature. To set the Feature Type, go to Edit Codes and then the General tab of the Edit Field Code Definition dialog.

Process Settings Tab:

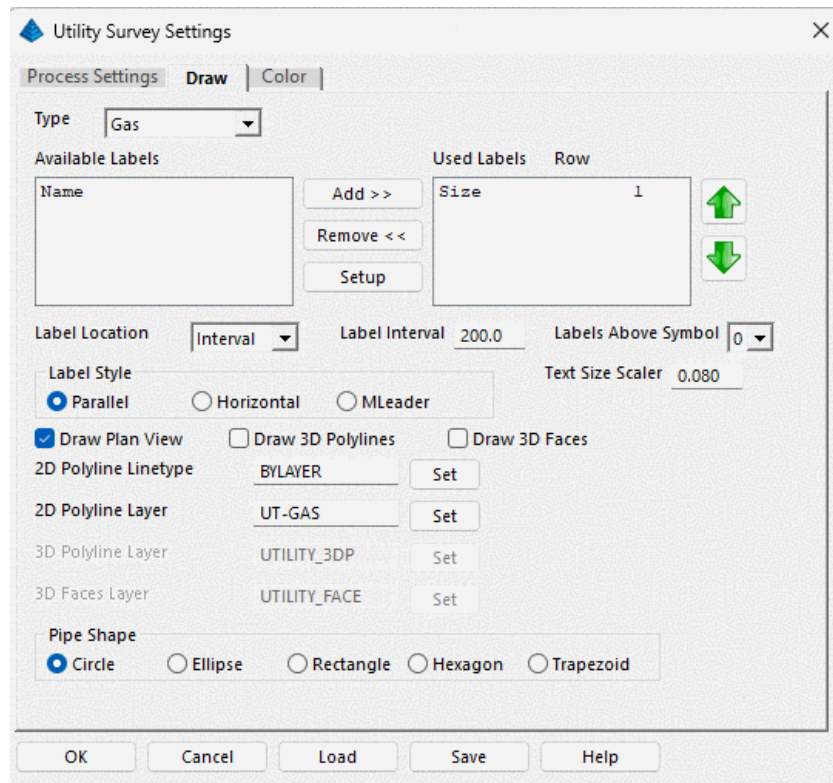


Elevation Measurement - sets the reference for the point location as either Top or Bottom of the pipe

Description Sequence - Sets the order of the utility size in the code description

GIS Attributes - using GIS attributes mimics the process described above. The difference is how the values are input into the Utility feature. Using the GIS attribute option requires that the data be entered as an attribute that is carried with the point coordinate file.

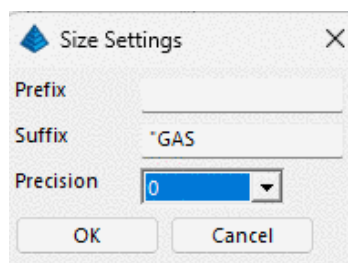
Draw Tab:



Type - selects the type of utility for the draw settings. There is a pull down menu with include utility types of:

- Gas
- Water
- Sanitary
- Storm
- Communication
- Electric
- Irrigation
- Fire
- Utility Duct
- and Other for user defined utilities

Setup - allows the additional control of adding prefix, suffix and the size precision to the utility line label. This can also be accomplished by double clicking the label field.



Available/Used Labels - to setup the Utility Line Labels, highlight any available field labels on the left side of the dialog box and either double click or hit the Add button to move it to the Used Labels portion of the dialog box on the right.

Label Location - sets the location of the label along the utility line

- None - does not add a label to the line
- Ends - adds a label to each end of the utility line
- Interval places a label at a distance interval
 - Label Interval is the distance in CAD units to place the label
- Both - places a label at each end as well as at intervals

Label Style - controls the placement of the text label for the size and/or name of the utility.

- Parallel - places the label parallel to the utility pipe
- Horizontal - places the label horizontal to the screen view, with respect to a Twist Screen if present
- Mleader - creates a Mleader that points to the utility line

Draw Options - are toggles to select options for the entities to be drawn. One or more may be selected. The options are:

- Draw Plan View - will draw the 2D line work as set in the code table with respect to the options herein
- Draw 3D Polylines - draws a 3D polyline between points with each vertex matching the point elevation
- Draw 3D Faces - creates 3D Faces in the shape and size of the pipe shape selected

2D Polyline Linetype - sets the linetype for the 2D polyline

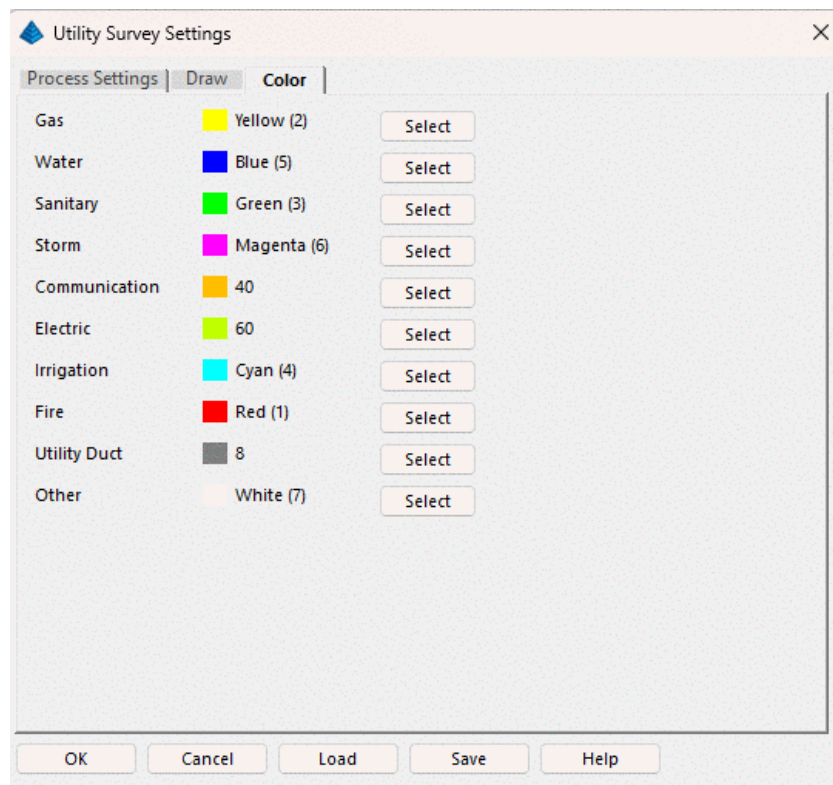
2D Polyline Layer - sets the layer for the 2D polyline to be placed

3D Polyline Layer - sets the layer for the 3D polyline to be placed

3D Faces Layer - sets the layer for the 3D faces to be placed

Pipe Shape - select the shape of the pipe to be created. These pipe shapes affect the 3D Faces created if that option is enabled.

Color Tab:



This dialog box contains the option to set individual colors for each utility type.

Symbol Library - accesses the Symbol Library also contained in the Settings Menu

Edit Points: - accesses the Edit Points Menu also contained in the Points Menu

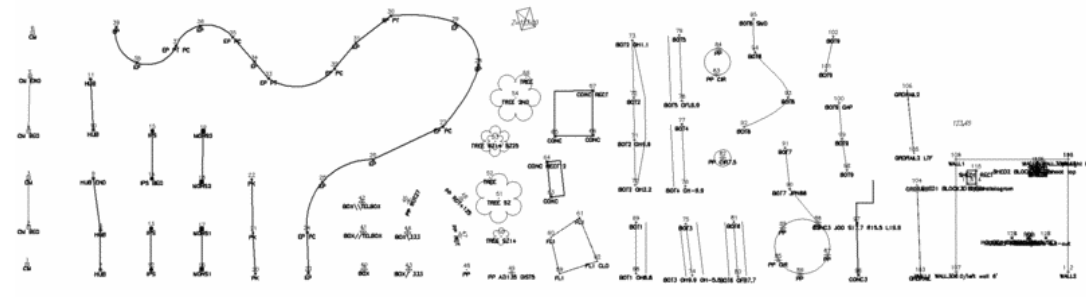
Report Codes/Points - Option to Report Codes and Points from the current Coordinate file and/or Field Code Table

Help - accesses the Help Menu

Exit - exits the Field Code Table Editor

Coding Examples

Under the Carlson Projects folder, there is an example that shows the different ways for linework coding along with examples for many of the special codes. The examples are in **f2f_example.crd** and **f2f_example.fld**. To see this example, create a new drawing, set to the supplied coordinate file and process using the supplied field code table.



Here is a breakdown of the features that the points above illustrate.

Point 1: Point Entity by itself

Points 2-3: Using Begin code to start a line; end line using Begin code for next line

Points 4-5: Using Begin and End to start and stop linework

Point 6: Point Entity by itself after End code

Points 7-11: Linework by code defined as Polyline entity type; using End as break between linework

Points 12-15: Linework by code defined as Polyline entity type; using Begin as break between linework

Points 16-19: Linework by code defined as Polyline entity type; using # after code instead of Begin/End to separate linework

Points 20-22: Linework by code defined as Polyline entity type without using Begin/End to start/stop linework

Points 24-26: 3 point curve using on PC code

Points 27-30: 3+ point curve using PC/Point codes

Points 32-33: 2 point tangent curve using PC/Point codes

Points 35-39: reverse curve using PC/Point codes

Point 40: Regular point without extra description

Point 41: Using // to use a code description as a suffix

Point 42: Using \ to use a code description as a prefix

Point 43: Using / to append a description

Point 44: Using \ to add a description as a prefix

Point 45: Using ROT and a Point# to rotate to that Point#

Point 46: Using ROT and a value to set the rotation

Point 47: Using ROT by itself to rotate to the next Point#

Point 48: Regular point without rotation

Point 49: Using AZI and DIST codes to offset the point

Point 50: Using SZ with value to set size of symbol

Points 51-52: Using SZ by itself to size symbol by the distance to the next point

Point 53: Using SZ with 2 values to draw multiple symbols at those sizes

Points 54-55: Using 2ND code to size the symbol

Points 56-58: Using 2ND and 3RD codes to size the symbol in 2 dimensions

Points 59-62: Using CLO to close the linework

Points 63-64: Using RECT with two points and a value to create a rectangle

Points 65-67: Using RECT with three points to create a rectangle

Points 68-69: Using OH to offset right a fixed amount

Points 70-73: Using OH on multiple points to offset various amounts

Points 74-75: Using multiple OH on the same point to offset polyline multiple times

Points 76-77: Using OH with negative value for offset to left

Points 78-79: Using OFL with value for offset left a fixed amount

Points 80-81: Using OFB with value to offset both left and right a fixed amount

Point 82: Using CIR to draw circle at specified radius

Points 83-84: Using CIR to draw circle using two points for center and perimeter

Points 85-89: Using CIR to draw best-fit circle through points on perimeter

Points 90-91: Using JPN to join linework to another Point#

Points 92-95: Using SMO to create smoothed linework

Points 96-97: Using JOG to create additional linework segment extensions

Points 98-102: Using GAP to create a break in the linework

Points 103-106: Using LFT to switch linetype to left side

Points 107-109: Using WALL3D with specified height value

Points 110-112: Using WALL3D with height from 2nd point

Points 113-115: Using BLOCK3D with height and three points to define parallelogram

Points 116-123: Using BLOCK3D with height and multiple points to define perimeter

Points 124-128: Using FACE3D with multiple points to make a surface

Points 129-132: Using HOLE3D with multiple points to define the perimeter of a hole in the FACE3D surface

Point 133: Using code definition with Attribute Format set to Text and only Elevation turned on with Label Decimal

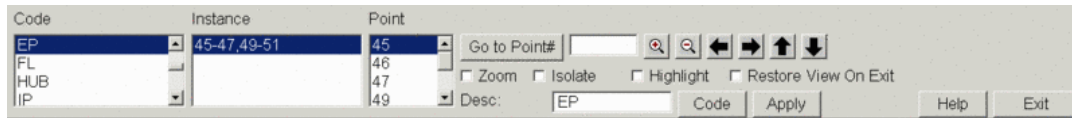
Menu Location: Survey

Keyboard Command: fld2fin

Prerequisite: A data file of points with descriptions

Field to Finish Inspector

This command reviews entities in the drawing created by Field To Finish. Point descriptions can be edited and the drawing is updated for both the point symbols and linework, using the Field To Finish coding.



Field to Finish Inspector docks a control panel dialog at the bottom of the screen which leaves the drawing view at the top.

Code: Lists the field codes that were found in the drawing. Clicking on a code causes the Instance list to show all of the linework and points that use the selected code.

Instance: Lists the linework and points of the currently selected code.

Point: Lists the points that make up the currently selected linework or point in the Instance list.

Go to Point#: Type in the point # to see in the drawing and then press this button to bring the point # into view. If the point # is not in the drawing, then a message will be printed at the top of the dialog box. If the Zoom toggle is on, then the point # will be brought to the center of the screen even if it was already visible on the screen. Likewise, if Isolate or Highlight are on, then those options will be applied, too.

Zoom: Check this checkbox to make the Field to Finish Inspector automatically zoom and pan the drawing so that the selected items in the above lists are viewable. Zoom is used on the Code and Instance lists. Pan is used for all three lists.

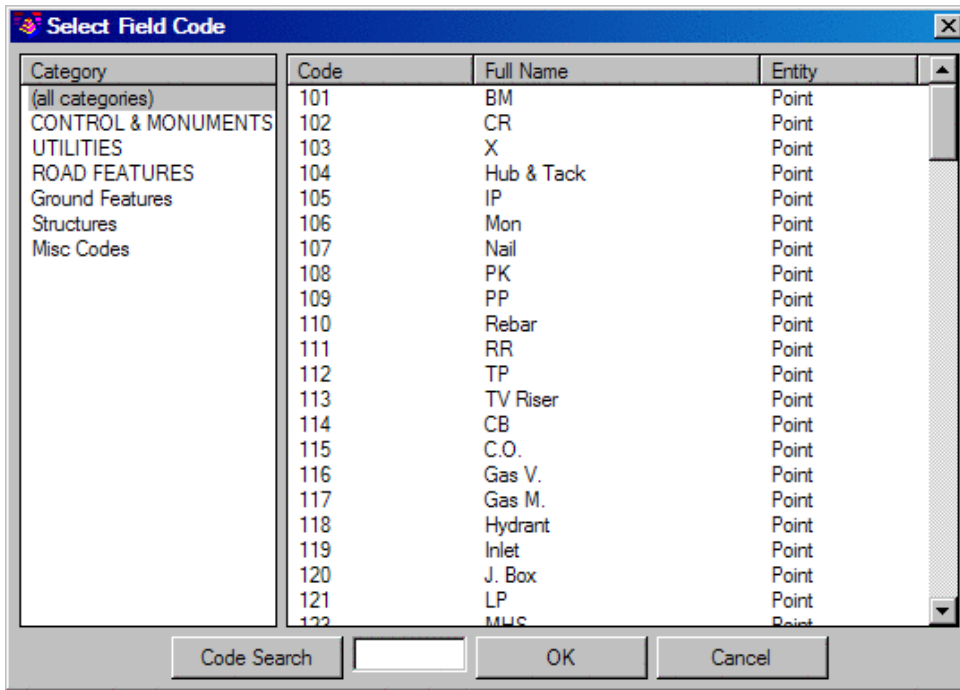
Isolate: Check this checkbox to make the Field to Finish Inspector only display the selected items in the above lists.

Highlight: Check this checkbox to make the Field to Finish Inspector highlight the selected items in the above lists.

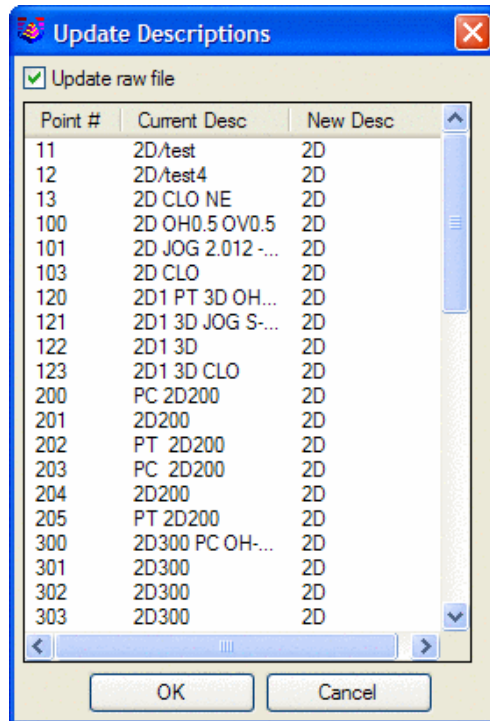
Restore View On Exit: Check this checkbox to make the Field to Finish Inspector restore the zoom and pan values when you exit.

Desc: This edit box will display the description field from the coordinate file used on the given point(s). If the points do not all have the same description in the coordinate file, **varies** is displayed instead. If you type in a new description and then click on Apply, the new description will be applied to the coordinate file and Field-to-Finish will be used to process the coordinate file and update the drawing, including linework. Press the Code button to place an existing field code into this Desc edit box.

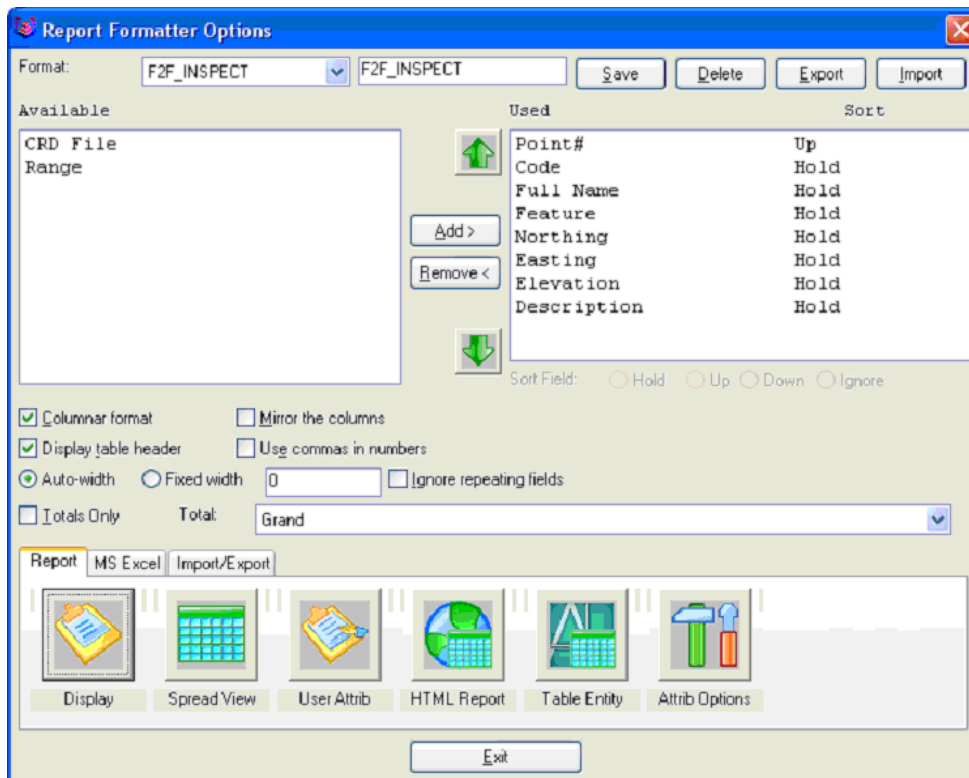
Code: Press this button to select a field code from the current field code definition (FLD) file. The following dialog box is an example of what you will see. The Categories on the left are the categories that are defined in the current field code definition file. The list on the right is all of the field codes in the selected category. Select (*all categories*) to see all of the codes in all of the categories. The selected field code will be placed in the Desc edit box.



Apply: Press this button to apply the modified description that is in the Desc edit box to the currently selected points. The below dialog box will come up that lists exactly what will be changed. Optionally, the raw file that was used to create the coordinate file will be updated as well. Press OK to continue. The description will be updated in the coordinate file and then Field-to-Finish will be used to process that coordinate file and finally the drawing will be updated to reflect the changes.



Code: This button generates a user-defined report with fields for the point number, coordinate, feature name and code. This report uses the Report Formatter to select which fields to include.



SAMPLE REPORT

Point# Code Full Name Feature Northing Easting Elevation Description

1 CM CONCRETE MONUMENT POINT 4922.730 5570.695 502.510 CM/(4" DIAM)

2 CM CONCRETE MONUMENT POINT 4739.612 5499.121 506.050 CM

5 SHED SHED POLYLINE 4794.880 5495.289 505.110 SHED

6 SHED SHED POLYLINE 4771.855 5486.661 505.530 SHED

7 SHED SHED POLYLINE 4782.648 5457.861 505.820 SHED

11 18D 18" TREE POINT 4889.990 5551.491 503.010 18D/OAK

Pulldown Menu Location: Survey

Keyboard Command: f2f_inspect

Prerequisite: Entities created by Field-to-Finish

Edit Field to Finish Codes

This command edits a Field to Finish code table. The program prompts for the .fd file to edit and then runs the code table editor. See the Field-to-Finish section of the manual for a description of the code table.

Pulldown Menu Location: Survey

Keyboard Command: editf2f

Prerequisite: None

Enter Deed Description

This command lets you enter line and curve data which is drawn and annotated as entered. When entering in data, the bearing quadrant and bearing value is input on the same line. For example, a bearing of N45-10-30E would be entered as 145.1030, where (1) represents the NE quadrant. The numeric codes for the quadrants are 1-4 beginning

with NE as (1) and continuing sequentially in a clockwise direction to the NW quadrant (4). Distance data can be entered in Varas, Meters, Poles, Chains or US Feet. In addition to typing in the angle and distance numbers, there is a Select option at the command line which prompts to select an existing text label that the program will read to get the angle or distance.

Curve data can be entered for Non-Tangent, Reverse-Tangent and Tangent curves. Data used to define curves includes but is not limited to Tangent Out Bearing, Radius data, Chord Bearing, DeltaAng and Tangent Length. Prompting for curve data is determined by what curve definition data is used. When you are finished, the closure and area of the figure is reported.

The program has the option to Undo the previous data entry in case you need to re-enter values. Also, the program auto-saves the data entered during the command so that if the command is canceled and restarted, there's an option to resume entering data. The command starts with the dialog shown here.

Line and Curve Layer: Specify the layer name for lines and arcs.

Points Layer: Specify the layer name for the points.

Traverse by: Select between entering bearings, azimuth, gons or point numbers. The points option recalls points from the current coordinate (.CRD) file. The prompt option adds a prompt for each angle to specify the angle format.

Point Format: Choose between creating Carlson points in the coordinate (.CRD) file at each point in the figure, drawing descriptions only or having no point labels.

Apply Rotation: This option adjusts the entered bearings by the specified rotation.

Label Lines and Arcs: Specify whether the annotation should be drawn on the lines and arcs or should be added to line and curve tables. The settings for the label styles are defined by the Annotate Defaults and Auto Annotate commands. Please see those commands in the manual for a description of those settings. You can either specify specific settings files from those commands or use the current settings which is the default.

Deed Name: Specify the beginning deed name. Only available when Store to Deed File is checked on.

Draw Linework: Specify whether or not to draw linework, if this is disabled then all annotation options are disabled also.

Create Polyline: This option creates a polyline of the deed perimeter instead of individual line and curve entities.

Prompt for Descriptions: Specify whether or not the program should prompt you for point descriptions. If this is not checked, then point descriptions are blank.

Prompt for Elevations: Specify whether or not the program should prompt you for point elevations. If this is not checked, then point elevations are set to zero.

Prompt for True Point of Beginning: With this option, the traverse begins before the TPOB. When the traverse reaching the TPOB, enter the POB option at the command prompt to indicate the true point of beginning and start the traverse loop.

Plot Point Symbols: If the Point Format is set to Descriptions Only or None, this option is available. It will place point symbols without creating points in the coordinate (.CRD) file.

Create Radius Points: When checked, radius points will be created for arcs. Radius points are given the description RADPT.

Store to Raw Data (.RW5) File: When checked, data entered will also be written to a raw data (.RW5) file that can be opened using the *Edit-Process Raw Data File* command. This file can be used to perform coordinate adjustments. The Compass rule, Crandall rule, Transit rule, Angle balance adjustment and Least-square adjustment routines are all available. See *Edit-Process Raw Data File* for more information.

Store to Deed File: When checked, data entered will be written to a deed (.PDD) file. This file can be processed later to correct errors, create deed reports or to redraw the deed. To use this option, set the deed file name by picking the Specify File Name button. Also set the Deed Name field.

Prompts

Pick point or point number: 1

```
PtNo. North(y) East(x) Elev(z) Description
1      8000.00 12000.00 0.00
```

In this example the coordinate for point number one has already been stored in the current coordinate (.CRD) file with the *Draw-Locate Points* command.

Undo/Exit/Curve/Select/<Bearing (Qdd.mmss)>: 145.3035

Varas/Meters/Poles/Chains/Select/<Distance(ft)>: 210.5 Enter P to input a distance in Pole format or C for Chains format.

Undo/Exit/Curve/Select/<Bearing (Qdd.mmss)>: C Enter C to traverse through a curve.

Tangent-out/Radius: R

Radius: 1103.5

Curve direction (Left/<Right>)? press Enter for right

Non-tangent/Reverse-tangent/Chord/Delta angle/Tangent/<Arc length>: N If the curve is non-tangent to the previous leg then enter the arc length, enter C for a chord length, D to enter the delta angle or T to enter the tangent distance. In this example we have a non-tangent curve so we entered N.

Curve direction input [<Chord>/Radial]?: C

Chord Bearing (Qdd.mmss): 245.2341

Length of Chord: 201.22

Undo/Exit/Curve/Select/<Bearing (Qdd.mmss)>: 345.3218

Varas/Meters/Poles/Chains/Select/<Distance(ft)>: 209.28

Undo/Exit/Curve/Select/<Bearing (Qdd.mmss)>: 445.2348

Varas/Meters/Poles/Chains/Select/<Distance(ft)>: 200.54

Undo/Exit/Curve/Select/<Bearing (Qdd.mmss)>: E Enter E to end the prompting and calculate the closure error.

Closure error distance> 1.35251089 Error Bearing> N 70d41'35" E

Closure Precision> 1 in 607.63 Total Distance Traversed> 821.82

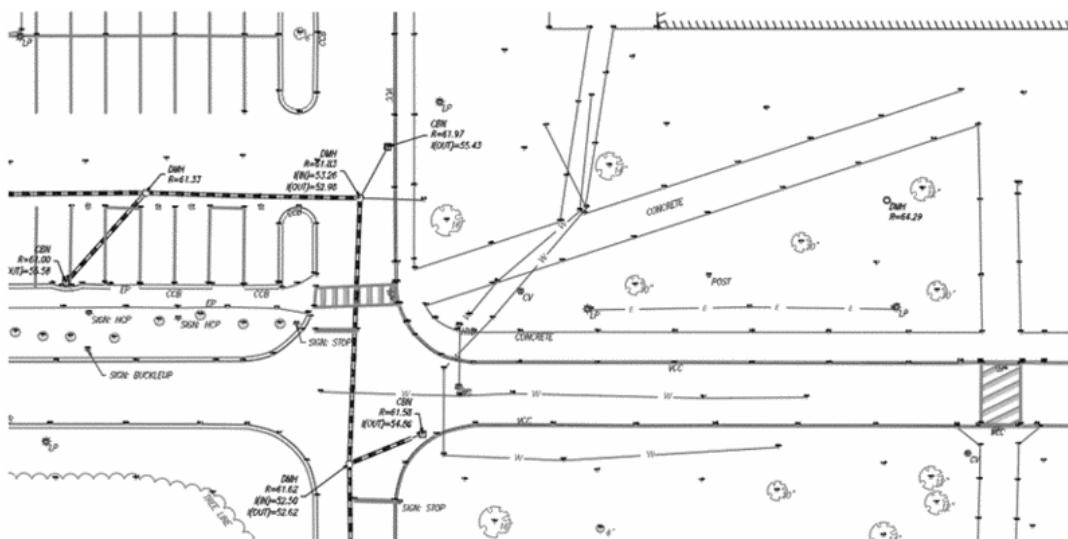
Pulldown Menu Location: Survey

Keyboard Command: PDD

Prerequisite: None

Draw Field to Finish

This command turns data collector field notes into a final drawing by matching the descriptions of the field points with user-defined codes. The points are brought into the drawing with attributes defined by the code, including the layer, symbol, size and linetype. Essentially, Field to Finish drafts a plan as you survey.



Example of a survey processed with Draw Field to Finish

Draw Field to Finish	Create/Edit a Field Code	See Some Examples
Create a Code/Point List	Use The Tree Feature	
Work with Code Tables	Use the Pipe Feature	
Use Special Codes	Use the Utility Feature	

Two main files are used in Draw Field to Finish - a coordinate file (**CRD/CRDB**) and a field code definition file (**FLD**). The coordinate file consists of point#, N,E,Z coordinates and text description fields. The description field contains the codes for the Draw Field to Finish processing. An ASCII data file can be converted into a coordinate file using the *Import Text/ASCII File* command. The field code definition file defines the layer, symbol, size and other actions to apply with each code. These file names are displayed at the top line of the Draw Field to Finish dialog box.

Field to Finish is an extremely versatile and robust program. The concept however is straight forward. Each point located in the field is coded with a description. That point description is related to a matching code in a Code Table (**FLD**) that instructs the Field to Finish program on how to process and draw it. Points can be placed in a drawing

using specific symbols, on particular layers and annotated with text or block attributes. A series of points can be used to draw 2D and/or 3D polylines for use in drafting or creating surface models. Beyond a large variety of settings for each defined code in the code table, there are also Special Codes that modify the behavior of the code, such as PC (point of curvature) and PT (point of tangent).

Field to Finish is a very powerful tool but it is not required to place points from a coordinate file into a drawing. *Draw-Locate Points* provides a simpler method for drawing points but it is much less versatile.

Using a drawing template: Field-to-Finish will "layerize" the points and linework according to the code definitions. If the layers to use are not already defined in the drawing, Field-to-Finish will create any layers defined in the code table when it processes that code. It will however, randomly assign a color, linetype etc. to that layer. To provide consistency and have the same properties for these layers in all your drawings, it is recommended you use a prototype drawing, or Template (DWT). Each new drawing created can then be started using this template which contains all layers needed matching those in the Code Table (FLD) file. See Carlson Configure -> Startup Settings to select a prototype Template.

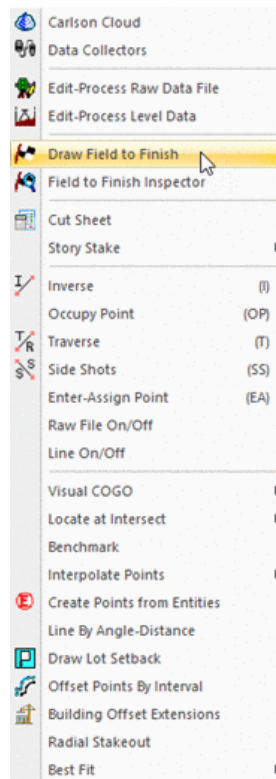
Connecting Line Work: If a code is created as a Line, 2D Polyline or 3D Polyline, all codes with that same description will be connected with no extra coding needed. For example if the code EP is defined as a 3D Polyline and points 100-110 all have EP as a description, Field to Finish will draw a 3d Polyline connecting all points 100 through 110. Stopping and starting line work is where additional coding is needed. There are basically two different methods for connecting linework and controlling the start/stop or gaps between line segments.

1. **Adding group numbers:** As described above the linetype is defined by the code as either lines, 2D polylines, 3D polylines (breaklines), or both 2D and 3D polylines. To control the connection of the linework, distinct lines with the same code are defined by adding a group number to the end of the code name in the data file. With this method, all points with the description CODE1 will be one line while points with CODE2 will be another line. Both CODE1 and CODE2 use the definition for CODE. For example, the code EP could be a code for edge of pavement that is to be connected as 3D polylines. If there are two separate edge of pavement lines on the left and right sides of a road, all the points for the left side could have the description EP1 and the points on the right side could be EP2. Besides having the number after the code, the number can be used as a prefix by defining the code with a # special character. For example, when the code is defined as #CODE, then the points with descriptions 10CODE and 20CODE get matched to this code.
2. **Adding Special Codes:** The second method is to add a special code to the end of a line segment or the beginning of a new line segment. For this method, you use the same code for all similar line segments and just instruct Field to Finish when to stop drawing. For example the end of an edge of pavement line would be coded EP END. (The code END is the default end line special code). Alternatively, you can instruct Field to Finish to start a new line segment by adding a beginning special code IE. EP BEG. The program will realize that a new line segment should not be connected to a previous one and will create an appropriate gap.

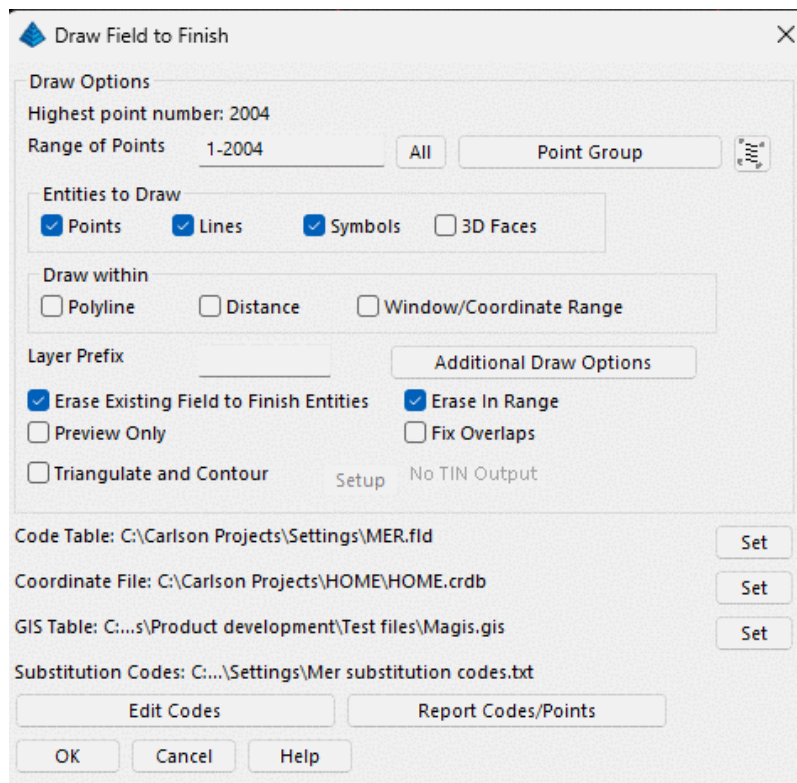
Legacy PointCAD format: This method also connects points with the same code. The difference is that instead of using a number after the code for distinct lines, you use the same code with an additional code for starting and ending the line. For example, +0 is used to start a line and -0 to end. So the coding for a segment of edge of pavement could be EP+0, EP, EP, EP-0. Another special code that has been added to Field to Finish is +7, -7. This 7 code will use the linetype definition of line, 2D polyline or 3D polyline defined by the Draw Field to Finish code. For example, if EP is defined as a 3D polyline, then the coding EP+7, EP, EP, EP-7 will create a 3D polyline. Otherwise codes like +0, -0, which is defined as start and end line, will draw EP as a line. Other PointCAD special codes are: +4 starts a curved 2D polyline, *4 starts a closed curved 2D polyline, +1 begins a 3-point arc, +5 starts a 3D polyline, *5 starts a closed 3D polyline, +6 starts a 2D polyline, *6 starts a closed 2D polyline, +7 starts a line whose type is specified by the field code definition, -05 starts a curved 3D polyline section, -50 ends that section, +8 starts a 2D and 3D polyline combination, *8 starts a closed 2D and 3D polyline combination, -08 starts a 2D and 3D polyline combination curved section, -80 ends that section. //, followed by a field code, concatenates that field code's description on to the point's description. For example, OAK//04 might become LIVE OAK TREE 4" if the

field code OAK translates to LIVE OAK TREE and the field code 04 translates to 4".

Draw Field to Finish is found in the **Survey** Pull-down Menu



Draw Field to Finish



Range of Points: Specify the range of points to draw.

Point Group: Specify the point group(s) to process.

Point List: Select points from a point list dialog box.

Entities To Draw: The Points option draws only the points and point attributes. The Lines option draws only the linework and the Symbols draws only the symbols. Any combination of these options can be processed as well as individual processing of each entity.

Draw Within: These options are methods to filter the points to draw. The Polyline method prompts for a closed polyline and only draws points inside this polyline. The Distance method uses a specified center point and distance to only draw points within this circle. The Window/Coordinate Range prompts for lower left and upper right points to define the rectangular area to draw points.

Layer Prefix: Optional layer prefix added to all entities drawn with Draw Field to Finish.

Erase Existing Draw Field to Finish Entities: When checked, this option will erase from the drawing any old entities created by previous Field-To-Finish runs before drawing the new entities.

Erase In Range: This option only erases and redraws those Draw Field to Finish entities that are within the specified range of points to process.

Preview Only: When checked, this option will temporarily draw the points and linework and allow you to review it with zoom and pan.

Fix Overlaps: This option checks the points drawn by Field-to-Finish for any point labels that overlap with other points or linework. For any overlaps, the point labels can be automatically moved or you can step through each overlap to decide how to handle it. See the command called Fix Point Attribute Overlaps in the Points chapter for more information on this feature.

Triangulate and Contour: This option uses the points and 3D polylines created by Field-to-Finish as input for

building a surface in Triangulate and Contour. Pick the Setup button for the options like contour interval and TIN file to create.

Code Table: Sets the FLD file to process which contains the code definitions.

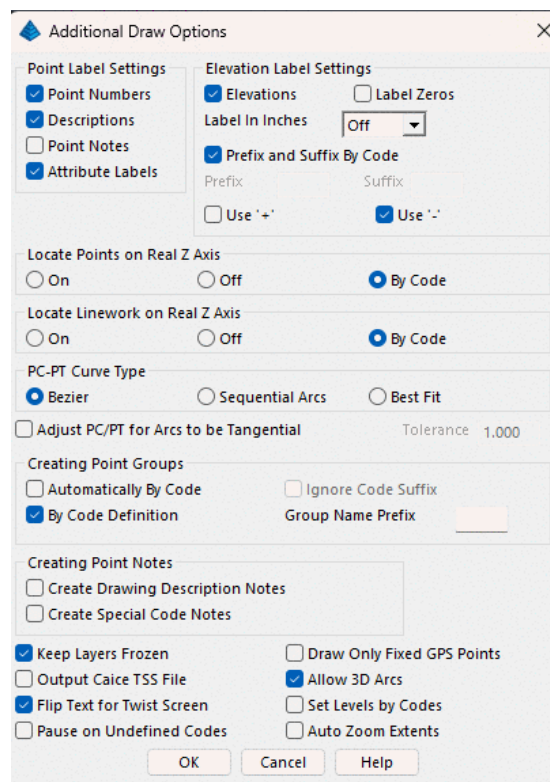
Coordinate File: Sets the CRD file to process which contains the point numbers, coordinates and descriptions.

GIS Table: Sets the GIS file which defines the GIS feature and attribute names. This file is optional and is used in GIS processing.

Edit Codes: Opens the **Field to Finish** Edit Code dialog box.

Report Codes: Opens the Report Code dialog box.

Additional Draw Options:



Point Label Settings: Specify whether you want Draw Field to Finish to label the **Point Numbers**, **Descriptions**, or **Points Notes** which are contained in the note (.NOT) file that is associated with the coordinate (.CRD) file. The **Attribute Labels** option controls whether or not to draw attribute labels. These attributes can be the description and/or additional comments in the form of block attributes or text and may or may not be associated with a GIS feature.

Elevation Label Settings: Specify the elevation labeling options. The Elevations toggle controls whether to draw the elevation attribute label for the points. The Label Zeros option will label the elevations of points with z=0. The Prefix and Suffix By Code sets whether to use the general Prefix and Suffix for all point elevation labels from this dialog or use the Prefix and Suffix defined in each code definition. Use '+' and Use '-' will place the appropriate symbol in front of the elevation. The Label In Inches option allows for elevation label in feet and inches instead of in decimals.

Locate Points on Real Z Axis: Choose between locating all the points at real Z elevation, all at zero elevation or to use the real Z setting as defined in the individual codes (**By Code**). These options can be used to override the setting contained in the FLD as part of the code and is global in nature. It is usually recommended to use the By Code option.

Locate Linework on Real Z Axis: Choose between drawing all the linework at real Z elevation, all at zero elevation or to use the 2D/3D polyline setting as defined in the individual codes (**By Code**). These options can be used to override the setting contained in the FLD as part of the code and is global in nature. It is usually recommended to use the By Code option.

PC-PT Curve Type: Sets the method for drawing curves with more than 3 points. The **Bezier** option draws a smooth polyline through all the curve points. The **Sequential Arcs** method draws multiple arcs with arc end points at each of the curve points. These arcs are tangent to the preceding line segment. The **Best Fit** method creates a single best-fit curve for all the curve points between the PC and PT.

Adjust PC/PT for Arcs to be Tangential: This option will adjust the PC and PT polyline vertices to make the curve tangential. The program will only adjust these points if the adjustment distance is less than the specified tolerance. This option applies to cases where the tangents are well defined and the PC/PT are harder to survey exactly.

Creating Point Groups: Point Groups can be created in one or two different ways. Each field code definition can specify Point Group(s) that all point numbers that use that code will be added to. Multiple field codes can use the same Point Group name. Check the By Code Definition checkbox for that option. The second method is to automatically create Point Groups for each code that is processed. Check the Automatically By Code checkbox for that option. Ignore Code Suffix, if checked, will cause the codes to be considered after removing the numeric suffix. For example, points with the EP10 and EP11 codes will both be automatically added to the Point Group named EP. No matter how the Point Group is created, the Group Name Prefix can be used to add a prefix to the group name. Note: if the Point Group already exists, it will be erased first before being created again by either of these two methods.

Creating Point Notes: These options append point notes to the coordinate file data for some of the data fields processed by Field-to-Finish. These notes can then be used by other commands like List Points to report these fields. For example, this enables List Points to report both the point coordinate file description as well as the point drawing description as generated by Field-to-Finish.

Keep Layers Frozen: This option will not thaw layers that Field-to-Finish uses for drawing entities. So when you have this option on and some layers are frozen, then you won't see the entities that Field-to-Finish creates until you thaw those layers.

Draw Only Fixed GPS Points: This option applies to coordinate files created by GPS measurements. The program checks for GPS status records and only uses points that have GPS solution status of Fixed.

Output Caice TSS File: This option creates a TSS file from the processing results.

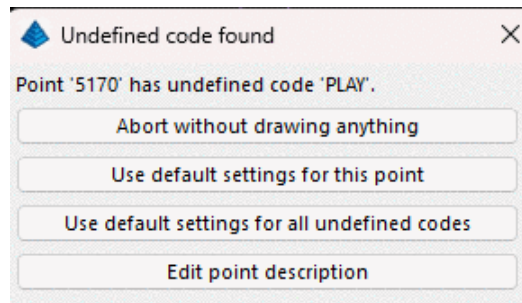
Allow 3D Arcs: When creating polyline arcs with the PC special code, this option checks whether to create the arc in 3D for tilted or vertical arcs such as a doorway or bridge arch. Otherwise, only the 2D coordinates are used to define the curvature.

Flip Text for Twist Screen: This option will rotate the point labels and symbol by 180 degrees when needed to make them right-side up readable relative to the current twist screen drawing view. This option applies to the Rotate To Line and Rotate special code (ROT).

Set Levels By Codes: This option assigns the optional Level Names for use by selection filters.

Auto Zoom Extents: When checked, this will force a zoom extents after Draw Field to Finish is done.

Pause on Undefined Codes: When checked, Draw Field to Finish will pause if it encounters a description that is not defined in the code table.

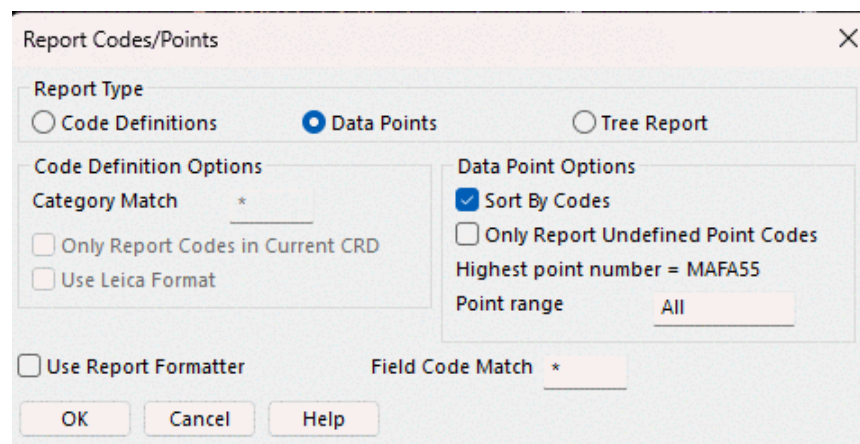


The options for handling undefended codes are:

- **Abort without drawing anything:** This stops the command. Run Draw Field to Finish again to correct the code table.
- **Use the default settings for this point:** This option draws a point using the current Point Default settings with no linework. To set your own default, define a code called "SC_DFLT" or "DEFAULT." This default code will instruct Field to Finish on how to process undefined codes.
- **Use default settings for all undefined codes:** This option will draw all undefined codes using the current Point Default settings with no linework. To set your own default, define a code called "SC_DFLT" or "DEFAULT." This default code will instruct Field to Finish on how to process undefined codes.
- **Edit Point Description:** Use this option to edit the point description or select a different code.

Report Codes/Points

This routine prints the code table or the data file to the screen, file, or printer. A useful option here is to print the data file (CRD Points) and choose Sort by Codes which will group the data points by distinct codes.



A useful option here is to print the data file (CRD Points) and choose Sort by Codes which will group the data points by distinct codes.

Report Type:

- Code Definitions: Creates a list of code definitions from the currently selected FLD file.
- Data Points: Creates a list of points grouped by codes of the current CRD (coordinate file)
- Tree Report: Creates a list of all points coded as a Tree Feature.

Code Definition Options

- Only Report Codes in Current CRD: Reports only the codes from the current FLD that are contained in the current CRD (coordinate file)
- Use Leica Format: Creates a list of code definitions using the Leica comma delineated format.

Data Point Options

- Sort by Codes: Sorts the list of points by code name
- Only Report Undefined Codes: Restricts the list to report only those points that have descriptions not contained in the code table. This is useful for reviewing a data set before processing.
- Point Range: Enter the range of points to be included in the list.

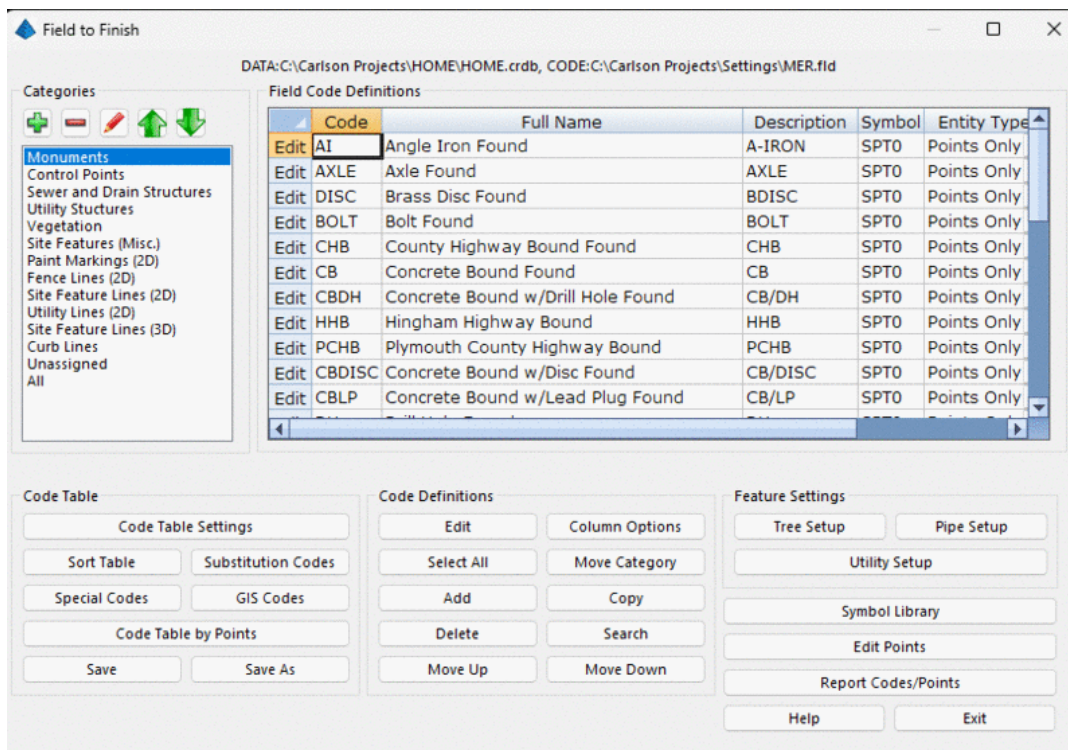
Use Report Formatter: Enables the option to use the Carlson Report Formatter for more control of the creation of the point list.

Field Code Match: Option to filter and limit the list to only select codes.

Edit Codes

Selecting Edit Codes takes you to the Field to Finish Code Table dialog box. This dialog box contains access to all of the tools and options for Field to Finish. It is the heart of this program.

The **Field to Finish** dialog box allows you to load the coordinate and field code definition file. Here you can add, view and edit the code definitions, view and edit the coordinate file, create reports, and then return to the Draw Field to Finish dialog box to process the files. The top section displays the code definitions. The bottom section has three columns of functions each pertaining to controls for different elements of the command. The **Code Table** section provides controls for settings, sorting and reporting of codes. The **Code Definitions** section provides tools for the creation and editing of codes. The **Feature Settings** section provides controls for the special tree, utility and pipe feature types.



The code table editor has a list of categories and a spreadsheet of codes with optional displays of related definitions. The spreadsheet shows the codes for the currently highlighted category. The category toolbar buttons allow you to add, remove, edit the names and change the order of the categories. There are two fixed categories. The **Unassigned** category shows any codes with blank categories. The **All** category shows all the codes. You can control which fields are visible in the spreadsheet by using the **Column Options** button. You can make edits to the fields in the spreadsheet or highlight a row and pick the **Edit** button to bring up a dialog to edit the code.

This section of the manual is separated into three categories:

Code Table

Code Definitions

Feature Settings **Quick Links for this section:**

Code Table Settings

Sort Table

Substitution Codes

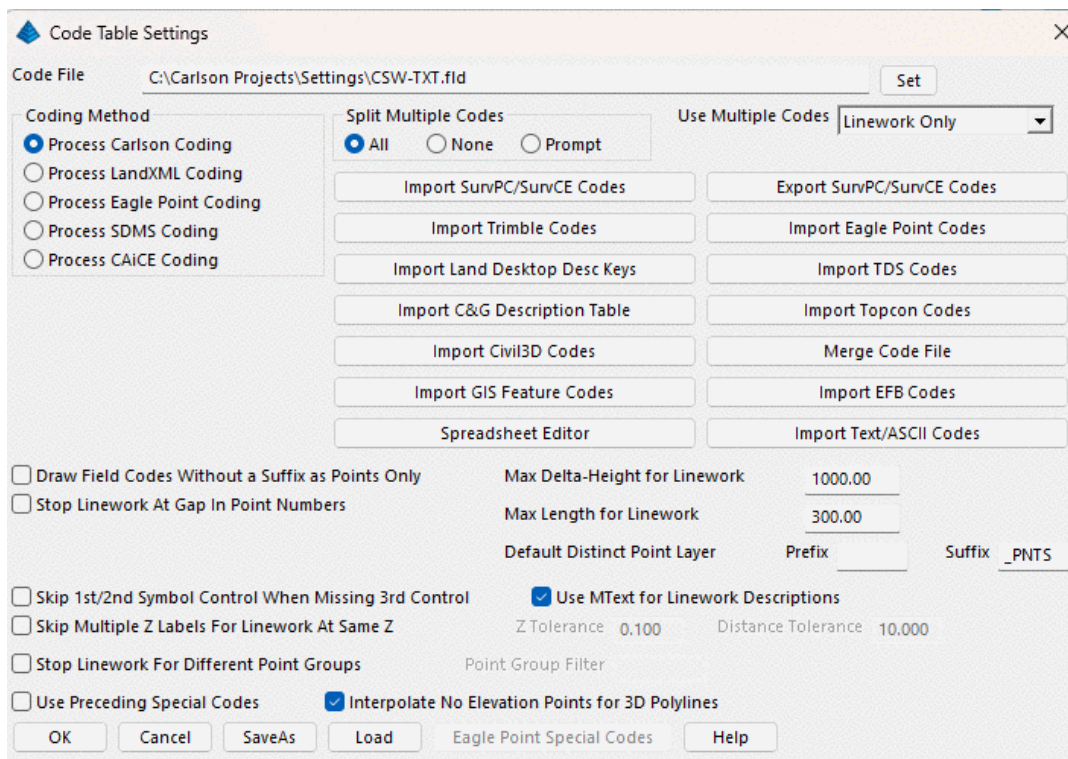
Special Codes

GIS Special Codes

Code Table by Points

Code Table Settings:

These options provide tools for defining the coding method to be used for processing of the point data. Various import tools allow for the importing of codes from different software packages. Controls for handling multiple codes are located on this dialog. All special codes can be replaced to other characters defined by the user. The special codes are listed and edited on this dialog.



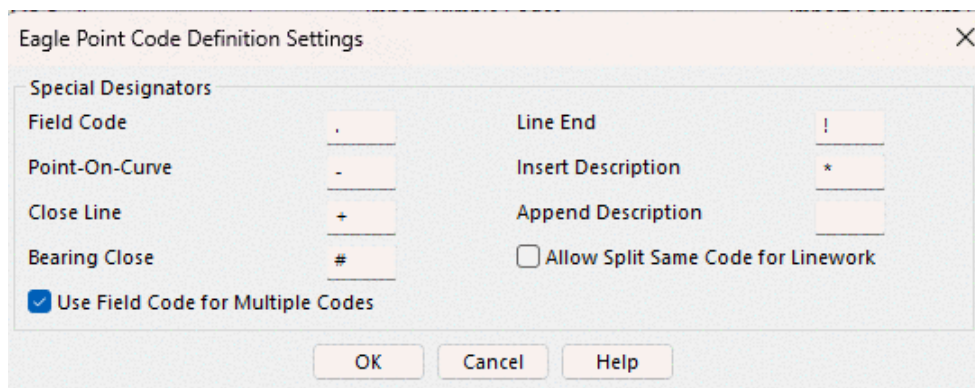
Set: Choose this button to select an existing or create a new code table. The name of the current table is shown in the field to the left of this button.

Coding Method:

Carlson Coding: When checked, this option interprets and processes coordinate files based upon the Carlson Coding method and data collection method.

LandXML Coding: This method prompts for a LandXML or Leica/Hexagon HeXML file to use with the processing. This method applies when the point descriptions do not have linework coding and the LandXML file has PlanFeatures that define how points are connected to create linework. This method reads the LandXML file to find the points that begin and end lines and curves.

Eagle Point Coding: When checked, coordinate files are processed based on the Eagle Point Data Collection method. When selected the *Eagle Point Codes* button at the bottom of the dialog box becomes available for selection and displays the following dialog. This dialog allows for customization of the eagle point special designators.



Currently the supported designators include, "Field Code", "Point-On-Curve", "Close Line", "Bearing Close", "Line End", "Insert Description" and "Append Description". Also supported is the ability to recognize overwriting of

descriptions just as Eagle Point does by using the space separator instead of the "Insert Description" designator. Examples of supported coding are as follows:

.TC Places a node and or line per the field code library.

TC Places a node and or line per the field code library.

-TC Specifies a point on a curve.

TC- Specifies a point on a curve.

..TC Stops the line.

TC! Stops the line.

.TC+ Closes the line back to the starting point.

TC+ Closes the line back to the starting point.

.TC# Typically coded on the third corner of a rectangle to close the figure with having to locate the fourth corner.

TC# Typically coded on the third corner of a rectangle to close the figure with having to locate the fourth corner.

WV.WI Places a node as specified by the code "WV" in the field code library and then begins a line as specified by code "W" in the field code library.

.TC.EP.FL Results in three lines coming together.

TC1.TC2.TC3 Results in three lines coming together. All three lines are specified by the definition of the single code "TC" in the field code library.

TC.TC1 When used in conjunction with the "Draw Field Codes Without a Suffix as Points Only" toggle, "TC" will be recognized as the node and "TC1" will be recognized as the line so that if the code "TC" in the field code library is defined as a polyline, line or 3D polyline, duplicate lines will not be unintentionally placed when this shot only pertains to a single element. Keep in mind that all line work must have a numeric suffix when using this toggle.

TREE * OAK Result on screen would be: TREE OAK

TREE OAK * Result on screen would be: OAK TREE

TREE OAK Result on screen would be: OAK

TC1!.TC2-.VLT6# Stops "TC1", continues "TC2" as a point on a curve and closes VLT6 as a rectangle using the "Bearing Close" code.

Note: The use of the "Use Multiple Codes for Linework Only" toggle is recommended when using Eagle Point Coding.

SDMS Coding: This option processes coordinate files based upon SDMS coding method. When active, the program will prompt for an SDMS .PRJ file to process.

CAiCE Coding: When checked, coordinate files are processed based on the CAiCE Data Collection method. Examples of supported coding are as follows:

169 is just the code 169.

145C10 is the code 145 and line #10.

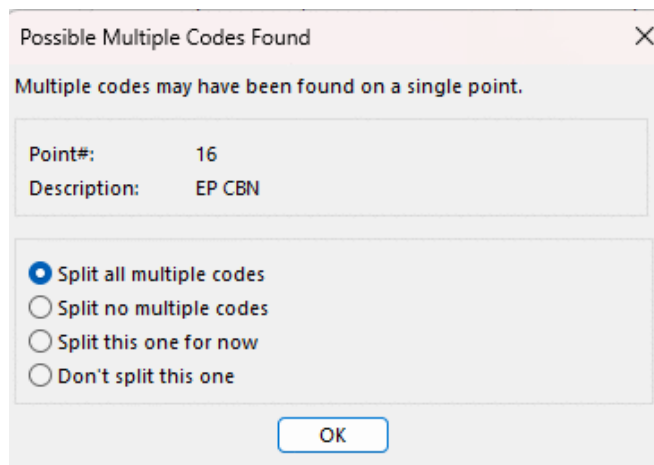
169C25C is the code 169, line #25, and the point is on a curve.

172C12B is the code 172, line #12, and this point closes the line.

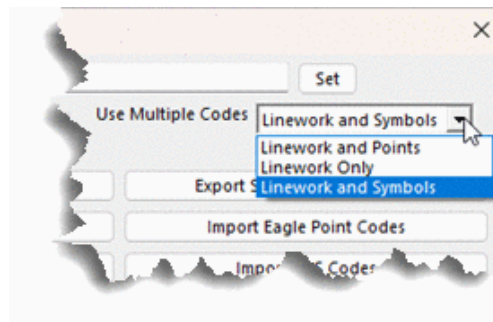
Split Multiple Codes:

Multiple codes are defined by including each code in the point description field separated by a space. A single data point can be used in different lines by assigning it multiple codes. For instance, a point might be part of both a curb line and a driveway line with a description of "CURB DRW". Field-to-Finish uses spaces as the delimiter for multiple codes. You should avoid spaces in the descriptions except for where multiple codes are intended or after the "/" character. For example, a code for light post should not be "LGT POST" but instead should be "LGTPOST".

There are three options for the handling of multiple codes when encountered. The **All** option will split all multiple codes and process each code based upon its code definition. When **None** is select the first code in the description string will be processed and the remaining characters left a part of the description (this is exclusive of any Special Codes used). If the **Prompt** option is checked on, when Field-to-Finish detects multiple codes on a point the following dialog will be displayed with options for handling the codes.



Use Multiple Codes:



Multiple Codes are simply more than one code separated by a space (default setting). Field to Finish has options to allow more control over the use of these multiple codes. This setting has three options:

- **Linework and Points** - Uses the space to connect multiple line codes as well as creating multiple point blocks at each location.
- **Linework Only** - Limits the multiple coding to be used for connecting line work only.
- **Linework and Symbols** - Uses the space to connect multiple line codes as well as creating multiple symbols at each location.

Import/Export Options:

Import SurvCE Codes: This option imports a SurvCE Feature Code List (fcl) into a Carlson Field to Finish (fld) code definition file.

Import Trimble Codes: This option imports Trimble .FXL file codes into the Carlson Field to Finish (fld) code definition file.

Import Land Desktop Desc Key: This option imports and converts a Land Desktop Description Key into a Carlson Draw Field to Finish (fld) code definition file. The Land Desktop Description Key file is a mdb file and is found in the Land Desktop Project file path. It is located in the under the COGO/DescKey directory.

Import C&G Description Table: This option imports C&G code tables (tbl) into the Carlson Field to Finish (fld) code definition file.

Import Civil 3D Codes: This option imports Civil 3D .fdb_xdef file codes into the Carlson Field to Finish (fld) code definition file.

Import GIS Feature Codes: This option imports features in a .GIS file from Define GIS Features into F2F codes.

Export SurvCE Codes: This option creates a SurvCE Feature Code List (fcl) from the current a Carlson Field to Finish (fld) code definition file.
Import TDS Codes: This option imports TDS codes into the Carlson Field to Finish (fld) code definition file.

Import TDS Codes: This option imports TDS codes into the FLD code definition file.

Import Eagle Point Codes: This option imports Eagle Point codes into the Carlson Field to Finish (fld) code definition file.

Import Topcon Codes: This function imports Topcon codes and layers from an XML file.

Merge Code File: This function adds code definitions from another (fld) code definition file into the current code table. The program shows a list of the codes from the other file that are different than the current code table. You can choose which codes to import. Any codes that conflict with an existing code definition are defaulted to not import.

Import EFB: This option imports the SDMS Electronic Field Book codes (.xml) into the Carlson Field to Finish (fld) code definition file.

Import Text/ASCII Codes: This option imports code definitions from a user-defined format. Each row in the text file should represent one code. The program will prompt for the delimiter (ie. comma separated) that is used in the text file and then for the field type for each of the columns (ie. "Layer" or "Description").

Spreadsheet Editor: This function lets you edit all the codes with all their settings in a single large spreadsheet. This method can be a handy way to use copy and paste to edit the codes. This method can also be quicker to make many changes than navigating the Edit Code dialog. This method is for expert users and you need to take care to make valid inputs.

Code	Description	Symbol	Symbol	Layer	Entity	Tie	Linetype	Text Size	Line	Processin	Nearest	Full Name	
1	CALC	SPT2	0.2000	PNT-STAK	0	1	BYLAYER	0.0200	0.0000	1	0	Calculated	
2	DEFAULT	SPT15	0.2000	PNT	0	1	BYLAYER	0.0200	0.0000	1	0	Default Sh	
3	Category	Monuments											
4	AIRON	A-IRON	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Angle Iron
5	ROS	RING OF STONES	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Ring of St
6	AXLE	AXLE	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Axle Foun
7	BD	BDISC	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Brass Disc
8	BOLT	BOLT	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Bolt Foun
9	CHM	CHM	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	County Hi
10	CM	CM	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Concrete
11	CMDH	CM/DH	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Concrete
12	CMLP	CM/LP	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Concrete
13	DH	DH	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Drill Hole f
14	HT	HT FND	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Hub and T
15	STK	STK FND	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Hub and T
16	IP	IP	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Iron Pipe
17	IROD	IROD	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Iron Rod f
18	LCD	LC DISC	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Land Cou
19	USCG	USC&G DISC	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Land Cou
20	LPEP	LP/EP	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Lead Plug
21	MN	MAG	SPT10	0.0400	PNT-MON	0	1	BYLAYER	0.0800	0.0000	1	0	Mag Nail

Additional Options:

Draw Field Codes Without a Suffix as Points Only: This option is useful for when wanting to use a field code sometimes for linework and sometimes for just points but it is preferred to number the lines rather than using start

and stop codes. For example, if the field code EP is defined to use the Line Entity type, then EP25 will be drawn as a Line, however if just EP is used, no linework will connect to that COGO point. When using this option, it is required to add a numerical value after each line code.

Stop Linework At Gap In Point Numbers: This option is a method for controlling the start and stop of drawing linework. This method will automatically stop linework where there is a gap in the point numbers for the linework code. For example, if there are points with code EP then points with code CL then more points with code EP, the EP linework for the first set of EP points will stop at the last EP before the CL points and then new EP linework will start after the CL points.**Max Delta-Height for Linework:** Use this option to specify the maximum elevation difference that Draw Field to Finish should draw any section of linework. This option is for use with 3d polylines and lines.

Max Length for Linework: Specify the maximum length that Draw Field to Finish should draw any section of linework. If two codes are distant beyond this limit, no linework will be drawn between them.

Default Distinct Point Layer: These settings control the default layer for when the Distinct Point Layer option is turned on for each code definition.

Skip 1st/2nd Symbol Control When Missing 3rd Control: For multi-point symbols when the 3rd control point is missing, this option centers the symbol on the 1st point instead of using the 1st point as a control point.

Use MText for Linework Descriptions: Controls whether to create MText or regular Text entities for linework description labels.

Skip Multiple Z Labels For Linework At Same Z: This option labels only the first point elevation for points on polyline or rectangle from the RECT code when the elevation difference between the points is less than the specified **Z Tolerance** and the horizontal distance is less than the **Distance Tolerance**. For example, for points on the corners of a level utility pad, this option will label the pad elevation just once.

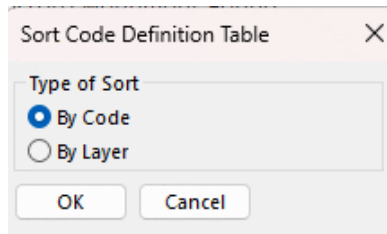
Stop Linework For Different Point Groups: This option applies to when you have points for the same job collected at different times or with different crews and you want to prevent linework connecting points between them. For example, this applies when you survey a job with two crews and combine their points into a single CRD and both crews used "EP1". In this case, to keep each set of "EP1" separate, you can put each set of points into a separate point group with Point Group Manager and use this Field-to-Finish option. The **Point Group Filter** is optional and is used to make the program only check groups that match the filter. A point can be in multiple groups such as "CONTROL" and "DAY1". Some groups could be used for different days such as "CONTROL". So the Point Group Filter could be set to "DAY" to filter out the common groups and only process the time related groups.

Use Preceding Special Codes: This setting tells the program to expect the special codes before the main code. For example, if "BEG" is the special code for begin linework and "EP" is a main code, then the program looks for "BEG EP". The default sequence is for the special code to come after the main code as in "EP BEG".

Interpolate No Elevation Points for 3D Polyline: For points tagged as "No Elevation" that are part of a 3D polyline, this option will interpolate an elevation for this point from the other points in the 3D polyline. Otherwise, this point will be skipped in the 3D polyline.

Sort Table

This sorts the code table by either code name or layer.



Substitution Codes:

This option defines a lookup table for translations of the raw point descriptions. This translation is done as a pre-processing step before the regular Field-to-Finish processing. For example, if you had a substitution setup for "25" = "EP", then a point description of "25" would get translated to "EP" and then this would be processed with Field-to-Finish.

This works for both numerical and alpha raw descriptions.



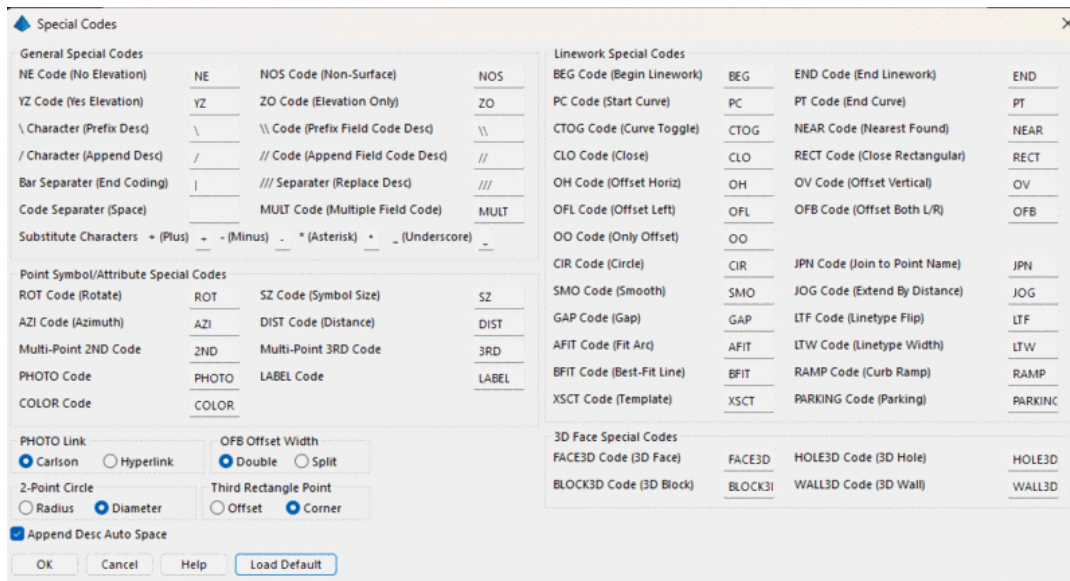
- The **Match With Number Suffix** option will match with a number following the string. For example, "CL" will match with "CL2". Otherwise, the strings have to be an exact match. The substitution codes are stored in a file named f2f_sub.dta in the %appdata%\Carlson...\USER folder. You can also use the **Import** and **Export** functions to load and save substitution codes to a comma separated text file.

Special Codes:

Special Codes are essentially modifiers of an original code. They perform additional operations and/or offer addi-

tional control over the original code. These special codes work with line work, point data and symbols. There are also options to control the placement of labels and add additional text to attributes.

Below shows the default dialog box containing the Special Codes. These can all be modified at the users discretion but the must be unique and not conflict with any codes defined in the code table.



A special code is placed after (or before if that option is enabled) the regular code with a space separating the code and special code. For example if an edge of pavement shot was taken at the beginning of a curve, it would be coded EP PC, whereas the PC tells Field to Finish to begin drawing an arc. Special codes may be strung together to modify a single code or added to multiple codes within the same string. For example, if a ling string ended at the PT of a curve it could be coded EP PT END whereas both PT and END are the special codes. An example of multiple codes combined with special codes could be a driveway intersecting the edge of pavement at the beginning of a curve: EP PC DRV SMO. The PC starts drawing an arc on the edge of pavement and the driveway begins with a smoothing code. It is important that any special code modifying another code must be placed immediately after the code it is modifying.

Quick Links for this Section

- General Special Codes
- Linework Special Codes
- Point Symbol/Attribute Special Codes
- 3D Face Special Codes

General Special Codes

NE: No Elevation - This special code represents no elevation. A point with this special code is located at zero elevation and the elevation attribute is set to None.

NOS: Non-Surface - This special code indicates that the point should be "non-surface"; that is, that it should be ignored when contouring or creating surfaces. This can also be controlled per-field code by turning on the Non-Surface toggle in the Edit Field Code Definition dialog box. When used, this special code tags the point to be non-surface but maintains the elevation attribute.

YZ: Yes Elevation: - This special code locates overrides the code setting for non-surface and locates the point and

maintains the elevation attribute.

ZO: Elevation Only - This special code represents elevation only (Z-Only). A point with this special code is used as part of a 3D polyline for elevating the 3D polyline without effecting the horizontal position of the polyline. For example, this code could be used on a grade break point along a curb where only the elevation should change and not the horizontal alignment.

"\": Prefix Description - This special code takes the part of the description after the "\" and puts it as the prefix before the point description. For example, a data point with the description "TR \ 24ft" and a "TR" code definition with a description of "Tree" would be drawn with a description of "24ft Tree".

"\\": Prefix Field Code Description - This special code causes text after the "\\" to be interpreted as a field code. That field code's description is then prefixed to the first field code's description. For example, if the field code O2 has the description 2" and the field code OAK has the description *oak tree*, then OAK\\O2 will result in the point having the description of 2" *oak tree*.

"/": Append Description - A special character "/" (the forward slash or divide key) can be used for an unprocessed description to append. Everything after the "/" is added directly to the point description and is not considered a code and no further substitution is done on it. For example, a data point with the description "UP/#150" for a code UP that has UP for its description would be drawn "UP #150".

"/": Append Field Code Description - This special code causes text after the "/" to be interpreted as a field code. That field code's description is then appended to the first field code's description. For example, if the field code O2 has the description 2" and the field code OAK has the description *oak tree*, then O2//OAK will result in the point having the description of 2" *oak tree*. If the "/" character has been replaced with a different character, for example with a & character, then the "/" code would become "&&".

"—": End Coding - The bar separator indicates the end of coding. Everything after the bar is ignored for Field-to-Finish processing. This is useful if additional information is needed for a point but no action should be taken and no description should be added.

"/": Replace Description - This special code takes the part of the description after the "/" and uses it as the point description label.

" ": Code Separator - The default for separating multiple codes is a space. This can be modified to another character if desired. Caution should be taken when modifying special codes to be sure there are no conflicts with other codes or special codes.

MULT: Multiple Field Code - This code applies when the Split Multiple Codes under Code Table Settings is set to None and you want to override this setting and explicitly split selected codes. Multiple codes apply to points with dual code definitions for drawing two different style points or for connecting different linework to the same point. For example, if a point is both a sidewalk and driveway corner, then the point description could be "SW MULTDR".

Substitute Characters - The characters (*, -, +, /, and _) can be used and substituted in Draw Field to Finish. The way these characters are used is that when the file is processed the description field is searched for these characters. If the "+" symbol was changed to "-" then the program would look for "-" and change it to "+". This is useful when a particular data collector may not have all the symbols available. With these substitutions you can make a character that is provided on the data collector generate the symbol needed. Multiple characters can also be used. For example "-" can be used to in order to produce a "/" character or any of the characters listed above.

Linework Special Codes

BEG: Begin Linework - This special code denotes the beginning of a new line string. If a code is already designated to be a line, 2D or 3D polyline, then Field to Finish will connect like codes without the need for the BEG code. However, BEG can be used to designate the start of a new line string or can be used to force linework

between codes designated as points only.

END: End Linework - This special code denotes the end of a line string. This is used when line numbers are not being used. See Connecting Line Work for description.

PC: Start Curve - This code begins a three point arc or a curved line when used with the "PT" code (see below). The point with this special code is the first point on the arc. The next point with the code is considered a point on the arc, and third point with the code is the arc endpoint. The PT code is not necessary when coding a 3 point arc.

Example of 3 point curve:

- 10, 500, 500, 0, EP PC - start curve
- 11, 525, 527, 0, EP - second point on curve
- 12, 531, 533, 0, EP - end point of curve

For curves beyond 3 points, the **PT** code needs to be used for the last point on the curve. The routine then looks back for the PC and "fits" a curve between all intermediate points.

PT: End Curve - This code can be used with "PC" to define a curve with more than three points or a tangent two-point curve. Starting at the point with the "PC", the program will look for a "PT". If the "PT" is found, all the points between the "PC" and "PT" are used for the curve which is drawn as a smoothed polyline that passes through all points and only curves the polyline between points. If no "PT" is found, then the regular three point arc is applied as explained above. If no points are found between the "PC" and "PT", then the point prior to the "PC" and the point after the "PT" are used to create tangents for the resulting curve.

Both the PC and PT codes used for drawing arcs are subject to the PC-PT Curve Type found in the Additional Draw Options.

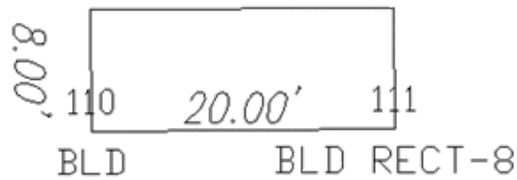
CTOG: Curve Toggle - This special code toggles curve mode on and off. Instead of using PC to start a curve, you can use CTOG. Likewise, instead of using PT to end a curve, you can use CTOG.

NEAR: Nearest Found - This special code sets the current polyline to Nearest Found connection order. This applies to codes that have the Connection Order set to Sequential and you want to override this setting to Nearest Found for the current polyline. This is useful in situations when a point on a polyline was taken out of sequential order.

CLO: Close - This code forces the lines drawn between a series of points with the same code to close back to the first point with the same code. For example, shots 1-4 all have the BLD description with the exception of point 4. Its description is BLD CLO. This will force the linework drawn for the BLD code to close back to point 1 which is the first point with the description of BLD.

RECT: Close Rectangular - The "RECT" special code creates a rectangle as a 2D or 3D polyline using one of two different methods.

- *2 Point Method*: Locate 2 points for one side of a structure and add the code RECT plus a dimension for the remaining dimension of the rectangle. For example, you are locating a shed with the dimensions 20x8. Take a shot at each corner on the 20' side and code the second shot with the 8' dimension. A positive dimension offsets to the right and a negative dimension offsets to the left. First code: **BLD** Second Code **BLD RECT-8**



- 3 Point Method: Locate 3 corners of the structure and code the 3rd shot as RECT. This tells Field to Finish to form a parallelogram out of the 3 shots taken by copying the 2 sides located respectively.



Offsets: OH, OV, OFL, OFB, OO

The codes "OH" and "OV" stand for offset horizontal and offset vertical. These offset codes apply to 2D and 3D polylines. A single set of offset codes can be used to offset the polyline a set amount. For example,

10, 500, 500, 100, EP OH2.5 OV-.5

11, 525, 527, 101, EP

12, 531, 533, 103, EP

This would create a polyline connecting points 10,11 and 12 and an offset polyline with a 2.5 horizontal and -0.5 vertical offset. The direction of the horizontal offset is determined by the direction of the polyline. A positive horizontal offset goes right from the polyline direction and a negative goes left. The horizontal and vertical offset amounts apply starting at the point with the offset codes until a new offset code or the end of the polyline. Only one horizontal and vertical offset can be applied to 2D polylines. For 3D polylines, multiple offset codes can be used to make a variable offset. For example,

10, 500, 500, 100, EP OH2.5 OV-.5

11, 525, 527, 101, EP OH5.5 OV-.75

12, 531, 533, 103, EP OH7.5

This would offset the first point horizontal 2.5 and vertical -0.5, the second point horizontal 5.5 and vertical -0.75 and the third point horizontal 7.5 and vertical -0.75.

Multiple Offsets: When there are multiple "OH" codes for the same point, the polyline is offset multiple times, one for each OH code.

Alternatively and in conjunction with the "OH" code, a second number after the offset amount repeats the offset by that number. For example "OH2 5" means offset by 2 and repeat 5 times. This multiple offset option is useful drawing stair steps.

The "OFL" code stands for offset left horizontal. The only difference with the "OH" code is that you don't have to enter the "-" to go left.

The "OFB" code stands for offset both left and right horizontal. For example, if the points follow the center of a ROW, the OFB code can be used to create the left and right edges of the ROW. There is a setting in Special

Codes for Offset Both for whether the offset value is for the full or half width between the two offset lines. Also, there is a setting in each code definition under the Linetype tab for whether to create a closed polyline from the left and right offset polylines. If 3D polylines are offset, they maintain the elevations of the original located 3D polyline.

The "OO" code stands for offset only. This option only draws the offset polylines and not the polyline connecting the points. The "OO" code is a companion code that can be combined with any of the other offset codes. For example, "EP OH2 OO" will offset the EP line horizontally by 2 but draw only the offset polyline.

CIR: Circle - The "CIR" special code causes the point to create a circle in one of three different ways.

- *Single Point and Radius method:* The first way uses just the current point as the center with the CIR special code followed immediately by the radius. For example "CIR7.5" will create a circle centered on this point with radius 7.5 and at the elevation of the current point.
- *Two Point method:* The second method uses two points, the first point specifying the center and the elevation, and the second point specifying the radius. Only the first point has the "CIR" special code and the second point is the next point with a matching field code. Another variation of method 2 is using 2 points that are on the perimeter and define the diameter. For the 2 point method, whether the points define the radius or diameter is defined on the Code Table Settings > Special Codes > 2-Point Circle.
- *3 Point method:* The third method uses 3 or more points that specify the perimeter of the circle in 2D with the first point specifying the elevation. For this method, the "CIR" special code is only on the first point and the rest of the points are the next points with matching field codes.

The "CIR" code can be used with all of the linetypes including "points only". The circles are always parallel to the X-Y plane unless the code linetype is set to "3D Polyline". Then the circle is drawn as a 3D polyline. Any active linework for the code is ended before processing the "CIR" special code.

Examples of drawing circles:

PointNo. Description

Method 1 (Single point at center with radius value)

82 PP CIR7.5

Method 2a (Point at center plus point at perimeter)

83 PP CIR

84 PP

Method 2b (2 Points on perimeter that define the diameter)

83 PP CIR

84 PP

See 2-Point Circle for options to designate methods 2a or 2b.

Method 3 (Points on perimeter)

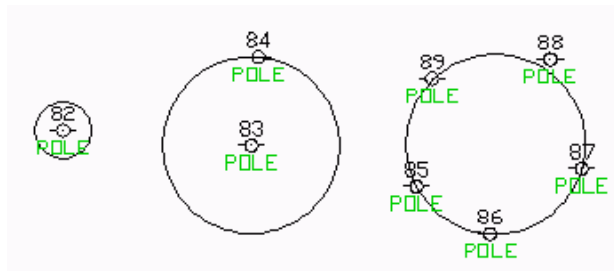
85 PP CIR

86 PP

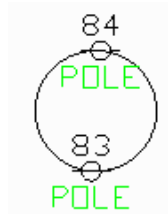
87 PP

88 PP

89 PP



Methods 1, 2a, 3



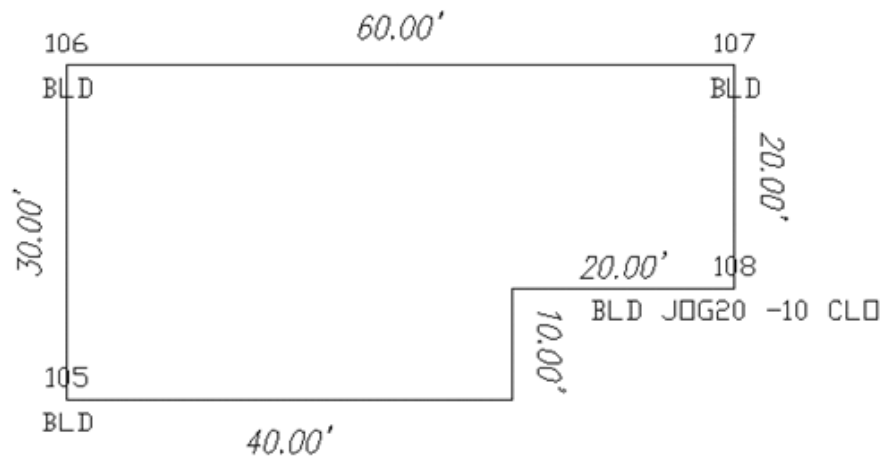
Method 2b

JPN: Join to Point Name - The "JPN" (Join to Point Name) special code joins to the point named immediately after the code. For example, "JPN205" causes a line to be drawn from the current point to the point "205". JPN is designed to work for adding a segment at the start of linework, but can be used at any point during the survey.

SMO: Smooth - This code is used to smooth the polyline. It is ideal for long winding roads or walkways that are generally curved as it prevents the need of adding PCs and PTs to define the arc segments. Tree lines and landscape areas are also good examples of utilizing this special code.

JOG: Extend By Distance - The "JOG" special code allows for additional points to be inserted into the line work at perpendicular or straight offsets. Only offsets should follow the JOG code. Positive numbers indicate a jog to the right and negative numbers indicate a jog to the left. Alternatively, "R#" and "L#" can be used where # is the distance to either the right or the left. Finally, "S#" can be used to make an offset straight ahead by using a positive # or behind by using a negative #. For example, "BLDG JOG S10.1 R5 L12.2 L5 L12.2" or equivalently "BLDG JOG S10.1 5 -12.2 -5 -12.2" advances 10.1 units and then draws a closed rectangle on the right hand side of an existing line. The offsets are always done in the X-Y plane. If the current line is vertical, an offset to the right is along the positive X-axis.

Example of building location using jog special. The command adds both a jog of 20 to the right and 10 to the left and then closes the building at point 108. **BLD JOG20 -10 CLO**



GAP: -This special code makes a single segment break in the current linework. For example, if you have a curb polyline that you want to break to skip over a driveway, then you could add the GAP code at the start of the driveway and continue the curb as normal on the other side.

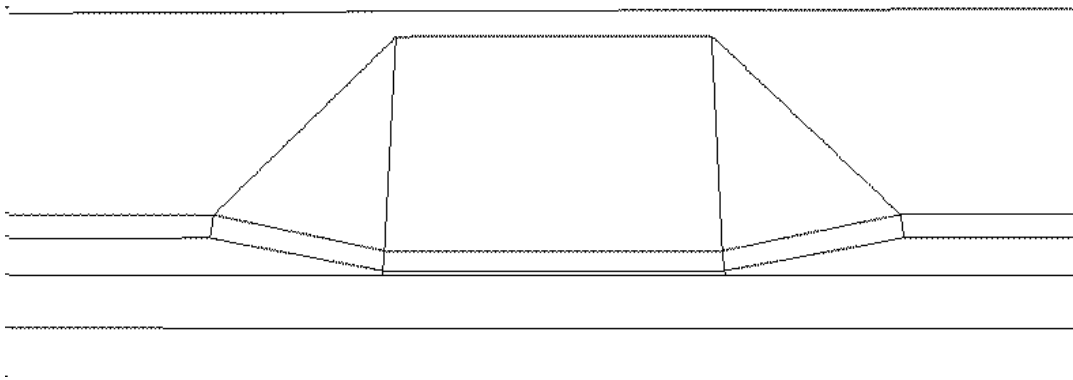
LTF: Linetype Flip - This special code switches the side for the linetype. This option applies to non-symmetrical linetypes like the treeline or guard rail for when you want the linetype to face the other way.

AFIT: Fit Arc - This special code adjusts the PC/PT points for the current arc to make the arc tangential. This special code is a way to individually control this tangential arc adjustment. To adjust all arcs within a specified adjustment tolerance, use the Adjust PC/PT setting under the Additional Draw Options from the first Field-To-Finish dialog.

LTW: Line Width - This special code sets the line width. The width value is entered after the LTW code. This width is applied to lines and 2D polylines.

BFIT: Best-Fit Line - This code creates a best-fit line using the points for the linework. This feature can be used when you have multiple points on a feature that you know is a straight line such as a sidewalk and you want a single line to be drawn. Each of the points that you want to include in the best-fit need to have the BFIT code.

RAMP: Curb Ramp - This special code adjusts the 3D polylines to create a curb ramp. The routine looks for parallel 3D polylines for the bottom of curb, top of curb and back of curb. These 3D polylines can be created in Field-To-Finish by using a Template under the Code > Linetype definition, or using the offset special codes, or by having three separate 3D polylines. The ramp is centered at the point with the RAMP code. The dimensions of the ramp follow the code in order of width, depth and taper. For example, a description of "BC RAMP 3 2 1" uses "BC" for a bottom of curb code then "RAMP" for the special code and 3 for the width, 2 for the depth and 1 for the side taper.



XSCT: Template - This special code defines a template from a series of points with this special code. This template is then applied like a template defined under Linetype for a code. This XSCT code is a way to define a template in the field instead of having a template with fixed dimensions defined in the code table. For example, you could have a code for BC for back-of-curb. Then store three points with a description of "BC XSCT" for three points to cross section the curb: back-of-curb, top-of-curb and flow line. After these three "BC XSCT" points, you could have single "BC" points along the curb and the program will apply the template along these curb points and draw three parallel lines.

PARKING: Parking - This special code draws parking stall lines using three points. Points one and two are used to draw the first parking line and define the length and angle of the lines. The third point defines that position of the last parking line. On the first point, a number needs to follow the PARKING code for the number of parking lines to draw. For example, PARKING8.



Point Symbol/Attribute Special Codes

ROT: Rotate - This code is used to set the rotation of the point symbol. If a point number follows the ROT code, then angle from the current point to this point number is used for the rotation. For example, "ROT45" would rotate the symbol towards point number 45. If there is no point number after the ROT code, then the rotation point is the next point number with the same code as the current point or a companion code for the current code. ROT can also be used to rotate towards an angle clockwise from north by using '+' or '-' in front of the number. For example ROT+45 rotates the point symbol to the northeast and ROT-90 rotates the point symbol to the west.

SZ: Symbol Size - This code is used to set a different symbol size. There are several ways to use this code. It can take multiple scale factors for different dimensions by putting an ID character after the factor.

- **SZ:** If nothing follows the SZ code, then the next point with the same field code as the current point will be used to determine the size.
- **SZ#:** The value of the new symbol size is specified after the SZ. This value is the actual size in drawing units. For example, SZ2.
- **SZ#X:** The value after the SZ is used to scale the symbol in the X dimension. For example, SZ2X.
- **SZ#Y:** The value after the SZ is used to scale the symbol in the Y dimension. For example, SZ2Y.
- **SZ#Z or SZ#V:** The value after the SZ is used to scale the symbol in the Z (Vertical) dimension. For example, SZ2Z.
- **SZ#H:** The value after the SZ is used to scale the symbol in the X,Y (Horizontal) dimensions. For example, SZ2H.
- **SZ#S:** The value after the SZ is a symbol size scaler that get multiplied by the drawing horizontal scale to determine the actual drawing units. For example, SZ0.2S.

The X, Y, Z, V and H can be combined. For example, to scale a symbol by 10 horizontally and 25 vertically, use SZ10H25Z. Or to scale a symbol by 2 in the X direction and 4 in the Y direction, use SZ2X4Y.

When multiple SZ codes are used in the same point description, the symbol is drawn multiple times at the different sizes. For example, a point description of "TREE SZ5 SZ10" will draw the tree symbol twice. One symbol will be size 5 and the other size 10.

AZI & DIST

The AZI and DIST codes are used together to offset the point. The AZI sets the offset azimuth and DIST sets the distance. The values should directly follow the code. For example, AZI25 DIST4.2 would draw the point offset 4.2 at an azimuth of 25 degrees.

2ND: Multi-Point Code - When used on the first point of a multi-point symbol, the "2ND" code indicates that the second point of the sequence (i.e., the next point after the current one) should be used as the second symbol insertion point for a multi-point symbol. Please refer to Symbol Points in the Edit Field Code Definition section below.

3RD: Multi-Point Code - When used on the first point of a multi-point symbol, the "3RD" code indicates that the third point of the sequence should be used as the third symbol insertion point. The "3RD" code should be used with the "2ND" code. Please refer to Symbol Points in the Edit Field Code Definition section below.

Note: The 2nd and 3rd codes are not necessary if the Automatic Multi-Point Special Codes option is enabled. Please refer to Symbol Points in the Edit Field Code Definition section below.

PHOTO - This code attaches a photo file to the point. The name of the photo file should be right after the PHOTO code. The PHOTO Link setting controls whether the photo is attached using a Carlson-format link or a CAD Hyperlink. Use the Image Inspector command to view photos attached to points by either link method. To use the Hyperlink, you can Ctrl-click or right-click on the point entity.

In addition to the PHOTO code, Field-to-Finish will also automatically create the photo links for SurvCE photos. The program looks for the photo database from SurvCE which should have the same name as the coordinate file with an extension of .phdb. This photo database file should be in a sub-folder of the coordinate file folder and called Pictures_X where X is the name of the coordinate file. For example, if your coordinate file is C:\Projects\Job1\Job1.crd, then the program looks for C:\Projects\Job1\Pictures_Job1\Job1.phdb.

When SurvCE stores photos, it creates this photo database using this naming. So to process with Field-to-Finish, copy the coordinate file and photo files from the data collector to your computer.

LABEL - This code controls the point attribute format using a number after the code. This number uses 0=attribute block, 1=text, 2=both, 3=none. For example, LABEL1 means draw that point using text attributes.

COLOR - This code sets the entity color. The color is specified after the "COLOR" code by either a CAD color number (1 to 256) or color letter. The color letters are R=red, Y=yellow, G=green, C=cyan, B=blue, M=magenta, W=white. For example, COLOR1 means use CAD color 1 (red) and COLORB means use color blue.

PHOTO Link - Sets the option for linking the photos. Carlson format links using the process described above. Hyperlink is a CAD link that is enabled through the CAD properties dialog box. You can also double click on any point and use the image tab to add an image file.

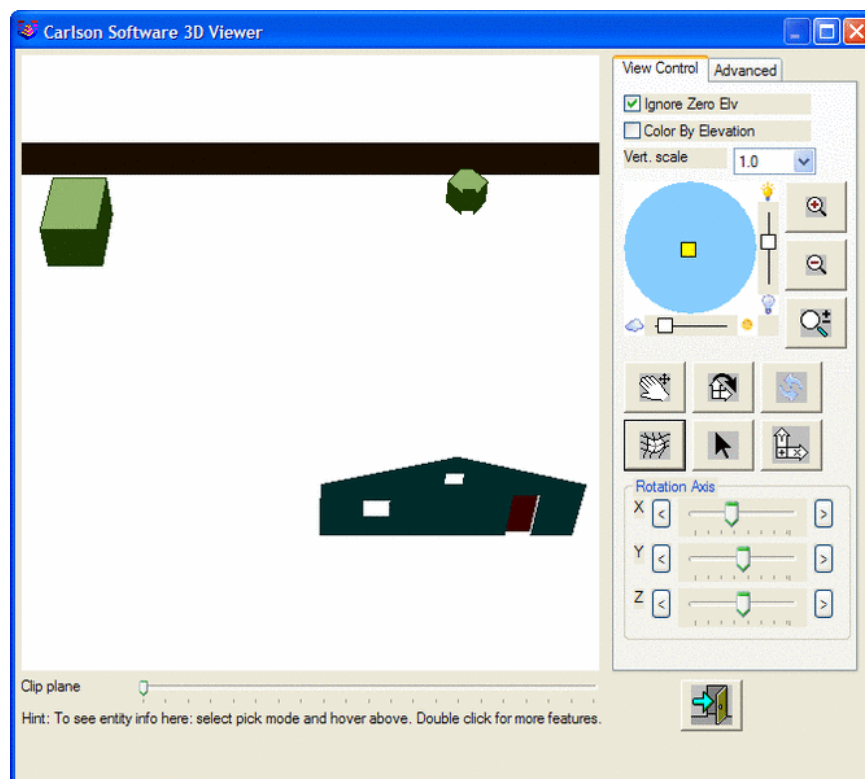
OFB Offset Width -This option works with the **OFB** special code.

- The Double option expects the user to enter half the total offset distance and the routine will double it. For example, if the total width of a pathway is 10, the OFB code will be 5.
- The Split option expects that the user will enter the total distance and the routine will split it in half. For example, if the total width of a pathway is 10, the OFB code will be 10.

2-Point Circle - This option works with the CIR special code. The radius option will use the first point with the CIR code as the radius point and the second located point as the outside of the circle. The Diameter option expects that the first point with the CIR code is one point identifying the outside of the circle and the second point another. Together they create a 2 point circle diameter.

Append Desc Auto Space -applies to the special codes that control the point description label. This option sets whether to insert a space in the description label when appending to the description.

3D Face Special Codes Below are the special codes that can be used for the easy creation of 3D surfaces. The resulting 3D face entities can be viewed in the Carlson 3D viewer by entering "cube" on the command line.



FACE3D - Makes a triangle mesh of 3D face entities by triangulating points starting with the current point and continuing until the line ends or another 3D special code is found. The points must be ordered along the perimeter. Although the mesh will be built if the points are clockwise or counterclockwise along the perimeter, the visible side in the Carlson 3D viewer, "cube", is the clockwise side by default. On the Advanced tab, the shading mode may be set to *Shade both* or *Shade back* if you would prefer to see both sides or just the counter-clockwise side.

HOLE3D - Makes an exclusion area within the triangle mesh identified by the point number following this code (e.g., "HOLE3D101" will start a hole in point # 101). If no point number is given ("HOLE3D"), the exclusion area is applied to the last mesh or if there is a mesh in the process of being constructed by the current sequence of points, it is ended and the hole is applied to it. Note that a hole can only be applied to a mesh that was created by FACE3D (not BLOCK3D or WALL3D). Note also that it can be difficult to predict what the "last mesh" was if it used a different field code since the points of the coordinate file are processed by order of field code first and then point number. There is no limit to how many holes can be applied to a FACE3D mesh. The points of the hole itself are not added to the FACE3D mesh; they are projected on to the best plane that contains the FACE3D mesh and then the hole is cut-out.

Example 1:

2500 HOUSE1 FACE3D */front of house*

2501 HOUSE1

2502 HOUSE1

2503 HOUSE1

2504 HOUSE1

2505 VENT1 HOLE3D2500 */applies 2505-2508 as a hole to last mesh that uses point #2500. So any point in the range 2500-2504 would have the same effect.*

2506 VENT1

2507 VENT1

2508 VENT1

Example 2:

2500 HOUSE1 FACE3D */front of house*

2501 HOUSE1

2502 HOUSE1

2503 HOUSE1

2504 HOUSE1

2505 HOUSE1 HOLE3D */stops the above mesh and applies 2505-2508 as a hole*

2506 HOUSE1

2507 HOUSE1

2508 HOUSE1

Example 3:

2500 HOUSE1 FACE3D */front of house*

2501 HOUSE1

2502 HOUSE1

2503 HOUSE1

2504 HOUSE1

2505 WINDOW1 FACE3D HOLE3D2503 */applies 2505-2508 as a hole to above mesh 2500-2504 and starts a new mesh using the WINDOW field code.*

2506 WINDOW1

2507 WINDOW1

2508 WINDOW1

Example 4 (same result as Example 3):

2500 HOUSE1 FACE3D */front of house*

2501 HOUSE1

2502 HOUSE1

2503 HOUSE1
2504 HOUSE1
2505 WINDOW1 FACE3D /starts a new mesh using the WINDOW field code.
2506 WINDOW1
2507 WINDOW1
2508 WINDOW1 HOLE3D2504 /makes the mesh 2505-2508 also be a hole in the mesh 2500-2504.

BLOCK3D - Makes a set of 3D faces to make a 3d block using the height value entered after the code (e.g., "BLOCK3D2.3" with height 2.3). Heights can be positive or negative. With 3 points, makes a parallelogram base that is extruded up (or down if height is negative) to form a 6-sided block, including top and bottom. With 4 or more points, makes a closed polygon for the base that is then extruded by the height. The points can be laid out in clockwise or counterclockwise order around the perimeter. The perimeter or base does not have to be a convex polygon.

WALL3D - Makes a set of 3D faces above the polyline using a height value entered after the code (e.g., "WALL3D2.3" with height 2.3). This height is a signed value so you can shoot either the top of the wall or the bottom of the wall. The height can be negative if the points on the top of the wall have been shot. If no height parameter exists, then the height is determined by the distance from the current point to the next point.

After the height value, you can have a width/thickness value. If no width value is entered, then a width of zero is used. When the width value starts with a minus "-", then the width is put on the left side of the points. When the width value starts with a plus "+", then the width is put on the right side of the points. When the width value has no plus or minus, then the points go through the center of the wall width.

Both sides of the wall will have triangles and so both sides will always be visible in the Carlson 3D viewer.

Example – 6' high wall shot along the bottom:

```
2000 1000.000 1060.000 100.000 WALL1 WALL3D6.0 /wall 6'  
2001 1100.000 1060.000 100.000 WALL1  
2002 1100.000 1160.000 100.000 WALL1
```

Example – 6' high and 1.5' wide wall shot along the bottom:

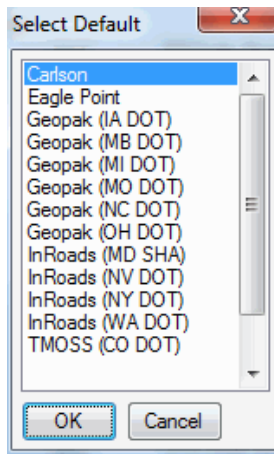
```
2000 1000.000 1060.000 100.000 WALL1 WALL3D6.0 1.5 /wall 6' high and 1.5' thick  
2001 1100.000 1060.000 100.000 WALL1  
2002 1100.000 1160.000 100.000 WALL1
```

Example – 6' high wall, height specified by 1st to 2nd point, shot along the bottom:

```
2020 1100.000 1160.000 100.000 WALL2 WALL3D /height by 2nd pt  
2021 1100.000 1160.000 106.000 WALL2  
2022 1000.000 1160.000 106.000 WALL2
```

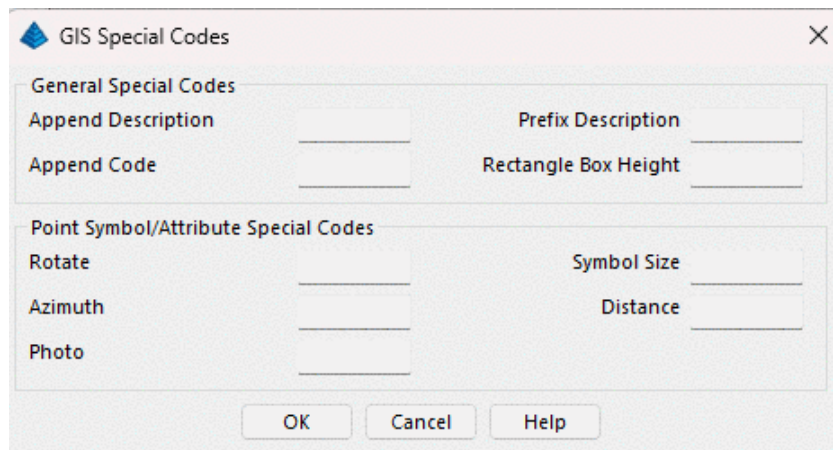
Load Default

This button sets the special codes to Carlson, Eagle Point, Geopak, InRoads or TMOSS defaults.

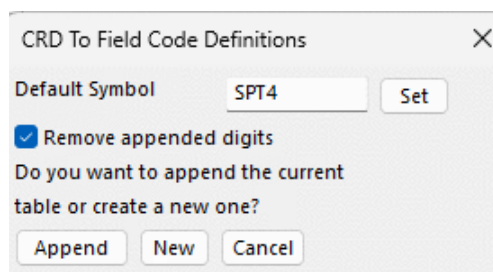


GIS Special Codes: This option allows you to use GIS attribute for Field-to-Finish special coding. For a select group of special codes, a GIS attribute can be assigned. When processing the points, if a point has GIS data for the specified attribute, then that attribute value is used for the special coding. For example, you can have a GIS attribute of COMMENT set to the Append Description special code. Then if a point has a GIS attribute for COMMENT, the value of that COMMENT will be added to the description label for that point.

The Append Code option adds the GIS code to the description code to make the code for processing. For example, for a point with description "ROAD" and a GIS attribute named "TYPE" with a value of "DIRT", then set the GIS Special Code for Append Code to "TYPE" and the program will process this point using the combined code of "ROADDIRT".



Code Table by Points - This command will create code table definitions based on the coordinate file field descriptions. This is useful when creating a code table from scratch



The options available when creating a code table from existing descriptions are:

- Default Symbol: Use the Set button to select a symbol to use for all of the codes created
- Remove appended digits: Removes the numeric group number from any code descriptions. For example EP1 would be created as EP. This avoids unnecessary duplication of code names.
- Append or New: allows you to append an existing code table or create a new one.

Save: Saves the Draw Field to Finish field code definition (.FLD) file.

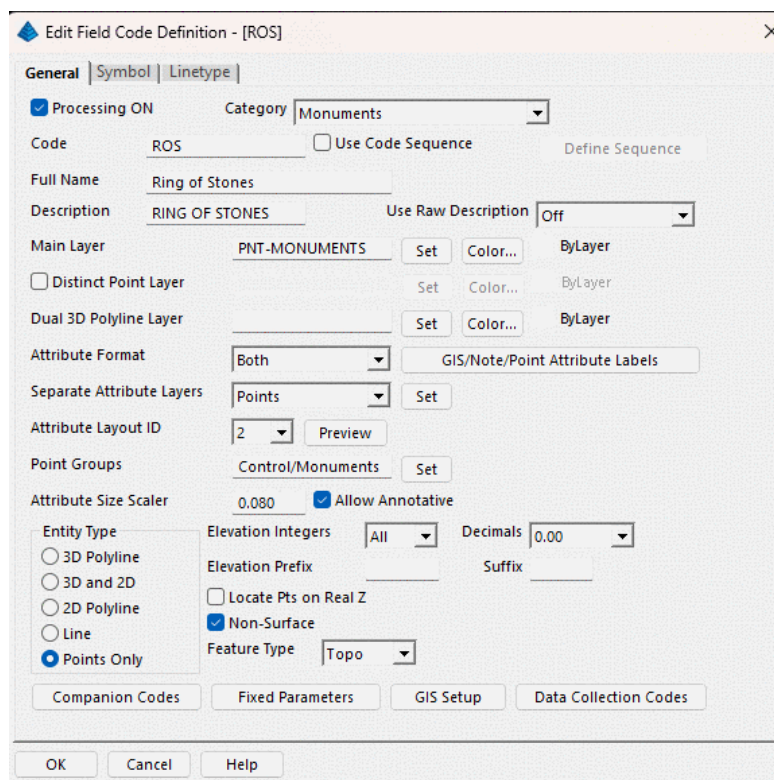
Save As: Reacts the same as Save but allows for specification of file name and location to save to.

Code Definitions

Edit: the Edit button reacts differently depending on whether a Single code is selected or Multiple codes are selected.

Edit (Single Code)

If only one field code is selected, the Edit Field Code Definition dialog box is opened.



The code definition dialog has three tabs: **General**, **Symbol** and **Linetype**. Here are the settings under General:

General Tab

Processing ON: This toggle controls whether this code will be processed. This allows you to process only specific codes if desired.

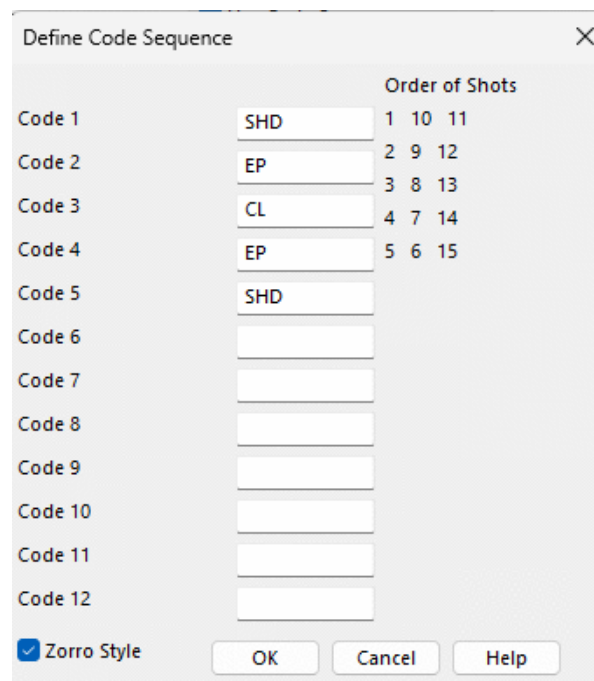
Category: Contains a "drop down" menu that displays the names of the Categories Created in the Draw Field to Finish dialog box for selection.

Code: This is the key name that identifies the code and is matched with the field data descriptions. It is important to note that the * character, used in this field, is regarded as a wildcard or "match anything" code. For example, a field code definition with the code defined as TREE* will be used for any raw description of TREE. Raw descriptions of TREEA, TREE12, TREE, etc. will match the TREE code definition. This will always be the case unless there is a more specific code is found. For example is there was a code TREEA in the code definition file, then that code would be used instead of the TREE code.

It is also important to note that the wild card is not necessary for line number grouping such as EP1, EP2 EP3 etc. The code for this example is simply EP. The program is designed to treat all codes with a numerical suffix as the code itself but separates each one into a separate line string.

Use Code Sequence: This specifies a sequence type code. Sequences are a way to simplify field entry of a sequence of codes. For example, a road cross-section could be SHD1 EP1 CL EP2 SHD2. Instead of entering these different descriptions, one sequence definition can store these descriptions in order. Then just the sequence code (such as RD) is used in the field. The cross-section can be shot in left to right then left right order, right to left then right to left order, or alternating left to right then right to left order. The alternating method is known as the Zorro style. The one restriction is that the shots always start from a right or left edge. To set up a sequence, choose the Sequence toggle in the Edit Code dialog. Then pick the Define Code Sequence button. This brings up a dialog for entering the sequence codes in order. These sequence codes should be defined as normal codes somewhere else in the Draw Field to Finish code table (ie SHD as a 3D polyline). In the field, the one template code is used for all the cross-sections shots (ie RD for all the points). Then Draw Field to Finish will substitute this template code with the sequence codes (ie substitute RD with SHD).

Define Code Sequence: This sets the code names that make up the sequence.

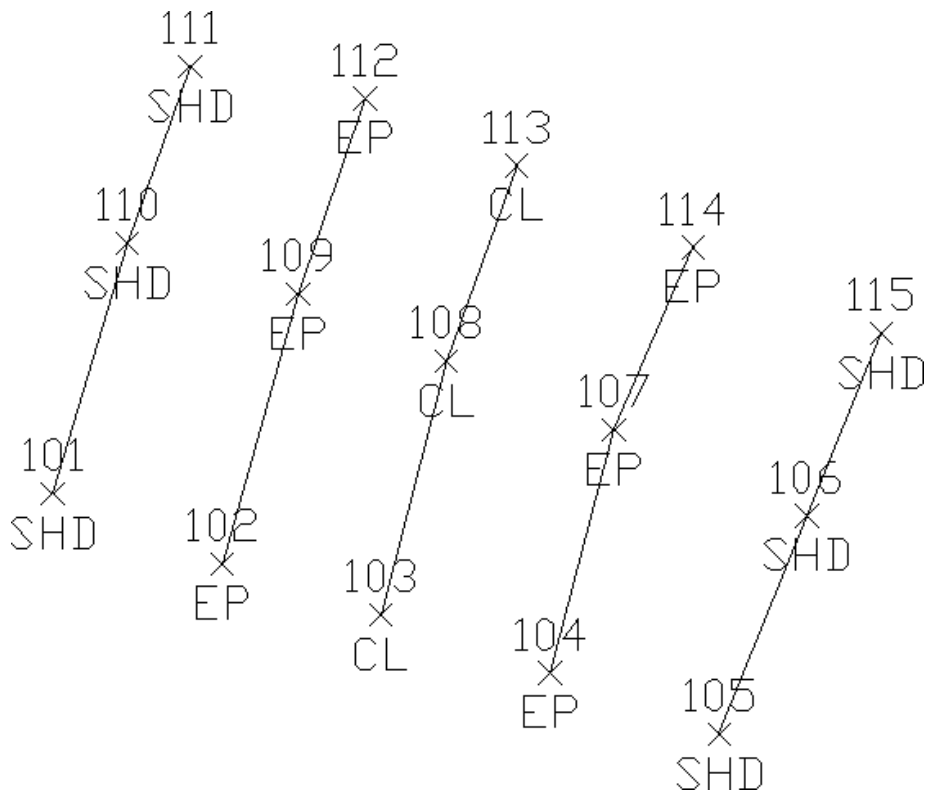


The dialog box titled "Define Code Sequence" contains a table for defining sequence codes. The table has three columns: "Code", "Code", and "Order of Shots". The "Order of Shots" column is further divided into three sub-columns with values 1, 10, and 11. The "Zorro Style" checkbox is checked. At the bottom are "OK", "Cancel", and "Help" buttons.

Code	Code	Order of Shots
Code 1	SHD	1 10 11
Code 2	EP	2 9 12
Code 3	CL	3 8 13
Code 4	EP	4 7 14
Code 5	SHD	
Code 6		
Code 7		
Code 8		
Code 9		
Code 10		
Code 11		
Code 12		

Zorro Style

OK Cancel Help



Resulting points and linework showing Zorro style template

Full Name: This is an optional field that describes the code for viewing and printing but also works with the **Legend** command.

Description: This value is assigned to the point description attribute when the point is drawn. This description can be different than the field code. For example, the code **IP** could have a description of **IRON PIPE** which will be applied to the attribute of the point.

Note: An additional description can be added to a point by entering it after a forward slash "/" for suffix or backwards slash "\" for prefix in the data description field.

Example **IP\3"** would be **3" IRON PIPE**.

Use Raw Description: This option turns off the Description field described above. Instead the points will be drawn with their original unprocessed descriptions.

- Attribute Block: The raw description will populate the *Description Attribute* of the point block.
- Text Attribute: The raw description will add text to the drawing in accordance with the **Attribute Format** options and the respective **GIS/NOTE/Point Attribute Labels** options.
- Both: will perform both of the above options.

Main Layer: The point and line work for the code will be created and placed on this layer.

- The **Set** button allows the selection of an existing layer contained in the current drawing.
- The **Color** button allows the selection of a color from the CAD color selection dialog box.

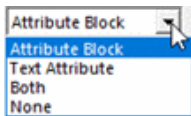
Distinct Point Layer: When this toggle is selected, the line work is created on the layer defined in the Main Layer field and the points are created in the specified distinct point layer. For example, you could have DRIVEWAY for linework and DRIVEWAY_PNT for the points. This option allows for greater control over the placement of points and linework.

- The **Set** button allows the selection of an existing layer contained in the current drawing.
- The **Color** button allows the selection of a color from the CAD color selection dialog box.

Dual 3D Polyline Layer: Displays the layer that the 3d polyline will be drawn on when using an Entity Type of 3D and 2D. The layer name can be typed in this field. The result is the 2D and 3D polylines being drawn on separate layers. For Example, you could have the 2D polyline drawn on the layer DRIVEWAY and the 3D polyline drawn on the layer BREAKLINES.

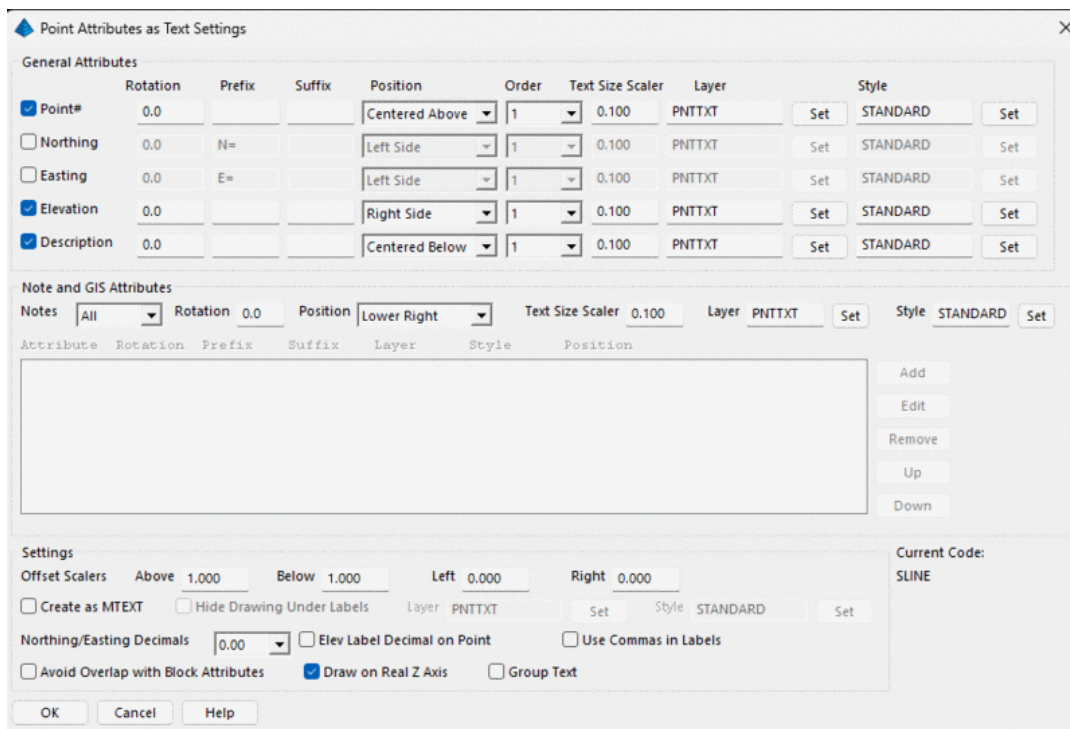
- The **Set** button allows the selection of an existing layer contained in the current drawing.
- The **Color** button allows the selection of a color from the CAD color selection dialog box.

Attribute Format:



This chooses the type of point entities to create. The **Attribute Block** format creates a Carlson point entity which is a block with attributes for point#, elevation and description. The layer of each attribute as well as the point block can be controlled by setting the Separate Attribute Layers.

The **Text Attribute** format creates text entities for each of the point attributes.



When the **Text Attribute** or **Both** format is selected, the placement, appearance, and function of each is controlled by selecting the GIS/Note/Point Attribute Labels button.

For the **General Attributes** control, there are toggles to control whether or not each attribute is displayed for the Point# (number), Northing, Easting Elevation and Description. In addition, there are controls for for each attribute that include:

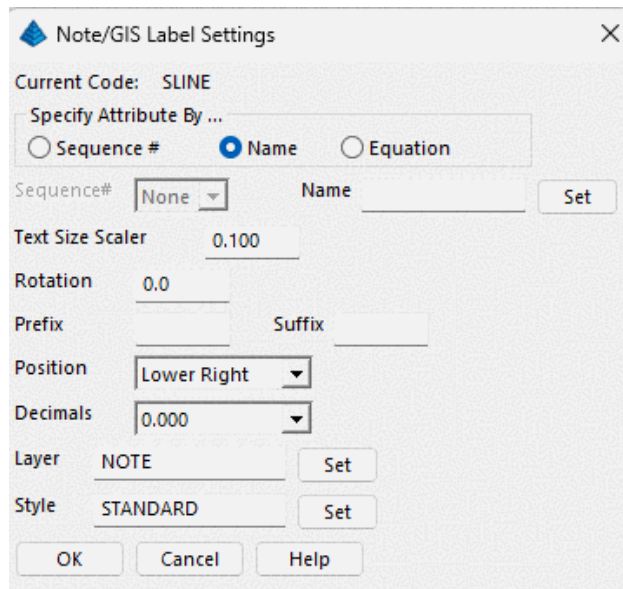
- Rotation - rotates the attribute display
- Prefix - adds a prefix to the attribute
- Suffix - adds a suffix to the attribute
- Position - controls the placement of the attribute in relationship to the point (node)
- Order - controls the order of each attributes.
- Text Size Scaler - sets the size of the attributes in relation to the drawing scale. Example 0.10 at a drawing scale of 1" = 20' will create a text size of 2.0
- Layer - sets the layer for the attribute to be placed
- Style - sets the text style for the attribute.

The **Note and GIS Attributes** section of the dialog box adds additional control over both Notes contained in the .NOT file and GIS attributes defined either by the GIS module, SurvCE/PC or other data collectors or created via Field to Finish fixed parameters. These notes/attributes are in addition to the point description but are directly related to each individual point. Making use of these additional attributes can add a great deal of power and flexibility to the Field to Finish program.

The **Notes** drop-down box allows 3 options.

- None - will not place any additional attributes into the drawing
- All (default) - will place all attributes contained in the .NOT file or created during the Field to Finish process into the drawing.
- Selected - will only place the specifically selected attributes into the drawing.

For **selected**, use the Add, Edit and Remove buttons to build the list of fields to label.



To specify the field to label, the *Sequence#* method sets the field by its order position. For example, a sequence of 3 would use the third attribute for the point. 5 Wires in the below example

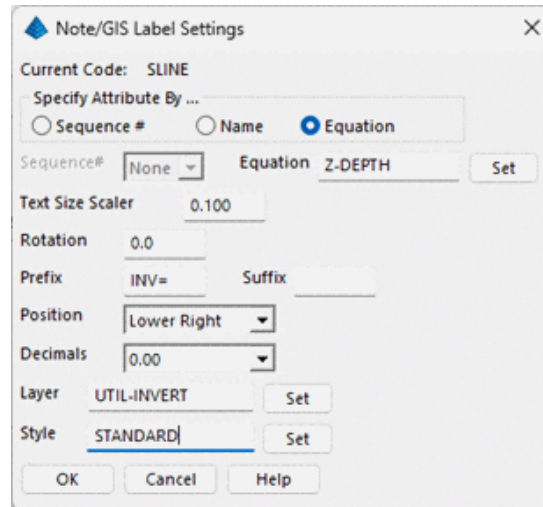
Note	
1	#43
2	WOOD
3	5 WIRES
4	
5	
6	
7	
8	
9	
10	

The **Name** method sets the field to label by field name. The field name can be predefined through the GIS feature file, by SurvCE/PC in the field or can be created via the Fixed Parameters in the Edit Field Code Definition dialog box. Example of Named Attributes:

Note	
1	NUMBER=#43
2	MATERIAL=WOOD
3	WIRES=5
4	
5	
6	
7	
8	
9	
10	

The **Equation** method sets the value by the specified equation of attribute names and numbers. Besides the attribute

names, there are keywords of "X", "Y" and "Z" for the values of the current point coordinates. For example, if there is an attribute named DEPTH, then you could define a label for the invert as an equation of "Z - DEPTH".



Another example is to make a label that is 1.5 higher than the point elevation using an equation of "Z + 1.5". Besides attribute names, you can make an equation by attribute sequence using "NOTE" plus the sequence number. For example, if the depth value is the 3rd attribute, then to define a label for the invert elevation, use the equation "Z - NOTE3".

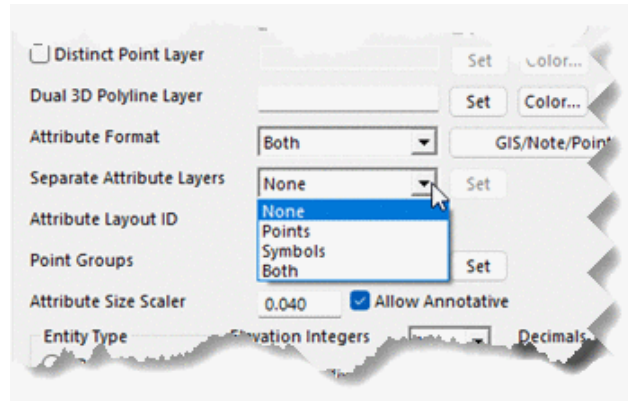
Note: Besides labeling attributes as text with this method, the Symbol > Custom Attributes feature is a way to label attributes as block attributes.

The **Settings** portion of the dialog box adds additional control of the placement and appearance of the attributes as they are placed in the drawing.

- Offset Scalers - control the distance for the text from the point for the different positions. These offset distances are calculated by multiplying the scaler by the horizontal scale for the drawing.
- Create as MTEXT - creates a Mtext entity containing all of the attributes.
 - Hide Drawing Under Labels - creates a background mask for the Mtext entity
 - Layer - with the *Create as Mtext* option enabled, all attributes will be placed on the same layer as set in this portion of the dialog box.
 - Style - with the *Create as Mtext* option enabled, all attributes will be use the same text style as set in this portion of the dialog box.
- Northing/Easting Decimals - sets the precision for the horizontal coordinates displayed
- The Elevation Label Decimal On Point option will place the elevation label so that the decimal point of the label is on top of the point location.
- Use Commas In Labels option will add commas for the thousandths place for the northing, easting and elevation labels. ie. N=1,000,000.00
- Avoid Overlap With Block Attributes option will attempt to resolve conflicts between the point block attributes and those set in this dialog box by shifting the horizontal location of the latter.
- Draw On Real Z Axis option controls whether to the text entities at the point elevation or at zero elevation.

- Group Text option creates a CAD Group for all the point labels.

Separate Attribute Layers:



This controls the layers of the point block, point attributes and/or point symbols.

- None - the point attribute layers are the default layers of, "PNTNO", "PNTELEV" and "PNTDESC", the parent layer for the point attribute block and symbol is the Main Layer designated in the Field Code definition.
- Points - allows the designation of each point block and associated attributes layers. For example if the code DWL has a layer name "DRIVEWAY", the point attributes could then be "DRIVEWAYNO", "DRIVEWAYELEV" and "DRIVEWAYDESC".
- Symbols allows the designation of the symbol associated with the point
- Both allows both of the above options.

Attribute Layout ID: Controls the location of the point number, elevation and description. These attribute layouts are defined in the drawings that are stored in the Carlson SUP directory with the file name of SRVPNO plus the ID number (i.e. SRVPNO1.DWG, SRVPNO2.DWG, etc.). If you want to change the attribute positions for a layout ID, then open and edit the associated SRVPNO drawing. Layout Ids of 1-9 are available.

Point Groups: This field is for the name of the point group that all points with this code will be added to. If the points for this code belong to multiple point groups, you can specify multiple point group names in this field separated by commas. Under Draw in Additional Draw Options, there is an option whether to automatically use the code name as the point group name or to use the name defined in the code definition. A point group does not have to exist for this option to work. It will create the designated point groups when processed.

Attribute Size Scaler: This is a scaler value that is multiplied by the horizontal scale to obtain the actual size for the point number, elevation and description attributes. This control is independent of the symbol size scaler and other text/attribute scalers allowing complete control over each element associated with the code.

Allow Annotative: This option will create the points as annotative when the annotative setting is on in the Point Defaults command.

Entity Type: This defines the line entity to be created.

- Points Only - does not connect points with any line work, however does create an option symbol is selected.
- Line - draws a CAD line entity
- 2D Polyline - creates a 2 dimensional Polyline drawn at elevation 0.

- 3D and 2D - creates both a 2D Polyline at elevation 0 and a 3D Polyline with each vertex matching the point elevation. The 3d polyline is placed on the layer specified in the **Dual 3d polyline Layer** setting and a 2d polyline in the layer identified in the **Main Layer** setting. Since 3d polylines do not display linetypes and are often difficult to draft with, this option is useful. 3D Polylines can be used for adding breaklines for surface modeling while 2d Polylines will maintain proper linetypes and can be edited through general CAD commands such as offset and fillet.

Note: The generation of Points, Symbols 2D and 3D Polylines can also be controlled through the Draw Field to Finish and Additional Draw Options.

Elevation Integers: This controls the number of digits to display to the left of the decimal point for the elevation label. The All setting will show the full elevation digits. The other settings allow you to limit the number of digits to display for the purpose of reducing the amount of space the elevation labels take up in the drawing. For example, if a site is in the 4000 foot elevation range, then this setting could be set to three digits (000) and an elevation of 4321 would be labeled as 321.

Elevation Decimals: This controls the display precision for the elevation label.

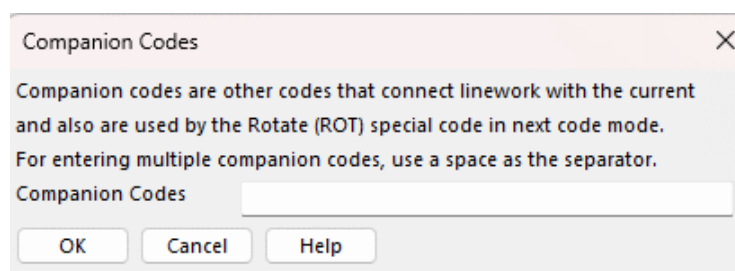
Elevation Prefix/Suffix: These set the prefix and suffix for the elevation label per code. In the Draw function under Additional Draw Settings, there is an override to set the elevation prefix/suffix for all the codes.

Locate Pts on Real Z Axis: This option will draw the points at the actual point elevation. Otherwise the points are drawn at zero elevation. For example, you could turn this option off for the FH for fire hydrant code to draw them at zero. Then the GND code could have this option on to draw the ground shots at their elevations. It is important to remember that Carlson's Triangulate and Contour routine will utilize points for creating a surface model regardless of whether they are set at Real Z or not. If the point is placed in the drawing at Real Z, Triangulate and Contour will use the point block elevation for the Z value whereas if they are drawn at zero (0), it will use the elevation from the CRD file. To ensure that a point is not used for surface modeling in Triangulate and Contour, use the Non-Surface option or utilize the NE or NOS special codes.

Non-Surface: Entities created with this flag are ignored when contouring or creating surfaces regardless of their elevation.

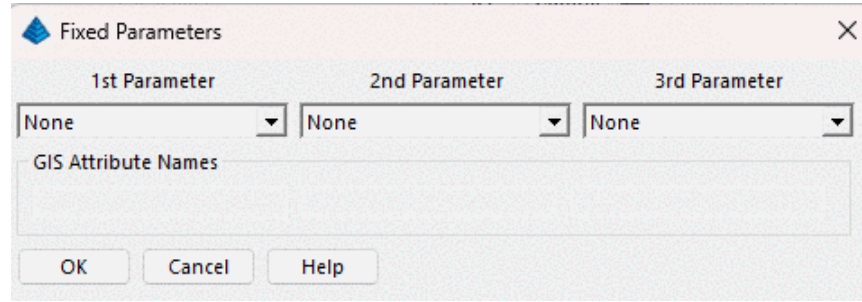
Feature Type: Controls how to process this code. The Default Feature Type is Topo. This is used for most survey work and places points using all of the above assigned settings. The Tree, Utility and Pipe Features are processed more specifically and are controlled via individual setups.

Companion Codes: This option allows different codes to connect when defined as line, polyline or 3d polyline. For example, a main line power pole code may be defined as PP while a service utility pole may be defined as UP. When processing Draw Field to Finish, it may be desired to connect all PP and UP codes together. This could be accomplished by defining a companion for UP as PP and a companion code for PP as UP. Each code needs to reference the other as a companion code.

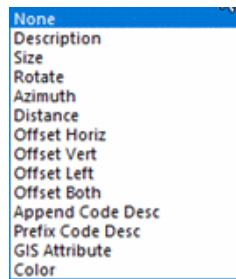


Companion Codes can also be used in conjunction with special codes such as the Rotate (RO) code.

Fixed Parameters: This option is a coding method where you specify a sequence of parameters that follow the main code.



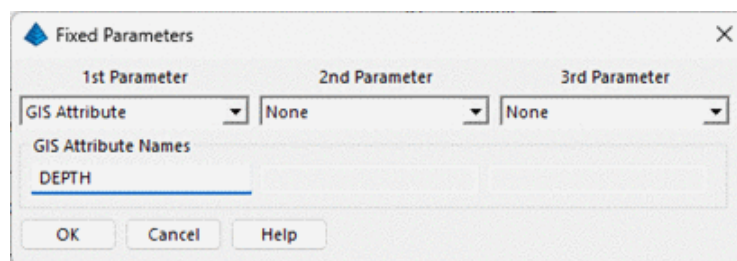
There can be up to three parameters each one having the options shown below:



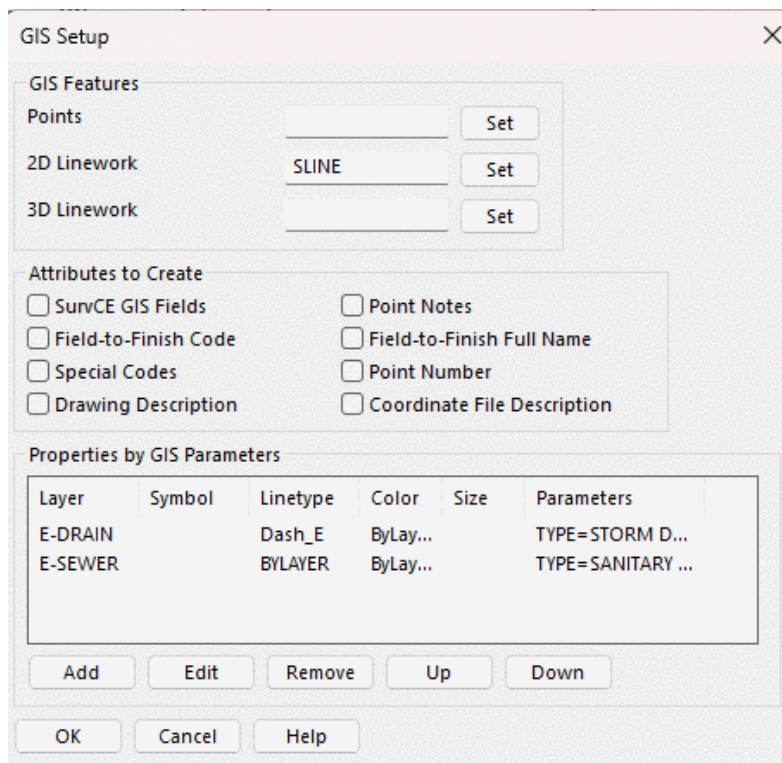
Each of these options take the place of keying in special codes in the field. Instead, the parameters can be entered directly following the main code followed by spaces.

For example: a code called CURB was used to create a consistent front and back of curb lines using the Horizontal and Vertical offset special codes. The first parameter is set as **Offset Horiz** and the second as **Offset Vert**. It would be coded in the field as **CURB 1 .5** for a horizontal offset of 1 and a vertical offset of 0.5 eliminating the need for the OH and OV special codes.

GIS attributes can also be created using Fixed Parameters by choosing the GIS Attribute parameter. This way the GIS attributes can follow the main code in the description. For example, you could have a main code of MH for manhole followed by the depth to store as a GIS attribute. The description would be "MH 4.4" which results in the creation of a named attribute, DEPTH with a value of 4.4.



GIS Setup:



With GIS processing activated, the entities created by Field-to-Finish are linked to a GIS feature name and attributes. The Features and respective attributes are contained in the .GIS file created in the GIS pull-down menu in the Survey module or within the full GIS module. There are two basic required files for using GIS, (beyond the use of Field to Finish), a .GIS file that contains the features and attributes and a database file to contain that data associated with those features and attributes. *See GIS module help for more information*

GIS processing in Field-to-Finish starts with the GIS Table setting in the initial Draw Field To Finish dialog.

Setting the GIS Table is optional but useful. The GIS Table is used as the reference in the Set functions for selecting a GIS feature name to assign to Field-to-Finish codes. Additionally, when processing the Field-to-Finish codes, any associated attributes from the GIS Table will be attached to the entities. Also, attributes generated from Field-to-Finish can be added to the GIS Table. Essentially, there is a "read/write" functionality between the FLD file and the GIS file.

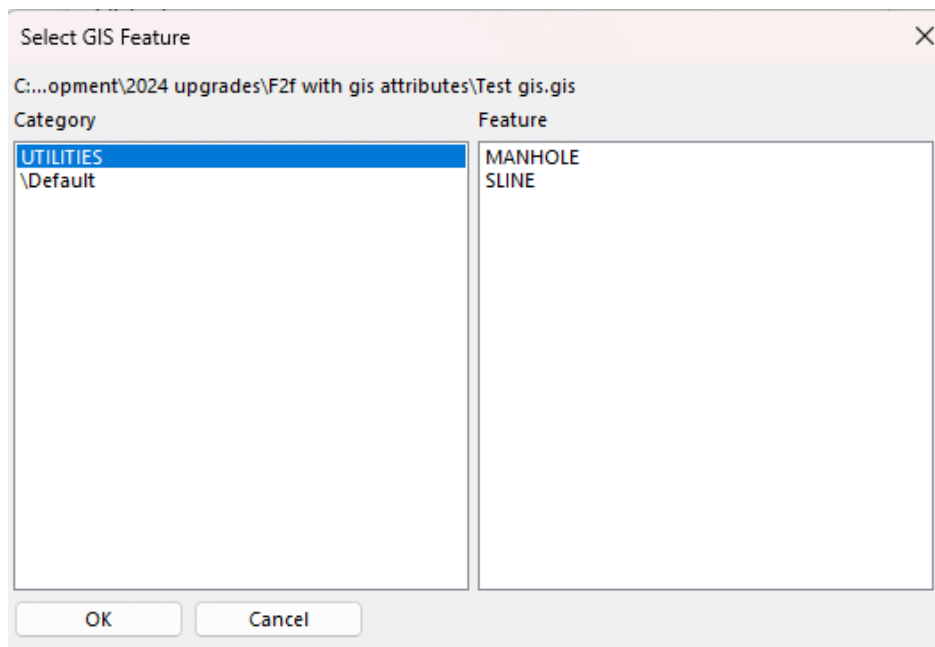
GIS Features: Each Field-to-Finish code has settings to assign GIS feature names. In the Edit Field Code Definition dialog, the GIS Setup button brings up a dialog for setting the GIS feature names and attribute options for the current code.

Points - selects a GIS feature defined as a Point

2D Linework - selects a GIS feature defined as a 2D polyline

3d Linework - selects a GIS feature defined as a 3D polyline

Set - in all 3 of the above options, the Set button will open a dialog box listing the available features in the active GIS table

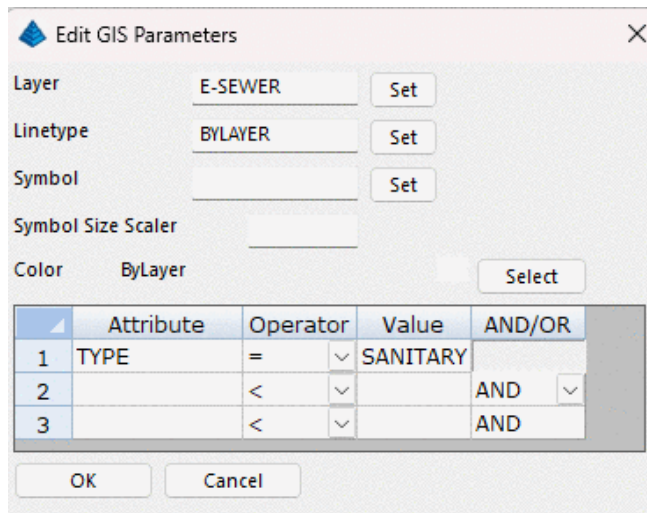


Since Field-to-Finish codes are capable of drawing both points and linework and GIS can have different features for points and linework, there are separate settings for the GIS feature names for points and linework. For example, a Field-to-Finish code UP for utility pole could be setup to draw both points with symbols at the poles and polylines between these points. Then you could have different GIS feature names for the pole points and linework with separate GIS attributes for each.

Attributes to Create: each option toggled on will create a GIS feature and add it to the .GIS file.

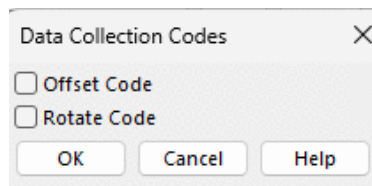
- **SurvCE GIS Fields:** This option uses the attribute data generated by SurvCE which is stored in a .vtt file with the same file name as the current coordinate file except with the .vtt extension.
- **Point Notes:** This option creates GIS attribute data generated by SurvCE from the .not file with the same file name as the coordinate file except with the .not extension.
- **Field-to-Finish Code:** This option creates an attribute named CODE with a value of the Field-to-Finish code name (ie. UP).
- **Field-to-Finish Full Name:** This option creates an attribute named FULL_NAME with a value of the Field-to-Finish Full Name (ie. Utility Pole).
- **Special Codes:** This option creates attributes for Field-to-Finish special codes including OH (Offset Horizontal), OV (Offset Vertical), SZ (Size), ROT (Rotation), AZI (Azimuth) and DIST (Distance).
- **Point Number:** This option creates an attribute named POINT_NAME with a value of the point number from the coordinate file.
- **Drawing Description:** This option creates an attribute named POINT_DWG_DESC with a value of the point description for the point block created in the drawing.
- **Coordinate File Description:** This option creates an attribute named POINT_RAW_DESC with a value of the point description from the coordinate file.

The Point Properties By GIS Parameters lets you control the layer, symbol, linetype, color or size of the points based on GIS attributes.

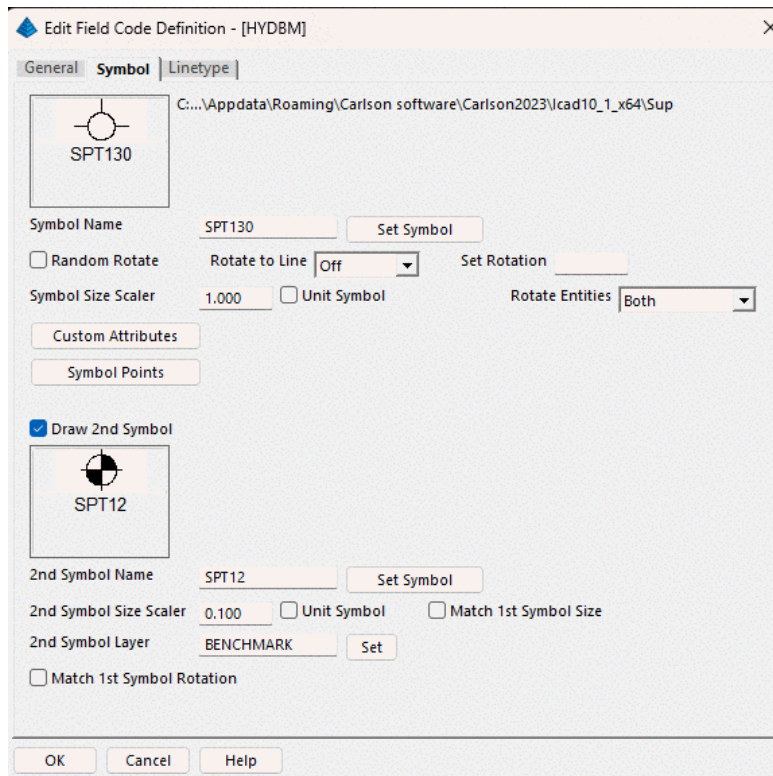


In the above example, a feature with the name SLINE (meaning sewer line) was selected. When Field to Finish finds 2 or more points with that code name and the attribute Type is set to Sanitary, it will draw a 2D polyline on the layer E-Sewer with a bylayer line type. Each code and respective feature can contain as many attributes as desired. Each attribute can then trigger different options for the placement of lines or symbols on different layers at different sizes.

Data Collection Codes: These settings apply to Carlson Field for turning on the Offset mode and Rotate mode automatically by F2F code.



Symbol Tab



Set Symbol: This is the point symbol for the code. The dialog allows you to select from the symbols defined in the Symbol Library which is setup with the Settings->Symbol Library command. Besides the symbols from the symbol library, you can also use any symbols that are defined as blocks in the current drawing by entering the block name in the symbol edit box. To have a point without a symbol, use the Carlson symbol named SPT0 which represents "no symbol". If you create a custom symbol (DWG file) and you want to process Field-To-Finish on other computers, be sure to load this symbol (DWG file) in the Settings > Symbol Library on each computer which will place it into the appropriate sub-folder. Or you can upload the symbol onto Carlson Community for the other computer to find. Field-To-Finish automatically checks the Carlson Community for any symbols that aren't found on a local computer.

Note: a great way to distribute custom symbols as well as all other custom settings and files is to use the Settings Server contained in the Project/Data Folders setup.

Random Rotate: This option will randomly rotate the symbol. For example, this option could be used for tree symbols to have the trees drawn in various orientations.

Rotate To Line: This option applies to points that are part of Field-to-Finish linework. This option will align the point attributes and symbol to the associated linework. The Middle mode uses the average angle for the line segments coming to and from the point. The Forward mode uses the line angle going from the point. The Backward mode uses the line angle coming into the point. The Ends mode uses the perpendicular angle for the line end segment and applies for culverts.

Rotate Entities: This setting controls whether to apply rotation to the symbol, the point attribute block or both.

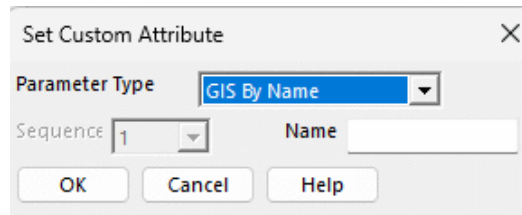
Set Rotation: Sets a specific rotation for a symbol. This rotation is subject to the current view as set in the Twist Screen command.

Symbol Size Scaler: This is a scaler value that is multiplied by the horizontal scale to obtain the actual size in the drawing. The horizontal scale can be set in *Drawing Setup*.

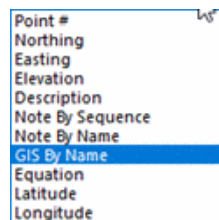
Unit Symbol: This option, when enabled, will draw the point symbol at unit (1:1) scale and will ignore the horizontal drawing scale. For example, this option could be used for a symbol that is already drawn to actual dimensions such as a car symbol.

Custom Attributes: This feature allows you to use customized blocks that have customized attributes (the tag/value pairs). This feature works for both point attribute blocks and symbols. For attribute blocks, Field-to-Finish looks for attributes with the tags "PT#", "ELEV2", and "DESC2". The custom attributes feature allows you to define additional attributes in their custom blocks on a per-field code basis. The dialog shows five attributes at a time. The number of attributes is unlimited. Use the Next and Back buttons to show more attributes.

For an example, the custom block could have an attribute with the tag "TREE_SPECIES" and there is a separate field code for each species of tree. Each of those field codes can specify the value that should be assigned to the attribute that has the TREE_SPECIES tag. Then when the points are drawn, the tree species is shown. Note that the custom attributes must have their Constant and Preset properties set to "no". The custom attributes settings in F2F should not use those tags that the software already handles (PT#, ELEV2, and DESC2), or the setting will be ignored.

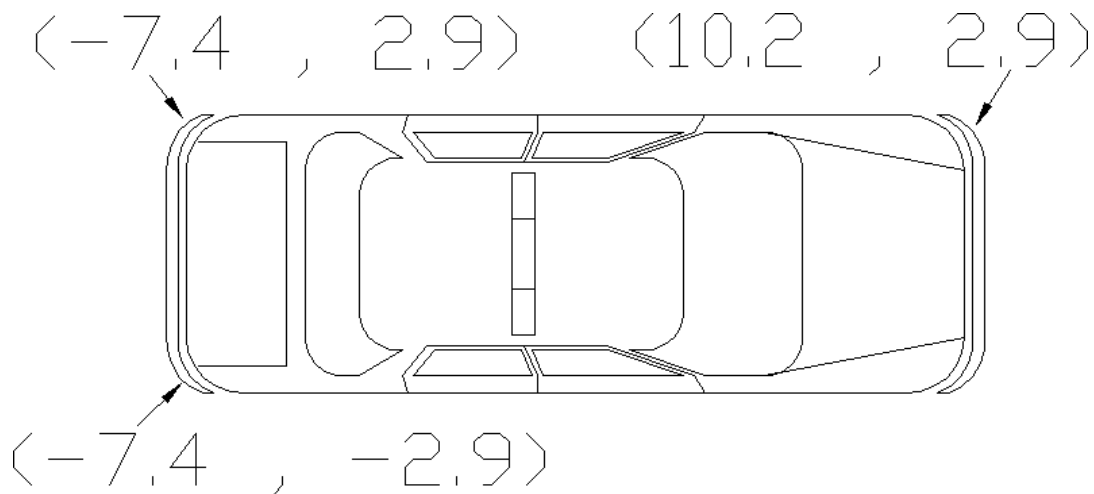
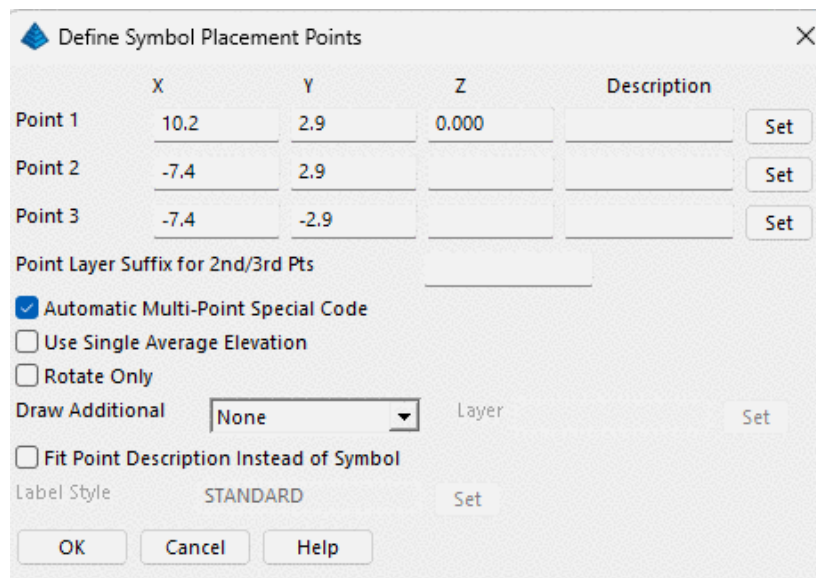


The Values for the attributes can be fixed strings that you enter in the dialog shown here. They can also be dynamic parameters including point number, northing, easting, elevation or description for the current point as well as a point note, GIS attribute or equation. The Equation type sets the value by the specified equation of attribute names and numbers. Besides the attribute names, there are keywords of "X", "Y" and "Z" for the values of the current point coordinates. For example, if there is an attribute named DEPTH, then you could define a value for the invert as an equation of "Z - DEPTH". To setup a parameter value, pick the Set button and then select the attribute. The Decimals setting applies to fields that are real values. The Prefix and Suffix fields are added to the Value for the attribute string. Available Parameter Types:



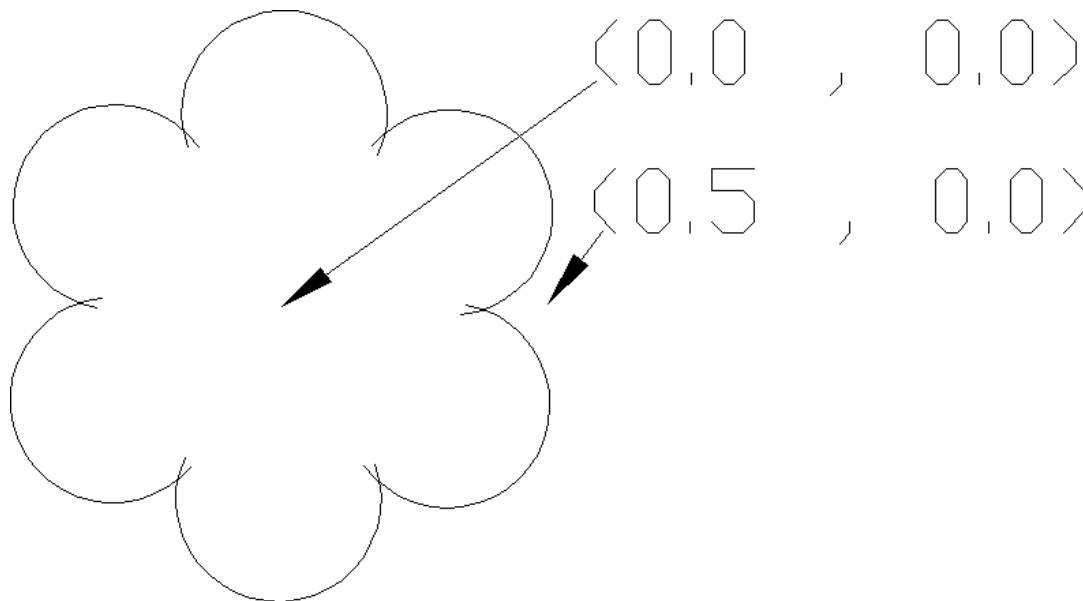
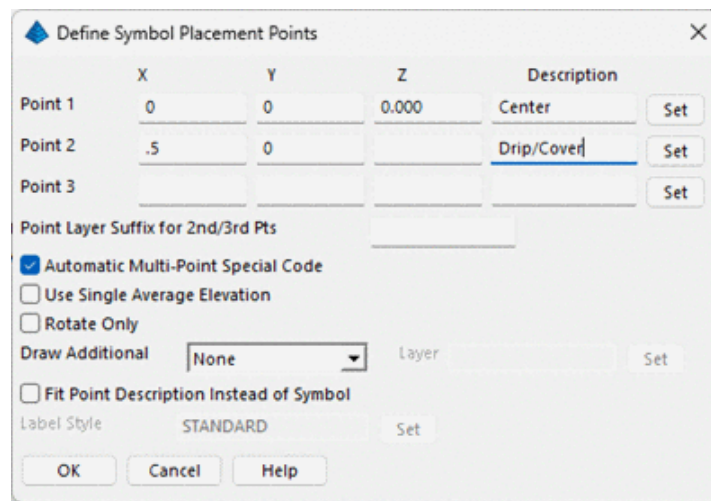
Besides labeling as block attributes, the Attribute Format method of Text mode is a way to label the attributes as text entities.

Symbol Points: For each code definition, the symbol insertion points can be defined with up to three points. To define the symbol insertion points, choose the Symbol Pts button in the Edit Code Definition dialog box. By default, the symbol insertion is defined by one point at the symbol center (0,0). A one point insertion definition can be used to insert a symbol offset from the center. With a two insertion point definitions, the program will rotate and scale the symbol. For example, two insertion points can be used to insert a tree symbol to size the tree, where the first point is for the tree center and the second is for the drip line. With three insertion point definitions, the program will rotate and scale the symbol in both X and Y. For example, three points can be used to insert a car symbol with the first point being the front drivers side, the second point as the back driver side (to rotate and scale the length) and the third as the back passenger side (to scale the width). Besides the insertion point coordinates, you can define a description for each point which is used for the drawn point description and is used for prompting in the Insert Multi-Point Symbol command and in Carlson Field data collection.



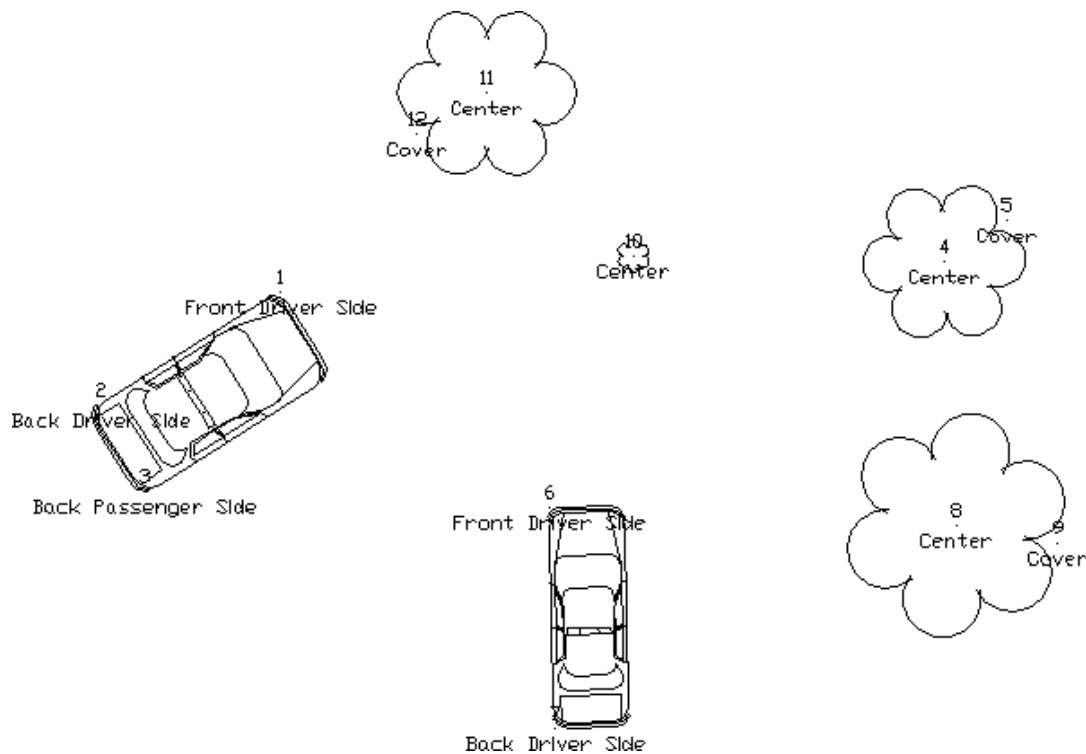
Three Point Symbol Drawing

The coordinates for the insertion point definitions are for the symbol at unit size. To figure these coordinates, you will need to open the symbol drawing (.DWG) file. By default, the symbols are located in the Carlson SUP directory. For example to make an insertion point for the tree drip line, open the tree symbol drawing and find the coordinate at the edge of the tree symbol (in this case 0.5,0.0).



Two Point Symbol Example

Not all of the symbol insertion points need to be used when drawing the points. If a code definition has a three insertion points, it is possible to use just the first two or first one. There are special codes to associate multiple points to the same symbol. The first code point is used as the first symbol insertion point. The "2ND" code is used to specify the second symbol insertion point. A point number can follow the "2ND" to identify a specific point. Otherwise without the point number, the program will use the next point with the current code. The "3RD" code is used to specify the third symbol insertion point and similar to the "2ND" code, a point number after the "3RD" is optional. The "2ND" and "3RD" codes should be assigned to the first point. For example, consider a code of "CAR" with a three point symbol insertion definition. If point #1 has a description of "CAR 2ND 3RD", then point #1 will be used as the first symbol insertion point and the next two points with the "CAR" description will be used as the second and third symbol insertion points.



Multi Point Symbol Drawing

The **Point Layer Suffix for 2nd/3rd Points** option allows for having different layers for the 2nd and 3rd points used with the multi-point symbol.

The **Automatic Multi-Point Special Code** option saves from having to enter the 2ND or 3RD special codes. This option applies when the code is always used from multi-point symbols.

The **Use Single Average Elevation** option averages the elevations of the symbol points for the point elevation label.

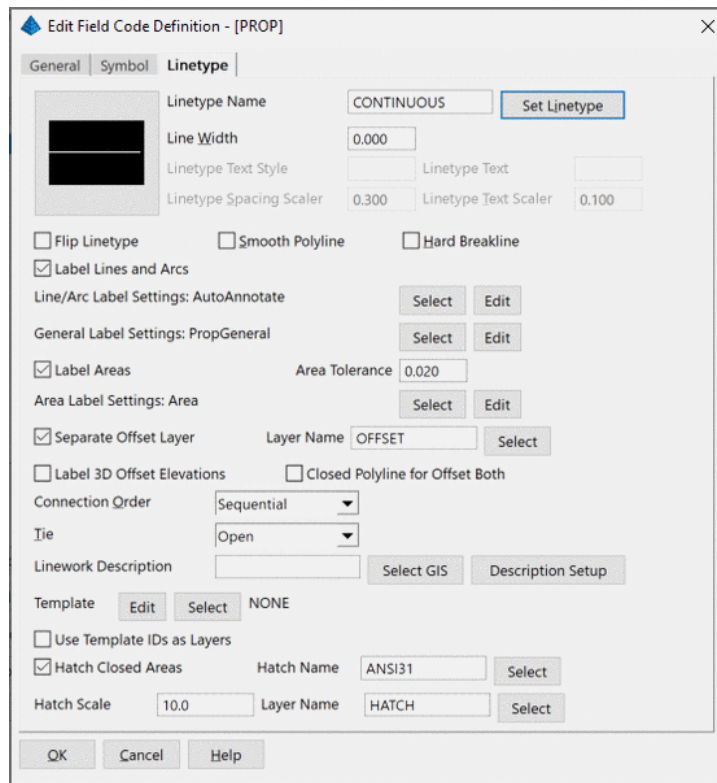
The **Rotate Only** option uses the symbol points only to rotate the symbol and does not scale.

The **Draw Additional** has options for drawing a 4th point or rectangle. The **Create 4th Corner Point** option draws another point when using a three multi-point symbol. This option applies when the symbol is rectangular and you shoot three corner points and want the program to draw a point at the 4th corner. The **Rectangle** option creates a rectangle to fit the 3 symbol points and applies to creating a breakline for surface modeling around the symbol.

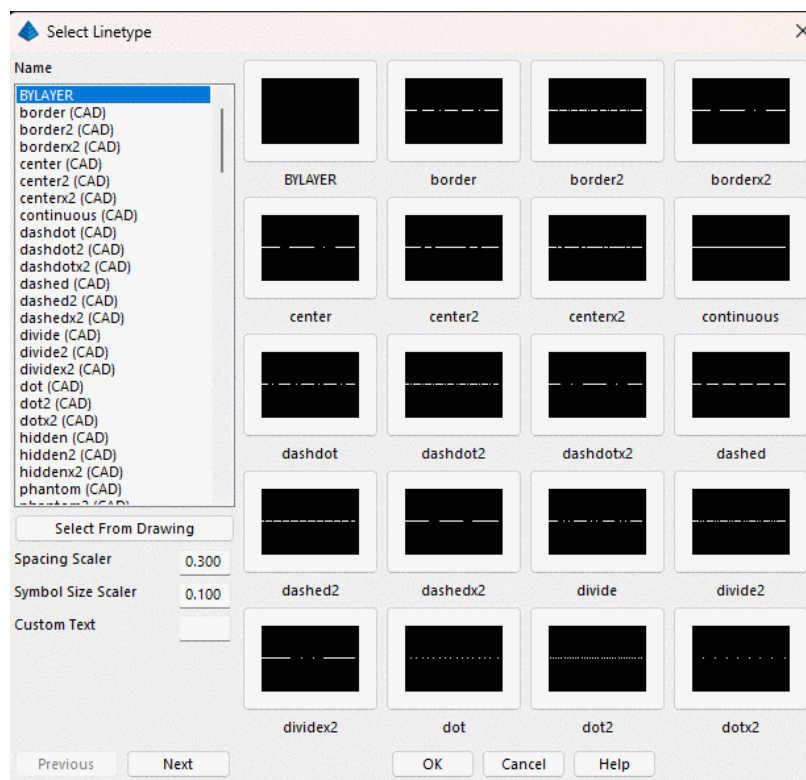
The **Fit Point Description Instead Of Symbol** option is a method to drawing labels that are positioned by these points such as road markings. To set a custom text to label, you can use the `///` (Replace Description) special code. For example with a code called MARK, the description could be "MARK 3RD `///`BUS LANE". Or turn on the Automatic Multi-Point Special Code and then the "3RD" isn't needed and the description could be "MARK`///`BUS LANE". For the text control points, lower-left = 0,0, upper-right = 1,1, upper-left=0,1, and lower-right = 1,0.

Draw 2nd Symbol: This option creates a second symbol on each point. For example, this additional symbol can be used to add a 3D symbol to a 2D symbol used as the first symbol or simply add a second symbol to the point. Besides selecting the symbol name, there are settings for the Symbol Size Scaler Layer and Rotation which mimics the settings described for the first symbol with the addition of matching the 1st symbol size and rotation.

Linetype Tab



Set Linetype: Line work can be drawn in any of the special linetypes or with the linetype for the layer ("BY-LAYER"). There are three types of pre-defined linetypes: **CAD**, **Entity** and **Continuous**.



The type is shown as part of the linetype names in the list. The **CAD** linetypes are the default linetypes available

in AutoCAD and IntelliCAD. The **Entity** linetypes insert text or symbol entities at the linetype interval. These linetypes are the same as used with the Annotate->Polyline To Special Line command. The **Continuous** linetypes define a special linetype in CAD and create continuous polylines with that special linetype. These linetypes are the same as with the Annotate->Change Polyline Linetype command. Besides these pre-defined linetypes within Field-to-Finish, you can also use any linetype that is defined in the drawing by entering that linetype name in the linetype edit box or by picking the **Select From Drawing** button within the Set Linetype dialog. The spacing and size of the special linetypes is determined by the CAD LTSCALE system variable and by the 3 controls shown above or in the field code settings:

- Line Type Spacing Scaler - controls the spacing between the text and/or symbols inserted in the linetype for either the Entity or Continuous linetypes.
- Line Type Text Scaler - controls the size of the text and/or symbols inserted in the linetype for either the Entity or Continuous linetypes.
- Custom Text - allows text entry for the "User Defined" linetype.

Note: The special linetype "**Hedge**" is drawn with a width set by Line Width. The offset special codes of OH or OFB can also be used to control the width of the "hedge" linetype. The special linetype "**userdash**" is drawn with user specified distances for the length of the dash and the length of the gap between dashes.

Line Width: This controls the width for the linework. Only applies to 2D polylines. The LTW special code can also be used to set the line width for a specific line.

Linetype Text Style: Sets the text style created in the *Style* command in CAD

Linetype Text: This is the text that is used for the user-defined linetype. To use, Set Linetype to either Other_E or UserDef_C. Then this text will be used for the linetype. For example, if you have a code for a 8" PVC pipeline, then you could set this text to 8" PVC. This linetype text will use the font set in the **Linetype Text Style**. If this style is blank, the text will use the current font.

Linetype Spacing Scaler: This is a scaler value that is multiplied by the CAD LTSCALE system variable to give the distance between text or symbols in the line.

Linetype Text Scaler: This is a scaler value that is multiplied by the CAD LTSCALE system variable to give the size of the text or symbol in a line.

Flip Linetype: This option switches the side for the linetype which applies to non-symmetrical linetypes like the treeline or guard rail. The Special Code LTF can be added to the first code in the line sequence to accomplish this as well.

Smooth Polyline: This option applies a modified Bezier smoothing to the polyline. The smoothed polyline will pass through all the original points.

Hard Breakline: This option will tag the 3D polylines created with this code as hard breaklines. In *Triangulate & Contour*, contours are not smoothed as they cross hard barriers.

Label Lines and Arcs: This option will label the lines and arcs drawn using the specified Line/Arc and General Label Settings.

- **Line/Arc Label Settings:** *Selected Setting* Select an existing Label Settings file (AAN) by clicking the **Select** button or modify the file by clicking the **Edit** button.
- **General Label Settings:** *Selected Setting* Select an existing Label Settings file (ADF) by clicking the **Select** button or modify the file by clicking the **Edit** button.

Label Areas: This option will label closed areas within the specific tolerance. The areas will be labeled using the specified Area Label Settings.

- **Area Tolerance:** This is the tolerance used to determine if an area is closed and should be labeled. If a polyline closure is greater than this tolerance, the area will not be labeled.
- **Area Label Settings:** *Selected Setting* Select an existing Label Settings file (ARS) by clicking the **Select** button or modify the file by clicking the **Edit** button.

Separate Offset Layer: This setting uses another layer name for polylines created using an offset special code.

Label 3D Offset Elevations: This option labels the elevations and symbols of the vertices for polylines created using an offset special code. The Attribute Format must be set to Text Attribute or both and the Elevation attribute enabled for this option to take effect.

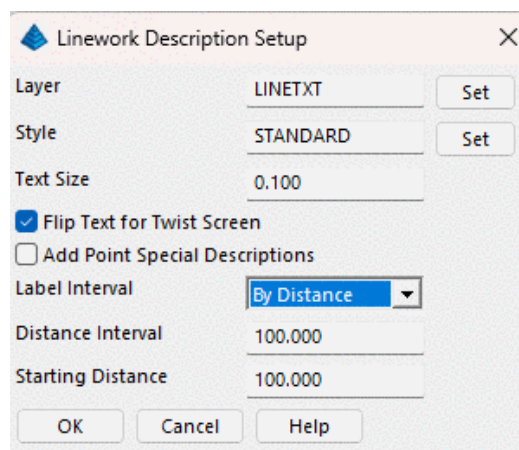
Closed Polyline for Offset Both: This option will create a closed polyline from the left and right offset polylines when the OFB (Offset Both) special code is used.

Connection Order: The points of a distinct code can be connected in their point number order, sequential, or by nearest found which creates the line by connecting each point to the next closest point.

Tie: The options for this is **Open** or **Closed**. When set to Open, the polyline created will end at the last point in the line string. When set to Closed, the last point in the line string will close back to the first. For example if you have points 1, 2, 3, and 4 with the code BLDG and Tie is set to Closed, the linework will be drawn from point 1 to 2 to 3 to 4 and then back to point 1, closing the figure.

Linework Description: This allows for an optional description which is labeled along linework created by the code.

- The **Select GIS** button chooses a GIS attribute to use for the linework description from the GIS Table. The program checks all the points used for the linework for the GIS attribute for this description. To setup the GIS attributes, the Linework Description string is formatted with "[GIS:attribute]". For example, if you have GIS attributes for "Client" and "Weather", then the string to label both attributes with prefixes could be "Client=[GIS:Client], Forecast=[GIS:Weather]" or without prefixes could be "[GIS:Client] [GIS:Weather]".
- **Description Setup** button displays a dialog to control the layer, style and size for these labels.



Layer - Sets the layer for the line text

Style - sets the text style for the line text

Text Size - sets the text size scaler for the line text

Flip Text for Twist Screen - orients the linework labels to be right side up for the drawing current twist view

Add Point Special Descriptions - adds a special description created using the prefix or suffix special codes and adds that to the description. For example if a sewer line were drawn with the Linework Description of Sewer and a prefix was added to the code of /8", the linework would be labeled as 8" Sewer

Label Interval - sets the option for the frequency of labeling the linework description. The options are from 1 to 4 labels per line string or by a Distance Interval.

- Distance Interval - sets the distance between labels in terms of drawing units.
- Starting Distance - sets the distance from the beginning of the line for the first description label.

Edit and Set Template: For 3D polyline codes, this option allows you to assign a template (.TPL) file to the code. The code points act as the base line for the template and the program will draw parallel 3D polylines for each break point (grade ID) in the template. The template file is defined in the Civil Design module or by using the Edit button. In the Edit Template dialog, there's a special setting for Field-to-Finish for the Offset Target Grade. This Offset Target Grade controls which grade in the template to use as the center line for the code points. If this field is blank, then the start of the template is used.

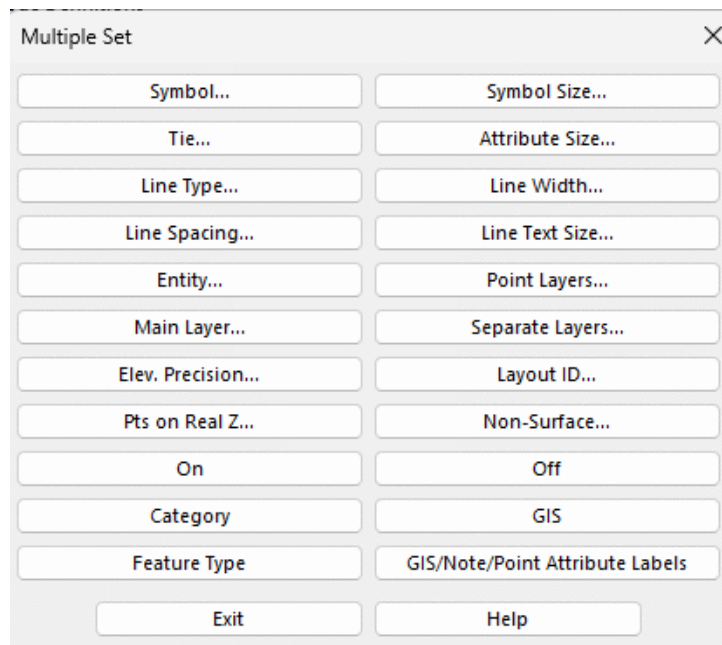
The templates are dynamic and can be adjusted along the 3D polyline by using the OH (offset horizontal) and OV (offset vertical) special codes. For example, when there is a curb template that flattens when it comes to a driveway, then an OV0 code could be used at the driveway start point and then use OV.5 at the driveway end point to restore the curb to 0.5 feet high.

Use Template IDs as Layer option will use the grade IDs from the template for the layers of the polylines created.

Hatch Closed Areas: This option applies for codes used to draw closed linework which is used as the perimeter for a hatch area. The hatch settings include the hatch pattern name, scale and layer.

Edit (Multiple Codes)

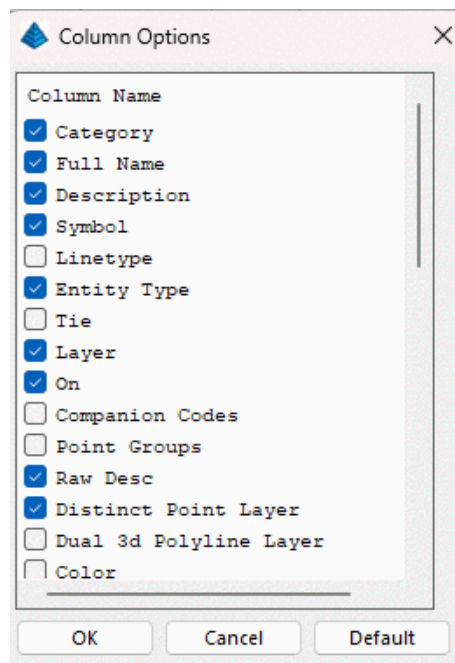
If more than a single code is selected, the Edit button will allow a limited number of "Global Edits" to the codes.



Each of these options will effect all codes selected and is a great way to make changes to a large number of codes at the same time. For example, if you wanted to set all the codes to a particular attribute size, use the Select All option and then Edit. You can then set all codes in the library to the same size attribute.

Code Definitions (continued)

Column Options: Controls the displayed columns in the display window.



Each item can be toggled on or off and the display window will update accordingly. It is important to remember that each field in each column can be modified directly from the display window. This helps greatly in the ease of editing codes in the table. Turn the columns on of interest and use copy/paste or a drop down selection as is available.

Select All: This option selects all the codes within a selected category. This can be used when only wanting to process a couple of codes. For example, use the select all option to select all the codes and then turn them off. Now select the codes for processing and turn them on. Also it can be used to make a global change to all the codes.

Move Category: Moves the selected code(s) to a predefined category.

Add: Adds a new empty code definition which is inserted in the list at the position after the currently selected one. If none are selected for positioning, the new code is placed at the top. Only one code definition may be highlighted before running this routine.

Copy: This option copies the definition of a selected code. It opens the Edit Field Code Definition dialog and copies the definition of the selected code to the appropriate settings. It does not copy the name of the code. It is a time saving tool to use when creating codes that are similar with only a couple of differences.

Delete: This command will remove the highlighted code definitions from the list.

Search: Allows you to search for a specific code in the list.

Move Up/Move Down: allows for the manual resorting of the code list by moving the selected code either up or down.

Feature Settings

Tree Setup:

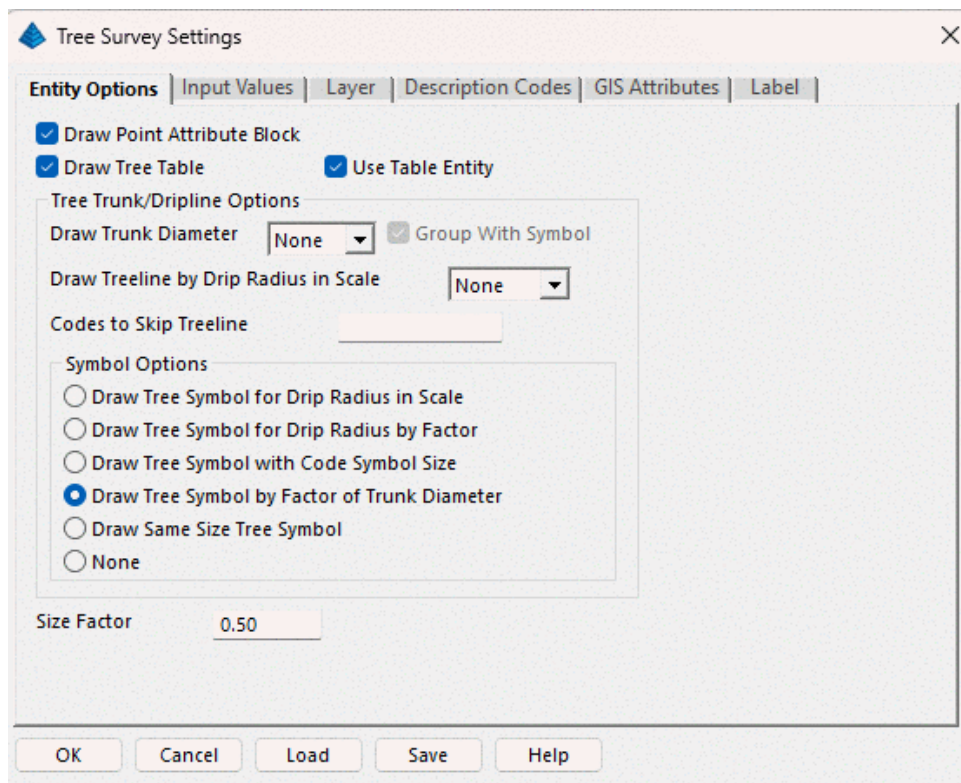
Tree surveys can be coded simply by using general Field-to-Finish coding methods such as defining a code for a tree ("OAK") with a tree symbol and using the SZ special code for sizing the symbol. The **Tree Feature** provides more power and flexibility in cases where a simple symbol is not sufficient. The tree survey works with three optional attributes for each tree: trunk, drip and tag. Trunk is the diameter of the tree trunk. Drip is the radius of the tree canopy. Tag is an id for the tree for reporting.



Tag	Trunk	Drip	Comment
T100	12"	15	MAPLE
T101	10"	10	CEDAR

Important: The Tree Survey Settings apply to codes that are set to a Feature Type of Tree. To set the Feature Type, go to Edit Codes and then the General tab of the Edit Field Code Definition dialog.

Entity Options Tab:



Draw Point Attribute Block -controls whether to draw the point block with the point #, elevation and description attributes.

Draw Tree Table - This option makes the program prompt for whether to draw a tree table when the program finds a couple points with tree codes. Otherwise, there must be many tree codes to have the program prompt for creating a tree table. The Use Table Entity option draws the table as a Carlson Table Entity. Otherwise the table is drawn with regular CAD lines and text.

Draw Trunk Diameter - Sets whether to create a circle or solid with the trunk diameter. The Group With Symbol option creates a CAD group to combine the tree and trunk symbols.

Draw Treeline by Drip Radius in Scale - shrink wraps the tree drip lines to get the overall treeline perimeter. The perimeter polyline can be drawn either as Bubbles or Smooth. The Bubbles creates a treeline style polyline with a series of arcs. The Smooth creates a regular polyline.

Codes to Skip Treeline - allows you to skip specific Field-to-Finish codes from using with the treeline. If you have more than a single code to skip, use either a space or comma to separate the codes.

Draw Tree Symbol for Drip Radius in Scale - draws individual symbols for each tree using the symbols defined in the code table and scaled by the drip size attribute.

Draw Tree Symbol for Drip Radius by Factor - draws individual symbols for each tree using the symbols defined in the code table and scaled by the drip size attribute and the Size Factor from this dialog.

Draw Tree Symbol with Code Symbol Size - draws individual symbols for each tree using the symbol name and size defined in the code table.

Draw Tree Symbol by Factor of Trunk Diameter - draws individual symbols for each tree using the symbols defined in the code table and scaled by the trunk size attribute multiplied by 12. For example, a 10" trunk size is drawn as a 10ft symbol.

Draw Same Size Tree Symbol - draws individual symbols for each tree using the symbols defined in the code table and at size of 6.

None - simply does not draw a tree symbol

Input Values Tab:

The screenshot shows the 'Tree Survey Settings' dialog box with the 'Input Values' tab selected. The dialog has a title bar with a close button (X) and a tabbed interface with the following tabs: 'Entity Options', 'Input Values', 'Layer', 'Description Codes', 'GIS Attributes', and 'Label'. The 'Input Values' tab contains the following settings:

- Default Tree ID to Point ID
- Begin Tree ID from: 100
- Input Trunk Value: Diameter
- Input Drip Value: Radius
- Canopy Companion Code: DRIP

At the bottom of the dialog are five buttons: 'OK', 'Cancel', 'Load', 'Save', and 'Help'.

Default Tree ID to Point ID - option uses the point number for the tree tag unless the point description contains a tree tag.

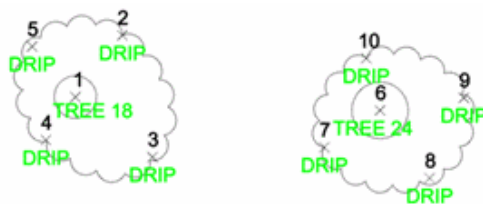
Begin Tree ID From - this is the number to start the incremental numbering of tree tags from. Each point with a tree feature code will be numbered consecutively thereafter.

Input Trunk Value - controls whether the trunk size is entered as a radius or diameter.

Input Drip Value - controls whether the canopy drip size is entered as a radius or diameter.

Canopy Companion Code - this tree coding method handles a point at the trunk and then a series of 3 or more points for the canopy. The point description coding uses separate codes for the trunk and the canopy points.

The program draws a Bezier smooth polyline thru the points for the canopy. In this example, points 1 and 6 have the tree feature code from the Field-to-Finish code table, and the rest of the points have the Canopy Companion Code.



Layer Tab:

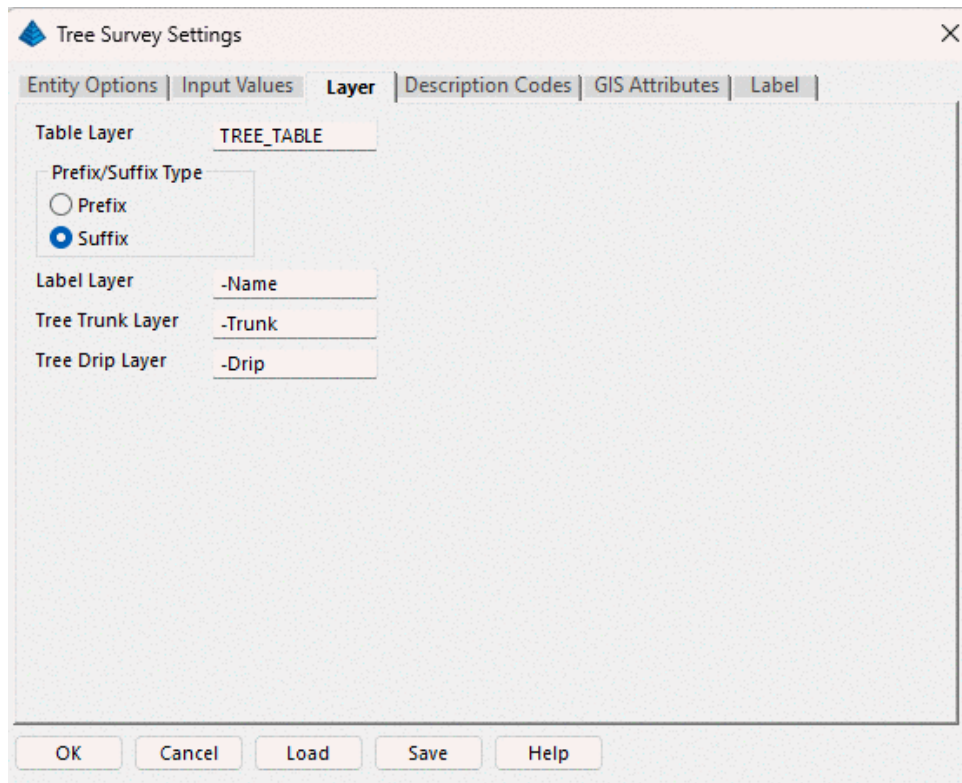


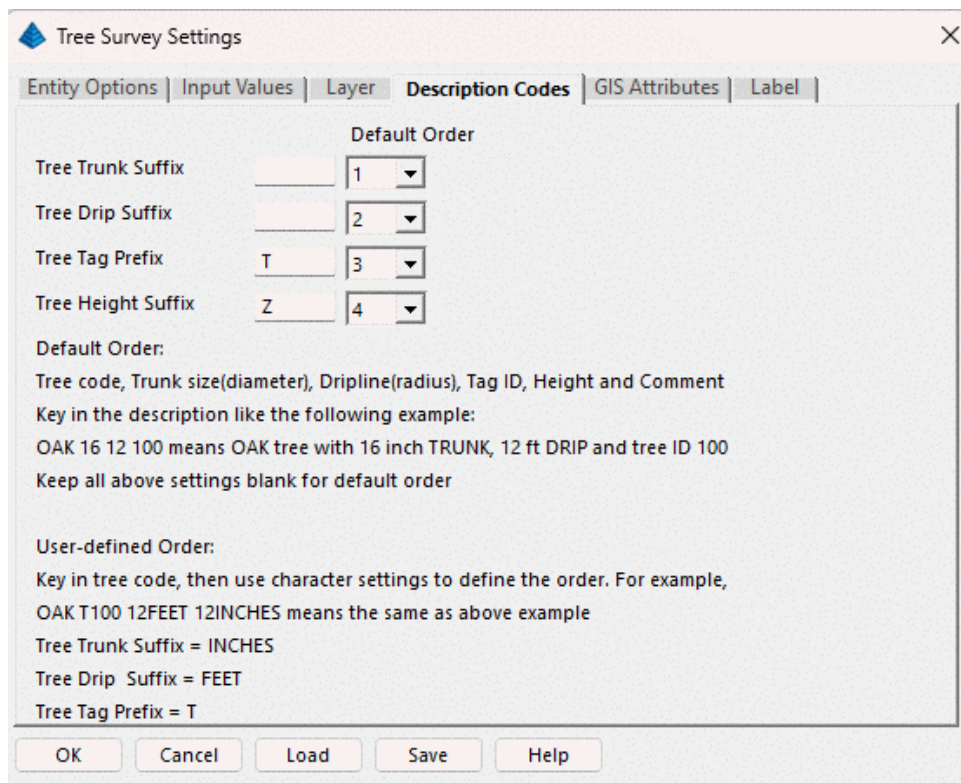
Table Layer - sets the layer for the optional tree table to be drawn

Prefix/Suffix Type - designates whether the layer names will be added as a suffix or prefix to the Main Layer as designated in the code itself.

- Label Layer - added to the layer name for the tree label ie. Maple, Oak etc.
- Tree Trunk Layer - added to the layer name for the label of the trunk size
- Tree Drip Layer - added to the layer name for the label of the drip line size

Example: the main layer for the code DT is set to Tree. The tree labels would be placed on the layers Tree-Name, Tree-Trunk and Tree-Drip

Description Codes Tab:



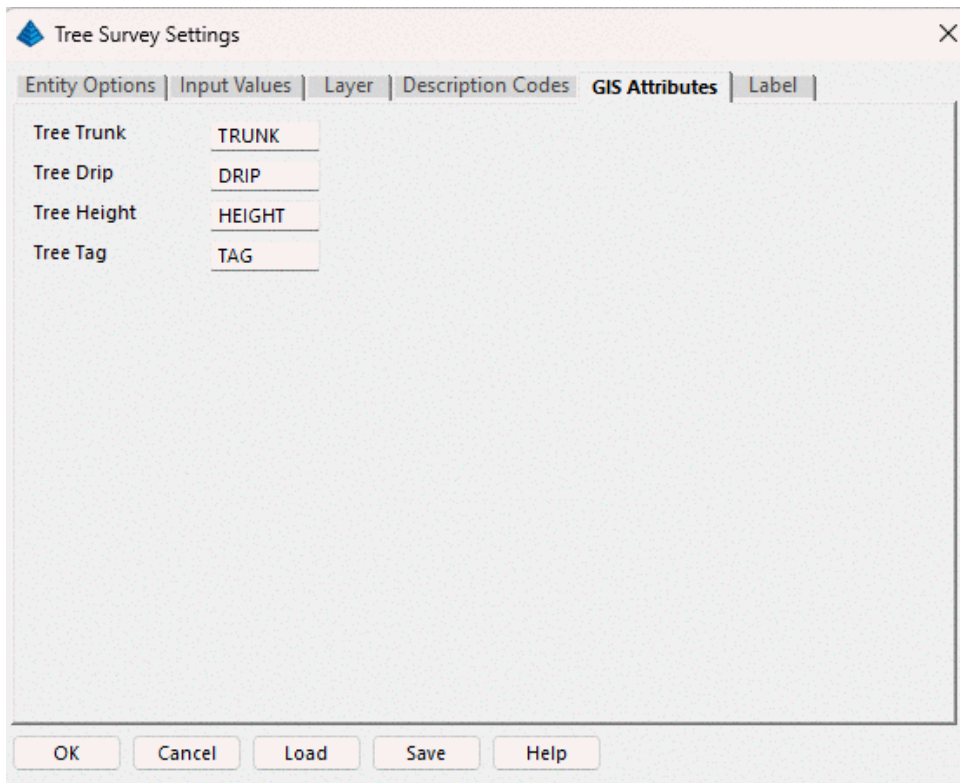
There are settings to help identify the tree attributes in the point description. The program looks for the trunk size, drip size, tag ID and height in the point description after the tree code. By default, the program expects the attributes to be in the order of trunk size, drip size, tag ID and height. Here's an example default order:

OAK 16 12 100 28

where OAK is the tree code from the code table, 16 is the trunk diameter, 12 is the drip radius, 100 is the tag ID and height is 28.

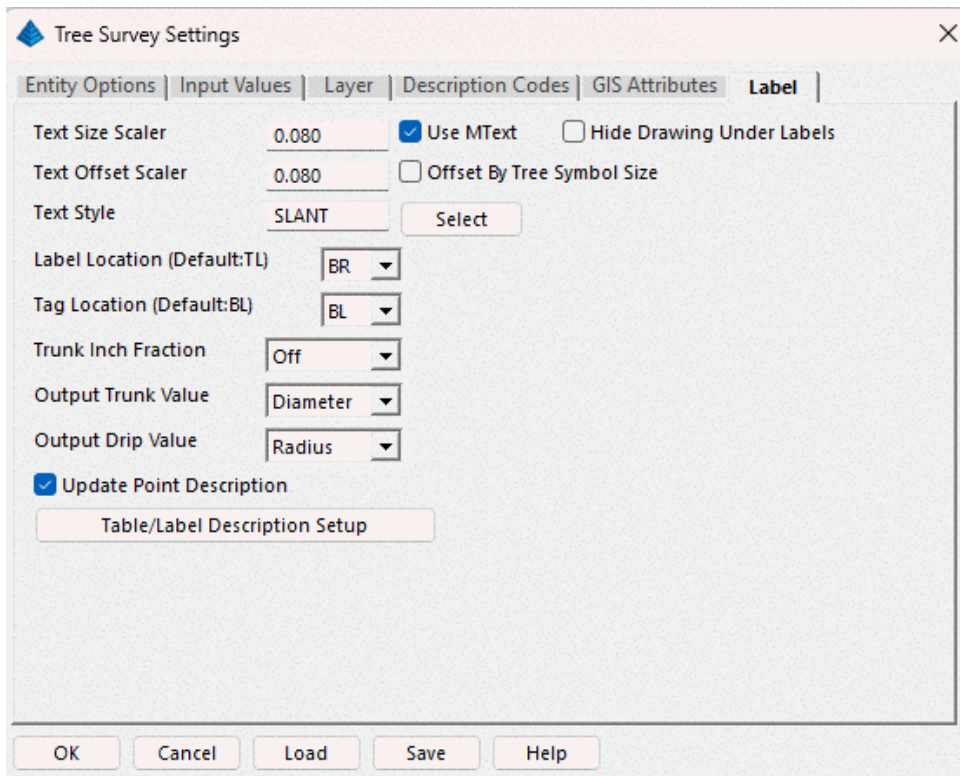
You can change the order that the program looks for the attributes by setting the Default Order numbers. Another way to change the attribute order is to use the suffix/prefix settings to identify the attributes. When the program finds a specified prefix or suffix, it tells the program which attribute to use. For example, if the Trunk Suffix is "in" and the Drip Suffix is "ft" and the Tag Prefix is "T", then OAK T100 16in 12ft means tag ID of 100, trunk diameter 16 and drip radius 12 feet.

GIS Attributes Tab:



In addition to looking for the tree attributes in the point description, the program can also read these attributes from GIS fields. On the **GIS Attributes** dialog tab, you can set the GIS field names for the tree attributes. The data collected in the field would be stored in the .vtt file which would contain the values for each of the defined attributes. Normally, this option would be used in place of adding the tree data directly to the code description.

Label Tab:



There are settings for the tree text labels for the size, offset from trunk center, style and location. When creating a tree table, only the tag text is labeled. Otherwise, the label is drawn.

Text Size Scaler - sets the scale size of the text the value of which is multiplied by the horizontal scale as set in the drawing setup. Options to **Use Mtext** and **Hide Drawing Under Labels** (applies a wipeout or text mask) are available.

Offset Scaler - applies a scale factor to control the placement of the text as it pertains to the point. **Offset By Tree Symbol Size** moves the labels beyond the tree symbol to avoid overlap between the labels and the symbol. The amount to move is factored by the Text Offset Scaler.

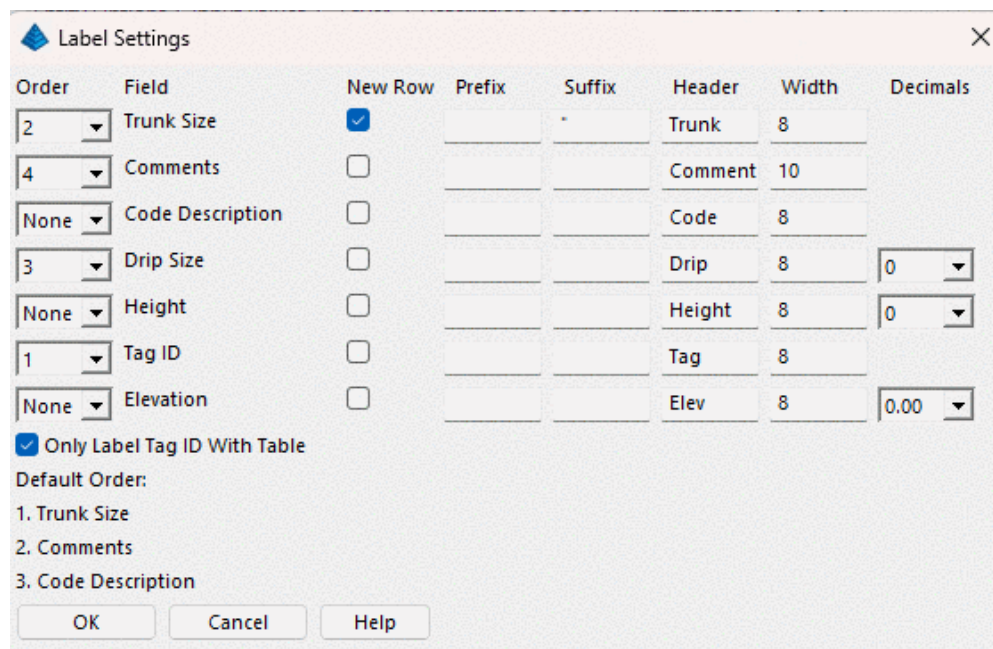
Trunk Inch Fraction - sets the precision in inches to use for the trunk size labels.

Output Trunk Value - sets whether to label the trunk size as a radius or diameter.

Output Drip Value - sets whether to label the canopy (drip line) size as a radius or diameter.

Update Point Description - sets whether to only use the label setup for the table and not the point drawing description. With this toggled on, the description attribute in the drawing will be updated to reflect the values to be used in the table as opposed to the coding as input in the field. The code description in the coordinate file is not affected by this setting.

Table/Label Description Setup - opens a Label Settings dialog box that controls the fields to be labeled in the drawing as well as included in the tree table.



For each field, there are settings to control the placement, prefix, suffix and the caption for the table columns. These settings effect both the labeling in the drawing and the table.

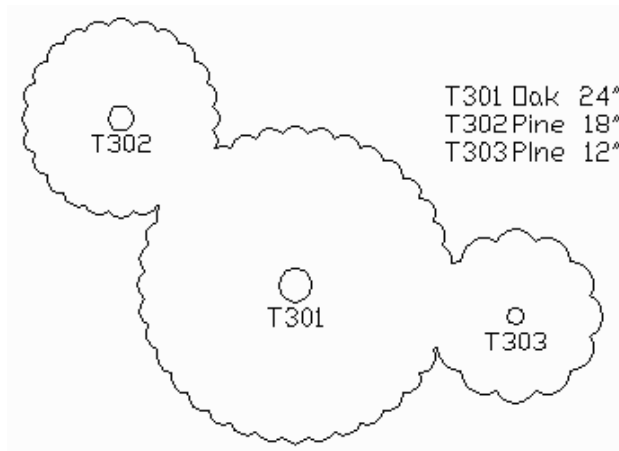
- Order - sets the order for the field
- New Row - when toggled on, places the field on a new row for the description label. This does not affect the tree table.
- Prefix - adds an optional prefix for the label

- Suffix - adds and optional suffix for the label
- Header - designates the caption for the top of the column in the tree table
- Width - sets the width of the field for the tree table column
- Only Label Tag ID With Table - when enabled, tree tags will only be used when the option for a tree table is enabled.

When Field-to-Finish draws entities, the program checks for codes set as tree features and applies the settings from the Tree Survey dialog. When tree features are found, the number of trees are reported along with a prompt for whether to draw a tree table. The tree table has the tag ID, code description and trunk diameter.

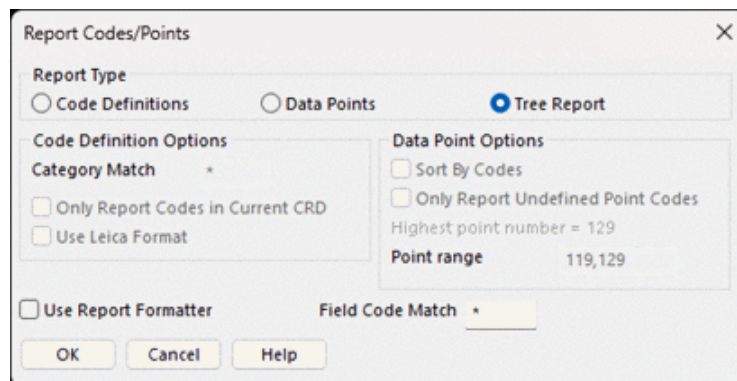
Here is an example with the following three points:

Point#	Northing	Easting	Description
1	4994.73	4923.15	OAK 24 38 301
2	5034.59	4881.40	PINE 18 24 302
3	4987.32	4975.79	PINE 12 20 303

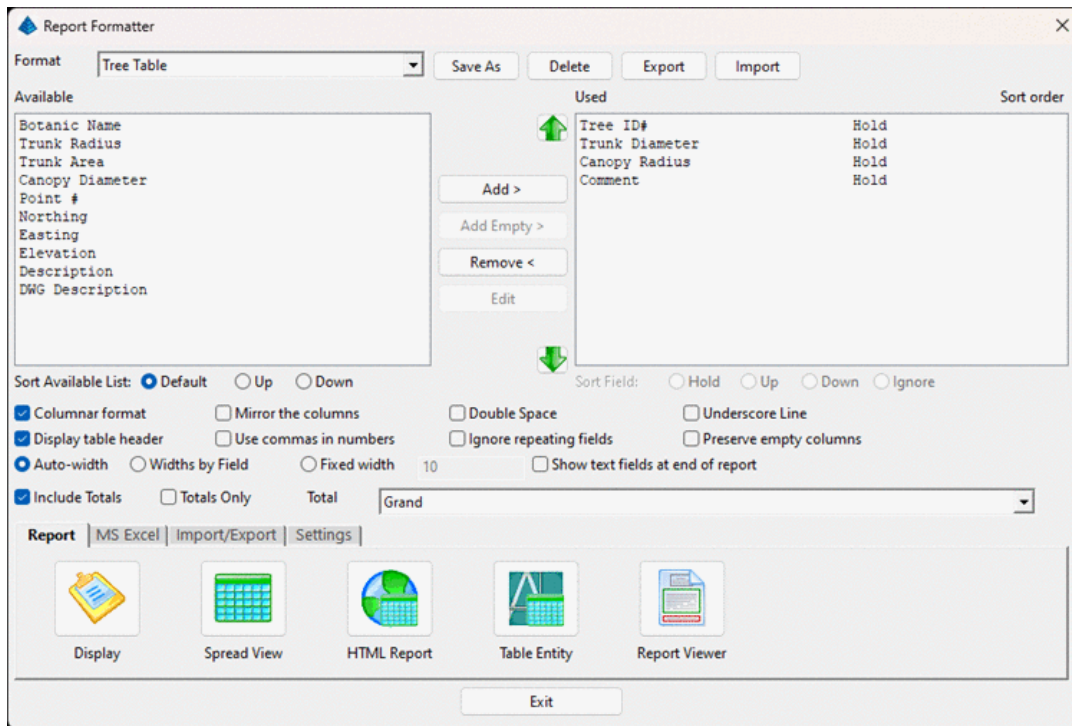


Drip line drawn as Treeline method along with a tree table.

Another feature of Tree Survey is the **Tree Report** under the Report Codes/Points function. The Report Formatter option can be used to make a custom report and output to Excel or create a custom table in the drawing. To use this feature, use the Report Codes/Points option found in the Code Table or Draw Field to Finish dialog box.



Select the Tree Report option and click <OK>. Alternatively, enable the Use Report Formatter option and click <OK> to reveal additional options for the report.

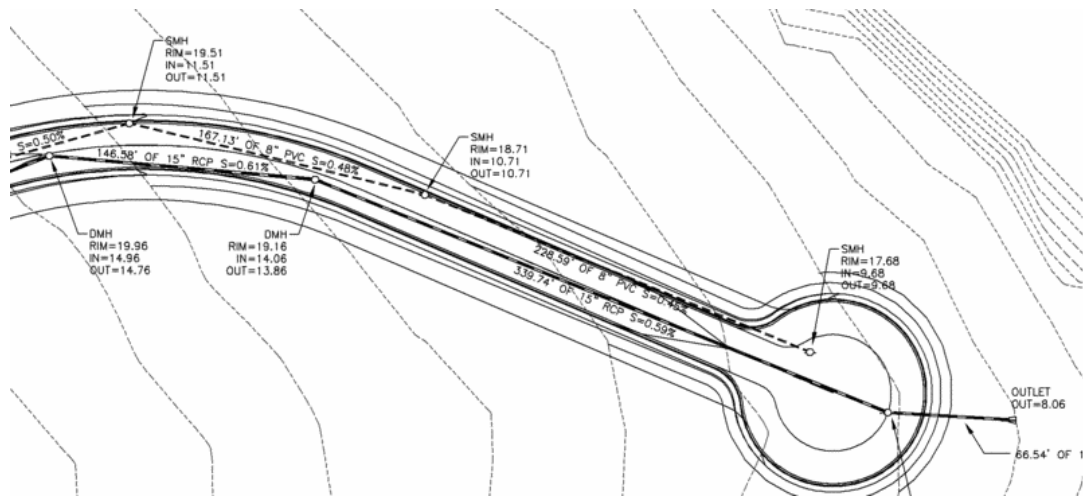


Example Tree Table Report

Tree ID# Trunk Diameter Canopy Radius Comment
 T100 12" 15' MAPLE
 T101 10" 10' CEDAR

Pipe Setup:

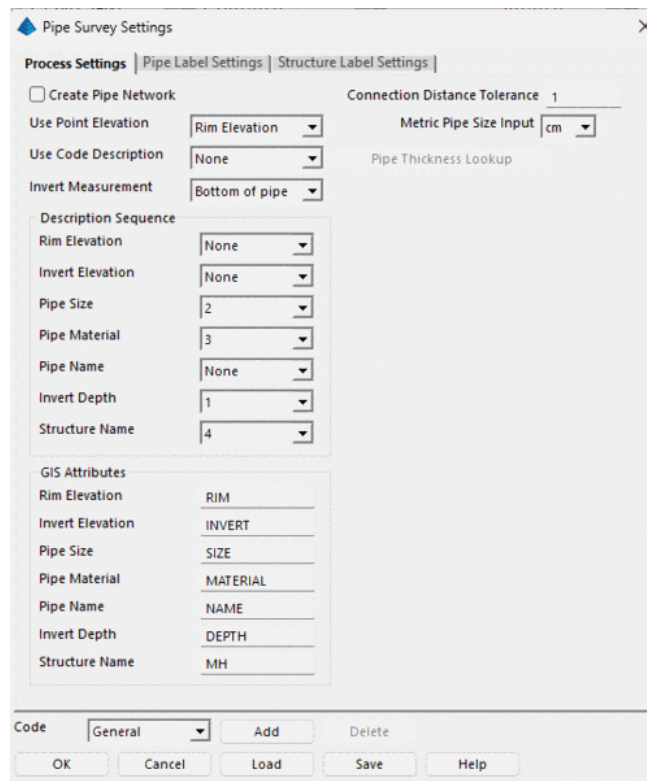
The **Pipe Feature** adds additional properties and options to the drawing a pipe line or network. Size, material and name are some of those options. in addition, the Pipe Feature allows more control over labeling of the pipe parameters and ability to draw the pipe in 3D.



Important: The Pipe Settings apply to codes that are set to a Feature Type of **Pipe**. To set the Feature Type, go to Edit Codes and then the General tab of the Edit Field Code Definition dialog.

When a code is tagged as a Pipe Feature, the program looks for additional parameters after the code. The parameters are separated by spaces in the point description and in the order specified in the Description Sequence in the settings dialog. For example, if code 'P' is defined as a Pipe Feature and the order of size, material and name, then a description of 'P 8 PVC 31' would define a pipe point for an 8 inch pipe of material PVC and name of 31. These pipe parameters are optional. The program will use the provided parameters and leave the rest blank. Besides using the point description, the pipe attributes can alternatively be set from GIS attributes using the attribute names setup in the dialog.

Process Settings Tab:



Create Pipe Network - when enabled, a SewerNET pipe network (.SEW) file will be created from the points connected with a pipe feature type. The **Connection Tolerance** limits the distance between points to be considered pipes belonging to the same structure. This is particularly useful when locating the actual invert for each pipe within a structure. The .SEW file can be utilized in the Sewer Network (**SewerNET**) contained in the **Hydrology** module. From there, the network can be edited, analyzed, and used for plan and profile creation.

Use Point Elevation - sets the control parameter for the point elevation. The field located point can be used as the **Rim Elevation**, the **Invert Elevation** or can be set to **None**.

Use Code Description - the code description can be used as the **Pipe Material**, **Pipe Name** or set to **None**.

Invert Measurement - when a pipe is field located directly, this control if the elevation will be used as the **Top** of Pipe or **Bottom** of Pipe.

Description Sequence - sets the order for the code descriptions as they are input into the code. The order of these setting must match the method in which the code is created. The fields that can be used and ordered are:

- Rim Elevation

- Invert Elevation
- Pipe Size
- Pipe Material
- Pipe Name
- Invert Depth
- Structure Name

For a structure with multiple inverts, pipe sizes and/or materials, the values should be in the description separated by commas. For example, for a structure with two incoming PVC pipes with sizes of 15" and 24", and inverts at depths 5.9 and 5.5, the description would be 'P 15,24 PVC 5.9,5.5'. If a value, such as the material does not change between structures, it need not be reentered.

GIS Attributes - using GIS attributes mimics the process described above. The difference is how the values are input into the pipe feature. Using the GIS attribute option requires that the data be entered as an attribute that is carried with the point coordinate file. The advantage to this method is that it does not necessitate long code description. Instead, the point attribute data can be entered in in a tabular format.

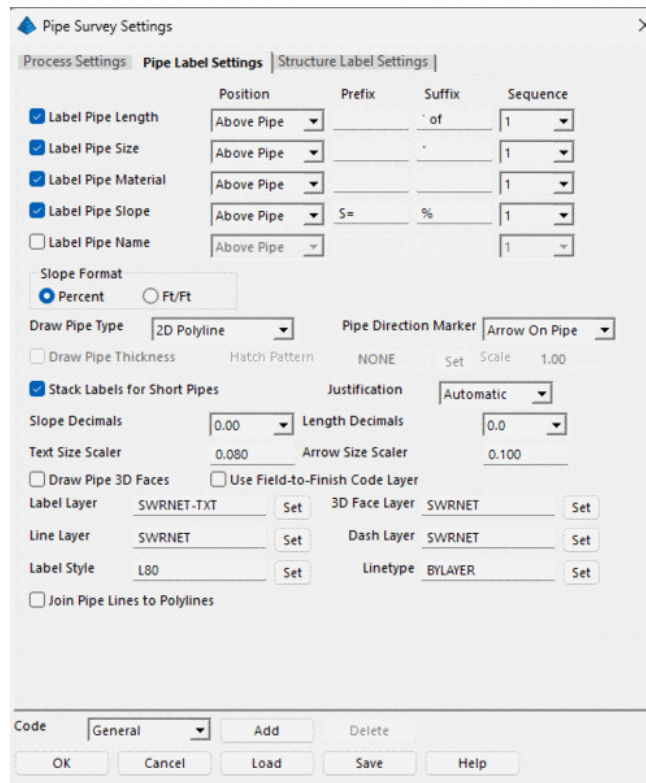
Example:

```
DEPTH=5.0  
SIZE=12  
MATERIAL=CONCRETE
```

The attributes can be created in advanced or in the field using SurvCE/PC and can be set up for prompts during the collection of the data.

Code - the Code defaults as General which would apply to all pipe feature codes. You can also Add a code that is contained in the Code Table as a Pipe Feature which can then have individual settings. For example, you could have a code called STORM for storm sewers and one called SANITARY for sanitary sewers. Each pipe network can have unique settings for layers and labels.

Pipe Label Settings Tab:



Label Pipe Length, Size, Material and Slope - all have controls over the position and content of the labels:

- Position - sets the position of the label as above or below the pipe
- Prefix - adds a prefix to the label
- Suffix - adds a suffix to the label
- Sequence - sets the sequence of the pipe label

Slope Format - can be expressed as a percentage of slope or foot per foot (ft/ft)

Draw Pipe type - sets the type of pipe to be drawn.

- 2D Polyline - draws a consistent 2D polyline at elevation 0
- 3D Polyline - draws a 3D polyline with each vertex matching the point elevation
- Double/Width - draws 2 dashed lines offset from each other the size of the pipe with no scale exaggeration.
- Dashed/Width - draws a solid and dashed line whose width is the size of the pipe with no scale



- None - does not draw line work between points
- Field-to-Finish - uses the linetype as selected in the Field to Finish Code Table

Pipe Direction Marker - draws an arrow in the direction of flow for each pipe segment. The choices are an **Arrow** on the Pipe, **Parallel** to the pipe or **None**

Draw Pipe Thickness - a pipe thickness (if available) can be drawn outside the pipe with the exception of the 2D or 3D polyline options.

- **Hatch Pattern** - an optional Hatch Pattern can be used to depict the pipe thickness. The selection and scale of the hatch pattern is also available.

Stack Labels for Short Pipes - automatically stacks the selected labels to make them look more readable.

Justification - the text justification for the labels can be set as Automatic, Left/Right or Centered.

Slope Decimals - sets the precision for the pipe slope labels

Length Decimals - sets the precision for the pipe distance labels

Text Size Scaler - sets the text size in terms of a scale factor which is applied to the horizontal scale as set in the Drawing Setup. (text scaler x horizontal scale = text size)

Arrow Size Scaler - sets the Arrow size in terms of a scale factor which is applied to the horizontal scale as set in the Drawing Setup. (text scaler x horizontal scale = text size)

Draw Pipe 3D Faces - when enabled, a 3 dimensional pipe as 3D Faces is drawn separately from the other pipe lines which can be viewed in Orbit view or Carlson's Drawing Viewer'

- **Use Field-to-Finish Code Layer** - draws the 3D pipes on the layer established in the Code Tablee

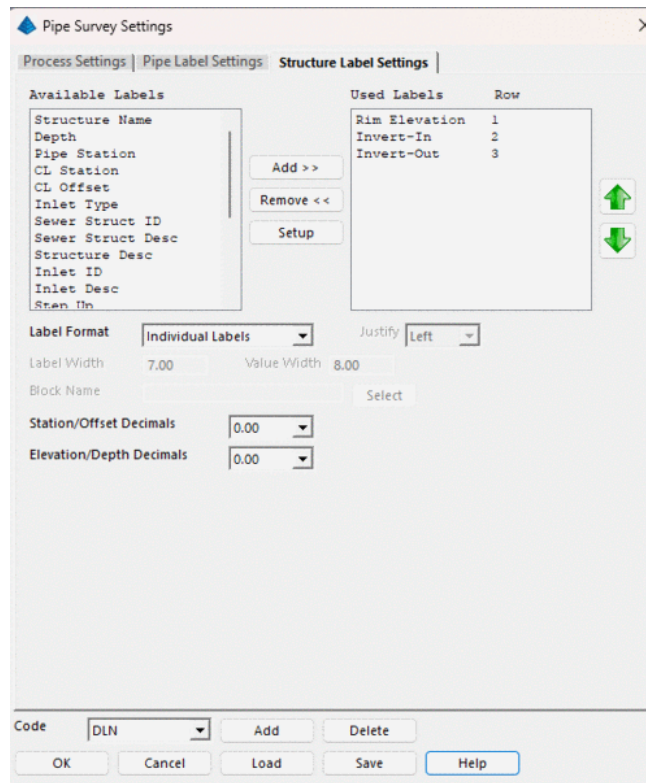
Layers for the Labels, Lines, 3D Faces and the Dashes (when using the Dashed/Width option) can be typed in or selected using the Set button.

Label Style - sets the text style for the pipe labels

Linetype - sets the linetype for the line work. This will override the linetype defined in the layer unless bylayer is selected and does not apply to the Dashed/Width option.

Join Pipe Lines to Polylines - joins all connected pipes into a single polyline. This does not work for the Double/Width or Dashed/Width pipe types

Structure Label Settings Tab:



Available/Used Labels - to setup the Pipe Structure Labels, highlight any available pipe structure labels on the left side of the dialog box and either double click or hit the Add button to move it to the Used Labels portion of the dialog box on the right.

Label Format - select the desired label format from the drop down list.

- Individual Labels - labels text at each structure location
- Data Table - creates a Data Table at each structure location. When this option is used, you are able to control the **Justification** of the table as well as the **Width** of the Labels and Values. These options only become available when the Data Table option has been selected.
- Attribute Block - adds the data for each structure to a predefined block containing the relative attributes. When this option is used, there is a Select button to select the block to be used.
- Along Pipe - labels the structure information along each respective pipe segment.

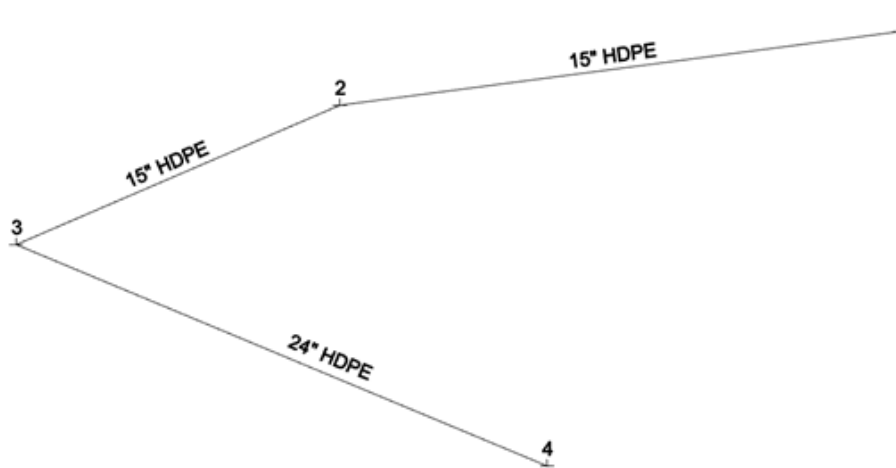
Station/Offset Decimals - sets the precision for the CL Station and Offset Labels

Elevation/Depth Decimals - sets the precision for Elevation labels of Rim, Inverts, and Depth.

Example coding of a pipe network. PIPE code is defined as a Pipe Feature. The 1st description is the pipe size. The 2nd description is the pipe material. The 3rd description is the pipe depth. Point 3 has two sizes and two depths separated by commas because two pipe connect at this point.

Point	Description
1	PIPE1 15 HDPE 4.75
2	PIPE1 15 HDPE 6.15

- 3 PIPE1 15,24 HDPE 5.96,5.5
- 4 PIPE1 24 HDPE 9.9



Example coding for 2 separate pipes using END code to separate the pipes. CUL code is defined as a Pipe Feature. 1st description is the Pipe Material and 2nd description is the pipe size. In this case, turn OFF the Create Pipe Network because the separate pipes are using the same code of CUL.

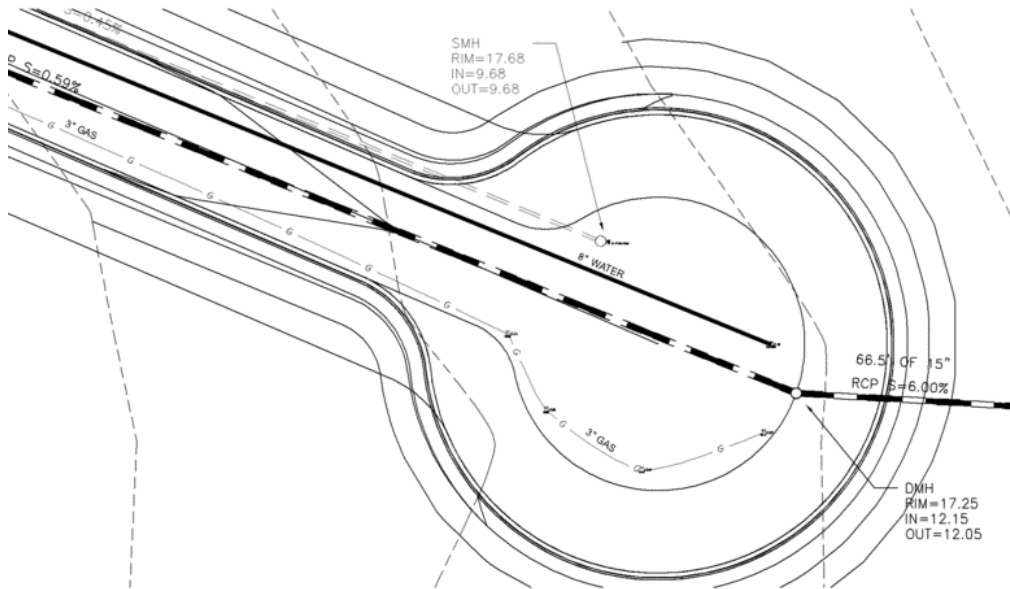
Point	Description
7	CUL RCP 18
8	CUL END RCP 18
14	CUL RCP 18
15	CUL END RCP 18

Example coding for 2 separate pipes using number suffix on codes to separate the pipes. CUL code is defined as a Pipe Feature. 1st description is the Pipe Material and 2nd description is the pipe size. In this case, the Create Pipe Network can be applied. The separate pipes are using separate codes of CUL1 and CUL2.

Point	Description
7	CUL1 RCP 18
8	CUL1 RCP 18
14	CUL2 RCP 18
15	CUL2 RCP 18

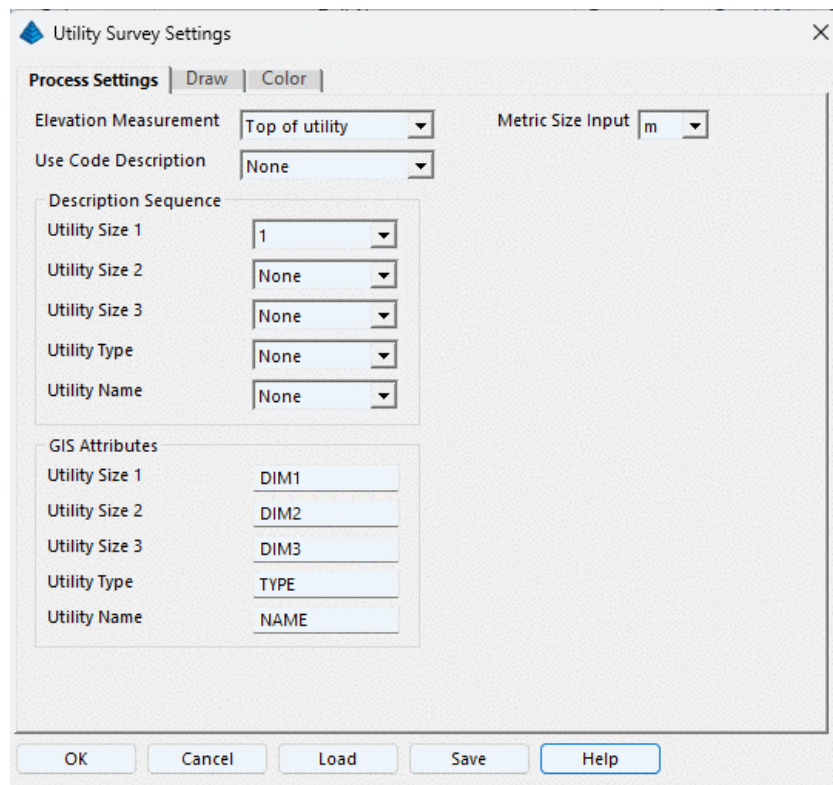
Utility Setup:

The Utility Feature works similar to the Pipe feature. The differences with utility are supporting different pipe shapes (circle, rectangle, etc), different types of utilities (gas, electric, etc), and not having manhole structures. For each utility type, there are settings for pipe shape, color, labels and drawing 2D Polylines, 3D Polylines and 3D Faces.



Important: The Utility Settings apply to codes that are set to a Utility Feature. To set the Feature Type, go to Edit Codes and then the General tab of the Edit Field Code Definition dialog.

Process Settings Tab:

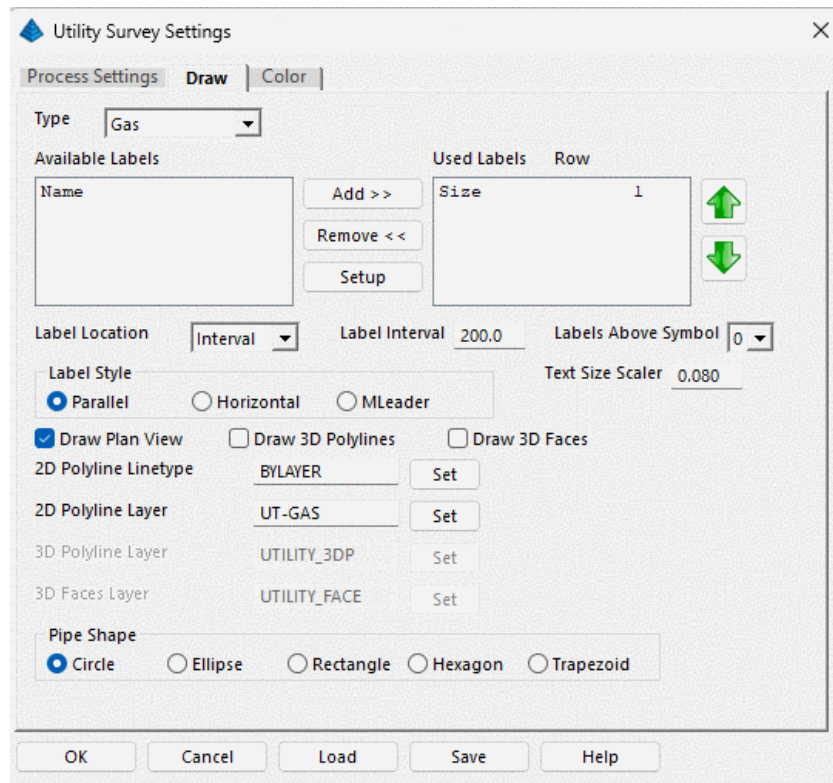


Elevation Measurement - sets the reference for the point location as either Top or Bottom of the pipe

Description Sequence - Sets the order of the utility size in the code description

GIS Attributes - using GIS attributes mimics the process described above. The difference is how the values are input into the Utility feature. Using the GIS attribute option requires that the data be entered as an attribute that is carried with the point coordinate file.

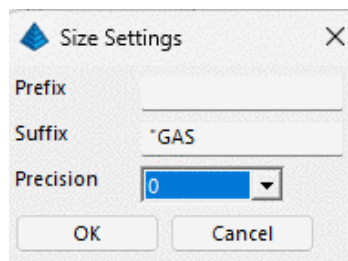
Draw Tab:



Type - selects the type of utility for the draw settings. There is a pull down menu with include utility types of:

- Gas
- Water
- Sanitary
- Storm
- Communication
- Electric
- Irrigation
- Fire
- Utility Duct
- and Other for user defined utilities

Setup - allows the additional control of adding prefix, suffix and the size precision to the utility line label. This can also be accomplished by double clicking the label field.



Available/Used Labels - to setup the Utility Line Labels, highlight any available field labels on the left side of the dialog box and either double click or hit the Add button to move it to the Used Labels portion of the dialog box on the right.

Label Location - sets the location of the label along the utility line

- None - does not add a label to the line
- Ends - adds a label to each end of the utility line
- Interval places a label at a distance interval
 - Label Interval is the distance in CAD units to place the label
- Both - places a label at each end as well as at intervals

Label Style - controls the placement of the text label for the size and/or name of the utility.

- Parallel - places the label parallel to the utility pipe
- Horizontal - places the label horizontal to the screen view, with respect to a Twist Screen if present
- Mleader - creates a Mleader that points to the utility line

Draw Options - are toggles to select options for the entities to be drawn. One or more may be selected. The options are:

- Draw Plan View - will draw the 2D line work as set in the code table with respect to the options herein
- Draw 3D Polylines -draws a 3D polyline between points with each vertex matching the point elevation
- Draw 3D Faces -creates 3D Faces in the shape and size of the pipe shape selected

2D Polyline Linetype - sets the linetype for the 2D polyline

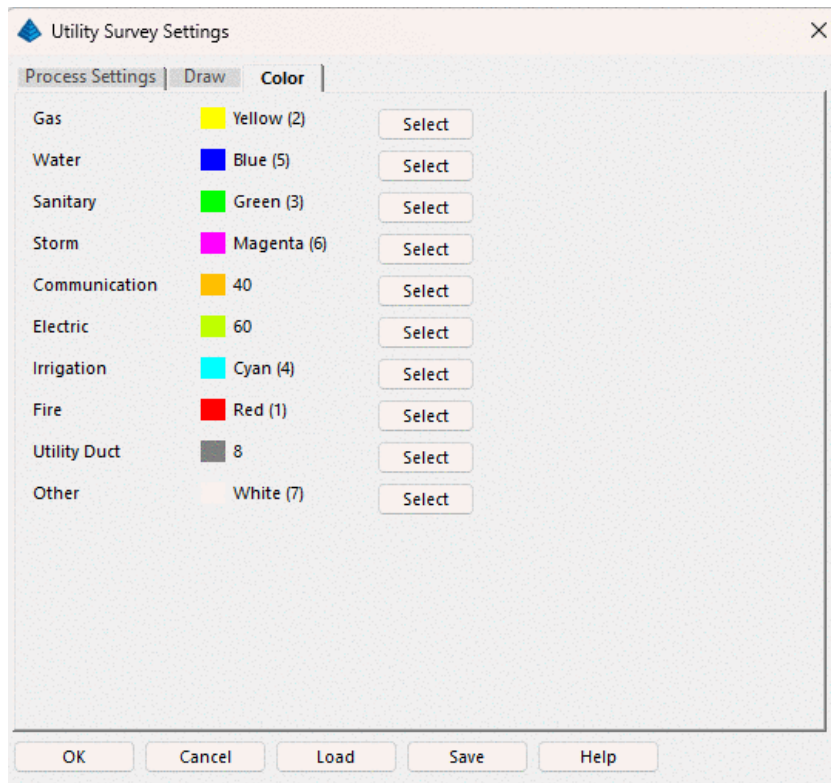
2D Polyline Layer - sets the layer for the 2D polyline to be placed

3D Polyline Layer - sets the layer for the 3D polyline to be placed

3D Faces Layer - sets the layer for the 3D faces to be placed

Pipe Shape - select the shape of the pipe to be created. These pipe shapes affect the 3D Faces created if that option is enabled.

Color Tab:



This dialog box contains the option to set individual colors for each utility type.

Symbol Library - accesses the Symbol Library also contained in the Settings Menu

Edit Points: - accesses the Edit Points Menu also contained in the Points Menu

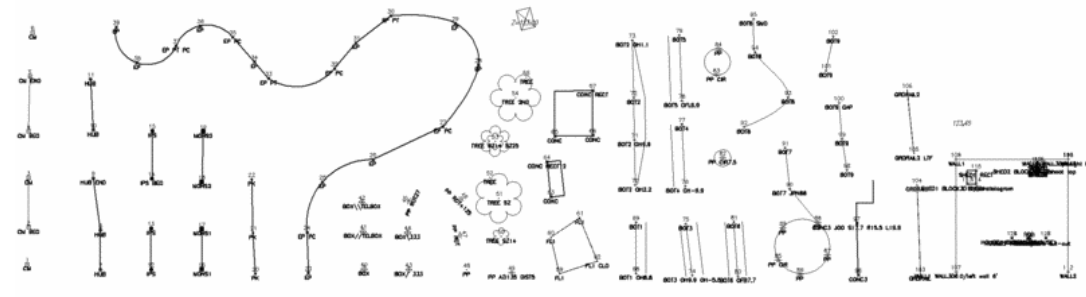
Report Codes/Points - Option to Report Codes and Points from the current Coordinate file and/or Field Code Table

Help - accesses the Help Menu

Exit - exits the Field Code Table Editor

Coding Examples

Under the Carlson Projects folder, there is an example that shows the different ways for linework coding along with examples for many of the special codes. The examples are in **f2f_example.crd** and **f2f_example.fld**. To see this example, create a new drawing, set to the supplied coordinate file and process using the supplied field code table.



Here is a breakdown of the features that the points above illustrate.

Point 1: Point Entity by itself

Points 2-3: Using Begin code to start a line; end line using Begin code for next line

Points 4-5: Using Begin and End to start and stop linework

Point 6: Point Entity by itself after End code

Points 7-11: Linework by code defined as Polyline entity type; using End as break between linework

Points 12-15: Linework by code defined as Polyline entity type; using Begin as break between linework

Points 16-19: Linework by code defined as Polyline entity type; using # after code instead of Begin/End to separate linework

Points 20-22: Linework by code defined as Polyline entity type without using Begin/End to start/stop linework

Points 24-26: 3 point curve using on PC code

Points 27-30: 3+ point curve using PC/Point codes

Points 32-33: 2 point tangent curve using PC/Point codes

Points 35-39: reverse curve using PC/Point codes

Point 40: Regular point without extra description

Point 41: Using // to use a code description as a suffix

Point 42: Using \ to use a code description as a prefix

Point 43: Using / to append a description

Point 44: Using \ to add a description as a prefix

Point 45: Using ROT and a Point# to rotate to that Point#

Point 46: Using ROT and a value to set the rotation

Point 47: Using ROT by itself to rotate to the next Point#

Point 48: Regular point without rotation

Point 49: Using AZI and DIST codes to offset the point

Point 50: Using SZ with value to set size of symbol

Points 51-52: Using SZ by itself to size symbol by the distance to the next point

Point 53: Using SZ with 2 values to draw multiple symbols at those sizes

Points 54-55: Using 2ND code to size the symbol

Points 56-58: Using 2ND and 3RD codes to size the symbol in 2 dimensions

Points 59-62: Using CLO to close the linework

Points 63-64: Using RECT with two points and a value to create a rectangle

Points 65-67: Using RECT with three points to create a rectangle

Points 68-69: Using OH to offset right a fixed amount

Points 70-73: Using OH on multiple points to offset various amounts

Points 74-75: Using multiple OH on the same point to offset polyline multiple times

Points 76-77: Using OH with negative value for offset to left

Points 78-79: Using OFL with value for offset left a fixed amount

Points 80-81: Using OFB with value to offset both left and right a fixed amount

Point 82: Using CIR to draw circle at specified radius

Points 83-84: Using CIR to draw circle using two points for center and perimeter

Points 85-89: Using CIR to draw best-fit circle through points on perimeter

Points 90-91: Using JPN to join linework to another Point#

Points 92-95: Using SMO to create smoothed linework

Points 96-97: Using JOG to create additional linework segment extensions

Points 98-102: Using GAP to create a break in the linework

Points 103-106: Using LFT to switch linetype to left side

Points 107-109: Using WALL3D with specified height value

Points 110-112: Using WALL3D with height from 2nd point

Points 113-115: Using BLOCK3D with height and three points to define parallelogram

Points 116-123: Using BLOCK3D with height and multiple points to define perimeter

Points 124-128: Using FACE3D with multiple points to make a surface

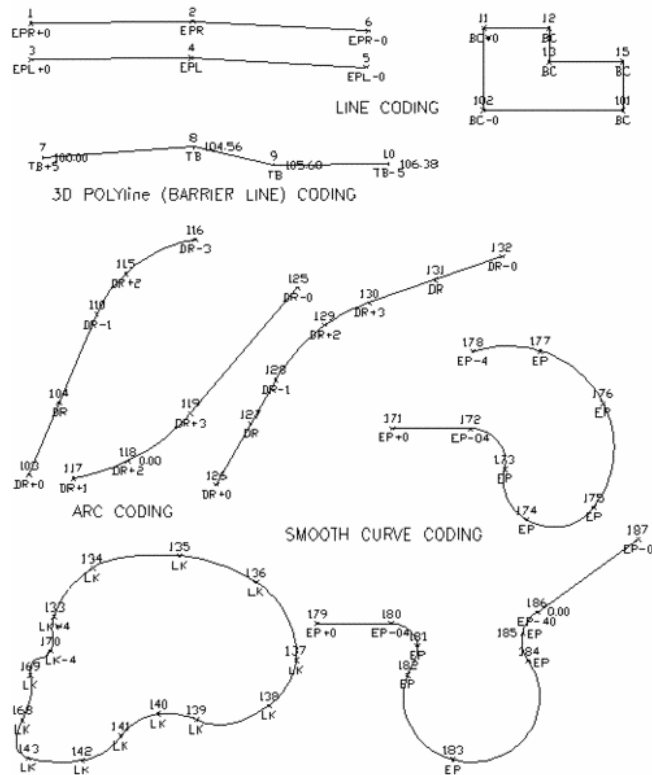
Points 129-132: Using HOLE3D with multiple points to define the perimeter of a hole in the FACE3D surface

Point 133: Using code definition with Attribute Format set to Text and only Elevation turned on with Label Decimal

PointCAD Coding

Field-to-Finish supports an early Carlson style of linework coding called PointCAD. The PointCAD codes use numbers with +,-,* symbols as follows:

- +0 Starts a regular 2D line (not a polyline) that is open.
- *0 Starts a regular 2D line that is closed.
- +4 Starts a curved 2D polyline that is open.
- *4 Starts a curved 2D polyline that is closed.
- +1 Begins a 3-point arc.
- 0 or -1 or -3 or -4 or -5 or -6 or -7 Ends a line.
- +5 Starts a 3D polyline that is open.
- *5 Starts a 3D polyline that is closed.
- +6 Starts a 2D polyline that is open.
- *6 Starts a 2D polyline that is closed.
- +7 starts line whose type (2D line, 2D polyline, 3D polyline) is specified by the point's field code definition. If the field code definition is to use points, then a 2D line is started.
- +2 Middle point of 3 point arc
- 05 starts a curved 3D polyline section.
- 50 ends a curved 3D polyline section.
- +8 starts a 2D and 3D polyline combination that is open.
- *8 starts a 2D and 3D polyline combination that is closed.
- 8 ends a 2D and 3D polyline combination.
- 08 starts a 2D and 3D polyline combination curve that is open.
- 80 reverts back to a straight 2D and 3D polyline combination.



PointCAD linework coding examples

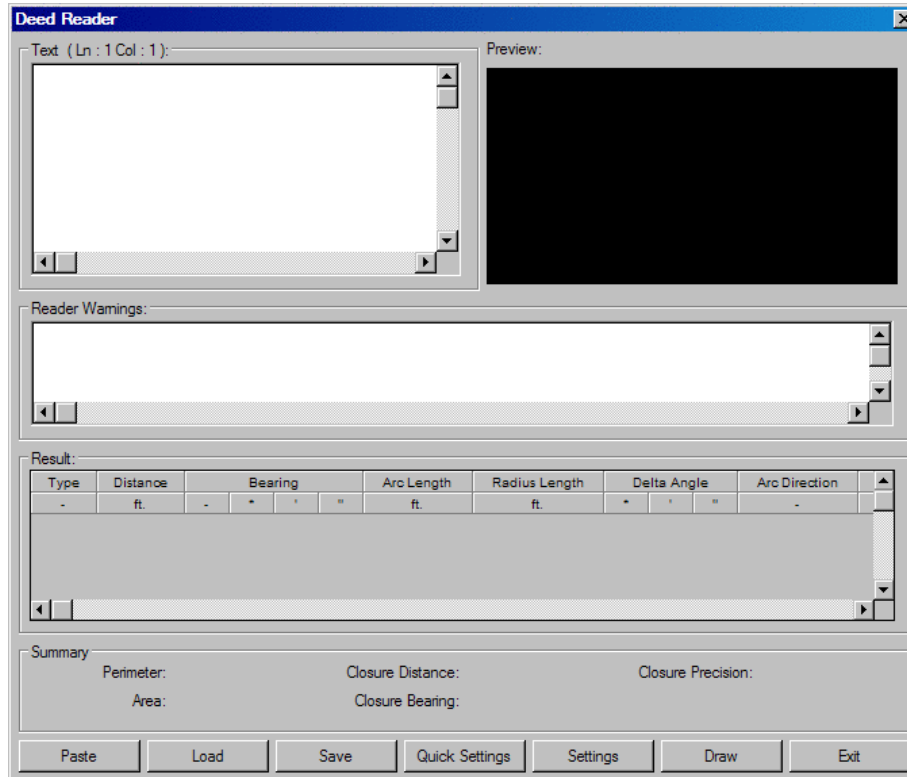
Menu Location: Survey

Keyboard Command: fld2fin

Prerequisite: A data file of points with descriptions

Deed Reader

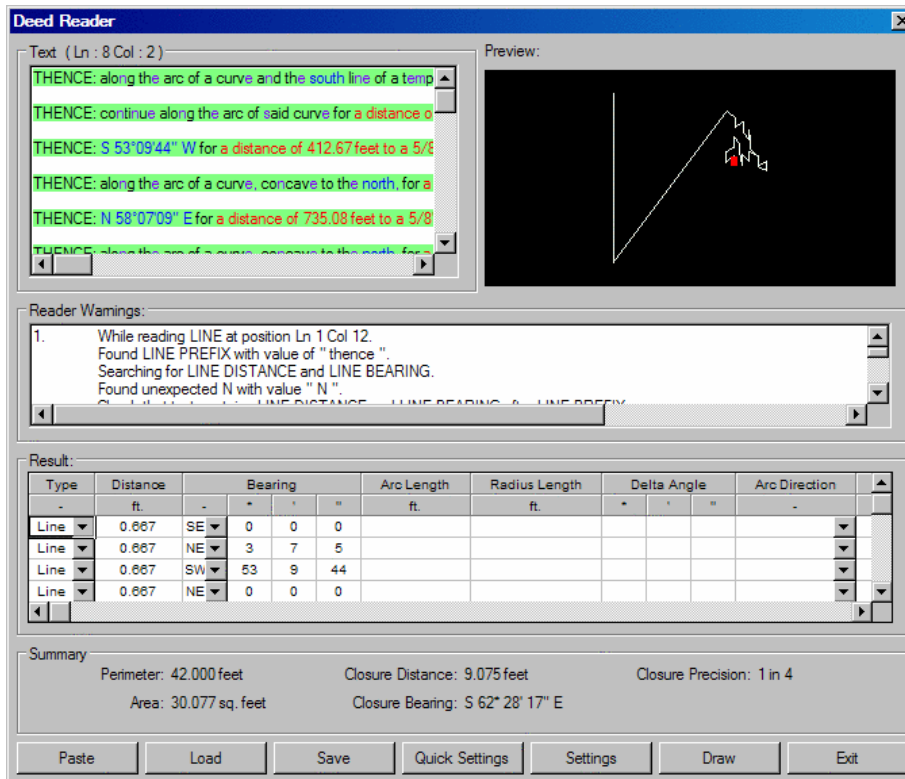
This command is used to extract deed line and curve data from the text of a deed. It shows the deed data in a spreadsheet and also graphically. The deed data can be saved to a deed file, drawn and reported. A blank Deed Reader dialog box appears as soon as the command is chosen.



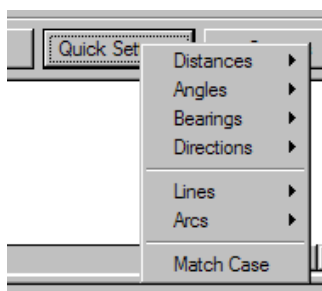
The **Text** section is for entering in ASCII/TXT data for the deed. This can be accomplished by using the Paste button at the bottom of the dialog, or loading a filing using the Load button. You can also type information directly into this screen. **Reader Warnings** indicates irregularities in the deed text. The **Result** section is below that. This section will give you a detailed, editable spreadsheet of the deed, which can be saved. At the very bottom of the dialog is a section called **Summary**. Here is where you will see the mathematical and closure data for this deed displayed.

Paste: This is for pasting in copied information.

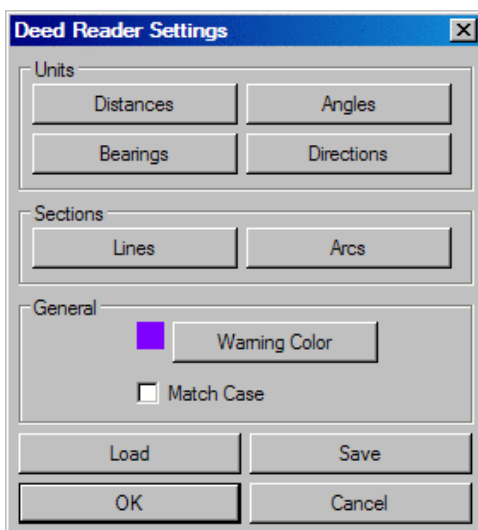
Load: This option will load an existing deed text (.TXT) file. Here is an example.



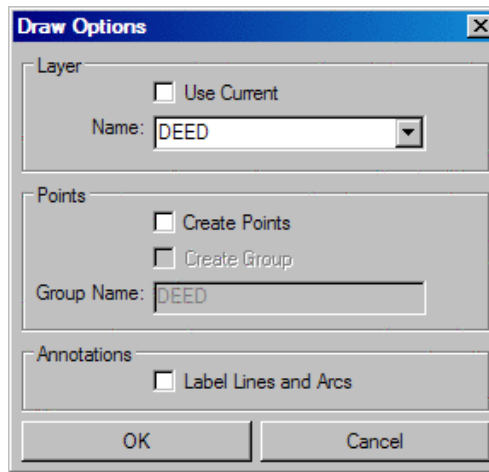
Quick Settings: This option allows you to set up, in a speedy fashion, the detailed criteria for this *Deed Reader* command.



Settings: A more formal settings feature, which is more methodical and dialog box driven.



Draw: This option will provides you choices as to how the date will be translated to the drawing screen.



It is in the Draw Options dialog that you can make decisions as to how detailed and involved your drawing will be. The Points section is key if you desire to have points created to a new coordinate file, or if you want to append an existing one. In the Annotations section, if Label Lines and Arcs is clicked on, the next dialog that you see, after choosing a point of origin, will be Auto-Annotate. Finally, click OK.

Prompts

Deed Reader dialog: *enter in or load the deed text*

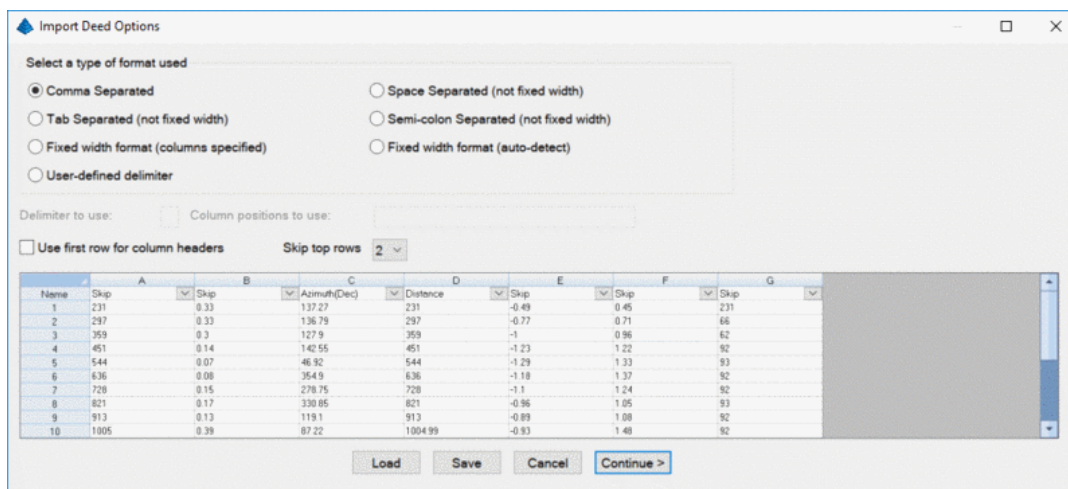
Pulldown Menu Location: Survey

Keyboard Command: read_legal

Prerequisite: Deed text

Import Angle/Distance File

This command creates a Deed File (PDD) using the angles and distances from a text file. The text file must have one set of angle and distance on each row with the values separated by a delimiter such as a comma. After selecting the data file to import and the deed file to output, the program has a dialog to define the format of the input file. Choose the file delimiter and select the columns for the Azimuth and Distance.



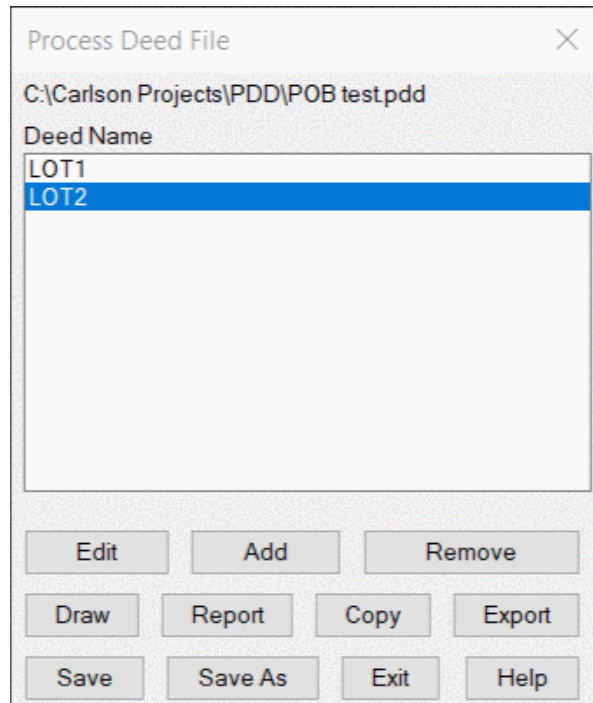
Pulldown Menu Location: Survey > Deed Tools

Keyboard Command: csv2pdd

Prerequisite: File to import

Process Deed File

This command contains several functions for deed (.PDD) files. A deed file consists of one or more deed descriptions. Each deed description includes a deed name, starting coordinate and line/curve data. This deed data can be created with the *Enter Deed Description* command. This command begins with the Process Deed File dialog.



Edit Opens the Edit Deed dialog where you can view or modify the deed name, starting coordinates, or line/curve data. Within this dialog the following commands are available.

Add Opens the Edit Deed dialog where you can add a new deed.

Remove Removes the currently highlighted deed.

Draw Draws the currently highlighted deed in the drawing and returns to the main dialog. The actual geometry will not appear in the drawing until you exit *Process Deed File*. There is an option to label the deed using the settings from the Annotate Defaults and Auto-Annotate commands.

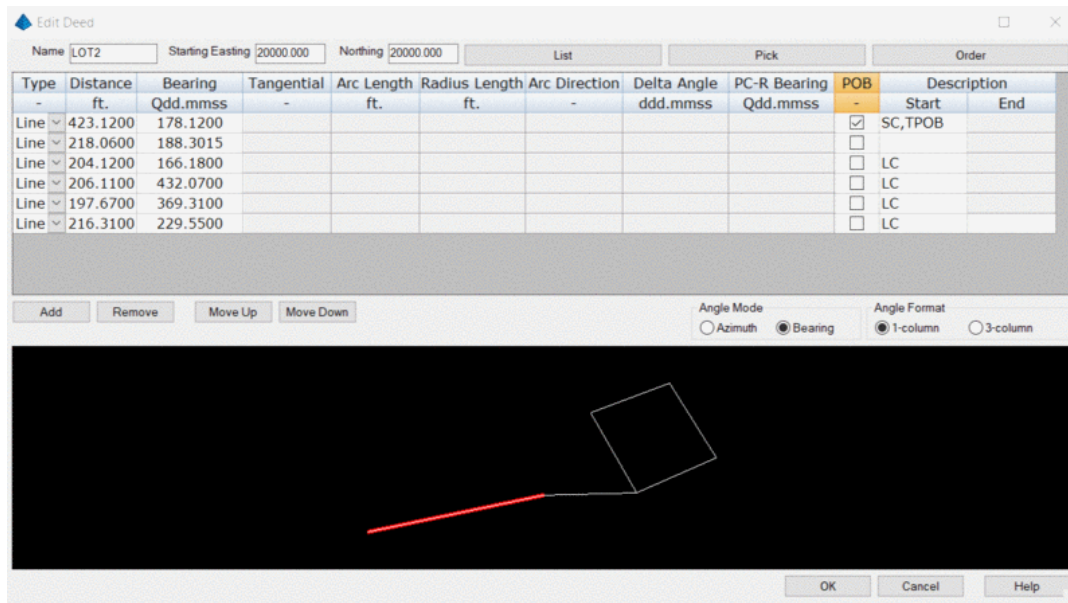
Report Generates a report for the currently highlighted deed. The report is displayed in the Standard Report Viewer unless Use Report Formatter is active which allows for customized reports and Excel output using the Report Formatter. For the Report Closure, the Start-End Coordinates option uses the difference between the POB (point of beginning) and the coordinates of the last call to calculate the closure error. The Angle-Distance Precision option determines the closure error by starting with the POB and calculating each point in the deed using the angle and distance values entered. Distance values are further affected by the precision in the report settings.

Copy Creates a new deed by copying the geometry of the highlighted deed.

Export Saves the selected deed data (.PDD) in raw file format (.RW5) that can be used with Edit-Process Raw File.

Save Saves the currently loaded deed (.PDD) file.

SaveAs Allows you to save the currently loaded deed (.PDD) file to another file name.



List Selects the starting coordinate from a point selection list from the current coordinate file.

Pick Allows you to screen pick the starting coordinate.

Order Allows you to set the sequence of the columns in the spreadsheet editor.

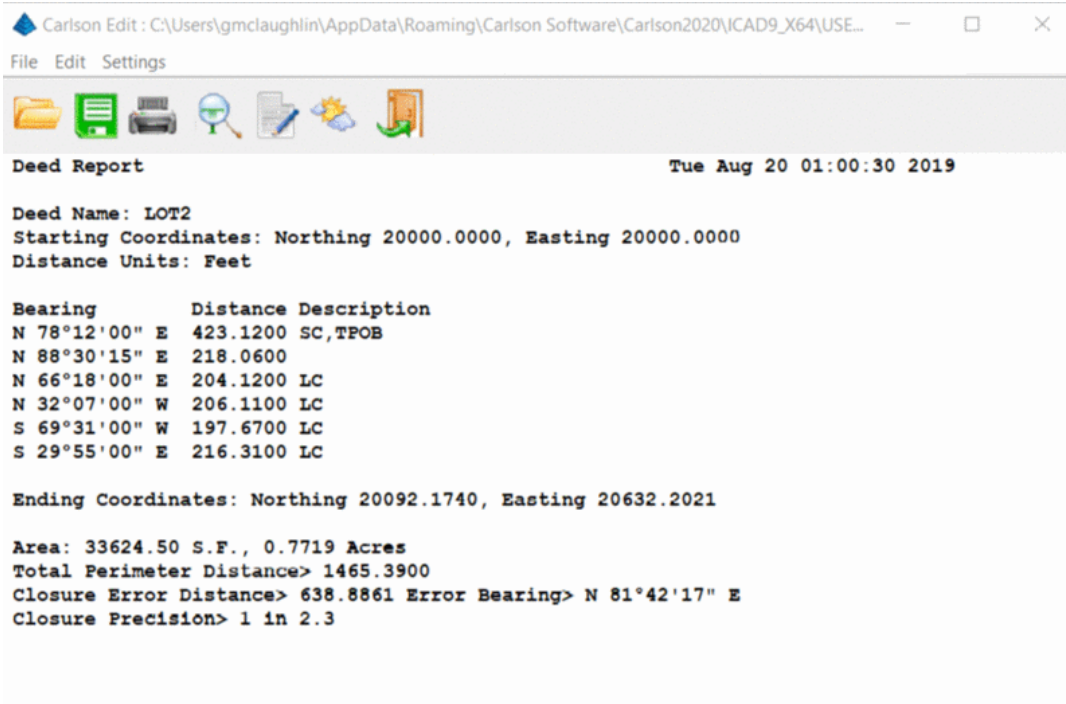
Add Allows you to add a new deed call (line or curve).

Remove Removes the highlighted deed call.

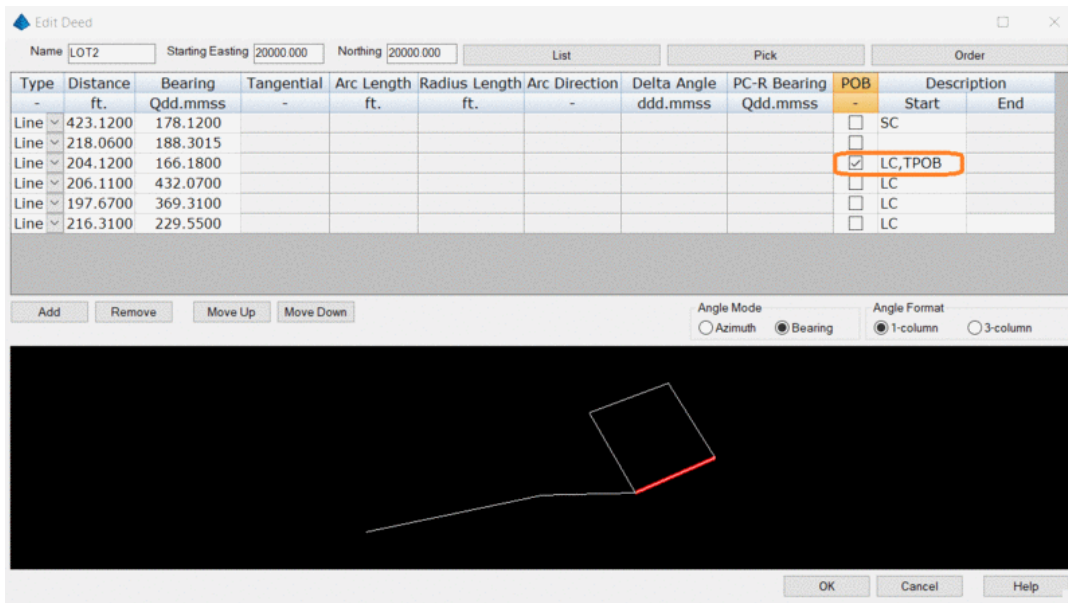
Move Up/Down Change the list order of the data records for the currently highlighted row.

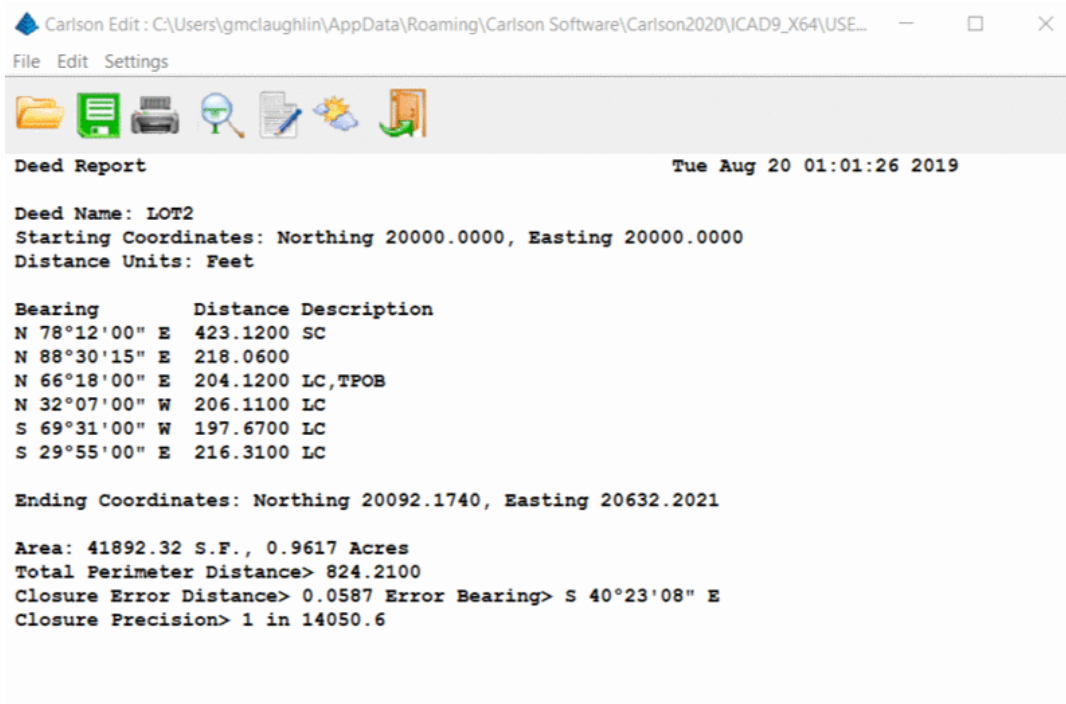
Angle Mode Choose between Azimuth or Bearing inputs.

Angle Format Choose between using one spreadsheet cell for the angle in dd.mmss format or using three cells with dd, mm and ss in separate cells for easier editing.



The POB location is variable on the deed. This means when entering or editing a deed, you may select any call in the deed to be the POB (Point of Beginning). The total perimeter, closure error and precision are determined from this point to the last call of the deed. Other calls leading up to the POB may be included in the deed as reference or commencement calls but not used in the closure calculations.





Pulldown Menu Location: Survey

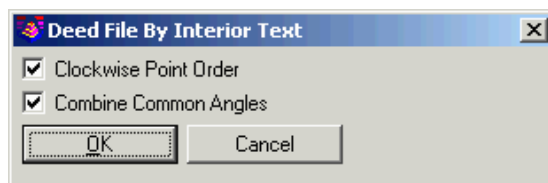
Keyboard Command: deed

Prerequisite: none

Deed File By Interior Text

This command stores deed data to a deed file (.pdf) using linework boundaries around selected text of the deed names. You can create a single deed or process multiple deeds at once. From the selected linework, the program finds all the closed areas and then looks in the closed areas for text to use as the deed name. The deed boundaries can be defined by any combo of lines, arcs and polylines.

In the options dialog, the Clockwise Point Order option chooses between creating the deed points in clockwise or counter. The Combine Common Angles option will join sequential boundary line segments together that have the same angle.



Prompts

Deed File to Process dialog: choose .pdd file to save deed data

Deed File By Interior Text options dialog

Select deed linework and text.

Select objects: *pick the deed labels and surrounding linework*

Pull-down Menu Location: Survey > Deed Tools

Keyboard Command: txt2pdd

Prerequisite: Deed labels surrounded by linework

Deed Linework ID

This command is used to report the deed name associated with selected linework. Since the Carlson deed commands that draw deeds attach the deed name to the linework, this command will extract that information and list it out. You can choose to select more than one deed linework entity before ending out of the command.

Prompts

Select deed linework to identify: *select deed linework*

Deed Name: Out Lot3 - Carlson Property

Select deed linework to identify (Enter to end): *select Enter*

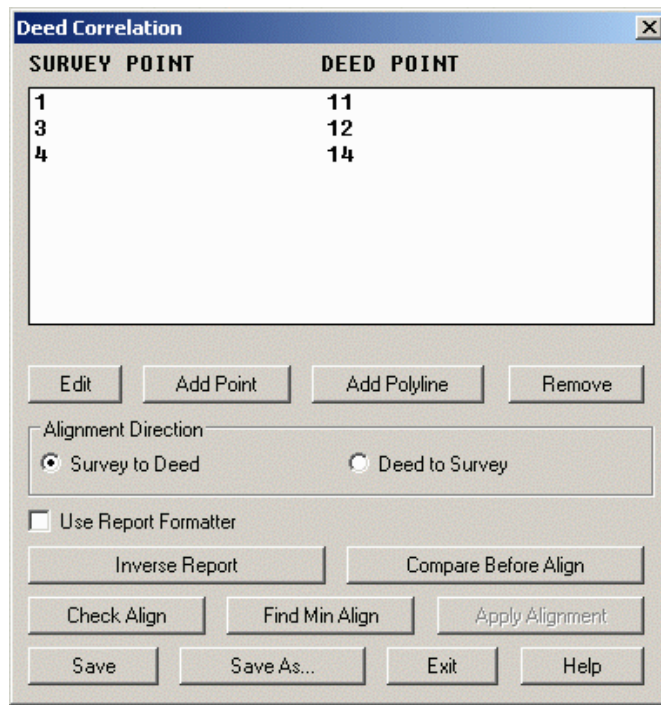
Pull-down Menu Location: Survey

Keyboard Command: deed_id

Prerequisite: A deed name assigned to the entity

Deed Correlation

This command takes a set of field and design/deed points and creates an inverse report, such as radial stakeout, for each pair of points. The Align functions combine a translation and rotation. The Alignment Direction settings controls whether to go from the survey points to the deed points, or vice versa. The command includes a routine to find the best point to hold and the best point to rotate to. This command provides tools for the correlation of surveyed points with that of deed input points. Different points can be specified as hold points, or rotation points, and provide a report showing the bearing and distance of all sides of the traverse/deed, based upon the hold and rotation points. This allows for the review of different scenarios based upon hold and rotation points. Perhaps two points in the field are in good shape, and seem to meet all the descriptions thereof. You decide to hold these two points as good, but you would like to see what holding these points will do to each side/call of the tract/description. This is what this routine is designed to do. In addition to allowing user specified trials of different hold and rotation points, the routine also provides a Find Minimum Rotation option that will report which points specified as the hold and rotation points will result in the minimum rotation of all sides of the tract/description. All points must be contained in the same coordinate file, and the points to be used in the correlation must be specified as either Survey points or Deed points. The Use Report Formatter option chooses between a fixed, automatic report, or a user-defined report format.



Edit: Edits the highlighted Survey and Deed point. Once selected the dialog above is displayed allowing for changes to be made.

Add Point: Click this button to specify the points as either Survey or Deed points. Then fill out the Edit Points dialog as desired or use the **Screen Pick** button to pick point entities in the drawing.

Add Polyline: This function prompts to select polylines for the survey and deed perimeters and uses the polyline vertices for the survey and deed points.

Remove: Removes the highlighted Survey and Deed points from the correlation setup. This does not delete the points from the coordinate file.

Inverse Report: This generates a report showing the inverse data from each point, both survey and deed, to every other point specified in the correlation set up. For example if there were four points in the survey points (1-4) then the report would show inverse data from 1 to 2, 3,4; from 2 to 1,3,4; from 3 to 1,2,4 and from 4 to 1,2,3. This would be the same for the corresponding deed points.

Compare Before Align: This option compares the survey information to the deed information.

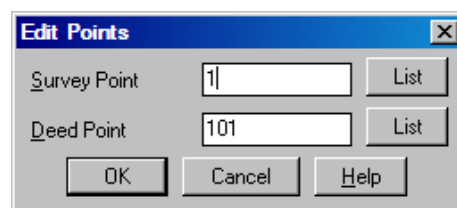
Check Align: This option that allows for user specified hold and rotation points, and then reports the inverse data of each side of the tract/description. The hold point and rotation point must be points from the specified survey point group.

Find Min Align: Determines the hold and rotation points that would result in the minimum rotation to each side of the tract/description. When selected the Minimum Deed Rotation Report is displayed.

Apply Alignment: This option can be issued after the Min Align criteria is set.

Save: Saves the point list to the current dcf file.

Save As: This option prompts for a user specified file name and allows for a user specified location to save the file. The file extension for the deed correlation file is dcf. When executing the program you have the option of using an existing file or creating a new file for the deed correlation.



After specifying the hold and rotation points, the deed correlation report will display again, showing the bearing and distance of each side of the tract/description.

Deed Correlation Report 2/23/2005 14:11

Survey Points				Deed Points			
To Point	Bearing	Distance	Desc	To Point	Bearing	Distance	Desc
7	N62°41'33"E	478.353	sur	2	N62°38'22"E	481.363	deed
8	N87°21'09"E	784.656	sur	3	N87°16'09"E	787.390	deed
9	S69°32'02"E	919.915	sur	4	S69°41'09"E	921.341	deed
10	S28°24'42"E	597.129	sur	5	S28°42'56"E	597.605	deed

Survey Points				Deed Points			
To Point	Bearing	Distance	Desc	To Point	Bearing	Distance	Desc
6	S62°41'33"W	478.353	sur	1	S62°38'22"W	481.363	deed
8	S62°56'56"E	402.846	sur	3	S62°53'55"E	403.259	deed
9	S38°54'44"E	695.407	sur	4	S38°53'41"E	695.218	deed
10	S10°42'59"W	757.876	sur	5	S10°40'01"W	758.444	deed

Survey Points				Deed Points			
To Point	Bearing	Distance	Desc	To Point	Bearing	Distance	Desc
6	S62°41'33"W	478.353	sur	1	S62°38'22"W	481.363	deed
8	S62°56'56"E	402.846	sur	3	S62°53'55"E	403.259	deed
9	S38°54'44"E	695.407	sur	4	S38°53'41"E	695.218	deed
10	S10°42'59"W	757.876	sur	5	S10°40'01"W	758.444	deed

Check Deed Rotation

Hold/Pivot Point:

Rotation Point:

OK Cancel Help

Check Deed Rotation Report 2/24/2005 14:34

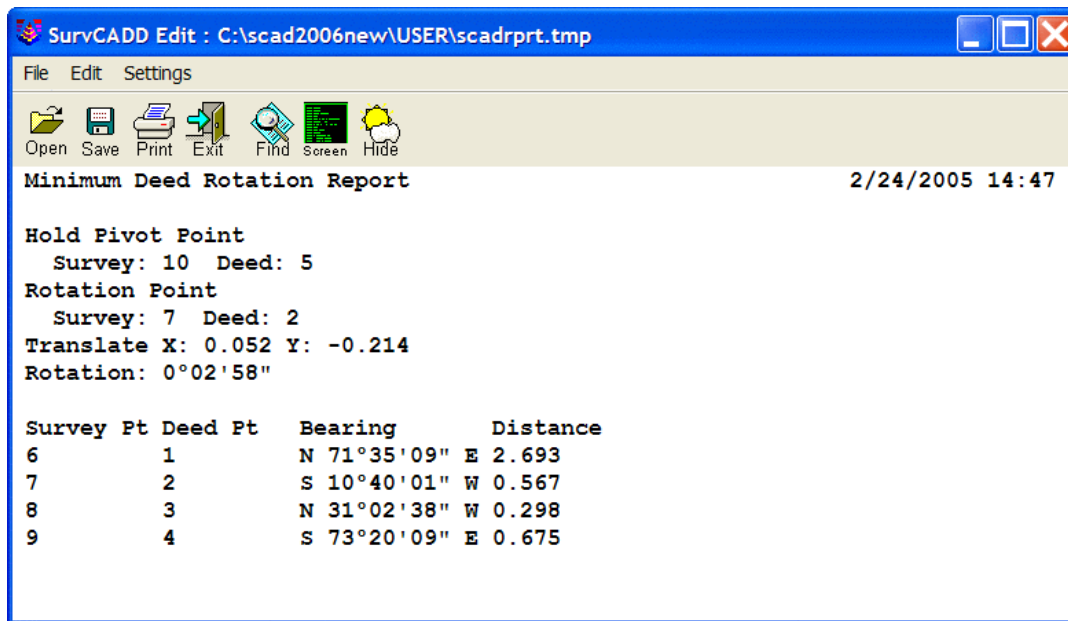
Hold Pivot Point
Survey: 6 Deed: 1

Rotation Point
Survey: 7 Deed: 2

Translate X: -2.956 Y: -1.310

Rotation: 0°03'10"

Survey Pt	Deed Pt	Bearing	Distance
7	2	S 62°38'22" W	3.009
8	3	S 78°33'32" W	2.766
9	4	S 62°16'06" W	2.134
10	5	S 71°43'06" W	2.658



Pulldown Menu Location: Survey

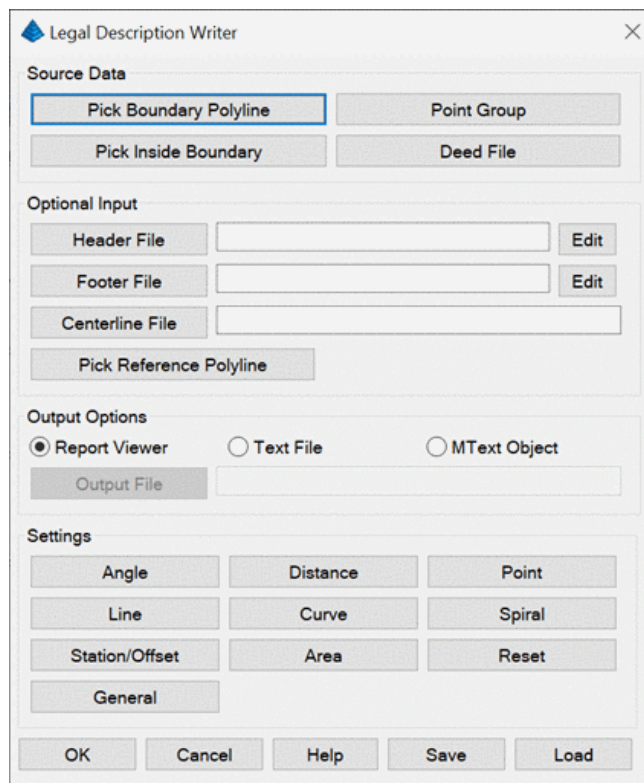
Keyboard Command: deed_align

Prerequisite: A coordinate file (.CRD)

Legal Description Writer

The *Legal Description Writer* gives you the ability to create a detailed legal description from a polyline. This description consists of calculated calls, point descriptions from Carlson points, and numerous user defined terms. The programs values for these terms are easily replaced, and are stored as defaults with each use. When a scale factor is specified under Drawing Setup, the Legal Description distances will apply the scale factor which is a way to report grid distances from ground drawing coordinates or vice versa.

In addition to this command, you can also generate legal descriptions by point numbers with the Report function within the Lot File Manager command.



Legal Description Writer Dialog

This initial and primary dialog box is shown above, and described below.

Pick Boundary Polyline: This button is used to designate the polyline boundary used. The boundary should be a closed polyline. Tools are provided in the Edit menu if you need to reverse the polyline or change its origin point. You can also select multiple polylines to process at the same time by entering M for Multiple at the Select Boundary Polyline prompt in this routine.

Pick Inside Boundary: This button is another way to designate the polyline boundary. With this method, the boundary can be defined by multiple linework entities. You pick inside the boundary area and the program will figure the boundary perimeter from the surrounding linework. This method uses the same technique as the Draw->Boundary Polyline command. The boundary perimeter that the program finds is highlighted for visual confirmation.

Deed File: This method selects the deed to report from a deed file (.pdd).

Pick Reference Lines: Used to select lines that tie into the polyline boundary used for the legal description. These should be LINE objects that have one endpoint exactly the same as the beginning point of the boundary polyline. If a Carlson point exists at the end of the line away from the boundary, the routine will pick up its description, otherwise you will be prompted for the description. You can choose any number of reference lines, simply press enter to conclude the selection of reference lines.

Point Group: This method defines the perimeter by a series of points from a group defined by the Point Group Manager command. In Point Group Manager, a group can be defined as a point list including the ability to have radius points.

Header File: This button and edit field are used to designate the optional header text file. If a valid file is selected

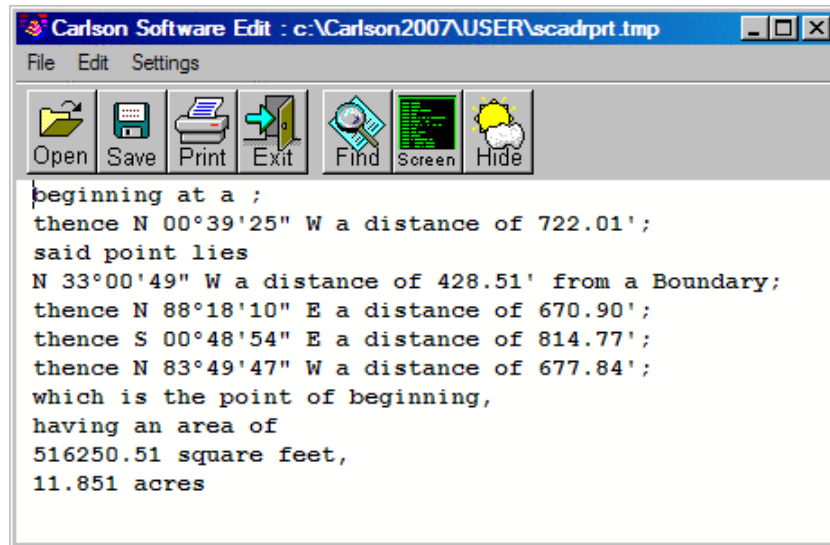
it will be written into the top of the output.

Footer File: This button and edit field are used to designate the optional footer text file. If a file is selected it will be written at the end of the output.

Centerline File: This allows you to set an optional reference centerline for reporting station/offset for each point in the boundary.

Output Options allows you to select where Legal Description Writer should send the output.

Report Viewer: The output is sent to the report viewer specified under Configure Carlson->General settings: Carlson Standard Report Viewer, Windows Notepad or Microsoft Word.



Text File: The output is sent to an external text file as designated in the output file section described below.

Mtext Object: This creates a mtext object in the current drawing. Upon choosing OK you will be prompted for a starting point (which is the upper left corner) and well as a second point that determines the width and angle. By default ortho is turned on for this second point. Press the F8 key to toggle its status.

Output File: This button and edit field are used to designate the necessary output text file. This file can then be brought into your word processor and finalized. Note that the appearance of the output file can be affected by the status of the 'Use Paragraph Format' toggle in the Legal Description's General settings.

Angle Specifications

This section is used to establish the appearance of the bearings that are output with the description, and allows detailed control over each aspect.

Angle Format: The Format chooses between Bearing, Azimuth, Decimal and Fraction. Designate the character or word used in each bearing direction. Standard values are the letters N, S, E, or W. One possible option is the entire words NORTH, SOUTH, EAST, and WEST. It is important to keep in mind that spaces are literal, meaning that if you don't have a literal space after N/S, and before E/W, a space will not be formatted into the bearing. To use Azimuth, place a check in the Use Azimuth box and the General Prefix will be set to AZ.

Cardinal Angles: For example bearings that are due NORTH, the default is to generate N 00_ 00' 00 E. If the Label Cardinal Angles toggle is turned on, the program will substitute the single word (which you can change) for the direction, these usually being NORTH, or DUE NORTH.

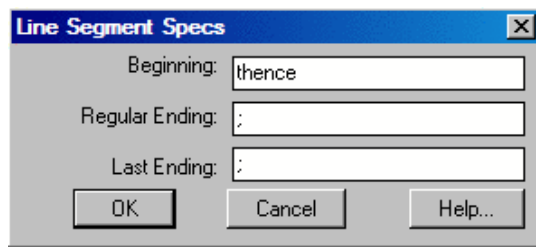
Interior/Exterior Angles: Controls whether to report the interior and exterior angles at corners.

Quad Directions: These names are used for the Concave Direction for curves and for the direction when using the Report Interior Angle option.

Symbols: This section allows you to designate the precision for bearings, as well as the symbols used. Turn on/off the toggles for degrees, minutes, and seconds to control the precision. For example, if you wish to round to the nearest minute, simply clear the toggle from the second field. For each field (degrees, minutes, seconds), you can supply the character or word to be used. You can quickly fill in these fields with the two buttons to the right.

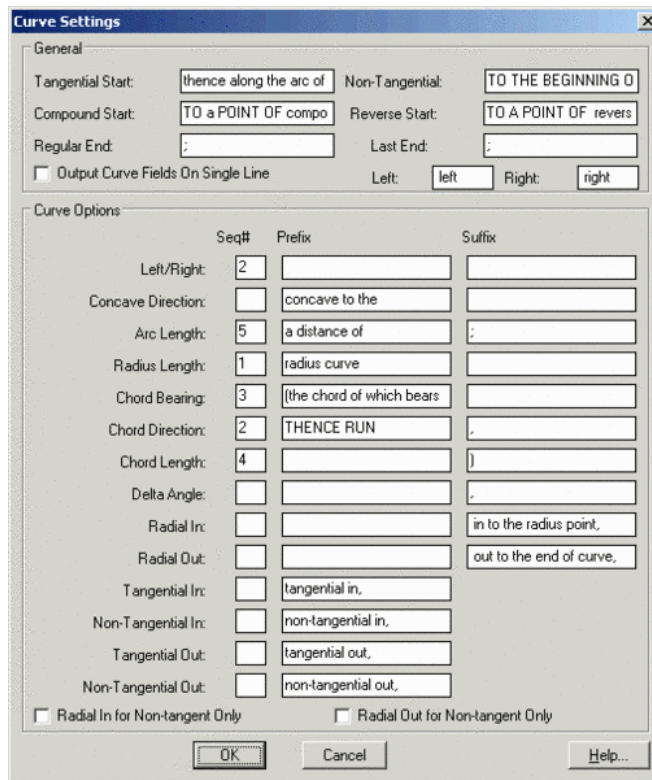
Line Segment Specifications

This section is used to establish the terms used when the course of a call is a line segment, as is often the case. Simply supply the beginning and ending terms for these line calls.



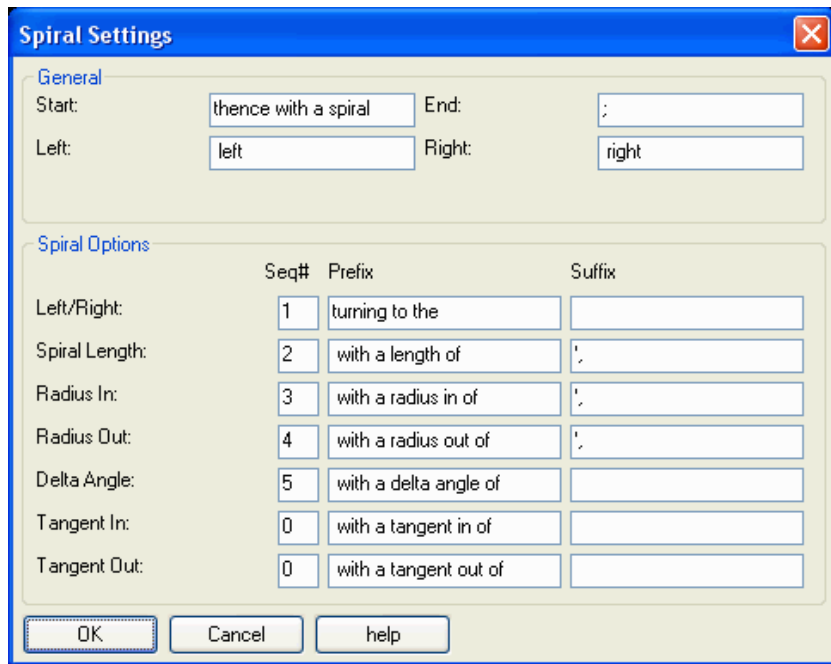
Curve Segment Specifications

This large dialog is used to establish the terms and options used when creating the course of a curve. Basic options include beginning and ending terms, as well as the words for left and right if chosen. In the large table of curve options, you can choose the items you wish to report, in the order you want them to appear. Simply place a number in the sequence field indicating the items you wish to report, making sure that there are no duplicate numbers. In the example below, the program would output the curve direction, arc length, radius length, chord bearing, and chord length, radius length, chord bearing, and chord length, and in that order. Each field can also have a unique prefix/suffix. There are four different possible phrases for the start of the curve description for whether the curve is tangential, non-tangential, compound or reverse. The Radial In/Out for Non-tangent Only option applies to the Radial In/Out fields and tell the program to only use these fields when the curve is non-tangent. Otherwise, these fields are always used when the Radial In/Out fields are in the sequence.



Spiral Specifications

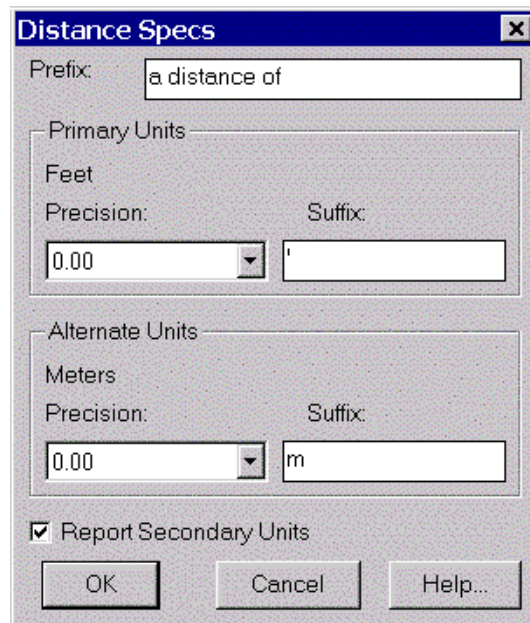
This dialog has the setting for reporting spiral portions of the boundary. In order to pick up the spiral, a centerline (.CL) containing the spiral must be drawn using the Draw Centerline File command. Then the program will pick up the spiral definition for any portion of the legal description boundary that follows the spiral on the centerline.



Distance Specifications

This dialog is used to establish the terms and precision used when creating a distance for the course of a call. The precision and suffix apply to curves as well. Simply choose the desired distance precision from the popdown, and supply the beginning and ending terms for the line calls.

Note the availability of dual distance reporting. If you would like to report dual distances such as feet/metric, turn on the toggle in the lower left corner of the dialog. Note that the primary units are the units set in the Settings menu, Drawing Setup. If you have English set as your units in Drawing Setup, then the alternate units will be metric. The opposite also applies. If your units in Drawing Setup is set to metric, then the alternate units will be English.



Point Specifications

In the process of following the polyline definition for a boundary, the legal description writer can look for descriptions of the points at the endpoints of the polyline. These can be extracted by setting the data source to the corresponding point from the coordinate (.CRD) file, meaning the points do not have to be plotted on the screen. A second option is point block, in which the program will read the information from the drawing, and not require the presence of a coordinate (.CRD) file.

Besides point descriptions, you can also report the point coordinates.

Point Settings

Descriptions

Description Source

CRD File Point Block None

Prefix: to a

Suffix:

Tolerance: 0.01000

Unknown:

Coordinates

Report Coordinates

North,East East,North None

Northing Prefix: at Northing

Northing Suffix:

Easting Prefix: and Easting

Easting Suffix:

Precision: 0.00

OK Cancel Help

Prefix: General term applied before the actual description.

Suffix: General term applied after the actual description.

Unknown: The text designated here will be placed in the description if the program does not find a valid description at that coordinate location. The words 'Unknown Point' may be used.

Tolerance: The point must be within this distance of the polyline vertex to use the description.

Report Coordinates: Choose whether to report coordinates and the order for the northing and easting.

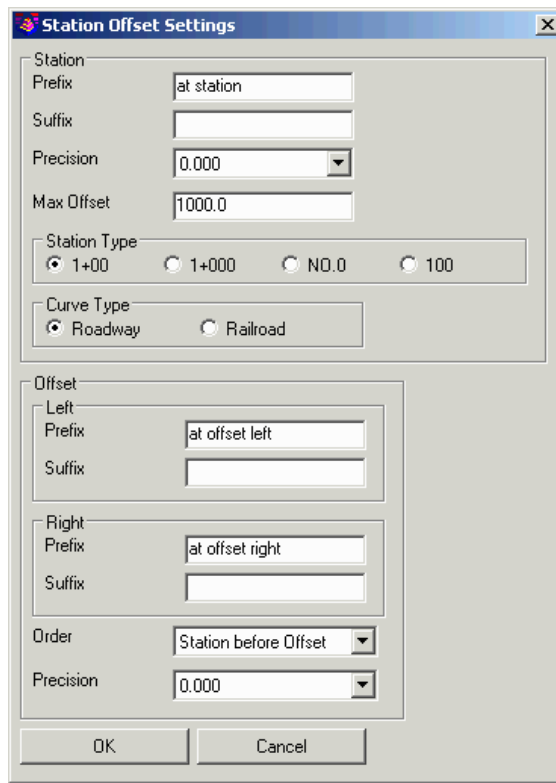
Northing Prefix/Suffix: Strings to report before and after each northing value.

Easting Prefix/Suffix: Strings to report before and after each easting value.

Precision: Number of decimals for the northing and easting values.

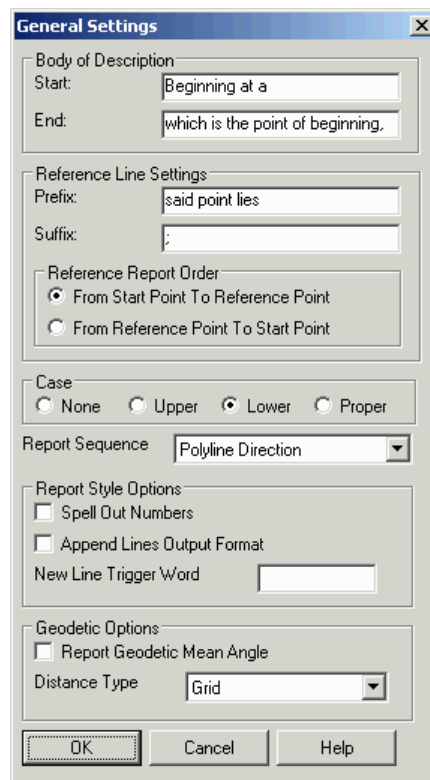
Station/Offset Specifications

When using a reference centerline file, the report will include the station and offset for each of the points in the boundary. The settings in this dialog control the format of these values. The program will skip reporting the station/offset for points with an offset more than the specified **Max Offset**.



General Specifications

This dialog controls general specifications which can affect the entire description. Each group of items are explained in detail below.



Body of Description: Enter the beginning and ending terms for the description.

String Case: Choose the button corresponding to the string case conversion desired. If you want no changes made, choose none. Choosing upper, lower, or proper case conversion will affect the case of all text throughout the description, except bearing letters.

Report Sequence: This option controls the sequence to report the boundary segments either in the direction of the polyline, clockwise or counter-clockwise.

Spell Out Numbers: This option writes numbers as words instead of digits. For example, a distance of 123 would be written as one hundred twenty three.

Append Lines Output Format: If this toggle is on, the program will output the description without carriage returns after each line. This approach makes a nice paragraph style when brought into a word processor with word wrap. If the toggle is cleared, the program will place carriage returns at the end of each call.

Geodetic Options: In order to use these options, the grid projection for the drawing must be set using the Settings > Drawing Setup command. The **Report Geodetic Mean Angle** option reports the geodetic mean angle which is the average of the geodetic bearings at the two point instead of reporting the direct coordinate bearing between two points. **Distance Type** controls whether to report grid distances or geodetic distances at zero or mean elevation.

Report Course Numbers adds a number that increments for each course in the legal description. There are settings for the Prefix and Suffix for this number.

Area Specifications

The legal description writer can output several types of areas. Basic options include beginning and ending terms. In the large table of area options, you can choose the items you wish to report, in the order you want them to appear. Simply place a number in the sequence field indicating the items you wish to report, making sure that there are no duplicate numbers. You can edit the prefix/suffix for each and control decimal precision of each field output. For geodetic areas, the grid projection needs to be defined in the Settings > Drawing Setup command.

	Seq#	Prefix	Suffix	Precision
Sq. Feet	1		Square Feet.	0.0
Sq. Yards	0		Square Yards	0.00
Sq. Miles	0		Square Miles	0.000
Acres	2		Acres	0.000
Sq. Meters	0		Square Meters	0.0
Sq. Kilo	0		Square KM	0.00
Cuerdas	0		Cuerdas	0.00
Hectares	0		Hectares	0.00
Geodetic Sq. Feet	0		Geodetic Sq. Feet	0.0
Geodetic Sq. Meters	0		Geodetic Sq. Mete	0.0
Geodetic Acres	0		Geodetic Sq. Acre	0.00

Reset: This option will reset all settings to their original default values.

Save: This option saves the legal description settings to a file. The file will be saved with an extension of (LGL).

Load: This option loads previously saved legal description (*.LGL) files.

Pulldown Menu Location: Survey

Keyboard Command: legal

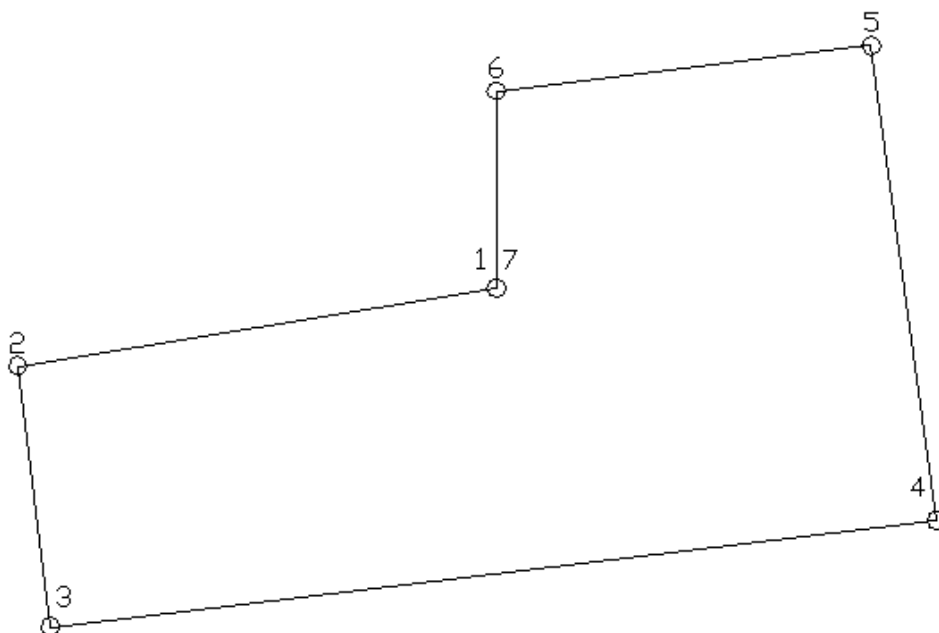
Prerequisite: Polyline boundary

Closure by Point Numbers

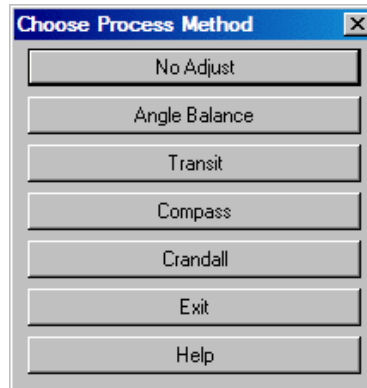
This command allows for traverse entry by point numbers, reports the closure and supports traverse adjustments. Using an existing coordinate file, the traverse is defined by a series of point numbers. The angle and distance for each traverse segment is calculated using the coordinates of the points. The traverse can be processed using all adjustment routines. Refer to the *Edit-Process Raw Data File* command for more detail on adjustment procedures. After selecting *Closure By Point Numbers* from the Survey menu, the Closure By Point Numbers dialog will appear.

Point#	North	East	Elevation	Desc
1	5000.000	5000.000	0.000	
2	4950.156	4692.914	2.716	XCUT
3	4783.049	4713.613	-1.716	XCUT
4	4851.222	5281.506	11.852	MNSET
5	5155.546	5239.733	8.928	MN-SET
6	5126.433	4999.524	7.655	MN-6
7	5000.056	4999.506	3.414	MN-1

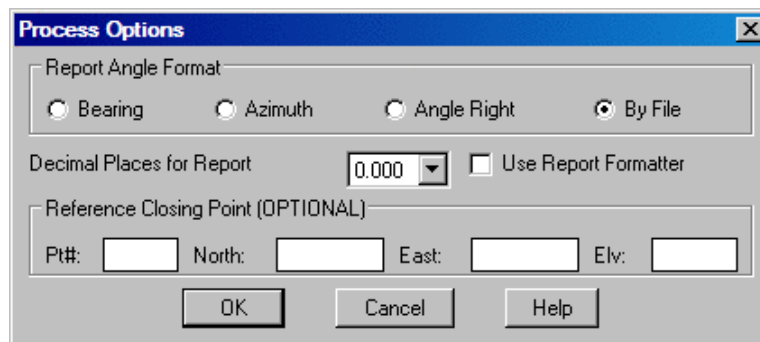
Point Number(s)



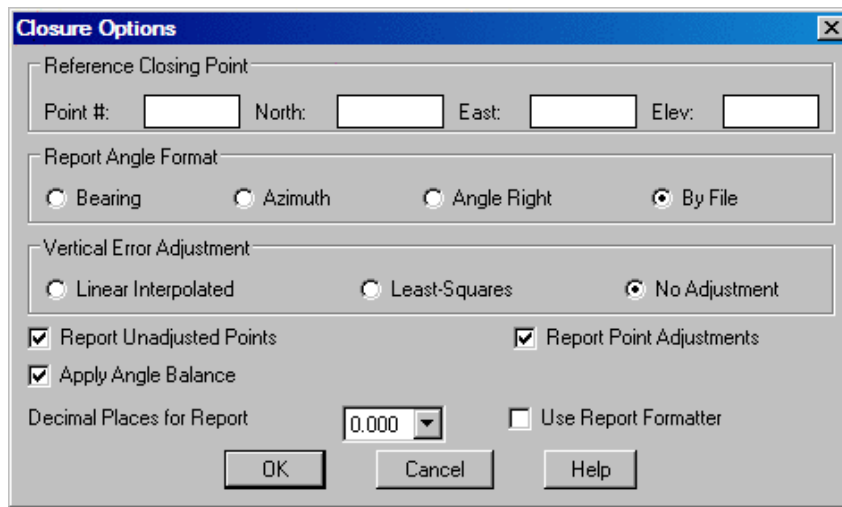
In this dialog shown above, add the point numbers that make up the traverse. This can be done by entering the point number, a range of points, or a point group into the Point Number(s) field. You can also choose points from a list by clicking the List button. Once each point, or group of points, is entered, click the Add button. Continue in this fashion until all of the point numbers are entered in. Clicking the Process button will display the Choose Process Method dialog. Choose the desired process method.



After selecting the process method for any of the adjustment methods, the dialogs and prompts will follow. They all start out with an "options" dialog box. These dialogs are titled either Process Options or Closure Options, depending on which process method you chose. The prompts that follow for any of the methods are subset of, and are very similar to, the prompting found in the *Edit-Process Raw Data File* command. After you have made your selections within these dialog boxes, click OK.

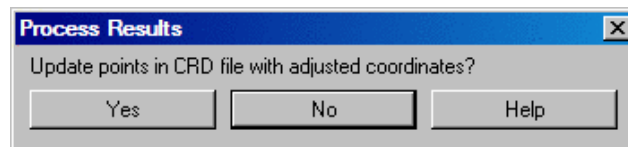


When you choose No Adjust of Angle Balance



When you choose Transit, Compass or Crandall

Each of the process methods will display a report that details the closure before the adjustment, and after the adjustment. Options to save and print this report are available. After a review of the report, pressing Exit will remove the report from the screen. At this point a Process Results dialog, prompting whether to Update points in CRD file with adjusted coordinates, will appear. If you choose Yes, the active coordinate (.CRD) file will be updated with the adjusted coordinates. Choosing No will leave the active coordinate (.CRD) file in its existing state, with the coordinates unchanged. It is important to remember that the starting and ending point in this routine must be a different point number. For example, if the traverse starts at point 1 and ends at point 1, then another point number should be used for the tie in shot to point 1. This logic is different in *Edit-Process Raw Data File*, where the starting and ending point can be the same point number.



Pulldown Menu Location: Survey

Keyboard Command: ptrawedit

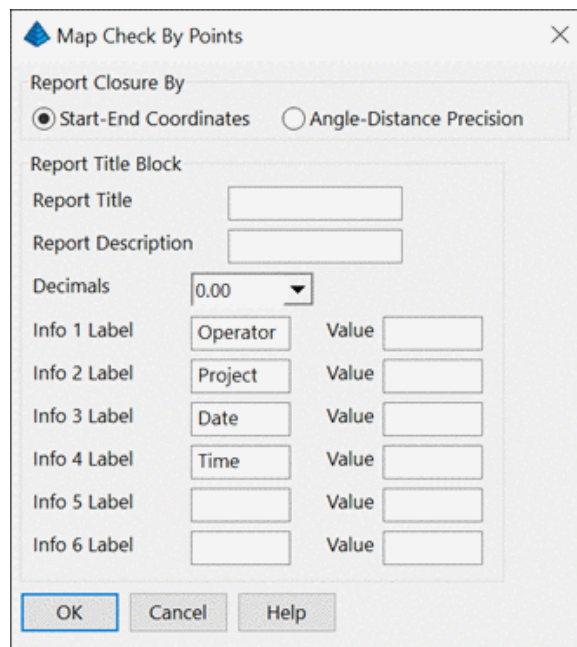
Prerequisite: Coordinate (.CRD) file

Map Check by Pnts

This command allows you to check the closure of a figure and produce a report. The points used for the map check should already be stored in a coordinate (.CRD) file, by using commands such as *Traverse*, *Locate by Bearing*, *Curves* menu, *Locate by Angle* – or perhaps a file from an electronic data collector.

In the dialog, you can set the labels for the report and the closure method.

Report Closure By: For the Start/End Coordinates method, the closure error distance is the distance between the start and end points which is 0 (perfect closure) if starting and ending at the same point number. For the Angle/Distance Precision method, the actual bearings and distances in the report are used, and the rounding used in the report will result in small closure errors.



Prompts

Beginning Point Number: 903

PointNo. Northing(Y) Easting(X) Elev(Z) Description

903 4940.73 2490.40 0.00

eXit/Curve/<point number>: 904

PointNo. Northing(Y) Easting(X) Elev(Z) Description

904 4850.89 2388.01 0.00

BEARING> S 48d43'58" W Hz DIST> 136.21

eXit/Curve/<point number>: 905

PointNo. Northing(Y) Easting(X) Elev(Z) Description

905 4699.39 2423.32 0.00

BEARING> S 13d07'04" E Hz DIST> 155.56

eXit/Curve/<point number>: 906

PointNo. Northing(Y) Easting(X) Elev(Z) Description

906 4653.59 2582.19 0.00

BEARING> S 73d55'04" E Hz DIST> 165.34

eXit/Curve/<point number>: 910

PointNo. Northing(Y) Easting(X) Elev(Z) Description

910 4941.88 2492.50 0.00

BEARING> N 17d16'54" W Hz DIST> 301.93

eXit/Curve/<point number>: X

Closure error distance> 2.39476609 Error Bearing> N 61°10'45" E

Closure Precision> 1 in 316.96 Total Distance Traversed> 759.04

SQ. METERS: 30403.0 SQ. KILOMETERS: 0.03

HECTARES: 3.04 CUERDAS: 7.74 PERIMETER: 759.04

Pick area label centering point: *pick point on screen for label text*

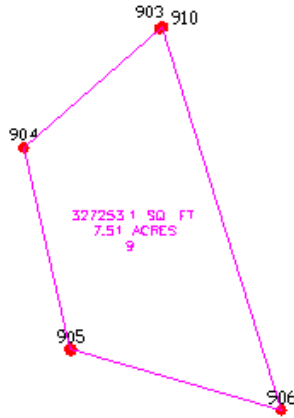
Erase Polyline Yes/No <Yes>: N

Typical Map Check Report:

```

Map Check                04/19/2006 13:14
Description
COURSE   BEARING        DISTANCE  PT#  NORTHING  EASTING  DESCRIPTION
903-904  S 48°43'58'' W  136.21   904  4850.89   2388.01
904-905  S 13°07'04'' E  155.56   905  4699.39   2423.32
905-906  S 73°55'04'' E  165.34   906  4653.59   2582.19
906-910  N 17°16'54'' W  301.93   910  4941.88   2492.50
Closure error distance> 2.39476609 Error Bearing> N 61°10'45'' E
Closure Precision> 1 in 316.96 Total Distance Traversed> 759.04
327253.1 SQ. FT.
7.51 ACRES

```

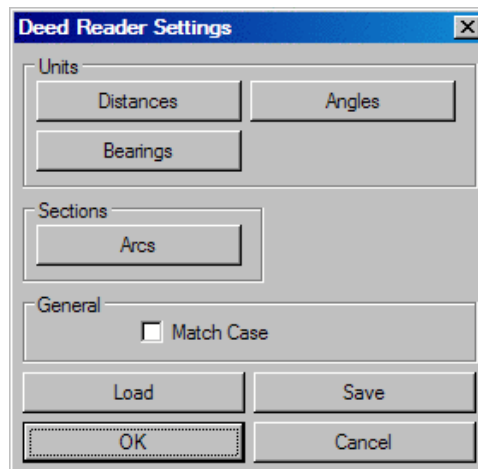
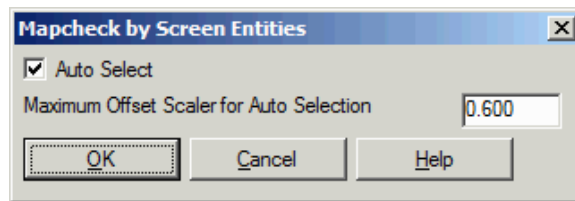
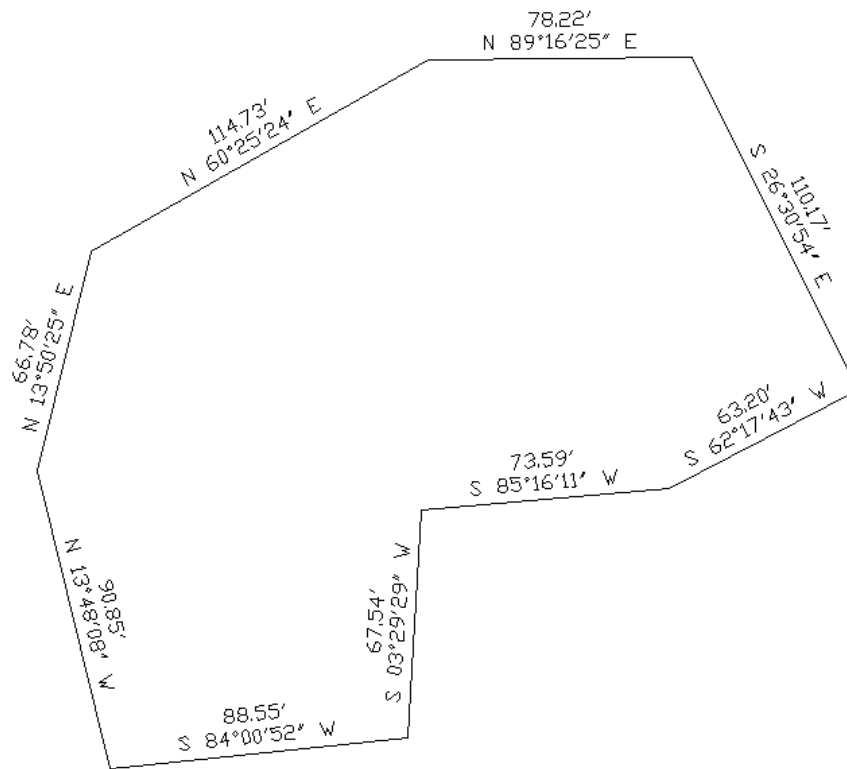


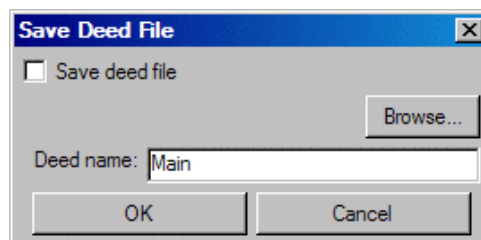
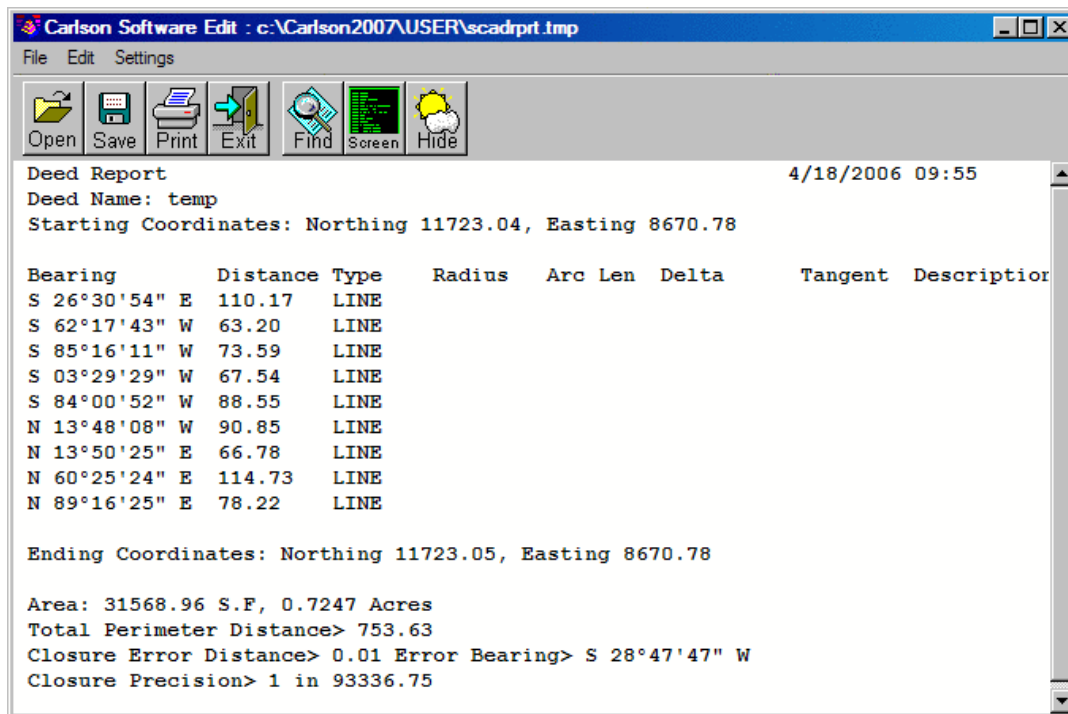
Pulldown Menu Location: Survey
Keyboard Command: mc
Prerequisite: Current coordinate (.CRD) file

Mapcheck by Screen Entities

This command allows you to check the closure of a figure, and produce a report from the Distance and Bearing labels in the drawing. The command works by prompting for a polyline and a sample of the text labels. Then the program looks for text on the sample layer and matches the text labels to the polyline segments. The text to process can be selected manually or automatically using an offset factor from the polyline. In addition to labels along the polyline, line or curve tables can also be used when the table are created using the Table Entity method in Table Defaults. The *Deed Reader* command is used here also, for extracting the deed line and curve data from the text of the deed. This polyline can be an open polyline in which case it's like doing an open traverse. The closure is based on the length of the polyline compared to the difference between the ending coordinate of the polyline and the ending coordinate from the label angles and distances. The deed data can then optionally be saved to deed file.

The report includes the line and curve data for every segment along with the starting and ending coordinates, the area and the closure. The format for the area values is set by the Area Defaults command.





Prompts

Select linework to process: *select polyline for the figure*

Select sample text: *select a label*

Pulldown Menu Location: Survey

Keyboard Command: textdeed

Prerequisite: Distance and Bearing labels

Cut Sheet

This command creates a report of the horizontal distance and elevation difference between points and a design. The design elevation can be defined by a grid file, triangulation file, 3D polyline, section file, note file, road template file, runway airway clearance or design points. The station and offset of the points can also be reported when a centerline is applied.

The data for the cut sheet is shown in a spreadsheet. You can edit or enter data in all the fields except for the Cut/Fill and Hz Error fields which are calculated. The cut sheet data can be saved and loaded with a .CUT file. The functions for processing the data are in the pull-down menus. Here's an outline of a typical workflow:

1. Import the survey data using Import > Points, or Import > SurvCE.
2. Assign the target design elevation using a method from the Grade menu.
3. If station-offset are needed, use a method from the Centerline menu.
4. Run Report > Create Report.

5. Run File > SaveAs to save the cut sheet data.

Point#	Description	Survey Z	Design Z	Cut/Fill	Station	Offset	Comment	Ht Error	Survey Northing	Survey Easting	Design Northing	Design Easting	Design PII
1	201	3.0230	3.0270	C0.004				0.005	500.0030	1000.0030	499.9970	1000.0010	229
2	202	2.9280	2.9290	C0.001				0.007	507.1260	1000.1610	507.1200	1000.1580	230
3	203	2.8730	2.8750	C0.002				0.008	515.6990	1000.3100	515.6940	1000.3040	231
4	204	2.8280	2.8390	C0.011				0.006	524.2220	1000.4950	524.2160	1000.4830	232
5	205	2.8380	2.8400	C0.002				0.008	532.6920	1000.6110	532.6850	1000.6070	233
6	206	2.8430	2.8430	F0.000				0.004	541.1150	1000.9370	541.1120	1000.9340	234
7	207	2.8470	2.8470	F0.000				0.003	548.8540	1001.7280	548.8520	1001.7260	235
8	208	2.8570	2.8560	F0.001				0.008	556.5090	1001.4540	556.5050	1001.4470	236
9	209	2.8760	2.8720	F0.004				0.005	565.0350	1001.0860	565.0300	1001.0860	237
10	210	2.8270	2.8200	F0.007				0.004	574.1200	1000.4850	574.1160	1000.4840	238
11	211	2.8310	2.8210	F0.010				0.024	581.7900	998.8380	581.7810	998.8620	239
12	212	2.8950	2.8880	F0.007				0.009	590.2710	998.6650	590.2640	998.6710	240
13	214	2.8910	2.8890	F0.002				0.005	605.8310	1000.5520	605.8320	1000.5570	241
14	215	2.8970	2.8860	F0.011				0.010	612.8220	1001.1010	612.8240	1001.1110	242
15	216	2.8700	2.8650	F0.005				0.006	618.3560	1000.0230	618.3580	1000.0290	243
16	217	2.8960	2.8890	F0.007				0.008	624.0820	998.0750	624.0760	998.0800	244
17	218	2.8970	2.8950	F0.007				0.004	628.4190	990.4730	628.4150	990.4740	245
18	219	2.8950	2.8920	F0.006				0.001	634.5950	986.8270	634.5950	986.8280	246
19	220	2.8390	2.8330	F0.006				0.004	641.8180	986.8070	641.8200	986.8100	247
20	221	2.8640	2.8590	F0.005				0.003	648.6800	988.3530	648.6820	988.3510	248

File > Open: Loads data from a .CUT file.

File > New: Clears the spreadsheet.

File > Save: Saves the spreadsheet data and report format settings to the current .CUT file.

File > SaveAs: Prompts for a .CUT file and saves the data.

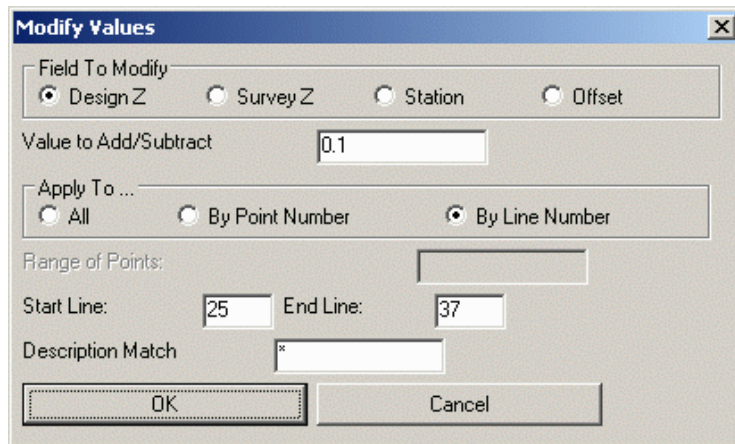
File > Exit: Quits the program.

Edit > Undo: To under the last edit in the spreadsheet.

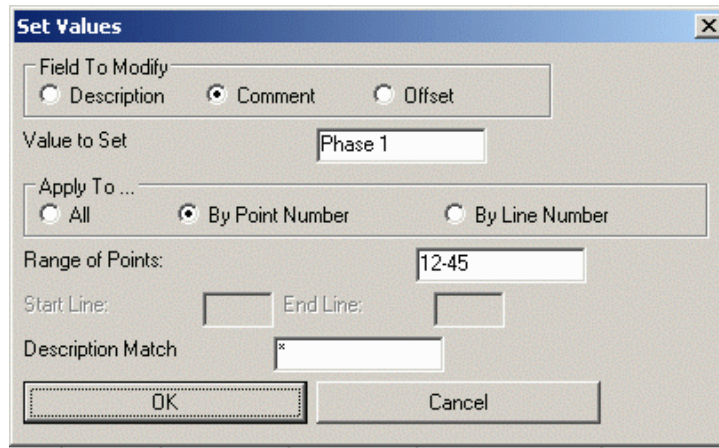
Edit > Delete Row: Deletes the currently highlighted spreadsheet row. You can also use the Delete key to delete the current row.

Edit > Insert Row: Inserts a new row above the current row. You can also use the down arrow key from the last row to add rows to the bottom of the spreadsheet, and use the Insert key to add a row above the current row.

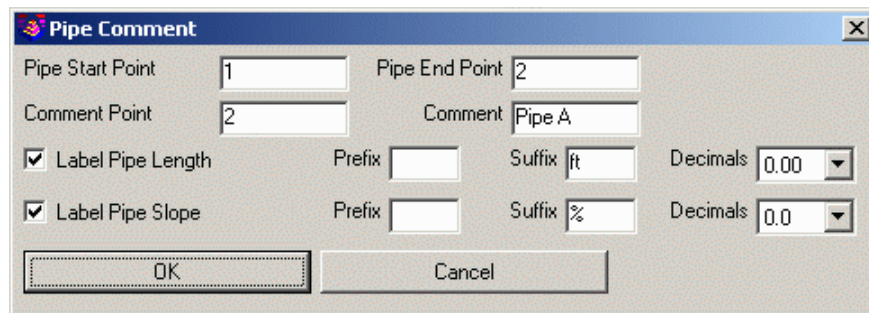
Edit > Modify Values: Changes values in the spreadsheet for the elevation, station or offset. In the dialog, set the amount to add or subtract (use negative value to subtract) and set which field to modify. There are controls for whether to update all the values or by point number range or by line number range. The Description Match is a way to filter which values to update.



Edit > Set Values: Sets values in the spreadsheet for the description, comment or offset field. In the dialog, choose the field to set, the value and which rows to set.



Edit > Set Pipe Command: This function fills in a Comment record of length and slope (ie "81.00ft - 15" RCP @ 2.4%") using the distance and slope between two points. The two points to use and the formatting options are set in this dialog.



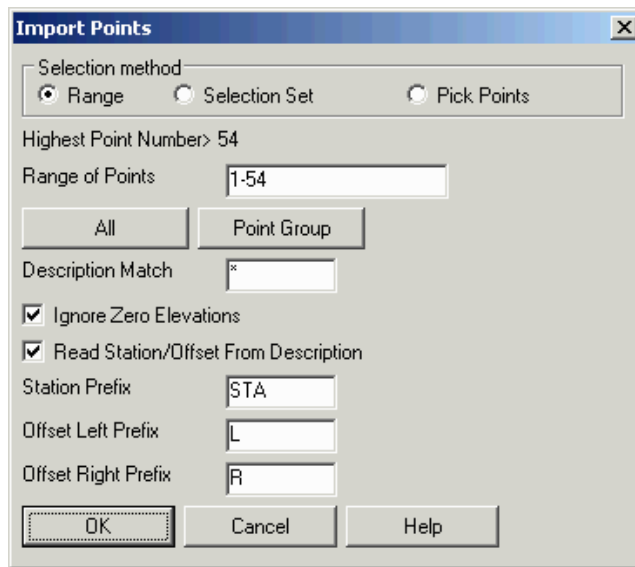
Edit > Cut: Blanks out the data for the currently highlighted cells and puts this data into the Windows clipboard.

Edit > Copy: Copies the data for the currently highlighted cells into the Windows clipboard.

Edit > Paste: Puts data from the Windows clipboard into the spreadsheet starting at the currently highlighted cell.

Import > Coordinate File: Imports survey data from a coordinate file for the Point#, Northing, Easting, Survey Z and Description fields of the spreadsheet. This function first prompts for the coordinate file to import. Then there is a dialog to choose whether to select the points by point number range, by selecting point entities from the drawing, or by screen picking points. The Description Match and Ignore Zero Elevations are options for filtering out points.

Besides reading the coordinate data, this import function can also read station/offset data from the point descriptions by looking for values after the prefixes defined in this dialog.



Import > Point Text File: This method is similar to Import by Coordinate File except a text file is used instead of a CRD file. The text file should have one point per row with fields for the northing, easting, elevation, description and point number.

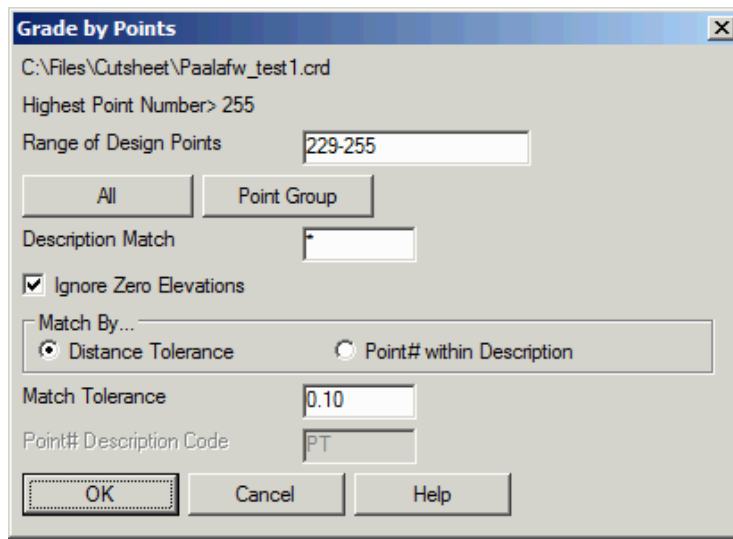
Import > Note File: This method reads the survey data along with the grade elevation from the note (.NOT) file that is associated with a coordinate file. For example, if the coordinate (.CRD) file is job3.crd then the note file name is job3.not. In Carlson Software's data collection programs (SurvCE and Field), there is an option to store stakeout data to the note file under the Stakeout options. When storing a point in the stakeout routines (using SurvCE or Field), the target point number, coordinates and elevation can be stored to the note file. This results with the as-staked coordinate stored in the coordinate (.CRD) file and the target coordinate stored in the associated note file. The Cut Sheet report can display this stakeout data using the Stakeout Point Comparison report option. The horizontal difference between the staked point and the target point can be reported in Bearing-Distance, Delta X-Y or North-South-East-West format. Also, in SurvCE and Field, the elevation difference routines can record the design grade elevation and station-offset to the note file when a point is stored to the coordinate (.CRD) file. This grade data can be reported using the Grade Elevation Report option. The note file records that the Cut Sheet report uses are TARGET_X, TARGET_Y, TARGET_Z, TARGET_DESC, TARGET_PT, STATION, OFFSET, VOFF1 and VOFF2.

Import > RW5 File: This method imports cut sheet data from a RW5 file of measurement data created by SurvCE. The stakeout functions in SurvCE store all the data needed to fill out the whole cut sheet including the survey data, design data and station-offset.

Import > SurvCE Cut Sheet: Imports data from a SurvCE Cut Sheet file (.CSV or .TXT). The setup for these cut sheet files in SurvCE is under File > Job Settings > Stake > Cut Sheets.

Import > TDS Raw: Imports data from a TDS RAW or RW5 file by reading the TDS stakeout records.

Grade > Points: The reference points to compare can be in the same coordinate file or a separate file. The reference/design points need to be matched with the survey points. The Match By Distance Tolerance method matches the design point that is closest to the survey point and within the specified Match Tolerance. The Point# Within Description method looks for the specified Point# Description Code in the descriptions of the design points and gets the survey point number from the suffix of the description code. When the Point# Description Code is found, the number after this code is used as the point# to match from the other file. For example, if description code is "PT" and the description for point# 101 in the first coordinate file is "CURB PT303", then point# 303 from the second coordinate file is used for the match. For the separate file option, there is a third method to match points between the files which is to use point numbers to match points between the files. The Match By Manually method has a dialog for picking pairs of point numbers for the survey and design to add. The list of available survey points to match with comes from points in the spreadsheet that have Point#, Survey Northing and Survey Easting but not the design data.



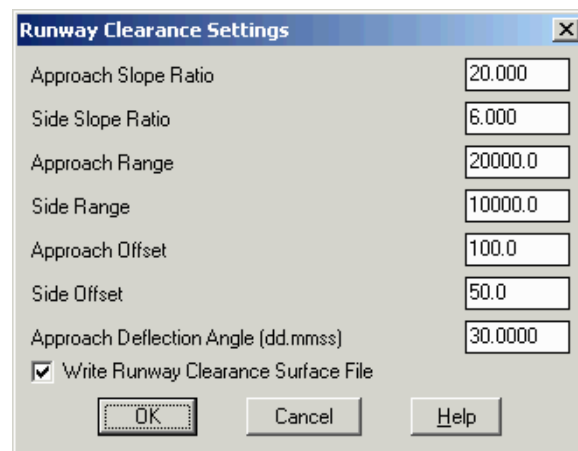
Grade > Triangulation File: the design elevation is determined by the elevation of the triangulation surface at each point.

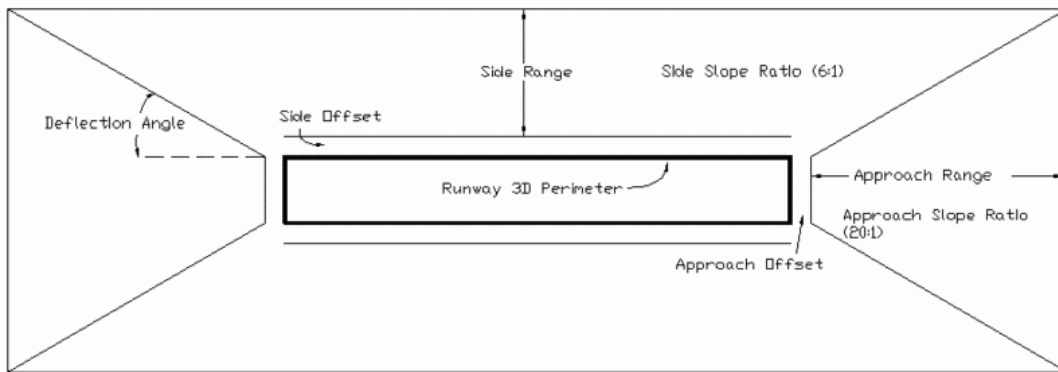
Grade > Grid File: the design elevation is determined by the elevation of the grid surface at each point.

Grade > 3D Polyline: When using a **3D Polyline** for the grade elevation, the program calculates the elevation along the 3D polyline at the position perpendicular from the point selected. This calculated elevation is then compared to the point(s) selected to determine the cut/fill values.

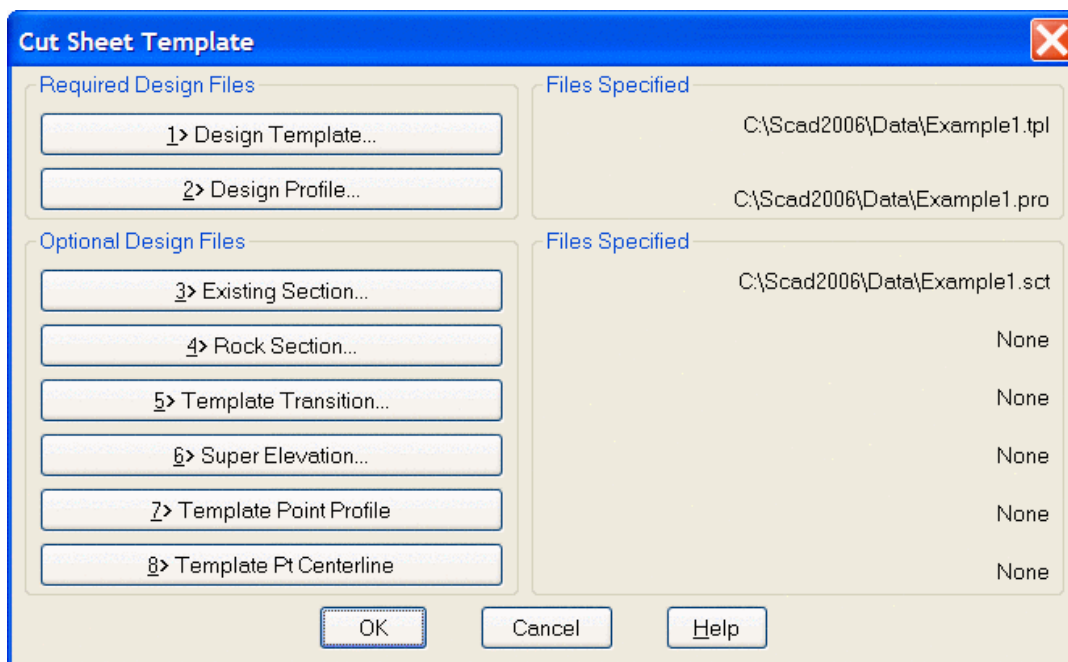
Grade > Cross Sections: With **Section Files**, the grade elevation is interpolated from the offset-elevation data in the section file based on the station-offset of the point along the centerline. When using this method, a centerline file (*.cl) must be specified for station-offset data.

Grade > Runway Clearance: This option defines the target surface as the airway clearance around a runway. This method is for clearance reports for tree and building tops by comparing points to this runway clearance surface. The runway surface is built from a 3D perimeter polyline of the runway along with slopes for the approach lanes and runway sides. The runway sides are offset level from the runway perimeter for the specified distances before starting the slopes. The parameters for the runway are defined in the dialog and illustrated in the graphic shown here. The Write Runway Clearance Surface File creates a triangulation surface file that you can draw or inspect for verification of a correct target surface.





Grade > Road Design: This option defines the grade elevation using road design files. For each point, the program finds the station-offset for the point along the centerline and then applies the road design at that station to determine the grade elevation. **Grade to Process** is used to define the surface to use for the cut sheet comparison. These grades are defined as Top Surface, usually final grade, or subgrades and correspond to the defined grades and subgrades within a template file. The required design files include a centerline (.CL) file, a template (.TPL) file, and a profile (.PRO) file. There are also several optional design files such as Superelevation, Template Point Profile and Template Point Centerline. The design files are created in the Civil Design module. Using the design files in *Cut Sheet* is similar to the *Process Road Design* command.



Centerline > Centerline File: This function assigns the Station and Offset fields in the spreadsheet by prompting for a centerline file (.CL) and locating each point along the alignment.

Centerline > Polyline: This function assigns the Station and Offset fields in the spreadsheet by picking a polyline, entering the starting station, and locating each point along the alignment.

Centerline > Points: This method defines the alignment by entering two points to define a line.

Report > Create Report: This function display a report of the cut sheet data using the current report settings. When Use Report Formatter is off, the report is shown directly in the standard viewer. Otherwise, the Report Formatter dialog is shown for customizing the report and outputting to different formats such as Excel.

Report > Report Settings: There are several settings for the report including decimal precision, prefix for cut and fill and distance units. For the Horizontal and Vertical Tolerance, the report highlights any points that exceed these tolerances. The **Distance Format** chooses between Angle-Distance, Delta X/Y, and North-South-East-West deltas. The **Cut/Fill Direction** chooses whether to report cut/fill as Survey relative to Design or vice versa.

The **Report Zero Elevations** option controls whether to skip or report points with zero elevation.

The **Report Statistics** option reports the min, max, average and standard deviations of the deltas at the bottom of the report.

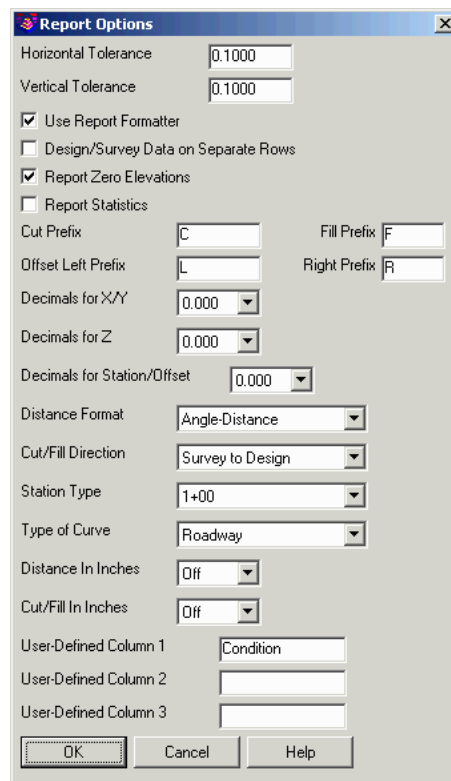
The **Design/Survey Data on Separate Rows** option creates two rows in the report for each record with the Design coordinate on one row and the deltas on the second row as shown here.

```
Point# Northing Easting Elevation Description Comment
229 499.997 1000.001 3.027
201 -0.005 -0.002 0.004 A

230 507.120 1000.158 2.929
202 -0.006 -0.003 0.001 B

231 515.694 1000.304 2.875
203 -0.005 -0.006 0.002
```

There are options for up to three **User-Defined Columns** which add extra fields to the spreadsheet and report.

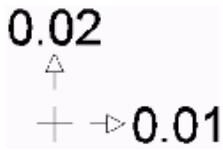


Draw > Draw Labels: This function uses the cut sheet data and draw settings to create labels in the drawing.

Draw > Label Settings: There are three types of labels to draw. The Mark Points Outside Tolerance draws a symbol at each point that exceeds the tolerances setup in Report Settings. The Colors Labels Outside Tolerance sets the color for points outside the tolerance. Otherwise these points are labeled using BYLAYER for the color.

The Draw Delta Symbol has three styles. The DX/DY style draws a symbol to show the direction of the delta X and delta Y along with the values. The deltas are due north-south and due east-west. The Rotate Deltas By

Centerline style draws the same DX/DY symbol but this style prompts for a centerline to align the deltas. The Delta Distance style draws an arrow from the design towards the survey point.

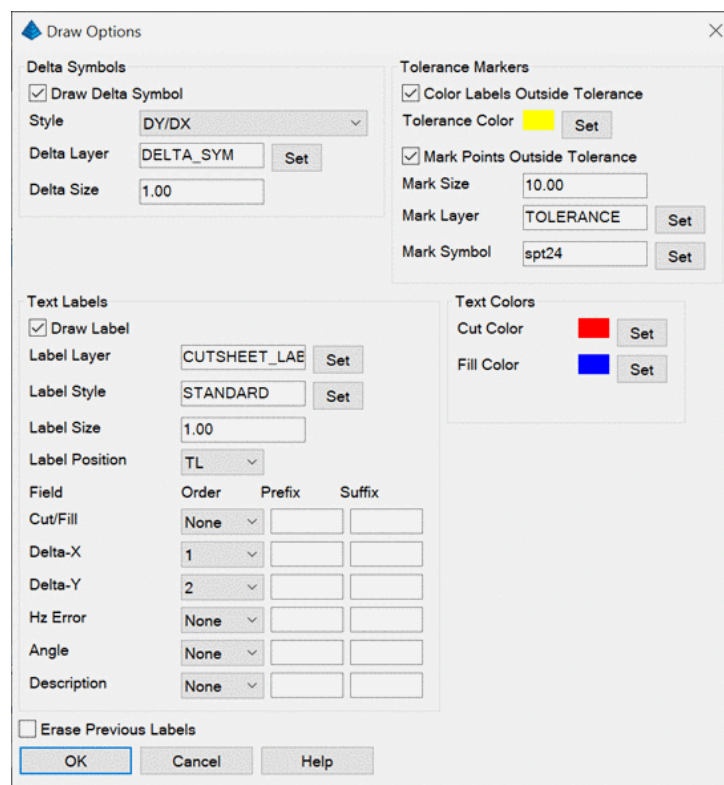


DX/DY symbol style



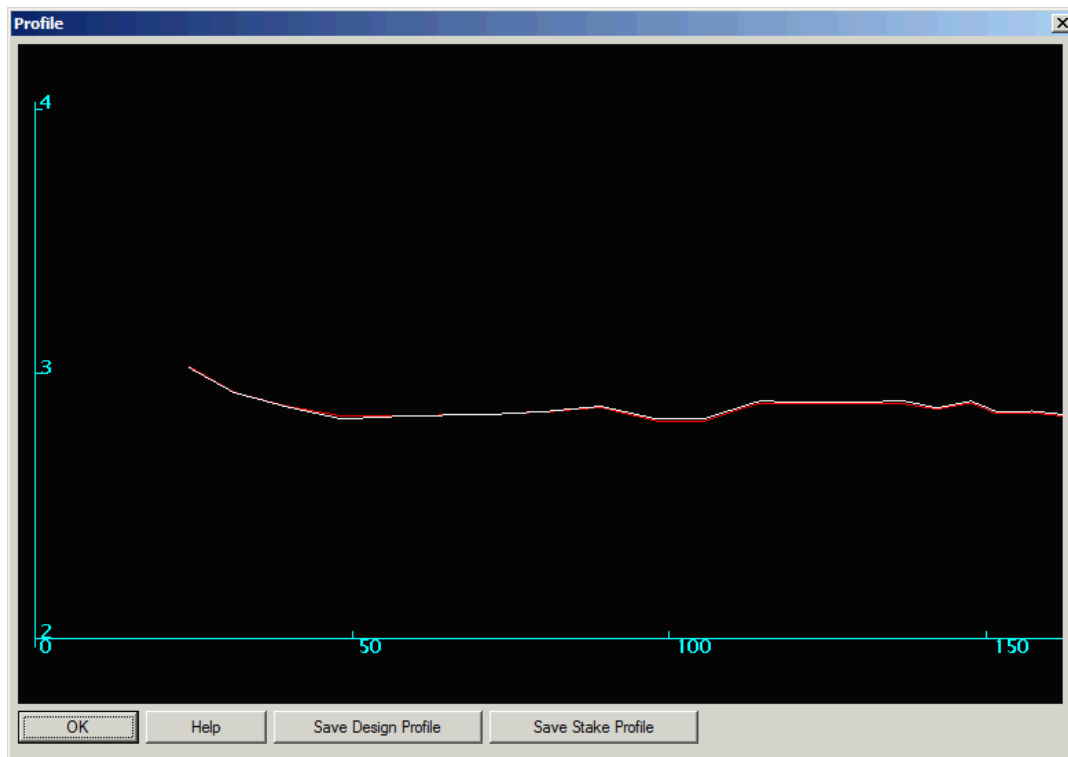
Delta Distance symbol style with Hz Error text label

The Text Labels creates text labels for cut/fill, delta-x, delta-y, horizontal distance error or description. There are settings for the sequence order, prefix, suffix, layer, style, color, size and position.



View > Zoom Plan View: This function zoom centers the drawing on the currently highlighted point.

View > Profile: Creates profiles connecting the survey and design points. The profiles are shown in a graphic preview dialog which has functions to save the profiles to .PRO files.



View > Hide By Point Numbers: This function is a way to filter the spreadsheet by point numbers. The function prompts for a range of point numbers and then only displays those points in the spreadsheet.

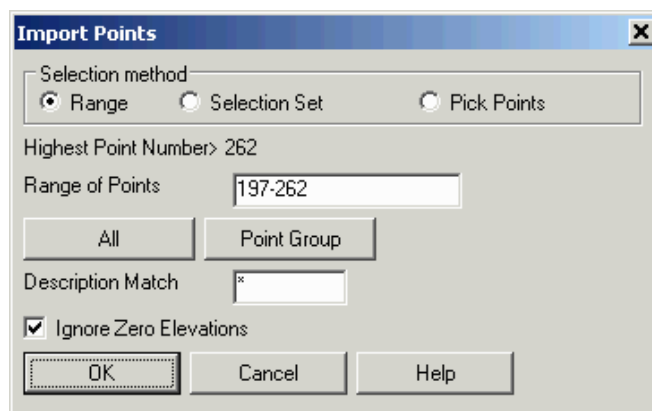
View > Hide By Description: This function is a way to filter the spreadsheet by description. Wildcard matching applies to the user-defined description filter.

View > Show All Rows: This function shows all the spreadsheet rows including those hidden by the above functions.

Examples of Cut Sheet reports comparing points are shown next.

Example 1: Cut Sheet comparing points to surface and using Report Viewer for report

1) Run Import > Points and select sample_cusheet.crd from Carlson Projects. Select the points to import by range as shown.



2) Run Grade > Triangulation File and choose sample_cutsheet.tin from Carlson Projects.

3) Run Centerline > Centerline File and choose sample_cutsheet.cl from Carlson Projects. Now all the data is ready for reporting.

	Point#	Description	Survey Z	Design Z	Cut/Fill	Station	Offset	Comment	Hz Error	Survey Northing	Survey Easting
1	197	STK BC	210.0000	210.5493	F0.549*	0000	-1.0000			323271.4614	533400.4033
2	198	STK BC	209.6000	210.1964	F0.596*	25.0000	-1.0000			323271.4614	533425.4033
3	199	STK BC	209.5000	210.1823	F0.682*	26.0128	-0.9997			323271.4614	533426.3905
4	200	STK BC	209.4500	210.1089	F0.659*	31.2170	-0.9977			323271.9804	533431.3544
5	201	STK BC	209.4200	210.0330	F0.613*	36.4211	-0.9959			323273.5148	533436.1036
6	202	STK BC	209.3900	209.9578	F0.568*	41.6253	-0.9945			323275.9983	533440.4328
7	203	STK BC	209.3500	209.8864	F0.536*	46.8295	-0.9933			323279.3235	533444.1547
8	204	STK BC	209.3200	209.8095	F0.489*	52.0336	-0.9925			323283.3466	533447.1084
9	205	STK BC	209.2800	209.7398	F0.460*	57.2379	-0.9920			323287.8937	533449.1661
10	206	STK BC	209.2700	209.6983	F0.428*	62.4421	-0.9918			323292.7680	533450.2389
11	207	STK BC	209.2500	209.6834	F0.433*	65.2353	-0.9998			323295.4614	533450.3905
12	208	STK BC	209.7000	209.6003	C0.100	90.2529	-1.0000			323320.4614	533450.3905

4) Run Report > Report Settings and make sure the settings match the dialog shown here.

Report Options ✕

Horizontal Tolerance

Vertical Tolerance

Use Report Formatter

Design/Survey Data on Separate Rows

Report Statistics

Cut Prefix

Fill Prefix

Decimals for X/Y

Decimals for Z

Distance Format

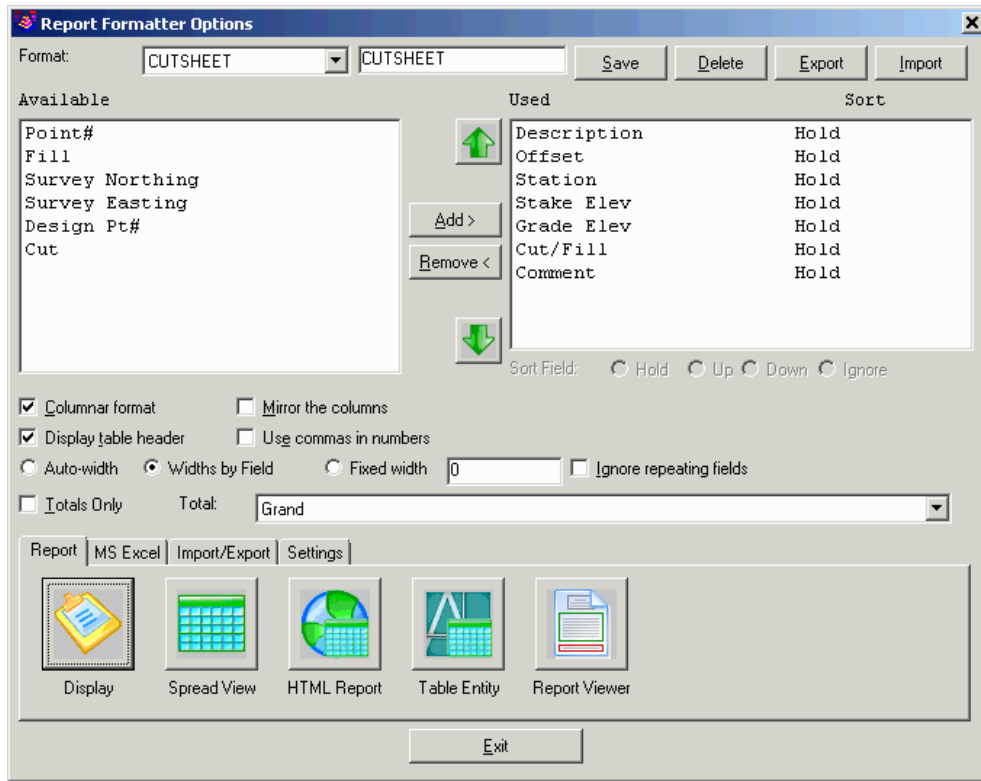
Cut/Fill Direction

Station Type

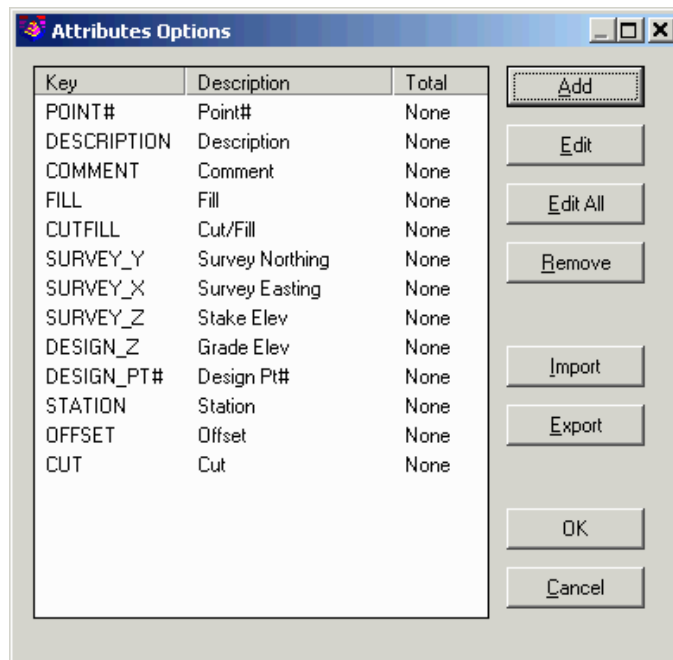
Type of Curve

Cut/Fill In Inches

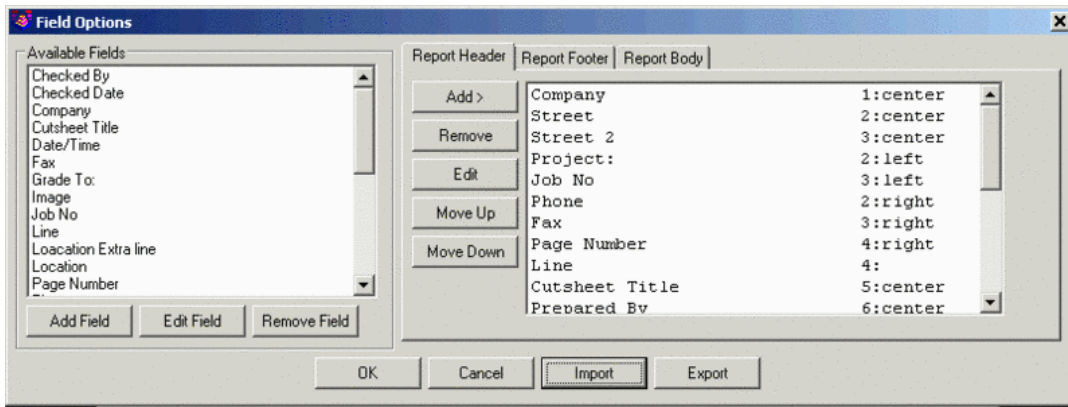
5) The next steps will format the report. Run Report > Create Report which brings up the Report Formatter. Move the Available fields over to the Used fields list as shown. Also, be sure that Widths by Field is toggled on.



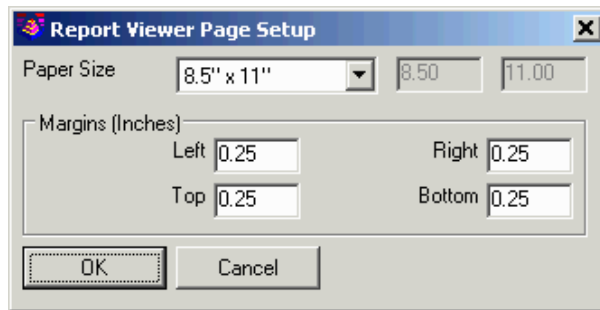
6) On the Report Formatter dialog, pick the Settings tab and then the Attrib Options button which controls how the values are formatted in the report. Then on the Attribute Options dialog, highlight Description and pick the Edit button. On the Attribute Display Options dialog for Description, turn off Auto Width and set the Fixed Width to 18. Then pick OK. Next highlight Comment in the list and pick Edit. Set the Fixed Width to 21 for Comment. Then pick OK on the Attribute Options to finish the attribute edits.



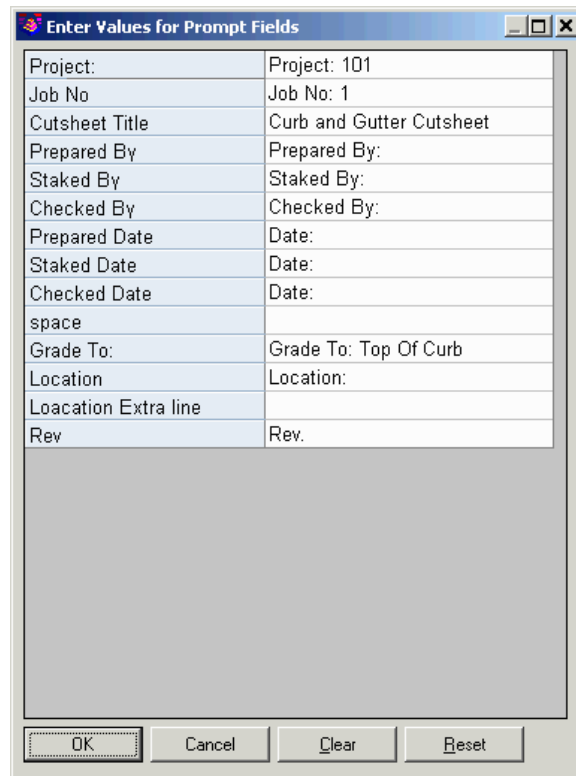
7) On the Settings tab for Report Formatter, pick the Field Options button which setups up the report header and footer. On the Field Options dialog, pick Import and select sample.cutsheet.fds from the Carlson Projects\Settings folder. Then pick OK.



8) On the Report tab for Report Formatter, pick the Report Viewer button. There is a dialog for setting the report page size. Pick OK.



Next is a dialog for entering report fields for the report header as setup in the previous step with the Field Options. Fill in the report fields and then pick OK.



Now the report is displayed in the Report Viewer which can be printed or saved to PDF, MS Word or Excel.

Carlson Report Viewer

1 of 2 100% Find | Next

Carlson

Project: 101 102 West Second St Tel #
 Job No: 1 Maysville, KY Fax #
 Page 1

Curb and Gutter Cutsheet

Grade To: Top Of Curb Prepared By: Date:
 Location: Staked By: Date:
 Rev. Checked By: Date:

Description	Offset	Station	Stake Elev	Grade Elev	Cut/Fill	Comment
STK BC	L1.000	0+00.000	210.000	210.549	F0.549*	
STK BC	L1.000	0+25.000	209.600	210.196	F0.596*	
STK BC	L1.000	0+26.013	209.500	210.182	F0.682*	
STK BC	L0.998	0+31.217	209.450	210.109	F0.659*	
STK BC	L0.996	0+36.421	209.420	210.033	F0.613*	
STK BC	L0.995	0+41.625	209.390	209.958	F0.568*	
STK BC	L0.993	0+46.830	209.350	209.886	F0.536*	
STK BC	L0.993	0+52.034	209.320	209.809	F0.489*	
STK BC	L0.992	0+57.238	209.280	209.740	F0.460*	
STK BC	L0.992	0+62.442	209.270	209.698	F0.428*	

SurvCADD Edit : c:\scad2006\USER\scadrprt.tmp

File Edit Settings

Open Save Print Exit Find Screen Hide

Cut Sheet 3/26/2005 12:35

Coordinate File> C:\scad2006new\DATA\extest.crd

Survey Pt#	Survey Elevation	Design Pt#	Design Elevation	Delta-Y	Delta-X	Cut/Fill	Description
2	392.969	102	393.779	0.300	0.250	F9 3/4"	B-TOE
3	394.812	103	394.152	0.300	0.250	C7 15/16"	B-TOE
4	395.287	104	394.627	0.300	0.250	C7 15/16"	B-TOE

Example 2: Cut Sheet Report comparing points from the Current Coordinate File and with the Use Feet-Inches For Cut/Fill options on.

SurvCADD Edit : c:\scad2006\USER\scadrprt.tmp

File Edit Settings

Open Save Print Exit Find Screen Hide

Cut Sheet 3/26/2005 13:00

Survey Coordinate File> C:\scad2006new\DATA\extest.crd
 Design Coordinate File> C:\SCAD2006NEW\DATA\EXAMPLE1.CRD

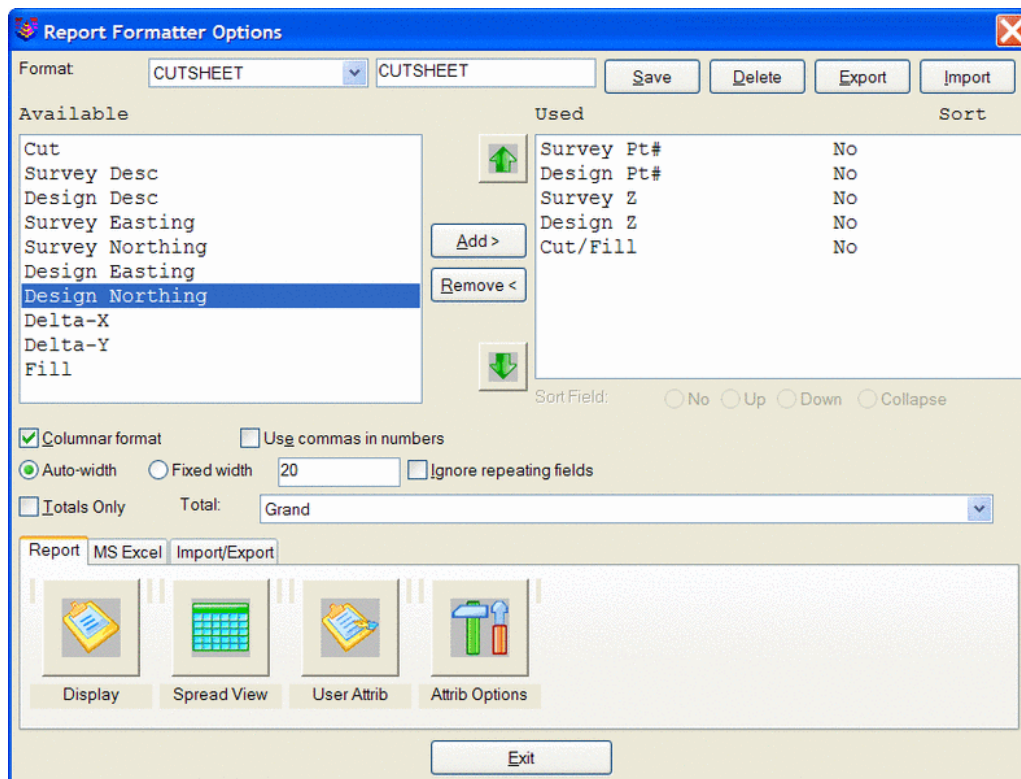
Pt#	Northing	Easting	Survey Elevation	Design Elevation	Delta-Y	Delta-X	Cut/Fill	Description
4	4060.37	8306.64	395.29	394.63	0.00	-0.15	C0.66	B-TOE
5	4068.79	8323.54	394.04	395.50	-0.12	-0.25	F1.46*	B-TOE
6	4080.35	8342.97	394.26	395.72	-0.12	-0.05	F1.46*	B-TOE

* = Exceeded tolerance
 Horz Tolerance: 1.000 Vert Tolerance: 1.000

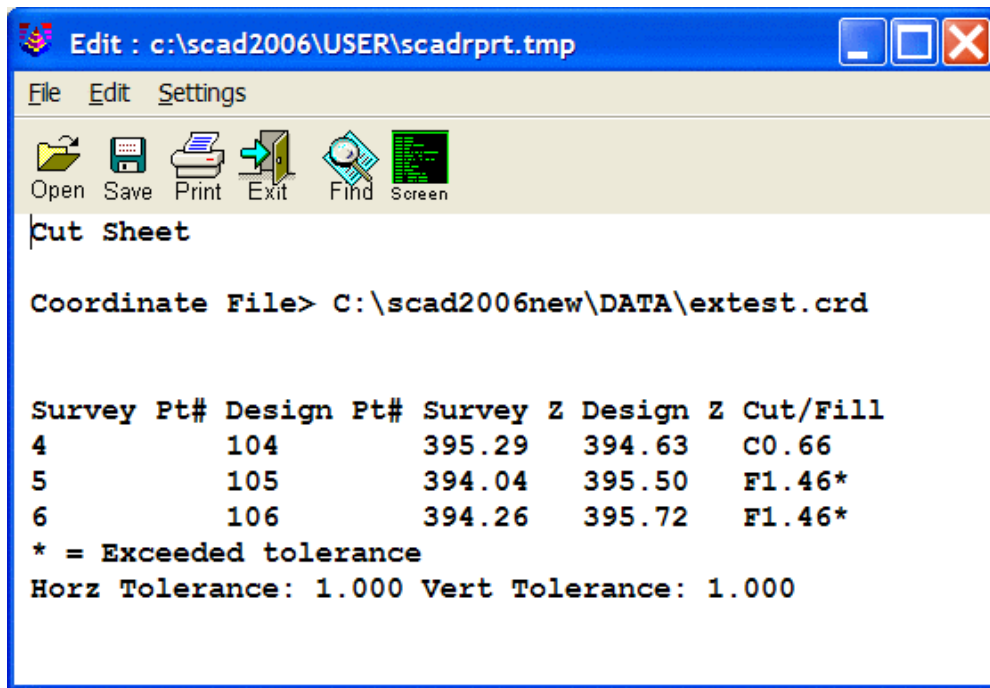
Example 3: Cut Sheet Report comparing points from Another Coordinate file, reporting coordinates for the points.

Example 4: Steps for Comparing Points in Current Coordinate file and using Report Formatter Option to customize report output to user preference.

- 1) Check on Use Report Formatter on the Cut Sheet Report dialog.
- 2) Specify points to compare by one of the four methods described above for comparing points within the current coordinate file.
- 3) Select report content by highlighting the desired data from the Available list on the left side of the dialog box and then pressing the Add button to place the selected data in the Used list. Standard window selection methods using the Ctrl and Shift keys can be used to select more than one item at a time. After moving the selected data to the Used window it may be necessary to move data up or down to obtain the desired order of your report. To do this use the up and down arrows located on the left of the Used window.



4) When the desired data has been specified in the Used window press the Display button at the bottom left of the dialog. For more detailed information on using the report formatter see the Report Formatter section of this manual.



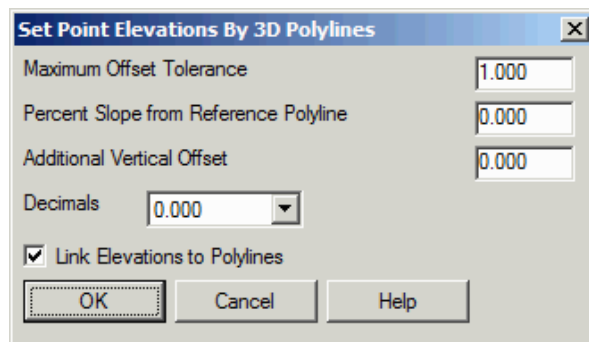
Pulldown Menu Location: Survey

Keyboard Command: cutrprt

Prerequisite: A coordinate (.CRD) file

Set Point Elevations by 3D Polyines

This command assigns elevations to points by referencing 3D polyines. The station-offset is calculated for each point to the nearest reference 3D polyline. The point must be within the specified Max Offset Tolerance in order to be elevated. The elevation is calculated from the elevation of the reference 3D polyline at the station combined with the specified percent slope times the offset plus the vertical offset. The Decimals setting is for the elevation label of the point. The elevation for the coordinate file always uses full precision. The option to Link Elevations To Polyines will update the point elevations when the reference polyline is changed.



Prompts

Options Dialog

Select points from screen, group or by point number [**<Screen>/Group/Number**]? *press Enter*

Select points to elevate.

Select objects: *pick the points to elevate*

Select reference 3D polyines.

Select objects: *pick the reference 3D polyines*

Elevating points...

Elevated 10 points.

Pulldown Menu Location: Survey and 3D Data in Civil

Keyboard Command: 3dpts_3dp

Prerequisite: 3D polylines

Set Point Elevations by Surface Model

This command assign elevations to points by a triangulation or grid surface model. For each of the points, the routine looks up the elevation from the surface model at the point x,y location. The option to Link Elevations To Surface Model will update the point elevations when the reference surface model is changed.

Prompts

Choose Grid or Tmesh file to process dialog *choose existing GRD, TIN or FLT file*

Select points from screen, group or by point number [**<Screen>/Group/Number**]? *press Enter*

Select points to elevate.

Select objects: *pick the points to elevate*

Elevating points...

Elevated 10 points.

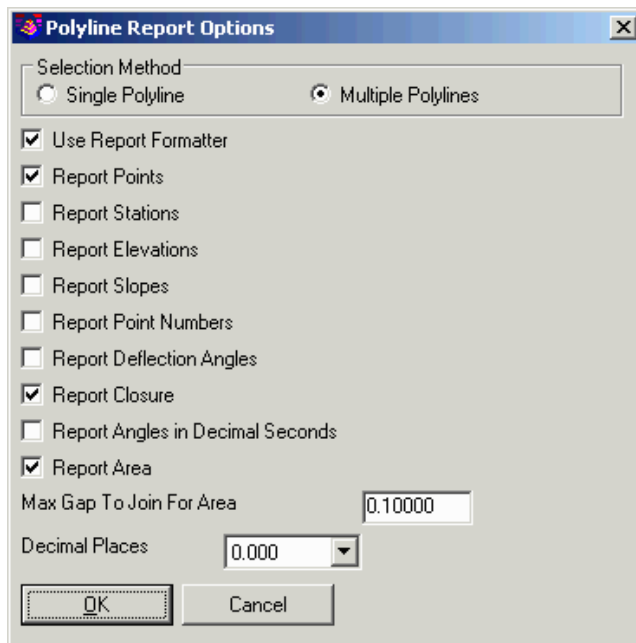
Pulldown Menu Location: Survey and 3D Data for Civil

Keyboard Command: 3dpts_tin

Prerequisite: A surface model

Polyline Report

This command generates a report of data in the selected polylines. After starting the command, press O for Options to set various report options. When the Report Points option is active, the report includes the coordinates, angle-distance and curve data for all the points. With Report Points turn off, the report includes just the perimeter, area and layer names of the polylines. The closure can be reported between the starting and ending points of the polyline. The polyline area can also be reported. With the Multiple Polylines option, you can report a selection set of polylines with their total length at the end of the report.



Polyline Report

Northing	Easting	Bearing	Distance
4657.495	5452.844		
		N 40d45'51'' E	84.323
4721.362	5507.902		
		Radius: 175.795 Chord: 249.282 Degree: 32d35'33'' Dir: Right	
		Length: 277.088 Delta: 90d18'35'' Tangent: 176.747	
		Chord BRG: N 85d55'08'' E Rad-In: S 49d14'09'' E Rad-Out: S 41d04'26'' W	
		Radius Point: 4606.577,5641.050	
4739.102	5756.552		
		S 24d29'28'' E	122.817
4627.336	5807.466		
		S 74d29'33'' W	199.062
4574.114	5615.650		
		N 62d53'05'' W	182.885
4657.470	5452.866		
		Closure Error Distance> 0.03419 Error Bearing> N 41d22'21'' W	
		Closure Precision> 1 in 25333.8 Total Distance> 866.174	
		Polyline Area: 47735.6 sq ft, 1.1 acres	

Prompts

Options/Select polyline to report: *pick a polyline*

Standard Report Viewer Displays the report for the selected polyline.

Options/Select polyline to report (Enter to End): *press Enter*

Pulldown Menu Location: Survey > Polyline Tools

Keyboard Command: plreport

Prerequisite: A polyline

Polyline to Deed File

This command generates a deed (.PDD) file from the geometry of a selected polyline. This file can be opened using *Process Deed File* which allows you to edit the deed data and generate reports.

Prompts

Deed File To Write: *choose file location and name*

Select Polyline To Process: *select polyline*

Done.

Pulldown Menu Location: Survey > Polyline Tools

Keyboard Command: pl2pdd

Prerequisite: a polyline

Polyline to RW5 File

This command generates a raw data (.RW5) file for the selected polyline. This file can be opened using *Edit Process Raw Data File*, which allows you to process the raw data (.RW5) file to generate coordinate points, calculate closure and perform coordinate adjustments by the compass, crandall, transit and least squares adjustment routines.

Prompts

RW5 File to Write (Standard Windows File Selection Dialog): *choose file location and name*

Select Polyline To Process: *select polyline*

Done.

Pulldown Menu Location: Survey > Polyline Tools

Keyboard Command: pl2rw5

Prerequisite: a polyline

Crandall Polyline Adjustment

This command applies a crandall adjustment to a perimeter defined by a polyline. This is a way to distribute rounding errors on perimeters without altering the record dimensions. The command draws a new polyline of the adjusted perimeter and creates a report. The closure error is the difference between the end point of the polyline and the specified closing point that the program prompts for. For a closed loop adjustment, the closing point is the starting point of the polyline.

Here is an example report for a closed loop.

Crandall Polyline Adjustment

Closure Results

Starting Point : N 5212.097 E 4476.155

Ending Point : N 5209.953 E 4491.172

New Point : N 5212.097 E 4476.155

Azimuth Of Error : N 81°52'24" W

North Error : -2.144

East Error : 15.017

Distance Error : 15.169
Distance Traverse: 4093.830
Closure Precision: 1 in 269.9

Original Data

Angle Distance
S 01°56'05" E 826.057
S 84°59'27" E 1154.279
N 02°17'10" E 914.234
N 89°29'16" W 1199.260

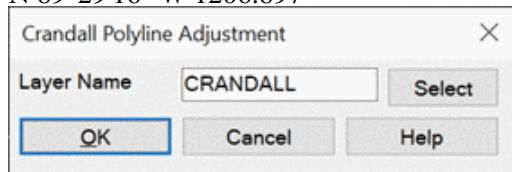
Adjusted Point Comparison

Original Adjusted

Northing Easting Northing Easting Dist Angle
5212.097 4476.155 5212.097 4476.155 0.000 N 90°00'00" E
4387.393 4504.014 4387.393 4504.014 0.883 N 01°56'05" W
4287.253 5646.513 4287.253 5646.513 7.558 N 78°20'03" W
5201.307 5683.005 5201.307 5683.005 7.666 N 74°17'17" W
5212.097 4476.155 5212.097 4476.155 15.169 N 81°52'24" W

Adjusted Data

Angle Distance
S 01°56'05" E 825.175
S 84°59'27" E 1146.879
N 02°17'10" E 914.782
N 89°29'16" W 1206.897



Prompts

Pick polyline for Crandall Adjustment: *pick polyline*

Reverse polyline [Yes/<No>]? *press Enter*

Close polyline to start point or pick new closing point [<Close>/Pick]? *press Enter*

Pulldown Menu Location: Survey > Polyline Tools

Keyboard Command: crandalladj

Prerequisite: A polyline

Compass Polyline Adjustment

This command applies a compass adjustment to a perimeter defined by a polyline. After selecting the polyline, the program displays arrows to show the polyline direction and you have an option to reverse the polyline. The command draws a new polyline of the adjusted perimeter and creates a report. The closure error is the difference between the end point of the polyline and the specified closing point that the program prompts for. In the dialog, you can set the Traverse Type as Open or Closed. For a closed loop adjustment, the closing point is the starting point of the polyline. For an open traverse, the program will prompt for the closing point.

For the Geodetic Report option, the program reports the geodetic distances and angles. To use this option, the grid

projection must be define in Settings > Drawing Setup.

Here is an example report for a closed loop.

Compass Polyline Adjustment

Closure Results

Starting Point : N 4854.766 E 5357.221

Ending Point : N 4854.636 E 5357.533

New Point : N 4854.766 E 5357.221

Azimuth Of Error : N 67°22'29" W

North Error : 0.130

East Error : -0.312

Distance Error : 0.338

Distance Traverse: 1029.503

Closure Precision: 1 in 3050

Original Data

From To Angle Distance

4 5 S 13°01'45" E 392.218

5 11 N 44°23'59" E 296.297

11 N 60°02'20" W 340.987

Adjusted Point Comparison

Original Adjusted

Point# Northing Easting Northing Easting Dist Angle

4 4854.766 5357.221 4854.766 5357.221 0.000 N 90°00'00" E

5 4472.645 5445.645 4472.695 5445.526 0.129 N 67°22'29" W

11 4684.343 5652.952 4684.430 5652.744 0.226 N 67°22'29" W

4854.636 5357.533 4854.766 5357.221 0.338 N 67°22'29" W

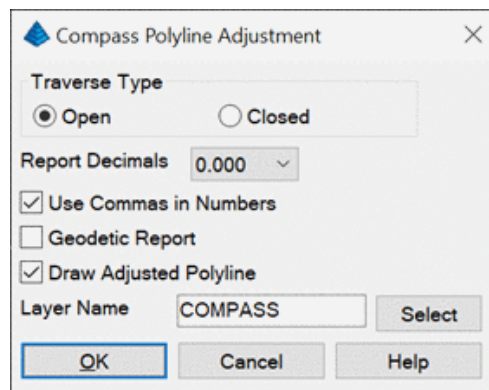
Adjusted Data

From To Angle Distance

4 5 S 13°00'50" E 392.143

5 11 N 44°22'56" E 296.261

11 N 60°02'29" W 341.098



Prompts

Pick polyline for Compass Adjustment: *pick polyline*

Reverse polyline [Yes/<No>]? *press Enter*

Pick new closing point: *pick point*

Pulldown Menu Location: Survey > Polyline Tools

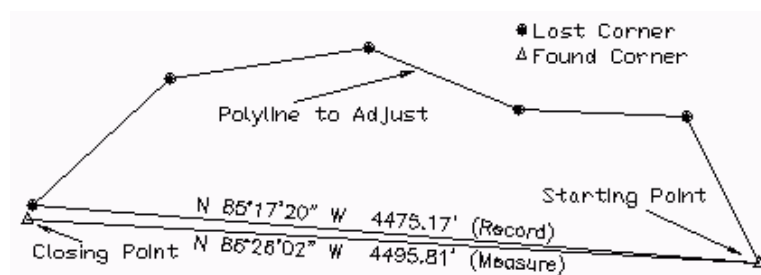
Keyboard Command: compassadj

Prerequisite: A polyline

Grant Boundary Adjustment

This command applies a Grant Boundary Adjustment by rotating and scaling a polyline. Before running this command, the grid projection must be set in Drawing Setup and a polyline must be drawn.

The Grant Boundary method is used to set lost corners on perimeters within public lands. Distances between the record and measured are compared to define a ratio for adjustments. A rotation is defined by the difference between the record and measured bearings to preserve the angular relationship at the lost corners and to adjust the distance at the same ratio through each lost corner.



Prompts

Pick polyline for Grant Boundary Adjustment: *pick polyline*

Layer name for adjusted polyline <GRANT>: *press Enter*

Reverse polyline [Yes/<No>]? *press Enter*

Pick new closing point: *pick point*

Pulldown Menu Location: Survey > Polyline Tools

Keyboard Command: grantadj

Prerequisite: A polyline

4 Sided Building

Often only one or two sides of a building are surveyed in the field. This routine completes the building by drawing the other sides. *4 Sided Building* creates a parallelogram given two connecting lines, or given a polyline with two segments. With two lines, there is an option to make the parallelogram as a polyline or as four lines. When only one side is defined, the program will prompt for the building width. Besides using linework to define the sides, you can use points by entering P at the prompt to switch to points mode.

Prompts

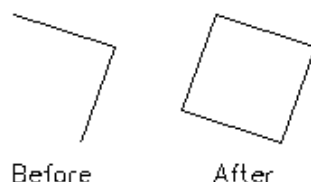
Options/Points/<Pick a line or polyline>: *pick a line*

Pick another side (Enter for none): *pick a line*

Convert the lines into a polyline [<Yes>/No]? *press Enter*

Options/Points/<Pick a line or polyline>: *press Enter*

Entering O for options lets you choose whether or not to be prompted to set the new polyline width, and for whether to default the width to make a square building with one sided input.



Pulldown Menu Location: Survey

Keyboard Command: 4sided

Prerequisite: A polyline with two segments or two adjoining lines

CAPD Command Reference

Cadastral Automated Plat Drafting System (CAPD) was developed by the BLM. Here are a list of command names from CAPD and the equivalent commands in Carlson. This list is to help CAPD users find these functions within Carlson.

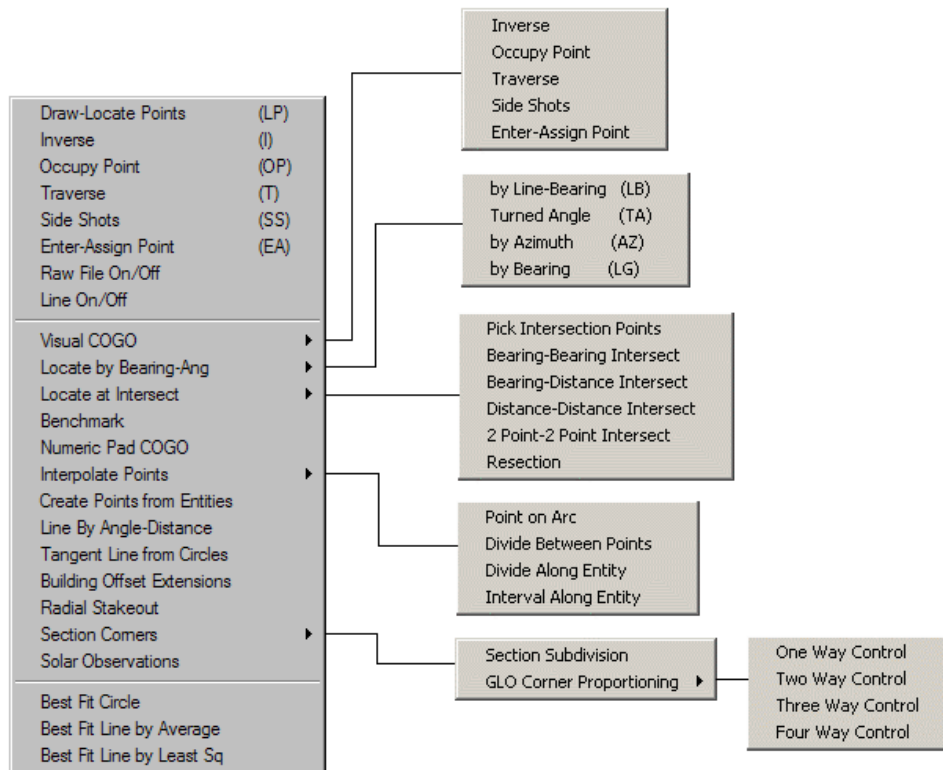
CAPD Command	Carlson Command	Menu Location	Description
ALIGN	TXTLINEUP	Edit > Text > Line Up Text	Used to align text
BAREA	DEFAREA, PLAREA	Area/Layout > Area De- faults, Area By Closed figure Polyline	Computes area of a closed figure
BBI	BBINT	COGO > Locate At In- tersect > Bearing-Bearing Intersect	Bearing-bearing intersection
BDI	BDINT	COGO > Locate At Inter- sect > Bearing-Distance Intersect	Bearing-distance intersection
BDA, BDT	AUTOANN	Annotate > Auto-Annotate	Bearing distance labels
BTRV	IA	Area/Layout > Inverse With Area	Boundary traverse
CADINIT	LDEF	Annotate > Annotate De- faults	Set up default layers, fonts, line type
CHELEV	CHGELEV	Edit > Change > Elevations	Change elevations of entities
CMP	COMPASSADJ	Survey > Polyline Tools > Compass Polyline Adjust- ment	Compass adjustment

DBK	BREAKAT	Edit > Break > At Intersection	Break lines that cross using grid coordinates
DBK2	GEODBK	COGO > Geodetic > Double Break	Geodetic double break lines that cross
DDI	DDINT	COGO > Locate At Intersection > Distance-Distance Intersect	Distance-distance intersection
DDSETANN, DS	LDEF	Annotate > Annotate Defaults	De- Set up default layers, fonts, line type
DL	LDEL	Edit > Erase > Erase By Layer	Delete layer
DPROP	GEODPROP	COGO > Geodetic > Double Proportion Line Division	Double proportion computation
EV	RMVERTEX	Edit > Polyline Utilities > Remove Polyline Vertex	Delete vertex in polyline
FLIP	FLIPTEXT	Edit > Text > Flip Selected Text	Flips text 180
FTCRV	2TANLIN	Draw > Arc > 2 Tangents, Fit curve Radius	
GETPRO	SETUP	Settings > Drawings Setup	Set up projection
GID	INVERSE	COGO > Inverse	Display geodetic coordinates of points
GLBL	LABLAT	Annotate > Label Lat/Long	Label geodetic coordinates
GMID	GEOMID	COGO > Geodetic > Middle Break	Breaks line at geodetic midpoint
GPOINT	LPOINT	Points > Draw/Locate Points	Input point by lat/lon/elv
GRIDRT	TWIST4	View > Twist Screen > Restore Due North	Switch between true north and grid north
INIDGEO	READPT, POINTTBL	Points > Import Text/ASCII File, Annotate >	Import points and create table of points Point Table
IPG	LABLAT	Annotate > Label Lat/Long	Create block of coordinates
IRRB	GEOIRRB	COGO > Geodetic > Irregular Boundary Adjustment	Irregular boundary adjustment
ITRAV	INVERSE	COGO > Inverse	Inverse between points
MEDLIN	MEDOFF	Edit > Offset > Offset	Median Create midline between 2 polylines

PRCRV	CINFO	Inquiry > Curve Info	List curve info
REV	BRGQUAD	Annotate > Flip Labels > Reverse bearing Switch Bearing Quadrant	
SETSL	PICKTXTSTYLE	Draw > Text > Pick Text Sets style for text Style	
SHOPRO	SETUP	Settings > Drawings Setup	Shows projection information
SPROP	GEOSPROP	COGO > Geodetic > Single Single proportion breaks Proportion Line Division	line geodetically
TRAV	GEOTRAV	COGO > Geodetic > Geodetic traverse for input Geodetic Traverse	of bearings and distances

COGO Menu

This chapter provides information on using the commands from the COGO menu to perform coordinate geometry operations in your drawing. The top section provide basic COGO routines, with optional quick keys. The bottom section provides numerous survey functions, including the easy-to-use Visual COGO and also Numeric Pad COGO.



Inverse

This command reports the bearing/azimuth and horizontal distance between two points. The command prompts for a series of points. Use the appropriate object snap mode to select the points from the screen, or use the point numbers to reference coordinates stored in the current coordinate (.CRD) file. The results are then displayed. This command is also used in conjunction with the *Traverse* and *Sideshot* commands to occupy and backsight two points. The last two points you Inverse to are the Backsight and the Occupied point for the Traverse and Sideshot commands. An

attractive feature of *Inverse* is that you can enter T or SS within the command and go directly to *Traverse* or *Sideshot*. Even a single S will transmit to *Sideshot*. Hotkeys are not case sensitive. Press [Enter] at the point prompt to end the command.

You can also inverse around an arc by inverting to the Point of Curvature (PC), and then entering an A for Arc option. The program will ask for the radius point, the curve direction left or right and the PT point. The curve data is then reported. There is an unequal PC-Radius and PT-Radius distance check. The tolerance for this is set in the *Area Label Defaults* command.

After picking the first point, there is a keyboard option for Multiple which will prompt for a range of point numbers to report as a sideshot inverse.

There are several input options for *Inverse* that are set by entering O for Options on the command line. Sideshot inverse holds the current occupied point and calculates the bearing/distance to each entered point. The Pairs option reports the bearing/distance between pairs of points and not for every entered point. For example, if points 1,2,11,12 were entered, the bearing/distance would be reported for 1,2 and 11,12 but not 2,11. The Auto Increment option uses the next point number by just pressing Enter. To exit the routine with Auto Increment active, End must be entered.

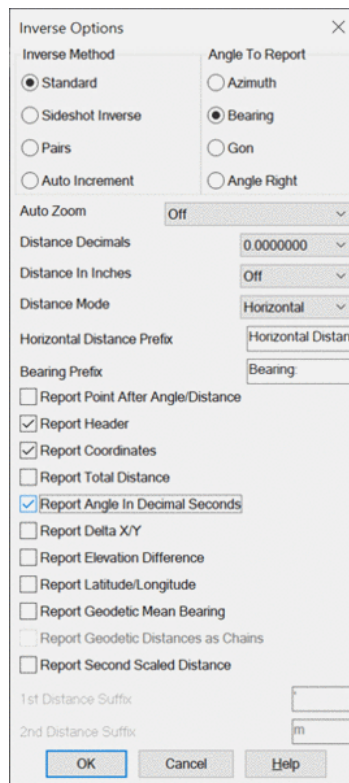
The *Auto Zoom* settings under Inverse Options will zoom the display as needed to have the occupied point or both the occupied and backsight points visible. The *Report Total Distance* option displays a running total of all inversed distances during the current run of the routine.

The Report Geodetic Mean Bearing option reports the geodetic bearing at the to point (forward), at the from point (back) and the mean bearing. The geodetic distance is also report for the geodetic distance at zero elevation and at ground elevation. The coordinates are converted to lat/lon using the projection setup under the Drawing Setup command. The program reports that lat/lon, convergence angle and grid scale factor at the from and to points. Here's an example for SP83 VT,
Northing(Y) Easting(X) Elev(Z)
218623.2996 485210.2502 0.0000
Northing(Y) Easting(X) Elev(Z)
218439.0529 487144.1875 0.0000
Bearing: S 84°33'28" E Horizontal Distance: 1942.6941325
Lat: 43°01'05.81806" Long: -76°49'09.53807"
Convergence: N 02°56'59" E Scale: 1.0014892493
Lat: 43°01'04.98404" Long: -76°48'43.45145"
Convergence: N 02°56'41" E Scale: 1.0014841465
Geodetic Forward Bearing: S 87°30'28" E
Geodetic Back Bearing: S 87°30'09" E
Geodetic Mean Bearing: S 87°30'18" E
Geodetic Distance: 1942.984 Zero Elev, 1942.984 Ground Elev

There are also several angle output options that are set at the second prompt in Options. The angle can be reported as either Bearing, Azimuth, Gon or Angle Right. You can also specify to report angles with decimal seconds. The distance settings include the number of decimals for distances, whether to report slope or horizontal distance and whether to report distances in feet and inches format. The Report Total Distance option will report the cumulative distance for all the inverses. The Report Delta X/Y will distances as delta north-south-east-west instead of angle and distance. For Report Latitude/Longitude, the grid projection must be set in Drawing Setup. The Report Point After Angle/Distance controls whether the point coordinates are reported before or after the angle and distance. The Report Header controls whether to have a header label line for the point data. The Report Coordinates option choose whether to report the northing, easting and elevation of the points. The Report Elevation Difference option will report the delta Z between the pairs of points. The Report Second Scaled Distance option will report a second

distance value that is scaled from the first distance value using the scale factor defined in Drawing Setup. When the Second Scaled Distance option is on, there are settings for the suffix to use for both the first and second distance to help identify them separately in the report.

For instruction on how to insert either new or existing points into the drawing, see Draw-Locate Points in the Points Commands section of the General Commands chapter.



Prompts

Calculate Bearing & Distance from starting point?

Traverse/SideShot/Options/Arc/Multiple/Pick point or point number: *pick a point*

Traverse/SideShot/Options/Arc/Multiple/Pick point or point number: 9

PtNo. Northing(Y) Easting(X) Elev(Z) Description

9 4909.25 4648.37 0.00

Bearing: N 81d8'54" E Azimuth: 81d8'54"

Horizontal Distance: 261.17407461

Pulldown Menu Location: COGO

Keyboard Command: inverse, i

Prerequisite: None

Occupy Point

This command sets the occupied point and backsight angle for other COGO commands such as *Traverse*. For setting the occupied point, you have the option of picking a point on the screen, entering coordinates at the command line or typing in a point number that will be read from the current coordinate (.CRD) file. Four options are available for determining the backsight direction: Azimuth, Bearing, None and Point. For the default Point option, you may pick

a point on the screen, input coordinates, or type a point number that will be read from the current coordinate file. For the Azimuth and Bearing option, you enter the backsight angle in the selected format. The None option sets the backsight to an azimuth of 0 (zero) or North. You can also set the occupied point by using the *Inverse* command. If you inverse from point 3 to point 1, you have set point 1 as the occupied point and point 3 as the backsight. For more information, see the *Inverse* command.

The current occupied point and backsight are shown in the lower right hand corner of the AutoCAD status bar just below the command line.

Prompts

Set Occupied Point

Pick point or point number: *pick a point (5000 5000 0.0)*

Set backsight method [Azimuth/Bearing/None/<Point>]? *press Enter*

Set Backsight Point

Pick point or point number: *pick a point (5184.76 5381.3 0.0)*

For instruction on how to insert either new or existing points into the drawing, see Draw-Locate Points in the Points Commands section of the General Commands chapter. This feature can be found in the Points pulldown of all menus.

Pulldown Menu Location: COGO

Keyboard Command: occpoint, op

Prerequisite: None

Traverse

This command allows the user to input any combination of turned angles, azimuths or bearings to define a traverse or figure. The command prompts for an Angle-Bearing Code which defines the angle or bearing type. This command always occupies the last point it calculated and backsights the point before that.

Codes 1 through 4 define the bearing quadrants:

- 1 = Northeast
- 2 = Southeast
- 3 = Southwest
- 4 = Northwest

The remaining codes define as follows:

- 5 = Azimuth
- 6 = Angle turned to the left
- 7 = Angle turned to the right
- 8 = Deflection angle left
- 9 = Deflection angle right

For both the Angle-Bearing Code and the Distance prompt, the user can enter point-defined responses: two points separated by an asterisk, as in 2*3 for the bearing (or distance) defined by 2 to 3. You can also add math expressions. For angles, 2*3+90 would deflect 90 degrees right from 2 to 3. For distances, 2*3/2 would mean half the distance of 2 to 3. You do not need to enter N before entering a number-defined distance. Just bring up the number inverse prompt. For distances, the # symbol can also be used to indicate point numbers like in SurvCE. So 2#3 means the distance between points 2 and 3.

The command draws lines between located points (if the Line On/Off in the COGO menu is set to on) and plots the points calculated and stores them in the current coordinate (.CRD) file if point numbering is On. The point settings

are defined in the *Point Defaults* command. If Point Protect is turned on, *Traverse* checks if the point numbers are already stored in the file. Point Protect is set in the *Coordinate File Utilities* command.

There are Angle-Bearing code input options for *Traverse* that are set by entering O for Options. The Angle Right option prompts for the angle right and skips the angle-bearing code prompt. The Azimuth option prompts for the azimuth and skips the angle-bearing code prompt.

Prompts

Occupied Point ?

Pick point or point number: *pick a point*

You will only be prompted for the occupied point the first time you use the command.

Use the *Inverse* command to set the occupied and backsight points.

Exit/Options/SideShot/Inverse/Enter Azimuth (ddd.mmss) <>: o

Angle prompt angle right or azimuth only [Right/Azimuth/Prompt]? p

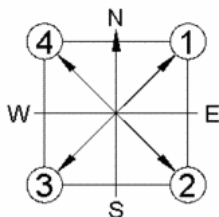
Exit/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <5>: *press Enter* Pressing Enter uses the default angle right code.

Enter Angle (dd.mmss) <90.0000>: *88.1324* You can also enter L or R to define an angle 90 degrees Left or Right.

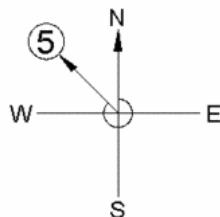
Backsight Point ?

Pick point or point number: *pick a point*

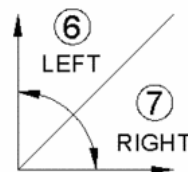
Number inverse/<Distance>: *100*



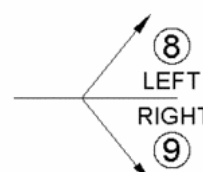
Codes used for
BEARINGS



Codes used for
AZIMUTHS



Codes used for
Angles
LEFT & RIGHT



Codes used for
Deflection Angles
LEFT & RIGHT

Select Coordinate (.CRD) File This dialog only appears if there is not a current coordinate (.CRD) file.

Exit/Options/Line/Side Shot/Inverse/<Angle-Bearing Code <7>>: *14*9-45.2045* Uses the bearing defined by point numbers 14 & 9 and subtracts the angle 45 degrees, 20 minutes, and 45 seconds. You can use a + or - in this type of entry.

Number inverse/<Distance>: *N* (*note: you can enter 14*9/2 here, as well*)

Point number inverse (i.e. 10*20): *14*9/2* This causes the command to recall the distance from point number 14 to 9 and divide it by 2.

Exit/Options/Line/Side Shot/Inverse/<Angle-Bearing Code <7>>: *L*

Select Line or Polyline that defines Bearing: *select line that defines bearing*

Number inverse/<Distance>: *100*

Exit/Options/Line/Side Shot/Inverse/<Angle-Bearing Code <7>>: *E* Enter E to end the command. Enter S or SS to execute the *Side Shots* command or I to execute the *Inverse* command.

For instruction on how to insert either new or existing points into the drawing, see Draw-Locate Points in the Points Commands section of the General Commands chapter. This feature can be found in the Points pulldown of all menus.

Pulldown Menu Location: COGO

Keyboard Command: T, Traverse

Prerequisite: None

Side Shots

This command allows the user to input any combination of turned angles, azimuths or bearings while remaining on an occupied point. The command prompts for an Angle-Bearing Code which defines the angle or bearing type. Codes 1 through 4 define the bearing quadrants; 1 being North-East, 2 South-East, 3 South-West, and 4 North-West. Code 5 is a north-based azimuth, 6 an angle turned to the left, 7 an angled turned to the right, 8 a deflection angle left and 9 a deflection angle right. The command plots the points calculated and stores them in the current coordinate (.CRD) file if point numbering is On. If Point Protect is turned On, *Side Shots* checks if the point numbers are already stored in the file. All points calculated radiate from the occupied point. Use the *Traverse*, *Inverse*, or *Occupied Point* commands explained previously to define the occupied and backsight points. Options allows you to select your angle entry method.

Prompts

Occupied Point ?

Pick point or point number: *screen pick a point or enter a point number*

Exit/Options/Traverse/Inverse/Enter Azimuth (ddd.mmss) <A>: *O for options*

Angle prompt angle right or azimuth only [Right/Azimuth/Prompt]? P for prompt

Exit/Options/Points/Line/Traverse/Inverse/<Angle-Bearing Code <7>: *6 Code 6 for angle turned to left.*

Pick point or point number: *pick a point*

Enter Angle (dd.mmss) <6>: *22.3524 Angle of 22 degrees, 35 minutes, 24 seconds.*

Points/<Distance>: *120.91*

Enter Vertical Angle (dd.mmss) <0.0000>: *88.2548*

These prompts only come up if you have Instrument and Rod height prompting turned on.

Instrument Height <5.000>: *5.12*

Rod-Target Height <5.120>: *press Enter*

Enter Point Description <>: *Topo Shot*

Exit/Options/Points/Line/Traverse/Inverse/<Angle-Bearing Code <6> E

For instruction on how to insert either new or existing points into the drawing, see Draw-Locate Points in the Points Commands section of the General Commands chapter. This feature can be found in the Points pulldown of all menus.

Pulldown Menu Location: COGO

Keyboard Command: sideshot, ss

Prerequisite: None

Enter-Assign Point

This command creates a point at the user-entered coordinates. The point is both stored to the current coordinate (.CRD) file and drawn on the screen. The program will prompt for the northing and easting. This routine will prompt for point number, elevation and description, depending on the settings in the *Point Defaults* command. *Point Defaults* also allows you to set the point symbol and layer. *Point Defaults* is found under the Points pulldown.

Prompts

Enter North(y): *5000*

Enter East(x): *5000*

Select/<Enter Point Elevation <0.00>: *Enter 100 for elevation, or press S and enter to select text to set elevation.*

Enter Point Description <>: *START*

N: 5000.00 E: 5000.00 Z: 0.00

Enter North(y): *press Enter to end*

For instruction on how to insert either new or existing points into the drawing, see Draw-Locate Points in the Points Commands section of the General Commands chapter. This feature can be found in the Points pulldown of all menus.

Pulldown Menu Location: COGO

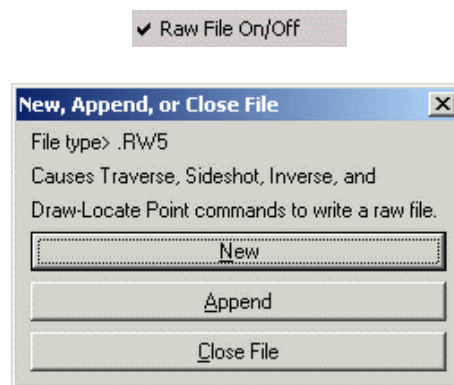
Keyboard Command: eapoint, ea

Prerequisite: None

Raw File On/Off

This menu selection toggles raw file (.RW5) creation. When this option is active, commands such as *Traverse* create entries in the current raw data (.RW5) file. If Raw File is turned on, the pulldown menu option will have a check mark character in the menu. A dialog will appear, allowing you to create a New, Append an existing, or Close the .RW5 file.

To begin this routine, select the COGO pulldown and observe the Raw File (On or Off) toggle for check. Click the command and the dialog appears.



New: Allows you to create a new raw traverse file (.RW5).

Append: Allows you to append an existing raw traverse file.

Pulldown Menu Location: COGO

Keyboard Command: openraw

Prerequisite: None

Line On/Off

This menu selection toggles line plotting on and off for the commands such as *Traverse*, *Locate by Line Bearing*, etc.. If line drawing is turned on, the pulldown menu option will have a check mark character to the left of the command.

Command: set_lonoff

Line ON

Command: set_lonoff

Line OFF



Pulldown Menu Location: COGO

Keyboard Command: set_lonoff

Prerequisite: None

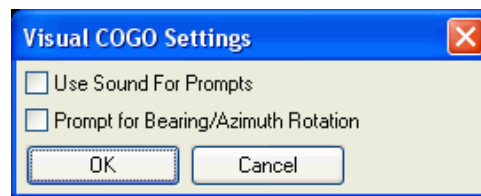
Visual COGO

This command contains COGO routines for Inverse, Occupy Point, Traverse, Side Shots, Bearing-Bearing Intersect, Bearing-Distance Intersect, Distance-Distance Intersect and Enter-Assign Point. Choosing Visual COGO from the COGO pull-down menu provides you with quick access to any of the Visual COGO routines.

A dialog for command input docks on the side of the graphic window when any of the five options from the pulldown menu are selected. Points are drawn to the screen as they are created. Linework can also be drawn. CAD and Carlson commands can be activated with the Visual COGO dialog active. This allows for quick switching between Visual COGO commands and any other command. You can also switch between Visual COGO commands within the dialog by entering the 2 character function name in any edit box. For example, from Visual COGO Inverse, you can enter SS in the point number field to switch to Side Shots.



The function names OC, EA, IN, BB, BD, DD, TR and SS are also available as function buttons across the top of the dialog. The second row of buttons are functions for zooming in/out and panning. The final button brings up Visual Cogo options. The **Use Sound** option is for whether to have sounds cues. The Prompt for **Bearing/Azimuth Rotation** adds an additional angle input in the Sideshot and Traverse functions. This angle is added to the bearing or azimuth angle input and is a way to handle North rotation where the orientation of the angles that your entering is different than the target coordinate system.



Prompts

When in Visual COGO, you will have a very different user interface from other areas of Carlson. This user-friendly screen will guide you through various COGO data entry procedures such as Inverse, Occupy Point, Traverse, Side Shots and Enter-Assign Point. You will still be able to follow the command on the command line at the bottom of your Carlson screen. Using Visual COGO is an alternative and easy method to entering in such information. The top half of the COGO pulldown menu offers you the more traditional Carlson data entry method. Your results will be the same.

IN (Inverse): This command reports the bearing/azimuth and horizontal distance between two points. The points can be entered manually or by picking from a point list by picking on the list button. The resulting report of bearing/azimuth is dependent upon the Angle Mode setting in the drawing setup options.

OC (Occupy Point): Used to specify the point number of the instrument setup point. The point can be specified by manually entering in the point number in the Occupied Point data field, or by selecting the List button and choosing from the list of points contained in the coordinate file.

Backsight Method can be either by Point Number or by Azimuth. If angle right/left or deflection right/left is being used for traverse or sideshot entry, a backsight method must be specified. If using Bearing or Azimuth entry, no backsight method is required. The Backsight Point can be specified by manually entering in the point number in the Backsight Point data field, or by selecting the List button and choosing from the list of points contained in the coordinate file.

Instrument Height: Use this field to set the height of the instrument.

Accept (F2): Selecting this button or pressing the F2 function key accepts the data entered in the fields above. After accepting the data, until changed, the points specified will remain the occupied and backsight points. If the dialog is exited without Accepting the settings the Occupied and Backsight points will have to be specified when the OC dialog is revisited.

Exit: Cancels the command

TR (Traverse): This command allows data entry using any combination of turned angles, deflections, azimuths or bearings to define a traverse or figure. This command always occupies, moves up to, the last point it calculated and backsights the point before, or the previous occupied point.

Point Number: This is the number of the point to be created.

Rod Height: Height of target to be located.

The horizontal angle component can be input in various formats. The format label will change with the option chosen. Choose the format by selecting the down arrow and picking from the list.

NE=Northeast
SE=Southeast
SW=Southwest
NW=Northwest
AZ=Azimuth
AL=Angle Left
AR=Angle Right
DL=Deflection Left
DR=Deflection Right

The vertical angle component can be input in various ways (the format label will change with the option chosen). Choose the format by selecting the down arrow and picking from the list.

VA=Vertical Angle. Zero (0) degrees is level.

ZE=Zenith Angle. Ninety (90) degrees is level.

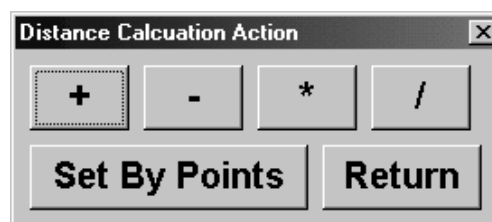
DZ=Elevation Difference. The difference in elevation either plus or minus from the instrument setup to the target.

The distance component can be entered as either Slope or Horizontal Distance. Choose the format by selecting the down arrow and picking from the list.

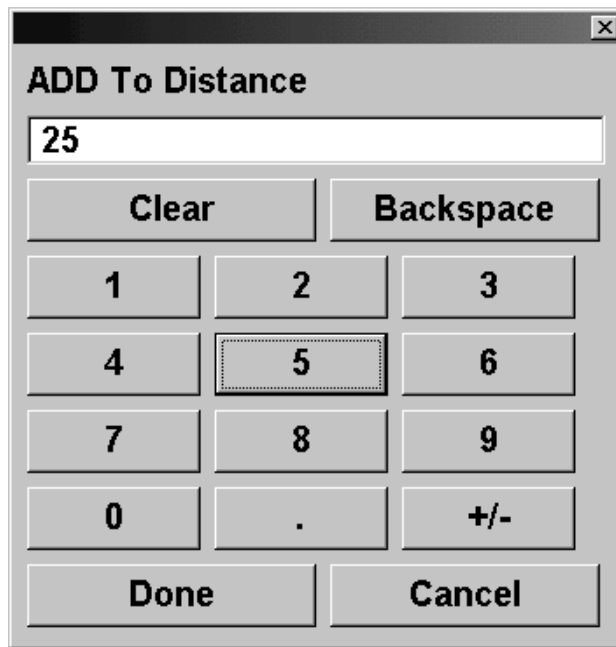
SD=Slope Distance

HD=Horizontal Distance

Distance can be defined by Point Numbers by selecting the calculator button to the far right of Angle Right and Slope Distance.

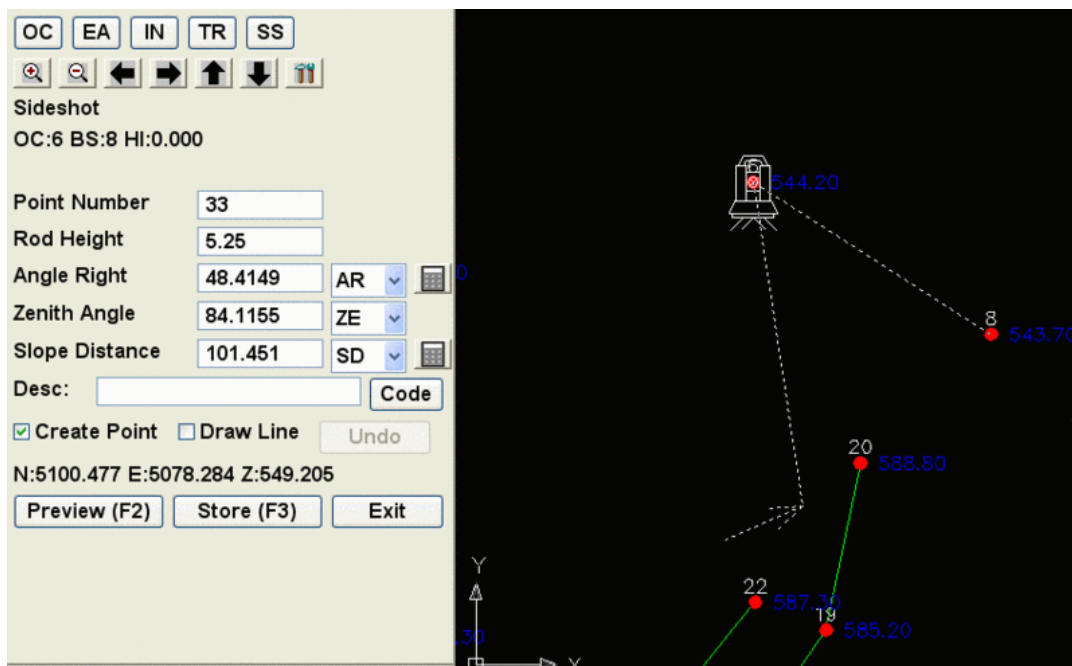


Additional mathematical calculations of addition, subtraction, multiplication and division can be performed on the input distance by selecting the appropriate button and filling out the function dialog.



For example to add 25 to the Slope distance value on the traverse dialog, select the + button, enter 25 and then select Done. The same steps apply to any of the other mathematical functions.

Side Shots: This command works in the same way as the traverse command. All the available options contained in the traverse command are available in this command. The only difference in the commands is that the side shot command does not move the setup point to last shot input. Refer to the traverse command for further details.



Desc: Defines the description for the point to be created.

Create Point: Option whether to store the point to the CRD file and draw a point.

Draw Line: Option to draw line to the traverse point.

Preview (F2): Previews the traverse point location, without storing the point to the coordinate file.

Store (F3): Stores the traverse point based upon the entered data to the coordinate file.

Undo: After storing the point, the point can be deleted from the screen and coordinate file by selecting the undo button.

Exit: Exits the Visual COGO command and closes the dialog box.

EA (Enter Assign): Use this function to enter and assign coordinate values for new and existing points.

OC EA IN TR SS

Zoom In Zoom Out Left Pan Right Pan Up Pan Down Pan Tool

Snap None

Enter-Assign Point

Point Number: 1

Northing: 5000

Easting: 5000

Elev: 100

Desc: START

Screen Pick Point

Store (F2) Undo Exit

BB (Bearing-Bearing Intersect): Enter two base points along with the angles from each and the routine calculates the bearing-bearing intersection point. The dialog has fields for the output point number and description.

BD (Bearing-Distance Intersect): Enter two base points along with the angle from the first and distance from the second and the routine calculates the bearing-distance intersection point. The dialog has fields for the output point number and description.

DD (Distance-Distance Intersect): Enter two base points along with the distances from each and the routine calculates the distance-distance intersection point. For the two possible solutions of the intersecting circles, the intersection clockwise from the first point is used. The dialog has fields for the output point number and description.

Zooming and panning functions are also available from the Visual COGO dialog box:

Plus (+) magnifier: Zooms the display window in. Use to view an area up close.

Minus (-) magnifier: Zooms the display window out. This shows more of the drawing.

Left arrow: Pans the display window to the left.

Right arrow: Pans the display window to the right.

Up arrow: Pans the display window up.

Down arrow: Pans the display window down.

Pulldown Menu Location: COGO

Keyboard Command: vcogo_inverse, vcogo_setup, vcogo_traverse, vcogo_sideshot, vcogo_store

Prerequisite: Coordinate file to process

Locate by Line Bearing

This command calculates and plots a line (if the Line On/Off is set to Line On) and point from an occupied point. The bearing can be defined by picking two points, selecting a line, inputting two point numbers, or typing in a bearing or azimuth. The command always occupies the last point calculated.

Prompts

Press [Enter] to use preview point/or select occupied point.

Pick point or point number: 14

PointNo.	North(Y)	East(X)	Elev(Z)	Description
14	4869.06	4390.31	0.00	

Pick points that define bearing.

Define Bearing by, Line/Bearing/Numbers/<pick 1st point>: B

At this prompt the default is to pick the first point that defines the bearing. If you pick a point, you are then prompted for a second point. You can input B to type in a bearing or azimuth or L to select a line or polyline that defines the bearing, or N to input two point numbers that define the bearing.

[A]zimuth/<Bearing (Qdd.mmss)>: A

Azimuth (ddd.mmss): 45.2349

Number inverse/<Distance>: 188.27

Enter Vertical Angle (dd.mmss) <0.0000>: press Enter

The horizontal distance is given.

Enter Point Description <stk>: press Enter

The coordinates are given.

Pulldown Menu Location: COGO > Locate by Bearing-Ang

Keyboard Command: locbrg, lb

Prerequisite: None

Locate by Turned Angle

This command locates a point by turned angle and distance.

Prompts

Define occupied & backsight points by [L]ine or [P]oints <P>: L

Select Line or Polyline near end point that defines occupied point: select line

Occupied point: (4078.44 4610.89 0.0)

Backsight point: (4390.31 4869.06 0.0)

Enter Angle (ddd.mmss) <45.2349>: 22.5632

Pick or Type Distance <188.27>: 40.32

Enter Vertical Angle (dd.mmss) <0.0000>: hit Enter

Pulldown Menu Location: COGO > Locate by Bearing-Ang

Keyboard Command: turnang2, ta

Prerequisite: None

Locate by Azimuth

This command locates points by azimuth and distance. The AutoCAD text screen provides the horizontal distance and coordinates.

Prompts

[Enter] to use preview point/ or Select occupied point ?

Pick point/<point Number>: _endp of (pick a point)

Enter Azimuth (ddd.mmss) <22.5632>: 277.1259

Enter or pick Distance <40.32>: 104.39

Enter Vertical Angle (dd.mmss) <0.0000>: Enter

Pulldown Menu Location: COGO > Locate by Bearing-Ang

Keyboard Command: locazi2, az

Prerequisite: None

Locate by Bearing

This command locates points by bearing and distance. Additionally, the AutoCAD text screen provides the horizontal distance and coordinates.

Prompts

[Enter] to use preview point or Select occupied point ?

Pick point/<point Number>: 24

PointNo.	Northing (Y)	Easting (X)	Elev (Z)	Description
24	4922.37	4544.81	0.00	

Enter Bearing (Qdd.mmss) <277.1259>: 435.2317

Enter or pick Distance <104.39>: 200

Enter Vertical Angle (dd.mmss) <0.0000>: Enter

Pulldown Menu Location: COGO > Locate by Bearing-Ang

Keyboard Command: locbrg2, lg

Prerequisite: None

Locate by Delta

This command locates points by specified delta x, y, z from a reference point. The point style and whether to prompt for a description is set in *Point Defaults*.

Prompts

[Enter] to use preview point/ or Select occupied point ?

Pick point/<point Number>: *pick a point*

Delta Northing (dy): 23.45

Delta Easting (dx): 12.34

Delta Elevation (dz) <0.0>: *press Enter*

Enter Point Description <>: *press Enter*

N: 11687.04 E: 10095.31 Z: 0.00

Delta Northing (Enter to end): *press Enter*

Pulldown Menu Location: COGO > Locate by Angle-Distance

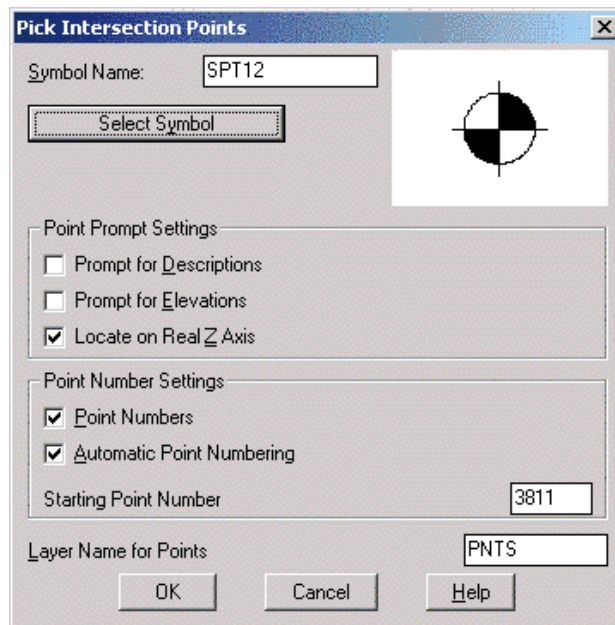
Keyboard Command: locdelta

Prerequisite: None

Pick Intersection Points

This command locates points at screen picked intersections. The object snap mode is set to intersection. This routine is similar to the Locate Point command, with an additional check that makes sure there is an intersection at

the picked point. If there is not an intersection at the picked point, then no point is created.



Prompts

Pick Intersections Points dialog

APParent intersection on [<Yes>/No]: *Y*

This first prompt is very important. Apparent Intersection snaps to the apparent intersection of two objects (arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, spline, or xline) that do not intersect in 3D space, but may appear to intersect in the current view. This allows you to locate a point at the theoretical intersection of two 3D entities. You should answer No to this prompt if you want to ignore theoretical 3D intersections.

[app on] Pick intersection Point: *pick a point*

[app on] Pick intersection Point: *press Enter to end*

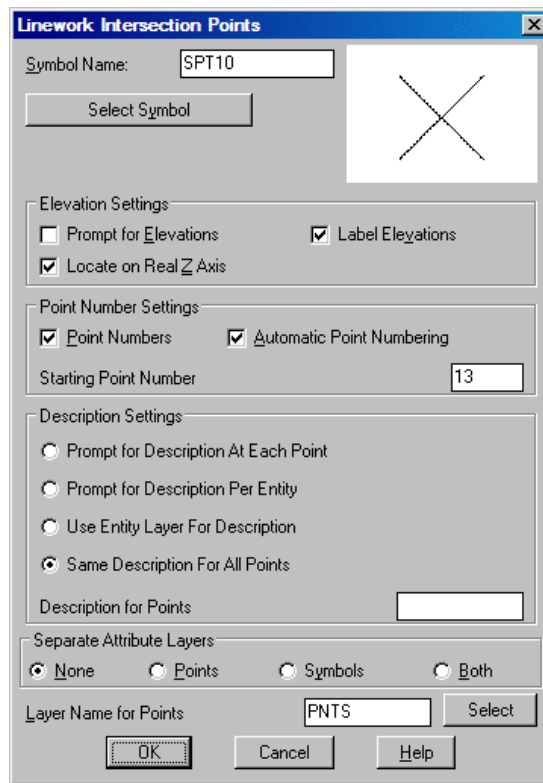
Pulldown Menu Location: COGO > Locate at Intersect

Keyboard Command: pickint

Prerequisite: Intersection of two entities

Linework Intersection Points

This command is used to create points at all of the intersections between selected linework entities.



Prompts

Select lines and polylines to process.

Select objects: Specify opposite corner: *pick objects*

Pulldown Menu Location: COGO > Locate at Intersect

Keyboard Command: ADDINTPTS

Prerequisite: None

Bearing-Bearing Intersect

This command locates a point at the intersection of two lines. The lines can be defined by picking two points, selecting a line or typing in a bearing. After the lines are defined a point symbol is located at the point of intersection. When a grid projection is defined in Drawing Setup, there is a prompt for whether to use the mean, forward or back geodetic bearing.

Prompts

[Enter] to use preview point or select 1st Base point ?

Options/<Pick point or point number>: *press Enter*

Define 1st angle by (Line/Points?Right/Azimuth/Bearing) <Bearing>: *L*

Select Line or Polyline that Defines 1st Bearing: *select*

Enter 1st Offset Distance <0.0>: *press Enter*

2nd Base point?

Pick point or point number: *pick a point*

Define 2nd angle by (Line/Points/Right/Azimuth/Bearing) <Line>: *P*

[Enter] to use preview point/or pick 1st point that defines 2nd bearing.

Pick point or Point number: *pick point*

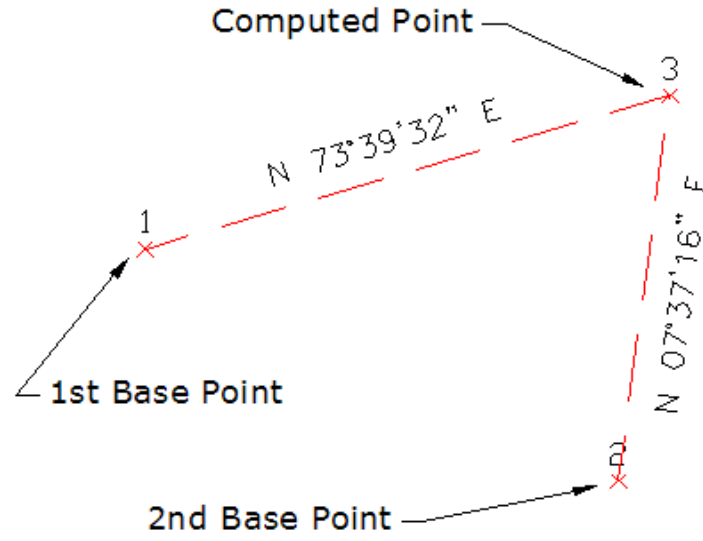
2nd point that defines 2nd bearing ?

Pick point or Point number: *pick a point*

Enter 2nd Offset Distance <0.0>: *press Enter*

Enter/<Select text of elevation>: *select*

The point is then located at the computed point of intersection.



Bearing-Bearing Intersect example

Pulldown Menu Location: COGO > Locate at Intersect

Keyboard Command: bb

Prerequisite: None

Distance-Distance Intersect

This command creates a point at the distance-distance intersection from two base points. The program prompts for two distances and two base points. The two possible intersections (A,B) are shown on the screen. You can either pick near the desired intersection or type in the letter A or B. The A intersection is clockwise from the first point. The Options choice brings up a small dialog that allows you to be prompted for angle method or for offsets, or both. When a grid projection for the drawing is defined under Drawing Setup, the program will prompt for whether to use the grid or geodetic distances.

Prompts

Select 1st base point

Options/<Pick point or point number>: 1

Points/<1st distance>: 46.72

Enter 1st Offset Distance <0.0>

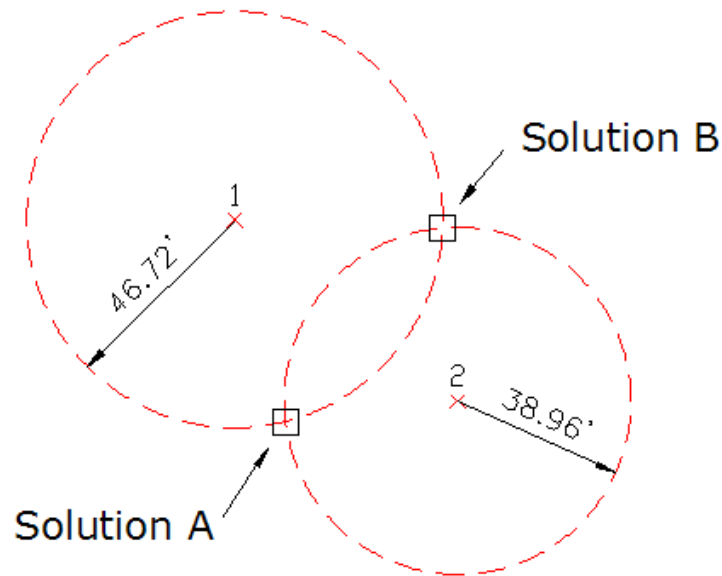
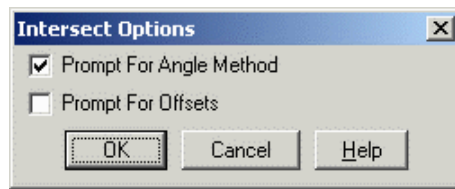
Select 2nd base point

Pick point or point number: 2

Points/<2nd distance>: 38.96

Enter 2nd Offset Distance <0.0>: *press Enter*

Pick near solution or Enter [A] or [B]: *pick a point*



Pulldown Menu Location: COGO > Locate at Intersect

Keyboard Command: DD

Prerequisite: None

Bearing-Distance Intersect

The Bearing-Distance Intersect command prompts the user for a base point from which the known bearing intersects. It then defines the bearing by one of three methods. The bearing can be defined by picking two points, selecting a line with the same bearing or by typing in the bearing in the form of Qdd.mmss (similar to the *Locate by Bearing* command). Next the user is prompted for a base point from which the known distance radiates. After entering the known distance a circle is drawn radiating from the selected base point, and a line defined by the bearing is extended to intersect the circle. The user then picks the correct point for the solution desired and a point symbol is located at the selected intersection. The command then erases the temporary circle and line. The Options choice allows you to be prompted for angle method or for offsets, or both.

When a grid projection for the drawing is defined under Drawing Setup, this command will prompt for whether to use the grid bearing or geodetic mean, forward or back bearing. Also, the program will prompt for whether to use the grid or geodetic distance.

Prompts

[Enter] to use preview point or select known Bearing base point

Options/Pick point or point number: *pick point*

Define 1st bearing by (Line/Points/Azimuth/Bearing)<Bearing>: *l*

Select Line or Polyline that Defines Bearing: *pick entity*

Enter 1st Offset Distance <0.0>: *press Enter*

Known distance base point.

Pick point or point number: *pick point*

Points/<Enter Distance>: 40.41

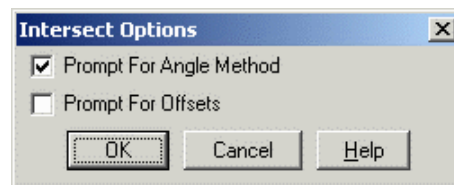
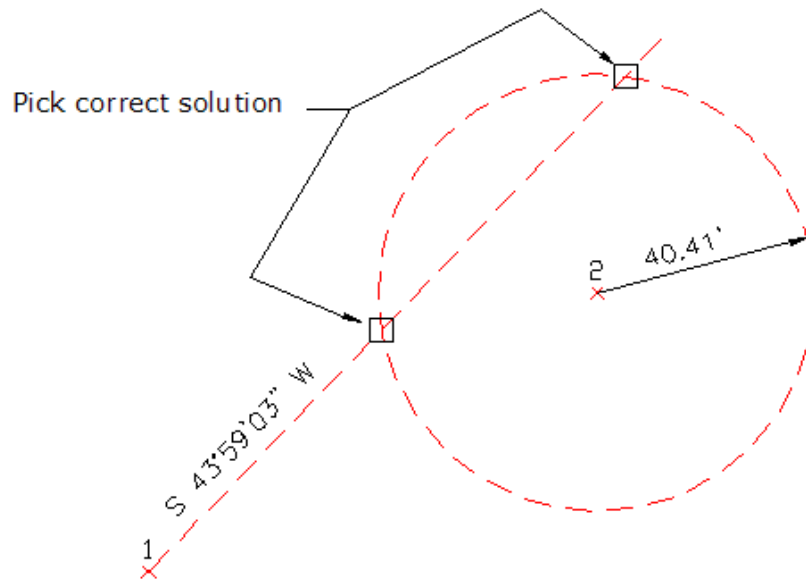
Enter 2nd Offset Distance <0.0>: press Enter

[int on] Pick Intersection point ([Enter] to cancel): pick point

Enter Point Number <55>: press Enter This prompt appears only if Automatic Point Numbering is turned off.

See Point Defaults

Enter Point Symbol Number <4>: press Enter This prompt appears only if point symbol prompting is turned on. Symbol number 4 is located at the computed coordinate and labeled point number 55.



When Options (O) is selected

Pulldown Menu Location: COGO > Locate at Intersect

Keyboard Command: bdint

Prerequisite: None

Perpendicular Intersect

This command creates a point at the perpendicular intersection from an offset point to a line. This command prompts for a base point from which the known bearing intersects. Then the bearing is defined by typing in the bearing in the form of Qdd.mmss (similar to the *Locate by Bearing* command). Next the user is prompted for an offset point.

Prompts

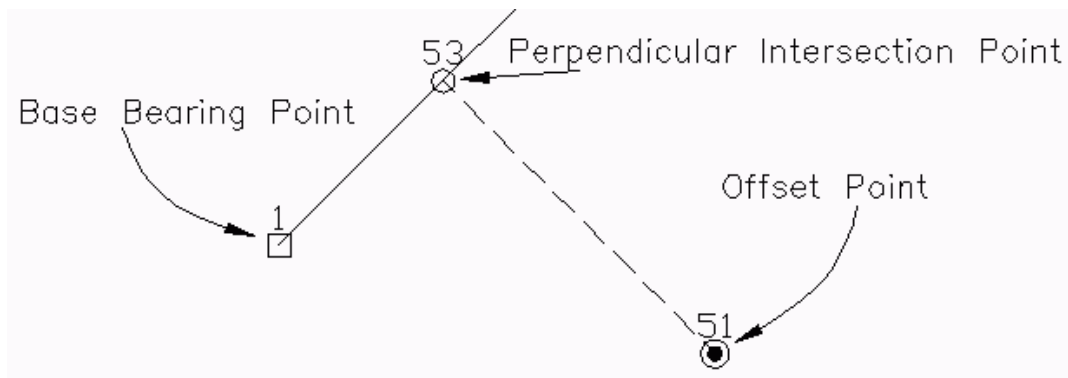
[Enter] to use preview point or select known Bearing base point

Pick point or point number: 1

Bearing (Qdd.mmss): 145.0000

Offset point.

Pick point or point number: 51



Pulldown Menu Location: COGO > Locate at Intersect

Keyboard Command: perpint

Prerequisite: None

Tangent Intersect

This command creates a point at the tangential intersection from a point to another point and distance. This command prompts for a base point and then the distance. Next there is a prompt for the second point. Since there are two possible tangential solutions on the circle from the base point, the program displays the two possible solutions and prompts for which one to use. To choose the solution, pick close to the solution point.

Prompts

[Enter] to use preview point or select known base point

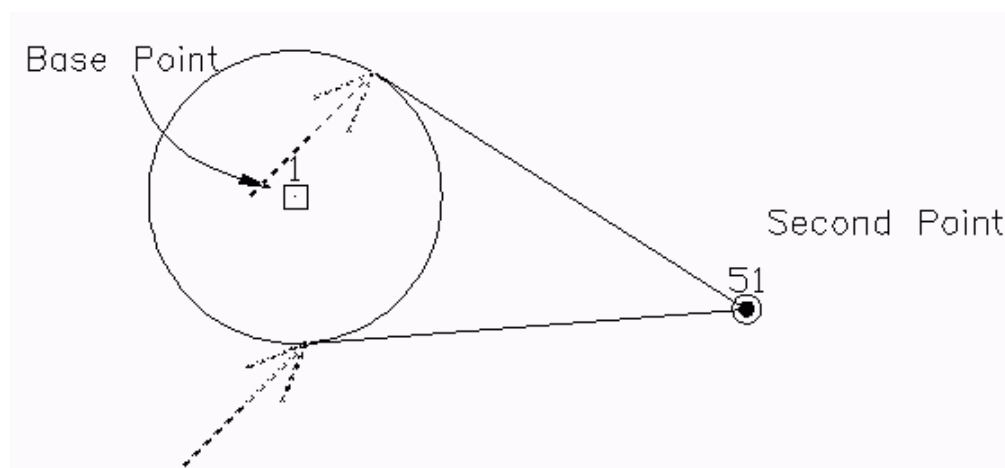
Pick point or point number: 1

Points/<Enter Distance>: 25

Second point.

Pick point or point number: 51

Pick tangent point solution: *pick a point*



Pulldown Menu Location: COGO > Locate at Intersect

Keyboard Command: tangint

Prerequisite: None

2 Point - 2 Point Intersect

This command is similar to Bearing-Bearing Intersect except that in this command bearings are defined by specifying two point numbers. In the example shown below, the first two points specified are 3838 and 3839, the second pair are 3841 and 3840. Point 3842 is located at the intersection.

Prompts

Specify 1st base point.

Pick point or point number: 3838

Specify 2nd point that defines 1st direction.

Pick point or point number: 3839

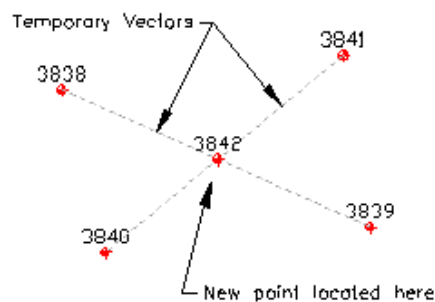
Specify 2nd base point.

Pick point or point number: 3841

Specify 2nd point that defines 2nd direction.

Pick point or point number: 3840

Select/<Enter Point Elevation>: *Enter value*



Pulldown Menu Location: COGO > Locate at Intersect

Keyboard Command: bbint2

Prerequisite: None

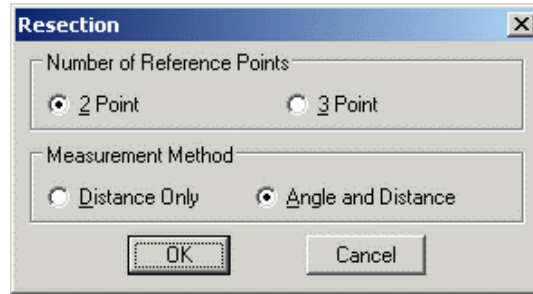
Resection

This command calculates point coordinates given the angle and distance from two or three reference points. The Z coordinate can also be calculated in addition to the X,Y. If you only need the 2D solution, then enter the instrument and rod heights as 0.0, the zenith angle as 90 and the distance as the horizontal distance. The reference points are specified by point number. These reference points need to be stored in the current coordinate (.CRD) file before running this command.

After entering the reference point, there is a dialog to enter the horizontal angle, zenith angle and slope distance. The horizontal angle is the horizontal azimuth or angle right from the unknown point to the reference point. In the example, the backsight azimuth is 0 (due north), but this is not a requirement since the backsight can be any angle. The program calculates the coordinate by averaging the distance-distance and angle-angle solutions. Since there is redundant data, the final calculated coordinate will differ slightly from the individual measurements. For example, in a 3-point resection, there are two different distance-distance solutions (between the first-second point and between the second-third points). The program reports the difference between the final coordinate and the

individual solutions as the residuals which act as an indicator whether the data is good. High residuals suggest a problem with the input data. In the dialog that displays the final coordinates and residuals, there is a button to store the coordinates to the current coordinate (.CRD) file with a specified point number.

In the first Resection dialog box, you can choose to use two or three reference points.



In the second Resection dialog box, you assign the reference point.



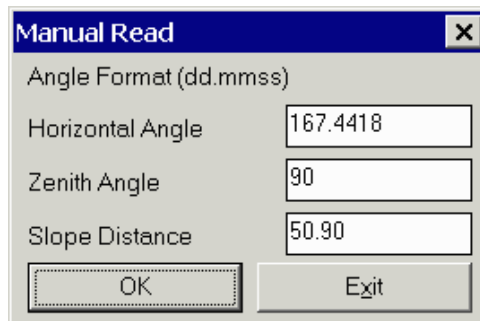
Point: You must enter the point number of your reference point. These reference points need to be stored in the current coordinate file before you run this command.

Inst. Height: You must enter the instrument height.

Target Height: You must enter the target height.

If you need only the 2D solution, then enter the instrument and target heights as 0.0.

In the Manual Read dialog box, you must specify parameters for the calculation.



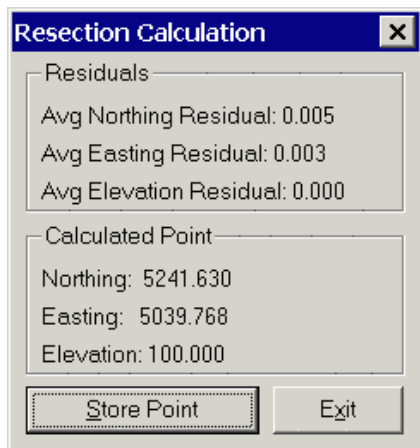
Horizontal Angle: You must enter a horizontal angle from the resection to the reference points. The horizontal angle is the horizontal azimuth, or angle right, from the unknown point to the reference point.

Zenith Angle: You must enter a zenith angle. For a 2D solution, set the zenith angle to 90 degrees.

Slope Distance: You must enter a slope distance from the reference points to the resection.

You are prompted for additional reference points and parameters.

The Resection Calculation dialog box that displays the final coordinates and residuals. You can select the option to store the coordinates in the current coordinate file with a specified point number.



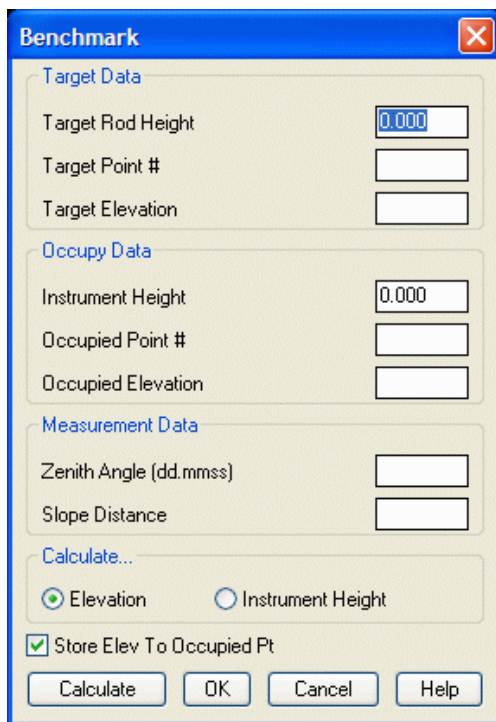
Pulldown Menu Location: COGO > Locate at Intersect

Keyboard Command: cresection

Prerequisite: Two or three reference points

Benchmark

This command is similar to the data collector routine, where a measurement with a total station is taken from an unknown elevation to a known elevation foresight. The unknown elevation of the occupied point is then calculated based on the measurement. Either the Occupied Elevation or the Instrument Height can be calculated. Note that a check box is located at the bottom of the dialog box to "Store Elev To Occupied Pt". This will automatically change the elevation of the occupied point.



Prompts

Coordinate File to Process dialog If required, this dialog will appear and you must select a file.

Benchmark dialog *Fill in variables, click Calculate*

Pulldown Menu Location: COGO

Keyboard Command: benchmark

Prerequisite: None

Numeric Pad COGO

Using only the keys on the numeric pad, this command does several COGO commands. The program cycles through six prompts. Only respond to the prompts that apply and the program will perform the correct action. The prompts are: First point? First angle? First distance? Second point? Second angle? Second distance?

To *inverse*, give a first point and second point.

To *traverse*, give a first point, first angle and first distance.

To do *bearing-bearing intersect*, give a first point, first angle, second point and second angle.

To do *bearing-distance intersect*, give a first point, first angle, second point, and second distance. Or give a first point, first distance, second point, and second angle. The point is calculated at the closer intersection.

To do *distance-distance intersect*, give a first point, first distance, second point, and second distance. The point is calculated at the first intersection going clockwise from the first point's distance circle.

Points can be screen picked or entered as point numbers that reference the current coordinate file. The last point is used as a default when you press Enter at the prompt for the first point. Which point is being used is indicated by a ghost arrow pointer.

Angles can be specified by picking two points or entering an angle code which begins with a single digit code followed by the degrees and the minutes and seconds after a decimal point. The digits codes are (1 - Northeast, 2 - Southeast, 3 - Southwest, 4 - Northwest, 5 - Azimuth). For example, Northwest 50d10'2" would be *450.102*.

Distances can be specified by picking two points or entering the distance value.

Prompts

Enter coords/Quit/<Pick 1st point or point number>: 5

Pick or Type 1st Direction by 2 Points: *145.0135* (Northeast 45d1'35")

Pick or Type 1st Distance by 2 Points: *50.0*

A point is created from the values for this traverse. The prompts for the second point don't appear because all the information for this action is entered.

Enter coords/Quit/<Pick 1st point or point number>: *press Enter* to use the point created by the traverse.

Pick or Type 1st Direction by 2 Points: *50.0*

Enter coords/Quit/<Pick 1st point or point number>: 4

Enter/Pick 2nd Direction by 2 Points: *press Enter*

Enter/Pick 2nd Distance by 2 Points: *75.0*

This creates a point by distance-distance intersect.

Enter coords/Quit/<Pick 1st point or point number>: *Q*

Pulldown Menu Location: COGO

Keyboard Command: ccogo

Prerequisite: None

Line by Angle-Distance

This command draws a line from an occupied point at a given angle and distance, where the angle format supports the standard 1-9 angle-bearing codes. It holds the current occupied point and calculates a line by angle-distance to each entered point. As for the angle formats, the Options choice allows for angle right, azimuth only or prompt entry (Right/Azimuth/Prompt) methods. The Prompt method allows you to enter the 1-9 angle-bearing codes.

Prompts

Occupied Point ?

Pick point or point number: *pick point*

Exit/Options/SideShot/Inverse/Enter Azimuth (ddd.mmss) <90.0000>: *112.3024*

Points/Distance>: *290*

Exit/Options/SideShot/Inverse/Enter Azimuth (ddd.mmss) <112.3024>: *O*

Angle prompt angle right or azimuth only [Right/Azimuth/Prompt]? *R*

Exit/Options/SideShot/Inverse/Enter Angle (dd.mmss) <112.3024>: *88*

Points/Distance>: *300*

Exit/Options/SideShot/Inverse/Enter Angle (dd.mmss) <88>: *O*

Angle prompt angle right or azimuth only [Right/Azimuth/Prompt]? *P*

Exit/Options/Points/Angle-Bearing Code <7>: *Enter*

Enter Angle (dd.mmss) <88>: *31.4340*

Points/Distance>: *419*

Exit/Options/Points/Angle-Bearing Code <7>: *E*

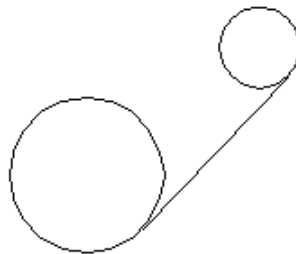
Pulldown Menu Location: COGO

Keyboard Command: travline

Prerequisite: None

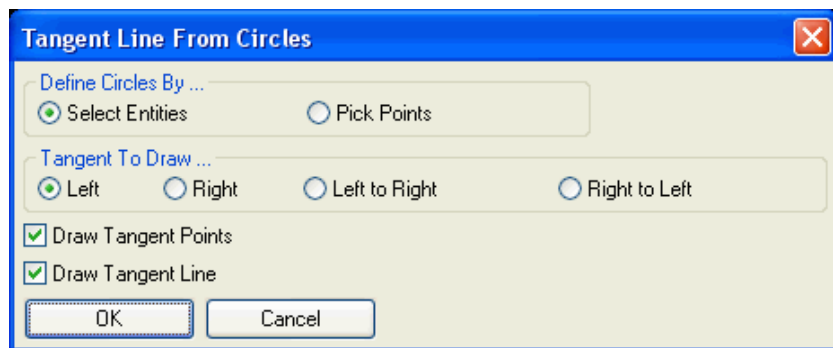
Tangent Line from Circles

This command draws a line that is tangent to two circles or arcs. The circles can be defined either by picking the radius point and entering the radius, or by selecting circle or arc entities. The tangent line can be drawn to either outside on the left or right side, or across the middle between the circles from left to right or from right to left. The line and the circles are drawn in the current layer. There is also an option to create two points at the ends of the tangent line.



Tangent line Between two circles

Prompts



Tangent Line From Circles dialog

Pick center point of first circle: *pick a point*

Pick first radius: 25

Pick center point of second circle: *pick a point*

Pick second radius: 35

Pulldown Menu Location: COGO

Keyboard Command: linecircle

Prerequisite: None

Geodetic Traverse

This command creates points by traversing by bearing and distance. The program starts by prompting for a starting point. Then the bearing is entered as either grid bearing or geodetic bearing for mean, forward or backward. Then the distance is entered either as grid distance or geodetic distance at mean elevation or zero elevation, and the distance units can be either feet, meters or chains. The resulting point is then drawn using the point settings from the Point Defaults command. Before running this command, the grid projection for the drawing must be defined in the Drawing Setup command.

Prompts

Select 1st line to split: *pick a line*

Select 2nd line to split: *pick a line*

Pulldown Menu Location: COGO > Geodetic

Prerequisite: None

Keyboard Command: geotrav

Geodetic Double Proportion Line Division

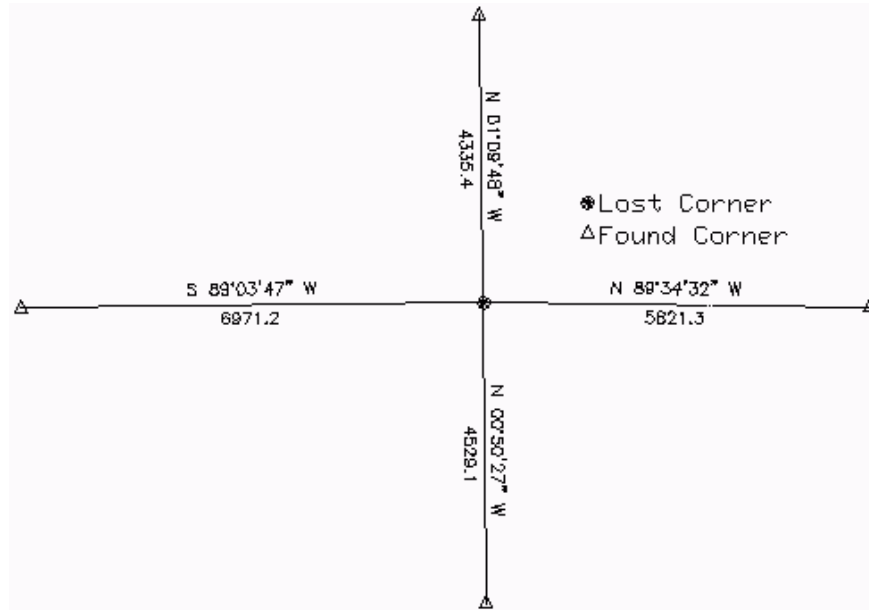
This command is used to restore a lost corner from new measurements between four known corners with two measurements each on intersecting meridional and latitudinal lines. The program prompts for the four known corner points to establish the retracement. Plus the program prompts for the four sets of record bearings and distances from the known points to the lost corner. The report then shows the calculated point along with the input data.

Double Proportion

Retrace Point	Northing	Easting
1st North-South	243758.936	471259.132
2nd North-South	252622.467	471144.220
1st East-West	248221.216	477033.581
2nd East-West	248208.591	464242.295
Record	Bearing	Distance
1st North-South	N 00°05'03''	W 4529.100
2nd North-South	N 01°09'48''	W 4335.400

1st East-West N 89°34'32" W 5821.300
 2nd East-West S 89°03'47" W 6971.200

 Northing Easting
 New Point 248287.938 471213.134



Prompts

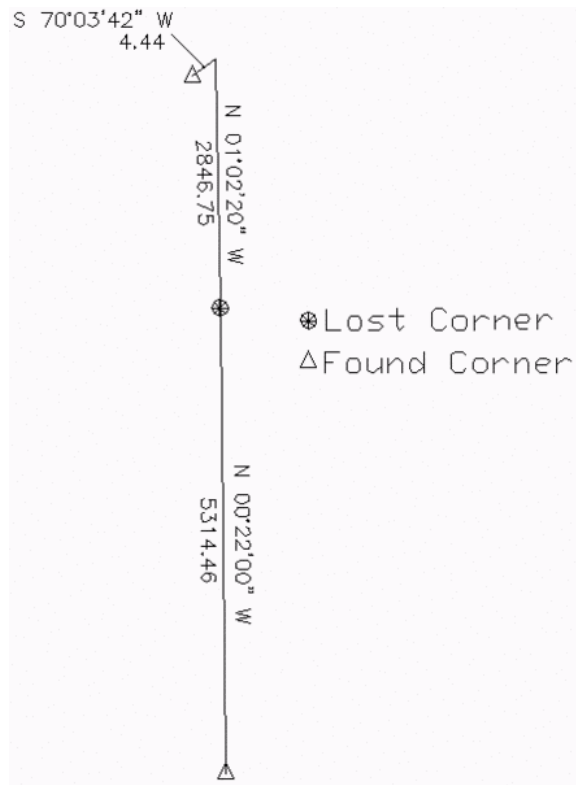
Pick First point for North-South line: *pick a point*
Pick Second point for North-South line: *pick a point*
Mean Bearing from Record (Qdd.mmss): 40.5027
Enter Record Distance [<Meters>/Feet/Chains]: 4529.1
Mean Bearing from Record (Qdd.mmss): 41.0948
Enter Record Distance [<Meters>/Feet/Chains]: 4335.4
Pick First point for East-West line: *pick a point*
Pick Second point for East-West line: *pick a point*
Mean Bearing from Record (Qdd.mmss): 489.3432
Enter Record Distance [<Meters>/Feet/Chains]: 5821.3
Mean Bearing from Record (Qdd.mmss): 389.0347
Enter Record Distance [<Meters>/Feet/Chains]: 6971.2

Pulldown Menu Location: COGO > Geodetic
Prerequisite: Grid projection under Drawing Setup
Keyboard Command: geodprop

Irregular Boundary Adjustment

This command adjusts angle/distance courses between two control points. This adjustment applies to boundaries that are not established as straight lines and are termed "irregular". A modified form of single proportionate measurement is used by this method to restore the lost corners. To apply the geodetic lengths and angles with this adjustment, the grid projection must be set under Drawing Setup before running this command.

For input, this command takes reference coordinates for the starting and ending points either by screen pick or by point number. Then the record angles and distances between these two reference points are entered. The program reports the input data, the recalculated points and the adjusted angles and distances.



Prompts

Pick Starting point or point number: *pick point*
Pick Ending point or point number: *pick point*
Mean Bearing from Record (Qdd.mmss): 40.2200
Enter Record Distance [<Meters>/Feet/Chains]: 5314.46
Mean Bearing from Record (Qdd.mmss): 41.0220
Enter Record Distance [<Meters>/Feet/Chains]: 2846.75
Enter another record [Yes/<No>]? *press Enter*
Draw adjusted lines [<Yes>/No]? *press Enter*

Irregular Boundary Report

Retrace Point

Northing Easting
222637.518 477292.438
230796.636 477202.638

Record

Bearing Distance
N 00°22'00" W 5314.460
N 01°02'20" W 2846.750

New Point

Northing Easting
227950.882 477255.709

Adjusted

Bearing Distance
N 00°23'46" W 5313.490
N 01°04'06" W 2846.248

Pulldown Menu Location: COGO > Geodetic
Prerequisite: Two control points and record courses
Keyboard Command: geoirrb

Geodetic Single Proportion Line Division

This command breaks a line into two lines that have the same mean geodetic angle. The length of the first new line is proportional to the specified part distance relative to the total distance. Before running this command, the grid projection must be set under Drawing Setup.

Prompts

Select a line near beginning point: *pick a line*
Enter Record Part Distance [Meters/<Feet>/Chains]: 500
Enter Record Total Distance [Meters/<Feet>/Chains]: 2000

Pulldown Menu Location: COGO > Geodetic
Prerequisite: A line
Keyboard Command: geosprop

Geodetic Double Break

This command breaks two crossing lines at their intersection such that the two segments of the first line have the same geodetic mean bearing and the two segments of the second line have the same geodetic mean bearing. Before running this command, the grid projection must be set under Drawing Setup.

Prompts

Select 1st line to split: *pick a line*
Select 2nd line to split: *pick a line*
Pulldown Menu Location: COGO > Geodetic
Prerequisite: Two crossing lines
Keyboard Command: geodbk

Geodetic Middle Break

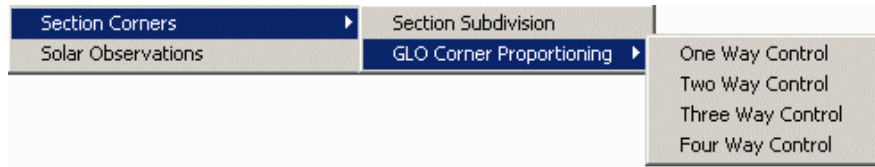
This command breaks a line into two lines that have the same mean geodetic angle and same geodetic length. Before running this command, the grid projection must be set under Drawing Setup.

Prompts

Select line to split at geodetic midpoint: *pick a line*
Pulldown Menu Location: COGO > Geodetic
Prerequisite: A line
Keyboard Command: geomid

GLO Corner Proportioning

The GLO Corner Proportioning commands calculate section and 1/4 section corners by one, two, three or four way control. GLO plats are the official plats of the U.S. Government Land Office (GLO) executed after July 1946. The Department of the Interior, Bureau of Land Management (BLM) is the successor agency to the GLO.



Section Subdivision

This command calculates and stores unknown Section corners that can be calculated given the data specified. The calculated points will be plotted on the screen and saved to the coordinate file.

A screenshot of the 'Section Subdivision' dialog box. It contains several input fields and groups. The 'Section Location' group has fields for Section (3), Township (67), and Range (18). The 'Specify Field Located Corners' group has radio buttons for 'Section & Quarter Corners' (selected) and '1/16th Corners'. The 'Section Linework' group has a checked 'Draw Section Lines' box and dropdown menus for 'Section Lines' (V-PROP-SECT), 'Quarter Section Lines' (V-PROP-QTRS), and 'Sixteenth Section Lines' (V-PROP-SXTS). The 'Government Chainages' group has input fields for 'A to H' (39.15) and 'C to D' (39.98). A grid of 12 input fields is arranged in a 3x4 grid, labeled C, B, A, D, H, E, F, G. The top row contains 5744, 5745, and 4386. The bottom row contains 5764, 5765, and 5766. 'OK' and 'Cancel' buttons are at the bottom.

Section Location Group: Input the required Section number, Township and Range.

Specify Field Located Corners Group: Indicate whether Section & Quarter Corners were located in the field or if 1/16th corners were located in the field.

Section Linework Group: Indicate whether the calculated Section lines should be drawn and indicate the desired layer(s) for the Section Lines, Quarter Section Lines and Sixteenth Section Lines.

Inputs A - H: Specify the point IDs of corners that have been located in the field.

Government Chainages Group: Specify the government chainages as documented/required.

Note:

- The choices in the **Specify Field Located Corners** section of the dialog merely make it more convenient to enter the Section & Quarter corners and the 1/16th corners. This choice allows you to enter the corners in order by just typing the point ID of a corner then just press Enter to move to the next corner. You may enter any type of corner located in the field by changing the types of corners selection in the **Specify Field Located Corners** section.

For each calculated corner, the **Saving Point** dialog box will be displayed. Depending on the Point Default settings, this dialog may allow you to accept or change the default point ID. Also, depending on the Point Default settings, the description and elevation may also be changed or accepted.

Pulldown Menu Location(s): COGO > Section Corners

Keyboard Command: cg_section_subd

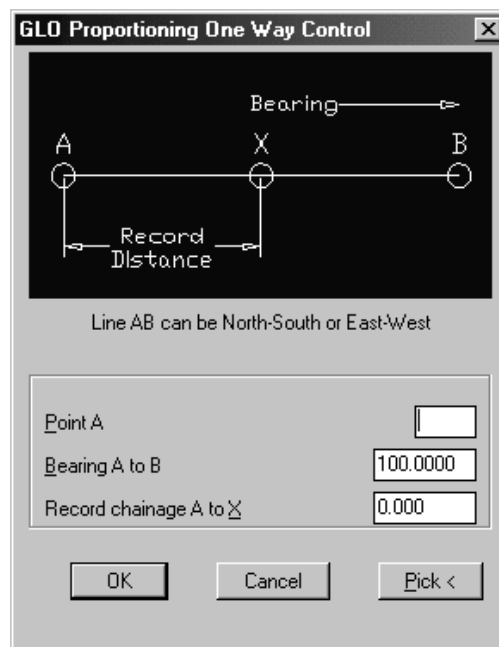
Prerequisite: Coordinate File with at least eight points

One Way Control

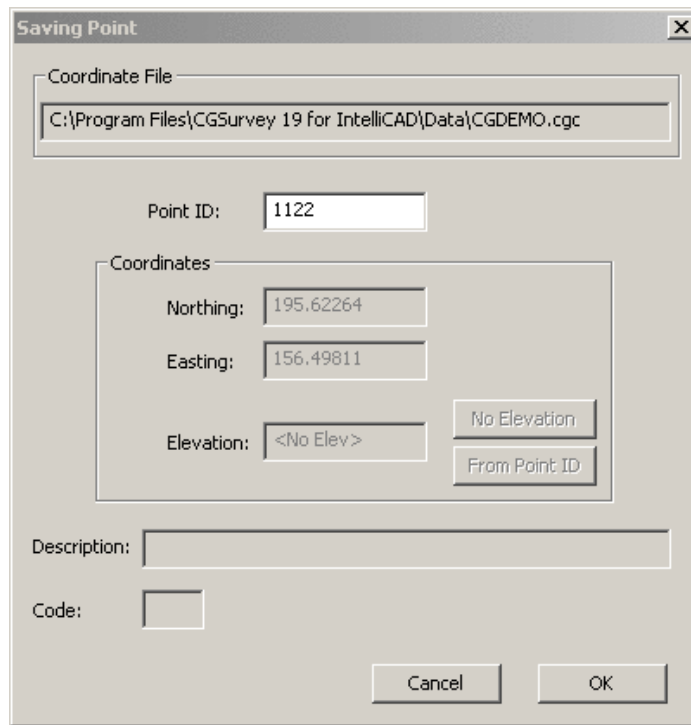
This routine calculates section and 1/4 section corners by one way control. First, enter the point number for Point A. This number can be entered in manually or picked from the screen by selecting the Pick radial button at bottom right. In a like manner, the Bearing from A to B can be entered manually or by using the Pick radial button to pick from the screen. The distance from A to X can be specified in the same manner as above. After selecting OK, a dialog box will display where the Point number, description and elevation can be edited. The point default settings determine the available data for editing. For example, if the option for Automatic Point Numbering is turned off in the Point Defaults, then the field for the point number will be grayed out. If elevations are turned off in the point defaults, then the elevation field will be grayed out. This also applies to the description of the point as well.

Prompts

GLO Proportioning One Way Control dialog



Saving Point dialog



Pulldown Menu Location: COGO > Section Corners > GLO Corner Proportioning

Keyboard Command: cg_glo_one_way

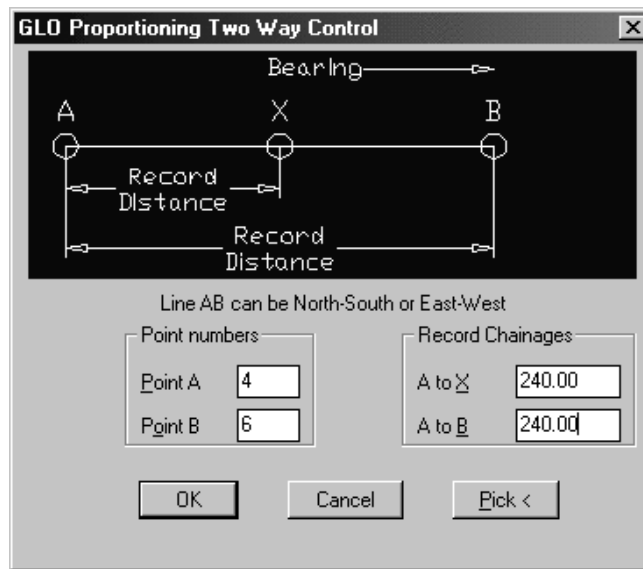
Prerequisite: A coordinate file

Two Way Control

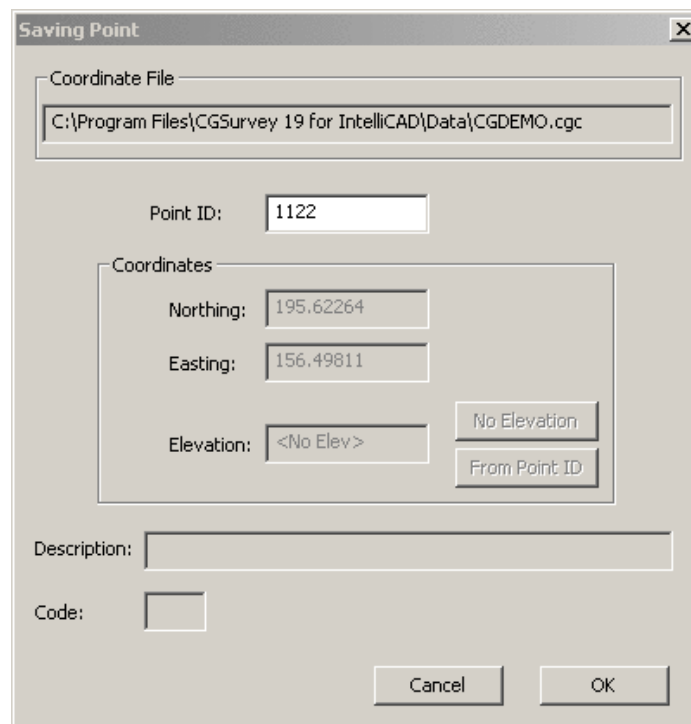
This routine calculates section and 1/4 section corners by two way control. Enter the point numbers for Point A and B. These numbers can be entered in manually or picked from the screen by selecting the Pick radial button at bottom right. In a like manner, the Record Chainages from A to X and from A to B can be entered manually or by using the Pick radial button to pick from the screen. After selecting OK, a dialog box will display where the Point number, description and elevation can be edited. The point default settings determine the available data for editing. For example, if the option for Automatic Point Numbering is turned off in the Point Defaults, then the field for the point number will be grayed out. If elevations are turned off in the point defaults, then the elevation field with be grayed out. This also applies to the description of the point as well. GLO is an acronym for Government Land Office.

Prompts

GLO Proportioning Two Way Control dialog



Saving Point dialog



Pulldown Menu Location: COGO > Section Corners > GLO Corner Proportioning

Keyboard Command: cg_glo_two_way

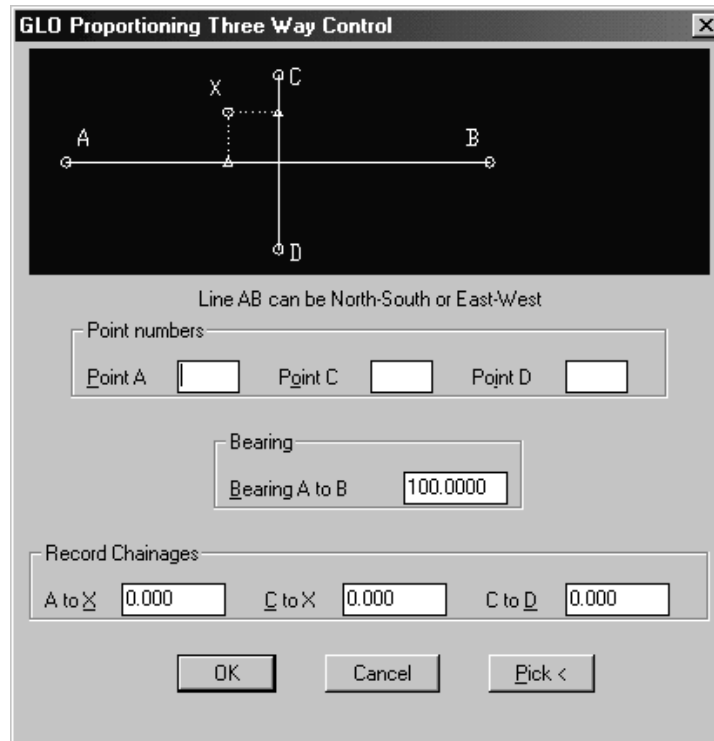
Prerequisite: A coordinate file

Three Way Control

This routine works as the previous GLO Proportioning methods described. Fill out the required data fields on the dialog box and select OK. After selecting OK, a dialog box will display where the Point number, description and elevation can be edited. The point default settings determine the available data for editing. For example, if the option for Automatic Point Numbering is turned off in the Point Defaults, then the field for the point number will be grayed out. If elevations are turned off in the point defaults, then the elevation field will be grayed out. This also applies to the description of the point as well. GLO is an acronym for Government Land Office.

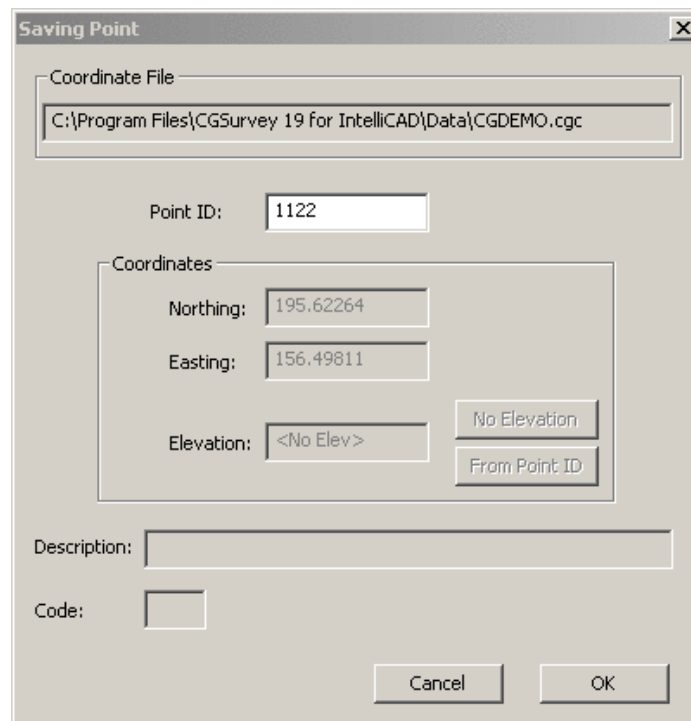
Prompts

GLO Proportioning Three Way Control dialog



The dialog box titled "GLO Proportioning Three Way Control" features a diagram at the top showing a horizontal line segment AB and a vertical line segment CD intersecting at point X. A dashed line connects point X to point C. Below the diagram, the text "Line AB can be North-South or East-West" is displayed. The "Point numbers" section contains three input fields: "Point A", "Point C", and "Point D". The "Bearing" section has a "Bearing A to B" input field with the value "100.0000". The "Record Chainages" section includes three input fields: "A to X" (0.000), "C to X" (0.000), and "C to D" (0.000). At the bottom are "OK", "Cancel", and "Pick <" buttons.

Saving Point dialog



The "Saving Point" dialog box shows the "Coordinate File" path as "C:\Program Files\CGSurvey 19 for IntelliCAD\Data\CGDEMO.cgc". The "Point ID" is "1122". The "Coordinates" section includes "Northing" (195.62264), "Easting" (156.49811), and "Elevation" (<No Elev>). There are "No Elevation" and "From Point ID" buttons. The "Description" and "Code" fields are empty. "Cancel" and "OK" buttons are at the bottom.

Pulldown Menu Location: COGO > Section Corners > GLO Corner Proportioning

Keyboard Command: cg_glo_three_way

Prerequisite: A coordinate file

Four Way Control

This routine works as the previous GLO Proportioning methods described. Fill out the required data fields on the dialog box and select OK. After selecting OK, a dialog box will display where the Point number, description and elevation can be edited. The point default settings determine the available data for editing. For example, if the option for Automatic Point Numbering is turned off in the Point Defaults, then the field for the point number will be grayed out. If elevations are turned off in the point defaults, then the elevation field will be grayed out. This also applies to the description of the point as well. GLO is an acronym for Government Land Office.

Prompts

GLO Proportioning Four Way Control dialog

GLO Proportioning Four Way Control

Line AB is East-West

Point numbers

Point A	<input type="text" value="1"/>	Point C	<input type="text" value="4"/>
Point B	<input type="text" value="2"/>	Point D	<input type="text" value="6"/>

Record Chainages

A to X	<input type="text" value="100.00"/>	C to X	<input type="text" value="135.00"/>
A to B	<input type="text" value="100.00"/>	C to D	<input type="text" value="142.00"/>

OK Cancel Pick <

Saving Point dialog:

Pulldown Menu Location: COGO > Section Corners > GLO Corner Proportioning

Prerequisite: A coordinate (CRD) file

Keyboard Command: cg_glo_four_way

Solar Observations

This feature calculates true north and/or grid north bearings by solar observation. It uses the Local Hour Angle (LHA) method. The routine calculates Ephemeris data, thus alleviating the necessity of obtaining a Solar Ephemeris. The True North option calculates the true north bearing to mark. This option requires no zone/ellipsoid information. The True North & Grid North option calculates both true north and grid north bearings to north. The convergence angle is also shown.

Note: There is a description of solar observation field procedures at the end of this section.

True North Prompts

Calculate true north, or true north and grid bearing (<True north>/Grid Bearing): type T, press Enter

Choose field method (Leading edge/Trailing edge/<Center>): choose method, press Enter

If a Roelofs prism is being used, the Center Method should be selected. If not, select one of the other options. The Trailing Edge Method is the more popular of the two remaining methods.

Date of observation as MM/DD/YY or MM-DD-YYYY: For example 04/08/03.

Enter latitude of instrument point as DD.MMSS: For example 36.0545

Enter longitude of instrument point as DD.MMSS:

The following input loop will begin:

Obs. #1 - Time of observation as HH.MMSS: For example 15.3030

Enter angle to mark as DD.MMSS: Angle in the instrument when backsighting the mark.

Enter angle to sun as DD.MMSS: Clockwise angle from mark to sun.

The angle to the mark always has a default value of the last entered Angle to Mark. Each observation is numbered and the true bearing to the mark will be calculated. There is not limit as to the number of observations that can be made from a setup. After data entry is complete, press Enter.

The following options appear:

[Edit/Ok/Quit] <O>:

If you choose Edit, you will have the following options:

ADD/Change/Delete/eXit:

Add: Allows for addition observation data entry.

Change:

Allows editing of existing data. When selected a prompt for Enter observation to change will be displayed. Choose which observation number to edit. You will then be prompted with the initial input prompts for the observation again. The original input values will be the default values for each prompt. To change the value, simply enter new data.

Once Delete: This will delete the specified observation data. Choose the observation number to delete.

eXit: This exits the change routine.

If you type 0 and Enter or just enter for **OK**, the bearings from all the observations will be averaged and shown as well as the True Bearing. For example:

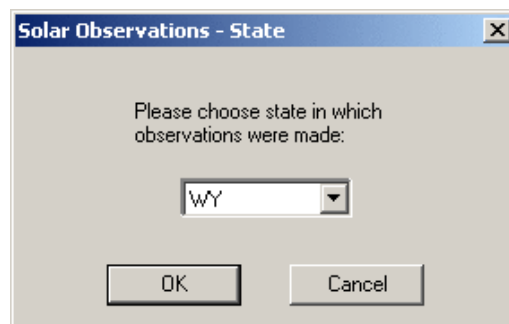
No. Time Angle-@-Mark Angle-to-Sun True-Brg-to-Mark

1 12.3030 0°00'00" 20°00'00" N 73°05'43"E
2 12.4456 0°00'00" 21°00'00" N 74°17'15"E
Average True Bearing: N 73°41'29"E

True North & Grid Bearing Prompts

Type of calculation [True-north/true-north-and-Grid-bearing] <T>: G

The following dialog will be displayed.



Select the state in which the observations were made. All fifty states are available, as well as PR for Puerto Rico and UTM for Universal Transverse Mercator.

If the state is divided into zones, you will be prompted for the zone you are working in.

Enter zone (N,S): Enter the zone.

If you are using a UTM , you will see the following prompt:

Enter ellipsoid to use [GRS-1980/Other] <G>:

Type "R" and Enter or just Enter for Reciprocal flattening, "S" and Enter for Semi-minor axis, or "E" and Enter for ellipsoid ECC squared.

if you typed O and Enter for **Other**, you will see the following prompt:

Ellipsoid constant [Reciprocal flattening/Semi-minor axis/ellipsoid ECC squared] <R>:

Depending on what was entered at the last prompt you will see one of the following prompts: **Enter reciprocal flattening constant:** Type the constant.

Enter semi-minor axis: Enter the axis.

Enter ellipsoid ECC squared constant: Enter the constant.

After entering the zone and ellipsoid information (if applicable) the date, latitude, longitude and time input loop will begin (as described above for the True North calculation).

After data entry is completed the **[Edit/Ok/Quit] <O>:** prompt will be displayed (see the True North section for more details on this prompt).

If you type 0 and Enter or just Enter for Ok, the information for all the observations is displayed along with the Average True Bearing, Average Grid Bearing and the Convergence Angle as follows:

No. Time Angle-@-Mark Angle-to-Sun True-Brg-to-Mark

1 12.2222 0°00'00" 20°00'00" N 72°57'31"E

2 12.4444 0°00'00" 22°00'00" N 74°20'51"E

Average True Bearing: N 73°39'11"E

Average Grid Bearing: N 72°15'12"E

Convergency Angle: 1°23'59"

Field Procedures for the Local Hour Angle (LHA) method

This section explains Universal Time and then explains two ways of pointing, Trailing Edge Tangency and Roelofs Prism.

Universal Time

The Universal Time can be obtained on certain radio bands. On the radio channel there will be a signal beep every minute. Set a watch to the Universal Time or, when in the field, start a stopwatch at the beep (for a known Universal Time).

In order for the solar observation method to produce accurate results, it is essential that you record the precise Universal Time for an observation. Thus, when making the field observations, record the stopwatch elapsed time in order to calculate the Universal Time or, if you set your watch to Universal Time, record the time directly.

Trailing Edge Tangency

While pointing at the ground mark, set the horizontal circle to read about 00-00-30, perfect pointing. With the scope direct, record the horizontal circle reading to the mark.

Attach the eyepiece filter and sight the sun. After locating the sun, do the following:

Set the horizontal reticle line near the center of the sun's image with the vertical reticle line leading the trailing edge of the sun (slightly right for a direct image).

Clamp the horizontal motion and watch the image of the sun as the trailing edge approaches tangency with the vertical reticle line.

Stop the timer at the time of tangency.

Record the time and the horizontal circle reading.

Repeat the pointing for a total of four pointings in the direct position.

Unclamp the horizontal motion, rotate the instrument 180 degrees, plunge the scope, and then obtain data for four reverse readings.

Unclamp the horizontal motion, point at the ground mark with the instrument reverse and record the horizontal circle.

The timer must be checked-in on a radio signal. Some quartz regulated electronic watches are accurate for extended periods of time, allowing several hours of check-in to check-out on the radio. Otherwise, most timers should be started and stopped on a radio signal at the beginning and ending of the observation set.

Roelofs Prism

Attach Roelofs prism and sight the sun (you can center the shadow of the telescope between the standards as an aid in locating the sun). Be sure that the hinged tube is closed when pointing at the sun. After locating the sun through the scope, do the following:

Rotate the prism until the four overlapping images of the sun are symmetrical with the instrument's reticle lines.

Point on the ground mark with the instrument direct and the Roelofs prism tube swung open, perfect pointing on the ground mark. Record the horizontal circle reading to the mark.

Point at the sun with the prism closed. After locating the sun, do the following:

Set the horizontal reticle line near the center of the sun's pattern with the vertical reticle line leading the center of the moving pattern (slightly to right of the sun for direct optics).

Clamp the horizontal motion and watch the pattern move to the point of coincidence. This is the intersection of the vertical reticle line with the apex of the small dark square formed in the center of the pattern by the overlapping parts of the four images formed of the sun.

Stop the timer at the moment of coincidence.

Record the time and the horizontal circle reading.

Repeat the pointing for a total of four readings in the direct position.

Unclamp the horizontal motion, rotate the instrument 180 degrees, plunge the scope, and then obtain data for four reverse readings.

Unclamp the horizontal motion, point on the ground mark with the instrument reversed and record the angle on the horizontal circle.

The timer must be checked-in on a radio signal. Some quartz regulated electronic watches are accurate for extended periods of time, allowing several hours of check-in to check-out on the radio. Otherwise, most timers should be started and stopped on a radio signal at the beginning and ending of the observation set.

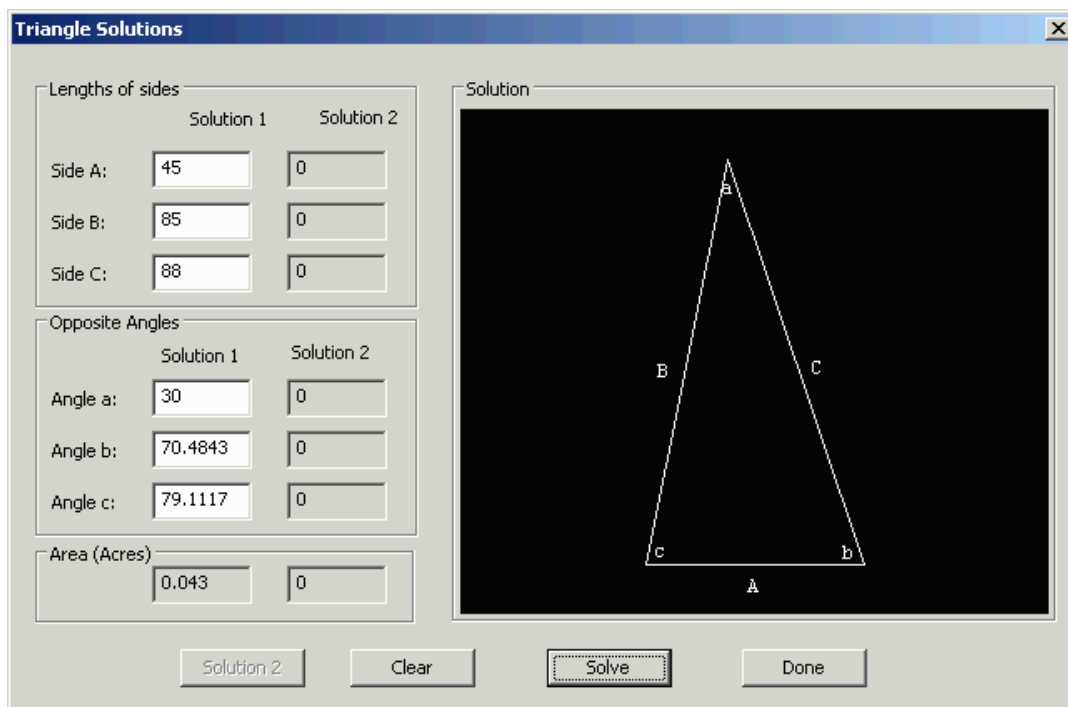
Pulldown Menu Location: COGO

Keyboard Command: cg_solar_obs

Prerequisite: None

Triangle Solutions

Triangle Solutions solves for the remaining sides and angles of a triangle given the known side and angles. The upper case letters A, B and C represent the distances. The lower case letters a, b and c represent the angles. Distance A is the leg of the triangle opposite the angle 'a'. Likewise, distance B and C are the legs opposite the angles of 'b' and 'c', respectively. Enter any three known values of the six possible parameters and the three unknowns will be calculated and displayed. If you enter three angles, you will be shown proportional distances since there is an infinite number of distances that would solve a three angle triangle.



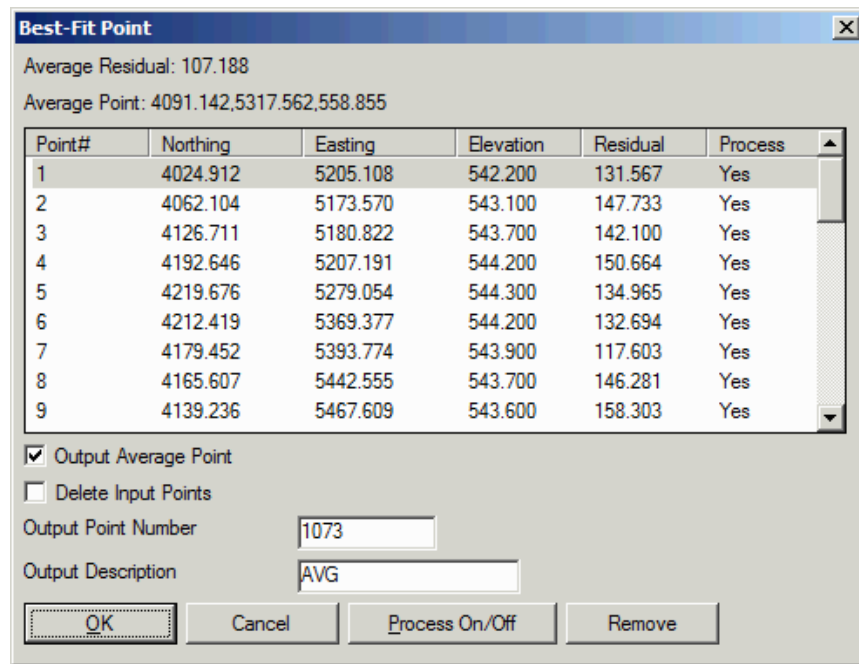
In this example, The sides A & B are known as is angle 'a'. After entering the three parameters, press the Solve button and the remaining three will be calculated and shown. The area in acres or hectares and feet or meters will also be calculated and shown. Press clear to enter data on a new triangle after the triangle has been solved. The solution for each triangle area is then displayed at the command line. You may press F2 to display the command line window and view the results

```
Side A Side B Side C Angle a Angle b Angle c
45.00 85.00 88.40 30°00'00" 70°48'43" 79°11'17"
Area = 1878.550 sq. ft. (0.043 acres)
```

Pulldown Menu Location: COGO
Keyboard Command: trianglesolutions
Prerequisite: None

Best Fit Point

This command calculates the average point from a selection of input points and reports the residual statistics. The input points can be specified by point number, by point group or by screen selection. The program displays the input points with residuals in a dialog where you can toggle on/off whether to include points in the point average using the Process On/Off button. The Remove button removes a point from the average and the residual report. There is an option whether to output the average point to the current coordinate file. The option to delete all the input points applies when there are several points that are meant to be the same point and you want to replace them with a single averaged point. The command shows a report of the input points, residuals and average point.



Prompts

Select points from screen, group or by point number [*<Screen>/Group/Number*]? *press Enter*

Select Carlson Software Points.

Select objects: *pick points*

Best Fit Dialog

Sample Report:

Source Coordinates

Point#	Northing	Easting	Elevation	Residual
1	4024.912	5205.108	542.200	131.567
2	4062.104	5173.570	543.100	147.733
3	4126.711	5180.822	543.700	142.100
...				

Residuals Standard Deviation: 37.128

Average Residual: 107.188

Average Point: 4091.142,5317.562,558.855

Pulldown Menu Location: COGO

Keyboard Command: bfitpt

Prerequisite: Two or more points

Best Fit Circle

This command draws a least-squares best-fit circle based on points on the perimeter. The program handles four or more perimeter points. A design point for the circle center can optionally be specified as a reference to compare with the best-fit center in the report. The report shows the residuals for each point, the residuals standard deviation, the difference between the design point and the circle center, and the circle parameters. The residuals are calculated as the perpendicular distance from the point to the circle. The best-fit circle can be calculated in 2D or 3D. In 2D mode, the elevation of the points is not used. In 3D mode, a best-fit plane is calculated for the points. Then the points are projected onto the plane and the best-fit circle is calculated on this plane. Then the resulting circle

is projected back into world coordinates and drawn as a 3D polyline with short chords to represent the 3D circle since CAD doesn't support a 3D circle entity. Applications for 3D circles are tunnel sections and architectural arches.

After specifying the points, the program calculates the best-fit circle and shows the results in the dialog show here. You can toggle each point for whether to include in the calculations. You can also modify the radius.

Prompts

Create 2D or 3D circle [**<2D>/3D**]? *press Enter*

Select points from screen or by point number [**<Screen>/Number**]? *N*

Point numbers: *2-6*

Point numbers (Enter to continue): *press Enter*

Enter design center point# (Enter for None):

Point#	Northing	Easting	Residual	Process
37	11533.784	10597.813	0.686	Yes
38	11562.410	10630.352	-0.993	Yes
39	11568.790	10668.917	0.264	Yes
40	11556.029	10699.437	0.735	Yes
41	11534.760	10716.946	0.150	Yes
42	11506.638	10724.753	-0.295	Yes

Sample Report:

Source Coordinates

Point#	Northing	Easting	Residual
2	5253.198	5070.233	0.126
3	5246.623	5084.077	0.045
4	5232.963	5078.608	-0.131
5	5235.610	5065.105	0.217
6	5247.392	5064.165	-0.264

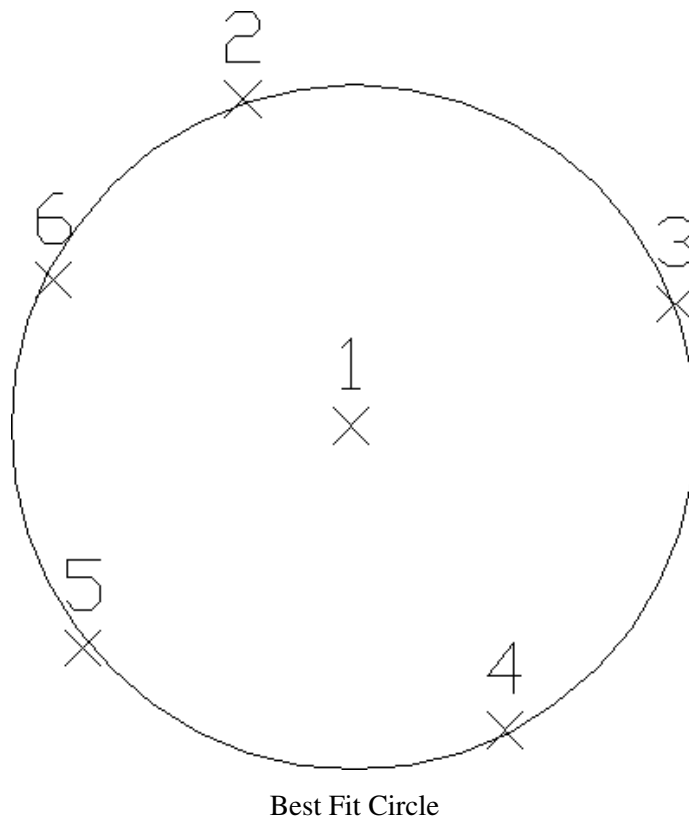
Residuals Standard Deviation: 0.174

Circle Center: 5242.678,5073.785 Radius: 10.977

Design Center Point#: 1

Design Center: 5242.718,5073.688

Center Distance Difference: 0.105



Pulldown Menu Location: COGO

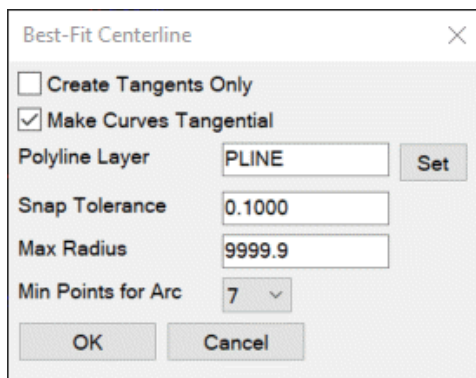
Keyboard Command: bfitcir

Prerequisite: Four or more points

Best Fit Centerline

This command processes a group of points to compute the best fitting centerline by least squares. The points can be input from Carlson points or from vertices of a polyline. For points input, the points can be selected by screen selection or point number range. Each line segment in the centerline is calculated by the best-fit line method and each arc segment is calculated by the best-fit arc method.

In the process options dialog, the **Snap Tolerance** is the max offset from the point to the line or arc segment in order to be counted as part of that segment. The **Polyline Layer** is used for the output polyline. The **Max Radius** controls the maximum radius for arc segments that the program will fit to the data. The **Create Tangents Only** option skips the step of calculating the best-fit arcs and outputs only the lines. Arcs can be added later using routines like Fit Curve in the Centerline Editor command. The **Make Curves Tangential** option fits the line and arc segments to be tangential. The **Remove Outliers** applies when the input data is noisy. This option will remove points that seem too far off the centerline to improve the centerline match with the remaining point. The **Min Points For Arc** setting controls how many points are required to calculate an arc. If this number is low that makes is more likely to create an arc where some points vary from linear but there really isn't meant to be an arc. If this number is high that makes is more possible to miss creating an arc in case the arc only has a few points on it.



The residual for each point is the perpendicular distance from the point to the best-fit centerline. The results are shown in a dialog and you can toggle each point for whether to include in the calculations. Points that are toggled off are not used for calculating the centerline but are still used in the residual report. The Remove function removes the point from both calculation and residual reporting.

Northing	Easting	Residual	Process
5074.084	5967.049	0.002	Yes
5079.623	5943.489	-0.003	Yes
5091.078	5894.816	-0.000	Yes
5102.531	5846.162	0.005	Yes
5113.979	5797.465	-0.006	Yes
5125.432	5748.805	-0.002	Yes
5136.889	5700.153	0.007	Yes
5148.335	5651.456	-0.005	Yes
5159.786	5602.817	0.002	Yes
5168.537	5557.053	-0.006	Yes
5172.974	5517.325	0.000	Yes

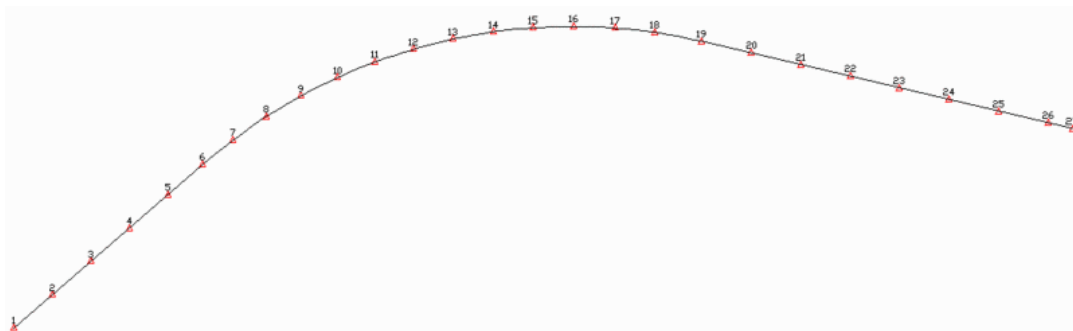
Prompts

Input existing data from points or polyline [<Points>/Line**]?** *press Enter*

Select points from screen or by point number [<Screen>/Number**]?** *S*

Select Carlson Software Points.

Select objects: *pick the centerline points*



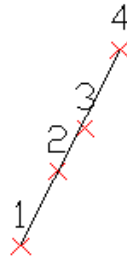
Pulldown Menu Location: COGO

Keyboard Command: bestcl

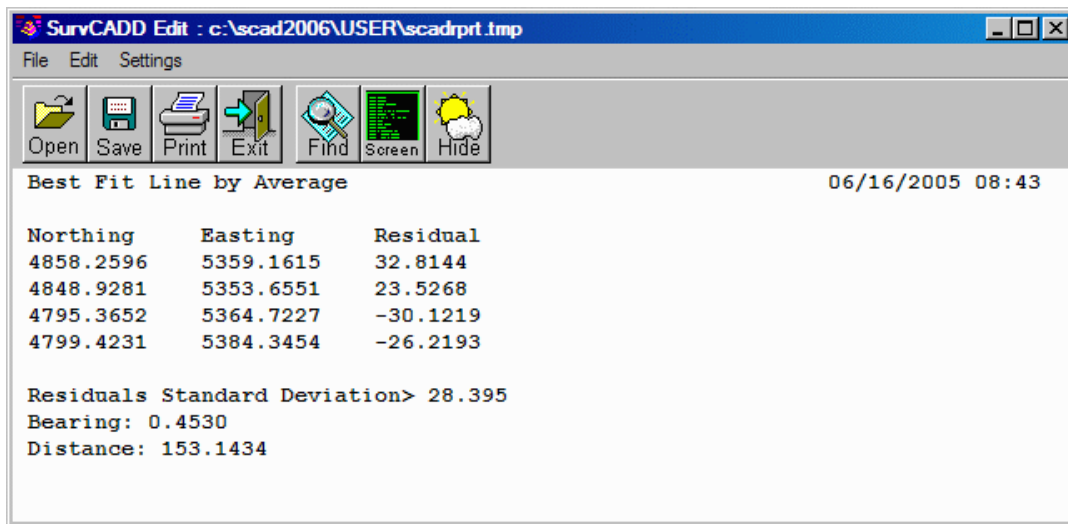
Prerequisite: Group of points or a polyline to sample

Best Fit Line by Average

This command will fit a line from a starting point by sampling a group of points. The routine averages the coordinates of the sampling group then draws the best-fit line. The program generates a report of the residuals, standard deviation, line bearing and line distance. The perpendicular distance from each point to the line is reported as the residual.



Screen selection of lines almost in line with one another



Sample report of Best Fit Line by Average with a different group of points

Prompts

Starting point ?

Pick point or point number: *pick starting point*

Select points from screen, group or by point number [**<Screen>/Group/Number**]? *press Enter*

Select points.

Select objects: *select group of points* Select points using Window or Crossing. The line is then drawn to the computed point.

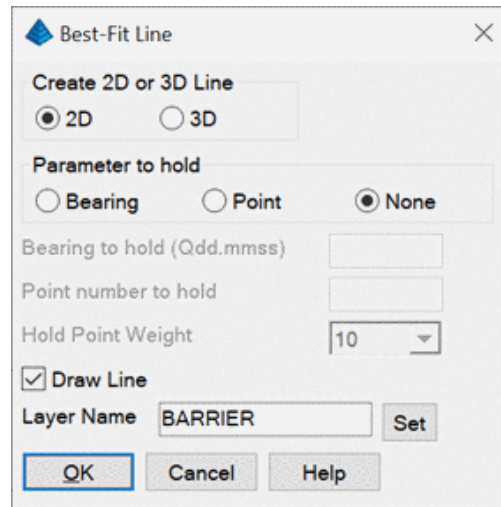
Pulldown Menu Location: COGO

Keyboard Command: bfitlin

Prerequisite: points to sample

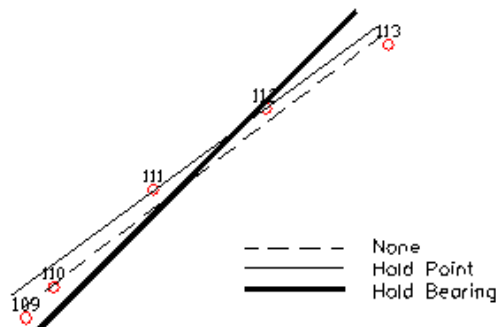
Best Fit Line by Least Squares

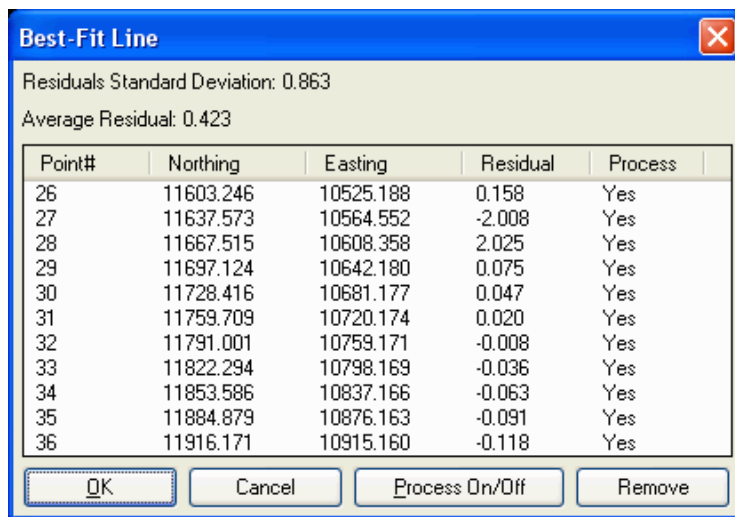
This command will process a group of points to compute the best fitting line by least squares. The points can be selected by screen selection of point entities, point number, point group name, screen pick or polyline vertices. The program can calculate either a 2D or 3D line. There are options to best fit with nothing held (None), to best fit by holding a point, and to best fit by holding a bearing. All three options are shown below in the graphic. When holding a point, you are prompted to enter the weight for the point. In this example, a weight of 1000 caused the line to pass to within 0.025 of point 111. With a weight of 5000, the line passed to within 0.005 of point 111. Increase the weight accordingly to obtain the desired precision. When holding a bearing such as N45E, you are prompted to enter the bearing in the form QDD.MMSS (e.g. 145.0000 or just 145). The program generates a standard report. The residual for each point is the perpendicular distance from the point to the best-fit line.



After specifying the points, the program calculates the best-fit line and shows the results in the image shown here. You can toggle each point for whether to include in the calculations. Points with Process set to No and not used for calculating the line but are still included in the report of residuals. Use the Remove button to remove a point both from calculation and reporting. Use the Add button to add another point into the calculations.

The Create Two Parallel Lines option applies when the points are for two line that are meant to be parallel such as a right-of-way. The program takes input for the distance between these two lines and then automatically sorts the points between the two lines, reports the residuals for the two lines and draws the two lines.





Prompts

Select points from screen, group or by point number [`<Screen>/Group/Number/Pick/Line`]? *Press Enter for Screen*

Options Dialog

Select Carlson Software Points.

Select objects: *pick the five points*

Point numbers (Enter to continue): *press Enter*

Parameter to hold [`<None>/Point/Bearing`]: *P*

Enter point number to hold: *111*

Enter weight for point: *5000*

Sample Report:

Best Fit Line By Least Squares

Holding point 111: (5227.721, 5149.482)

Coordinate File> c:\data\interval.crd

Source Coordinates

Point#	Northing	Easting	Residual
109	5103.542	5182.098	10.050
110	5114.634	5191.928	6.921
111	5149.482	5227.721	0.005
112	5178.703	5268.237	0.400
113	5201.666	5312.602	8.129

Residuals Standard Deviation: 6.559

Bearing: N 53d44'07'' E

Distance: 163.266

Pulldown Menu Location: COGO

Keyboard Command: bfitline1q

Prerequisite: Group of points to sample

Point on Arc

This command locates a point on an arc. You can select an arc entity, an arc polyline segment or enter three points to define an arc. After the arc is defined, the screen preview arrow shows the occupied point and the distance to solve

for is entered. The command then displays the curve information and locates/inserts a point symbol at the computed point. When prompted for the distance, use a positive value if the distance is from the 1st endpoint (PC the one highlighted by the screen preview arrow) and a negative value if from the 2nd endpoint (PT).

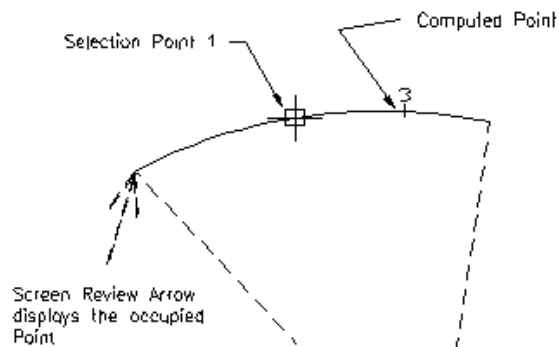
Prompts

Define arc by, Points/<select arc or polyline>: *pick arc or polyline arc segment* Pick a point on the arc somewhere near it's midpoint. The preview arrow points to the 1st endpoint.

Precede distance with minus sign if distance from 2nd endpoint.

Distance along arc from 1st point: *100*

The command then plots a point at the computed distance.



If a positive distance is entered for the distance then the point is measured from the occupied point. If a negative value is entered the point is measured from the opposite endpoint of the arc.

Pulldown Menu Location: COGO > Interpolate Points

Keyboard Command: ptarc

Prerequisite: None

Divide Between Points

This command divides the distance between two points and inserts one of the point symbols at the specified distances. It can also interpolate elevations (to interpolate the elevations, the points picked must be at their real Z axis elevation).

Prompts

Interpolate elevations [Yes/<No>]? *hit Enter*

Point to divide-interpolate from?

Pick point or point number: *1*

PointNo.	Northing (Y)	Easting (X)	Elev (Z)	Description
1	4252.76	4158.32	0.00	

Point to divide-interpolate to?

Pick point or point number: *pick a point*

Number of Segments-Divisions: *3*

Enter Point Description <>:*hit Enter*

The command then locates two points.

Pulldown Menu Location: COGO > Interpolate Points

Keyboard Command: divlin

Prerequisite: 2 points

Divide Along Entity

This command locates points along an entity such as a line, polyline, spline or arc. You must specify the number of divisions.

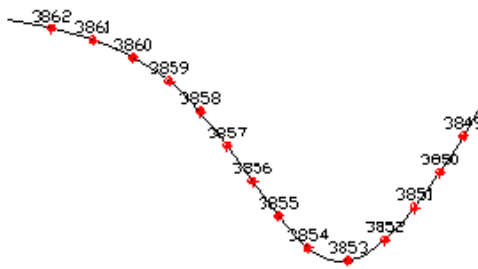
Prompts

Interpolate Elevations [Yes/<No>]: *press Enter*

Select Entity to Divide: *pick point on entity*

Number of Divisions/Segments: *15*

The command then locates 14 points.



Pulldown Menu Location: COGO > Interpolate Points

Keyboard Command: divent

Prerequisite: 2 points if you want to interpolate elevations

Elevation Along Entity

This command creates points at a specified elevations along 3D polylines. The points are stored in the current coordinate file and drawn on the screen. A single elevation can be entered or use an elevation interval.

Prompts

Select entities to process

Select objects: *pick a 3D polyline*

Elevation Range: *547.200 to 587.300*

Add single elevation or elevation interval [Single/<Interval>]?: *press Enter*

Enter elevation interval: *1*

Point Description: *EP*

Pulldown Menu Location: COGO > Interpolate Points

Keyboard Command: entptz

Prerequisite: 3D polyline

Interval Between Points

This command creates points by interpolating at a horizontal distance interval between two control points. There is an option for whether to interpolate the elevation or use zero for elevation. The point style and whether to prompt for a description is set in *Point Defaults*.

Prompts

Interpolate elevations [Yes/<No>]? press Enter

Point to interpolate from?

Pick point or point number: *pick a point*

Point to interpolate to?

Pick point or point number: *pick a point*

Interval Distance: *50*

Pulldown Menu Location: COGO > Interpolate Points

Keyboard Command: ptintpt


Prerequisite: None

Interval Along Entity

This command creates points at a specified distance along an entity such as a line, arc, spline or polyline. The points are listed out on the text screen, stored in the current coordinate (.CRD) file and drawn on the screen. For example, you might use this command to locate lot corner points along a frontage line. When Break Entity at Points is checked, the selected entity will be broken at every located point. The Create Points At Endpoints option locates points at the starting and ending points of the entity. When Create Point at Vertices is checked, points will also be located at each vertex of the selected entity. Horizontal Distance Between Points allow you to specify the distance between located points. There is also an option to create points on curved portions of the centerline at a different interval than on tangent portions (to reduce chord lengths, a shorter interval may be suitable for curves).

There is an option in this dialog allowing you to determine whether or not to label elevations on the new points. And for the purposes of describing the points, there is an option that allows you to set the same description to all of the points. For more options related to points, see *Point Defaults* under the Points pulldown.

Interval Along Entity

Symbol Name: 

Select Symbol

Point Number Settings

Point Numbers

Automatic Point Numbering

Starting Point Number

Elevation Settings

Prompt for Elevations

Label Elevations

Locate on Real Z Axis

Description Settings

Prompt for Description At Each Point

Same Description For All Points

Description for Points

Layer Name for Points Set

Break Entity At Points

Create Points At Endpoints

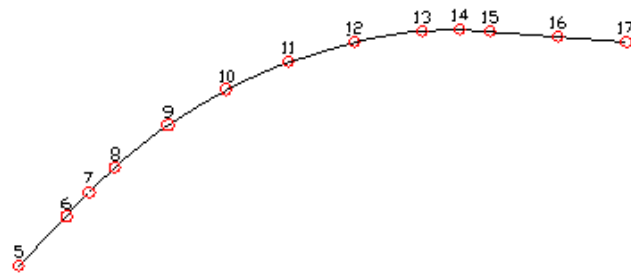
Reset Interval At Endpoints

Horizontal Distance Between Points

Distance On Line Segments

Distance On Curve Segments

OK Cancel Help



Create Points at Endpoints turned on

Prompts

Select entity near endpoint which defines first station.

[nea on] Select Entity to Interpolate Points: *select entity*

[nea on] Select Entity to Interpolate Points: **Locating 13 Points**

The command locates points along the selected entity.

Pulldown Menu Location: COGO > Interpolate Points

Keyboard Command: ptint

Prerequisite: An entity

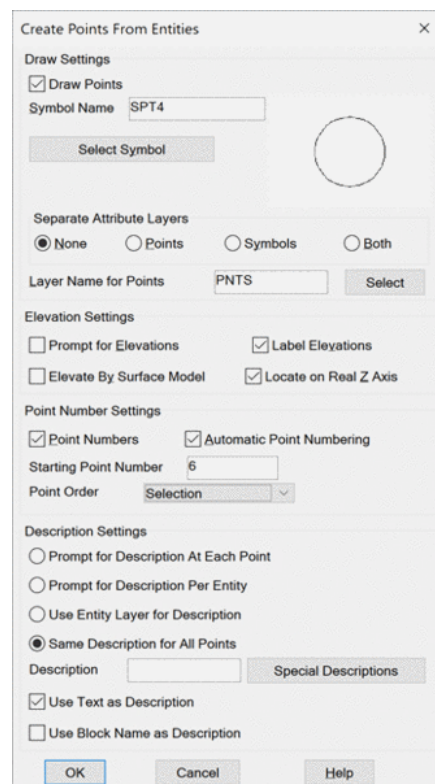
Create Points from Entities

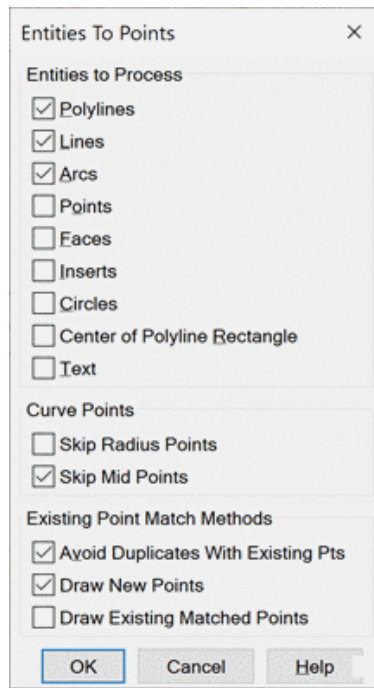
This command will create Carlson points on selected entities. The points are stored in the current coordinate (.CRD) file and drawn on the screen. For arcs and polylines with arc segments, points are created at the radius points of the

arcs as well as the PC and PT.

In the first options dialog, there are settings for the point attributes. To have points obtain their elevation from the selected entities, unselect the **Prompt for Elevations** toggle and select the **Locate on Real Z Axis** toggle. After you have specified the point options, a secondary dialog appears which allows you to specify the entity types to process. The **Point Order** setting controls the sequence of the new point numbers to be in the order of selection or in a direction like left to right. Under the **Description Settings**, **Prompt for Description At Each Point** will prompt you at the command line for a description for each individual point. **Prompt Per Entity** will ask you for a description per each highlighted entity. **Use Entity Layer for Description** will assign the layer name to the description. When Entity Layer for Description is checked, the layer name of the entity will be used as the description for the created point. **Same Description For All Points** will prompt you for a single description for all points. The **Use Text as Description** applies to points from Text entities where the description is set to the text string. The **Use Block Name as Description** applies to points from Insert entities and the program sets the point description to the block name.

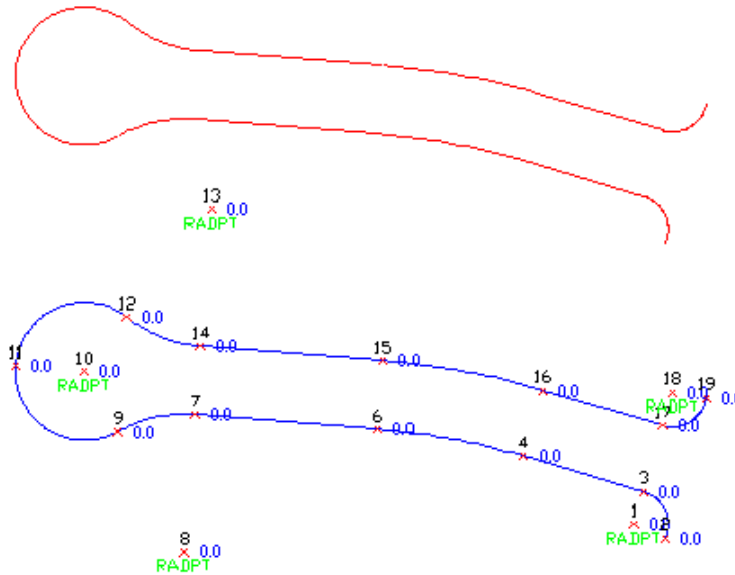
The second options dialog has processing settings. When **Avoid Duplicates with Existing Pts** is checked, this routine will not create a point if a point with the same coordinates already exists in the current coordinate (.CRD) file. The **Draw New Points** option creates point entities in the drawing. Otherwise, the new points are only stored to the coordinate file. The **Draw Existing Matched Points** option applies to the Avoid Duplicates option for the case when a duplicate is found in the coordinate file and not yet drawn.





Prompts

Create Points From Entities Dialogs Choose settings
 Select arcs, circles, faces, points, text, lines and polylines.
 Select objects: *pick entities*

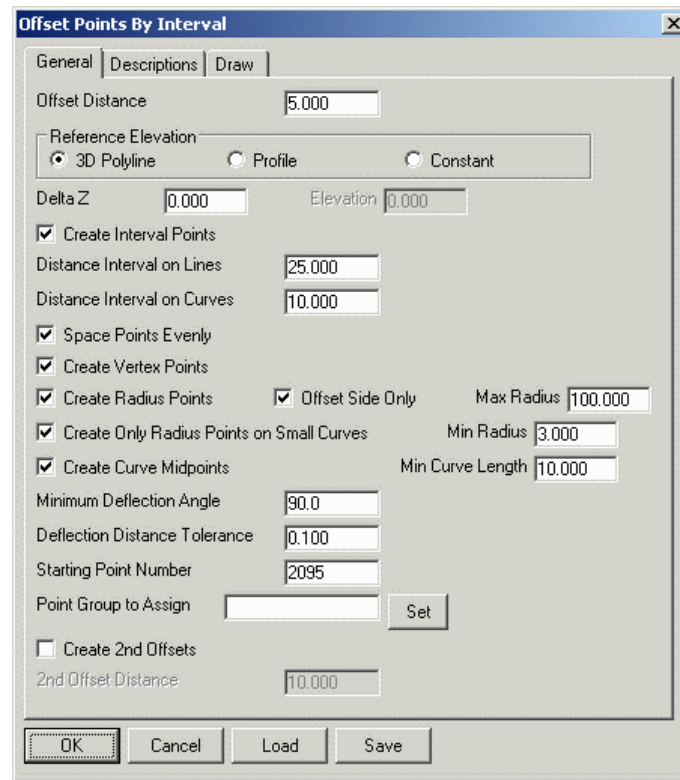


Before and after using Create Points from Entities. Points are created at each endpoint and radius point.

Pulldown Menu Location: COGO
Keyboard Command: autopnts
Prerequisite: drawing entities

Offset Points By Interval

This command creates offset points along an alignment which can be used for creating curb stakeout points. The alignment can be defined by either a 2D polyline or a centerline file (.cl). The options dialog has three tabs for the General, Description and Draw settings.



Offset Distance: Sets the offset distance from the alignment to the points. Use a positive value for offset right and negative for offset left.

Reference Elevation: This setting controls the point elevations. The **3D Polyline** method prompts to select a 3D polyline. The offset points get the elevations from the position of the nearest perpendicular offset on the 3D polyline. The **Profile** method prompts to select a profile file. The stations of the profile must match the alignment stations. The **Constant** method sets the point elevations to a fixed elevation. The **Surface Model** method uses a triangulation (.tin) or grid (.grd) surface model file.

Delta Z: Adjusts the point elevations from the reference elevations.

Create Interval Points: Creates points along the alignment at the specified interval. There are separate intervals for line and curve segments.

Space Points Evenly: For each segment, this option figures how many points fit at the interval and then adjusts the interval to make that number of points evenly spaced. For example, if the interval is 10 and the segment length is 48, then 4 points fit and this option will space them at an interval of 12.

Create Vertex Points: Creates points offset from the points in the alignment.

Create Radius Points: Creates radius points for curve segments in the alignment.

Offset Side Only: Applies to radius points and only creates the radius point when it is on the same side of the alignment as the offset direction.

Max Radius: Applies to radius points to only create the radius point when the radius of the curve is less than this amount.

Create Only Radius Points on Small Curves: When the curve radius is less than the **Min Radius**, only the radius point is created and not the PC, PT or mid points.

Create Curve Midpoints: Creates a point midway along the curve when the curve length is greater than the specified **Min Curve Length**.

Minimum Deflection Angle: At an alignment vertex between two line segments, when the deflection between the line in and line out is greater than this value, only a single offset point is created. Otherwise two offset points are

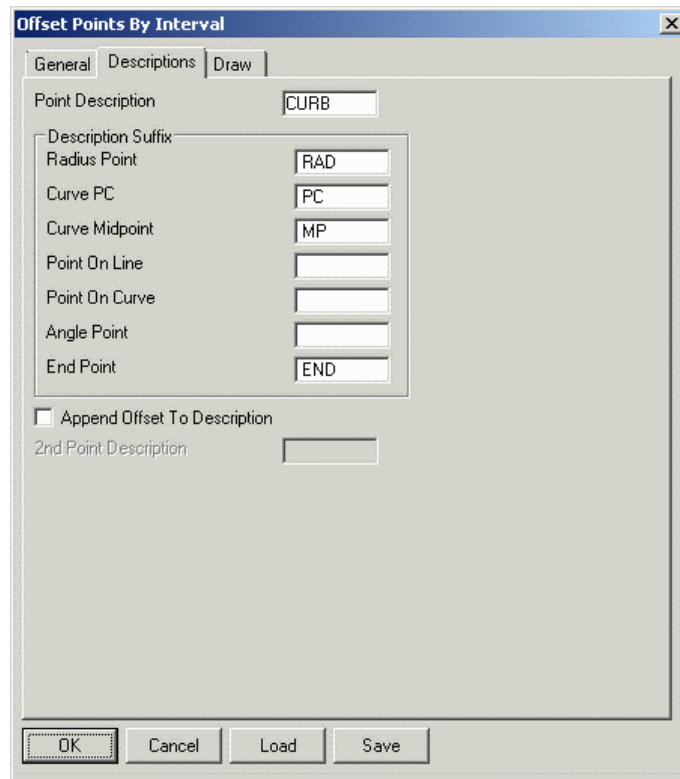
created, one for the line in and one for the line out. Set this value to 0 to always have only a single offset point. Set this value to 180 to always have two offset points.

Deflection Distance Tolerance: Sets how close together two offset points should be set on vertex deflection points. When the vertex point is set to get two offset points, but the offset points are within this tolerance of each other, then only a single offset point is created instead of two.

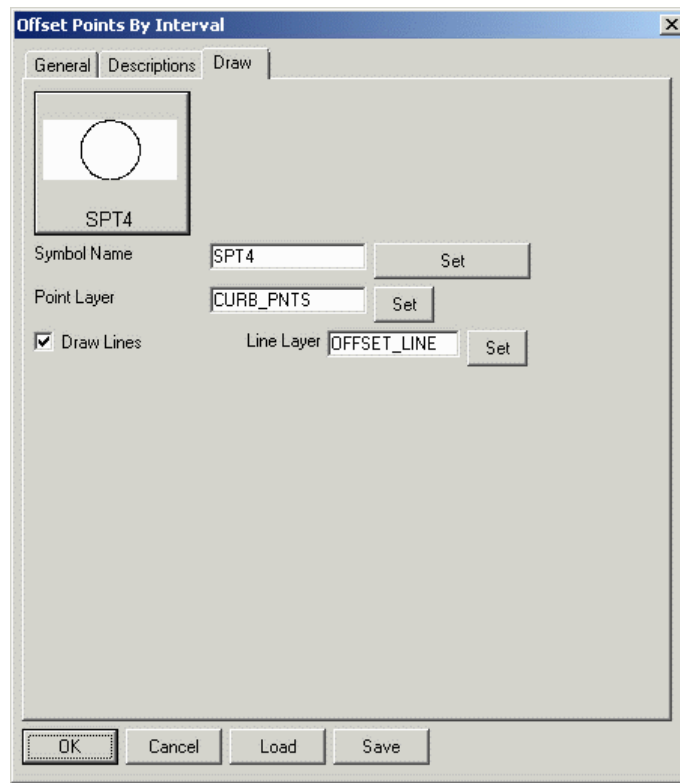
Starting Point Number: The new point numbers will increment starting from this number.

Point Group to Assign: Creates a point group for the new points created by this command.

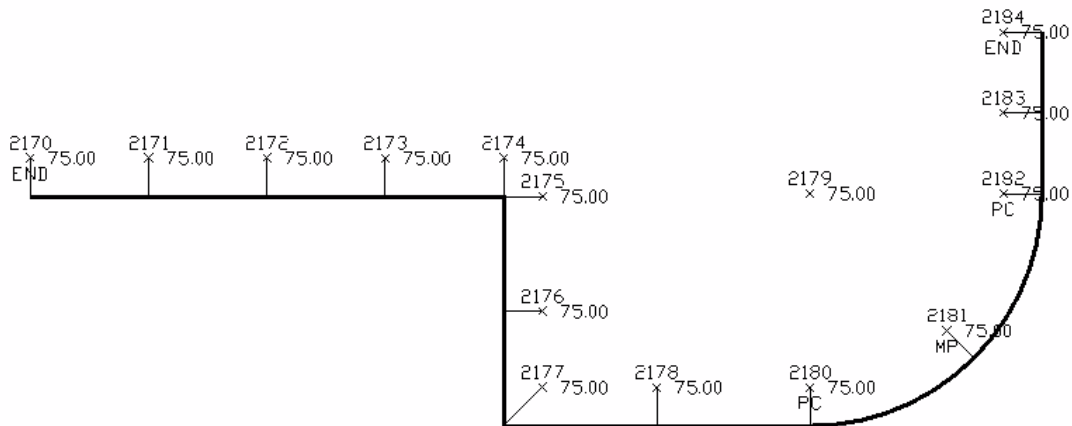
Create 2nd Offsets: Creates a second set of points at the **2nd Offset Distance**.



The Description settings control the point descriptions for the various types of new points. The **Append Offset To Description** option adds the offset amount to the description of each point.



The Draw settings have the symbol and layer for the points. The **Draw Lines** option is for drawing lines between the alignment and the points.



Prompts

CL File/<Select alignment polyline>: *pick a polyline*
Offset Points dialog

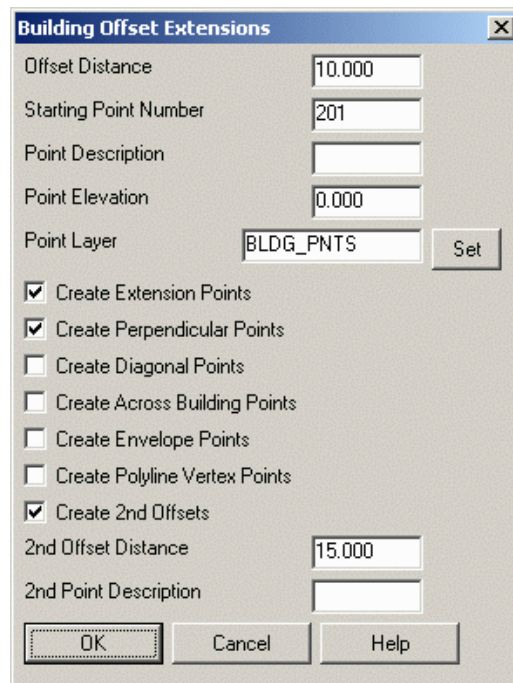
Pulldown Menu Location: COGO

Keyboard Command: curb_pnts

Prerequisite: Alignment polyline or centerline

Building Offset Extensions

This command is used to calculate building corner offset points that are extensions of the building faces. This command uses building perimeters that are drawn as closed polylines. The points are stored to the current coordinate file and drawn on the screen. There is a dialog for setting the parameters. The **Offset Amount** is the distance that the offsets are extended past the end of the building line. The **Starting Point Number** is the point number to begin storing from. The **Point Description** and **Elevation** are assigned to all the new points and the **Point Layer** is used for all the drawn points. The **Create Extension Points** controls whether offset points are created as extensions of the building lines at the corners. The **Create Perpendicular Points** applies to the extension points for creating a pair of perpendicular points at the corners. The **Create Diagonal Points** method creates points at the diagonals of corners. The **Create Across Building Points** creates points across to the other side of the building for inside corners. The **Create Envelope Points** calculates a rectangle that encloses the building and creates points at the four rectangle corners. The **Create Polyline Vertex Points** creates points at each vertex in the building perimeter. The **Create 2nd Offset** option makes a second point for each point at another offset and with another description.

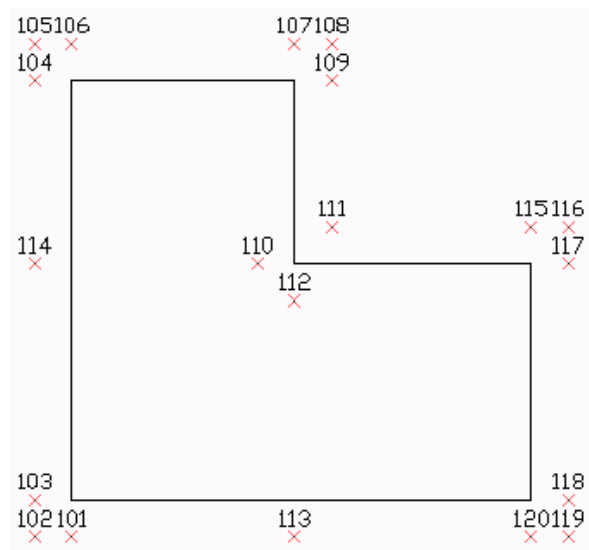


The screenshot shows the 'Building Offset Extensions' dialog box with the following settings:

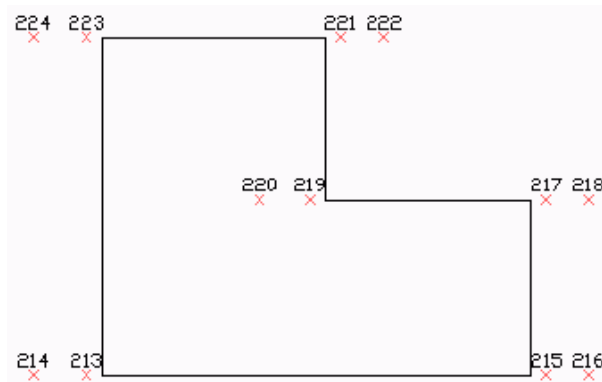
- Offset Distance: 10.000
- Starting Point Number: 201
- Point Description: (empty)
- Point Elevation: 0.000
- Point Layer: BLDG_PNTS (with a 'Set' button)
- Create Extension Points
- Create Perpendicular Points
- Create Diagonal Points
- Create Across Building Points
- Create Envelope Points
- Create Polyline Vertex Points
- Create 2nd Offsets
- 2nd Offset Distance: 15.000
- 2nd Point Description: (empty)

Buttons at the bottom: OK, Cancel, Help.

In the first example shown here, Create Perpendicular Points is on. Points 101, 103, 104, 106, 107, 109, 110, 112, 115, 117, 118 and 120 are corner extension offset points. Points 102, 105, 108, 111, 116 and 119 are diagonal points. Points 113 and 114 are across building points.



In the second example, Create Perpendicular Points is off and the program only creates points on one side. Also on this example, the Create 2nd Offsets option is on.



Prompts

Building Offset Extensions dialog
Select building perimeter linework.
Select objects: *make selection*

Pulldown Menu Location: COGO
Keyboard Command: bldg_pnts
Prerequisite: A polyline perimeter that represents a building

Radial Stakeout

This command creates a radial stakeout report using the current coordinate (.CRD) file. The program calculates the azimuth, angle right, horizontal distance and/or slope distance for a range of points relative to an occupied point and a backsight point.

Radial Stakeout

Occupied Point Number:

X: Y:

Backsight Point Number:

X: Y:

Maximum Hz Distance:

Range of points to Compute: Select Points from Screen

Number of Decimal Places for...
 Northing/Easting: Elevation:

Report Options
 Bearing Azimuth Angle Right Azi & Ang Right

Report Slope Distance
 Use Cut Sheet Format

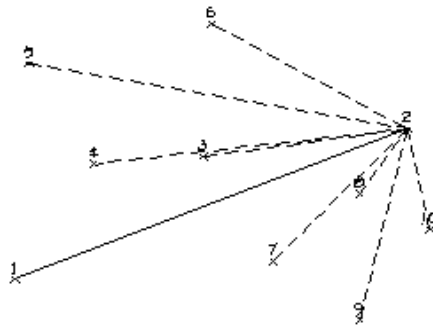
- Occupied Point Number:** Specify the occupied point number X and Y values will fill in automatically.
- Backsight Point Number:** Specify the backsight point number X and Y values will fill in automatically.
- Maximum Hz Distance:** This is the maximum horizontal distance from the occupied point that the program will include in the report.
- Range of points to Compute:** Enter the range of points to be included in the stakeout report If you check Select Points from Screen, this option is unavailable.
- Select Points from Screen:** This option allows you to select from the screen the points to be included in the stakeout report.
- Number of Decimal Places:** Specify the display precision for the report.
- Report Options:** Specify the direction format that the report should use.
- Report Slope Distance:** When checked, the slope distance is included in the report in addition to the horizontal distance.
- Use Cut Sheet Format:** When checked, adds columns to the report for Description, Hub Elev, and Elevation.

Point List

Select Point To Use

POINT#	NORTHING	EASTING	ELEVATION	DESCRIPTION
84	5607.342	7094.082	0.000	
85	5653.582	7113.105	0.000	
86	5699.474	7131.984	0.000	
87	6346.747	10082.366	4.782	
88	6910.131	9452.822	0.000	
89	6466.414	9333.032	0.000	
90	6301.674	8932.210	0.000	
91	5226.485	9780.132	0.000	

Results from clicking the List (point) button



Sample radial stakeout report:

Radial Stakeout

Occupied Point

2 7137.7248 9016.1417 500.000

Backsight Point

1 7075.7408 8875.7884 500.000

Backsight Azimuth= 246.1021

PtNo.	Azimuth	AngRight	HzDist	North (y)	East (x)	Elev (z)
3	261.0258	14.5237	74.061	7126.2022	8942.9830	500.000
4	262.4347	16.3327	113.032	7123.4208	8904.0181	500.000
5	281.1809	35.0748	137.858	7164.7435	8880.9572	500.000
6	301.4512	55.3452	82.296	7181.0342	8946.1639	500.000

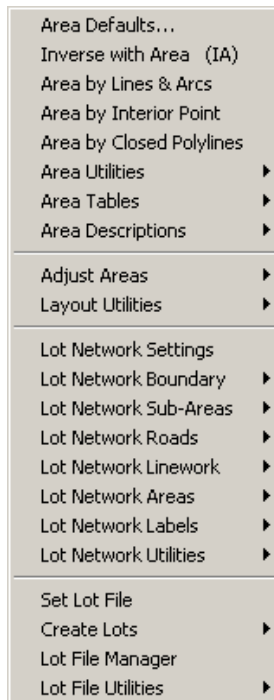
Pulldown Menu Location: COGO

Keyboard Command: radstake, rs

Prerequisite: A coordinate file (.CRD file) with points

Area/Layout Menu

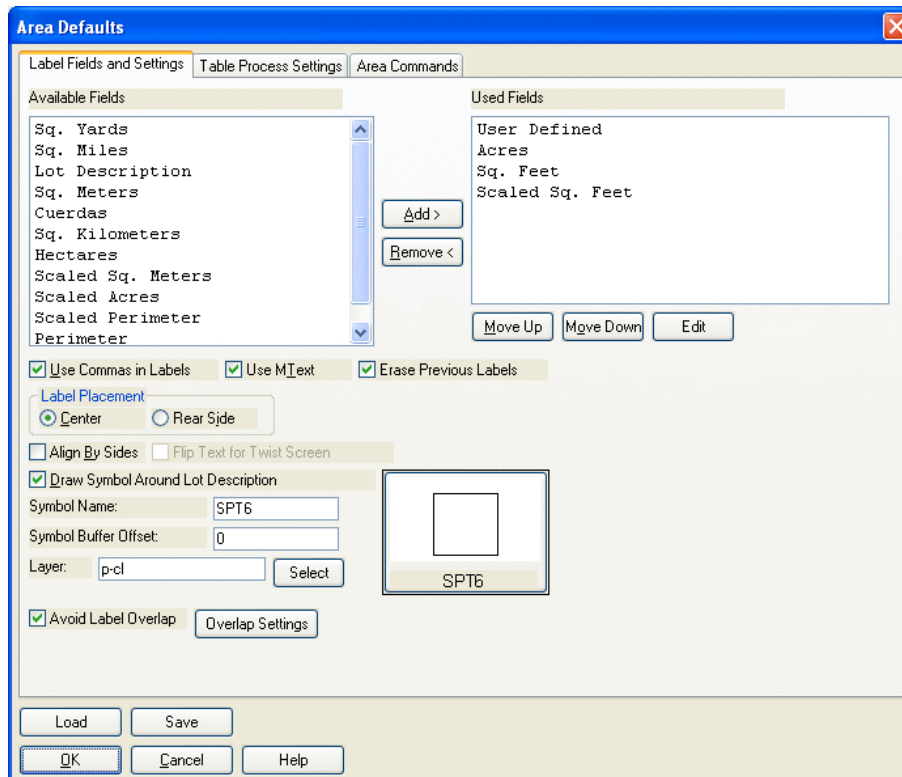
This chapter provides information on using the commands from the Area/Layout menu to calculate and label areas, and also to set and define lots. Commands for designing and drawing more complex configurations, such as cul-de-sacs and intersections, are available here as well.



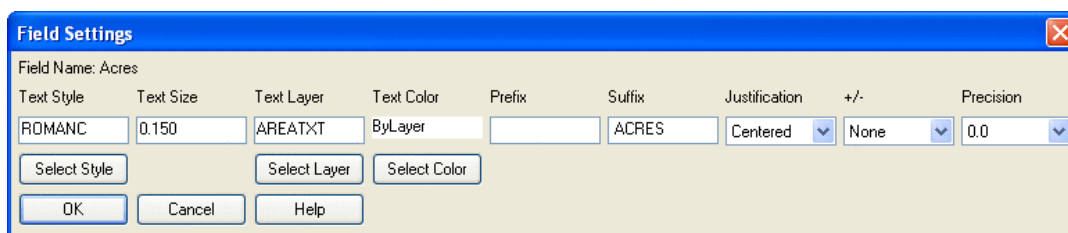
Area Defaults

This command allows you to specify default settings for area labeling. The Area Defaults dialog is divided into 3 tabs. The first is the Label Fields and Settings tab. The top portion of the Label Fields and Settings tab contains two listboxes which are used to control which of the possible ten area fields will be used for area labeling. You use the Add and Remove buttons to control which fields will be included in area labels. You can also add to the Used Fields list by double-clicking on items in the Available Fields list. The area label will include the values in the order as specified in the Used Fields listbox. To change the order you use the Move Up and Move Down buttons.

When a grid projection is defined in Drawing Setup, the Available Fields will include geodetic areas where the areas are adjusted by the projection. The Base Z from Drawing Setup is used for the elevation factor for this adjustment.

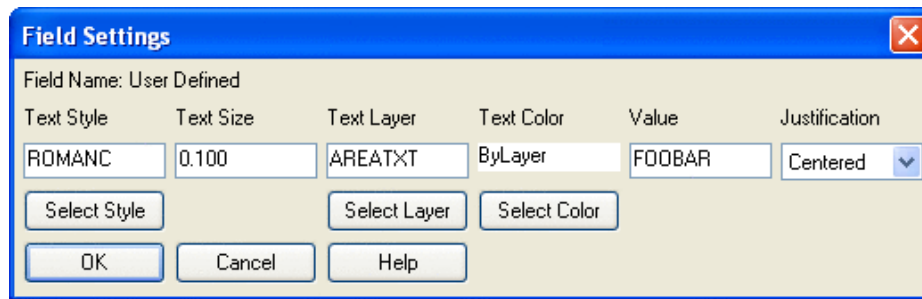


Field Settings Dialog: To control the appearance of the fields in the drawing, use the Edit button to edit the highlighted item in the Used Fields list, or double click on a field in the same list. This will call up the Field Settings Dialog.



User Defined: The Field "User Defined" can be added to place a custom fixed label in all areas. To control the value and appearance of the custom label in the drawing, use the Edit button to edit the "User Defined" item in the Used Fields list, or double click on a field in the same list. This will call up the Field Settings Dialog. In this case the "Value" setting becomes the custom label.

Scaled labels: The "Scaled Sq. Feet", "Scaled Sq. Meters", "Scaled Acres" and "Scaled Perimeter" fields can be used to include area labels that are scaled based on Drawing Setup "Report Scale Factor".



Text Style: This allows you to set a text style for the area labels. You can enter the name manually or use the Select Style button to call up a dialog which presents a list of known text styles.

Text Size: This value is multiplied by the horizontal scale to obtain the actual text size.

Text Layer: This allows you to assign a layer for the area text. You can enter the name manually or use the Select Layer button to call up a dialog which presents a list of known layers.

Text Color: This allows you to assign a color for the area text. Use the Select Color button to call up the standard color picker dialog. To use the default for the Text Layer, select ByLayer.

Prefix and Suffix: Although most area labeling uses the suffix, as in 1.25 Acres or 3.515 Hectares. But for those who prefer a prefix, as in Ac: 1.25, this routine can create that area labeling style automatically (see below for example of results of using a prefix with square feet and acres).

Justification: Use this to control whether the label field is left, centered or right justified.

+/-: This allows you to display + or - in the Prefix or Suffix of the area labels, or choose None.

Precision: Choose precision level for the currently selected field.

Below the Available and Used Fields lists the following items for further controlling area label generation:

Use Commas in Labels: This allows you to use commas in the area labels.

Use MText: Check this box to turn on the use of MText for area labels. If this is checked all area labels will be grouped into as few MText entities as possible. Area labels with different text styles, justification or layers will not be combined into the same MText entity.

Erase Previous Labels: When checked, previous area labels for the area being relabeled will be erased.

Label Placement: When auto placement of area labels is used, the labels can be placed either at the centroid of area or at the rear side. This is accomplished by selecting either the Center or Rear Side radio button, respectively. When Center is selected the user can choose to have the labels oriented according to the side lines of the area by checking the Align By Sides checkbox. When either Align By Sides or Rear Side is selected, the checkbox Flip Text for Twist Screen can be selected to have the label rotated 180 degrees to present it in the best reading orientation relative to the current Twist Screen rotation setting.

Draw Symbol Around Lot Description: When the Lot Description field is included in the Used Fields list, the user can check this checkbox to have a symbol drawn around the Lot Description field. When this box is checked, you specify the symbol name in the **Symbol Name** field or click on the current symbol (drawn to the right) to graphically choose the desired symbol. You specify the layer by entering the name in the **Layer** box or by clicking on the **Select** button to choose from a dialog that presents all known layers.

Symbol Buffer Offset: By default, the symbol will be automatically scaled according to the text length and size of the Lot Description value for the area. For additional control of symbol scaling, the user can enter a number in text size units in the Symbol Buffer Offset box. This value will be added to the automatically generated default scaling value.

Avoid Label Overlap: If this box is checked the area labels will be checked for overlaps after they are generated. Please see the Overlap Manager documentation for more information.

Overlap Settings: Click this button to go to the Avoid Label Overlaps dialog where you can review or modify the Overlap Manager settings. Please see the Overlap Manager documentation for more information.

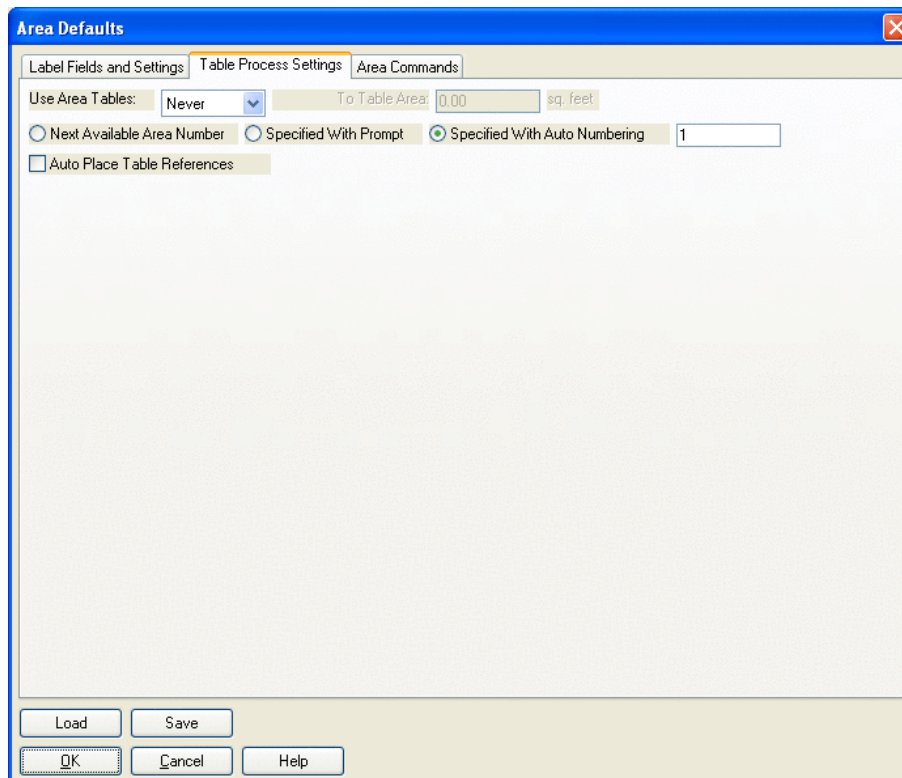


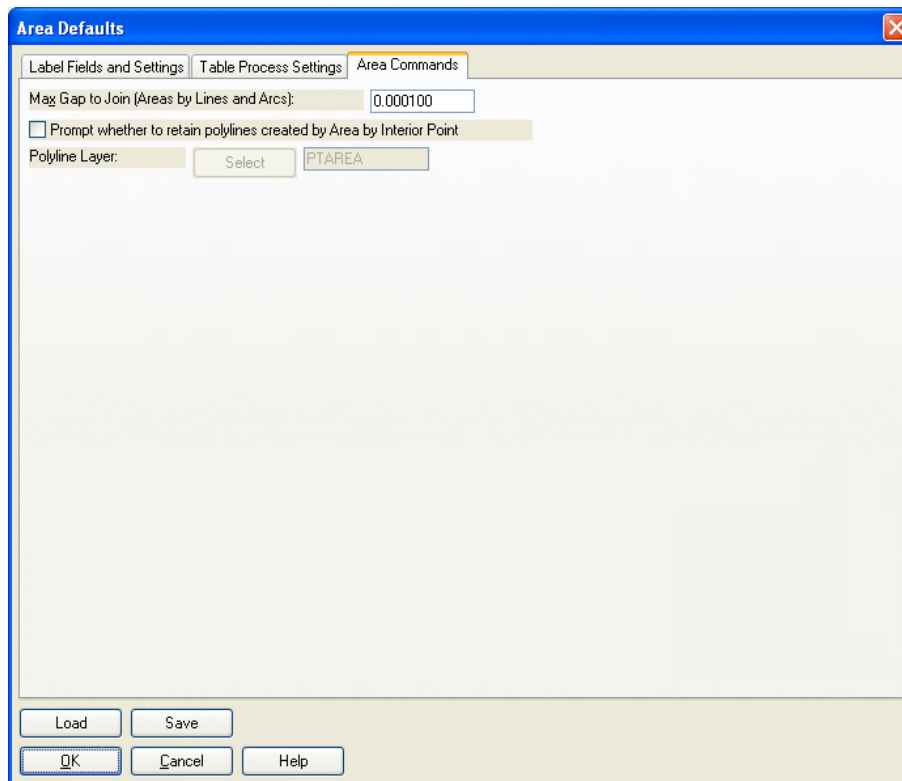
Table Process Settings Tab:

Use Area Tables: Use this control to determine whether area labels are sent to a table or not. Options are "Never", "Always", "By Scaler" or "Both". The By Scaler method uses the area table for areas less than the specified To Table Area. The Both method creates a table and also draws the labels within the area.

To Table Area: When the user has selected "By Scaler" in the "Use Area Tables" list this item is enabled. When "By Scaler" is selected and the area is less than this minimum, the area label is sent to a table.

Area Reference Numbering: There are three different methods for setting the reference number: **Next Available** will automatically use the lowest available number. **Specified With Prompt** will prompt you for a number for each area. **Specified with Auto Numbering** will automatically use the lowest available number starting with the specified number.

Auto Place Table References: When checked, will automatically place the area reference label according to the settings for the area labels as specified in the Label Field and Settings tab (see above). Otherwise you will be prompted to pick each label location manually.



Area Commands Tab:

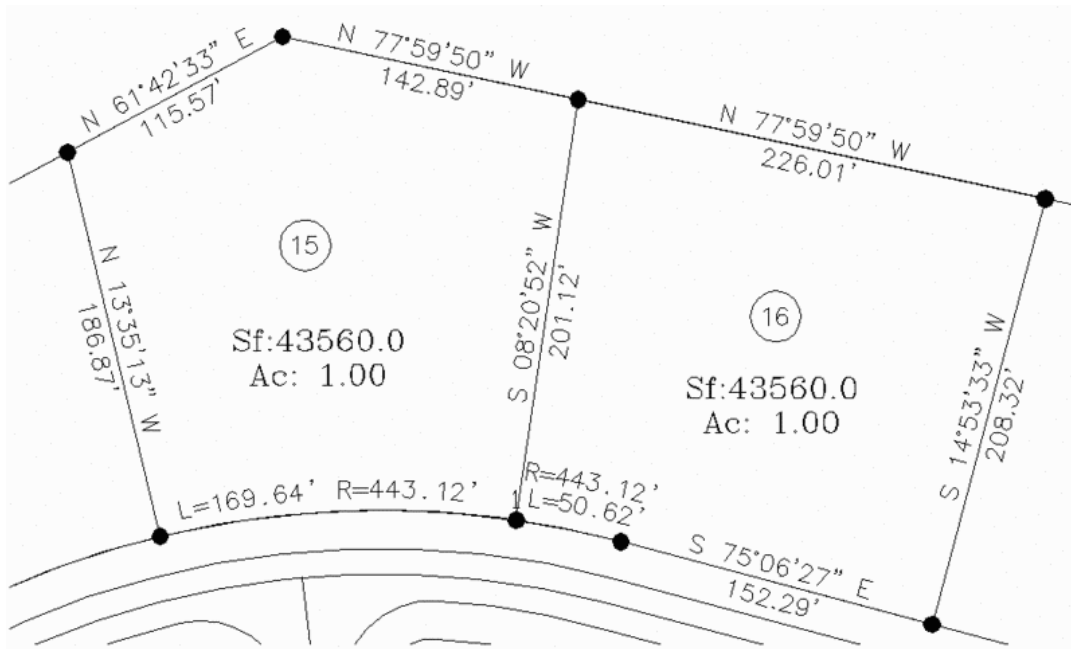
Max gap to join: You use this option during *Area by Lines & Arcs* command. When connecting lines and arcs that define the perimeter, the program will join endpoints if the distance between the two points is less than the specified gap. Otherwise the program will report an error and will not report an area.

Prompt whether to retain polylines created by Area by Interior Point: When checked the user will be asked whether to retain the polylines created by the "Area by Interior Point" command.

Polyline Layer: Will be enabled when "Prompt whether to retain polylines created by Area by Interior Point" is checked to allow the user to select the layer that any such created polylines will be placed in.

Load/Save: These buttons save and recall all the Area Default settings to a .ARS settings file.

Tip: Keep in mind that changes in Area Defaults, if changed from the Area/Layout pulldown menu, only apply to that work session. If changed within the Configure command, the changes apply to all new work sessions as well.



The results of using a prefix with square feet and acres

Pull-down Menu Location: Area/Layout

Keyboard Command: defarea

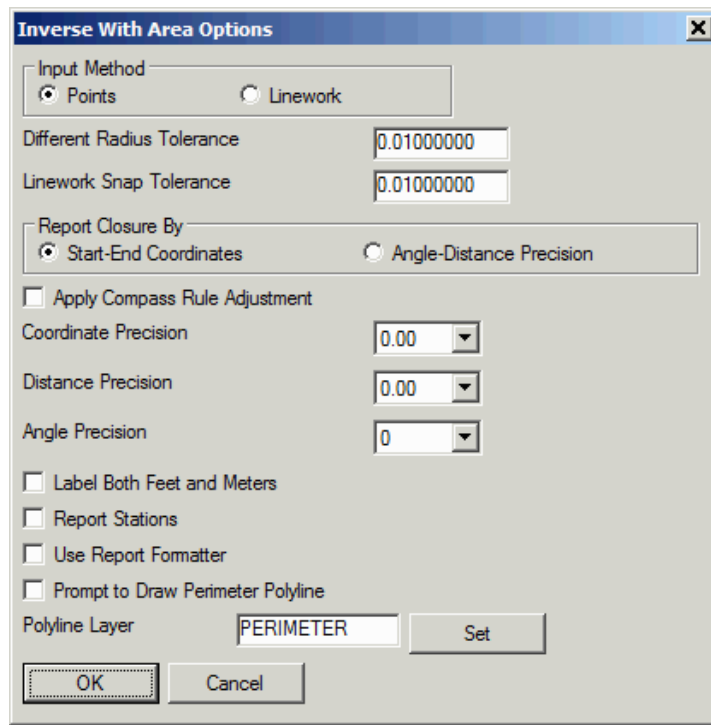
Prerequisite: None

Inverse with Area

This command generates a report of the angle and horizontal distance between a series of points and calculates the area of the closed figure defined by the points. The points can be entered individually or by selecting a point group or linework entities. To use a point group, type G for Group at the first prompt. To use a linework, type L for Line at the first prompt. For linework, a single closed polyline can be used or a series of connected lines, arcs and polylines that are selected one at a time.

Curve data can also be entered and reported. The points can be either picked on the screen, or entered by point number. You can also enter a range of point numbers (i.e. 1-9). The closure is reported using the total distance inversed, and the difference between the starting and ending points, as the closure error.

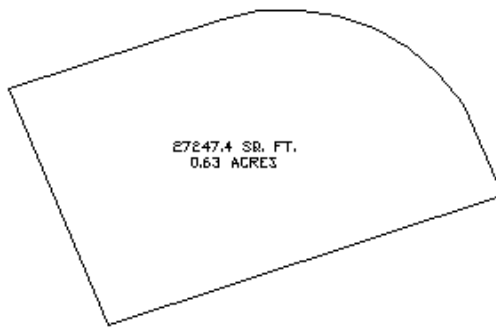
At the first command prompt, you can enter O for Options to bring up the command options. The **Input Method** determines the default input mode for the first command prompt. The **Different Radius Tolerance** checks that the distance between the PC and radius point and the PT and radius point match for curves. The **Linework Snap Tolerance** applies to the linework input mode and is used to check the distance between end points when connecting two linework entities. The Report Closed By choosing between using the difference between the starting and ending coordinates to calculate the closure error or angle and distance values in the report. The **Apply Compass Rule Adjustment** will adjust the perimeter as a closed loop for the closure error. The **Label Both Feet And Meters** is an option to report the distances in both feet and meters. The **Report Stations** option adds stations for each point into the report. The **Use Report Formatter** chooses between the standard report or customizing the report. You can also set the decimal precisions for the report and whether to report stations for the distances along the perimeter. This command creates a polyline of the figure which can be erased or kept in the drawing. The **Prompt To Draw Perimeter Polyline** and **Polyline Layer** settings apply to this polyline.



The area can be labeled in the drawing using the settings from the *Area Defaults* command. If you don't want to label the area, press Enter at the pick label point prompt.

Prompts

Options/Line/Group/<Pick Starting point or point number>: *pick a point*
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end): *pick a point*
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end): *R for radius*
Radius point number or pick point: *pick a point*
Curve direction [Left/<Right>]? *press Enter*
Pick End of Arc or point number (U-Undo,Enter to end): *pick a point*
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end): *pick a point*
Point number (R-RadiusPt,U-Undo,Enter to end): *pick a point*
Point number (R-RadiusPt,U-Undo,Enter to end): *pick a point*
Point number (R-RadiusPt,U-Undo,Enter to end): *press Enter*
SQ. FEET: 27247.4 SQ. YARDS: 3027.5 SQ. MILES: 0.0
ACRES: 0.63 PERIMETER: 668.35
Pick area label centering point: *pick a point*



Inverse with Area

CRD File> c:\data\newplat.crd

PNTNO	BEARING	DISTANCE	NORTHING	EASTING	STATION	DESC
903			4940.73	2490.40	0.00	StartPt
						S 48°43'58'' W 136.21 904

Closure Error Distance> 0.0000

Total Distance Inversed> 1008.07

AREA: 74664.6 SQ METERS

Pulldown Menu Location: Area/Layout

Keyboard Command: ia

Prerequisite: None

Area by Lines & Arcs

This command allows you to calculate the area of a perimeter or lot defined by lines, arcs, or polylines. Default settings for this command are set in *Area Defaults*. One of these settings is Max gap to join. If there is a gap greater than this value, the area is not reported, and the program will show where the gap is with a temporary X symbol. The area data shows up on the text screen. You can then choose to plot the area information to the drawing, or, by hitting *Enter*, just read it from the text screen.

Prompts

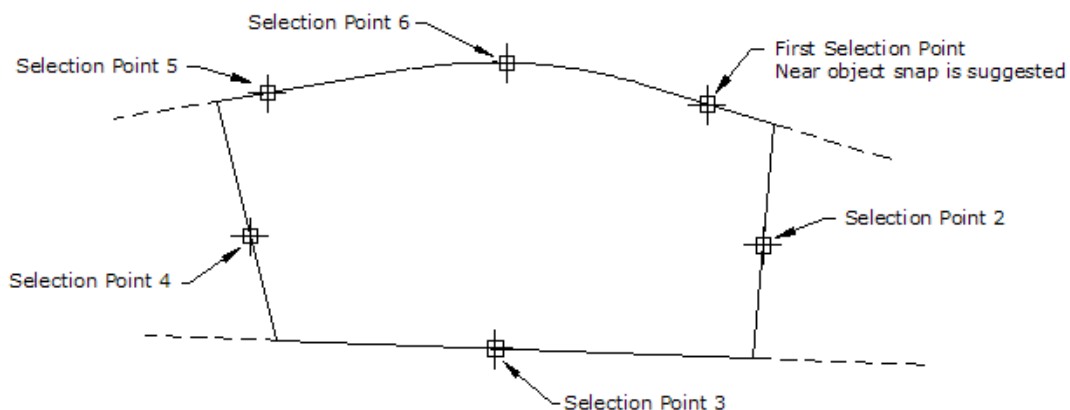
Select lines & arcs or polylines of perimeter for area calculation.

Select Objects: *select lines and arcs or polylines*

Lines and arcs are then joined together and the area calculated.

Pick area label centering point (Enter for none): *pick a point*

The area is then plotted at the point selected.



Pulldown Menu Location: Area/Layout
Keyboard Command: joinarea
Prerequisite: Lines, arcs, or polylines to define the area

Area by Interior Point

This command calculates and labels the area of the perimeter surrounding a picked interior point. The *Boundary Polyline* command is used to find the perimeter. Generally, this command will only work on closed or overlapping objects. Use *Area by Lines & Arcs* for other applications. The settings for the area label and for whether to prompt to create a closed polyline for the area are under the *Area Defaults* command.

Prompts

Pick point inside area perimeter: *pick a point*

Pick area label centering point (Enter for none): *pick a point*

The area is then plotted at the point selected.

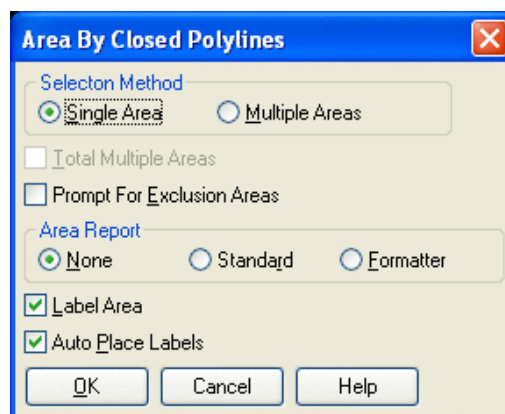
Pulldown Menu Location: Area/Layout

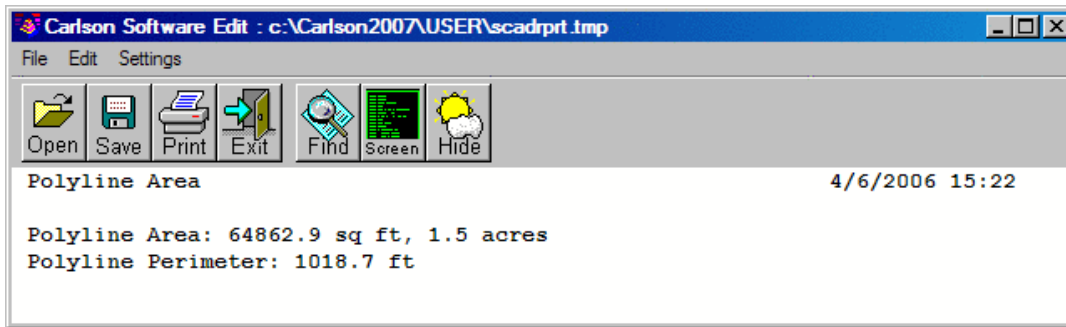
Keyboard Command: ptaarea

Prerequisite: Set Area Defaults

Area by Closed Polylines

This command will calculate and report the area of single area and multiple area closed polylines. In the case of multiple areas, the user can choose to have the areas totaled (**Total Multiple Areas**) into a single result or to generate data for each area separately. Area by Closed Polyline will also automatically find special Carlson attributes attached to the polyline, in addition to capturing the area itself. These attributes will appear in the report, which can be the standard report or which can be presented in the Report Formatter, which itself links to Excel and Access. For example, property names and owner names, as applied to a polyline using the Mine modules, will report out automatically using Area by Closed Polyline. The command "Draw Lots from File..." will apply "extended entity data" to the lot polylines, which includes the lot name, and this will also report out when using Area by Closed Polyline. In addition, lot names, or any interior text whatsoever, can be captured and included in the report. The plot of the area on-screen can be canceled if only the report is desired.





Prompts

Select Area Polyline: *select the area polyline*

SQ. FEET: 64862.9 SQ. YARDS: 7207.0 SQ. MILES: 0.0

ACRES: 1.5 PERIMETER: 1018.7

Pick area label centering point (Enter for none): *pick a location*

When auto-placing labels at the rear of lots or when aligning labels by the sides of the lot the user will also be prompted to pick one or more centerlines (**Select the Centerline Polyline**s). The routine will find the closest centerline and use this to determine the location of the front and back corners of the area.

When additional interior text is selected, the standard report will include that text:

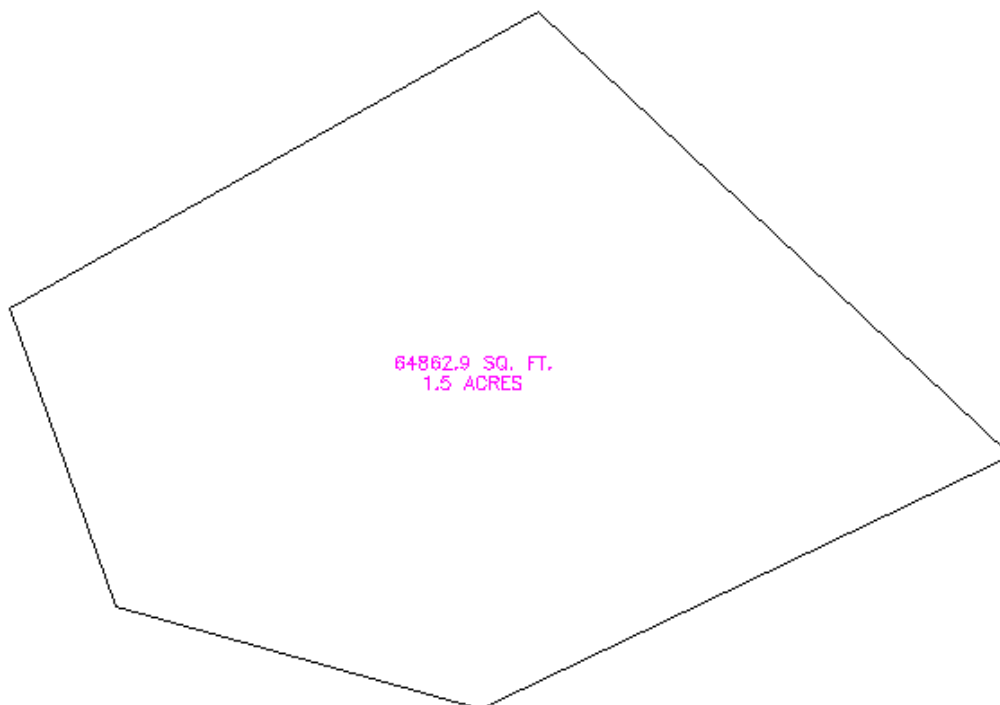
Polyline Area 11/17/2004 12:49

Polyline Area: 43560.0 sq ft, 1.00 acres

Polyline Perimeter: 838.35 ft

Text: 16 Sf: 43560.0; Ac: 1.00

In this case, the "16" refers to Lot 16, and appears in the report because the lot number and existing area labeling were selected along with the polyline for the lot.

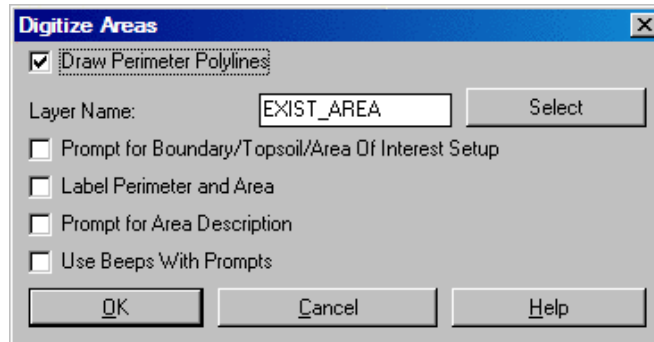


Pulldown Menu Location: Area/Layout

Keyboard Command: plarea
Prerequisite: Set Area Label Defaults

Digitize Areas

This command allows for digitizing areas. This routine includes an option for drawing perimeter polylines.



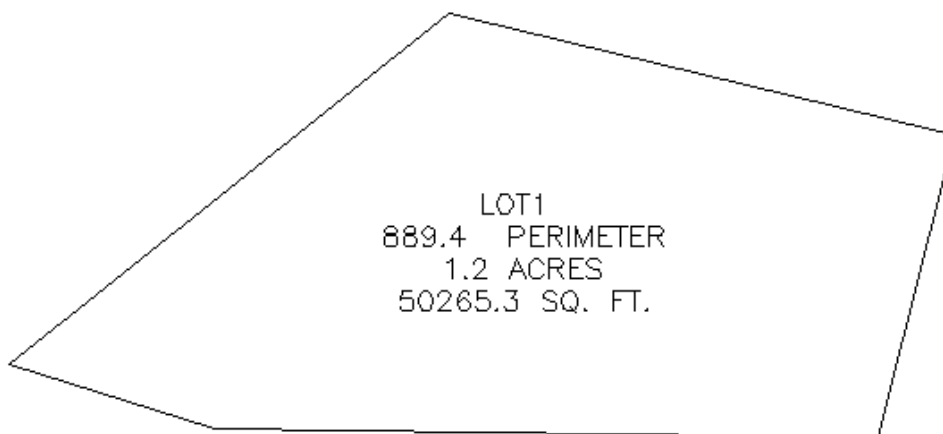
Pulldown Menu Location: Area/Layout
Keyboard Command: dig_area
Prerequisite: A digitizer

Label Last Area

This command will label the last area calculated with one of the Area commands in the manner defined in the *Area Defaults* dialog. The command prompts for a point where the label will be centered.

Prompts

SQ. FEET: 50265.3 SQ. YARDS: 5585.0 SQ. MILES: 0.0
ACRES: 1.2 PERIMETER: 889.4
Lot Description <2>: 1
Pick area label centering point (Enter for none): *pick a point*

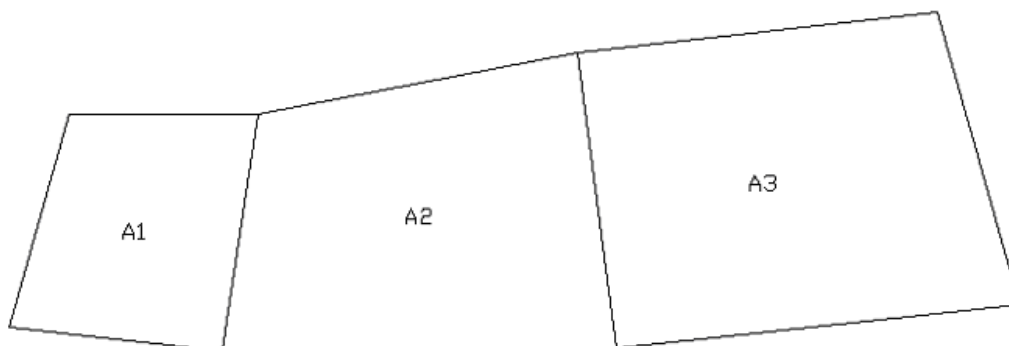


Pulldown Menu Location: Area/Layout
Keyboard Command: lastarea

Prerequisite: Set *Area Defaults*, and use one of the Area commands to calculate an area.

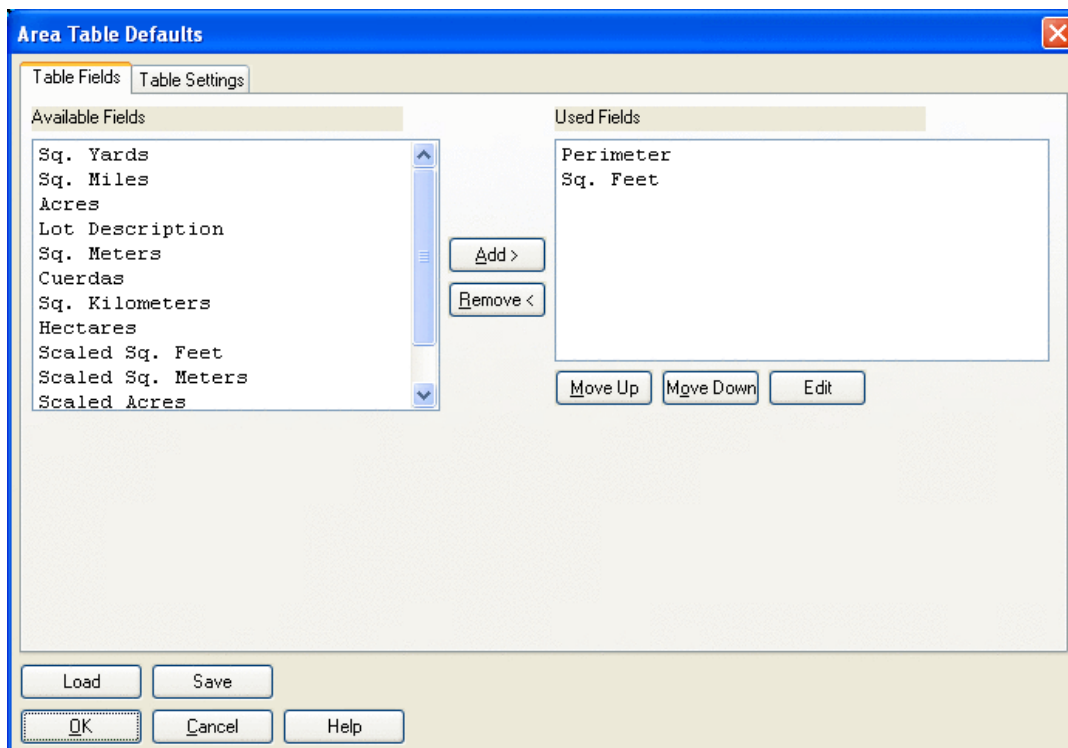
Area Table Defaults

This command allows you to specify table fields and format settings for area tables. Whether the Area Commands create an area table or label within the area is controlled by the Area Defaults command by the Use Area Tables setting. With the Area Defaults and Area Table Settings prepared, the various Area Commands will create tables according to the settings. When the Area By Closed Polyline routine is used to create the area table and the Link Linework With Labels option is on under Configure Carlson->General Settings, then the area table values are automatically updated when the polyline geometry is modified. Also, when using the Area By Closed Polyline command with the Lot Description field active for the table, the program prompts for an area description for each polyline. The rest of the area table fields are calculated from the polyline geometry.



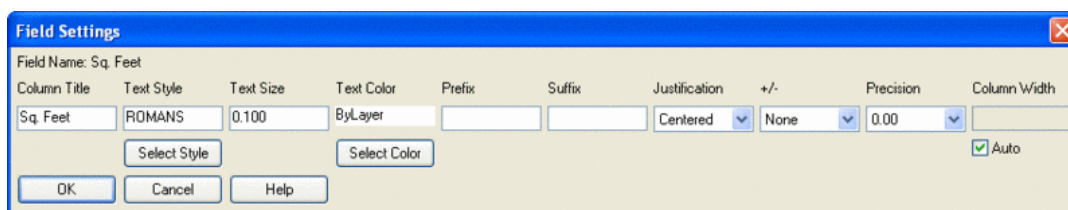
Area	Perimeter	Sq. Feet	Acres	Lot Description
A1	574.43	20212.71	0.46	Park
A2	835.85	41989.21	0.96	Lot 404
A3	910.72	50710.66	1.16	Lot 405

The Area Table Defaults dialog is divided into 2 tabs. The **Table Fields** tab brings up the Table Settings panel shown below. The area table option puts the area data in a table that is typically drawn outside the area and contains area data for multiple areas. Each row in the table has the data for one area and includes a reference number. The reference number is also labeled inside the area.



The **Table Fields** tab contains two listboxes which are used to control which of the area fields will appear in any table rows that are generated for areas. You use the Add and Remove buttons to control which fields will be included in area tables. You can also add to the Used Fields list by double-clicking on items in the Available Fields list. The area table will include the values in the order as specified in the Used Fields listbox. To change the order you use the Move Up and Move Down buttons.

Field Settings Dialog: To control the appearance of the fields in the table, use the Edit button to edit the highlighted item in the Used Fields list, or double click on a field in the same list. This will call up the Field Settings Dialog.



Column Title: This will be the tile name used for the field's column in the area table.

Text Style: This allows you to set a text style for the area labels. You can enter the name manually or use the Select Style button to call up a dialog which presents a list of known text styles.

Text Size: This value is multiplied by the horizontal scale to obtain the actual text size.

Text Layer: This allows you to assign a layer for the area text. You can enter the name manually or use the Select Layer button to call up a dialog which presents a list of known layers.

Text Color: This allows you to assign a color for the area text. Use the Select Color button to call up the standard color picker dialog. To use the default for the Text Layer, select ByLayer.

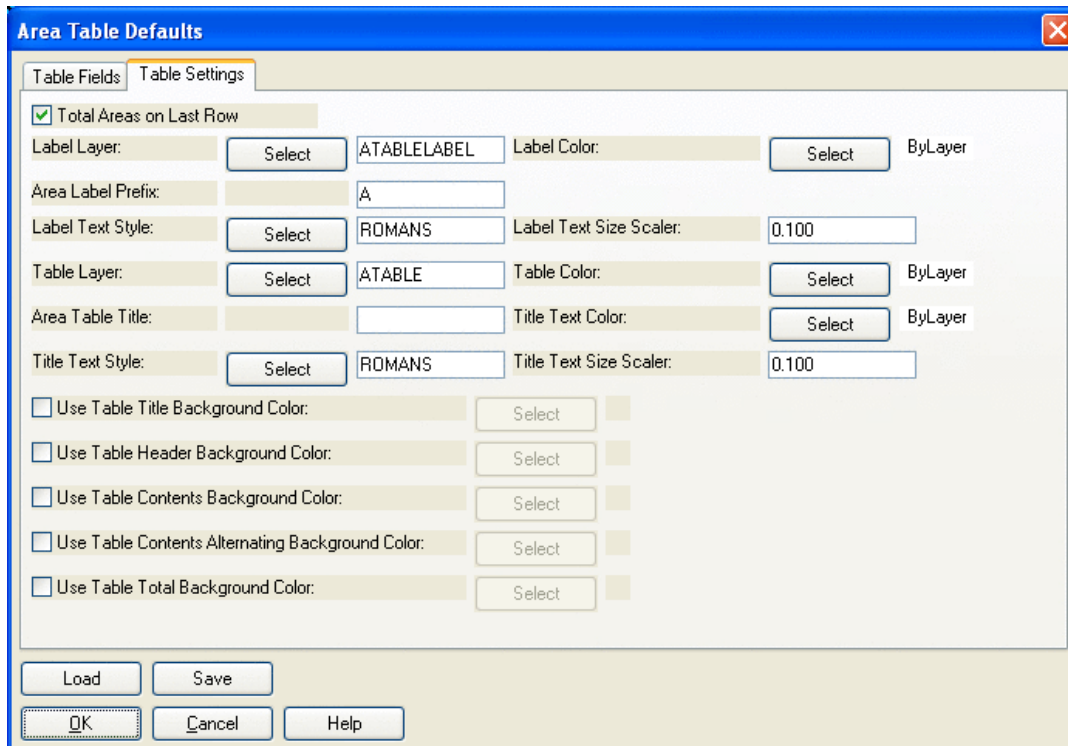
Prefix and Suffix: Although most area labeling uses the suffix, as in 1.25 Acres or 3.515 Hectares. But for those who prefer a prefix, as in Ac: 1.25, this routine can create that area labeling style automatically.

Justification: Use this to control whether the label field is left, centered or right justified.

+/-: This allows you to display + or - in the Prefix or Suffix of the area labels, or choose None.

Precision: Choose precision level for the currently selected field.

Column Width/Auto: The default behavior is that the column width is automatically set for best fit. The user can override this value by unchecking the Auto checkbox and setting the column with in text size units.



The **Table Settings** tab brings up the Table Settings panel shown above. The area table option puts the area data in a table that is typically drawn outside the area and contains area data for multiple areas. Each row in the table has the data for one area and includes a reference number. The reference number is also labeled inside the area.

Table Parameters:

Total Area on Last Row: Select this to have a total row placed at the bottom of the table which will contain the sum of all relevant table fields.

Label Layer: Use this to control the layer that the area table reference will be placed in. Use the **Select** button to pick from a list of all known layers.

Label Color: Use this to control the color of the area table reference. Use the **Select** button to pick from a color picker dialog. Select **ByLayer** to use the default color of the label layer.

Area Label Prefix: Use this to control the prefix of the area table references. Add a space after the prefix to have the prefix and the reference number separated by a space if desired.

Label Text Style: Use this to set the text style of the area table reference. Use the **Select** button to pick from a list of all known text styles.

Table Layer: This allows the user to set the layer that the table will be placed in. Use the **Select** button to pick from a list of all known layers.

Table Color: This allow the user to set the color of the grid lines of the table. Use the **Select** button to pick from a color picker dialog. Select **ByLayer** to use the default color of the table layer.

Area Table Title: To add a title row as the first row of the area table, enter a table title here.

Title Text Color: This allow the user to set the color of the table title. Use the **Select** button to pick from a color picker dialog. Select **ByLayer** to use the default color of the table layer.

Title Text Style: Use this to set the text style of the table title. Use the **Select** button to pick from a list of all known text styles.

Title Text Size: Use this to control the size of the table title text.

Background Colors: The area table is broken into 5 zones in respect to background color. Each zone can have its

own unique background color. The zones are Title, Header, Contents1, Contents2 and Total. To set a background color for each zone, first the respective "Use Table...Background Color" box must be checked. This enables the **Select** button, which is used to pick the respective background color from a color picker dialog. For the Contents zone all contents rows can either have the same background color, or by setting up an "Alternating Background Color", rows will have alternating colors.

Pulldown Menu Location: Area/Layout->Area Tables

Keyboard Command: defatab

Prerequisite: None

New Area Table

This command draws the column header labels for the Area Table commands. When prompted for the starting point, the user may enter a coordinate or pick a point on the screen. This table becomes the active area table. Any new area table entries will be added to this table until another table is created or the active table is changed with the atabset command (menu item Area/Layout> Area Tables> Set Active Table).

Area	Perimeter	Sq. Feet	Acres	Lot Description
------	-----------	----------	-------	-----------------

Prompts

Starting point of area table: *pick point*

Pulldown Menu Location: Area/Layout> Area Tables> Create New Table

Keyboard Command: atabnew **Prerequisite:** None

Set Active Area Table

This command allows the user to change the active area table. The table selected becomes the active area table. Any new area table entries will be added to this table until another table is created or the active table is changed with another invocation of this command.

Prompts

Select active Table: *pick area table*

Pulldown Menu Location: Area/Layout> Area Tables> Set Active Table

Keyboard Command: atabset

Prerequisite: None

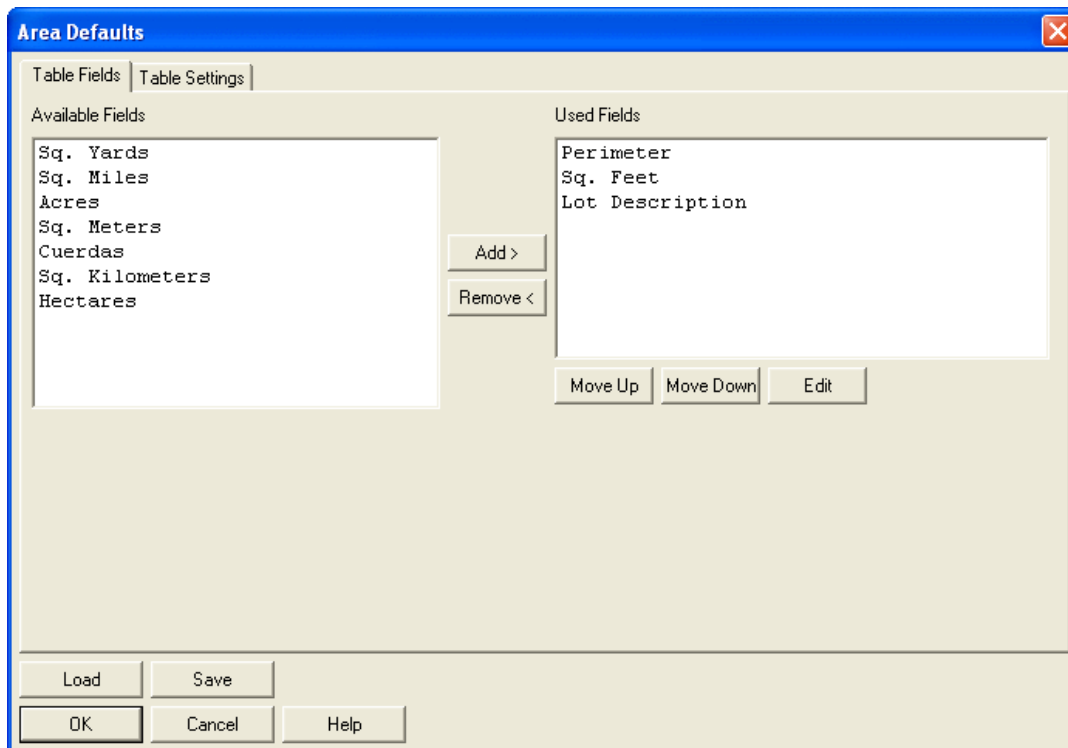
Edit Area Table Properties

This command allows the user to edit the properties of an area table.

Prompts

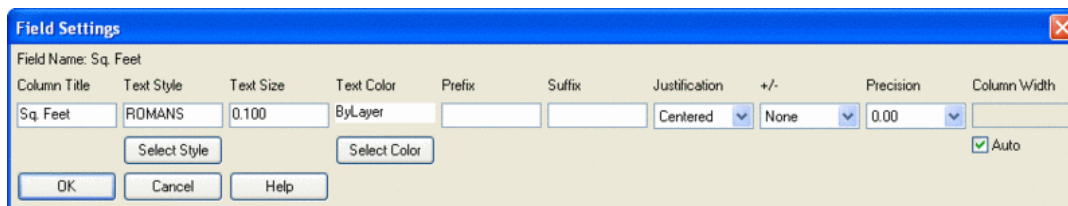
Select an area table to modify: *pick an area table*

After picking an area table the Area Defaults dialog will be displayed. Here you can change the settings of the selected table. The changes will be reflected once the user selects the OK button.



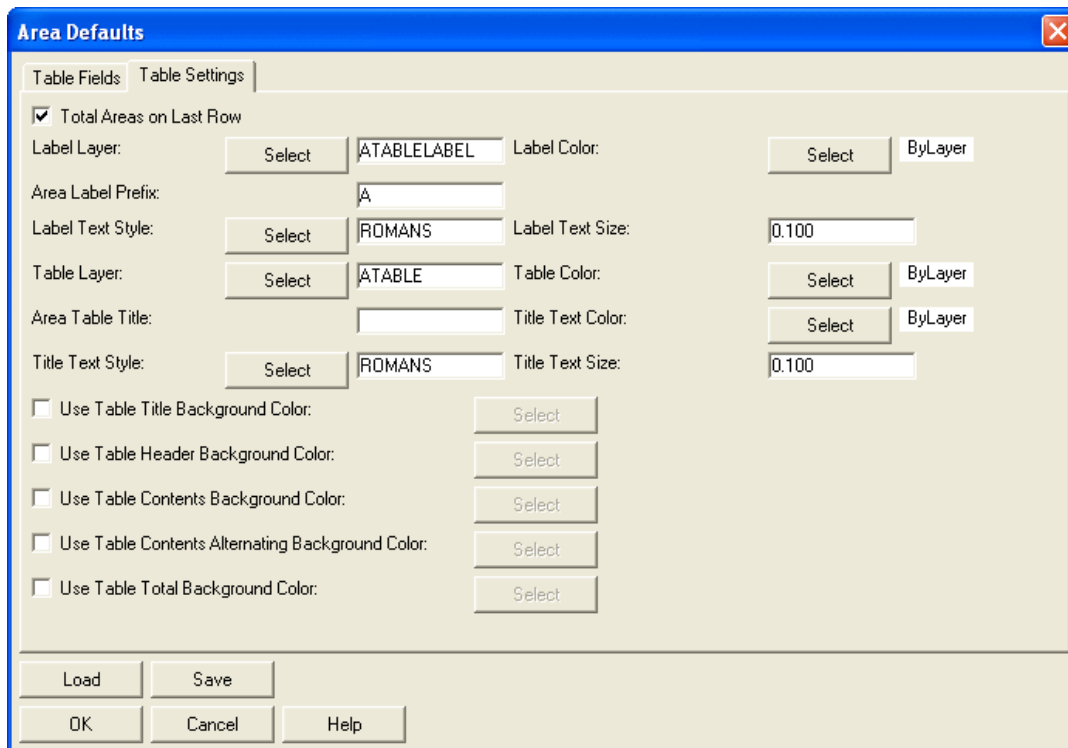
The Table Fields tab contains the Available Fields and Used Fields listboxes which are used to control which of the possible ten area fields will be used in the area table. You use the Add and Remove buttons to control which fields will be included in the table. You can also add to the Used Fields list by double-clicking on items in the Available Fields list. The area label will include the values in the order as specified in the Used Fields listbox. To change the order you use the Move Up and Move Down buttons.

Field Settings Dialog: To control the appearance of the fields in the table, use the Edit button to edit the highlighted item in the Used Fields list, or double click on a field in the same list. This will call up the Field Settings Dialog.



- Column Title:** This will be the tile name used for the field's column in the area table.
- Text Style:** This allows you to set a text style for the area labels. You can enter the name manually or use the Select Style button to call up a dialog which presents a list of known text styles.
- Text Size:** This value is multiplied by the horizontal scale to obtain the actual text size.
- Text Layer:** This allows you to assign a layer for the area text. You can enter the name manually or use the Select Layer button to call up a dialog which presents a list of known layers.
- Text Color:** This allows you to assign a color for the area text. Use the Select Color button to call up the standard color picker dialog. To use the default for the Text Layer, select ByLayer.
- Prefix and Suffix:** Although most area labeling uses the suffix, as in 1.25 Acres or 3.515 Hectares. But for those who prefer a prefix, as in Ac: 1.25, this routine can create that area labeling style automatically.
- Justification:** Use this to control whether the label field is left, centered or right justified.
- +/-:** This allows you to display + or - in the Prefix or Suffix of the area labels, or choose None.
- Precision:** Choose precision level for the currently selected field.
- Column Width/Auto:** The default behavior is that the column width is automatically set for best fit. The user can override this value by unchecking the Auto checkbox and setting the column with in text size units.

The **Table Settings** tab brings up the Table Settings panel shown below.



Total Area on Last Row: Select this to have a total row placed at the bottom of the table which will contain the sum of all relevant table fields.

Label Layer: Use this to control the layer that the area table reference will be placed in. Use the **Select** button to pick from a list of all known layers.

Label Color: Use this to control the color of the area table reference. Use the **Select** button to pick from a color picker dialog. Select ByLayer to use the default color of the label layer.

Area Label Prefix: Use this to control the prefix of the area table references. Add a space after the prefix to have the prefix and the reference number separated by a space if desired.

Label Text Style: Use this to set the text style of the area table reference. Use the **Select** button to pick from a list of all known text styles.

Table Layer: This allows the user to set the layer that the table will be placed in. Use the **Select** button to pick from a list of all known layers.

Table Color: This allow the user to set the color of the grid lines of the table. Use the **Select** button to pick from a color picker dialog. Select ByLayer to use the default color of the table layer.

Area Table Title: To add a title row as the first row of the area table, enter a table title here.

Title Text Color: This allow the user to set the color of the table title. Use the **Select** button to pick from a color picker dialog. Select ByLayer to use the default color of the table layer.

Title Text Style: Use this to set the text style of the table title. Use the **Select** button to pick from a list of all known text styles.

Title Text Size: Use this to control the size of the table title text.

Background Colors: The area table is broken into 5 zones in respect to background color. Each zone can have its own unique background color. The zones are Title, Header, Contents1, Contents2 and Total. To set a background color for each zone, first the respective "Use Table...Background Color" box must be checked. This enables the **Select** button, which is used to pick the respective background color from a color picker dialog. For the Contents zone all contents rows can either have the same background color, or by setting up an "Alternating Background Color", rows will have alternating colors.

Load/Save: These buttons save and recall all the Area Default settings to a .ARS settings file.

Tip: Keep in mind that changes made here only apply to the selected table. If properties are changed within the Configure command, the changes apply to all new work sessions as well

Pulldown Menu Location: Area/Layout> Area Tables> Edit Properties

Keyboard Command: atabedit

Prerequisite: An area table

Remove Area Table Rows

This command allows the user to remove rows from an area table. The routine will remove both the table row and the table reference label from the drawing.

Prompts

Select a table row to delete: *pick area table row*

Consolidate table [<Yes>/No]? If consolidation is chosen, row numbers will be renumbered to close up the gap created by this deletion. Consolidation will also update all relevant area table references in the drawing. If the user chooses not to consolidate the table at this time, the atabfix command (menu item Area/Layout> Area Tables> Consolidate Table) can be used at any time to perform consolidation.

Area	Perimeter	Sq. Feet
A1	470.70	9157.63
A2	629.20	15572.47
A3	542.18	16810.50
Total		41540.60

The drawing above shows the table before row removal. In the drawing below, row 2 has been deleted without table consolidation.

Area	Perimeter	Sq. Feet
A1	470.70	9157.63
A3	542.18	16810.50
Total		25968.13

The drawing below shows the results of deleting the same row 2, only this time the user has chosen to perform table consolidation.

Area	Perimeter	Sq. Feet
A1	470.70	9157.63
A2	542.18	16810.50
Total		25968.13

Pulldown Menu Location: Area/Layout> Area Tables> Remove Row

Keyboard Command: atabel

Prerequisite: None

Consolidate Area Table

This command allows the user to renumber area tables to eliminate numbering gaps left as the result of row deletions or other means.

Prompts

Select a table to consolidate:: *pick area table*

Row numbers will be renumbered to close up the gaps in the selected area table. Consolidation will also update all relevant area table references in the drawing.

Area	Perimeter	Sq. Feet
A1	470.70	9157.63
A3	542.18	16810.50
Total		25968.13

The drawing above shows the table before row removal. The drawing below shows the results of consolidating this table.

Area	Perimeter	Sq. Feet
A1	470.70	9157.63
A2	542.18	16810.50
Total		25968.13

Pulldown Menu Location: Area/Layout> Area Tables> Consolidate Table

Keyboard Command: atabfix

Prerequisite: None

Tag Area Descriptions

This command is used to assign a description to a closed polyline. The description is stored with the polyline in the drawing. This description is used for reports in routines like Area By Closed Polyline.

Prompts

Select polyline for area description: *pick a polyline*

Area description <AREA1>: *West Pond*

Pulldown Menu Location: Area/Layout > Area Descriptions

Keyboard Command: tag_area_desc

Prerequisite: A closed polyline.

Identify Area Descriptions

This command reports area descriptions for the selected polylines. There are two methods. The Pick method reports the area description for one selected polyline at a time. The Search method scans the whole drawing and highlights polylines with area descriptions.

Prompts

Pick polylines to check or search drawing [<Pick>/Search]: *press Enter*

Select area description polyline: *pick a polyline*

Description: West Pond

Select area description polyline (Enter to end): *press Enter*

Pulldown Menu Location: Area/Layout > Area Descriptions

Keyboard Command: id_area_desc

Prerequisite: A polyline with a tagged area description.

Untag Area Descriptions

This command removes an area description that has been assigned to a polyline.

Prompts

Select polylines to remove area description from.

Select entities: *pick area polylines*

Cleared 10 area descriptions.

Pulldown Menu Location: Area/Layout > Area Descriptions

Keyboard Command: untag_area_desc

Prerequisite: A polyline with a tagged description.

Area Descriptions By Interior Text

This command is used to assign a description to a closed polyline. The description is stored with the polyline in the drawing. This description is used for reports in routines like Area By Closed Polyline.

The program prompts to select closed polylines and text. For each closed polyline, the program looks for a text label inside the polyline and assigns that text as the description for the polyline.

Prompts

Select area polylines and text.

Select objects: *pick a polyline and text*

Pulldown Menu Location: Area/Layout > Area Descriptions

Keyboard Command: area_desc_text

Prerequisite: Text and closed polyline

Hinged Area

This command can be used to determine the dimensions of a figure when the area is fixed and three or more sides are known. The figure can be defined by a closed polyline or by picking the known points and curves. The command then prompts for the area to be solved for (in square units and acres).

Prompts

Define area by points or closed polyline [Points/<Linework>]? press Enter

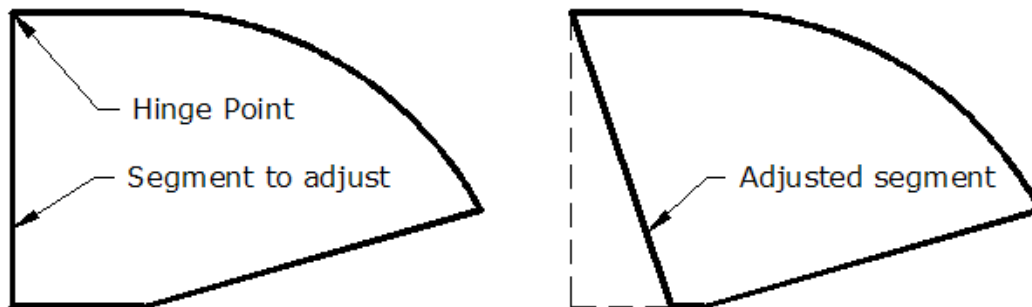
Select polyline segment to adjust: *select a polyline segment*

Select hinge point [endp]: Move the cursor around to find a hinge point.

Keep existing polyline [Yes/<No>]? N

Area: 47104.31 S.F, 1.0814 Acres

Remainder/Acres/<Enter target area (s.f.)>: 48000



Polyline method

Pulldown Menu Location: Area/Layout

Keyboard Command: harea

Prerequisite: A closed polyline or at least one known side. Two direction lines should be drawn.

Sliding Side Area

This command adjusts one side of a polyline to meet a specified area. The existing area can be defined by a closed polyline or by picking each point in the perimeter. The desired area can be entered in either square feet or acres. The area to adjust must be represented by a closed polyline. The program moves the selected segment of the polyline in or out. There are a few methods for defining the direction of the adjusted segment. With the Selected method, the original direction of the segment is maintained. The Line method prompts to pick another line segment to define the direction. The Angle method uses an entered angle for the direction. The Points method prompts for two points to define the direction.

Prompts

Define area by points or closed polyline [Points/<Linework>]? press Enter

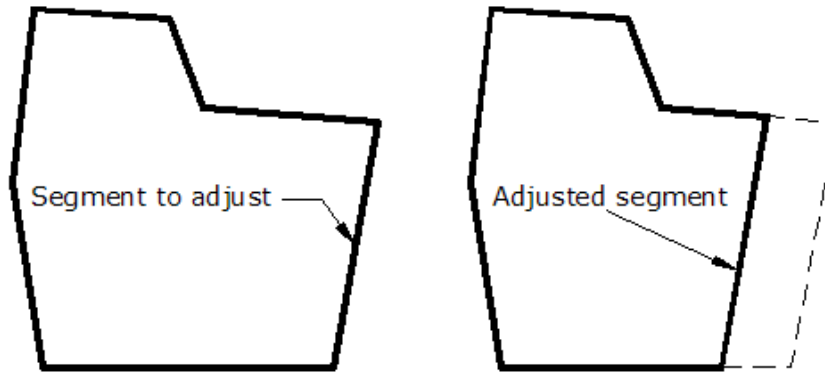
Select polyline segment to adjust: *pick a point on a closed polyline*

Keep existing polyline [Yes/<No>]? press Enter

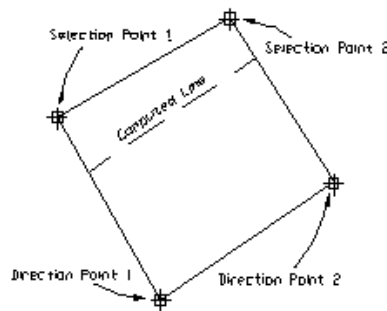
Define new line by selected line, another line, angle or points [<Selected>/Line/Angle/Points]? press Enter

Area: 176044.14 S.F, 4.0414 Acres

Remainder/Acres/<Enter target area (s.f.)>: 17800



Linework Polyline method:
Original perimeter polyline on left, adjusted perimeter on right



Points method

Pulldown Menu Location: Area/Layout

Keyboard Command: ssarea

Prerequisite: A closed perimeter polyline

Area Radial from Curve

This command swings a line radial from a curve to reach a predetermined area. The existing area can be defined by polylines or by picking each point on the perimeter. For the point method, the curve to radiate from should be the last entity selected when defining the figure. For the polyline method, front and back polylines are used. The computed line goes perpendicular from the front polyline and intersects the back polyline. This line is moved to find the target area. Both ends of the front and back polylines are connected to close the area. The options for the polyline method are set in the dialog shown.

Prompts

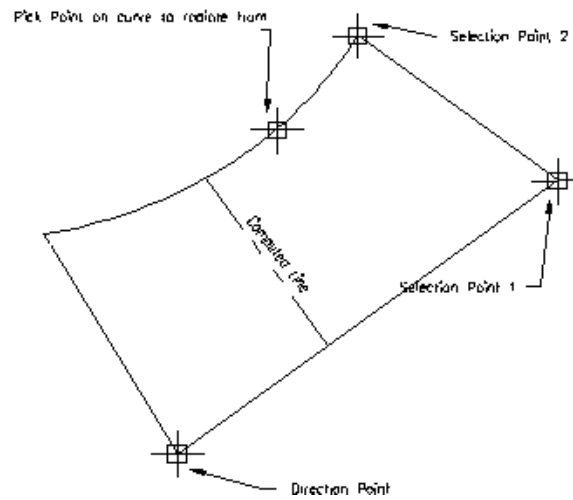
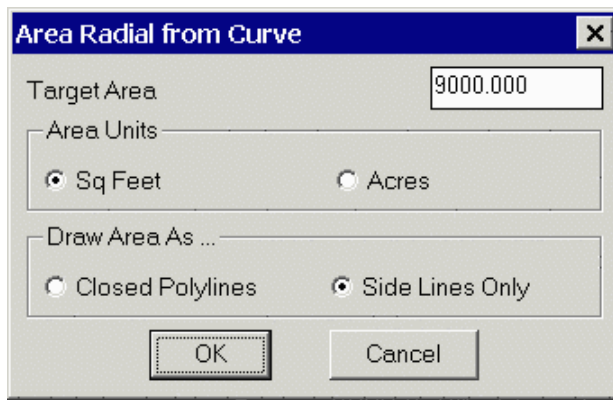
Define area by points or closed polyline [Points/<Linework>]? press Enter

Area Radial from Curve dialog Make choices and click OK.

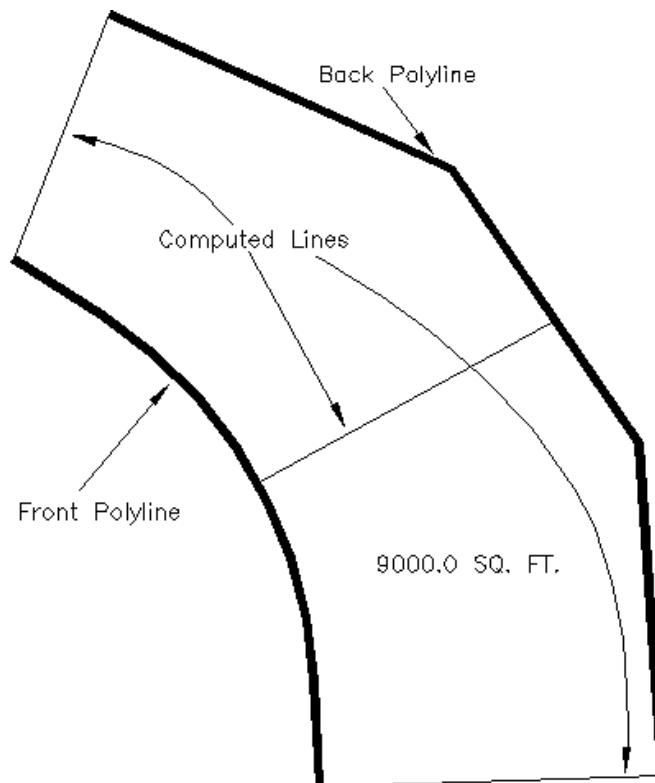
Select curve to radiate from: *pick the curve*

Select back polyline: *pick the back polyline*

Lot Area: 9000.00 S.F., 0.2066 Acres



Point Method



Polyline Method

Pulldown Menu Location: Area/Layout
Keyboard Command: arearc

Prerequisite: An existing area defined by points or polylines

Bearing Area Cutoff

This feature allows you to cut a predetermined area from a closed figure using a cut-off line having a specified bearing. The boundary intersected by the cut-off bearing line can be either a straight line or arc.

Enter area in ACRES [Sq. Feet/Done] <0.000000>: Enter the number of acres contained within the cut-off area.

To change from acres to square feet, type S and <Enter>.

Note: if units are set to meters, the prompt will be:

Enter area in HECTORS [Square meters/Done].

Enter bearing of cutoff line <100.000000>: Enter the bearing of the cut-off line through the property using Qdd.mmsss format.

where:

Q = quadrant (1 = NE, 2 = SE, 3 = SW, 4 = NW)

dd = degrees

mm = minutes

sss = seconds (The third s indicates that, if desired, you can optionally specify seconds to the nearest 0.1 second)

Note: Trailing zeros need not be entered.

Place area to right or left of bearing line [Right/Left] <R>:

"Looking" in the direction of the cut-off bearing allows you to determine which side is left or right. Type R and <Enter> or just <Enter> for right. Type L and <Enter> for left.

Method of defining the overall area to be divided [C&G Point-group/Manual-entry] <M>:

Type "P" and Enter if you wish to use a point group to specify the overall area or type "M" and Enter (or just press Enter) to specify the overall area interactively.

Defining the overall area using a C&G Point Group:

If you have a C&G Point Group that defines the area to be divided and you choose to use the point group option, you will then be asked to use a file dialog to browse to the point group file and select it.

Defining the overall area manually:

If you choose to type or pick the points defining the overall area individually, you will see the prompt:

Enter point ID or pick graphically [cLockwise/ccW/Polyline]: Specify a point ID or begin a curve by typing L or W. Type P and enter to pick a polyline.

Note: if you choose to pick a polyline, it must be a closed polyline and all its vertices must have coordinates matching points found in the coordinate file.

When specifying individual points, move around the parcel and pick or type in the points, in order, to define the overall area involved.

After all points have been entered, press Enter to end point input.

Calculate the Cut-off Line

No matter which method you use to specify the property being divided, once the overall tract is specified, the cut-off line is calculated and the points at which the cut-off line intersects the tract boundaries are saved.

The **Saving Point** dialog (below) will be shown for each intersection point.

The image shows a 'Saving Point' dialog box with the following fields and buttons:

- Coordinate File: C:\Program Files\CGSurvey 19 for IntelliCAD\Data\CGDEMO.cgc
- Point ID: 1122
- Coordinates:
 - Northing: 195.62264
 - Easting: 156.49811
 - Elevation: <No Elev>
- Buttons: No Elevation, From Point ID
- Description: (empty field)
- Code: (empty field)
- Buttons: Cancel, OK

Click the **OK** button to save the intersection point.

Depending on your settings for **Auto Line Plot** and **Auto Point Plot** on the **Graphics** tab of the **C&G Options** dialog box, you may see both the points and the cut-off line drawn on the screen.

This process can be repeated as many times as is necessary to further divide the overall area or to divide another area.

Press <Esc> or "D" at the **Enter area...** prompt to end the command.

Prompts

Enter area in ACRES [Sq. Feet/Done] <0.000000>:

Enter the number of acres contained within the cutoff area or type "S" and Enter to use square feet or "D" and Enter when done.

Enter bearing of cutoff line <100.000000>: Enter the bearing of the cut-off line through the property.

Place area to right or left of bearing line [Right/Left] <R>:

Type "R" and Enter or just Enter for right of line. Type "L" and Enter for left. The direction of the cut-off bearing determines which side is left or right.

Method of defining the overall area to be divided [C&G Point-group/Manual-entry] <M>:

Type "P" and Enter if you wish to use a point group to specify the overall area or type "M" and Enter (or just press Enter) to specify the overall area interactively.

for Manual entry:

Enter point ID or pick graphically [cLockwise/ccW/Polyline]: Specify the point ID or begin a curve by typing L or W. Type P and enter to pick a polyline.

Pulldown Menu Location: Area/Layout

Keyboard Command: baco

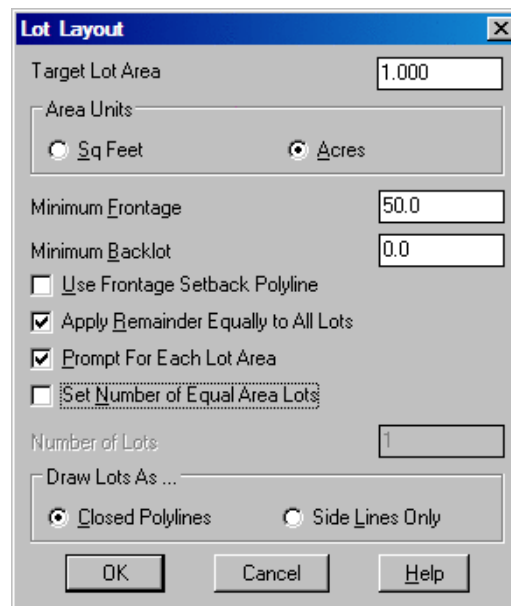
Prerequisite: Coordinate file.

Lot Layout

This command draws lots based on a front and back polyline. Starting from the front polyline, the program calculates two lot side lines perpendicular from the front polyline that intersect the back polyline and create the specified lot size. Lots are created along the front polyline in the order that the front polyline is drawn. If the front polyline needs to be reversed, use the *Reverse Polyline* command found on the Edit menu. The direction of the back polyline does not matter. The lots can be drawn as closed polylines or just the lot sides can be drawn. There is also an option to automatically create all the possible lots at the specified area between the front and back polylines or to prompt for each 0.4 acre lot.

In prompt mode, the program reports the remaining area between the front and back polylines and then asks for the lot size. The lot size can be specified either by area or frontage along the front polyline.

The lots are sized to meet the specified area and also meet the minimum frontage and backlot distances. The program starts by checking the lot area at the minimum distances. If this area is greater than the target, then the lot is drawn at the minimum distance and the resulting lot area will be greater than the target area. Otherwise the program will increase the frontage until the lot reaches the exact target area. The Use Frontage Setback Polyline option allows you to use another polyline besides the actual frontage polyline for the minimum frontage indicator. Typically, this Frontage Setback Polyline would be offset a set amount from the actual frontage polyline.



Prompts

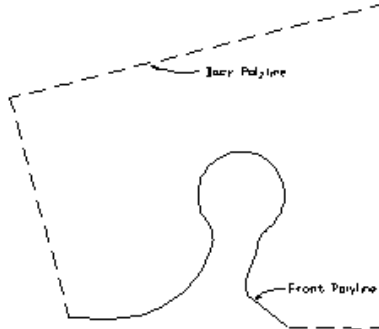
Lot Layout dialog

Select front polyline: *pick a polyline*

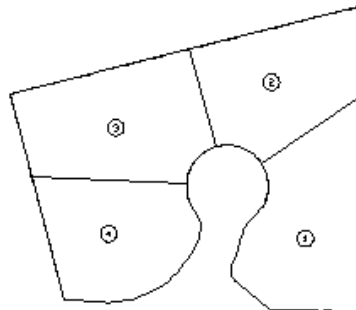
Select back polyline: *pick a polyline*

With prompt for each lot active:

Area remaining: 160326.88 S.F, 3.6806 Acres
Quit/Frontage/Enter lot area (Acres) <1.2269>: I
Area remaining: 116766.88 S.F, 2.6806 Acres
Quit/Frontage/Enter lot area (Acres) <1.0000>: F
Enter Frontage <50.00>: 75
Lot Area: 37807.50 S.F., 0.8679 Acres
Area remaining: 78959.38 S.F, 1.8127 Acres
Quit/Area/Enter frontage <50.00>: A
Quit/Frontage/Enter lot area (Acres) <1.0000>: press Enter
Area remaining: 35399.38 S.F, 0.8127 Acres
Quit/Frontage/Enter lot area (Acres) <1.0000>: Q



Polylines for Lot Layout
 The Front Polyline goes from right to left

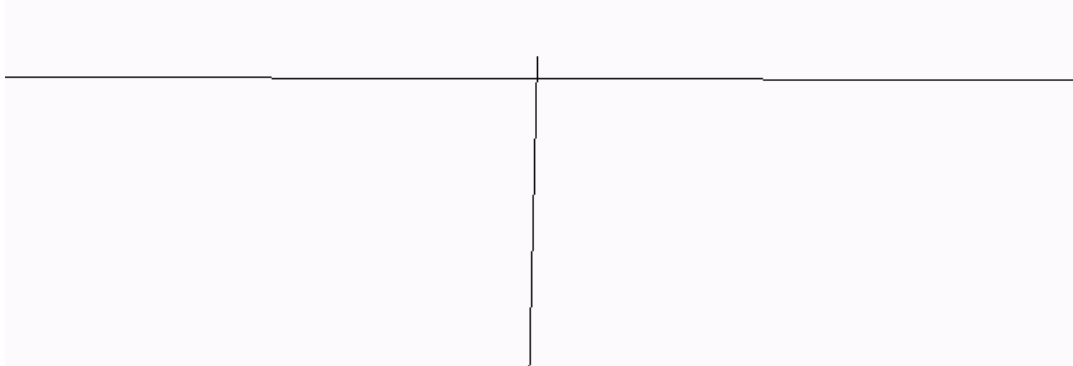
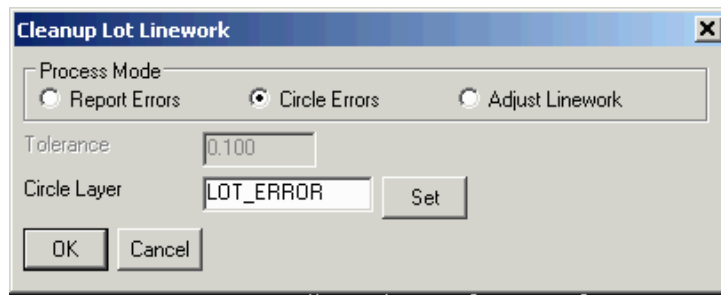


Resulting lots numbered using Sequential Numbers

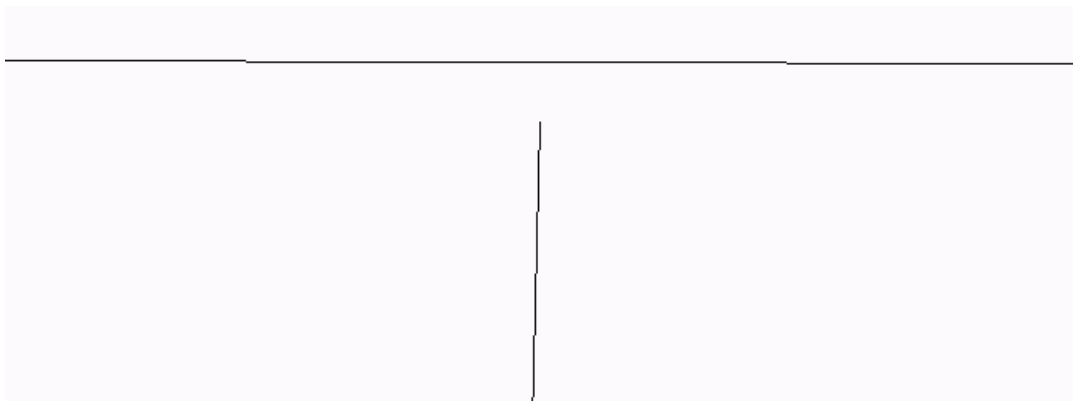
Pulldown Menu Location: Area/Layout
Keyboard Command: lotlay
Prerequisite: A frontage polyline and a backlot polyline.

Cleanup Lot Linework

This command finds any overshoots or undershoots in the lot linework. The Report Errors mode will report the location of the errors in the standard report viewer. The Circle Errors mode will draw circles on the specified layer around the errors. Then you can use CAD functions to review these errors and make edits. The Adjust Linework mode makes the program automatically adjust the linework coordinates to remove the errors as long as the coordinates move less than the specified Tolerance.



Example of overshoot where lot line goes past intersection



Example of undershoot where lot line falls short of intersection

Prompts

Cleanup Lot Linework dialog

Select polylines to process.

Select objects: *pick polylines*

Pulldown Menu Location: Area/Layout > Lot Layout Utilities

Keyboard Command: lotclean

Prerequisite: Linework

Set Linework Angles To Nearest Second

This command adjusts the coordinates of lines and polylines to set their bearings to the nearest second. This routine eliminates decimal seconds for the linework. Here's an example inverse showing decimal seconds on a line before running this routine.

```
Northing(Y) Easting(X) Elev(Z)
7054276.3676 11519401.0186 0.0000
7054104.6344 11519556.5360 0.0000
Bearing: S 42d09'47.5207" E Horizontal Distance: 231.6850115
```

Here's the inverse showing the bearing to the nearest second on the line after running this routine:

```
Northing(Y) Easting(X) Elev(Z)
7054276.3673 11519401.0182 0.0000
7054104.6344 11519556.5360 0.0000
Bearing: S 42d09'48.0000" E Horizontal Distance: 231.6850115
```

Prompts

Select lines and polylines to process.
Select objects: *pick lines and polylines*
Adjusted 1 polylines.
Maximum distance shift 0.000538

Pulldown Menu Location: Area/Layout > Lot Layout Utilities
Keyboard Command: pl2sec
Prerequisite: Linework

Set Linework Intersections To Perpendicular

This command processes a set of polylines to find any T-intersections and adjusts the polyline coordinates for any polylines that are not exactly perpendicular. A use for this routine is to select a right-of-way polyline and connected lot polylines that are meant to be perpendicular. To check polylines without adjusting, use the Edit > Polyline Utilities > Check Polylines > Highlight Non-Perpendicular Intersections command.

Prompts

Select polylines to check.
Select objects: *pick polylines*
Adjusted 1 polylines to make perpendicular.

Pulldown Menu Location: Area/Layout > Lot Layout Utilities
Keyboard Command: pl2perp
Prerequisite: Polyines

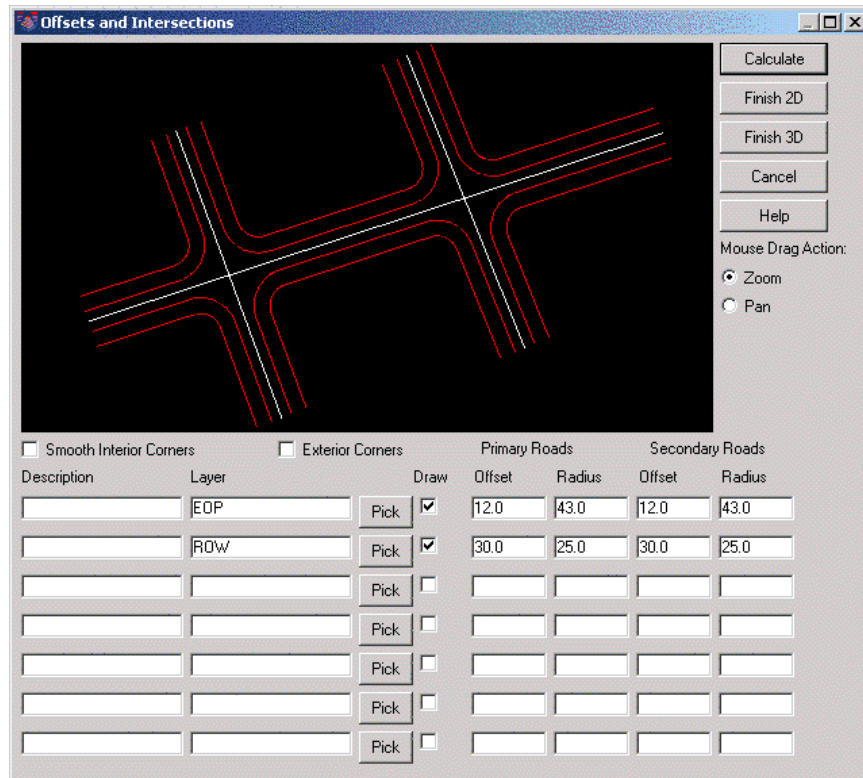
Offsets & Intersections

This command takes a set of centerline polylines and calculates the series of offset polylines using the user defined offset and fillet radius values. The function recognizes primary and secondary roadways which allows for different

offsets and fillet radii to be specified for each. Up to seven sets of offsets and radii can be defined for different features such as edge of pavement, right-of-way, sidewalk, etc.. Each set also has a layer name and description. The Pick button lets you set the layer name by picking an entity with that layer in the drawing. The description is for your own information and is not used by the program.

Multiple centerline polylines can be processed together which allows for the creation of an entire set of roadway offset polylines in one step. Intersections are calculated based on the centerlines selected and the fillet radii are applied at the intersections. The Smooth Interior and Exterior Corner options will fillet bends in the offset polylines. Otherwise turns without an arc in the original centerline will become straight corners in the offset polylines. The results of the calculations for the given parameters may be previewed in the dialog. Zoom and pan are available by clicking and dragging mouse on the preview image (zoom or pan mode is selected by a toggle). Once the satisfactory offsets are calculated, they are inserted into the drawing by clicking on Finish2D button. The Finish 3D button opens the *Elevate 2D Polylines* command, described in this chapter.

If it is preferable to handle intersections manually, you may run the command multiple times on non-intersecting centerlines. Another alternative is to use the *Offset* command in the Draw menu and the *Fillet* command in the Edit menu.



Prompts

Select all PRIMARY road polylines.

Select objects: *select polylines*

Select objects: *Enter*

Select all SECONDARY road polylines.

Select objects: *select polylines*

Select objects: *Enter*

Calculating offsets for layer EOP...

Calculating offsets for layer ROW...

Pulldown Menu Location: Area/Layout

Keyboard Command: wayint

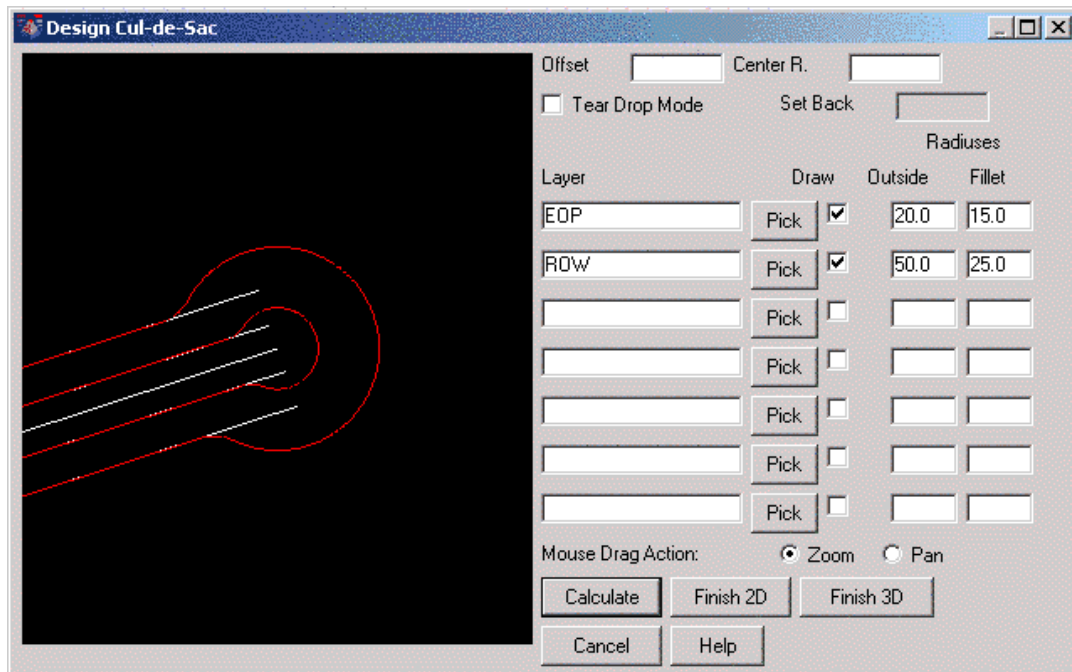
Prerequisite: Centerline polylines

Cul-de-Sacs

This command uses a polyline centerline and the offset polylines to create a cul-de-sac. These offset polylines can be generated by the *Offsets & Intersections* command, or with the standard *Offset* command. The layer names of the offset polylines must match the layer names set in the dialog.

To run this command, pick a set of polylines and point on roadway centerline where the cul-de-sac center is. For cul-de-sacs with an offset center, pick a projection of that center onto the centerline and specify an offset distance (positive value is offset to the right, negative - to the left). Like the *Offsets and Intersections* command, a preview is shown of the cul-de-sac being designed. Any of the cul-de-sac parameters may be modified and reviewed before the cul-de-sac is applied and the drawing is modified with the Finish 2D button. The Finish 3D button opens the *Elevate 2D Polylines* command described in this chapter.

Bend cul-de-sacs are created by selecting offset entities on one side of the centerline.



Prompts

Select all offset polylines to end with cul-de-sac.

Select objects: *make selections*

Pulldown Menu Location: Area/Layout

Keyboard Command: stdcul

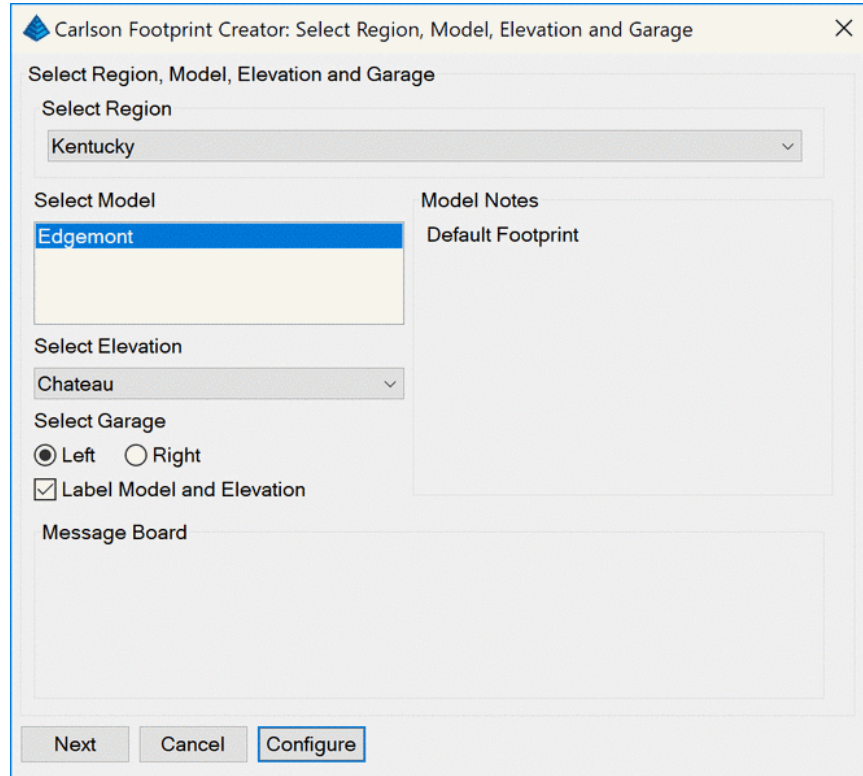
Prerequisite: A set of offset polylines and roadway centerlines.

Footprint Creator

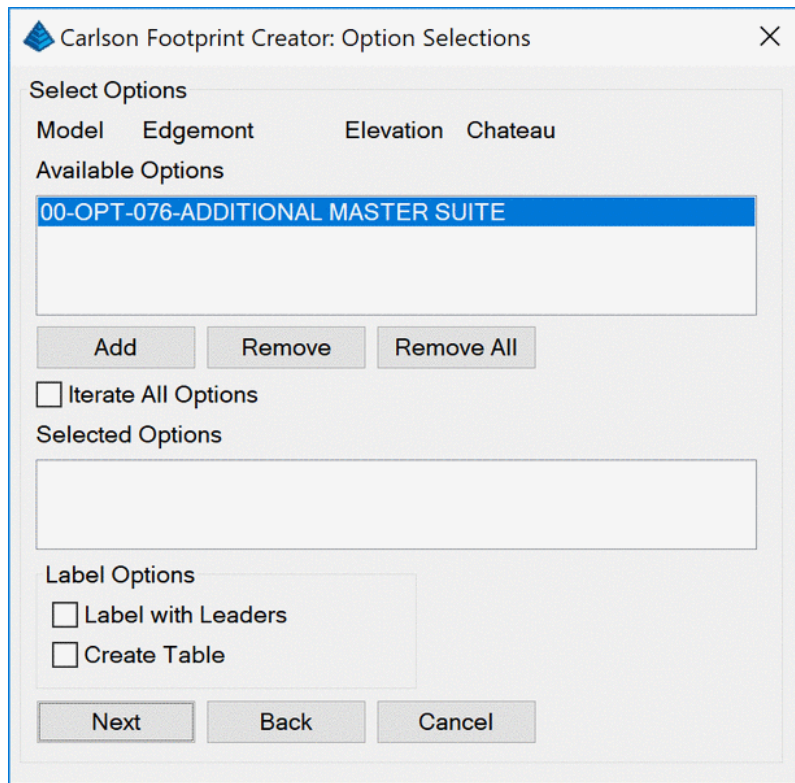
Footprint Creator is a command that allows for the creation of a library and placement of building footprints. The command is run from the Area/Layout menu and found in the Layout Utilities sub menu. The command inserts footprint drawings and uses the layering scheme in them to place the footprints with options. *Footprint Creator* allows the person producing plans to include optional aspects of designs such as; locating the garage left or right and including details such as driveways. Labeling tools are provided should it be desired that elements of the

footprints are called out in the plans.

Command Operation



The command starts with a dialog that allows for the selection of a footprint by a number of characteristics. These are region, model and elevation. Placement of the garage is able to be left or right. Labeling of model and elevation can be enabled or disabled. There is also a message board function that allows for team members to share information with each other about use and updates to footprints. For operation of the command select Next.



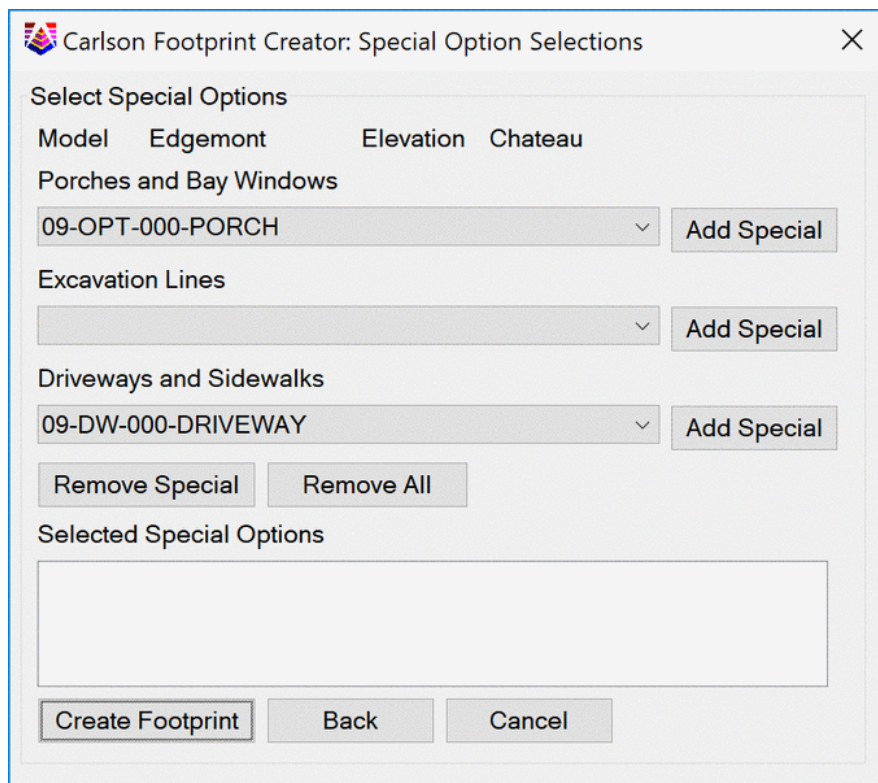
Footprint drawings may contain optional elements to be included upon placement. In this case an option is a master suite. This is read from the source drawings layering convention, layers containing "-OPT" display as options. To place an option select it by single clicking and choose the Add button. To remove it select it from the Selected Options pane and then select the Remove button. The Remove All button does just that and allows you to restart the selection of options.

Additionally, Iterate All Options if toggled along with the final creation method of Place on Building Pads will place and all options and combinations of all options. This option only has an effect if Place on Building pads is chosen.

The Label Options toggles will annotate the options either by leader, create table or both.

Use the Back button should you realize a different footprint is needed.

Select the Next button to select further options.



Porches and Bay Windows

The source drawing's layering scheme is used to allow for the placement of porches and bay windows. The command reads the drawing and displays any porches or bay windows available. The layer schemes for porches and bay windows are any of the following: "##-OPT-000-COVERED LANAI", "##-OPT-000-PORCH" "##-OPT-155" or "##-OPT-156". If none are included this pull down will be blank.

Excavation Lines

Excavation lines are also read from the source drawing. The layer scheme for excavation lines is "##-EXC". If none are included this pull down will be blank.

Driveways and Sidewalks

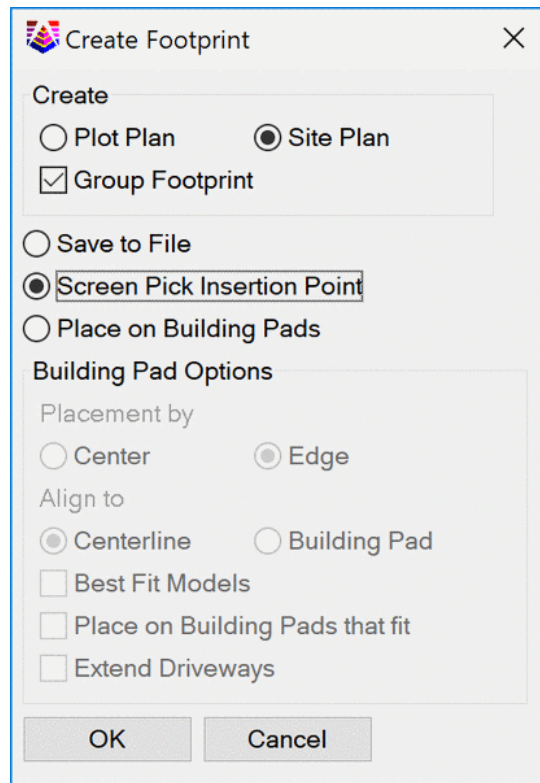
The source drawing can also include driveways. The layer scheme for driveways is "##-DW" and for sidewalks is "##-SW". Again, if none are present in the drawing this pull down will be blank.

Add Special, Remove Special, Remove All

The Add Special, Remove Special and Remove All buttons will add the option selected for the corresponding list. Remove them individually or remove them all.

Create Footprint

When ready to place the footprint select Create Footprint. You will be prompted to select the method for creation.

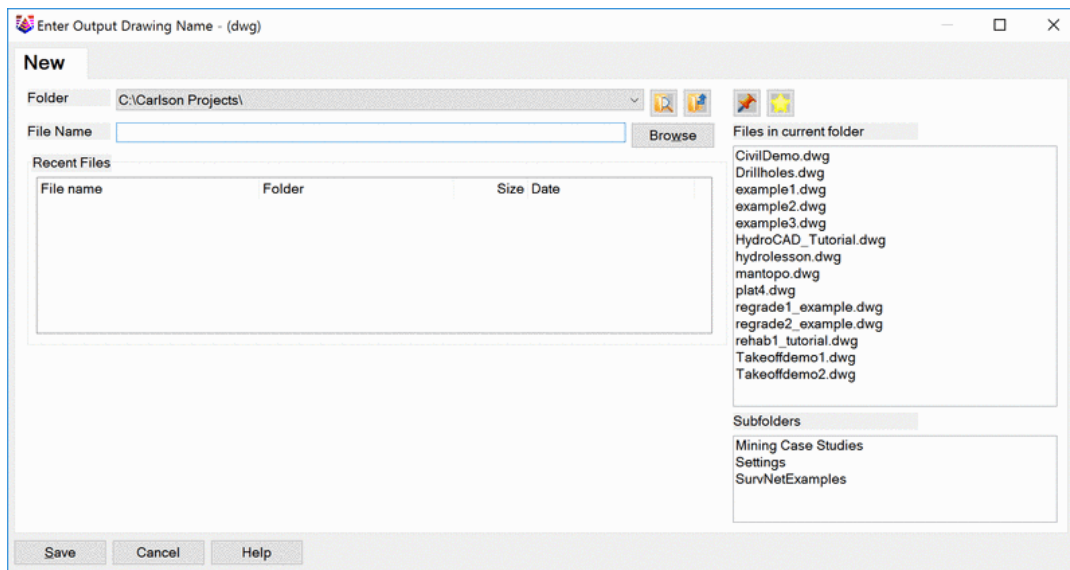


Create

Choose the option for placement. The footprint will include an enclosure line "Plot Plan" or simply place the footprint as drawn in the source drawing "Site Plan". The Group Footprint if selected will group the Footprint entities together.

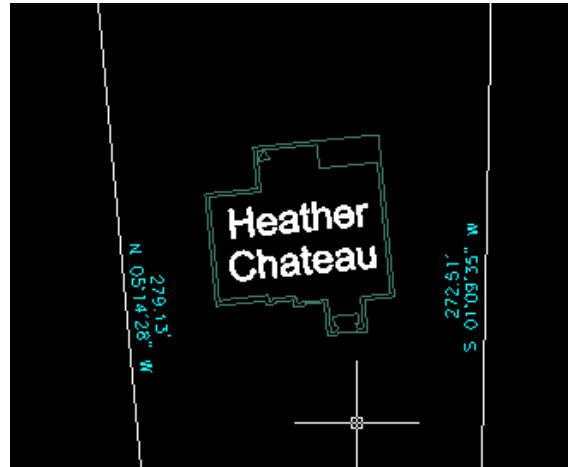
Save to File

Enter Output Drawing Name



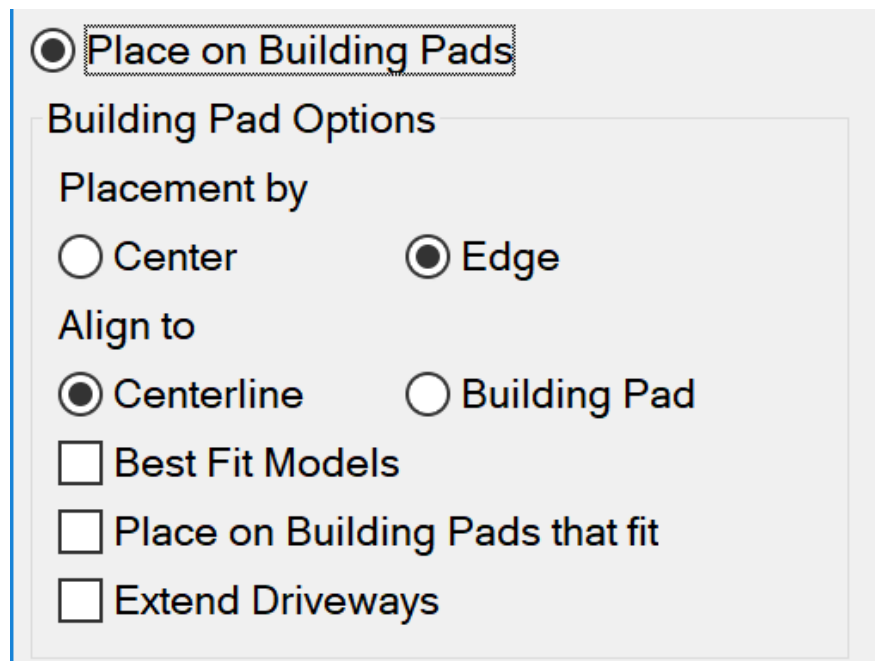
Screen Pick Insertion Point

Pick an insertion point and the rotation for the footprint. The command will continue to prompt for insertion points until canceled allowing for multiple placements of the same model, elevation and options within the same command execution.



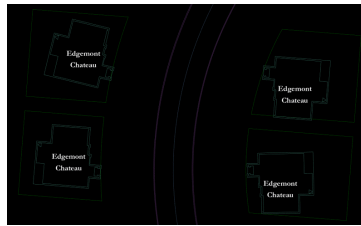
Place on Building Pads

For the Place on Building Pads to work properly, the footprint source drawing must be laid out properly. The insertion point of the footprint should be the front most point of the footprint with the garage on the left side. This option will prompt for the selection of Building Pads, these should be closed polylines. After selecting the building pads, the selection of Centerlines or Targets for Building Alignment will be prompted. Multiple targets may be selected and the tool will use these targets for determining the footprint front orientation.



Footprints will be placed in each building pad selected and rotated toward the selected alignment option. The footprints will be placed by edge (driveway side) or centered on the building pad. When the alignment is by centerline, the footprint will be placed perpendicular to the centerline. When alignment is by building pad it will rotated the footprint along the edge of the building pad.

Placement Center
Align to Centerline



Placement Edge
Align to Building Pad

Placement Center
Align to Building Pad

Placement Edge
Align to Centerline

Optional Toggles

If no option is selected, all Models and Elevations selected on the initial Select Region, Model, Elevation and Garage page will be placed within every Building Pad.

Best Fit Models

This option will adjust the footprint forward and back as well as side to side until the footprint fits within the building pad. If multiple models are selected on the initial Select Region, Model, Elevation and Garage page this will attempt to fit the highest priority model/elevation on each Building Pad. Once a footprint has placed on a building pad the building pad will no longer be processed. This is used to place the highest priority models in the selected building pads.

Place on Building Pads that fit

Like the Best Fit Models, this will only place footprints that fit within each building pads. This option will continue to process building pads that footprint have been placed in resulting with multiple footprints within any building pad that the footprint fits within.

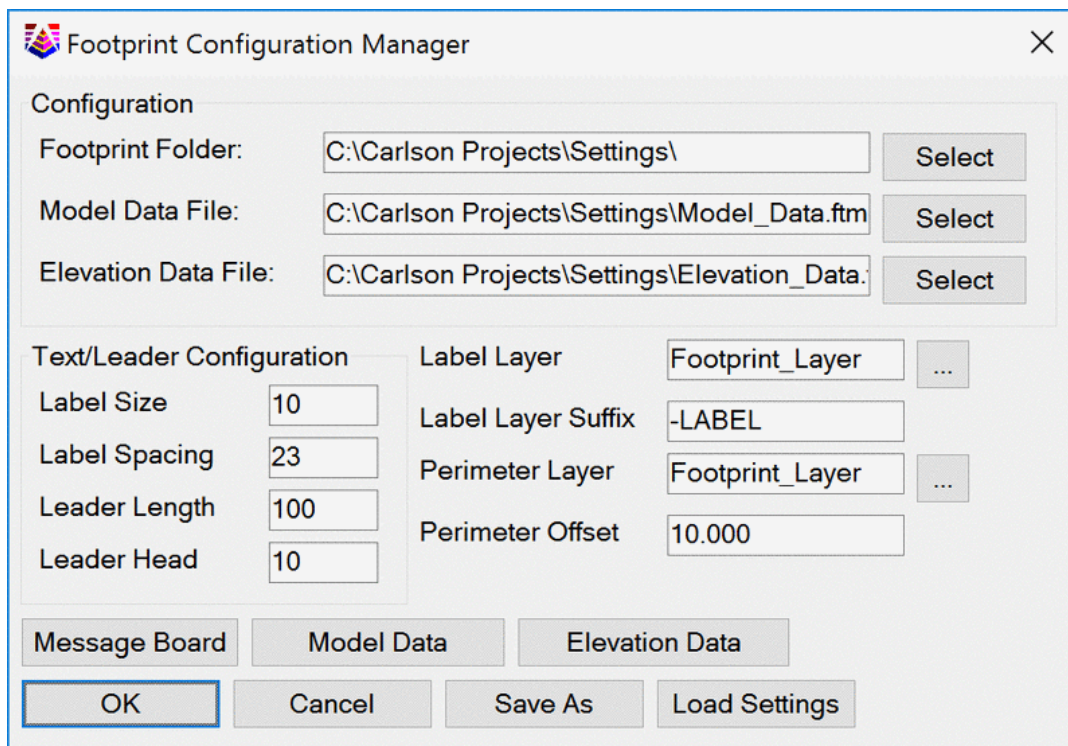
Extend Driveways

This option will prompt for Edge of Pavements for Driveway Extensions. Multiple targets may be selected, and the tool will use these targets while extending closed polylines representing the driveway. The layer scheme for the driveway extension must contain "DRIVEWAY" or "-DW". This includes the Special Optional Selections of driveways and sidewalks.

Configure

To configure *Footprint Creator*, select the Configure button on the first dialog. This must be coordinated with the drawings and directory structure for them for the command to work properly.

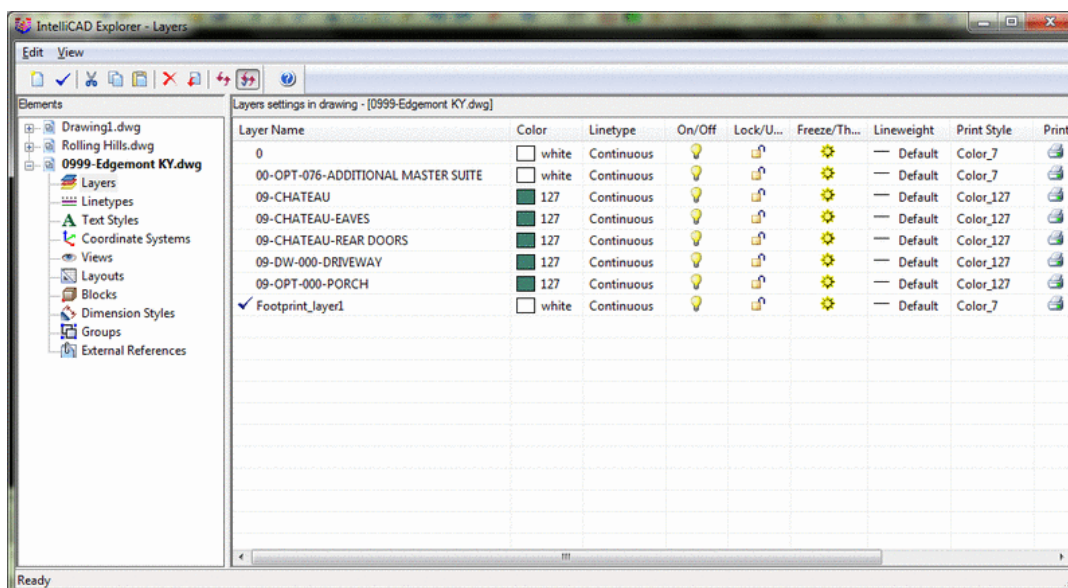
Footprint Configuration Manager



The Footprint Folder is the folder where all the sub folders for footprints will be located. The Model Data and Elevation Data Files by default will be in your Settings folder. Each footprint is stored in a folder with a name that corresponds to elements of the footprint. A footprint is included with the Carlson installation. It is 0999-Edgemont KY.dwg and is in a folder named 0999-Edgemont KY under the settings folder.

An additional model 0998-Remington KY.dwg is also included in the Carlson installation and is in a folder named 0998-Remington KY. This model has multiple elevations defined and can be accessed by choosing the Add_Model.ftm and Add_Elevation.fte files. The additional model and elevation files also include the Edgemont model and elevations.

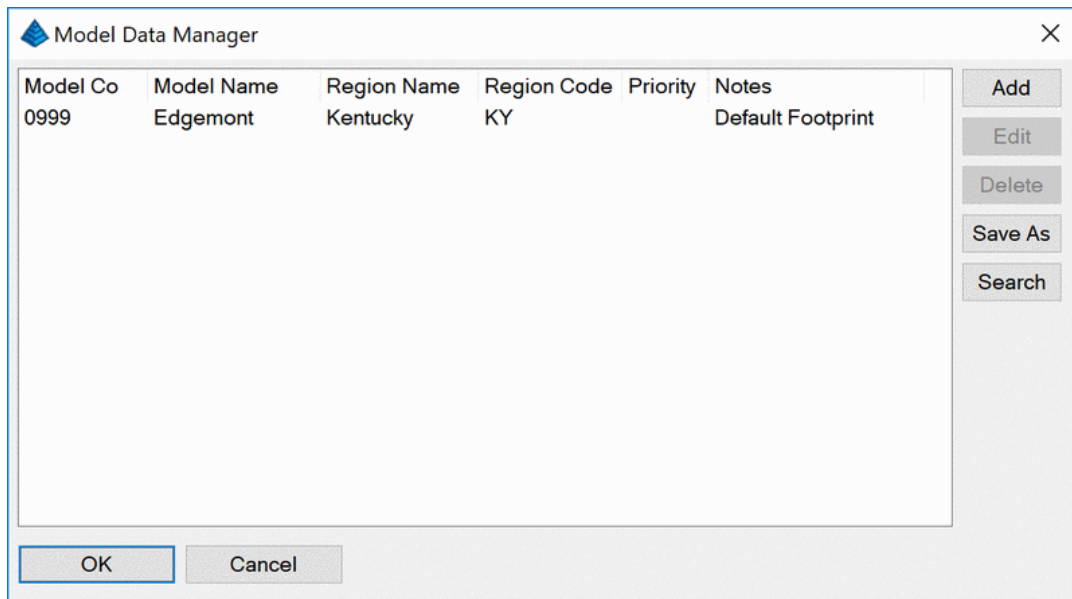
Layers are included that are read by the command to determine which elements of the drawing are placed when the command is run.



Other layers that can be used are those related to excavation. Layers with OPT are standard Options, they must be preceded by 00- as shown "##-OPT". Layers that include the Elevation name can be selected during command operation.

Model Data

To create or edit Models select the Model Data button.



The Model Code is a four-digit code that will be the first four characters of the folder and drawing file name. This is followed by a dash and then the Model Name which is followed by a space and then the Region Code. The command matches the Region Code to the Region Name for display in the dialog box for Region. The Notes are purely descriptive.

Add Model Data

To add a model select the Add button.

The 'Enter Fields' dialog box is a standard Windows-style window with a title bar containing a logo and the text 'Enter Fields' and a close button (X). The main area contains six labeled input fields stacked vertically: 'Model Code', 'Model Name', 'Region Code', 'Region Name', 'Priority', and 'Model Notes'. Each label is to the left of a rectangular text box. At the bottom of the dialog are two buttons: 'OK' and 'Cancel'.

The Model Code is four numeric characters. The Model name can be alpha or numeric. The Region Code is a two-character designation. The Region Name matches up with the Region Code and can be alpha or numeric. The notes are descriptive and are there purely to help understand information about the Model. To edit an existing model, select the Model and then select the Edit button. To save a model for modification, select the model and then select the Save As button. Should the list of models become extensive, use the Search button to locate the model desired.

Elevation Data Manager

To create or edit Elevation Data select the Elevation Data button.

The 'Elevation Data Manager' dialog box has a title bar with a logo and the text 'Elevation Data Manager' and a close button (X). The main area contains a table with the following data:

Code	Name	Abbreviation	Color Code
09	Chateau	CHAT	127

To the right of the table are five buttons: 'Add', 'Edit', 'Delete', 'Save As', and 'Search'. At the bottom of the dialog are two buttons: 'OK' and 'Cancel'.

The operation of this dialog is nearly identical to the Model Data Manager dialog. The Code and Name relate to layers in the source drawing.

Pulldown Menu Location: Area/Layout - Layout Utilities

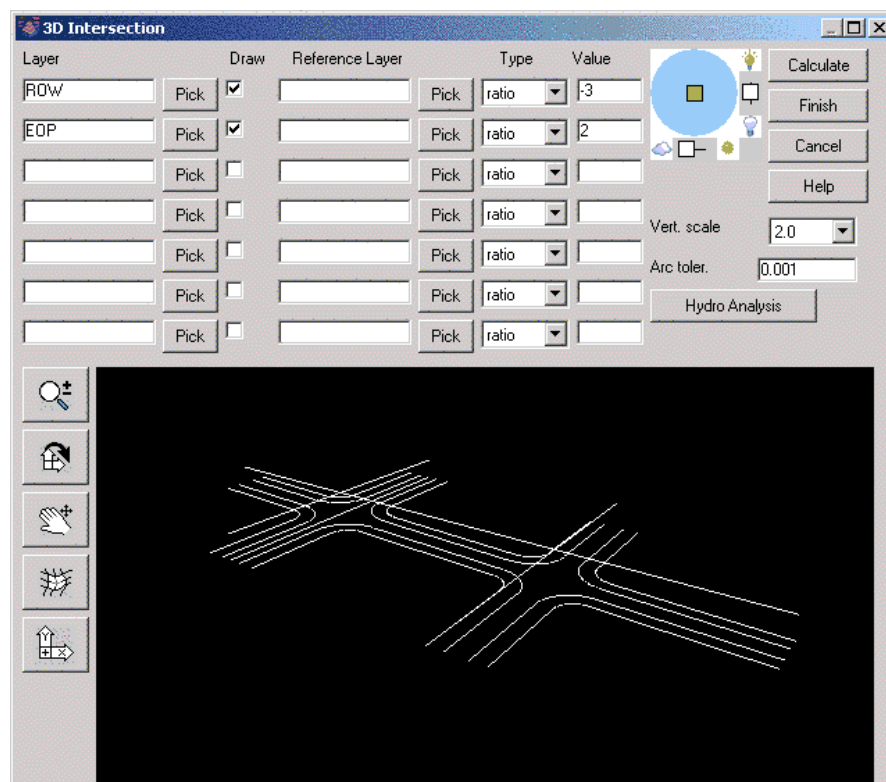
Keyboard Command: createfp

Prerequisite: Footprint drawings in properly named folders with layering convention matching the command configuration.

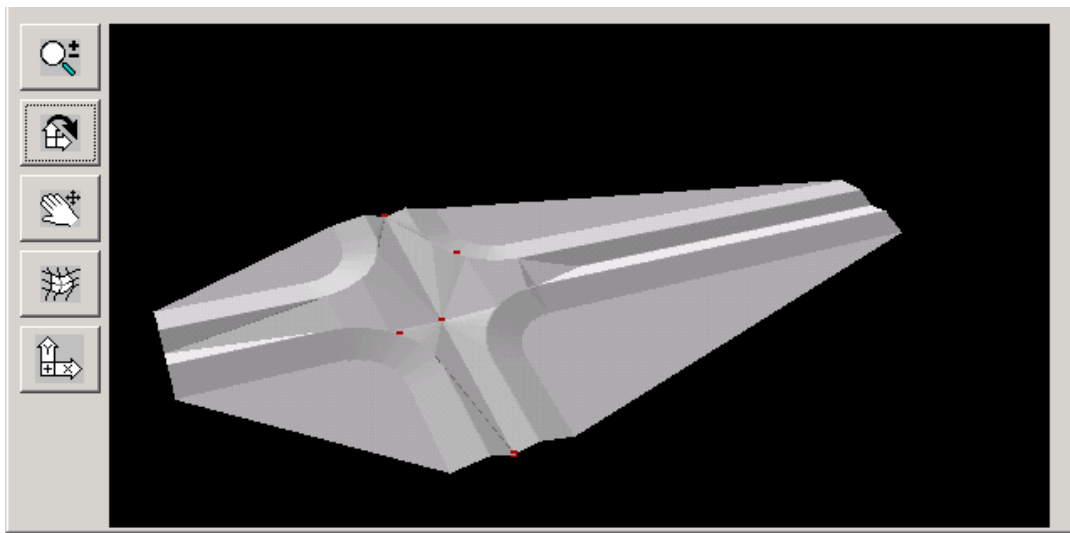
Elevate 2D Polylines

This command allows to assign elevations to a selection of polylines based on elevations along supplied 3D centerline and user-defined slopes. This routine calculates a distance from each vertex of 2D polyline to a specified 3D reference polyline and uses that distance and slope to calculate a 3D offset to a corresponding point on 3D polyline.

You can specify either the original centerline to be a reference 3D polyline or use another set of offset polylines. For example, you could specify the edge of pavement elevation to be relative to the curb elevation, while curb elevation is calculated based on the centerline elevation. You can view the resulting road/intersection design in 3D, making changes and updating picture on-the-fly. Local sink points can be reported instantly by evaluating a resulting triangulation to predict low points in the design leading to water retention.



After selecting entities



Another road/intersection design, this time in 3D

Prompts

Select all offset polylines for the intersection.

Select objects: *select entities*

Select all 3D profile polylines.

Select objects: *select entities*

Pulldown Menu Location: Area/Layout

Keyboard Command: 3dintersect

Prerequisite: A set of offset polylines and roadway centerlines

Parking

This command draws a series of parking stalls. The command prompts for stall width and length, stall parking angle, and side for stalls. Stalls can be located by the number of stalls in a direction, as many as fit between two points, or along a polyline.

Stall Layout Method: The Number of Stalls method creates the specified number of stalls at a fixed stall width along an alignment. The Fit By Width creates as many stalls as can be fit on an alignment between the specified Min and Max Stall Width. The Fit By Number is the same as Number of Stalls except the stall width is adjusted to span the full alignment with the stalls.

Min Stall Width: Indicate the minimum width a stall can be when the Fit on Alignment option is specified.

Max Stall Width: Indicate the maximum width a stall can be when the Fit on Alignment option is specified.

Stall Width: Indicate the stall width when the Number of Stalls option is specified.

Number of Stalls: Indicate the desired number of stalls when the Number of Stalls option is specified.

Stall Length: Indicate the desired length of each stall.

Angle of Parking: Controls the angle of the stalls from the alignment.

Line Width: Sets the line width property of the parking lines. Using a Line Width helps with viewing the parking lines in the 3D Viewer.

Adjust Length for Angle Method: When the stalls are not perpendicular, this setting controls whether the stall length is measured along the stall line or as the perpendicular offset from the curb alignment.

Side for Stalls: Indicate the side to which the stall lines should be placed. The Left and Right sides are relative to the direction of the baseline. Use the Pick option to screen pick the side for the stalls.

Stall Placement Method: Indicate the method by which the stall direction should be determined whether it be between two picked points or along an existing graphical alignment.

Draw Baseline: This option draws a baseline in addition to the stall side lines.

Draw Handicap Marker: This option draws a handicap symbol.



Locate on Real Z Axis: Controls whether the stall lines are created at zero elevation or the elevation of the alignment.

Draw Label: This option labels the number of stalls using the Text Size Scaler, Label Prefix and Suffix settings.

Layer: Specify the layer on which parking lines should be placed or click the Set button to choose an existing layer.

Prompts

Starting Point?

Pick point or point number: *Pick a point*

Ending Point?

Pick point or point number: *Pick a point*

Created 10 stalls.

Pulldown Menu Location: Area/Layout > Layout Utilities

Keyboard Command: parking

Prerequisite: None

Set Back Measure-Move

This command can be used to measure the perpendicular distance of 1 or 2 points to 1 or 2 lines. This can be helpful in placing buildings for proper setback from lot lines. After selecting the lot lines and the building, the command allows you to drag the building while a real time display on the side-bar menu shows the perpendicular distances to the lot lines. After experimenting you can press T to type in the values to move to. The second line and point are optional.

Prompts

Select 1st Lot line to measure perpendicular from.

Select object: *select line*

Select 2nd Lot line to measure perpendicular from ([Enter] for none).

Select object: *select line*

Select entity to move at 1st point to measure from:

Select object: *ENDPOINT of (Pick a point on polyline.)*

Pick a 2nd point on entity to measure from ([Enter] for none). *END of (Pick a point.)* Pick another endpoint of the polyline representing the building.

Drag-Pick new Location or [T]ype in Move distance(s) [C] to Cancel: *T* Either drag the building to a location and press the pick button on your pointing device or press T to enter the distances.

You may have to use a negative distance to move to the proper side of lot line!

Distance from 1st line: *10*

Distance from 2nd line: 20

The building is then moved to your specification.

Pulldown Menu Location: Area/Layout

Keyboard Command: setback

Prerequisite: Lot lines and polyline representing the building should be plotted.

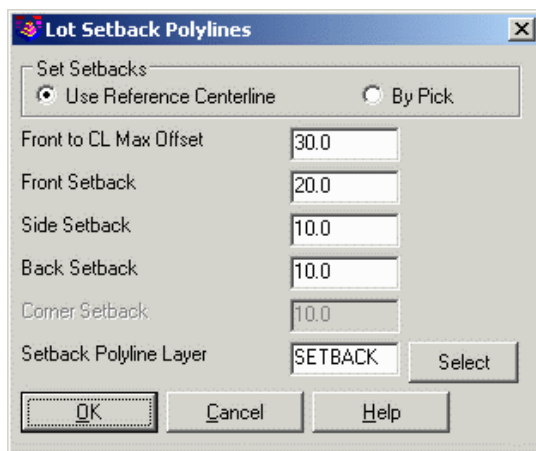
Draw Lot Setback

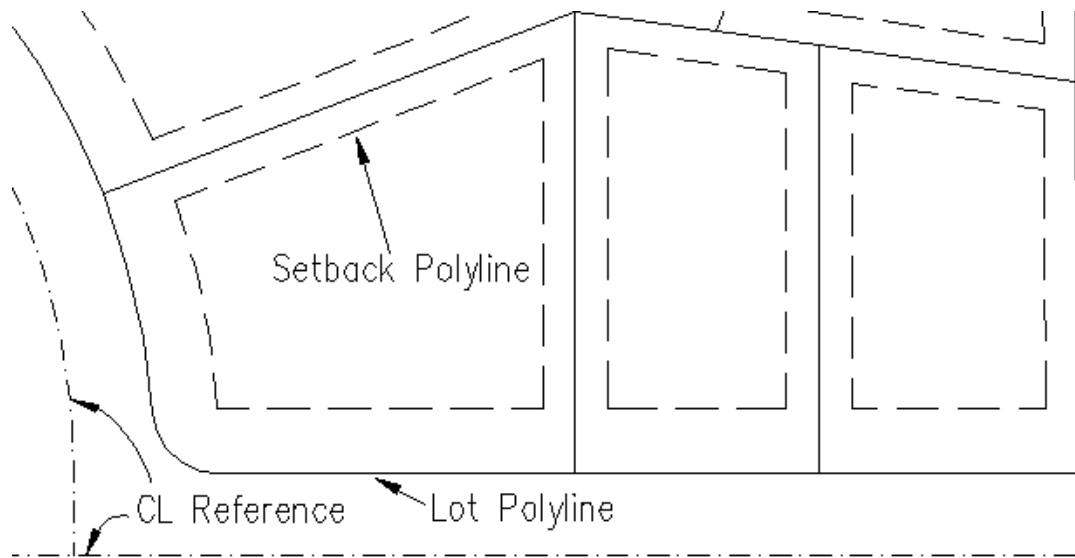
This command draws closed polylines inside lots to represent the building setback offsets. Before running this command, the lots need to be drawn as closed polylines. The command starts with a dialog for entering the setback offsets and the layer for the new setback polylines.

There are two methods for the program to define the edges of the lot as front, side, back or corner. For the Use Reference Centerline method, the Front to CL Max Offset is used to determine which lot edges are frontage. The program will prompt to select CL reference polylines and lot edges that are within this offset from these CL polylines are considered frontage edges. For the By Pick method, the program prompts to select the setback type for each lot edge. Only the By Pick method has the option for the Corner Setback type.

The Round Corners option will round the setback polyline using a radius of the setback offset amount for corners with an interior angle over 180 degrees.

The Front Setback is applied to the lot frontage edges. The Side Setback offset is applied to lot edges that have only one of their ends within the frontage offset. The Back Setback offset is applied to all other lot edges.





Prompts

Lot Setback Polyline dialog

Select reference centerline polylines.

Select objects: *pick the polylines*

Select lot polylines to setback.

Select objects: *pick the polylines*

Pulldown Menu Location: Area/Layout

Keyboard Command: lot_setback

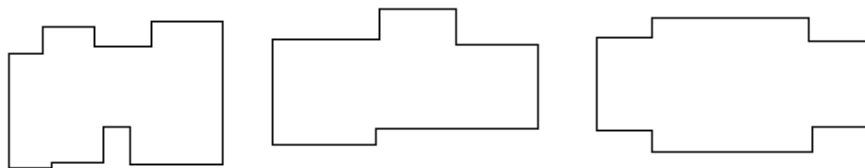
Prerequisite: Lot and CL polylines

Fit Structure

The purpose of the Fit Structure feature is to place a structure (or the footprint) within a bounding polygon. For example, a house foundation - the footprint - within the limits of the setback lines of a lot - the bounding polygon.

The user can easily insert a structure footprint within a lot or bounding polygon near its final location. The footprint then it can then be conveniently rotated and/or translated, in user definable increments, to the exact position desired.

Footprint Templates



The template, a full scale definition of a structure's footprint, must be defined prior to placing it within the bounding polygon.

Templates are not AutoCAD drawings but they can be imported from AutoCAD drawings.

The data specifying the dimensions of a template is stored in a binary format and cannot be manipulated without using the Template Manager. Once a template is placed in the drawing, it becomes a C&G footprint polyline. Since

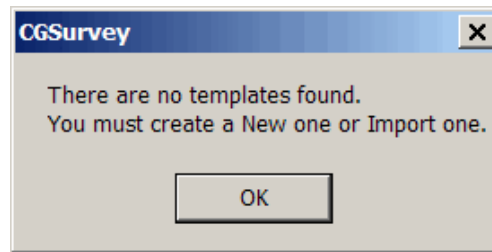
it is a C&G polyline, it can be queried and manipulated using ordinary AutoCAD and CGSurvey commands.

Template Manager

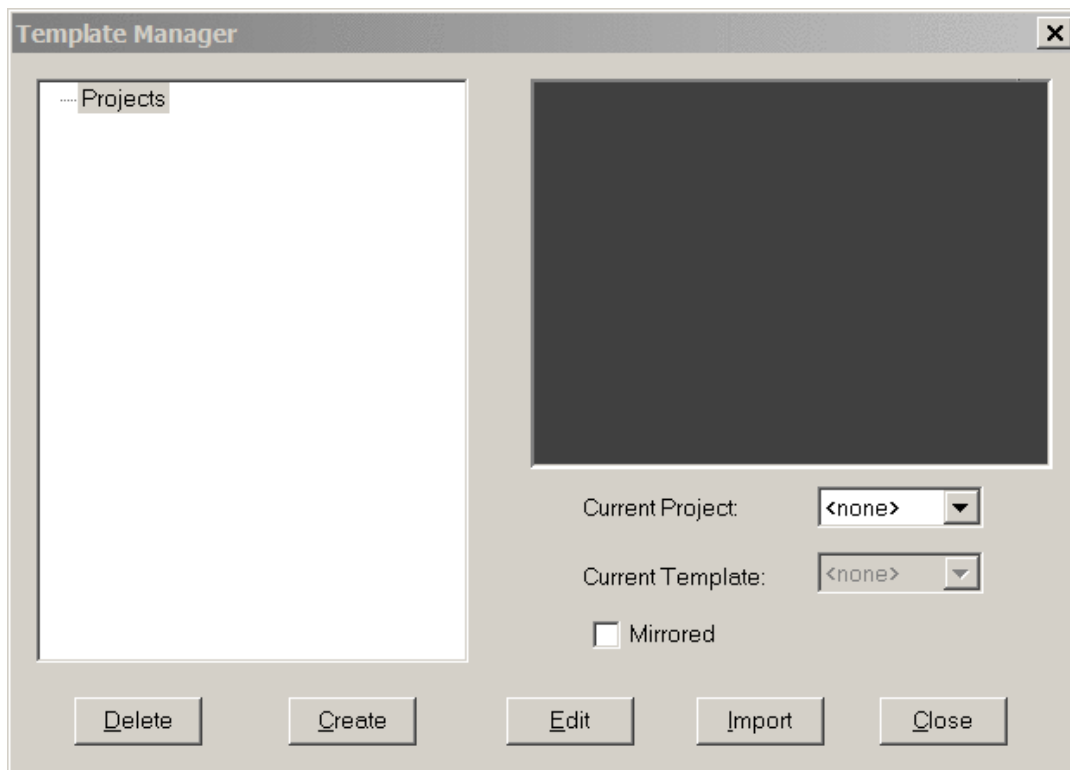
The Template Manager is used to manage the templates for the various projects you work on. For example, the various house footprints used in a given subdivision can be defined as templates. The templates can then be placed in a lot in an "as" or "reverse" orientation and reused as many times as necessary. You can use the Template Manager to define templates directly or import the templates from existing AutoCAD drawings.

The Template Manager allows you to organize your templates within projects. You can name the projects in a meaningful way then import the templates into the project "folder".

When you choose the Fit Structure feature from the Cogo menu for the first time, a dialog warns you that you have no templates defined then brings up the Template Manager.



In the dialog below you will notice that there is nothing listed under the Projects item. This means that you will have to either create a template or import one from an existing drawing.



Creating a Template:

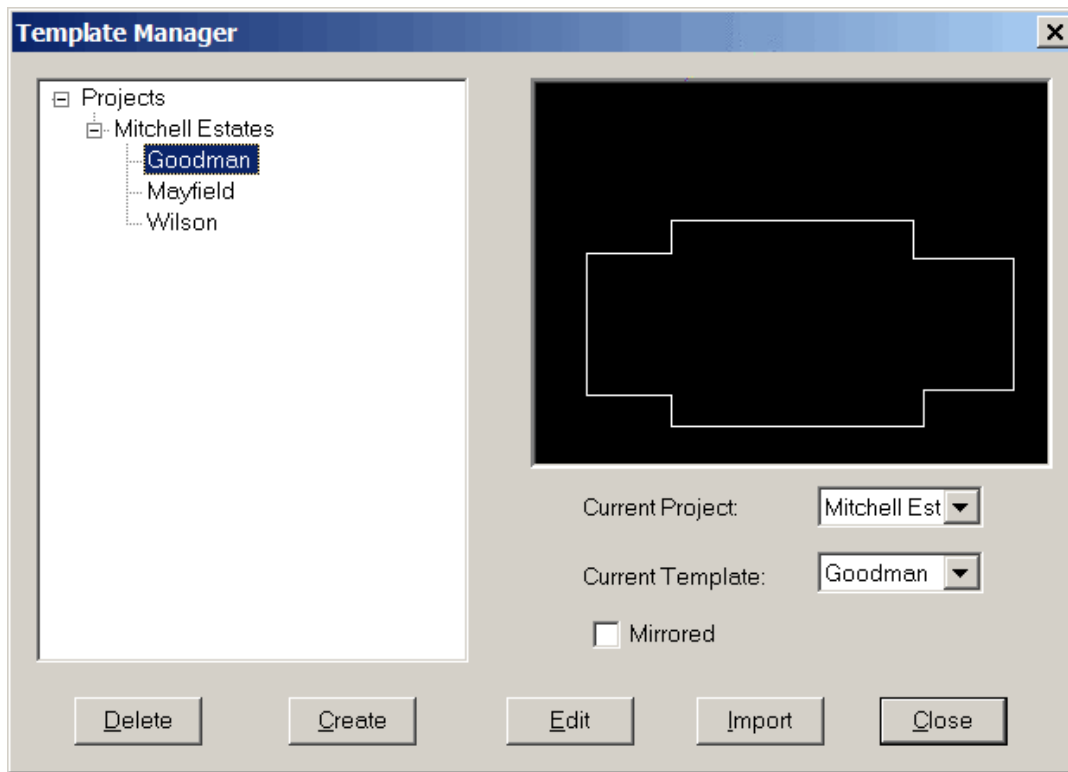
To create a new template, click the Create button. This brings up a dialog that allows you to configure the simple drawing in which you will create a template. This dialog allows you to specify the name of the project, the name of the template and asks about the approximate overall size of the template. If the structure template is made up of right angle segments you may want to specify a snap grid to aid you in laying out the template. You should be

aware that the create template method should only be used for very simple templates and that it does not allow you to edit the structure once it is added to the Template Manager.

The screenshot shows the 'Add a Template' dialog box. The title bar is blue with the text 'Add a Template' and a close button (X). The main area is light gray. It contains the following elements:

- 'Name of Project:' dropdown menu with 'Morristown' selected.
- 'Name of Template to Add:' dropdown menu with 'bldg 3' selected.
- A section titled 'Approximate Overall Dimensions of Template' containing:
 - 'Length:' input field with '125'.
 - 'Width:' input field with '75'.
- A section titled 'Grid' containing:
 - A checked checkbox labeled 'Use grid to aid in drawing the template'.
 - 'Grid interval:' input field with '5'.
- 'Cancel' and 'OK' buttons at the bottom.

When you are done configuring the create template drawing interface, click OK and you will see an empty AutoCAD screen upon which you can draw the template. The template is merely a closed polyline. The C&G Polyline by Points interface is used but in this case there will only be normal AutoCAD points picked (also known as graphic points designated as GR-PT). The polyline must be closed - so use the C for Close command line option for the last line segment in the template. Once you enter the Close option keyword for the polyline you are working on, the drawing window closes and the template is imported into the Template manager as shown below.



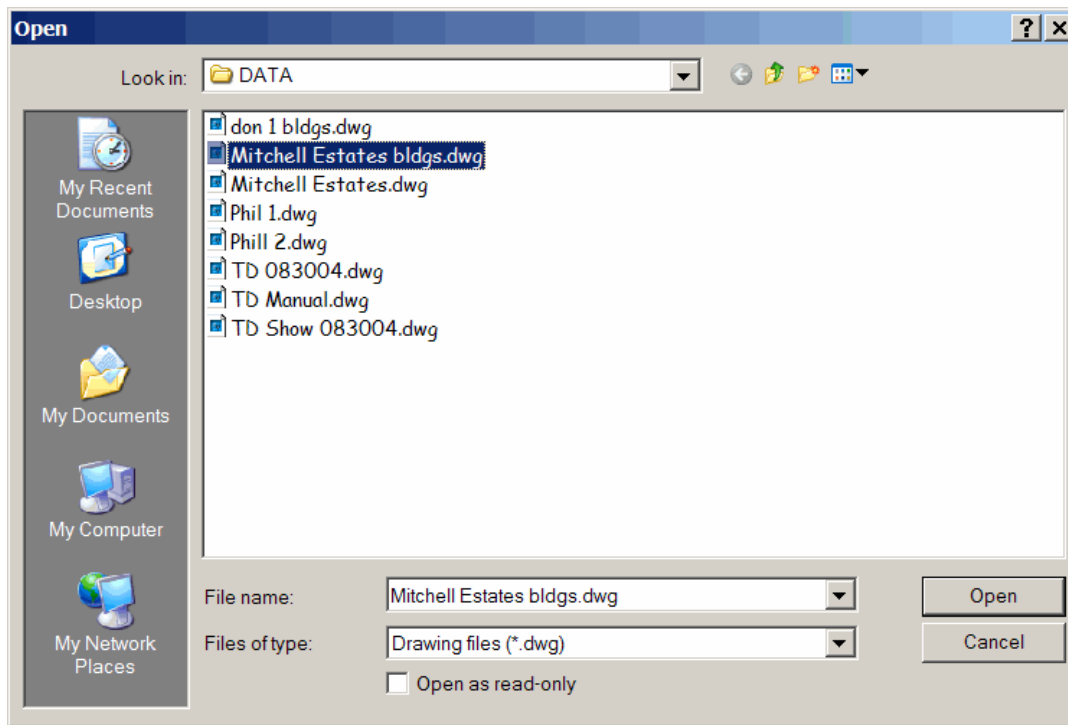
Note: You must click on the template name in order to see its shape in the window on the right and to choose it as the current template.

Importing a Template

The Create template method is only useful for very simple templates. For more complicated templates and projects with multiple structures, it is recommended that you use the Import method. To import a template you must create a separate drawing, then draw all of your templates at full scale on the layer specified for templates (see **Fit Structure Setup**).

Create a separate template drawing

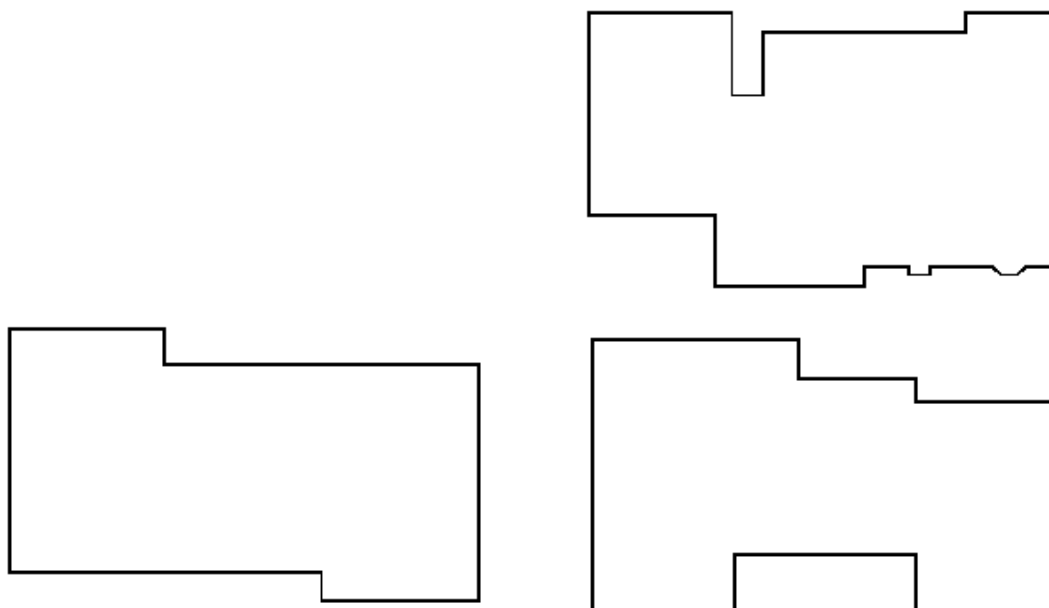
Begin the importing of a template by creating a new drawing file as a repository for all the structure template drawings used in a specific project. For this example we will create a new drawing named Mitchell Estates bldgs.dwg. This file will only contain the structure template drawings for this project. Select the CAD **File** menu then select the menu **New** item



You may be asked to choose a drawing template (not to be confused with the structural template polylines you are about to create). It is generally easiest to use the default acad.dwt drawing template but you may also specify one of your own choosing.

In this new drawing create a layer having the name specified in **Fit Structure Settings** and make this layer current. You can accomplish this by using the CAD Layer Manager. To open the Layer Manager, from the Format menu choose the Layer menu item.

In the new drawing, draw the templates (house footprints for example) you will be using in your project. A structure template must be a closed polyline and may contain arc segments.



Draw the individual house footprints. It is recommended that you use either the C&G **Polyline by Points** feature or use the standard CAD PLINE command - on the Draw menu choose 2D Polyline.

You could also use the C&G **Quick Traverse** feature to traverse around the building. However, if you use **Quick Traverse** to create the footprint you must then convert the C&G lines created by Quick Traverse to polyline. To do this you can use a utility on the CGTools menu, Join Nearest.

Once you have created the templates needed, close and save the template drawing file. You can come back to this drawing at anytime and add or modify templates as needed.

Note: If you change a template in the original template drawing, you must be re-imported using the Template Manager. First, use the Template Manager's **Delete** feature to delete the old template, then re-import the changed template from your template drawing file.

Placing a Footprint:

Return to the original drawing into which you wish to insert the footprint. In this example Mitchell Estates.dwg will be used to place structure footprints within lot setbacks.

Select the **Fit Structure** menu item.

If this is the first time you have run the command and no templates have been specified, you will be informed of this by a warning dialog. Click **OK** in the warning dialog and the **Template Manager** will come up.

If you have inserted a template prior to running this command, the following prompt will be seen at the command line:

Choose a structure template

[Set template/Current-template (Wilson)/Mirror-current/Done] <C> :

Select "S" for **Set template** to bring up the **Template Manager**.

The Template Manager

As mentioned earlier, the **Template Manager** is used to manage the structure templates you use for your various projects. In the left hand pane the projects and their associated templates are arranged similar to the directories in the Windows Explorer. On the right pane is a drawing showing an unscaled representation of the shape of the currently highlighted template. The highlighted template becomes the current template when you Close the Template Manager. The following describes the Template Manager functions in more detail.

Delete button: This allows you to delete a Project or an individual template.

Create button: This allows you to generate a template "on-the-fly" while in the current drawing file. This method of creating templates should only be used for the simplest of templates. In most cases it is recommended that you import pre-drawn templates from other existing drawings.

When the **Create** button is selected, the **Add A Template** dialog appears (shown earlier).

Name of Project: enter a new name or press the down arrow to select from existing projects.

Name of Template to Add: enter a new name or press the down arrow to select from existing projects.

Approximate Overall Dimensions of Template:

enter an approximate length and width. Make sure this overall dimension will include the entire template so you will be given enough room to draw the template - too large is better than too small.

Grid:

If you wish to have a snap grid as a drawing aid when you create a template, check the **Use grid to aid in drawing the template** checkbox and set the grid interval. You need not use a grid but it is useful in creating simple rectangular templates.

Click **OK** to begin creating the template. To create the template, pick the desired locations for the various building corners. Be sure to close the structure perimeter by typing C and Enter. After closing the template polyline you will be returned to the **Template Manager**.

Import button:

clicking this button allows you to import the template from another drawing file. As described earlier, you should create a separate template drawing. In that drawing draw the required templates as closed polylines. The templates may contain arcs.

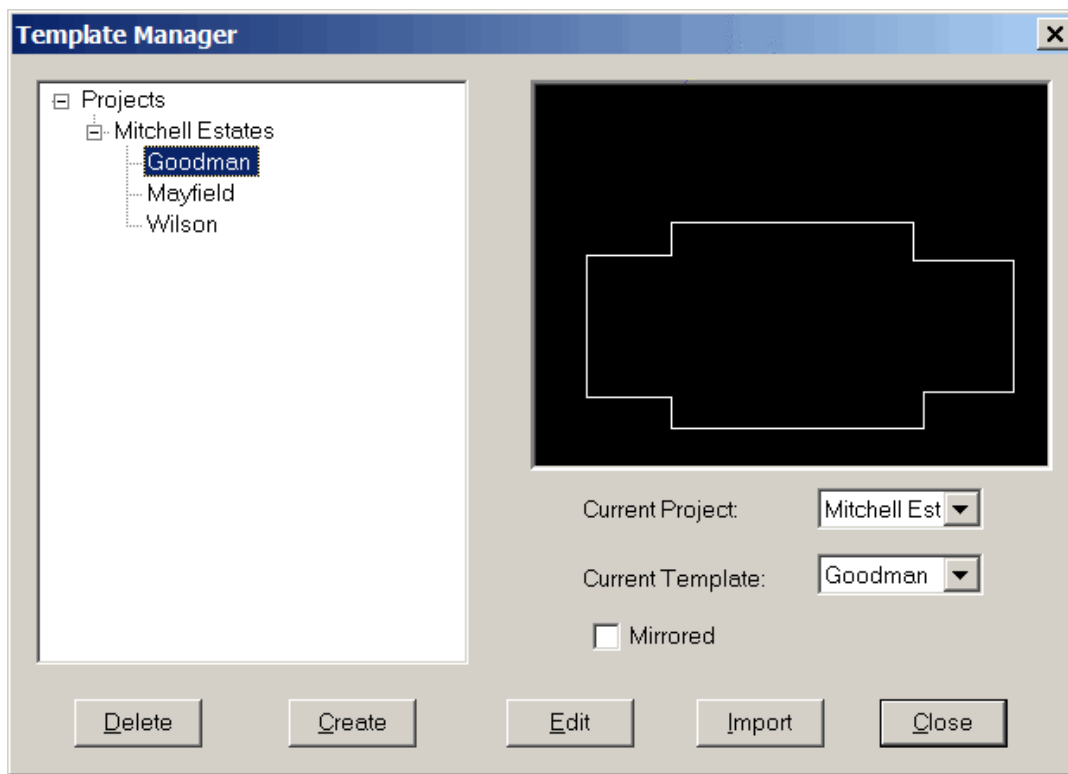
When you select the Import button a dialog (shown earlier) comes up asking you to enter or select a **Project Name** and to specify the **Name of Template to Add**.

The **Project Name** can be anything you wish but is often the name of the subdivision or the client name. The template name can also be anything you wish. It should generally reflect the type or style of structure the template represents.

After filling out the project and template names and clicking **OK**, a file dialog will be displayed. Choose the drawing file you created earlier containing the template(s) you wish to import.

After closing the file dialog the template drawing will be shown and you will be asked to choose the template polyline. When you pick the template polyline its geometry is stored in a special file reserved for template information and you will be returned to the **Template Manager**.

If you highlight the newly imported template on the left hand pane it becomes the current template and you should see it displayed in the right hand pane.



If you wish to import another template just repeat these steps as many times as necessary.

By highlighting the template name it is made the current template. You may choose to mirror the current template on the Y axis by checking the Mirrored checkbox. All you need do now is click the **Close** button to close the **Template Manager** and place the footprint in the drawing.

Fit Structure Example

The following file names will be used when describing the following example:

Coordinate File: Mitchell Estates.crd

Drawing File: Mitchell Estates.dwg

Template Drawing File: Mitchell Estates bldg.dwg

Note: The template drawing file may have several templates in the same drawing file. For example you may have a subdivision with many different house footprints.

Import the templates

Open the subdivision drawing file, in this case; Mitchell Estates.dwg, and the associated coordinate file: Mitchell Estates.crd.

The subdivision drawing should already exist and you should have already defined the bounding polygons within which the structures are to be placed. These bounding polygons can be defined either by polylines (arcs are allowed) or lines and arcs. The lots and setbacks (bounding polygons) can also be defined using a C&G Point Group or Groups.

Once the subdivision drawing is open and has been prepared for the placement of structures choose **Fit Structure** from the menu.

If you have not run the **Fit Structure** command and set a current template in this drawing session, the **Template Manager** dialog will appear.

The first task will be to create a project and import templates from the template drawing file, Mitchell Estates bldg.dwg.

Select the Import button and fill in the project name and template name.

In this example the subdivision name is Mitchell Estates and the house model being added to the template list is the Wilson.

Next a drawing file dialog will be displayed. Highlight the template drawing file (in this case Mitchell Estates bldg.dwg) and click the **Open** button and use the cursor to choose the polyline representing the template to be imported.

After choosing the template polyline, you will be returned to the **Template Manager**. You will notice that the template you just chose has been added to the template manager under the project you selected. To see its shape and make it the current template, click the template name under the current project.

You may continue to add templates as required. Click **Close** to begin placing the template in the subdivision drawing.

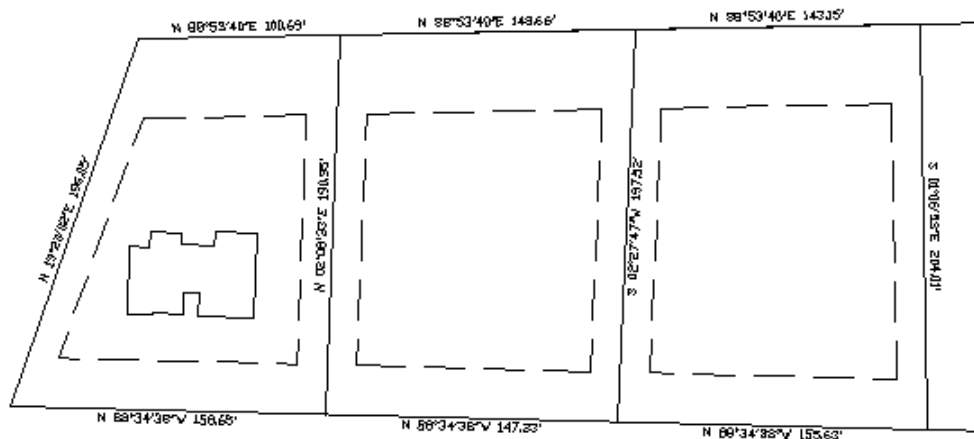
After the Template Manager closes you will return to the main drawing and see the following prompt:

Pick the lot within which the structure will be placed [cg-Point-group/Done] <pick>:

Pick a polygon or a series of lines that define a closed lot boundary or setback within which you wish to place the structure. Type P and Enter to use a C&G point group file to define the bounding polygon.

Place the structure in the bounding polygon

Once you have specified the bounding polygon you will be asked to place the structure near its final location in the bounding polygon. Move the structure near its desired location using the mouse and click the left mouse button to place it at that location. Once you have picked the approximate location for the structure you will then be allowed to rotate and move the structure to its exact final location.



Note: If you need to adjust a template further once it has been placed within the bounding polygon and you have exited the Fit Structure command, you can run the Fit Structure command again and pick the existing structure instead of using a template.

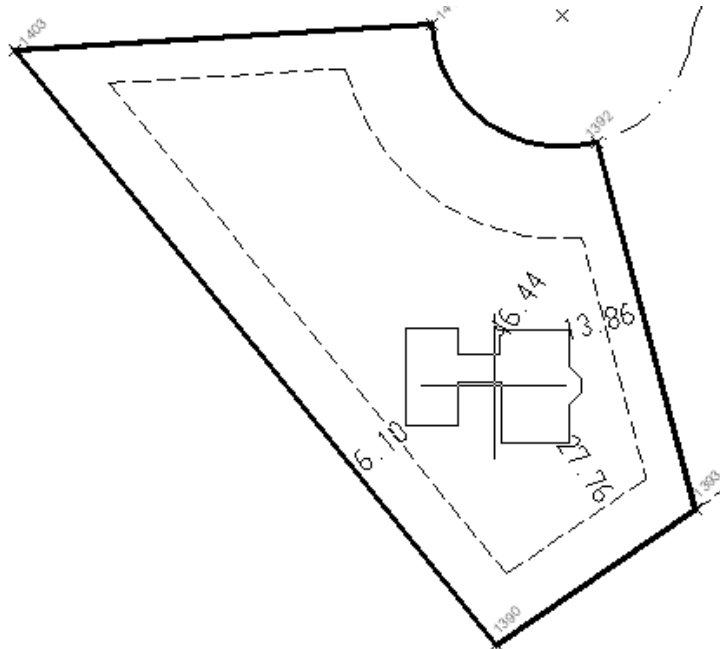
Adjust the structure

After placing the structure in the bounding polygon you will see the following prompt at the command line:

Adjust structure [Move/Step-move/step-Rotate/rotate/Ninety/Parallel/On-boundary/setup/Done] <D>:

You are now at the stage where the structure can be adjusted to its final desired location with relationship to the setback lines and its orientation with respect to the street and other features.

In all the commands used to adjust the structure, the distances to the bounding polygon may be displayed at the appropriate corners of the template (see example below). You may turn this distance display on or off or view or change other fit structure parameters using the setUp option (type U and Enter at various the prompts).



Move:

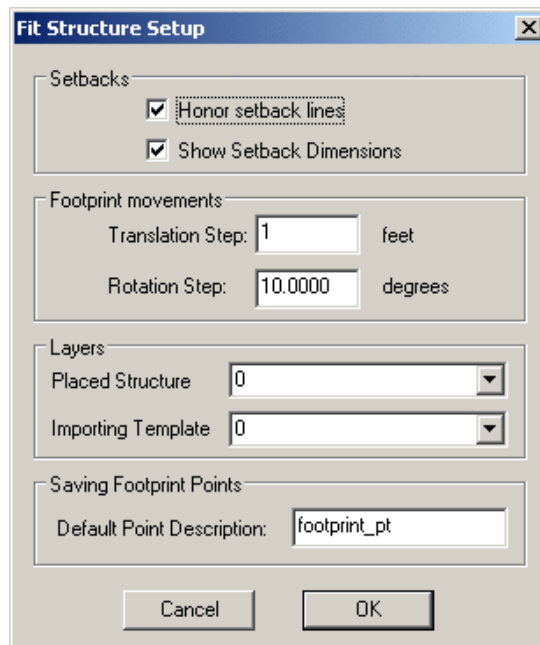
Type M and Enter to "drag" the structure around using the mouse cursor - similar to when you first placed the structure in the bounding polygon. This option is only meant for moving the structure in a gross, imprecise way and thus allow you to place it near its final location. After using this option the structure can be more finely adjusted using one of the other options described here.

Step-move:

To move the structure up, down, left or right, using the arrow keys on the keyboard, type S and Enter. The following prompt will appear:

Press arrow keys to move 1.000 dwg units [setUp/Done] <D>:

Now you can use the arrow keys on your keyboard to move the structure by steps in the X and Y directions. The distance moved per keystroke is indicated at the command line - in this case the structure moves 1 unit each time you press an arrow key. To change the per step increment, type U for setUp. This brings up the **Fit Structure Setup** dialog, allowing you to change the **Translation Step** setting (see below).



Click **OK** to return to the **Adjust Structure** command line.

Step-Rotate:

If you type R and Enter for **step-Rotate** you can then use the up and down arrow keys on the keyboard to rotate the structure by small rotational steps.

The following prompt will appear:

Use down/up arrow keys to rotate 10°00'00" clockwise/ccw [setUp/Done] <D>:

Rotate:

To rotate the structure, type R and Enter. The following prompt will appear:

Rotate structure to desired orientation: [setUp] <pick>:

Use this option to rotate the structure by moving the mouse. Left clicking will place the structure at the current rotated orientation. This method of rotation is not precise and is thus useful only for gross rotational movements.

rot-Ninety: Type N and Enter to rotate the structure 90 degrees in a clockwise direction.

Parallel:

Type P and Enter to rotate the structure so that one of its sides is parallel to a specified line segment on the bounding polygon.

First, select the side of the bounding polygon that you wish to be parallel to a selected side of the structure.

Next, select the side of the structure that is to be parallel to the previously selected line on the bounding polygon.

After picking the side on the structure the structure will be rotated into position.

Note: If the rotating the structure about its geometric center to make the selected sides parallel to one another will cause an encroachment, an error message will be displayed, no changes will be made, and you will return to the **Adjust Structure ...** prompt.

On-boundary:

Type O and Enter to choose a point on the structure that is to touch a selected point on the bounding polygon. This is accomplished by translation only.

Pick the point on the bounding polygon where you want the structure to touch: Pick the point where the structure touches the bounding polygon.

Choose the point on the structure that you want to touch the bounding polygon: Pick the point on the structure that touches the bounding polygon.

If choosing a structure corner as the point to touch the bounding polygon, you should use the end point snap. If you do not use end point snap, the translation of the point picked to the bounding polygon will likely cause the corner of the structure to encroach. You can specify end point snap when picking the point on the structure by typing in "end" and Enter at the prompt, then you merely need to pick a point on the structure near the desired corner to actually specify the corner point.

Note: If the translating the structure to make the selected point touch the bounding polygon at the selected point would cause an encroachment, an error message will be displayed, no changes will be made, and you will return to the **Adjust Structure ...** prompt.

Completing the adjustment process

Once you are satisfied with the location of the structure type D and Enter and you will see the following prompt:

Creating structure coordinate points:

Enter description for structure corner points <footprint_pt>:

You can accept the default description shown in brackets by pressing Enter or you may type in a description that will help you identify this particular structure and lot.

The corner and any radius points for the current location of the structure are stored in the current coordinate file and, if **Auto plot points** is ON, the points are drawn.

After storing the points for the previously placed structure you will see the following prompt:

Choose a structure template [Set template/Current Template/Mirror current/Done] <C>:

Press Enter or C and Enter if you wish to repeat the process and place the current structure template in the same or another bounding polygon.

If you wish to place a mirrored ("reverse") version of the current structure template in a bounding polygon, type M and Enter.

If you wish to place a different structure in a bounding polygon, type S and Enter to bring up the Template Manager, allowing you to pick a new template.

If you are done placing templates for now, type D and Enter for Done.

At any time you may adjust an existing structure by choosing **Fit Structure**. If there are existing structures in the drawing, it will be detected and the following the prompt will appear at the command line:

Pick existing structure to adjust or choose a structure template.

[Set-template/Current-template(Wilson)/Mirror-current/Done] <C>:

At this prompt you can use the mouse to pick an existing structure to adjust. You can now use any of the adjustment methods described above to further refine the location of the structure. After the adjustment process is complete the coordinate file is updated to reflect the adjusted locations of the structure's corner and radius points.

Note: When you pick an existing structure, any plotted corner point symbols are temporarily removed to facilitate the adjustment process. Once you are done adjusting the existing structure, these points are re-plotted at their new adjusted locations

At this prompt you may also choose to place a new structure in a bounding polygon. To use a different template, type S and Enter to bring up the **Template Manager** and allow to choose the desired template. If you

have already placed a template in the current drawing session, the prompt will indicate the current template. By typing C and Enter or just pressing Enter you can choose to place the current template in a bounding polygon or you can type M and Enter to place a mirrored version of the current template:

Prompts

Template Manager dialog: create or choose a template to place within a bounding polygon (a lot)

Add a Template dialog: Used in conjunction with the Template Manager dialog to add a template to a given project.

if you have already specified a template to use but no templates have been placed in the drawing:

Choose a structure template

[Set template/Current template (Wilson)/Mirror-current/Done] <C> : Type "S" and Enter to bring up the Template Manager dialog. Type "C" and Enter or just Enter to use the current template. Type "M" and Enter to mirror the current template. Type "D" and Enter when done placing templates.

if a structure/template exists in the drawing or you have already specified a template to use:

Pick existing structure to adjust or choose a structure template.

[Set-Template/Done] <S> : To adjust an existing structure pick it on the screen. Type "S" and Enter or just Enter to bring up the Template Manager dialog to choose a new template. Type "D" and Enter when done.

after you set a new template or chose to use the current one:

Pick the lot within which the structure will be placed [c&g-Point-group/Done] <pick> : pick the polyline or a series of lines that define a closed polygon within which the structure template will be placed. Type P and Enter to specify the bounding polygon using a C&G Point Group file.

Place the structure near its final location in the bounding polygon <pick> : Drag the structure template to the desired location and click the left mouse button to place the structure.

after you place a template or pick one to adjust:

Adjust structure [Move/Step-move/step-Rotate/roTate/rot-Ninety/Parallel/On-boundary/setUp/Done] <D> : Type "M" and enter to move the structure. Type "S" and Enter to use the arrow keys to move the structure in predefined steps. Type "T" and Enter to use the cursor to rotate the structure. Type "R" and Enter to rotate the structure template a predefined number of degrees using the up and down arrow keys. Type "N" and Enter to rotate the structure 90 degrees clockwise. Type "P" and Enter to translate and rotate the structure template parallel to a side of the bounding polygon. Type "O" and Enter to move the structure template so a chosen point on the structure touches a chosen point on the bounding polygon. Type "U" and enter to use the Setup dialog to change the step sizes, layer names and other configuration items for the fit structure command.

when saving the structure coordinate points:

Enter description for structure corner points <footprint_pt> : Specify a description for the structure template corner points to be saved in the coordinate file or just press Enter to use the default description.

Pulldown Menu Location: Area/Layout > Layout Utilities

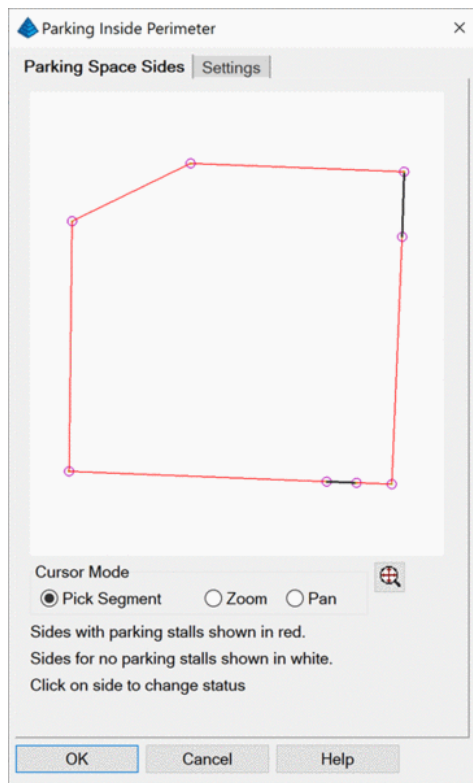
Keyboard Command: cg_fit_structure

Prerequisite: coordinate file, pre-drawn bounding polygon (lines and arcs or a polyline)

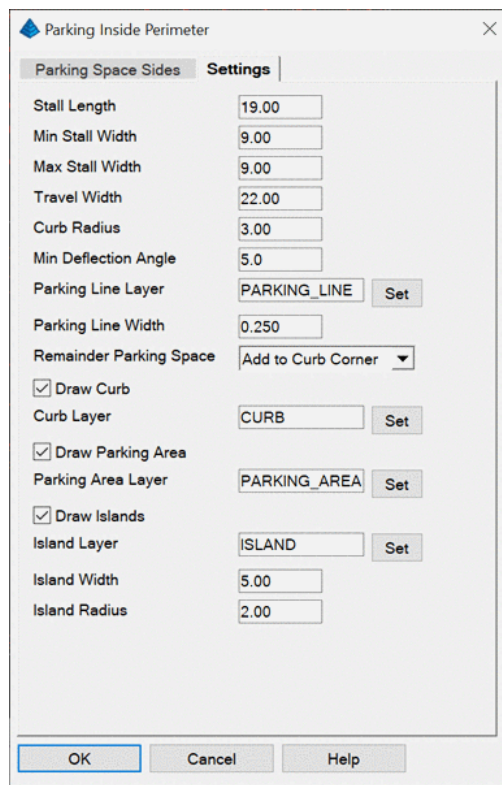
Design Parking Inside Perimeter

This command creates a parking lot layout inside a closed polyline. The program draws parking stalls along the perimeter and creates interior parking stalls. Before starting the command, create a closed polyline that has vertices for any locations along the perimeter where to skip creating stalls. For example, the part of the perimeter where the parking lot entrance is located should have vertices on either side of the entrance.

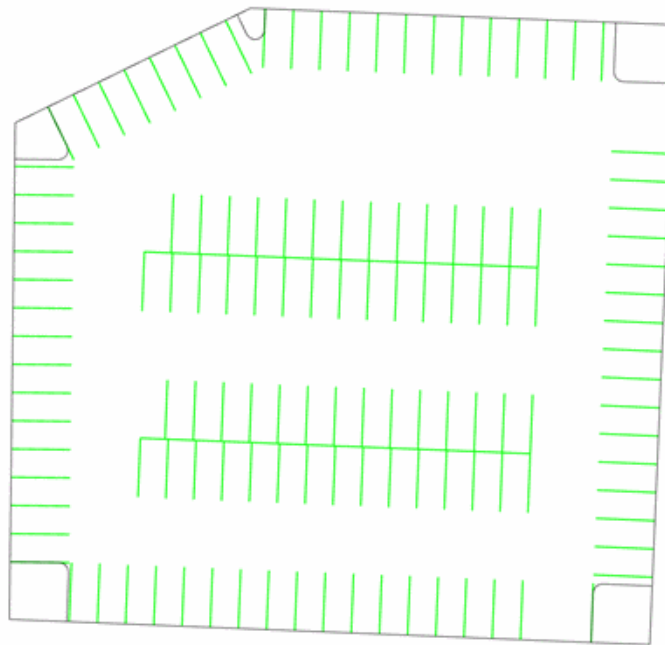
The command starts by selecting the parking perimeter. Then the dialog shows the selected perimeter. Pick on the perimeter segments for skipping parking stalls. In this example, the segments on the bottom-right and top-right are set for no parking stalls. Use the Create Parking Spaces Along Sides option to turn off all parking spaces around the perimeter and only create interior parking spaces.



The dialog has a Settings tab to set the dimensions of the parking stalls. The Min Stall Width and Max Stall Width set a range for the program to best fit the stalls along the perimeter. Set these widths to the same value to hold a fixed width. The Travel Width is the space between the rows of interior parking stalls. The Curb Radius is used for the fillet radius on interior corners. The Parking Line Layer and Parking Line Width are used for the parking stall lines. Using the Parking Line Width makes the lines show like paint stripes in the 3D Viewer command. When the parking stalls don't exactly fit a perimeter edge and there is left over space, the Remainder Parking Space setting chooses between leaving the space empty or making the curb corner larger to use up the space. The Draw Curb option draws a polyline around the perimeter with the interior corners and excluding the skip parking segments. The Draw Parking Area creates a closed polyline that makes the perimeter of the parking area. The Draw Islands option creates polylines for islands at the end of the parking space rows.



After the dialog, the program draws the parking lines and reports the number of parking stalls. The interior parking stalls are automatically created in the direction that makes the longest rows.



Prompts

Pick perimeter polyline: *pick driveway polylines*

Parking Perimeter dialog

Created 128 parking spaces.

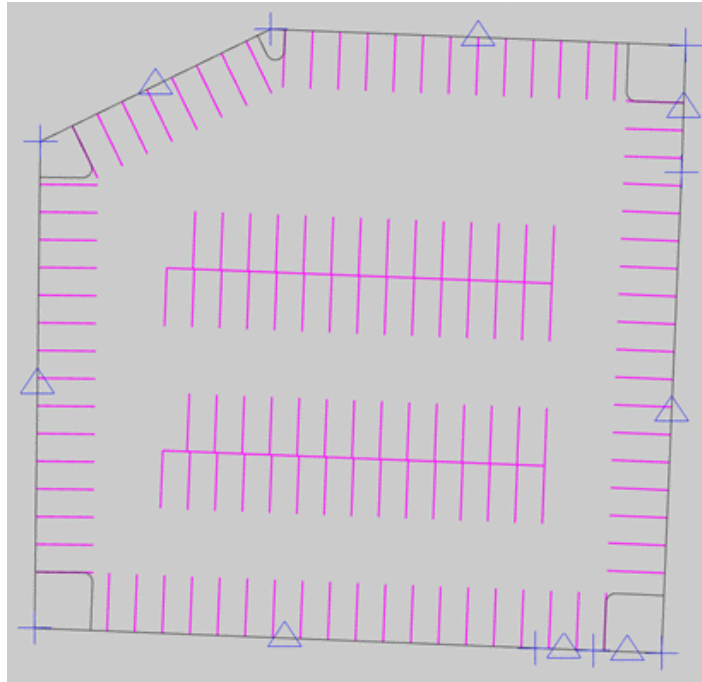
Pulldown Menu Location: Area/Layout > Parking Utilities

Keyboard Command: park_perim

Prerequisite: Closed polyline

Edit Parking Inside Perimeter

This command adjusts the perimeter polyline and recreates the parking layout from the Design Parking Inside Perimeter command. To start the command, pick any entity from the parking design. The program shows symbols along the parking perimeter. Pick near the plus symbol to adjust a single vertex and pick near a triangle to adjust a segment. After picking the vertex or segment to adjust, the program shows the new layout in real-time as you move the cursor to pick the new position. After picking the new position, the parking layout is redraw and the program reports the number of parking spaces and new parking area.



Prompts

Pick parking linework: *pick any entity from the parking design*

Pick perimeter point to adjust: *pick point on perimeter to adjust*

Created 127 parking spaces in area 38787.0 sf.

Pick perimeter point to adjust (Enter to end): *press Enter*

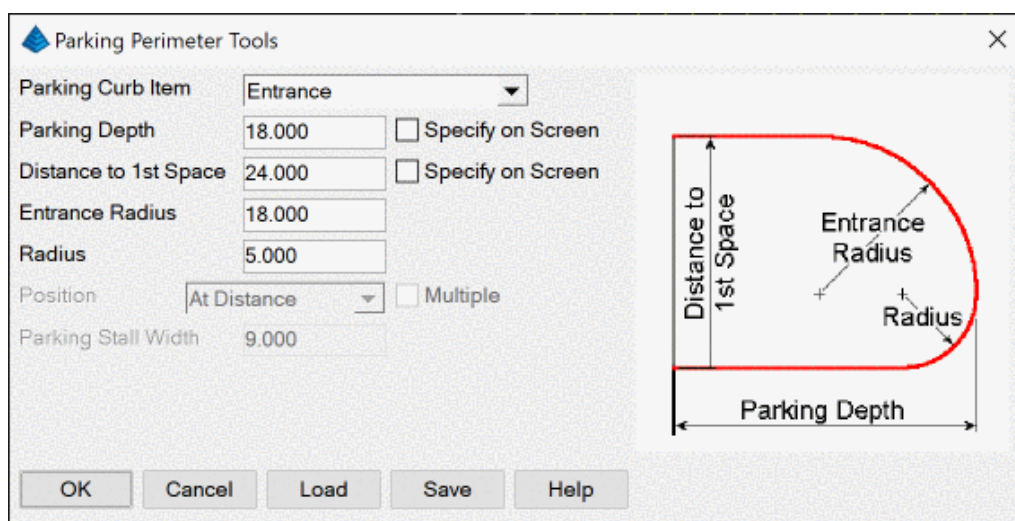
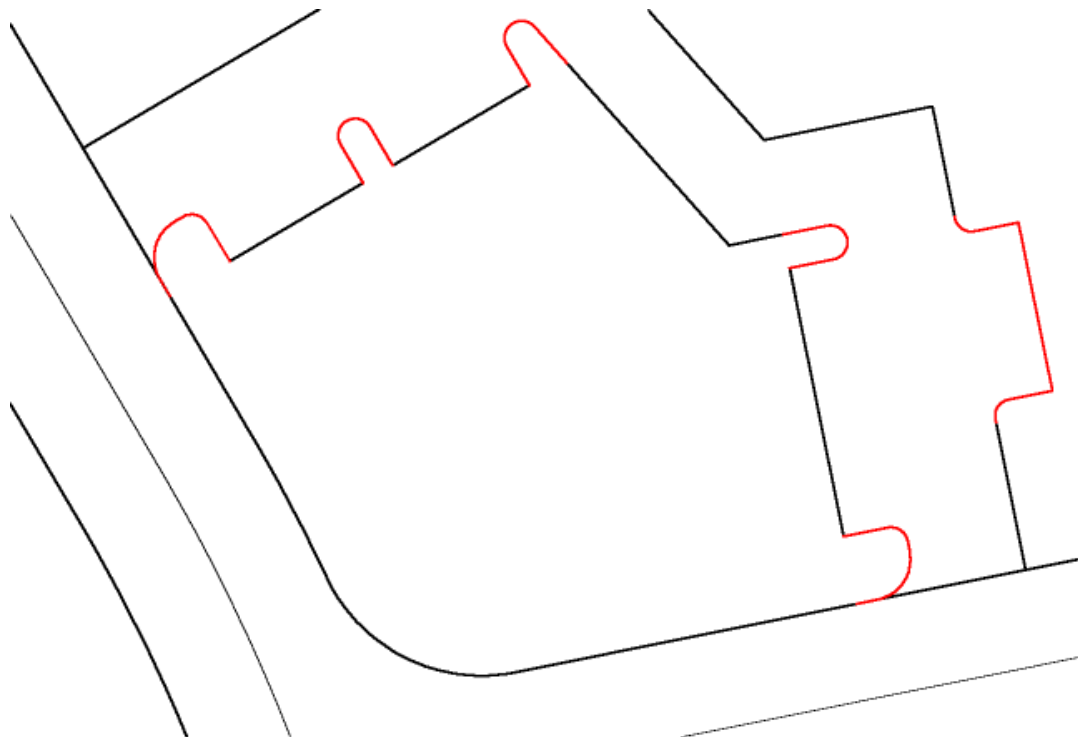
Pulldown Menu Location: Area/Layout > Parking Utilities

Keyboard Command: park_adj_perim

Prerequisite: Parking design

Parking Perimeter Tools

Parking Perimeter Tools are a set of tools to assist in the design of parking area curb linework.



Parking Curb Item: Tool items; Entrance, Half-Island, Bump-Out, Exterior Corner, , Extend Corner and Interior Corner

Depth: Depth of the curb item **Specify on Screen** Value will be input by prompt

Width: Width of the curb item **Specify on Screen** Value will be input by prompt

Exterior Radius: Radius on the exterior of the curb item

Interior Radius: Radius on the interior of the curb item

Position: Options for placement; By Pick, At Stall Count or At Distance

Parking Stall Width: Width of parking each stall

The **Save** and **Load** buttons save and recall the curb item settings to a .PARK file.

Parking Perimeter Tool dialog

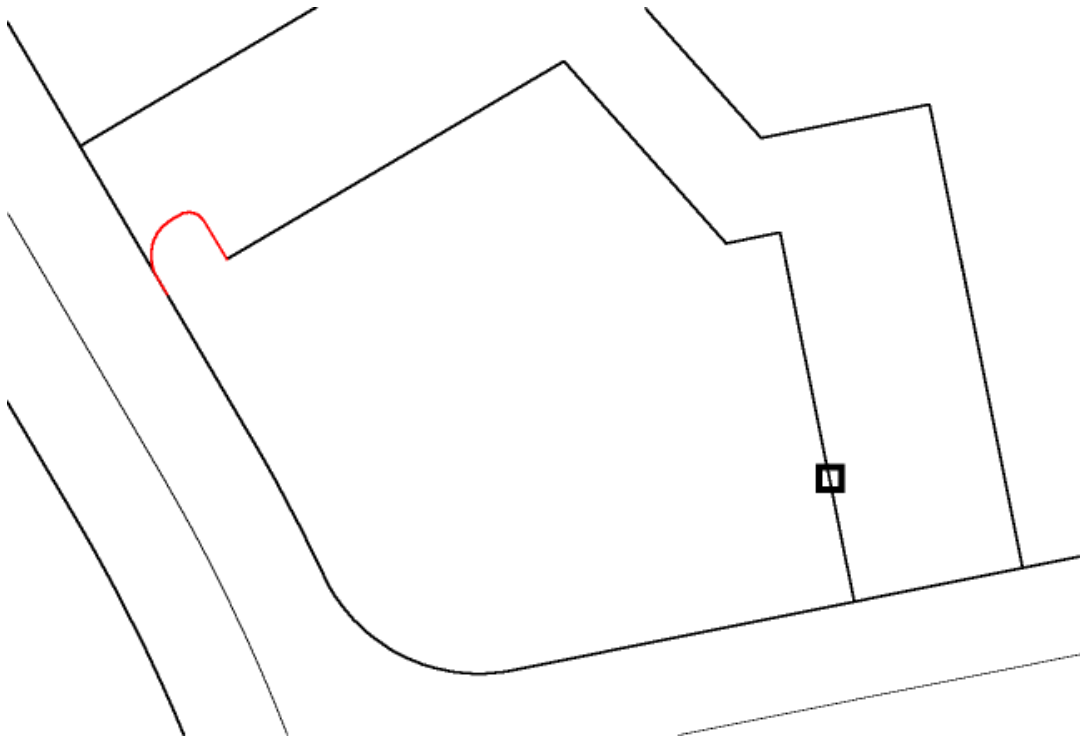
Select Parking Perimeter Polyline: *Select the entity to attach the parking curb item to. The command will use this initially selected polyline as the perimeter polyline through out the command.*

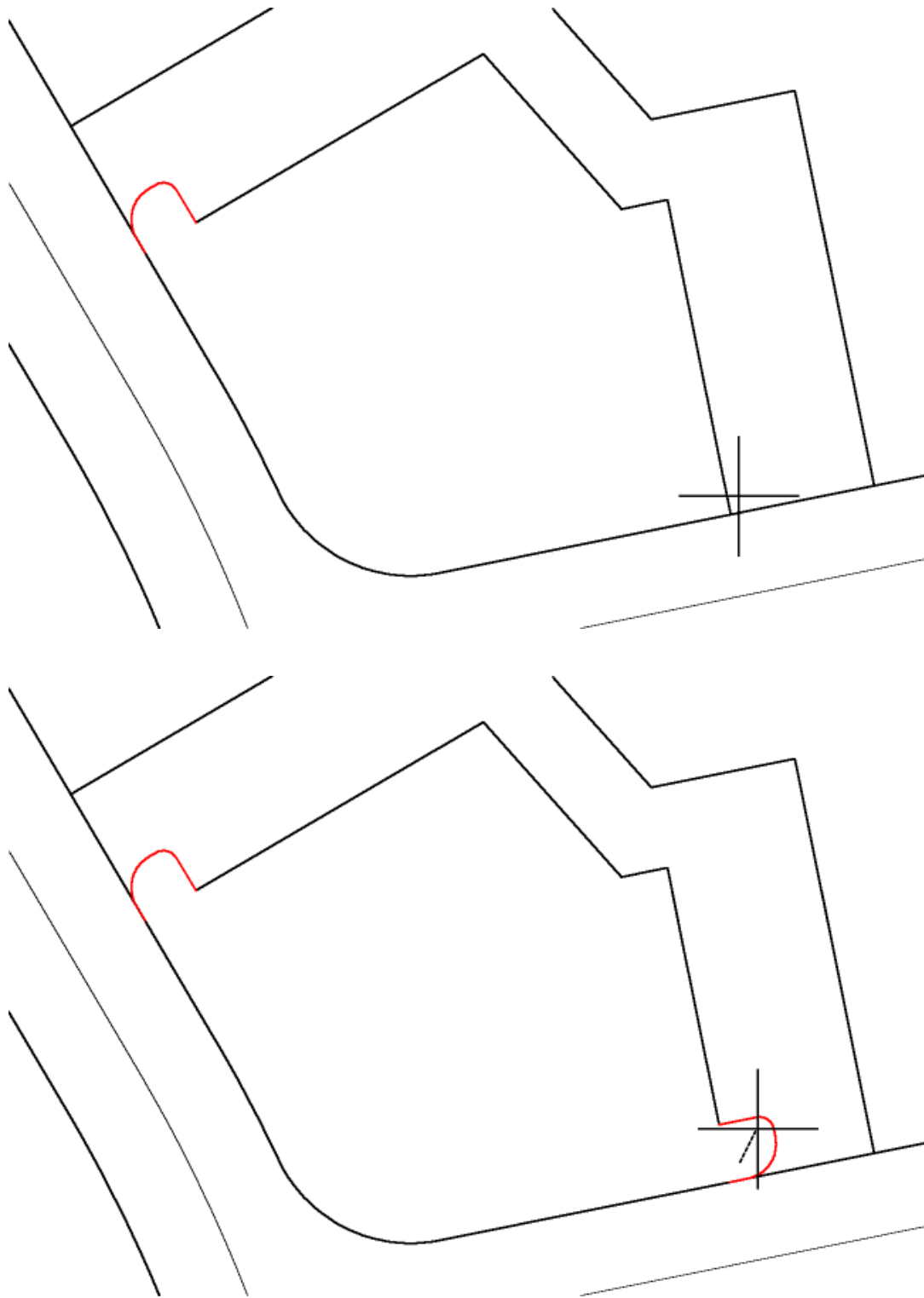
Entrance

Select end point to attach Entrance: *Select an end point of the entity. The entrance will be located at the closest end point of the point selected.*

Select polyline to align Entrance (Enter for None): *Select another polyline to align the entrance. If selected the entrance will join to the selected polyline.*

Pick side: *Pick a point for the entrance, the entrance will be drawn toward the point selected.*





Half-Island

Optional for By Pick

Pick center point for Half-Island: *Pick a point for the placement of the half-island.*

Pick Side: *Pick a point for the side of the half-island, the half-island will be drawn toward the point selected.*

Optional for At Stall Count and At Distance

Pick reference point for Half-Island: *Pick the point where the distance or stall count is calculated from along the*

entity selected.

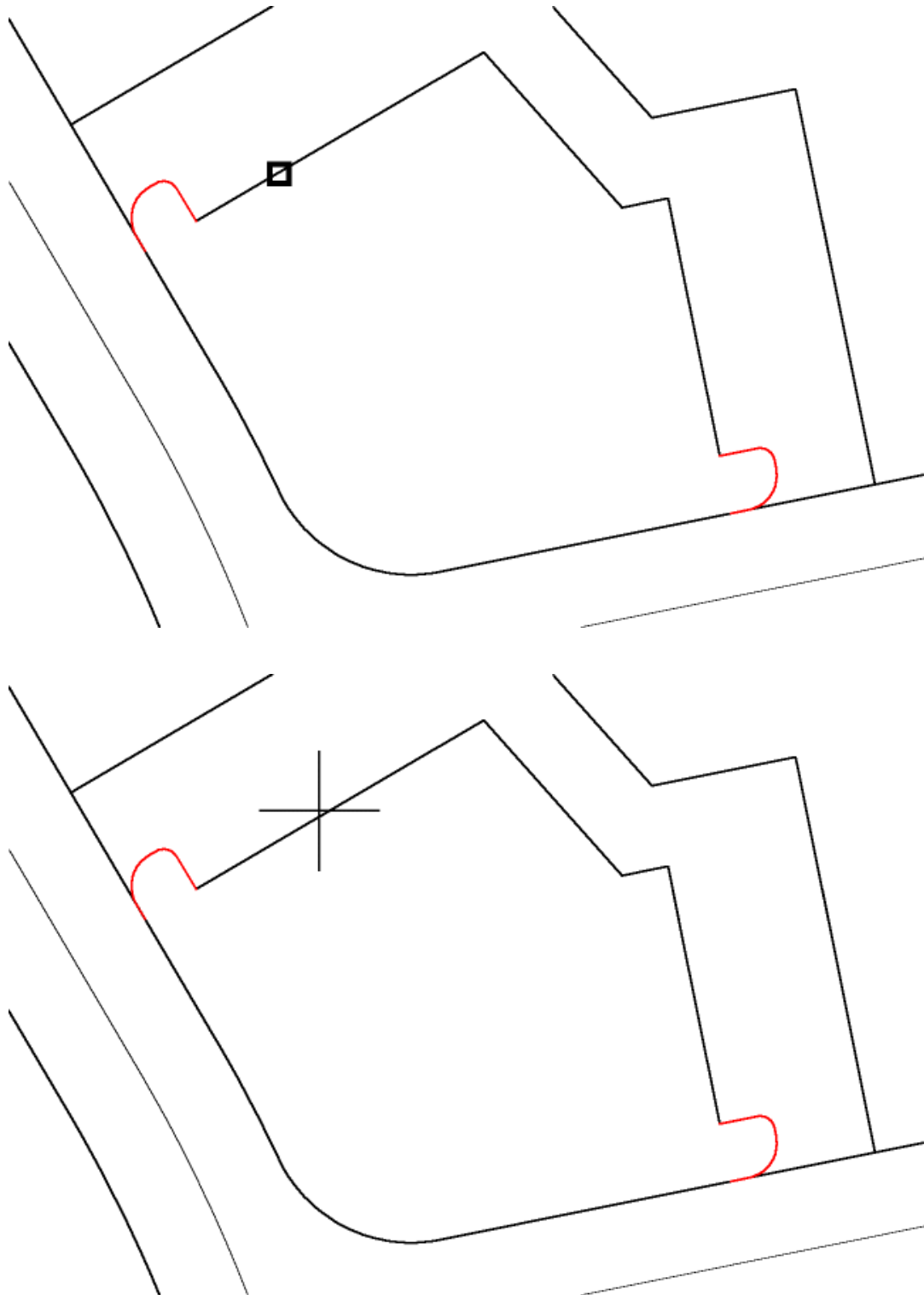
Pick side and direction: Pick a point for the side of the half-island, the half-island will be drawn on the side of the perimeter toward the point selected.

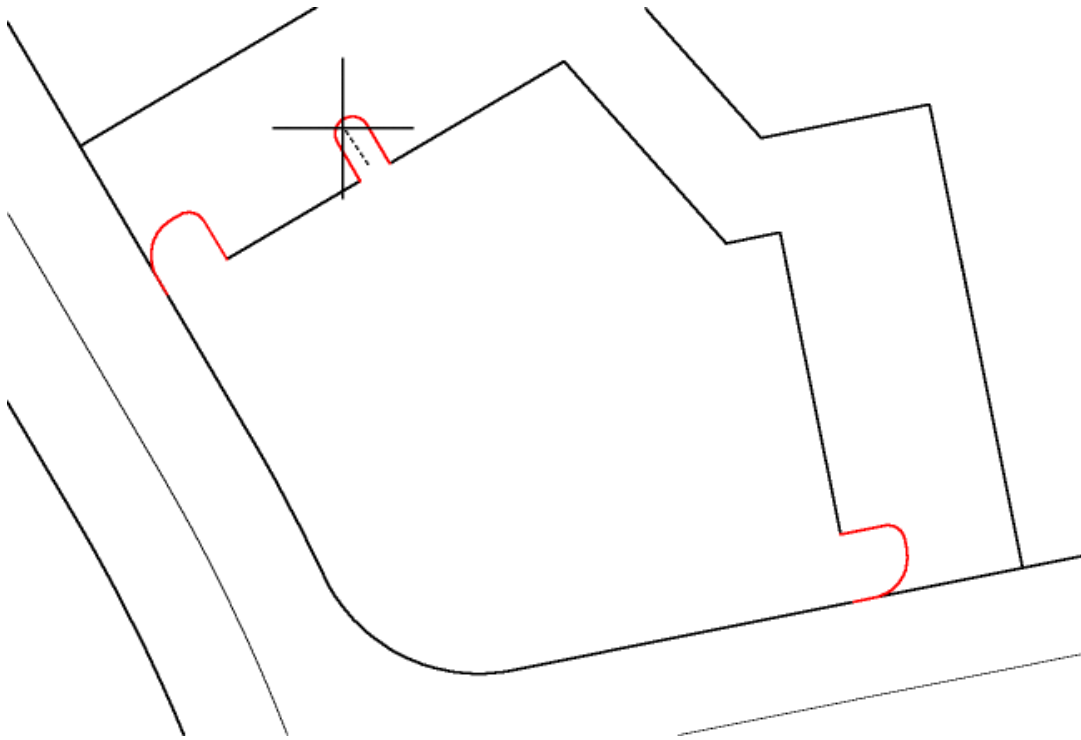
Optional for At Distance

Half-Island Spacing: Graphically pick or input the distance to the half-island.

Optional for At Stall Count

Parking Stall Count: Input the number of parking stalls fit before the half-island.

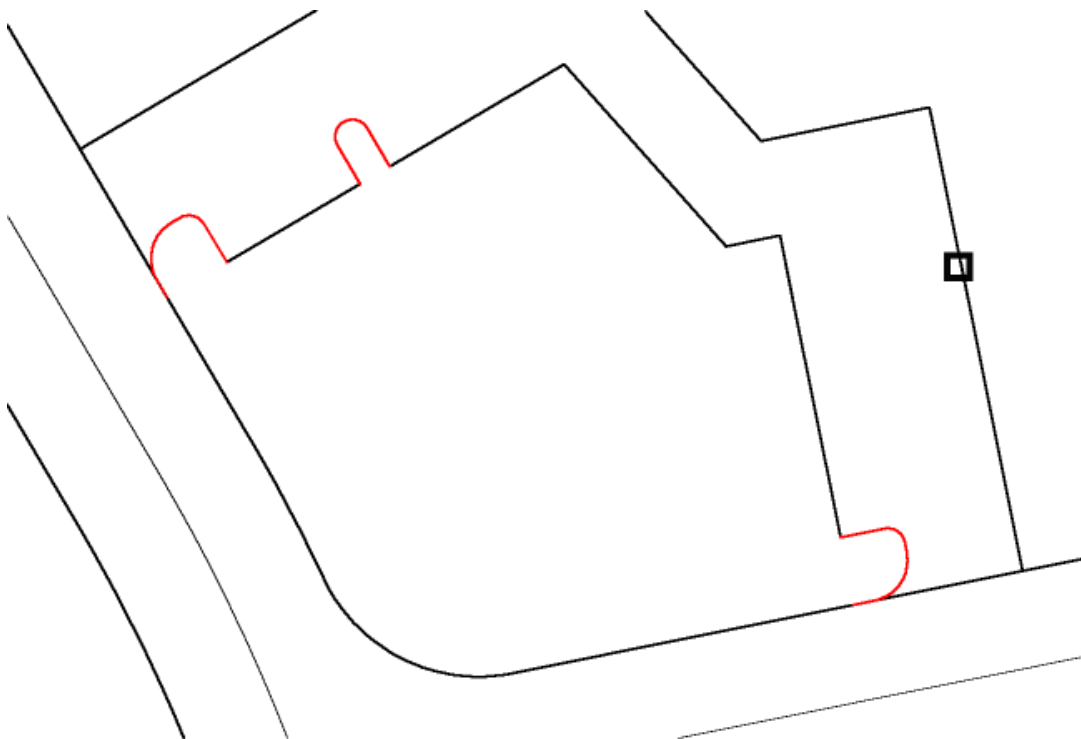


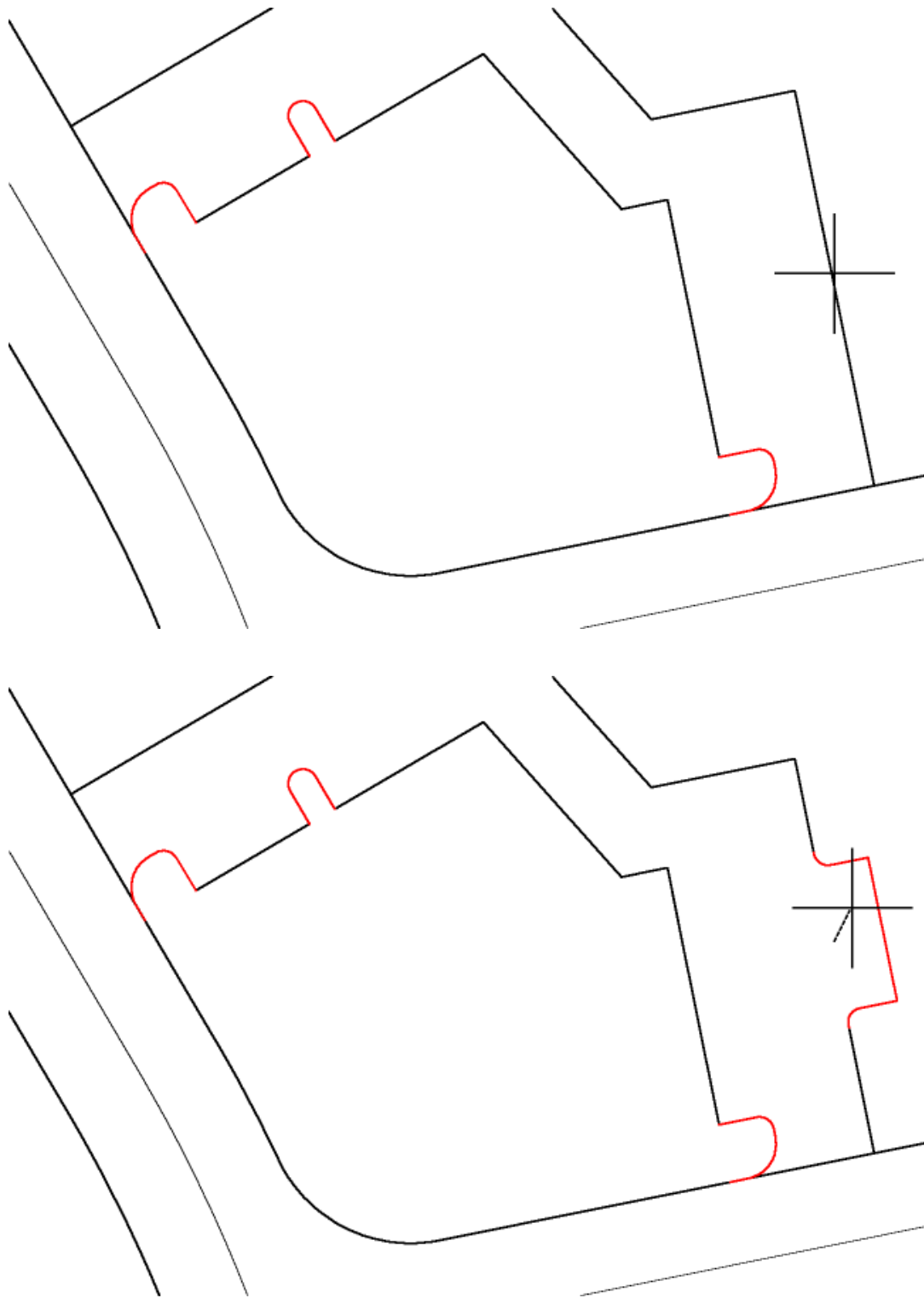


Bump-Out

Pick center point for Bump-Out: *Pick a point for the placement of the bump-out.*

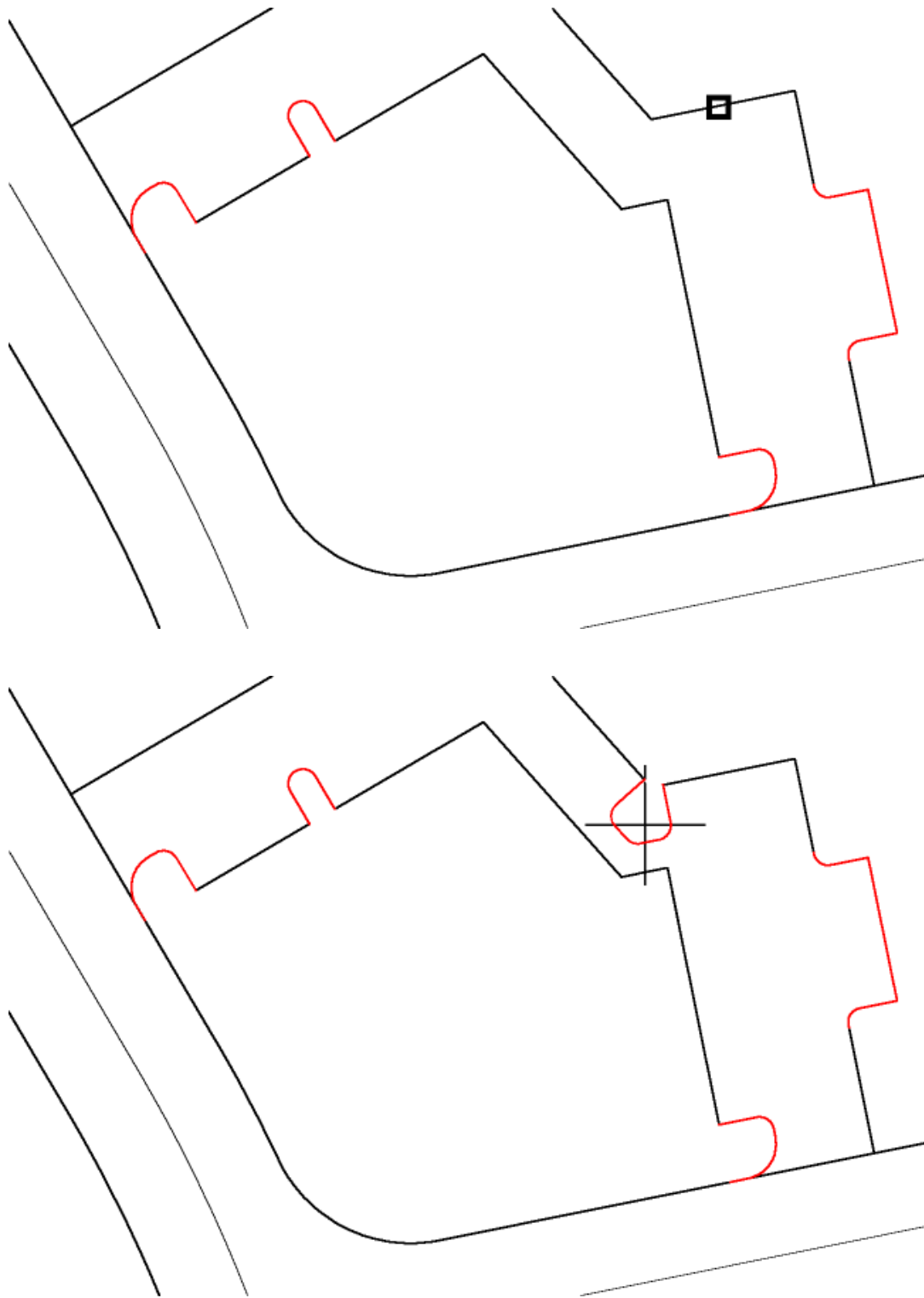
Pick Side: *Pick a point for the side of the bump-out, the bump-out will be drawn toward the point selected.*





Exterior Corner

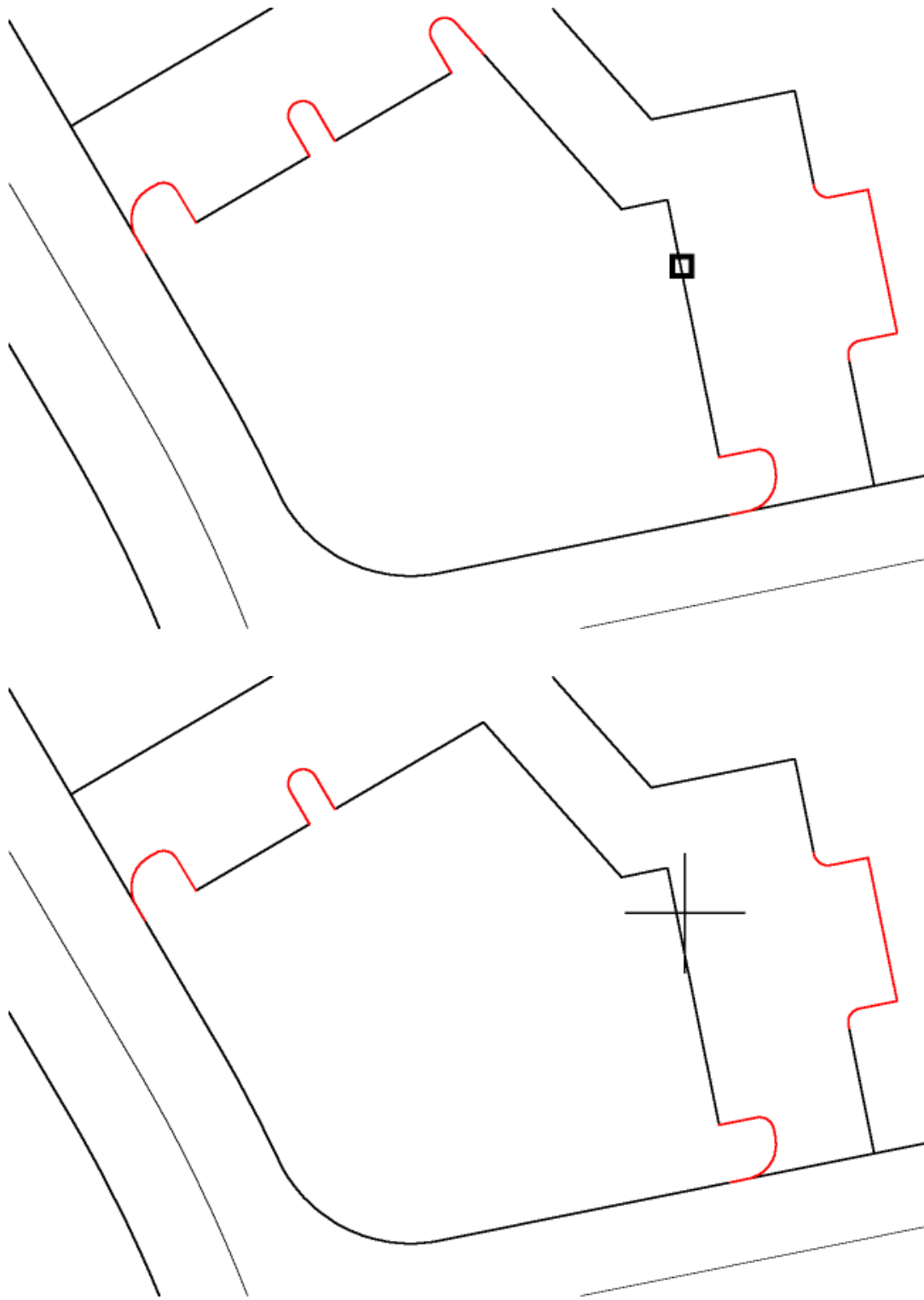
Pick a to attach Exterior Corner: *Pick a point, the corner will be located at the closest vertex of the point selected.*

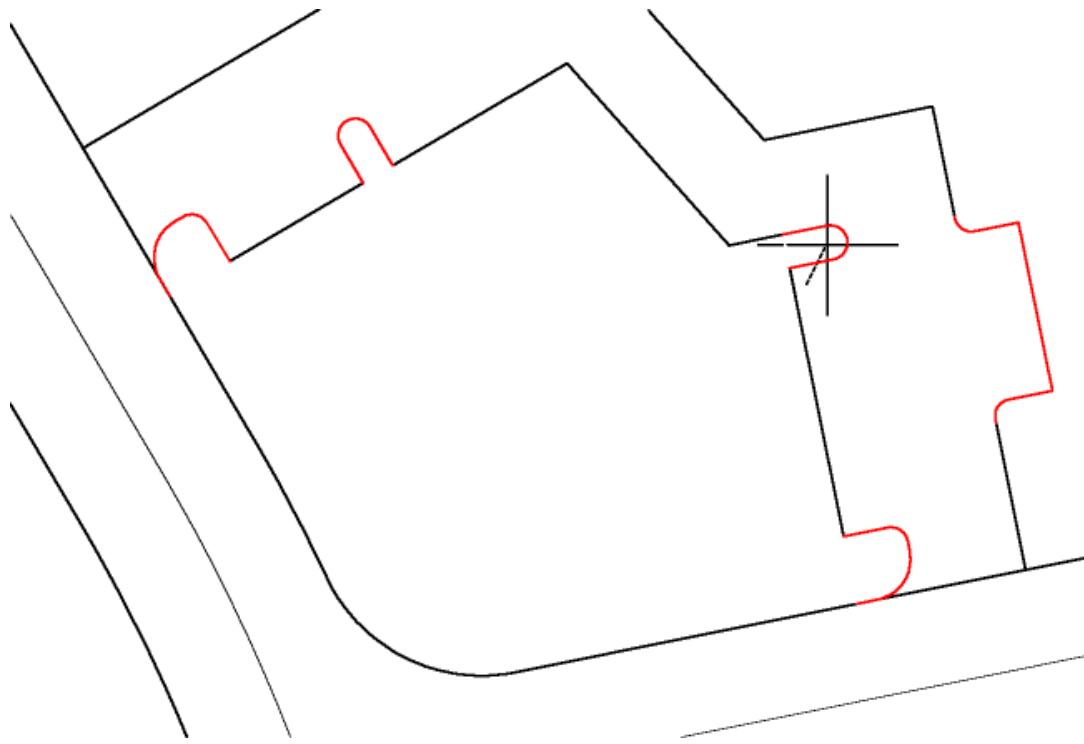


Extend Corner

Pick a point to attach Extend Corner: *Pick a point, the corner will be located at the closest vertex of the point selected.*

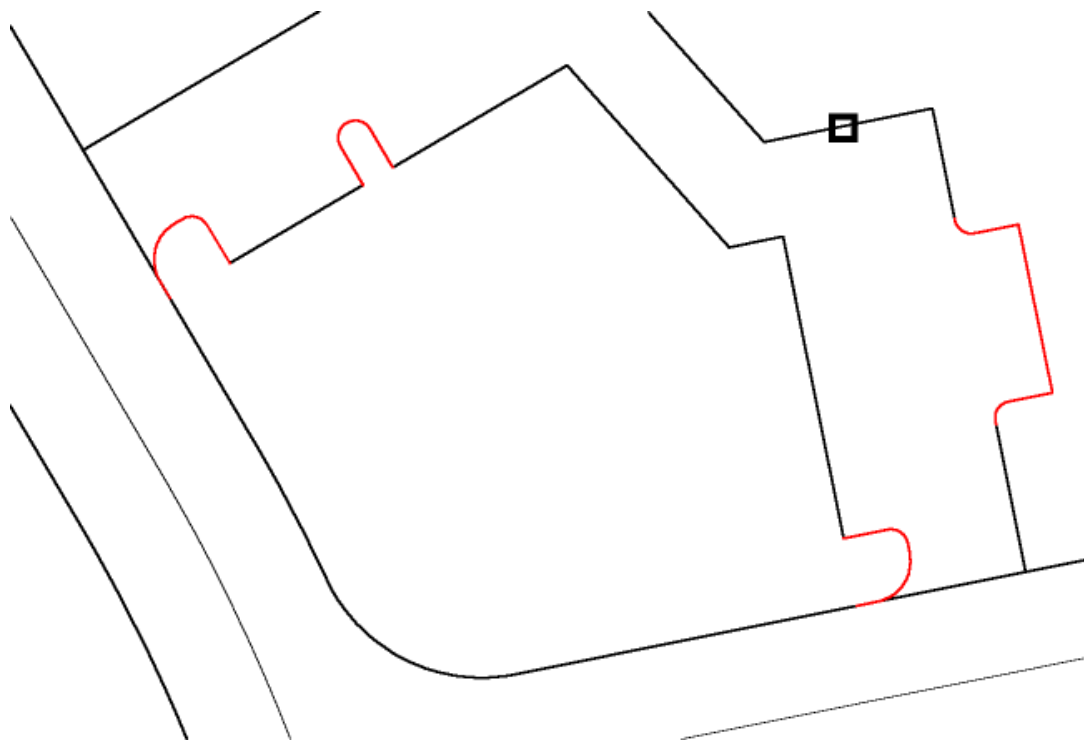
Pick Side: *Pick a side point for the corner to be extended.*

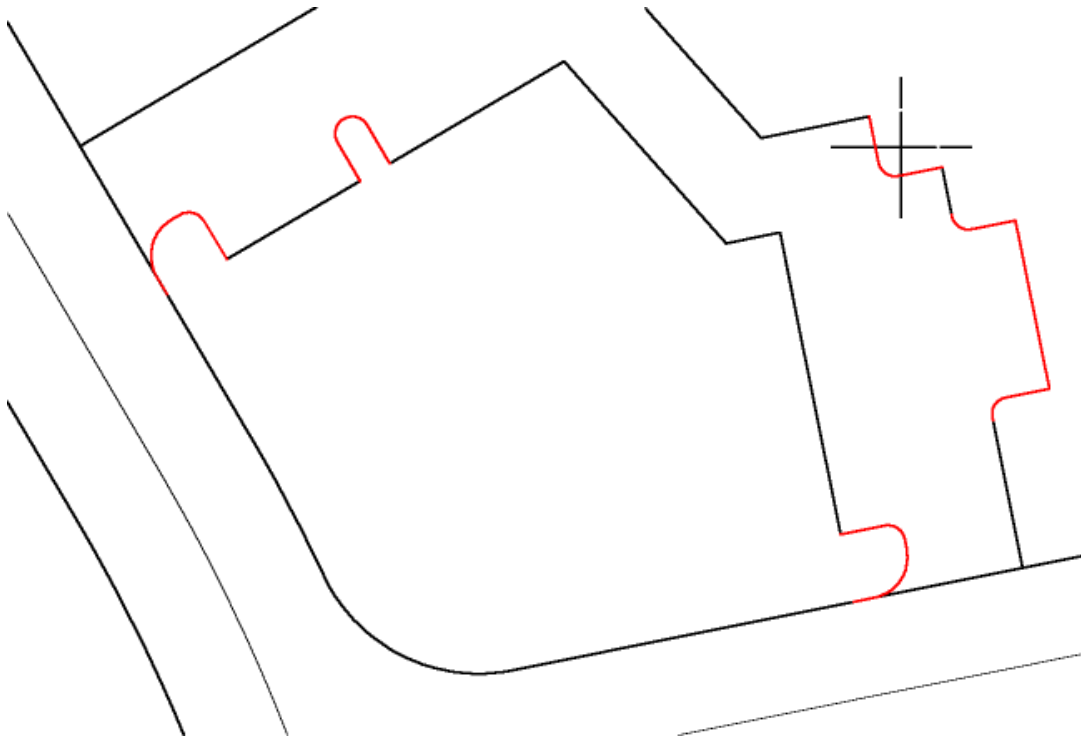




Interior Corner

Pick a to attach Interior Corner: *Pick a point, the corner will be located at the closest vertex of the point selected.*





Pick a point to attach *last curb item* or (Enter to End)[Options/Undo]:

Options will allow you to return to the Parking Perimeter Tool to change curb item or parameters.

Undo will undo the previous *curb item* placed.

Depth: *Graphically pick or input the depth of the curb item.*

Width: *Graphically pick or input the width of the curb item.*

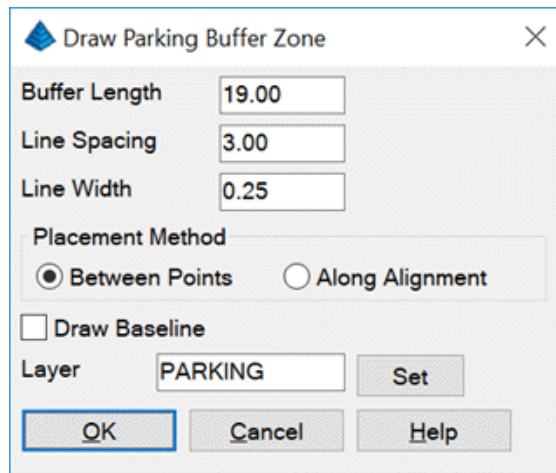
Pulldown Menu Location: Area/Layout > Parking Utilities > Parking Perimeter Tools

Keyboard Command: park_tool

Prerequisite: Parking perimeter linework

Parking Buffer Zone

This command draws a striped buffer zone for next to parking stalls.



Buffer Length: Sets the depth for the buffer which is typically the same as the parking stall length.

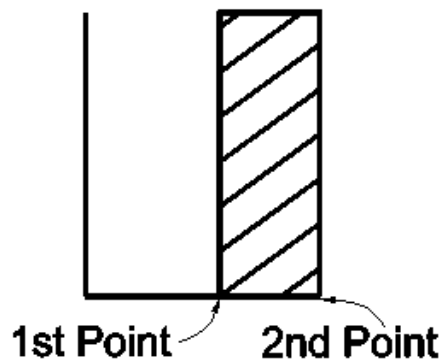
Line Spacing: Sets the interval for the stripes.

Line Width: Sets the line width property of the buffer lines. Using a Line Width helps with viewing the buffer lines in the 3D Viewer.

Placement Method: Indicate the method to set the baseline for the buffer zone which controls the buffer width and alignment. The Along Alignment method prompts to select a line, arc or polyline and supports an alignment that changes direction including a curve.

Draw Baseline: This option draws a baseline in addition to the buffer side lines.

Layer: Specify the layer on which parking lines should be placed or click the Set button to choose an existing layer.



Prompts

Starting Point?

Pick point or point number: *Pick a point*

Ending Point?

Pick point or point number: *Pick a point*

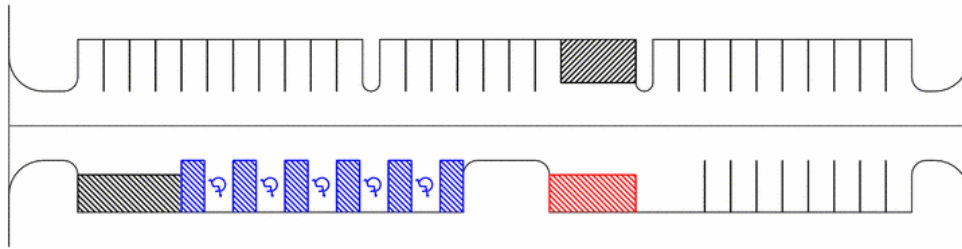
Pulldown Menu Location: Area/Layout > Layout Utilities

Keyboard Command: parking_buffer

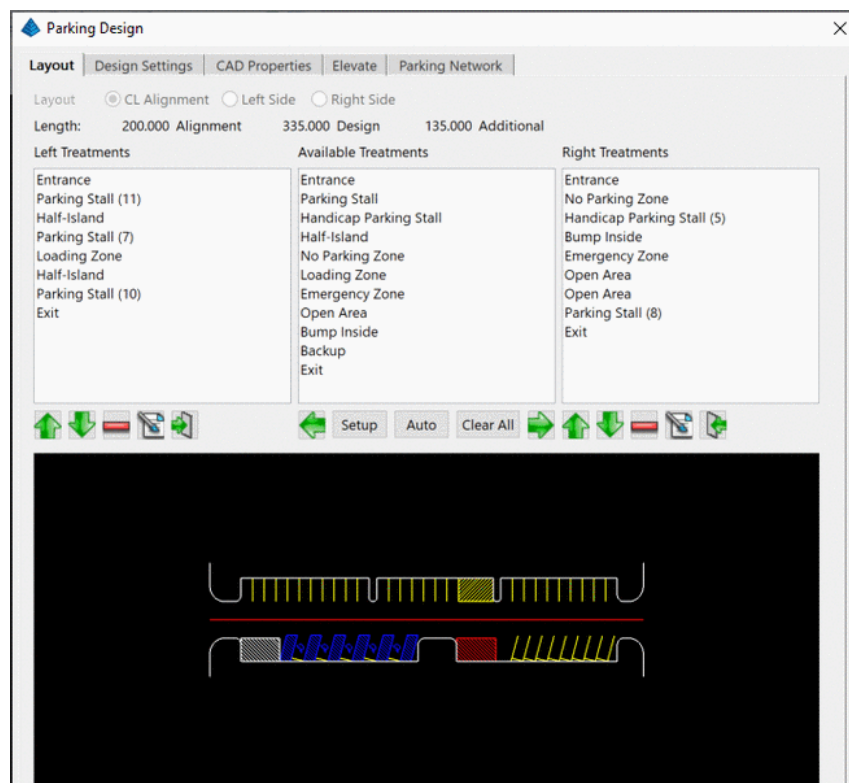
Prerequisite: None

Parking Design

Parking Design is a set of tools to create a network of parking treatments.



Layout tab



Layout:

CL Alignment Design by centerline path

Left Side Design Left side only

Right Side Design Right side only

Length:

Alignment Initial length of selection

Design Design length of treatments

Difference Additional length or Remaining length

Left Treatments List of parking treatments assigned to left side

Up/Down Reorder the selected parking treatment

Remove Remove the selected parking treatment

Edit Edit selected parking treatment

Mirror Mirror left parking treatments to the right

Available Treatments List of parking treatments available for design

Left Add selected treatment to the left side of design

Setup Edit selected parking treatment's default values

Auto Open the Automation settings

Clear All Remove all treatments from parking design

Right Add selected treatment to the right side of design

Right Treatments List of parking treatments assigned to right side

Up/Down Reorder the selected parking treatment

Remove Remove the selected parking treatment

Edit Edit selected parking treatment

Mirror Mirror right parking treatments to the left

Treatment Setup/Edits

Treatment edit is the same for edit of a side's treatment and setup of available treatments.

Entrance

The screenshot shows a dialog box titled "Parking Treatment" with a close button (X) in the top right corner. The dialog is divided into two main sections. On the left, there is a list of settings for the "Entrance" parking treatment:

Parking Treatment	Entrance
Parking Depth	18.000
Distance to 1st Space	24.000
Entrance Radius	12.000
Radius	5.000

On the right, there is a diagram of a parking stall with a semi-circular entrance. The diagram is outlined in red. Labels with arrows point to various parts of the diagram: "Distance to 1st Space" (the vertical distance from the entrance to the start of the parking space), "Parking Depth" (the horizontal distance from the entrance to the end of the parking space), "Entrance Radius" (the radius of the semi-circular entrance), and "Radius" (the radius of the inner curve of the entrance).

At the bottom of the dialog, there are five buttons: "Add Left", "Add Right", "OK", "Cancel", and "Help".

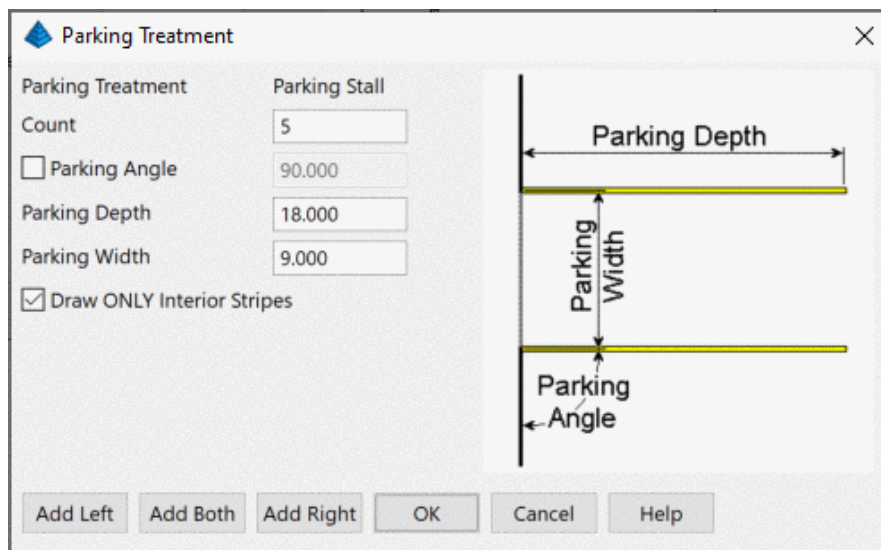
Parking Depth: Distance from parking perimeter to inside of entrance.

Distance to 1st Space: Distance from entrance to start of parking treatments.

Entrance Radius: Radius on the outside of entrance.

Radius: Radius on the inside of entrance.

Parking Stall



Count: Number of parking stalls.

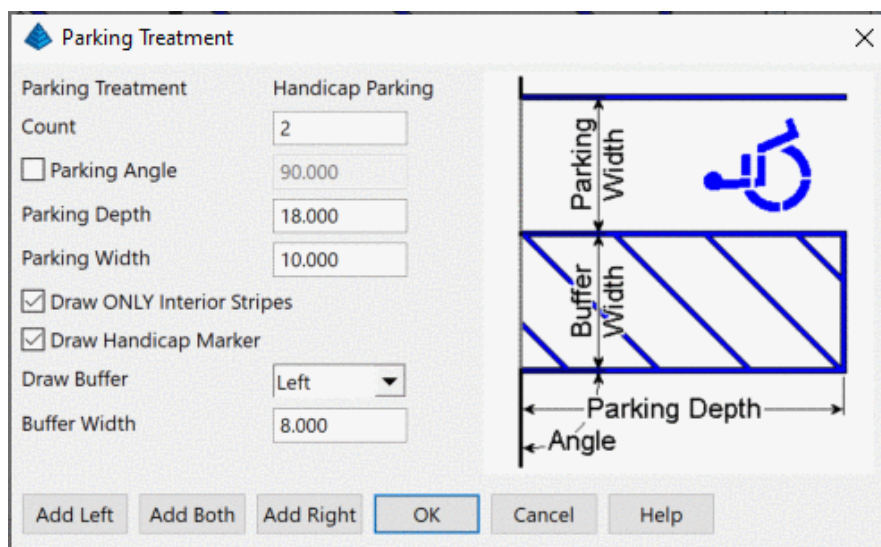
Parking Angle: Toggle to set the angle of parking stall.

Parking Depth: Depth of parking stall.

Parking Width: Width of parking stall.

Draw ONLY Interior Stripes: Include or exclude first and last stripe lines.

Handicap Parking



Count: Number of handicap parking stalls.

Parking Angle: Toggle to set the angle of parking stall.

Parking Depth: Depth of handicap parking stall.

Parking Width: Width of handicap parking stall.

Draw ONLY Interior Stripes: Include or exclude first and last stripe lines.

Draw Handicap Marker: Draw the handicap marker in the parking stall.

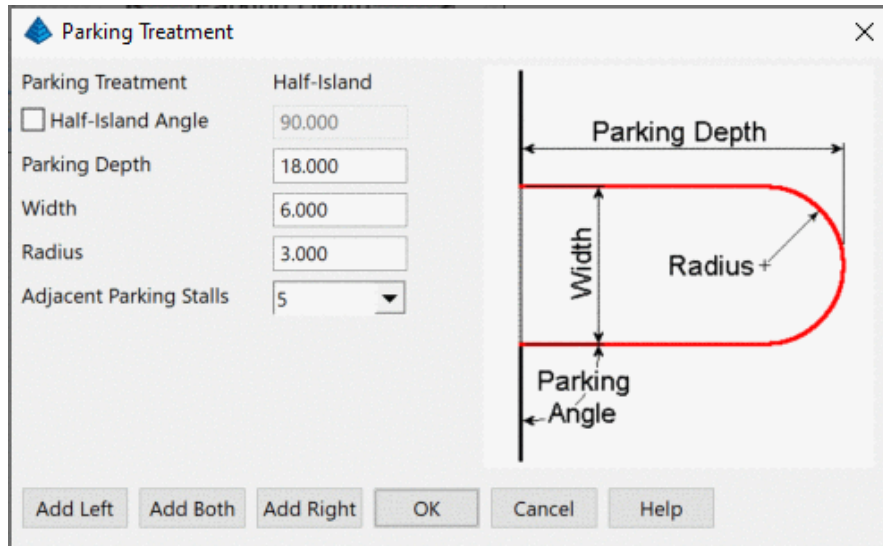
Draw Buffer: Location of parking buffer **None** No buffer is drawn

Left Buffer is drawn on the left side of stall

Right Buffer is drawn on the right side of stall

Both Buffer is drawn on both sides of stall
Buffer Width: Width of parking buffer.

Half-Island



Half-Island Angle: Toggle to set the angle of half-island.

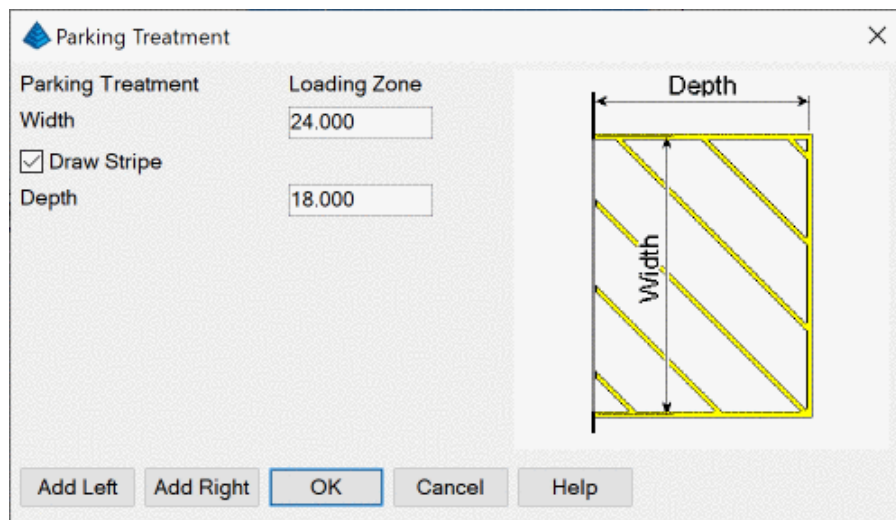
Parking Depth: Distance from parking perimeter to inside of half-island.

Width: Width of half-island.

Radius: Radius on inside of half-island.

Adjacent Parking Stalls: (Optional for Available Treatment's Edit) Number of parking stalls to add if none are adjacent to half-island.

No Parking Zone/Loading Zone/Emergency Zone

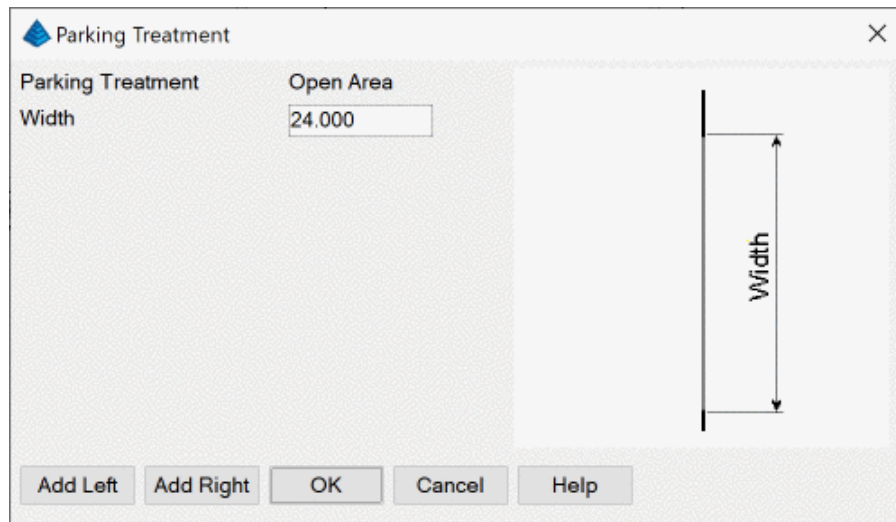


Width: Width of parking treatment.

Draw Stripe: Draw the Stripe for parking treatment.

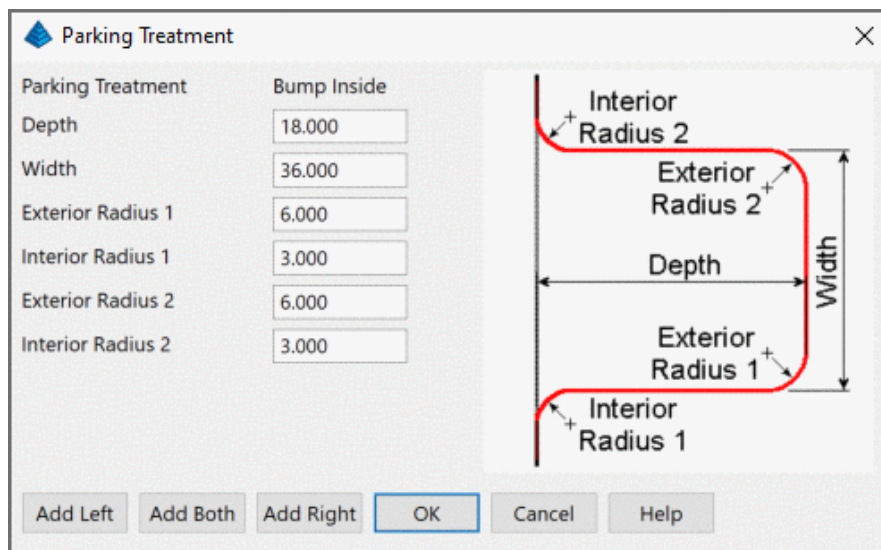
Depth: Distance from parking perimeter to inside of treatment.

Open Area



Width: Width of open area.

Bump Inside



Depth: Distance from parking perimeter to inside of bump inside.

Width: Width of bump inside.

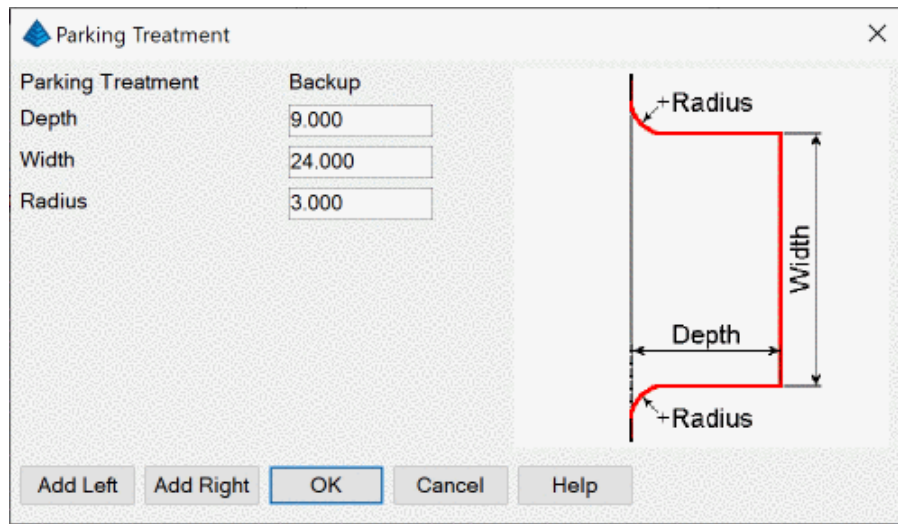
Exterior Radius 1: Radius of first side on the inside of bump inside.

Interior Radius 1: Radius of first side on the outside of bump inside.

Exterior Radius 2: Radius of second side on the inside of bump inside.

Interior Radius 2: Radius of second side on the outside of bump inside.

Backup

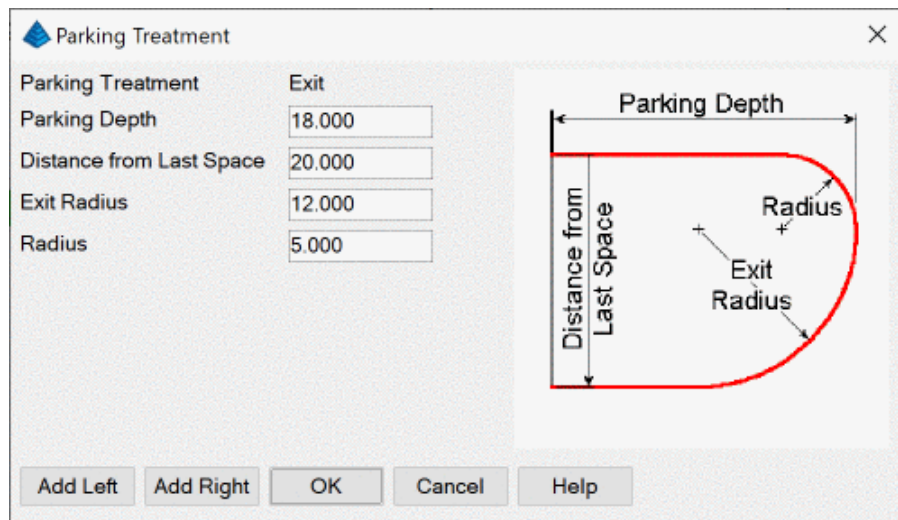


Depth: Distance from parking perimeter to inside of backup.

Width: Width of backup.

Radius: Radius on the outside of backup.

Exit



Parking Depth: Distance from parking perimeter to inside of exit.

Distance from Last Space: Distance from last parking treatment to exit.

Exit Radius: Radius on the outside of exit.

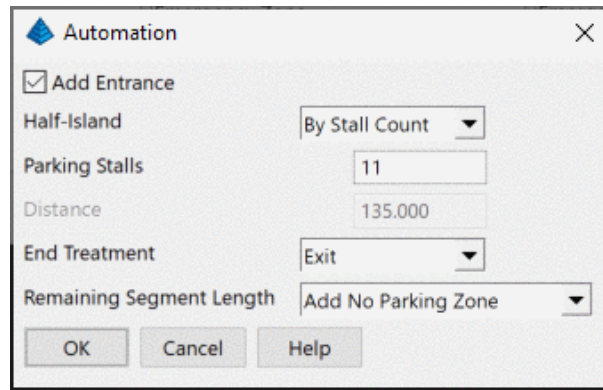
Radius: Radius on the inside of exit.

Add Left: (Optional for Available Treatment's Edit) Adds the treatment to the left list.

Add Right: (Optional for Available Treatment's Edit) Adds the treatment to the right list.

Automation dialog

The Automation dialog can automatically generated based of selected parameters. When placing treatments it will use the defaults of the treatment from the Layout tab. Various options can be selected to get a start of a parking design based on the initial selected length.



Add Entrance: If the default Entrance is added to both the left and right sides of the automatic design.

Half-Island: Method for adding half-islands to the automatic design.

None No half-islands will be automatically added to the automatic design.

By Stall Count Half-islands will be added after a number of stalls.

By Distance Half-islands will be added after a specified distance.

End Treatment: Type of end treatment to use when generating an automatic design.

None No end treatment will be automatically added to the automatic design.

Backup A backup is added to the design to allow for vehicles to backup and exit.

Exit An exit is added to both the left and right sides.

Remaining Segment Length: The way that the remaining length is used for treatment generation.

Add No Parking Zone Add no parking zone of remaining length.

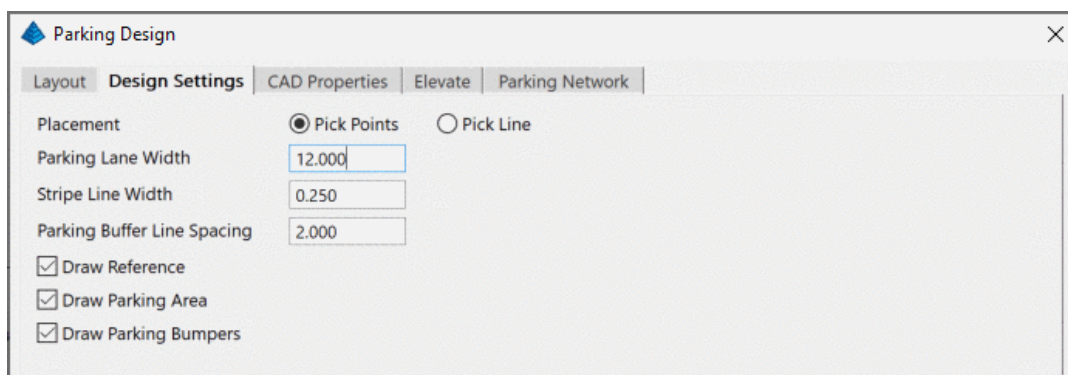
Add Open Area Add open area of remaining length.

Distribute Distribute the remaining length among last treatment.

Append to End Treatment Add remaining length to the segment end treatment.

OK Automatically generate the parking design based on the parameters specified.

Design Settings tab



Placement:

Pick Points Graphically select the start and end points for parking alignment.

Pick Line Graphically select a line for parking alignment.

Parking Lane Width: Width of the drive lane of parking.

Stripe Line Width: Line width of the parking stripes.

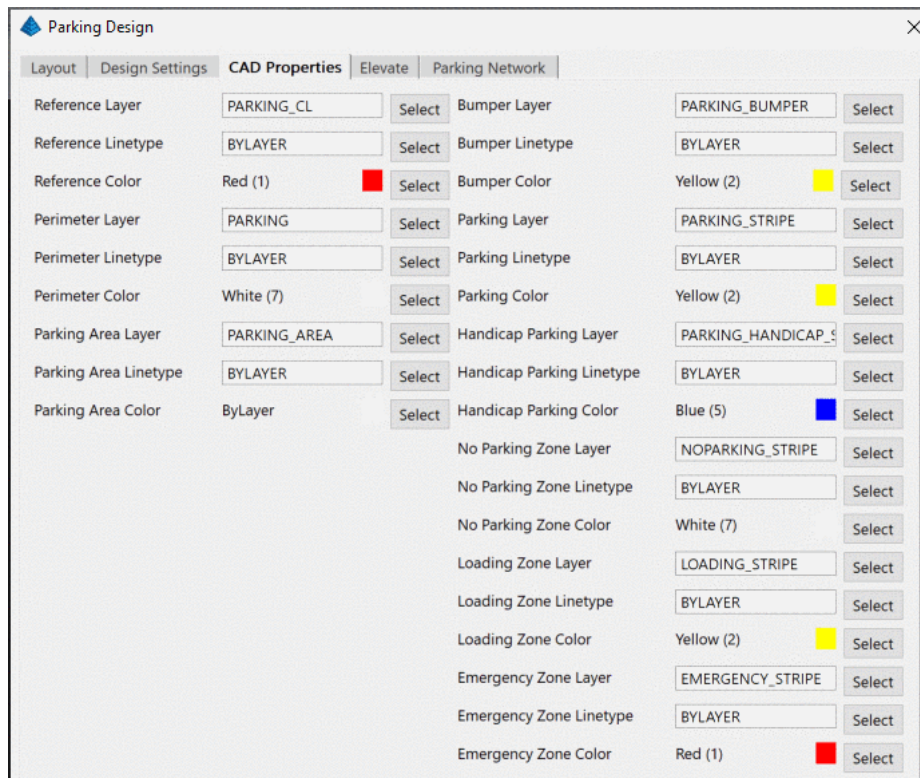
Parking Buffer Line Spacing: Distance between buffer line stripes.

Draw Reference: Toggle to draw the reference alignment.

Draw Parking Area: Toggle to draw the closed parking area.

Draw Parking Bumpers: Toggle to draw the parking bumpers on angled stalls.

CAD Properties tab



Reference Layer: Layer to draw the reference CL.

Reference Linetype: Linetype to draw the reference CL.

Reference Color: Color to draw the reference CL.

Perimeter Layer: Layer to draw the perimeter.

Perimeter Linetype: Linetype to draw the perimeter.

Perimeter Color: Color to draw the perimeter.

Parking Area Layer: Layer to draw the parking area.

Parking Area Linetype: Linetype to draw the parking area.

Parking Area Color: Color to draw the parking area.

Bumper Layer: Layer to draw the parking bumpers.

Bumper Linetype: Linetype to draw the parking bumpers.

Bumper Color: Color to draw the parking bumpers.

Parking Layer: Layer to draw the parking stripes.

Parking Linetype: Linetype to draw the parking stripes.

Parking Color: Color to draw the parking stripes.

Handicap Parking Layer: Layer to draw the handicap parking stripes, buffer and marker.

Handicap Parking Linetype: Linetype to draw the handicap parking stripes, buffer and marker.

Handicap Parking Color: Color to draw the handicap parking stripes, buffer and marker.

No Parking Zone Layer: Layer to draw the no parking zones.

No Parking Zone Linetype: Linetype to draw the no parking zones.

No Parking Zone Color: Color to draw the no parking zones.

Loading Zone Layer: Layer to draw the loading zones.

Loading Zone Linetype: Linetype to draw the loading zones.

Loading Zone Color: Color to draw the loading zones.

Emergency Zone Layer: Layer to draw the emergency zones.

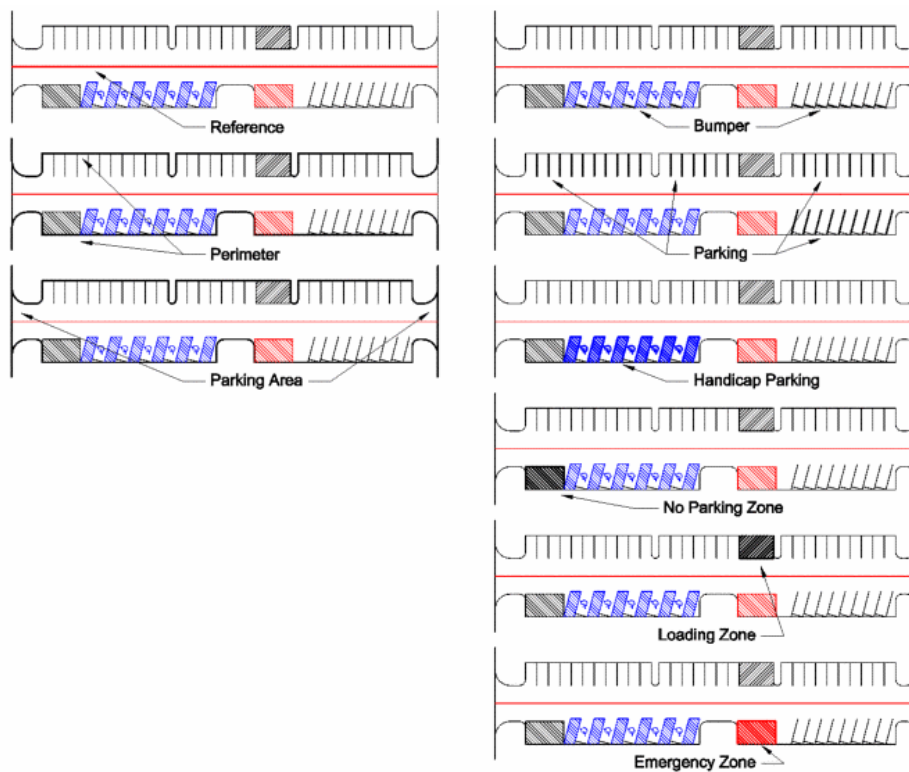
Emergency Zone Linetype: Linetype to draw the emergency zones.

Emergency Zone Color: Color to draw the emergency zones.

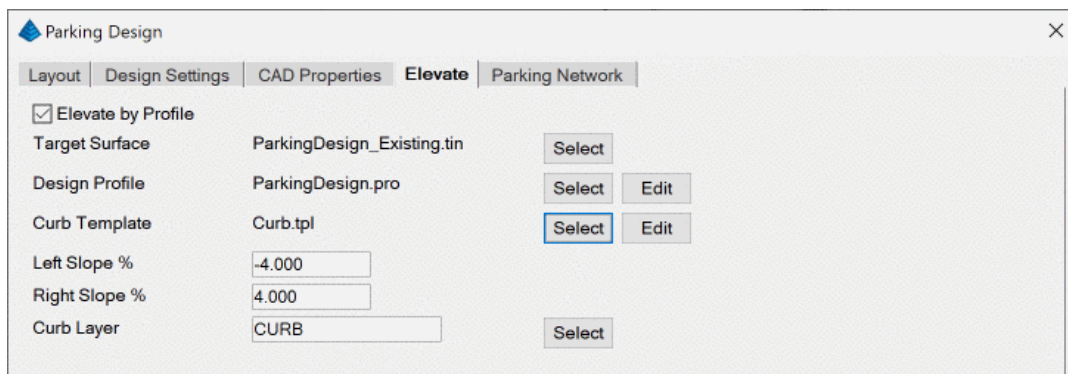
Select: (Layer) Select the corresponding layer from list.

Select: (Linetype) Select the corresponding linetype from list.

Select: (Color) Select the corresponding color from list.



Elevate tab



Elevate by Profile: Elevate the perimeter linework by the profile and side slopes.

Target Surface: Existing Surface to target. **Select:** Use this button to browse to and select the Existing Surface file to be used for the Road Network. Either a TIN or FLT triangulation file are accepted as valid surfaces, both of which can be made within the command Triangulate and Contour. For speed, it is recommended that the binary TIN file format be selected.

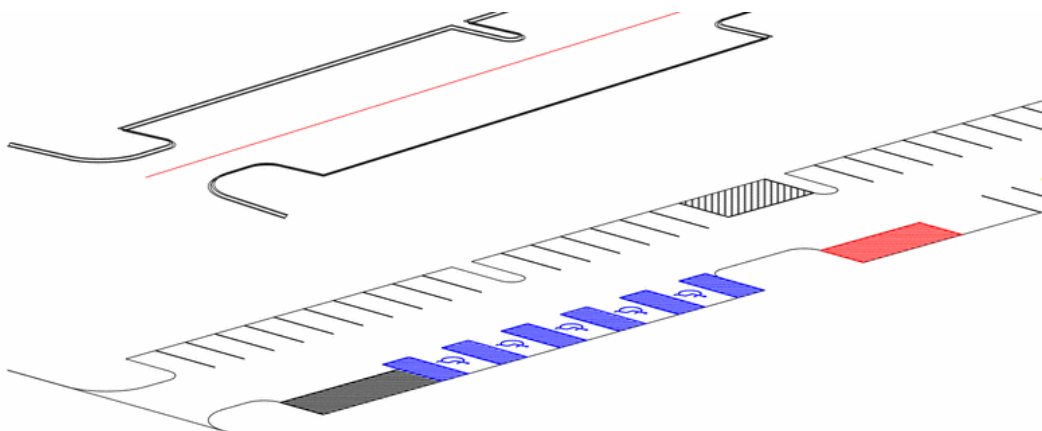
Design Profile: Design Profile of parking alignment. **Select:** Select the design profile file. **Edit:** Edit the design profile.

Curb Template: Curb Template to draw along perimeter. **Select:** Select the curb template file. **Edit:** Edit the curb template.

Left Slope %: Left slope from profile.

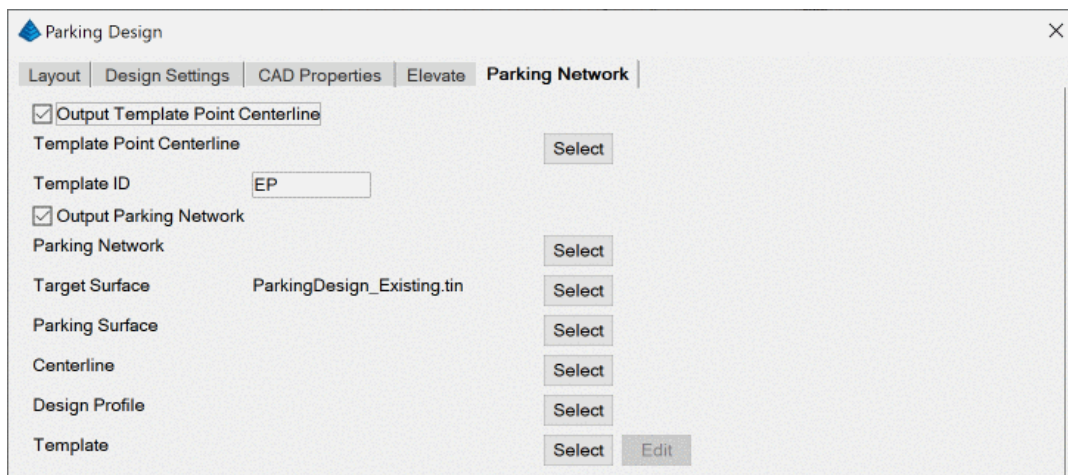
Right Slope %: Right slope from profile.

Curb Layer: Layer to draw the curb linework.



Parking Network tab

For integration with Road Network the parking design command can create the necessary data files to process with Road Network.



Output Template Point Centerline: Toggle for creation of a Template Point Centerline.

Template Point Centerline: Specified Template Point Centerline TPC file to create from the parking design.

Template ID: ID to use within the TPC file.

Output Parking Network: Toggle for creation of a Road Network RDN file from the parking design.

Parking Network: Specified Road Network RDN file to create from the parking design.

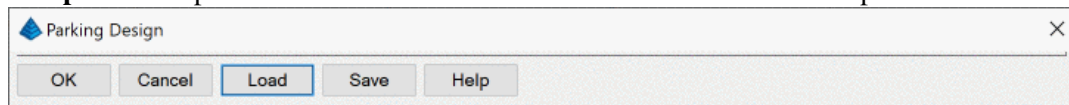
Target Surface: Existing Surface to target. **Select:** Use this button to browse to and select the Existing Surface file to be used for the Road Network. Either a TIN or FLT triangulation file are accepted as valid surfaces, both of which can be made within the command Triangulate and Contour. For speed, it is recommended that the binary TIN file format be selected.

Parking Surface: Parking Surface to create. **Select:** Use this button to browse to and select the Parking Surface file to be output from Road Network. Either a TIN or FLT triangulation file are accepted as valid surfaces For speed, it is recommended that the binary TIN file format be selected.

Centerline: Parking alignment centerline to create. **Select:** Use this button to browse to and select the Centerline file to be output for Road Network.

Design Profile: Parking alignment centerline's profile to create. **Select:** Use this button to browse to and select the Profile file to be output for Road Network.

Template: Template to use for the Road Network. **Select:** Select the template file. **Edit:** Edit the template.



The **Save** and **Load** buttons save and recall the parking design settings to a .PKD file.

Pick Start Point [CL-Centerline/P-Polyline] (Enter for none): Pick the start point for the parking alignment.

Pick Next Point: Pick the next point of the parking alignment.

Pick Next Point (enter to end): Pick additional points of the parking alignment or enter to end point selection.

Centerline Select the centerline file to use for the parking alignment.

Polyline Select the line for the parking alignment. The parking design will start at the line start point draw the direction that the line is drawn.

Optional Prompts

Select polyline to align Entrance (Enter for None): Select another polyline to align the entrance. If selected the entrance will join to the selected polyline.

Select polyline to align Exit (Enter for None): Select another polyline to align the exit. If selected the exit will join to the selected polyline.

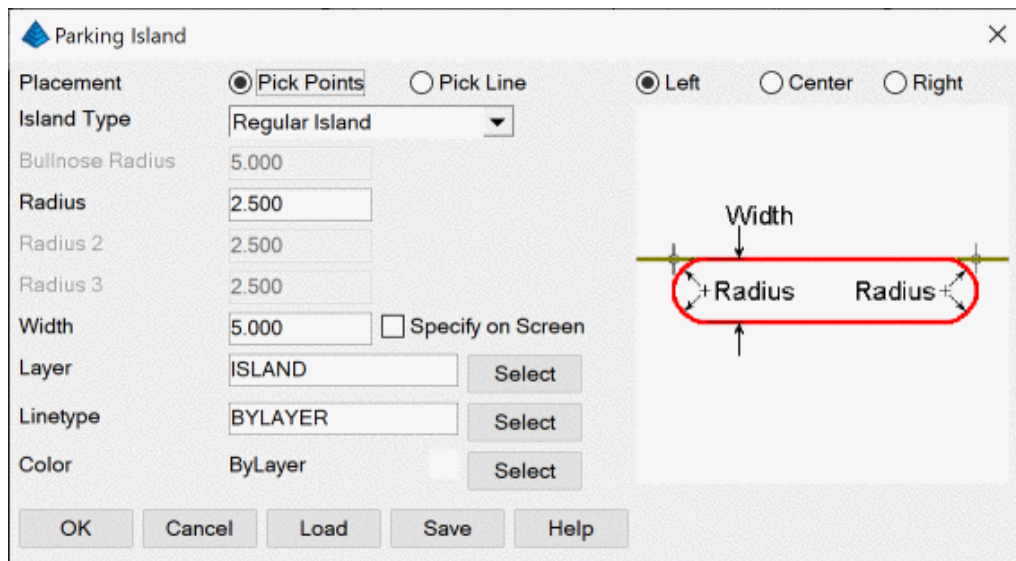
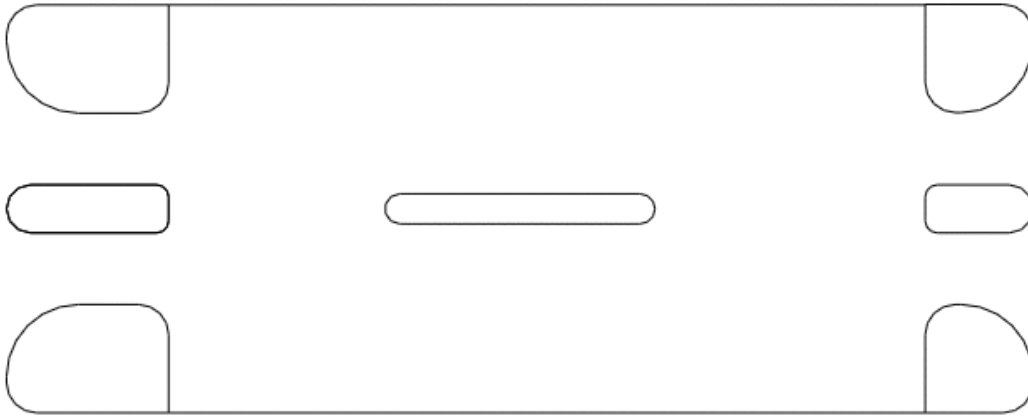
Pulldown Menu Location: Area/Layout > Parking Utilities > Design Parking Along Alignment

Keyboard Command: park_design

Prerequisite:

Parking Island

Parking Design is a set of tools to create a network of parking treatments.



Placement:

Pick Points Graphically select the start and end points for parking island.

Pick Line Graphically select a line for parking island.

Left The island will be aligned to the left of the selection.

Center The island will be drawn centered on the selection.

Right The island will be aligned to the right of the selection.

Island Type Type of island to create

Regular Island Regular cegar shaped island.

Bullnose Island Bullnose shape island.

Wedge Island Wedge shaped island.

Bullnose Radius Optional large radius on the bullnose island.

Wedge Radius Optional large radius on the wedge island.

Radius Radius on the regular island, smaller radius of bullnose island.

Radius 1 Optional radius 1 of the wedge island.

Radius 2 Optional radius 2 of the wedge island.

Radius 3 Optional radius 3 of the wedge island. **Width** Width of island.

The **Save** and **Load** buttons save and recall the parking island settings to a .ISLAND file.

Pick Start Point:

Pick End Point: *Pick the start and end points for the parking island.*

Select Line for Island: *Select the line for the parking island. The parking island will start at the line start point and be drawn the length of the line.*

Width: *Graphically pick or input the width of the island.*

Repeat prompt [**Options/Undo**] (**Enter to End**):

Options will allow you to return to the Parking Island to change type or parameters.

Undo will undo the previous island placed.

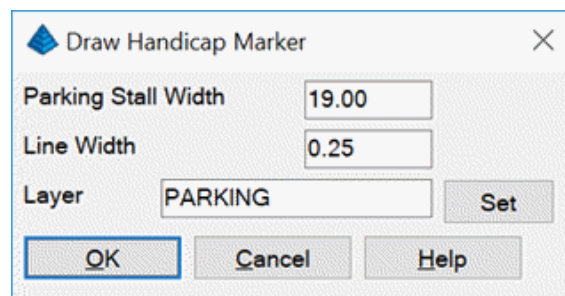
Pulldown Menu Location: Area/Layout > Parking Utilities > Parking Island

Keyboard Command: park_island

Prerequisite: None

Handicap Marker

This command draws a handicap symbol for parking stalls.



Parking Stall Width: Indicate the stall width which is used to size the marker.

Line Width: Sets the line width property of the parking lines. Using a Line Width helps with viewing the parking lines in the 3D Viewer.

Layer: Specify the layer on which parking lines should be placed or click the Set button to choose an existing layer.



Prompts

Pick point: *Pick a point for the marker center*

Pick rotation: *Pick a point*

Pulldown Menu Location: Area/Layout > Layout Utilities

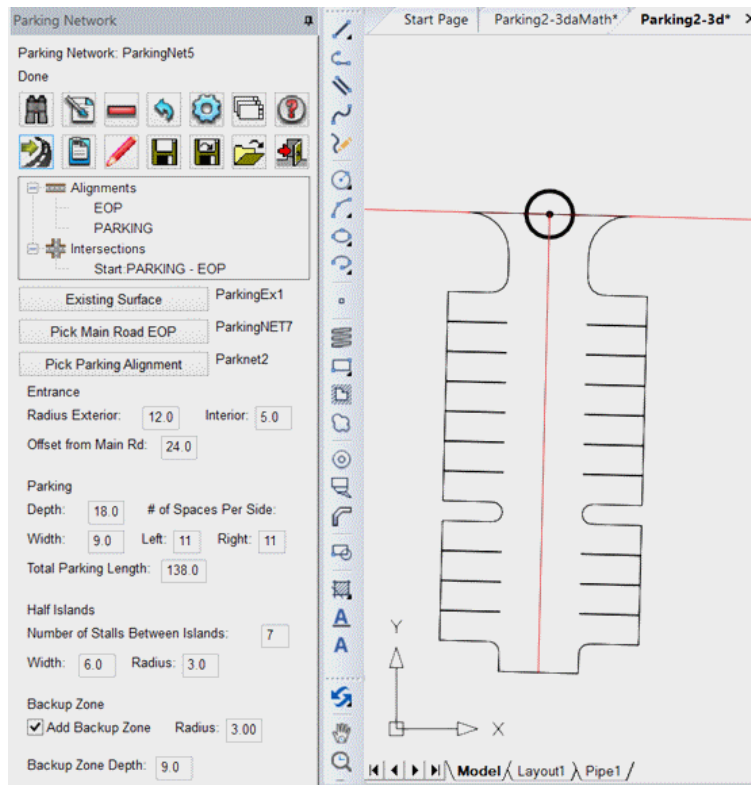
Keyboard Command: handicap_marker

Prerequisite: None

Parking Network

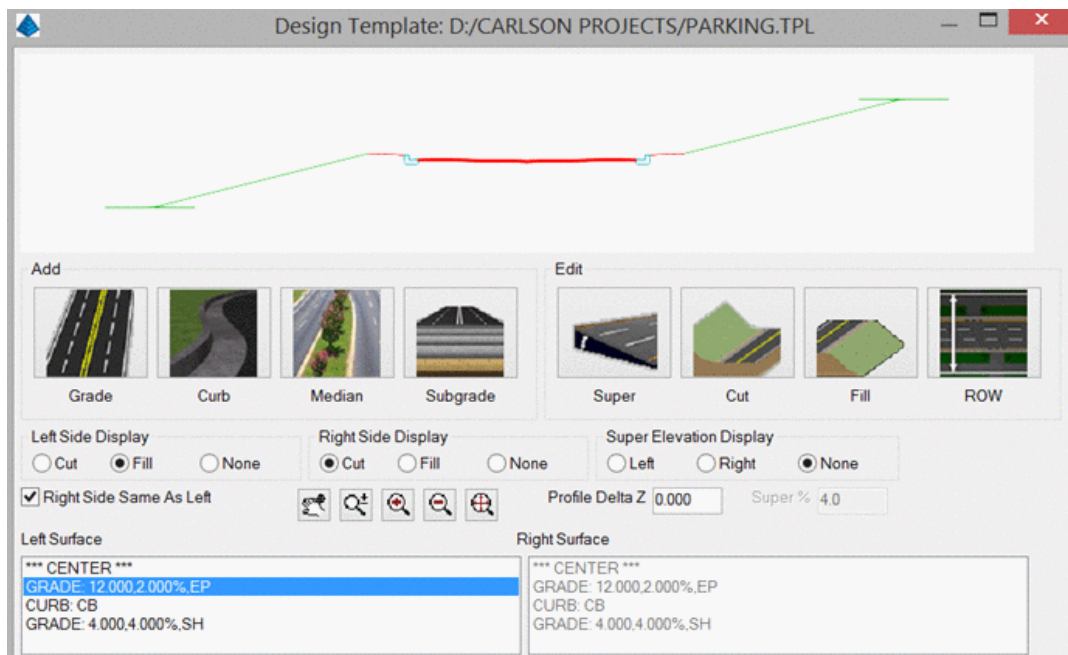
This command quickly designs parking based on a 2-point parking alignment connecting to a main road edge-of-pavement alignment. The typical application is to use a curb-based template for the parking, with dimensions for the parking stalls set within the main dialog.

A dialog pops up immediately that is similar to Roadnet, in which you work your way top to bottom starting by selecting the Existing Surface. After you complete the 3 items highlighted in red below, the Edit Alignment dialog appears, where you can review your Parking template (as well as the alignment data). The program automatically defaults to using the Parking.tpl file for the template, which sets the grade to the center of the driving lanes for the parking. You need to check the current default values set in the remainder of the dialog. These defaults recall your last use of the command. Then you launch the calculation by clicking the Process button as also highlighted in red.

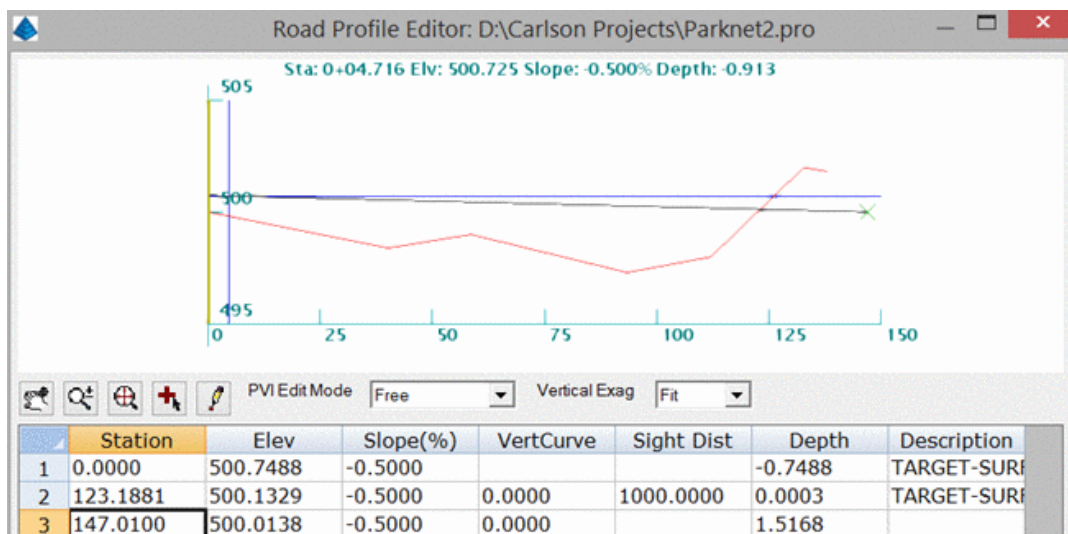


There are a number of default assumptions used in ParkingNet:

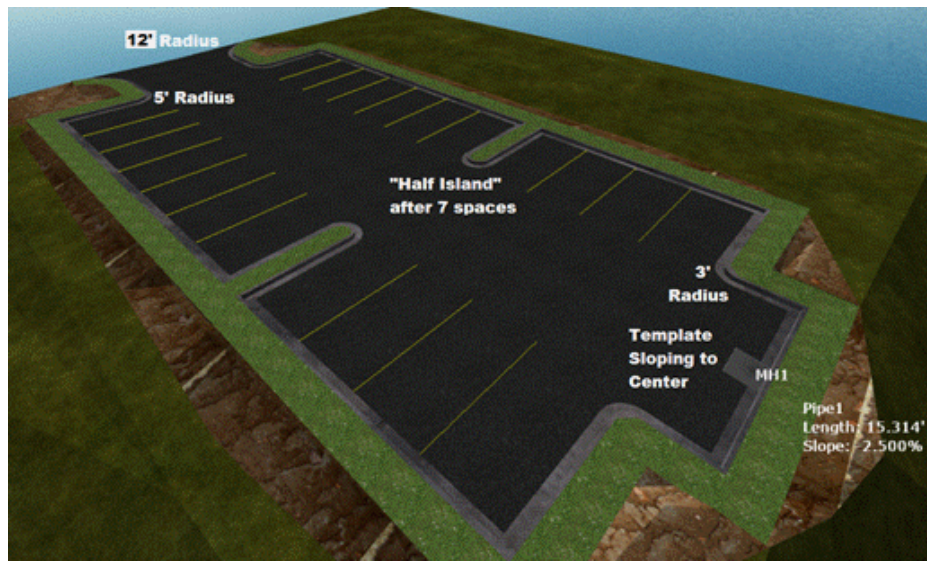
1. The Parking Alignment is a 2-point polyline.
2. Both the main road EOP alignment and the Parking alignment are given default names, "EOP" and "Parking"
3. Curb and Gutter parking perimeters are assumed.
4. The Main Road EOP (edge-of-pavement) is used primarily to transition the parking curb and gutter perimeter in an "apron" effect as shown above and below. For this, the "Zero" template is used automatically by the program, and provided to the user as Zero.tpl.
5. A Parking Template is used for the parking which assigns the width of the driving lane, and the slopes of the driving lanes. The default that is provided is designed to slope the water to the middle of the road, so that center of parking storm drain inlets can be used. If the parking profile slopes downhill, pipe lengths are minimized from end-of-parking drain to outlet as shown below. The template can be revised by the user by double-clicking "Parking" (default name for the parking alignment) and editing the template as shown below:



6. The Parking Profile defaults to a straight line from the point of connection to the Main Road EOP to the interpolated elevation of the end of the parking alignment, as derived from the selected existing TIN. You can add PVI elevations and vertical curves to the parking lot centerline profile by double-clicking "Parking" and editing the profile, as shown below:

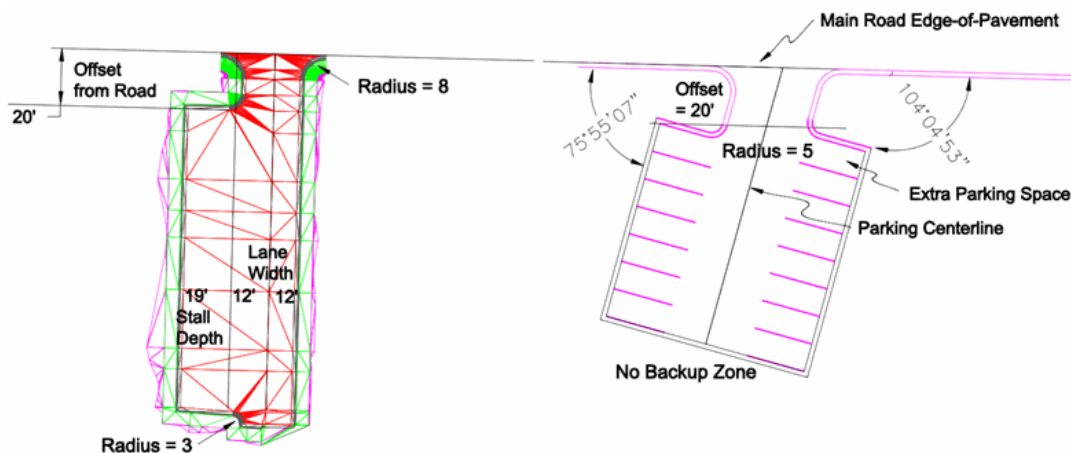


You can click the Settings button in the main dialog and save an Output TIN, merged with the existing surface. When this output TIN is viewed in the 3D Surface Viewer or in Precision 3D Topo/Hydro as below, you can see that the output TIN contains intelligent texturing:



When brought up in P3D Hydro (by running Dynamiccad with the TIN set in Surface Manager or by loading the output TIN in P3D), you can see the concrete curb and gutter and the typically asphaltic parking lot surface. Note that with a sloped template to the middle, the minimum length of pipe can be used (here, 15.3') to drain from the mid-point of the backup zone of the parking lot. As shown above, "half islands" can break up the parking for landscape purposes. Many counties and municipalities have rules for the spacing of half islands, with maximum distances between half islands of 150 feet being a typical maximum. In this case, a half island was placed after only 7 parking spaces to illustrate its use on a small parking lot example.

Show below are additional examples of output based on revised entries.



The Offset from Road prompt in the dialog sets how far from the main road EOP the first parking stall is placed. In the example above left, with an 8' radius to the main EOP and a 5' radius for the curve into the first parking space, that total of 13' leaves 7' additional for the tangent between the 2 curves, totaling a 20' offset from the road. The example parking alignment at left is perpendicular to the main EOP and calls for only parking to right of the Parking Alignment, as drawn from the main road EOP. This produces one-side only parking. (The parking stripe layer has been frozen in this case).

In the example at right, the alignment angles from the main road edge-of-pavement. An offset of 20' from the main road has been set for the first parking space. This offset is applied to the side that is closest to the main road. If the angle of parking alignment allows for an extra parking space as it does in the example above right, the user is prompted whether to place that extra parking space. If Yes, it appears as above. If no, the first parking

stall on the right would align with the first parking stall on the left, which was controlled by the minimum offset to the main road. Parking stalls and striping always line up on the left and right side in the ParkingNet command as designed. Note above, no backup zone was specified.

The "Binoculars" icon in the upper left of the dialog allows you to draw the 3D parking lines without grading the TIN, so they in effect preview things before you click the Process icon. The polylines drawn will all be erased with the next process or next use of the Binoculars or preview. You would need to Undo or erase these entities to remove them from the drawing.

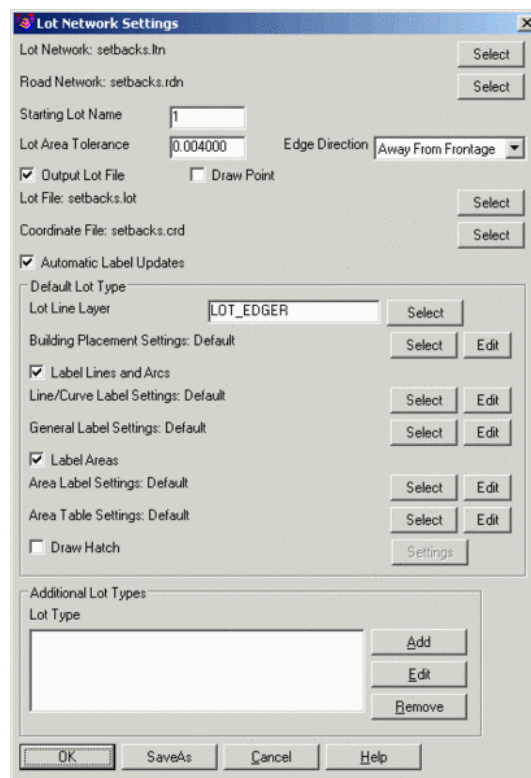
Pulldown Menu Location: Area/Layout > Parking Utilities

Keyboard Command: parkingnet

Prerequisite: TIN surface model

Lot Network Settings

This command displays a dialog for the current Lot Network Settings which specifies the lot network name, road network name, label settings, setback settings, hatch settings, building placement settings, lot type settings and lot area tolerance.



Lot Network: Click Select for the Lot Network name and choose the Lot Network file (.ltn).

Road Network: Click Select for the Road Network name and choose the Road Network file (.rdn).

Starting Lot Name: Indicate the starting Lot Name. As Lots are created, the trailing digit will be incremented by a value of 1.

Lot Area Tolerance: When creating lots to a target area, the program will finish adjustments when the area is within this tolerance of the target.

Edge Direction: Select an option to have new Lot lines drawn either *Away From Frontage* or *Toward Frontage*.

Output Lot File: This option automatically outputs a Lot file (.lot) from the Lot Network. When this option is enabled, set the Lot and Coordinate files to create. There is also an option to draw the lot file points. The settings

from Points > Point Defaults are used for drawing these points.

Automatic Label Updates: Enable this option if Lot labels should automatically update themselves if a Lot is altered or adjusted.

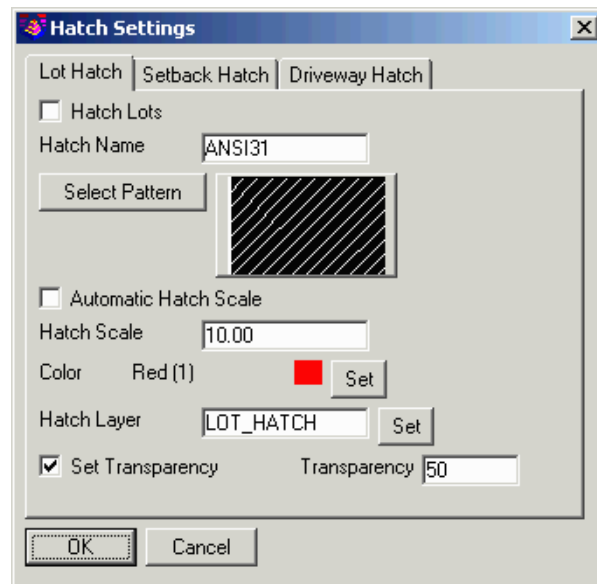
Default Lot Type: Settings in this section of the dialog box will be applied to all new Lots created using the "Default" Lot type.

Lot Line Layer: Specify a new layer or click Select to choose an existing layer for newly created Lot lines

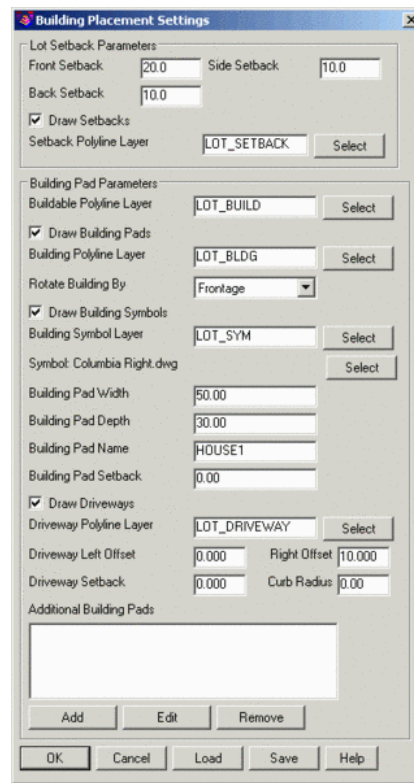
Label Lines and Arcs: Enable this option if you want this routine to label the newly created lines and arcs at the time new Lots are generated. For Line/Curve Label Settings, you can click Select to specify the Auto Annotate settings file (.aan) or click Edit to make changes to the Auto Annotate settings. For General Label Settings, you can click Select to specify the Annotation General Settings file (.adf) or click Edit to make changes to the Annotate Defaults settings.

Label Areas: Enable this option if you want this routine to label the areas at the time new Lots are generated. For Area Label Settings, you can click Select to specify the Area Defaults file (.ars) or click Edit to make changes to the Area Defaults settings. For Area Table Settings, you can click Select to specify the Area Table Settings file (.atb) or click Edit to make changes to the Area Table Defaults settings.

Draw Hatch: Enable this option if you want this routine to draw a Hatch pattern inside Lots at the time new Lots are generated. Click Settings to adjust the Hatch Settings as desired. In addition to hatching the lot area, you can also hatch the setback and driveway areas. Each area has separate hatch property settings.



Building Placement Settings: Click the Building Placement Settings button to specify the values that should be followed when building pads are placed using the Lot Network routines.



Front Setback: Sets the amount of setback for lot segments that are along the frontage.

Side Setback: Sets the amount of setback for lot segments that are partially along the frontage.

Back Setback: Sets the amount of setback for lot segments that are not along the frontage.

Draw Setbacks: Enable this option if you want this routine to draw Setback lines at the time new Lots are generated.

Buildable Polyline Layer: The Lot Network Report looks for closed polylines on this layer within the lots for reporting the buildable width and depth. This buildable area is a separate reporting option from the setback.

Draw Building Pads: Enable this option to draw building pad polylines.

Building Polyline Layer: Specify the layer on which building pad polylines should be drawn or click the Select button to choose an existing layer.

Rotate Building By: This setting controls whether to align the building pads to be perpendicular with the frontage or to be parallel with the lot edges.

Draw Building Symbols: Enable this option to draw building symbols (blocks).

Building Symbol Layer: Specify the layer on which building pad symbols should be placed or click the Select button to choose an existing layer.

Symbol: Click the Select button to specify the name of the building symbol.

Building Pad Width: Specify the width of the building pad polyline.

Building Pad Depth: Specify the depth of the building pad polyline.

Building Pad Setback: Specify the distance behind the Setback line at which to place building pad polylines. Enter "0" to have building pad polylines placed directly on the Setback.

Draw Driveways: Creates driveways as closed polylines between the building pad and lot frontage.

Driveway Polyline Layer: Sets the layer for the driveway polylines to create.

Driveway Left Offset: Facing the building pad from the front, this setting positions the left side of the driveway on the building pad.

Driveway Right Offset: Facing the building pad from the front, this setting positions the right side of the driveway on the building pad.

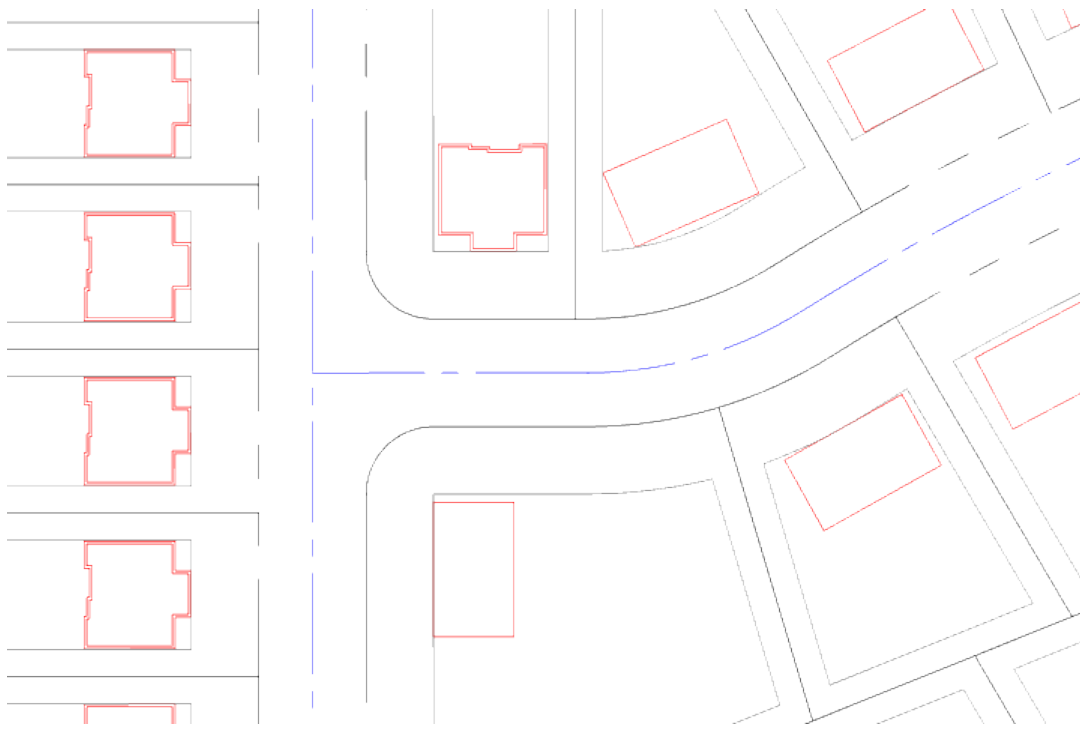
Driveway Setback: Sets the offset of the driveway from the building pad.

Driveway Curb Radius: Sets the curb return radius for where the driveway connects to the frontage.

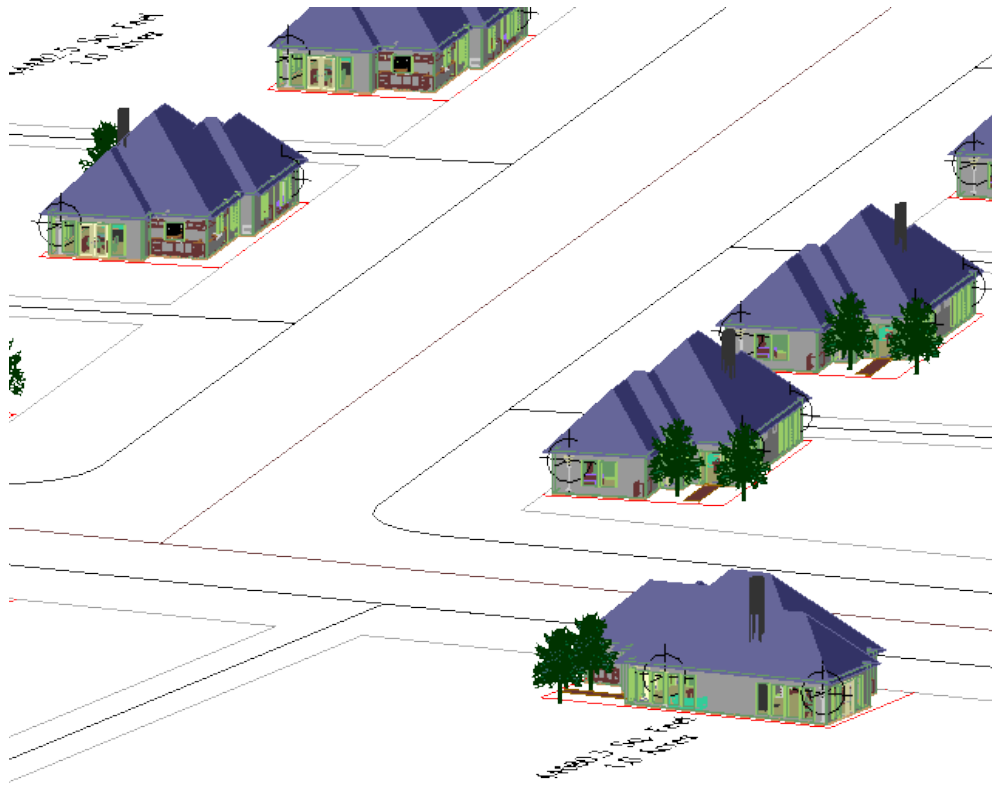
Additional Building Pads: Click the Add button to create additional building pads, the **Edit** button to modify existing building pads and the **Remove** button to delete building pads.

Note:

- When building pad creation is enabled, the initial Building Pad dimension will be attempted. If the initial building pad cannot be placed due to Lot size/placement restrictions, subsequent building pads in the Additional Building Pad list will be attempted.



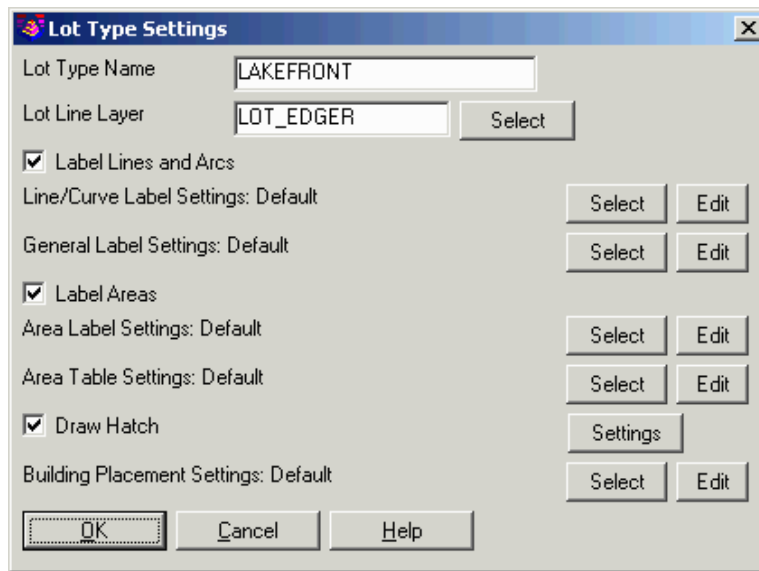
LotNet sample showing setbacks and examples of varying building sizes.



LotNet sample showing 3D building symbol.

Additional Lot Types: Settings established in this section of the dialog box allow you to create additional types of Lots in order to apply different Line/Curve, Area, Area Table, Setback, Hatch and Building Placement Settings according to their specified Lot Type. Additionally, running a Lot Network Report will break out Lot data based on Lot Type and Lot Network Inspector will display Lot Type.

Add, Edit and Remove: Use these buttons to create additional Lot types, edit or remove existing Lot types.



Pulldown Menu Location: Area/Layout

Keyboard Command: lotnet_config

Prerequisite: None

Lot Network Boundary

These are a collection of commands to assign and verify the site boundary for lot network.

Set Boundary: Sets the site boundary. It must be a closed polyline.

Highlight Boundary Perimeter: Indicates the boundary to the user by highlighting it.

Hatch Boundary Perimeter: Indicates the boundary to the user by hatching it.

Erase Hatch Boundary: Erases the hatched boundary for the user.

Clear Boundary: Deletes the boundary designation from the polyline.

Pulldown Menu Location: Area/Layout

Keyboard Command: lotnet_limit, lotnet_highlight_limit, lotnet_hatch_limit, lotnet_hatch_erase, lotnet_untag_limit

Prerequisite: None

Tag Sub-Area

This command provides the ability to establish "exclusion" areas (such as wetlands or drainage ponds) that limit where Lots or Lot Setbacks from the Lot Network routines can be created.

Prompts

Select polyline for sub-area: *Pick a closed polyline that defines the sub-area.*

Area Category: *Provide the name of a general category for the sub-area.*

Area Description: *Provide a more specialized description for the sub-area.*

Note:

- If a Sub-Area is created after a Lot Network has been processed, the existing Lot lines are kept and any associated setback lines are updated to honor the Sub-Area.
- If a Lot Network is processed after a Sub-Area has been created, the newly created Lots will honor the Sub-Area(s).

Pulldown Menu Location(s): Civil > Area/Layout > Lot Network Sub-Areas, Survey > Area/Layout > Lot Network Sub-Areas

Keyboard Command: tag_subarea

Prerequisite: A closed polyline

Untag Sub-Area

This command removes the from the selected polyline(s) the Sub-Area Category and Description information placed with the Tag Sub-Area command.

Prompts

Select sub-area polylines to remove sub-area tag.

Select objects: *Pick the polyline(s) whose Sub-Area information you wish to clear and press Enter when complete.*

Pulldown Menu Location(s): Civil > Area/Layout > Lot Network Sub-Areas, Survey > Area/Layout > Lot Network Sub-Areas

Keyboard Command: untag_subarea

Prerequisite: A closed polyline with appropriate Sub-Area data.

Identify Sub-Area

This command displays the Sub-Area Category and Description information found on polylines tagged with the Tag Sub-Area command and reports it to the Command prompt.

Prompts

Pick polylines to check or search drawing [<Pick>/Search]: Press Enter to individually select Sub-Area polylines or Type S and press Enter to search the entire drawing.

Select sub-area polyline: Pick the polyline whose Sub-Area information you wish to identify and press Enter when complete.

Pulldown Menu Location(s): Civil > Area/Layout > Lot Network Sub-Areas, Survey > Area/Layout > Lot Network Sub-Areas

Keyboard Command: id_subarea

Prerequisite: A closed polyline with appropriate Sub-Area data

Report Sub-Area

This command displays the Sub-Area Category and Description information found on polylines in the drawing that have been tagged with the Tag Sub-Area command and reports the information to the standard Report Viewer.

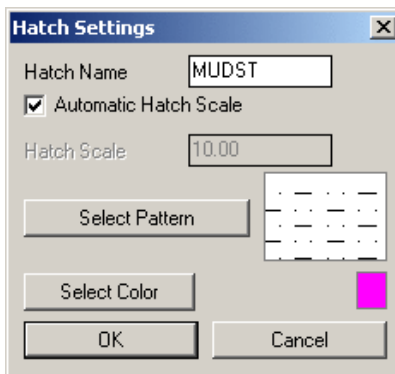
Pulldown Menu Location(s): Civil > Area/Layout > Lot Network Sub-Areas, Survey > Area/Layout > Lot Network Sub-Areas

Keyboard Command: report_subarea

Prerequisite: A closed polyline with appropriate Sub-Area data.

Hatch Sub-Areas

This command places a hatch pattern into polylines in the drawing that have been tagged with the Tag Sub-Area command.



Prompts

Hatch Name: Type in the name of a valid hatch pattern. A sample of the pattern will appear near the "Select Pattern" control.

Automatic Hatch Scale: Disable this toggle to manually control the size/density of the hatch pattern.

Hatch Scale: Specify the size of the hatch pattern. Larger Scale values create a less dense pattern.

Select Pattern: Use a visual dialog box approach to select a hatch pattern. The name of the hatch pattern selected displays in the Hatch Name control.

Select Color: Use a visual dialog box to specify the color of the hatch pattern.

Note:

- Any previous hatch patterns placed by the Hatch Sub-Areas command are first erased from the drawing.
- Hatch patterns are placed onto the LOTNET_HATCH.SUBAREA layer.

Pulldown Menu Location(s): Civil > Area/Layout > Lot Network Sub-Areas, Survey > Area/Layout > Lot Network Sub-Areas

Keyboard Command: hatch_subarea

Prerequisite: A closed polyline with appropriate Sub-Area data

Erase Sub-Areas Hatch

This command removes the hatch pattern(s) placed with the Hatch Sub-Areas command.

Pulldown Menu Location(s): Civil > Area/Layout > Lot Network Sub-Areas, Survey > Area/Layout > Lot Network Sub-Areas

Keyboard Command: erase_subarea_hatch

Prerequisite: A Sub-Area with an appropriately placed hatch pattern

Label Sub-Areas

This command displays the Sub-Area Category and Description information found on polylines tagged with the Tag Sub-Area command and uses the current text style to place the information as text into the drawing.

Prompts

Text Size <4.00>: *Press Enter accept the specified text size or Type an alternate numeric text size and press Enter.*

Label area size [Yes/<No>]?: *Choose whether or not the area of the Sub-Area(s) should be labeled in the drawing.*

Layer name <LOT.SUBAREA>: *Press Enter to accept the layer name specified or type in the desired layer name and press Enter when complete.*

Note:

- To remove Sub-Area Labels from the drawing, use the Lot Network Settings command.

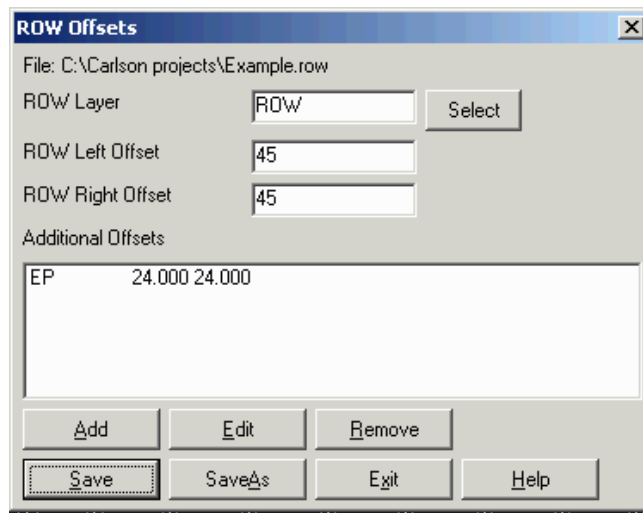
Pulldown Menu Location(s): Civil > Area/Layout > Lot Network Sub-Areas, Survey > Area/Layout > Lot Network Sub-Areas

Keyboard Command: label_subarea

Prerequisite: A closed polyline with appropriate Sub-Area data

ROW Offsets Editor

This command defines the ROW offsets for the Road Network for Lot Networks. The ROW offsets are for the frontage polylines to the left and right of the centerlines. Besides the ROW, you can also define additional offset polylines to be drawn. These additional offset polylines do not affect the lot network.



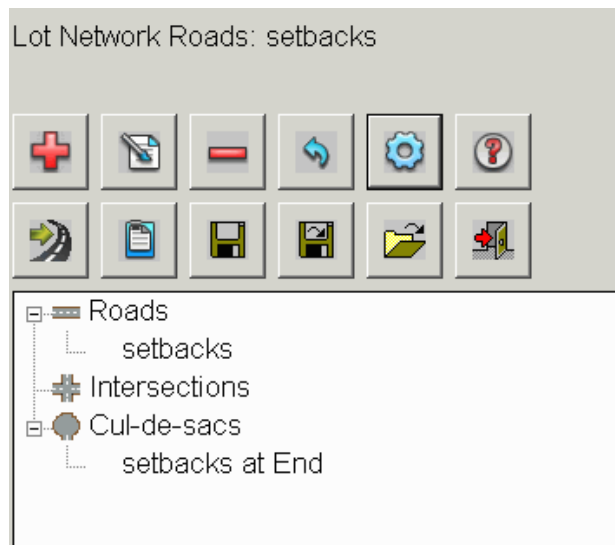
Pulldown Menu Location: Area/Layout > Lot Network Roads





Keyboard Command: lotnet_row

Prerequisite: None

Lot Network Road Network

This command develops the linework, geometry and labeling for subdivision, commercial and industrial sites by using the familiar Road Network interface and pre-defined settings. The program docks a dialog on the left of the screen identifying the geometry settings and all road files and leaves an active CAD screen and command line. You can save drawings and run virtually any standard AutoCAD command while within the docked dialog. Once you identify centerlines for the road network, the program detects intersections and end segments suitable for cul-de-sacs, and through input of design parameters for offset criteria, cul-de-sac dimensions and intersection transitions, the program will process the complete geometry layout, with output options including creating Lot files for later reference and a variety of labeling options for such items as Areas, Distances and Bearings. The road network settings are saved in a .RDN file.

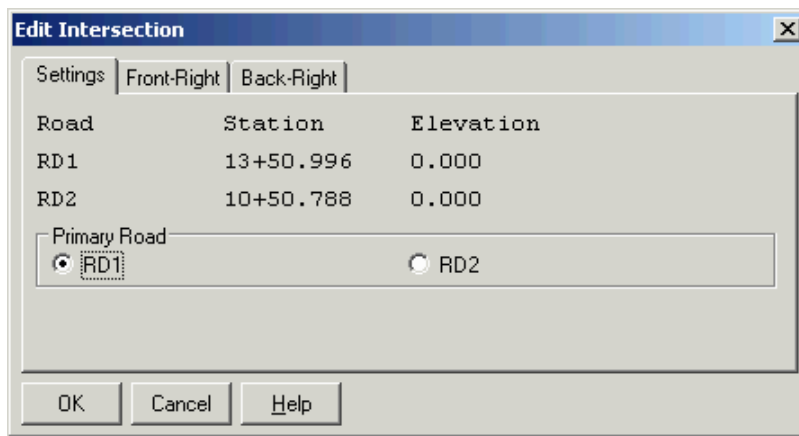


Before running the Road Network, use the following procedure to setup the lot labeling settings and site boundary.  Click the Lot Network Settings button. Note that you can use the Area/Layout Menu pulldown to access these commands as well. Select or create a lot network settings file.  Next, select the Set Boundary icon. Select a closed  polyline for the boundary around your site.  Next select the Road Network icon. When prompted,

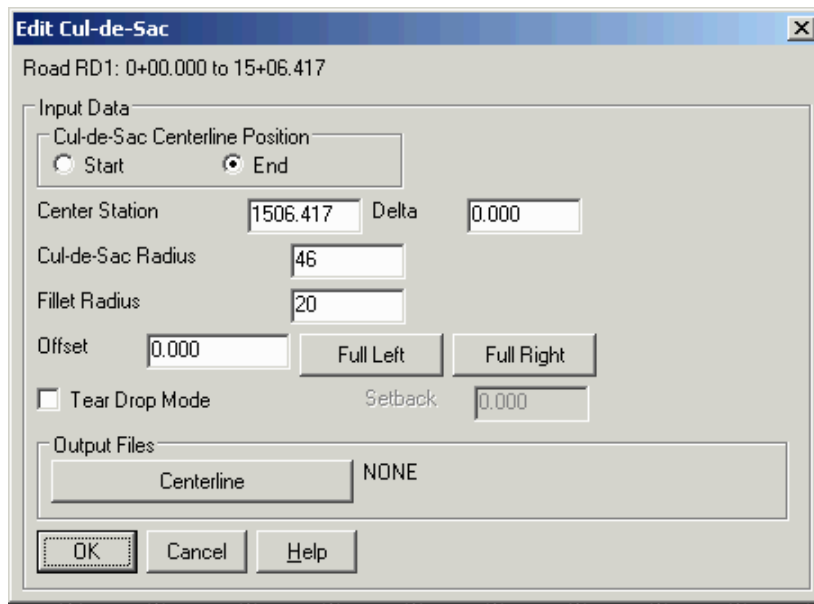
select the .RDN file from the Existing tab. This is where the centerlines involved for the subdivision will be defined and added to the Road Name area of the panel. These centerlines are standard Carlson .CL files. Click a centerline and choose Edit. If a CRD file is requested choose or create a .CRD file. The Edit Road dialog appears. The centerline can be selected here and these centerlines can be edited on the fly if needed. For ROW Offsets, we are using the Row-OFF-a.Row file. Click Edit. The ROW offsets dialog displays. Use the defaults of 45' left and right and note that additional graphics can be automatically generated by hitting Add and entering additional values, names and layers. Hit Exit.

Note also that Optional Input files can be attached to the process for roadway widenings based on the standard Carlson Road design tools of the same name. This is where a polyline indicating where the roadway template ID's should be tapered or widened is developed into a Centerline file and attached to the roadway template involved. Refer to the Road Design documentation for this information. Hit OK to close the Edit Road dialog. These settings can be set and altered for each road in the network.

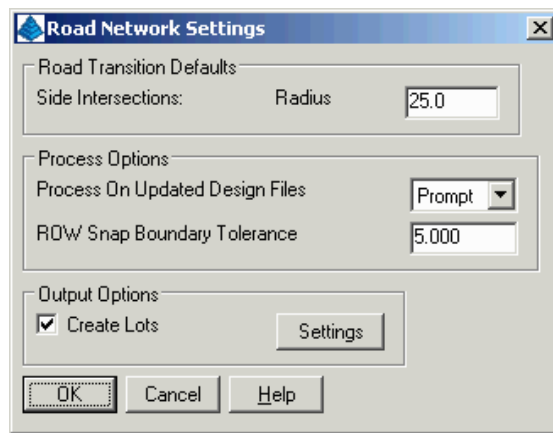
Next click on one of the intersections you may have and select Edit Intersection. In the Edit Intersection dialog, the intersection's radii can be set. Click on the Front Left or Front Right to verify this.



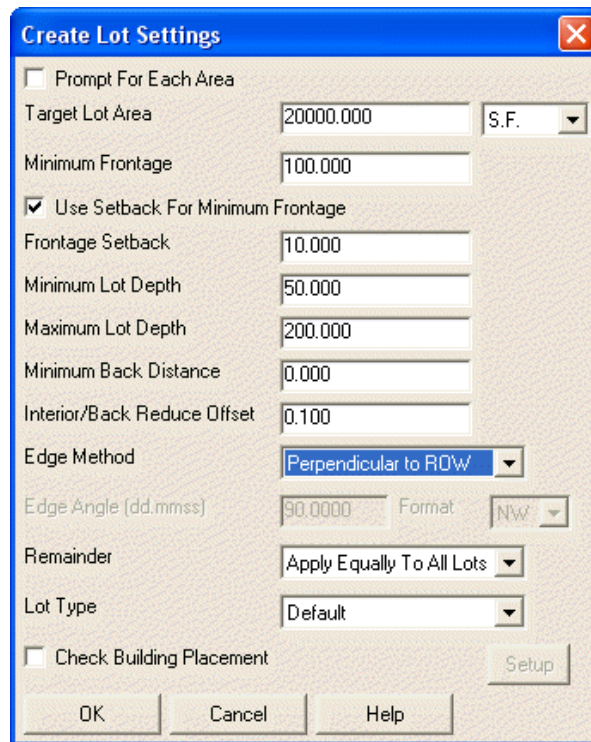
The program can also develop cul-de-sacs for the subdivision, although this example doesn't require one. To see how it works, click Add under Cul-de-sac's area of the panel. The Select Road for Cul-de-sac dialog appears. Select the road for the Cul-de-sac and the Edit Cul-de-sac dialog opens. Then as shown in the figure, choose whether the cul-de-sac occurs at the beginning or ending of the roadway, provide a cul-de-sac radius and file radius and any other criteria to develop the graphics as desired. Since we do not have a cul-de-sac in this example we will skip this step.



Next select Settings at the bottom of the Road Network panel.



The **Radius** is the default for new intersections. The radius for any existing intersection can be modified by selecting the intersection in the list and picking the Edit button. The **Process On Updated Design Files** controls whether to automatically run the Process function when an element of the road network is edited. The **ROW Snap Boundary Tolerance** is used to snap the ROW linework onto the Lot Network Boundary which is useful to clean up any undershoots between the ROW and boundary in case the centerline is not perfectly perpendicular to the boundary. The **Create Lots** setting draws linework for lots for the specified geometry parameters. Otherwise, only the ROW polylines are drawn. Use a radius of 25.0 and turn on the Create Lots toggle and click Settings. Set the values as shown in the Create Lot Settings dialog below. Then hit OK and OK to exit.



Prompt For Each Area: This option will pause to prompt for the target area as each lot is created.

Target Lot Area: The new lots will have this area +/- the Lot Area Tolerance under Lot Network Setting plus any effect from handling the Remainder.

Minimum Frontage: Controls the minimum lot perimeter length along the ROW.

Use Setback For Minimum Frontage: This option bases the Min Frontage check at the specified Frontage Setback from the ROW.

Minimum Lot Depth: This setting is the min distance from the ROW to the back of the lot for the lot side lines.

Maximum Lot Depth: This setting is the max distance from the ROW to the back of the lot for the lot side lines.

Minimum Back Distance: This setting is the min distance along the back of the lot perimeter between the two lot side lines.

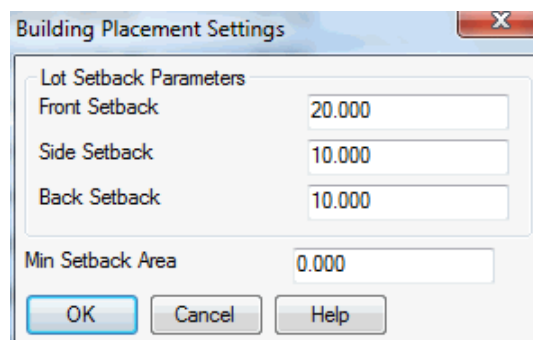
Interior/Back Reduce Offset: For interior boundaries generated by the program between lots, this option reduces the number boundary vertices. A vertex is removed if it doesn't affect the boundary by more than the specified offset amount. This method is similar to the Reduce Polyline Vertices command.

Edge Method: The lot sides can be created perpendicular to the frontage ROW, back boundary or at a specific angle.

Remainder: This option determines how to handle any remaining area that is less than the target area after fitting as many lots as possible. The **Create Separate End Lot** will make a lot with this remainder area. The **Apply Equally to All Lots** will spread the extra area to all the lots. The **Add To Last Lot** will add the remainder to the last lot created making it larger than the target area. The **Create Back Lot Edge** makes a back lot edge that meets the target area at min frontage.

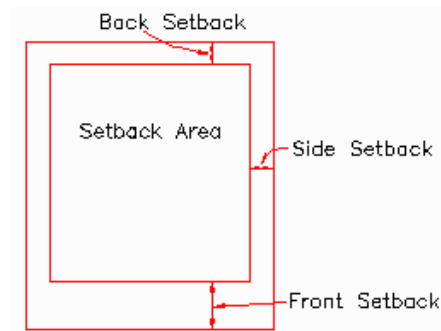
Lot Type: Sets the lot type for the new lots.

Check Building Placement: Checks that the building footprints fit within the lots for the specified setbacks.



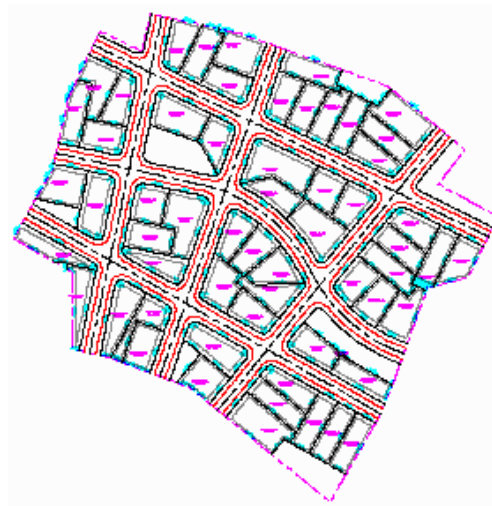
Lot Setback Parameters: These setting offset the lot perimeter inward for different sides of the lot.

Min Setback Area: This option checks that the lot area within the setbacks is at least this much.



In the Road Network panel click Save or Saveas to save these settings for your own experimentation.

Now Click Process to begin the lot layout. You will notice the ROW's and EOP's being generated, followed by the lot lines. Then areas are labeled and setbacks are created. Finally, the lotlines are labeled with distances, bearings and arc data.



Pulldown Menu Location: Area/Layout

Keyboard Command: lotnet_rdn

Prerequisite: centerlines and a site boundary polyline

Lot Network Linework

The following commands allow for the Lot Network to be manipulated after the processing. The commands allowing this are:

Adds a ROW polyline into the model. By clicking this command, the software asks for the user to select the new ROW polyline. It then reprocesses the site based on this new ROW data and relocates the EOP for this portion of the roadway.

This command adds a lot edge to the model. The software may request a Lotnet Settings file and if so create or select it. The software prompts with: Select Edge linework to add to model: Select the polyline you drew in the lot representing the new lot edge. The software reprocesses the site based on this new data and redevelops the lot layout accordingly.

This takes a property line out of the model. Select the edge in question when prompted.

This command allows for adding a new property corner to an existing lotline. Simply select the lot edge in question and then pick the point to be added using a snap or other means.

This command allows for moving a lot corner.

This command allows for eliminating a lot corner. Simply select the lot edge in question and then the corner to be removed.

Pulldown Menu Location: Area/Layout

Keyboard Command: Lotnet_Add_Row, Lotnet_Add_Edge, Lotnet_Remove_Edge, Lotnet_Point_Add, Lotnet_Point_Edit, Lotnet_Point_Remove

Prerequisite: None

ID Linework

This command identifies the type of Lot Network linework and which Lot Network it belongs to. For example, you can identify whether a line is a lot edge or ROW.

Prompts

Select Lot Network linework: *pick an entity*

Lot Edge: SETBACKS

Select Lot Network linework (Enter to end): *press Enter*

Pulldown Menu Location(s): Area/Layout > Lot Network Linework

Keyboard Command: lotnet_id

Prerequisite: Lot network linework

Untag Linework

This command removes entities from the Lot Network model by removing the Extended Entity Data (Xdata) related to Lot Network. You can use this command to remove a lot edge or ROW polyline that you no longer want to be part of the Lot Network.

Prompts

Select Lot Network linework to untag.

Select objects: *pick entities to untag*

Remove Lot Network tags for 2 entities

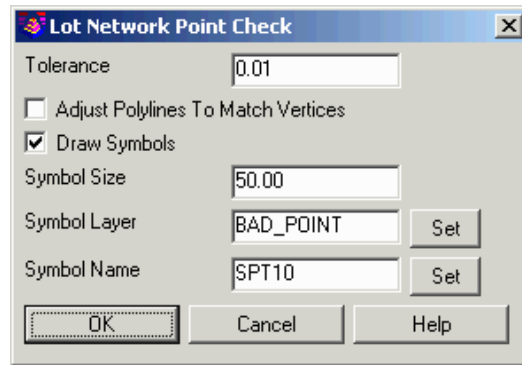
Pulldown Menu Location(s): Area/Layout > Lot Network Linework

Keyboard Command: lotnet.untag

Prerequisite: Lot network linework

Point Check

This command checks for points that are close together in the Lot Network. Any points that are within the specified Tolerance are reported. The command has an option to Draw Symbols at these points which helps with finding these points in the drawing. Also, the Adjust Polylines To Match Vertices option combines the close points to a single point by averaging them together and updates the linework entities. The Check T-Intersections option finds any points with the tolerance of a line segment.



Pulldown Menu Location(s): Area/Layout > Lot Network Linework

Keyboard Command: lot_pt.check

Prerequisite: Lot network linework

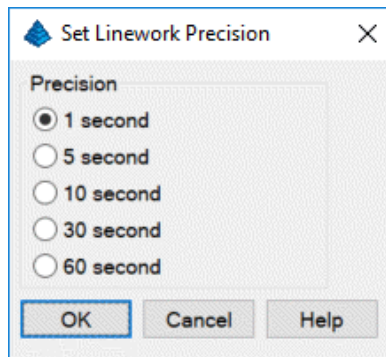
Set Lot Edge Angles To Nearest Second

This command adjusts the coordinates of Lot Network lines and polylines to set their bearings to the nearest specified precision between 1 and 60 seconds. This routine eliminates decimal seconds for the linework. Here's an example inverse showing decimal seconds on a line before running this routine.

```
Northing(Y) Easting(X) Elev(Z)
7054276.3676 11519401.0186 0.0000
7054104.6344 11519556.5360 0.0000
Bearing: S 42d09'47.5207" E Horizontal Distance: 231.6850115
```

Here's the inverse showing the bearing to the nearest second on the line after running this routine:

```
Northing(Y) Easting(X) Elev(Z)
7054276.3673 11519401.0182 0.0000
7054104.6344 11519556.5360 0.0000
Bearing: S 42d09'48.0000" E Horizontal Distance: 231.6850115
```

Prompts

Set Linework Precision dialog

Select lot edges to process.

Select objects: *Pick lot lines to adjust*

Pulldown Menu Location: Area/Layout > Lot Network Utilities

Keyboard Command: edge2sec

Prerequisite: A lot network

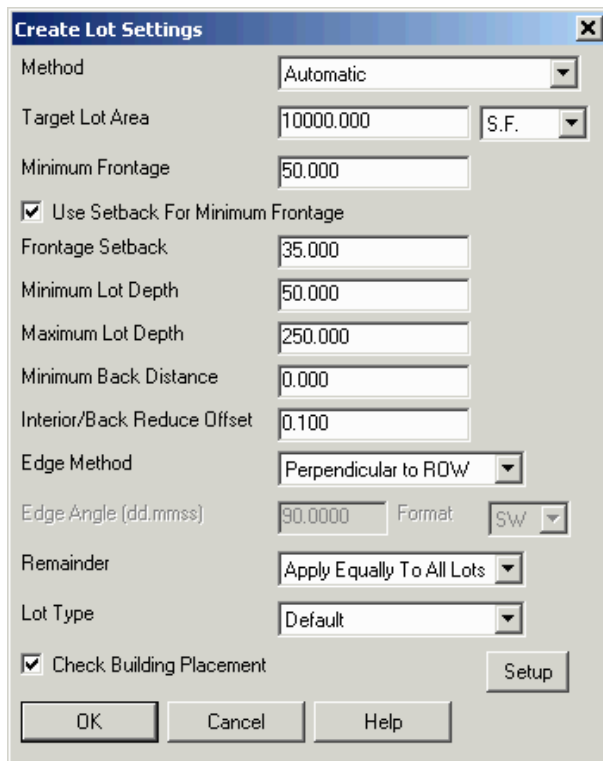
Lot Network Subdivide Area

This command subdivides an area into smaller parcels. The command displays the Create Lot Settings dialog. Once all settings are values have been entered, *click on the OK button.*

Next you are prompted to *Pick inside area to subdivide:*

Next, the right-of-way adjacent to your area to subdivide is highlighted. You are prompted to *Pick end of frontage to start lots:* . Use your left mouse button to pick a point near one end of the highlighted right-of-way. New lots will be created starting from this end of the right-of-way.

Note: All previously created lots will be re-drawn and re-labeled if the setting for "Automatic Label Updates" has been toggled ON in the LotNet Settings dialog box.



Method: The **Automatic** method will create the lots within the area according to the parameters in the dialog. The **Prompt For Each Area** method will prompt you to specify the area for each lot as it is created. The **Prompt For Each Frontage** will prompt you to specify the frontage for each lot as the method to size the lots.

Target Lot Area: This setting establishes the minimum lot area for each lot created using this command. Target Lot Area can be specified using Acres or Square Footage.

Minimum Frontage: This setting establishes the minimum width, along the front Right of Way, of the newly created lots.

Use Setback for Minimum Frontage Enabling this option measures the minimum frontage at the setback location instead of along the front Right of Way.

Frontage Setback: This setting establishes the distance off the front right-of-way for the front setback.

Minimum Lot Depth: This setting establishes the minimum depth of new lots created with this routine.

Maximum Lot Depth: This setting establishes the maximum depth of new lots created with this routine.

Minimum Back Distance: This setting establishes the minimum width of the rear lot line. Setting this to "0" allows for a pie-shaped lot.

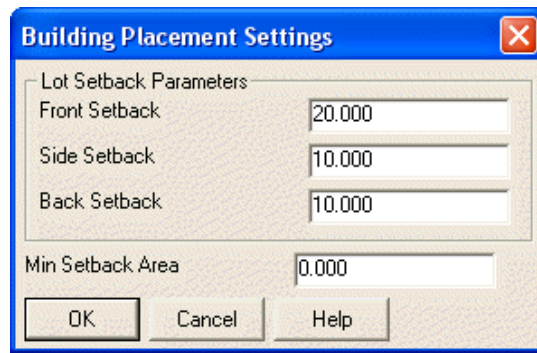
Interior/Back Reduce Offset: This setting establishes the maximum distance interior or back lot lines can be shifted or trimmed in order to meet other setback rules.

Edge Method: This setting establishes the angle between the front right-of-way and new lot lines. You are able to specify that new lot lines be drawn: Perpendicular to ROW, Perpendicular to Back Lot Line or at a Specific Angle.

Remainder: This setting allows you to distribute the area that is left over after creating new lots. The remaining area can be distributed using one of several methods: Apply Equally to all Lots, Create Separate End Lot, Add to Last Lot and Create Back Lot Edge.

Lot Type: This setting allows you to specify the Lot Type for new lots created using this routine. Lot Types are defined in the LotNet Settings dialog box.

Check Building Placement: This setting allows you to establish front, side and back setback distances for building placement along with a minimum allowable area for front setback. Building sizes are defined in the LotNet Settings dialog box.



Pulldown Menu Location: Area/Layout

Keyboard Command: lotnet_subdivide_area

Prerequisite: None

Size Lot by Frontage

This command provides the ability to resize a Lot associated with the current Lot Network .LTN file based on a user-specified amount of Lot Frontage.

Prompts

Pick inside lot to adjust: *Identify the interior portion of a Lot whose Frontage is to be adjusted.*

Select lot edge to adjust: *Choose a side Lot edge that is common to two Lots. The Current Area and Current Frontage of the selected Lot is reported.*

Frontage (ft): *Type in the new Frontage amount and press the Enter button.*

Note:

- To specify an alternate .LTN file, use the Lot Network Settings command.

Pulldown Menu Location(s): Civil > Area/Layout > Lot Network Areas, Survey > Area/Layout > Lot Network Areas

Keyboard Command: lotnet_ssfront

Prerequisite: A processed set of Lots and their graphical entities created by the Lot Network routines.

Lot Network Sliding Side Area

In this routine a lot side can be altered to reflect a new target area. It will hold its angle and slide along the front and back lot lines until it has achieved the desired area. When running the routine, Select the lot in question when asked to pick inside lot to adjust and then select the lot edge to adjust. Then the prompt asks Acres/<Enter Target Area(sf)>: Type in the desired area you are trying to obtain and the system computes it.

Pulldown Menu Location: Area/Layout

Keyboard Command: lotnet_ssarea

Prerequisite: None

Lot Network Hinged Area

In this routine a lot side can be altered to reflect a new target area. It will hold a lot corner and pivot, or rotate until it achieves the desired area. The procedure is as follows:

Pick inside lot to adjust: Select a point inside the lot to modify.

Select lot edge to adjust: Select the edge that will move.

The routine will report the current area to you and then ask for your desired area.

Current Area: 22494.5 SF 0.516 Acres

Acres/<Enter Target Area (sf)>: 10000

Pulldown Menu Location: Area/Layout

Keyboard Command: lotnet_harea

Prerequisite: None

Lot Network Labels

These are a collection of commands to draw lot network area, line and arc labels.

Deletes the labels in the model and re-labels the linework based on the LTN file settings.

Deletes the labels in the model and re-labels the linework.

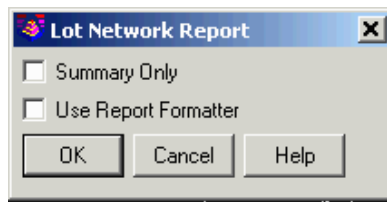
Pulldown Menu Location: Area/Layout

Keyboard Command: lotnet_update, lotnet_redraw

Prerequisite: None

Lot Network Report

This command generates a report with a summary of the areas and number of lots in the lot network model. When the **Summary Only** option is off, the report also include the area, perimeter and frontage for each lot. For a detailed report of the lot data including the lot corner coordinates, output the lot network to a .lot file and run the Report function inside Lot File Manager. The **Use Report Formatter** option allows for customized reports and output to different formats such as Excel. Otherwise a standard report is generated.



To get a report of buildings that fit within lots, first setup the different building dimensions under Building Placement Settings in the Lot Network Settings command. Then in this report, turn off Summary Only and the report will include the list of buildings for each lot.

Plan A	
Lot	Buildings
1	HOUSE1,HOUSE2,HOUSE3
2	HOUSE2,HOUSE3
3	HOUSE2,HOUSE3
4	HOUSE2,HOUSE3
5	HOUSE2,HOUSE3
6	HOUSE2,HOUSE3
7	HOUSE2,HOUSE3
8	HOUSE1,HOUSE2,HOUSE3

Lot Network Report

File: C:\Carlson Projects\Clearwater Oaks.ltn

Total Area: 20.520 acres, 893839.8 sf

Lot Area: 17.600 acres, 766648.4 sf

ROW Area: 2.920 acres, 127191.3 sf

Other Area: 0.000 acres, 0.0 sf

Number of Lots: 50

Pulldown Menu Location: Area/Layout

Keyboard Command: lotnet_report

Prerequisite: None

Lot Number Report

This command reports the range of lot numbers used in the current lot network and any gaps in the numbers. Here's a sample report:

Lot Number Report

File: C:\sample\setbacks.ltn

Used Numbers

1,26-37,39-52

Number of used numbers: 27

Unused Numbers

2-25,38

Number of unused numbers: 25

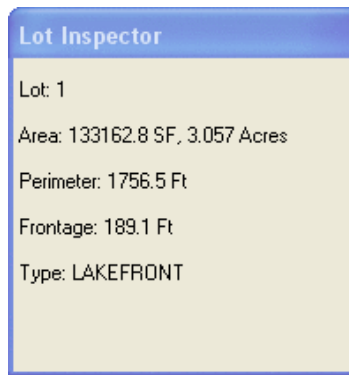
Pulldown Menu Location: Area/Layout > Lot Network Utilities

Keyboard Command: lotnet_number_report

Prerequisite: A Lot Network

Lot Network Inspector

This command shows a dynamic report of the lots as the cursor passes over them. The program has a small dialog that shows the lot number, area, perimeter, frontage and Lot type.



Pulldown Menu Location: Area/Layout

Keyboard Command: lotnet.inspector

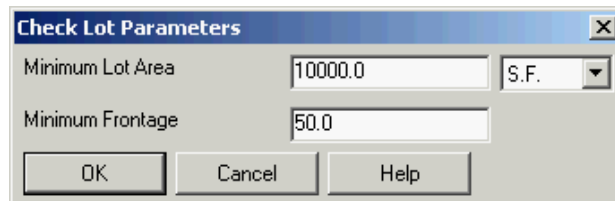
Prerequisite: a lot network

Check Lot Network Parameters

This command compares area and frontage of Lots associated with the current Lot Network file (.ltn) against user-specified area and frontage values.

Enabling the "Check ROW Offsets" option will also check for proper right-of-way distances using a specified centerline file (.cl) or (.rdn) file.

A detailed report is generated that displays the Lots that meet or do not meet the area and frontage minimums along with the coordinates of points along Lot frontage that violate the right-of-way value specified.



Minimum Lot Area: Specify the smallest acceptable area a Lot can be to "pass the test" and the appropriate unit of measure.

Minimum Frontage: Specify the smallest allowable amount of street frontage the Lot must have in order to "pass the test."

Check ROW Offsets: This setting allows you to specify the full right-of-way width for the road defined by the centerline file (.cl) you specify.

Note:

- To specify an alternate .LTN file, use the Lot Network Settings command.
- To "browse" over lots already in a drawing, use the Lot Network Inspector command.

Pulldown Menu Location: Area/Layout > Lot Network Utilities

Keyboard Command: lotnet.check

Prerequisite: A processed set of Lots created by the Lot Network routines

Find Lot Name

This command displays a temporary indicator in the drawing showing the location of a Lot associated with the current Lot Network .LTN file.

Note:

- To specify an alternate .LTN file, use the Lot Network Settings command.
- To "browse" over lots already in a drawing, use the Lot Network Inspector command.

Prompts

Lot Network File to Process dialog *Locate an existing .LTN file*

Lot name to find: *Type in the Name (usually the Lot Number) of the Lot you wish to locate and press Enter*

Pulldown Menu Location(s): Civil > Area/Layout > Lot Network Utilities, Survey > Area/Layout > Lot Network Utilities

Keyboard Command: lotnet_find

Prerequisite: A processed set of Lots created by the Lot Network routines

Lot Network Renumber Lots

This command allows you to renumber the lot number for selected lots. The program prompts for the Starting Lot Name: where the new value can be types, such as 200 for the new starting number. It then says Pick point inside lot to start renumbering: so you would pick inside the desired lot. The routine then asks for the Next direction point for renumbering: and you must pick into the next lot to continue or cross over several lots in one pick to include all of those lots in the renumbering process.

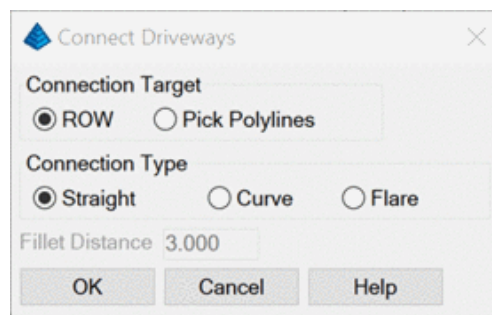
Pulldown Menu Location: Area/Layout

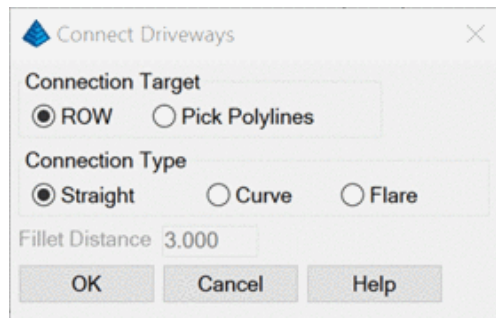
Keyboard Command: lotnet_renum

Prerequisite: a lot network

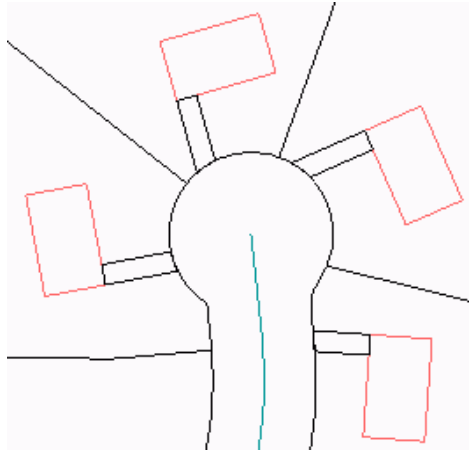
Connect Driveways

This command adjusts driveway polylines to connect them with a selected frontage polyline or the Lot Network ROW. The driveway polylines need to be closed polylines. The program will extend or trim the driveway to the frontage polyline and make the driveway follow the frontage for the width of the driveway. To run this routine, first select the driveway polylines to adjust. Then there is a settings dialog. Use the **Connection Target** to connect to either a selected polyline or the Lot Network ROW. The **Connection Type** controls how the driveway connects to the frontage polyline.





Before Connect Driveways



After Connect Driveways

Prompts

Select driveways to connect.

Select objects: *pick driveway polylines*

Connect Driveway dialog

Select polylines for driveway connections.

Select objects: *pick frontage polyline*

Pulldown Menu Location(s): Area/Layout > Lot Utilities

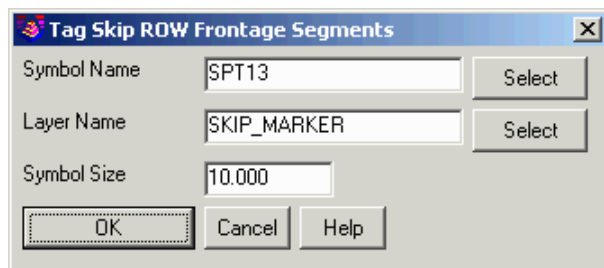
Keyboard Command: connect_driveways

Prerequisite: Driveway and frontage polylines

Tag ROW Segments to Skip Frontage

This command tags lot lines along the ROW to not use as frontage. When calculating setback, the program figures if a lot line is on the frontage based on whether the lot line is on the ROW. This command lets you tell the program not to use a lot line as frontage even when the lot line is on the ROW. This applies in cases of a corner lot where the lot side is on the side road ROW and you want this lot side to be considered as a side setback instead of front setback.

The tags are done by placing a symbol on the lot line. The program prompts for the symbol name, size and layer. Then the program prompts to pick the lot edges to tag. The symbol is drawn on the lot mid-point.



Prompts

Pick segment to skip: *pick a lot edge*

Pick segment to skip (Enter to end): *press Enter*

Pulldown Menu Location: Area/Layout > Lot Network Utilities

Keyboard Command: 3darc

Prerequisite: Lot lines

Tag No-Building Area

This command tags a polyline as a no-building area that Lot Network will avoid when placing building pads.

Pulldown Menu Location(s): Area/Layout > Lot Network Utilities

Keyboard Command: tag_no_build

Prerequisite: closed polyline

Untag No-Building Area

This command removes the no-building tag from selected polylines. You can use this command to update a polyline area that you no longer want to be counted as non-buildable.

Pulldown Menu Location(s): Area/Layout > Lot Network Utilities

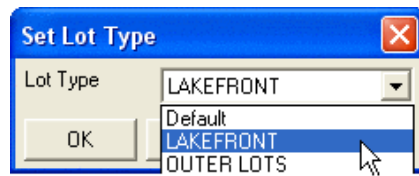
Keyboard Command: untag_no_build

Prerequisite: tagged no-building polyline

Lot Network - Assign Lot Type

This command allows you to assign a new Lot Type to Lots in a Lot Network by dragging a line across the Lots to be re-assigned.

Note: Lot Types must have already been defined through the LotNet Settings dialog box.



Prompts

Set Lot Type dialog: Select the "Default" or other pre-defined Lot Type.

Pick a point inside lot to start re-assigning: Use the left-mouse button to drag a multi-segmented line across all the Lots to be re-assigned to the selected Lot Type. *Press Enter to finish.*

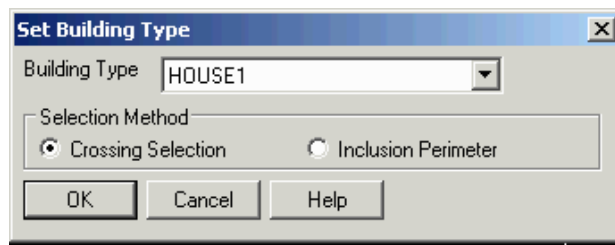
Pulldown Menu Location: Area/Layout → Lot Network Utilities

Keyboard Command: lotnet_type

Prerequisite: A lot network and pre-defined Lot Types

Assign Building Type

This command lets you change the building type within selected lots. The available building types are defined in the Lot Network Settings command under Edit for Building Placement Settings.



To run this routine, select the Building Type to use from the pull-down list in the dialog. Then select the lots to use this building type. The Crossing Selection method selects lots by picking within the lots to update. To update a single lot, pick a single point in that lot. To update a series of lots, pick a point in the first lot and another point in the last lot. Then every lot that crosses this line is updated. For the Inclusion Perimeter method, draw a closed polyline around the lots to update before running this command and pick this polyline as the inclusion and all lots inside are updated.

Pulldown Menu Location: Area/Layout > Lot Network Utilities

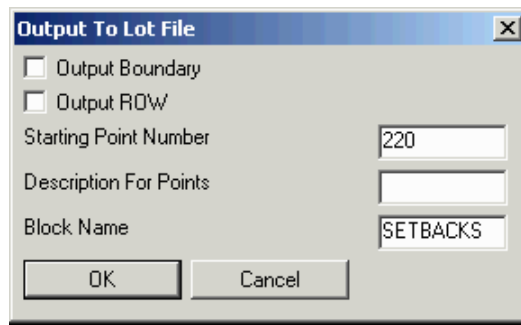
Keyboard Command: lotnet_bldg

Prerequisite: Lot network lots

Lot Network Output To Lot File

This command will create a .LOT file containing the points to define the lots. The points are stored into the specified coordinate file. The Output Boundary option creates a lot for the overall site perimeter. The Output ROW option creates lots for the ROW areas.

The .LOT file is used by the collection of Lot File commands including Lot File Manager.



Pulldown Menu Location: Area/Layout > Lot Network Utilities

Keyboard Command: lotnet_lotfile

Prerequisite: a lot network

Set Lot File

This command sets the lot (.LOT) file name that other lot routines will automatically reference. The lot (.LOT) file stores a list of lots with each lot being a list of point numbers which reference coordinates stored in a coordinate (.CRD) file.

Pulldown Menu Location: Area/Layout

Keyboard Command: setlot

Prerequisite: None

Design Lot

This command creates lot definitions that are stored in a lot (.LOT) file. The lots are defined by entering a sequence of point numbers. The point numbers reference coordinates from the current coordinate (.CRD) file. Each lot has a lot name and block name. The lots are not required to be closed perimeters and can also be used to represent other linework such as centerlines. Curves are entered by first specifying the PC point number, then type R for radius and enter the radius point number followed by the PT point number.

Prompts

Lot Name <1>: 105

Block Name <1>: *press Enter*

Lot Starting Station <0.0>: *press Enter*

If the figure that you are entering is a centerline, then you could use this as the starting station of the centerline.

Starting point number: 17

Point number (R-RadiusPt,U-Undo,Enter to end): 18

Point number (R-RadiusPt,U-Undo,Enter to end): 19

Point number (R-RadiusPt,U-Undo,Enter to end): R

Radius point number: 20

Use large included angle for curve (Yes/<No>)? *press Enter*

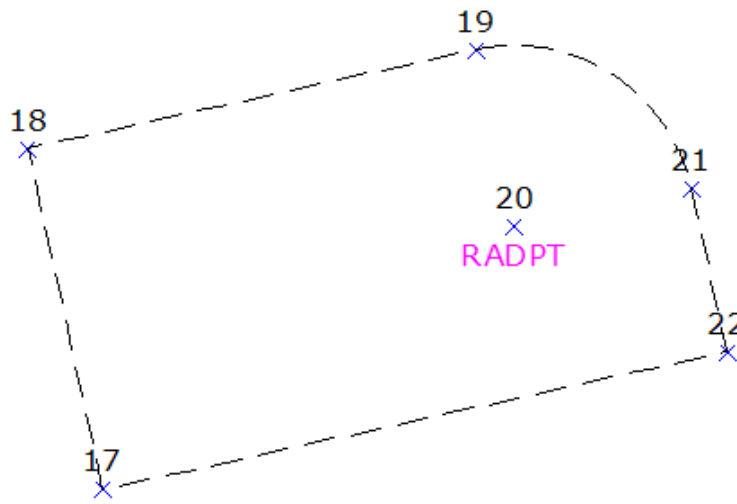
End of curve point number (R-RadiusPt,U-Undo,Enter to end): 21

Point number (R-RadiusPt,U-Undo,Enter to end): 22

Point number (R-RadiusPt,U-Undo,Enter to end): 17

Point number (R-RadiusPt,U-Undo,Enter to end): *press Enter*

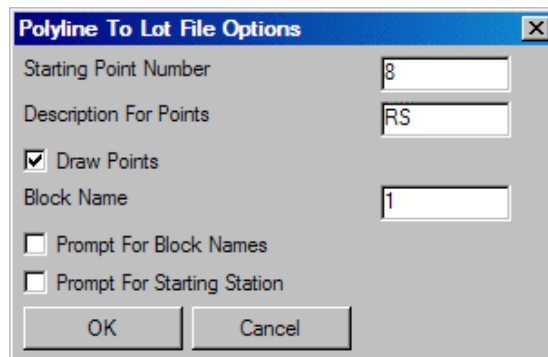
Enter another lot (<Yes>/No)? N



Pulldown Menu Location: Area/Layout > Create Lots
Keyboard Command: mklot
Prerequisite: Points in a coordinate (.CRD) file

Polyline to Lot File

This command will create lot (.LOT) files from selected polylines. The lots are defined by the series of point numbers. This command will create point numbers in the current coordinate (.CRD) file for each point in the polylines. Before creating a point number, the program will check to see if the point coordinates are already in the coordinate (.CRD) file and will use the existing point number if found. Each lot has a lot name and block name. Lots are not required to be closed perimeters and can also be used to represent other linework such as centerlines.



Prompts

Polyline To Lot File Options Dialog *enter in values*

After entering in the Starting Point Number, points will be automatically numbered starting from this value.

Select lot polyline: *pick a polyline*

Select lot polyline:

Lot Name <LOT 19>:

Created 3 lot points.

Select lot polyline (Enter to end):

Lot Name <LOT 20>:

Created 3 lot points.

Select lot polyline (Enter to end):

Lot Name <LOT 21>:

Created 3 lot points.

Select lot polyline (Enter to end):

Select lot polyline: *pick a polyline*

Lot Starting Station <0.0>: *press Enter*

Lot Name <106>: *press Enter* This defaults to the next available name.

Block Name <1>: *press Enter*

Lot Starting Station <0.0>: *press Enter*

Created 7 lot points.

Select lot polyline (Enter to end): *press Enter*

Pulldown Menu Location: Area/Layout > Create Lots

Keyboard Command: pl2lot

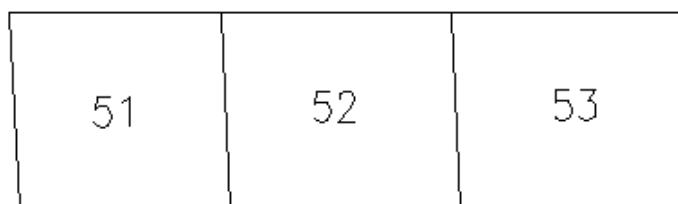
Prerequisite: A polyline

Lot File by Pick Interior

This command is used to create a lot by picking a point, and having the program figure the enclosing linework. The linework do not need to be closed themselves but selected together they should define closed areas. All the lots will have the same block name as entered and all lots will be assigned a starting station of 0.0.

The lots are defined by the series of point numbers. This command will create point numbers in the current coordinate (.CRD) file for each point in the bounding polylines. Before creating a point number, the program will check to see if the point coordinates are already in the coordinate (.CRD) file and will use the existing point number if found.

This command works well in conjunction with *Lot File Manager*. Once a lot (.LOT) file containing 1 or more lots is created, all lots can be redrawn automatically, with annotation, using *Lot File Manager*. Furthermore, since the lots are drawn from point numbers, if the point numbers for the lot corners are moved, the lots can be redrawn to the new point positions using *Lot File Manager*. If a point number is at the corner of four lots, moving that one point number will update all four lots.



Linework areas to convert into 3 lots

Prompts

Lot File By Pick Interior dialog

Pick point inside lot perimeter: *pick an interior point*

Lot Name <1>: *enter name for lot*

Pick point inside lot perimeter (Enter to end): *press Enter*

Pulldown Menu Location: Area/Layout > Create Lots

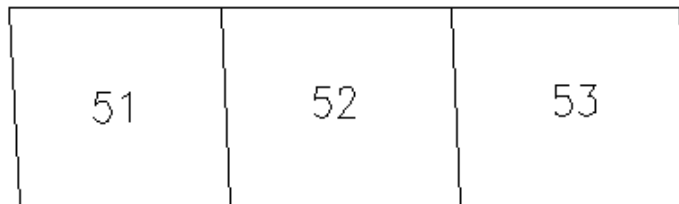
Keyboard Command: pt2lot

Lot File by Interior Text

This command creates lot definitions from the selected polylines and text. For each text entity, the program finds the bounding polyline around the text. The text is used as the lot name. The polylines do not need to be closed themselves but selected together they should define closed areas. Multiple lots can be created at once with this command. All the lots will have the same block name as entered and all lots will be assigned a starting station of 0.0.

The lots are defined by the series of point numbers. This command will create point numbers in the current coordinate (.CRD) file for each point in the bounding polylines. Before creating a point number, the program will check to see if the point coordinates are already in the coordinate (.CRD) file and will use the existing point number if found.

This command works well in conjunction with *Lot File Manager*. Once a lot (.LOT) file containing 1 or more lots is created, all lots can be redrawn automatically, with annotation, using *Lot File Manager*. Furthermore, since the lots are drawn from point numbers, if the point numbers for the lot corners are moved, the lots can be redrawn to the new point positions using *Lot File Manager*. If a point number is at the corner of four lots, moving that one point number will update all four lots.



Polylines and text to convert into 3 lots

Prompts

Lot File By Interior Text dialog

Select lot lines, polylines and text.

Select objects: *select the polylines and text*

Select objects: *press Enter*

Created 3 lots.

Pulldown Menu Location: Area/Layout > Create Lots

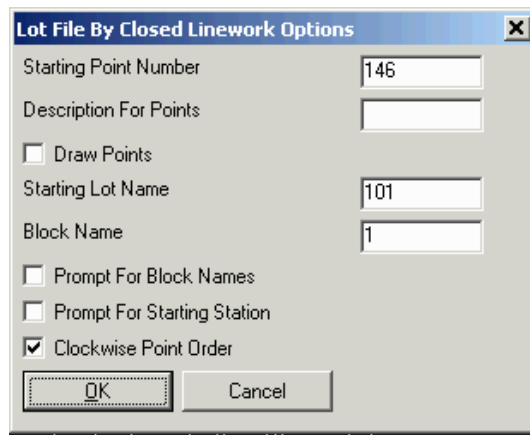
Keyboard Command: txt2lot

Prerequisite: Polylines and text

Lot File by Closed Linework

This command creates lot definitions from the selected polylines, lines and arcs. This command is similar to Lot File By Interior Text. The difference is that this routine does not process text for the lot names. Instead this command finds all the closed areas from the selected linework and then automatically names the lots.

For each lot, the program stores a series of points to the lot file. In the options dialog, there are settings for the point number, point description and whether to order the points clockwise around each lot. The Draw Points option will create point entities besides storing the points to the current coordinate file. The Starting Lot Name is used for assigning the lot names which then get incremented by one for each other new lot.



Prompts

Lot File By Closed Linework dialog

Select lot lines and polylines.

Select objects: *select linework*

Created 3 lots.

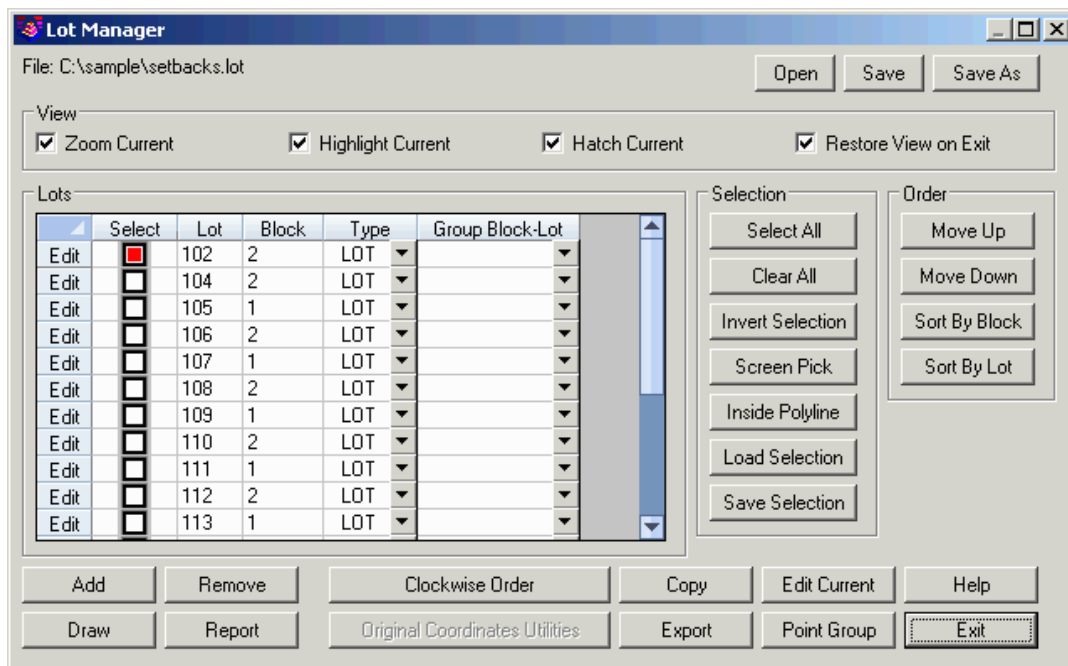
Pulldown Menu Location: Area/Layout > Create Lots

Keyboard Command: lwork2lot

Prerequisite: Polylines, lines or arcs

Lot Manager

This command combines input, edit, draw and report lot capabilities into one command.



Main Dialog

In the main dialog, there is a spreadsheet list for the lot names along with the block name, lot type and group assignment for each lot. You can edit these values directly in the spreadsheet. There are also function buttons as follows:

Open: selects another Lot File to process.

Save: saves the lot data to the current lot file.

SaveAs: prompts for another file name to save the lot data to.

View: The View options control drawing effects when you highlight lots in the spreadsheet list.

Zoom Current: zooms the display view to include the selected lot.

Highlight Current: highlights the perimeter of the selected lot as a dashed line.

Hatch Current: fills in the selected lot with a hatch.

Restore View on Exit: on leaving Lot Manager, this option sets the display to the original position before running Lot Manager.

Lot Selection: Many of the functions such as Draw process only the lots that are in selected mode. You can toggle which lots are selected with the buttons in the Selection spreadsheet column. You can also use the buttons in the Selection section to select the lots to process.

Select All: marks all the lots as selected.

Clear All: unselects all the lots.

Invert Selection: flips currently selected lots to unselected status and currently unselected to selected status.

Screen Pick: selects lots by picking inside the lots

Inside Polyline: prompts to select a closed polyline and then selects all the lots within this polyline

Load Selection: sets the current selection status from a .LSS file.

Save Selection: saves the current selection status to a .LSS file.

Add: creates a new lot. The new lot name is automatically generated by incrementing from the highest lot name.

Remove: deletes the currently selected lots.

Copy: creates new lots as copies of the currently selected lots.

Edit Current: brings up a dialog editor for the highlighted lot (see below).

Move Up/Down: changes the order of the highlighted lot in the list.

Sort By Block: sorts the lots by block name order first and then by lot name within each block.

Sort By Lot: sorts the lots by lot name only without using the block name.

Clockwise Order: sets the order for the lot points for the selected lots as either clockwise or counter-clockwise.

Point Group: creates a point group for the lot points of the selected lots.

Draw: draws the selected lot perimeters and annotation (see below).

Report: reports the selected lots (see below).

Original Coordinates Utilities: has methods for tracking lot coordinate transformations of the current coordinates relative to the original coordinates.

Export: output selected lots to a new lot file as a way to make a subset lot file.

Edit

This dialog allows you to edit the lot name, block name, group, coordinate file, starting station, ending station and the point numbers that define the lot. A curve is specified by the PC, radius point and PT point numbers. The Large Arc option indicates a curve with an included angle greater than 180 degrees. The Select button allows you to specify a new name or location for the coordinate file associated with the lot.

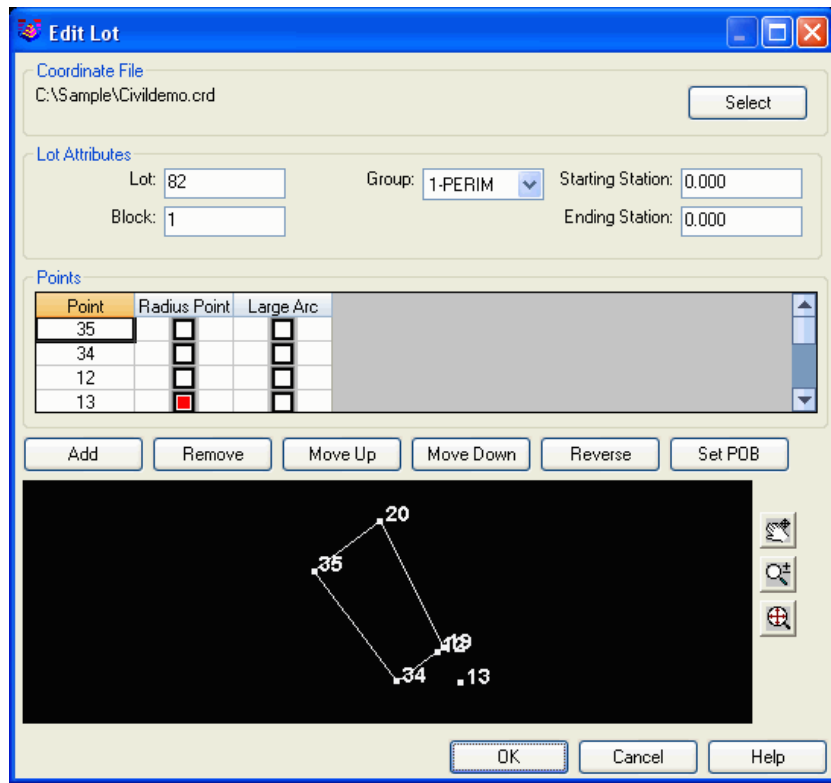
Add: adds a new point to the lot.

Remove: removes the highlighted point from the lot.

Move Up/Down: changes the order of the highlighted point in the list.

Reverse: reverses the order of the points.

Set POB: sets the point of beginning, starting point, to the currently highlighted point.



Draw

The Draw routine allows you to draw polylines for the lot perimeters as well as annotate the lot linework and areas.

Draw Lot Polylines: The Closed Polylines method creates a closed polyline for each lot. The Lines and Arcs method draws the sides of the lots as a series of lines and arcs. The advantage of the Closed Polylines method is that each lot is completely defined by the closed polyline. The advantage of the Lines and Arcs method is that there are no overlapping entities for common borders between lots. With the Lines and Arcs method, the program draws the common border entities only once. The layer for the polylines is set by the Lot Type which is defined in Define Lot Attributes. If the Lot Type is not defined, then the polylines are drawn in the current layer.

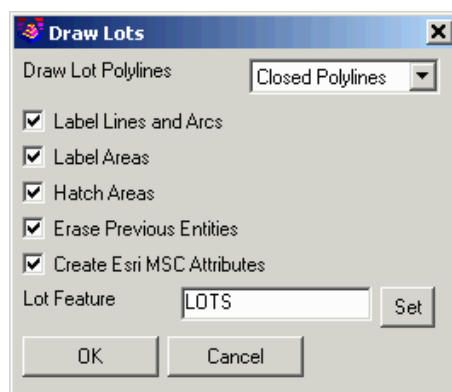
Label Lines and Arcs: Labels the bearing, distance, and curve data using the *Auto-Annotate* command. See *Auto-Annotate* for more details.

Label Areas: Labels the area, and optionally the name of the selected lots using the Area Settings dialog. See *Area Defaults* for more details.

Hatch Areas: Hatches the lot areas.

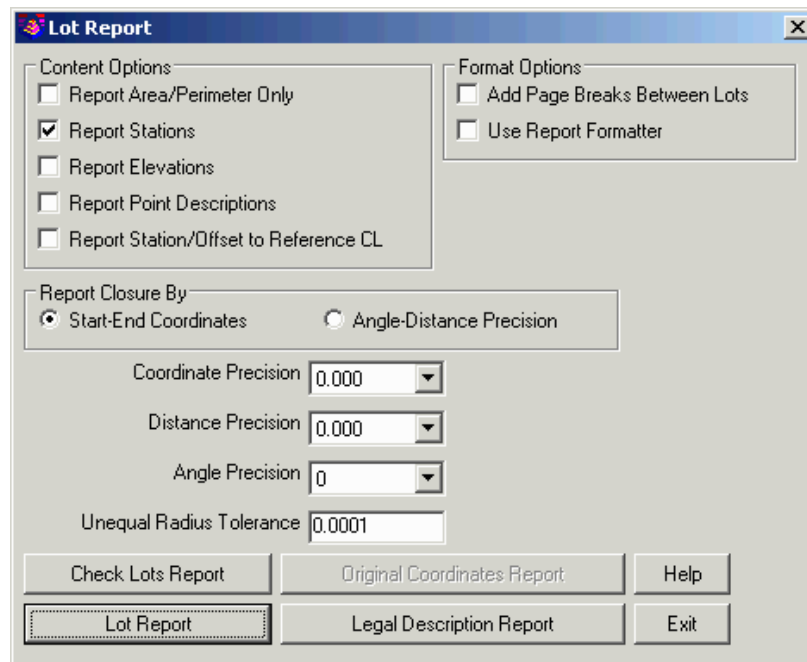
Create Esri MSC Attributes: Defines an Esri MSC format lot feature in the drawing with the lot attributes.

Erase Previous Entities: Erases lot polylines and labels from earlier runs of Draw to avoid duplicates.



Report

The Report routine has several types of reports.



Report Areas Only: When checked only the lot name, block name, and area are included in the report.

Report Stations: Controls whether to report the distance along the lot perimeter.

Report Elevations: Controls whether to report the elevation from the coordinate file for each lot point.

Report Point Descriptions: Controls whether to report the description from the coordinate file for each lot point.

Report Station/Offset To Reference CL: This option prompts for a CL file that is used to calculate stations and offsets for each of the lot points to include in the report.

Add Page Break between Lots: Formats the report so that each lot definition begins on a new page when printed.

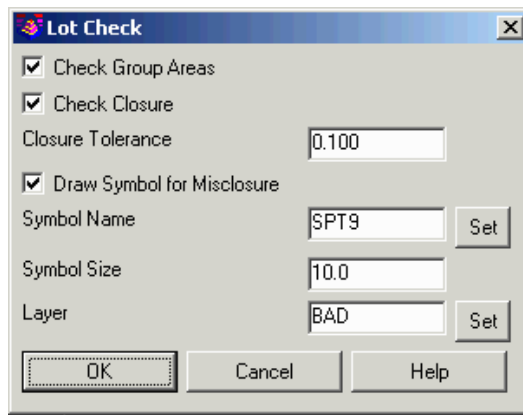
Use Report Formatter: When checked, the report is output to the *Report Formatter* where it can be customized as well as exported to Microsoft® Excel or Microsoft® Access. See *Report Formatter* in for more details.

Report Closure By: For the Start/End Coordinates method, the closure error distance is the distance between the start and end points which is 0 (perfect closure) if the lot starts and ends at the same point number. For the Angle/Distance Precision method, the actual bearings and distances in the report are used, and the rounding used in the report will result in small closure errors.

Report Precision: Specify the decimal precision for reporting coordinates, distance and angles on the report. The precision for the areas is defined in the Area Defaults command.

Unequal Radius Tolerance: When reporting the curve data for a lot, the two radial lengths are compared. If the difference in their length is more than this value, it is noted on the report.

Check Lot Report: The Check Group Areas method will check that the area for all the lots assigned to a Group Block-Lot add up to the area of the area of the enclosing group lot. The Check Closure method reports any lots where the beginning and ending coordinates are different than the specified tolerance. This closure check has an option to draw a symbol at the end point for lots outside the tolerance.



Lot Report: Creates a report using the report settings.

Legal Description Report: Writes a legal description using the same routine as the Legal Description Writer command. See the Legal Description Writer section of the manual for more details.

Lot Report

Lot File: C:\sample\CivilDemo.lot
 CRD File: C:\sample\CivilDemo.crd

```

LOT 55          OF BLOCK 1, TYPE: LOT
PNT#  Bearing          Distance  Northing  Easting    Station
36          3374.827    4631.668  0.000
  Radius: 642.845  Length: 85.660  Chord: 85.597  Delta: 07°38'05''
  Chord BRG: S 60°07'05'' W  Rad-In: N 33°41'58'' W  Rad-Out: N 26°03'53'' W
  Radius Pt: 8 3909.649,4274.994  Tangent: 42.894  Dir: Right
  Tangent-In: S 56°18'02'' W  Tangent-Out: S 63°56'07'' W
  Non Tangential-Out
41          3332.181    4557.451  85.660
  N 26°03'53'' W  175.000
42          3489.384    4480.558  260.660
  Radius: 467.845  Length: 62.341  Chord: 62.295  Delta: 07°38'05''
  Chord BRG: N 60°07'05'' E  Rad-In: N 26°03'53'' W  Rad-Out: N 33°41'58'' W
  Radius Pt: 8 3909.649,4274.994  Tangent: 31.217  Dir: Left
  Tangent-In: N 63°56'07'' E  Tangent-Out: N 56°18'02'' E
  Non Tangential-In  Non Tangential-Out
37          3520.420    4534.571  323.001
  S 33°41'58'' E  175.000
36          3374.827    4631.668  498.001

Closure Error Distance> 0.00032 Error Bearing> N 70°14'59'' E
Closure Precision> 1 in 1550913.7 Total Distance> 498.001
LOT AREA: 12950.1 SQ FT OR 0.3 ACRES
  
```

Pulldown Menu Location: Area/Layout

Keyboard Command: editlot

Prerequisite: None

Lot Inspector

This command activates a small pop-up window that when you place your pointer into a lot file area, the details of that lot file will be displayed in the Lot Inspector window.



Prompts

Move pointer inside lots (Pick to edit, Enter to End) hover crosshairs above lot(s)

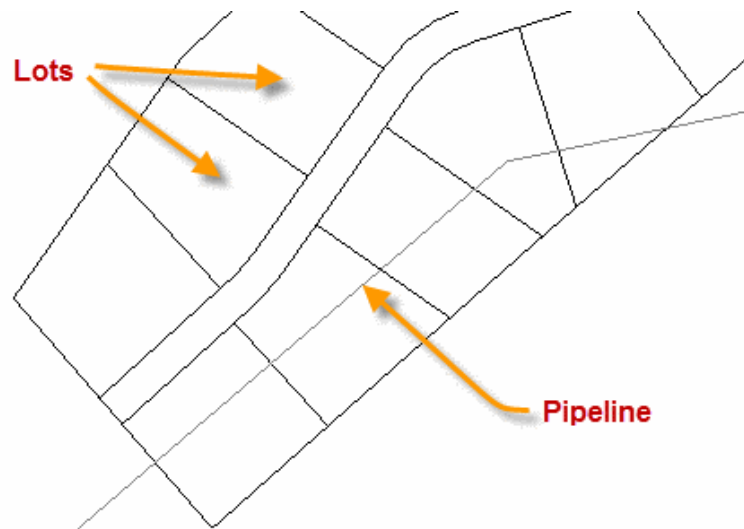
Pulldown Menu Location: Area/Layout

Keyboard Command: lotinspector

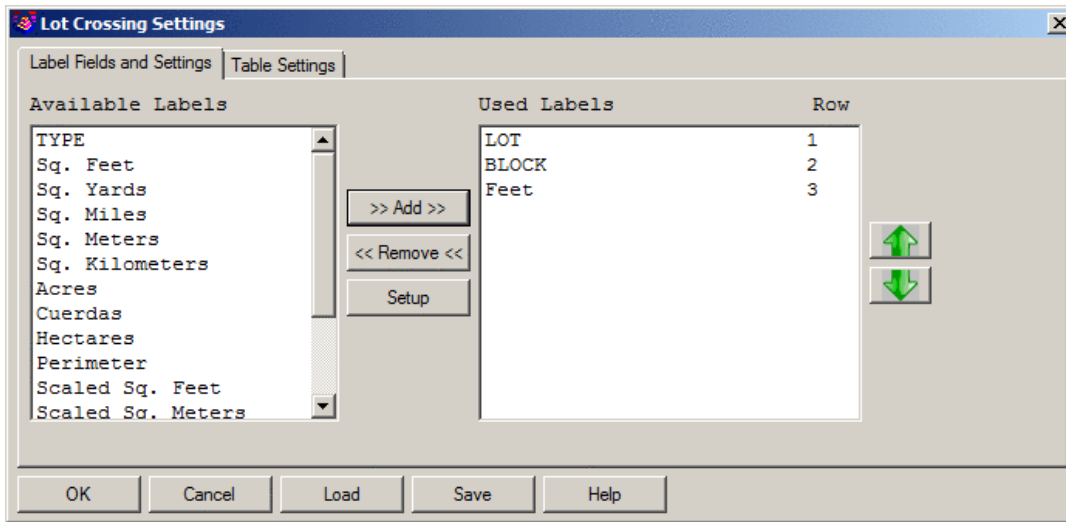
Prerequisite: None

Right-of-Way Crossing Table

This command will create a table using user selected information and user defined table features. A polyline is selected that crosses one or more lots. Lots must be defined in a Lot file prior to running the command. In the following example the polyline is labeled as a Pipeline.



When the command is started the user is presented with the Lot Crossing Settings dialog box. There two tabs; Label Fields and Settings and Table Settings.



Available Labels: This is the list of information that may be included in the table.

Used Labels: These are the items that have been selected to be in the table. They are placed in the table in the order listed. The green up and down arrows will move used labels up or down in the list.

Add: Clicking the Add button will add the highlighted labels in the available list to the used list.

Remove: Clicking the Remove button will remove labels highlighted in the used list and display in the available list.

Setup: Setup opens the Field Settings dialog for the Used Label that is highlighted.

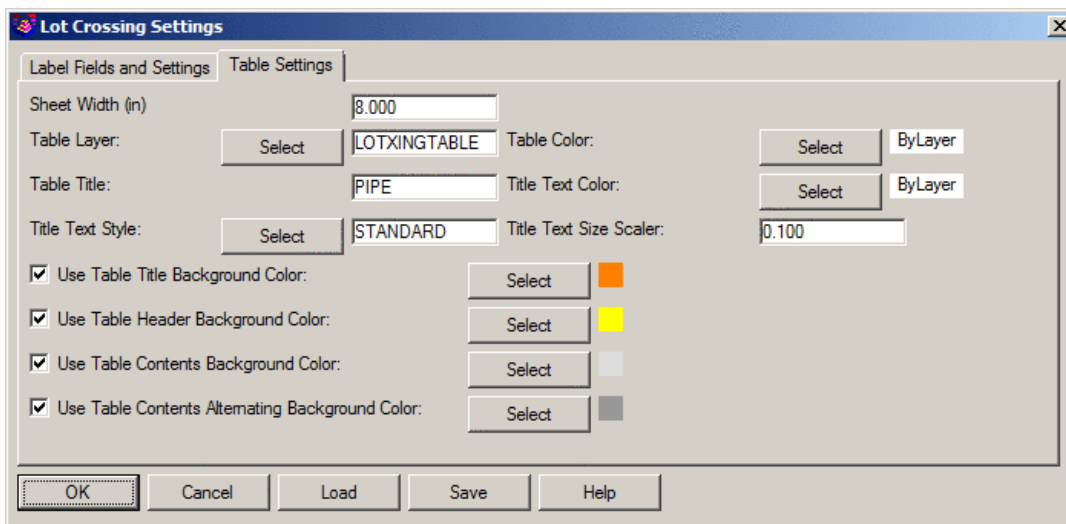
OK: Clicking the OK button will proceed to the selection of the crossing polyline.

Cancel: ends the command with no table being created.

Load: Loads previously saved settings so table created match previous tables.

Save: Saves the settings as currently displayed for use on future tables.

Help: Load this file.



Include Header: This draws a header row with the field names.

Include Horizontal Lines: This draws horizontal lines between the rows of the table.

Sheet Width (in): This value defines the width of the table. If set too small the text in the table will overlap.

Table Layer: Select an existing layer to draw the table on using the **Select** button or use a new layer by typing the name in the edit field.

Table Color: The user can specify a color for the table gridlines using the **Select** button. Bylayer will use the color assigned to the Table Layer for the grid lines.

Table Title: A title can be specified for the table by typing the desired title in the Table Title field. See example below.

Title Text Color: This specifies the color for the Table Title text.

Title Text Style: This specifies the text style for the Table Title text. Be sure the style specified is defined in your drawing.

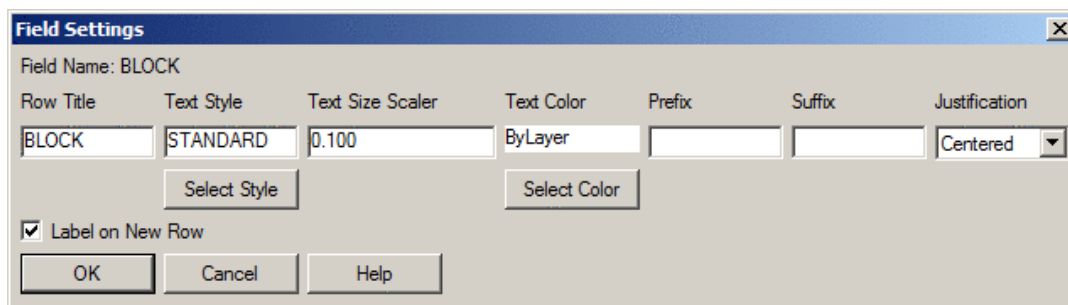
Title Text Size Scaler: This specifies the plotted height of the Table Title text. The table is drawn in model space. The height of the text in model space is the Text Size Scaler multiplied by the horizontal scale in Drawing Setup.

Use Table Title Background Color: This option allows the user to specify a background color for the Table Title.

Use Table Header Background Color: This option allows the user to specify a background color for the Table Header row.

Use Table Contents Background Color: This option allows the user to specify a background color for body of the table.

Use Table Contents Alternating Background Color: If the Table Contents Background color is being used, This option allows the user to specify a second color to use on alternating rows of the table body.



The Field Settings dialog box is opened by double-clicking a Used Label or highlighting a Used Labels and clicking the **Setup** button.

Row Title: Row Titles are the Used Labels that were selected Label Fields and Settings tab.

Text Style: This specifies the text style to be used for the current row text. You may use the **Select** button to choose a text style. Be sure the selected text style is loaded in the drawing.

Text Style Scaler: This specifies the plotted height of the current row text. The table is drawn in model space. The height of the text in model space is the Text Size Scaler multiplied by the horizontal scale in Drawing Setup.

Text Color: This specifies the color for the current row text. You can use the **Select Color** button to choose the color from a pallet. Bylayer uses the color of the Table layer for the text.

Prefix: This places the user provided prefix text with the row entries. An example would be prefixing lot numbers with the word Lot.

Suffix: This places the user provided suffix text with the row entries. An example would be using ft for feet as a suffix for a length.

Justification: Users can specify, Left, Center or Right text justification.

Calculated Numeric Values

Table numeric values that are calculated, like area or lengths, have the two following controls in addition to those listed above.

+/-: Users may specify a +/- be used as a prefix or suffix. The default is None, not used.

Precision: Decimal precision for calculated numeric values can be set to zero and up eight decimal places.

OK: Saves changes and closed the Field Setting dialog box.

Cancel: Closes the Field Settings dialog box without saving changes.

Help: Accesses this documentation.

PIPE					
LOT	Lot 16	Lot 15	Lot 14	Lot 13	Lot 12
BLOCK	1	1	1	1	1
Feet	219.14 Ft	203.93 Ft	176.67 Ft	143.92 Ft	172.48 Ft

Prompts

Pick a polyline for lot crossings: Select polyline crossing lots

Starting Station <0.0 >: Enter desired starting station

Ending Station <1642.88> Accept full length or enter ending station to process a part of the polyline.

Pick location for report table: Select location in drawing for table

Pulldown Menu Location(s): Survey Module: Area/Layout > Lot File Utilities > Right of Way Crossings Table

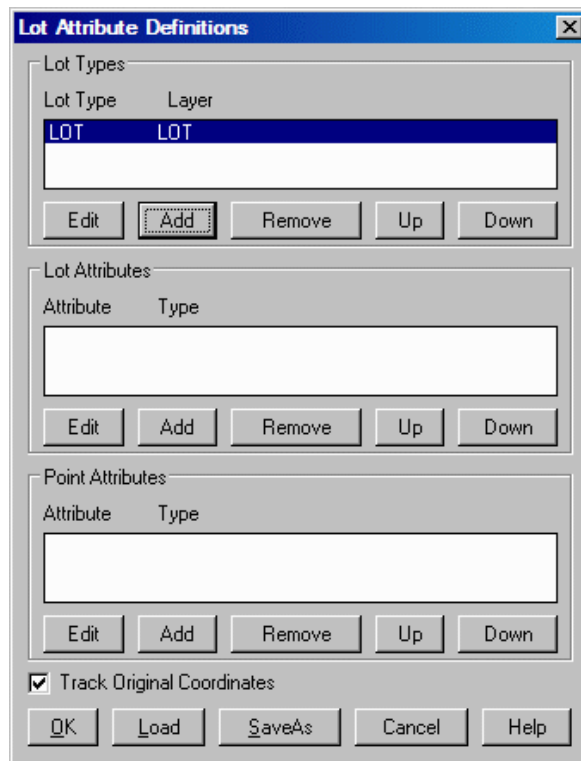
Keyboard Command: lotcross

Prerequisite: Polyline and Lot File

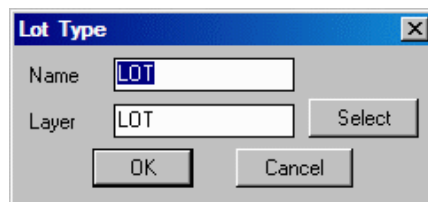
Define Lot Attributes

This command allows the user to define the Lot Type, Lot Attributes and Point attributes. With the use of the opening Lot Attribute Definitions dialog box, shown below, this routine allows you to edit, add, remove or reposition all of these definition types. You can save the selected data to a new Lot Attribute Definition file (LTD). You are also able to load an existing LTD file to work with.

The **Lot Types** section of the dialog lists out the Lot Type and the layer associated with it.

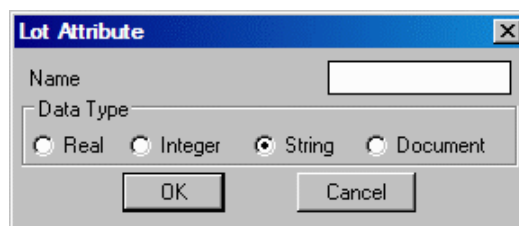


You can set up different lot types and a layer. When the lots are drawn, the layer name is used per lot type. Also, Lot Types are used in the lot report. There are also Lot Attributes, which are additional fields that you can define for the lots, such as deed number. And there is also Point Attributes.



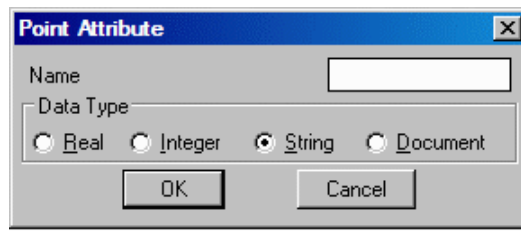
Edit/Add: Both the Edit and the Add buttons bring up the same Lot Type dialog, shown here. You can edit an existing lot or add a new one.

The **Lot Attributes** section asks for the Name and to enter the Data Type.



Edit/Add: Edit or add the name of the lot attribute. Choose from one of the four options for Data Type: Real, Integer, String or Document.

Similarly, the **Point Attributes** section also asks for the Name and to enter the Data Type.



Remove: Any of the Remove button will remove a lot type, lot attribute or point attribute from the list above it, depending upon which Remove button you use.

Up/Down (all three): Types and attributes can be repositioned.

Track Original Coordinates: This option will track the original coordinates of the lot so that this record may be kept for your future usage and needs.

Load: A Lot Attribute Definition file (LTD) can be loaded.

SaveAs: A new Lot Attribute Definition file (LTD) can be saved.

Pulldown Menu Location: Area/Layout

Keyboard Command: lotattr

Prerequisite: None

Import Lot File From MDB Database

This command will import a lot file from a Microsoft Access database file (.MDB) format.

Prompts

Database File to Import dialog *select existing .MDB file*

Lot File to Write dialog *select existing or create a new .LOT file*

Pulldown Menu Location: Area/Layout > Lot File Utilities

Keyboard Command: lotimport

Prerequisite: A lot (.LOT) file

Export Lot File to MDB Database

This Lot File Utilities command will export a lot file to a Microsoft Access database file format.

Prompts

Lot File to Export dialog *select existing .LOT file*

Database File to Write dialog *select existing or create a new .MDB file*

Pulldown Menu Location: Area/Layout > Lot File Utilities

Keyboard Command: lotexport

Prerequisite: A lot (.LOT) file

Export Lot File To Old SurvCADD

This Lot File Utilities command will export a Carlson lot file to SurvCADD .LOT file format.

Prompts

Source Lot File to Export dialog *select existing .LOT file*
Destination Lot File To Write dialog *create a new .LOT file*

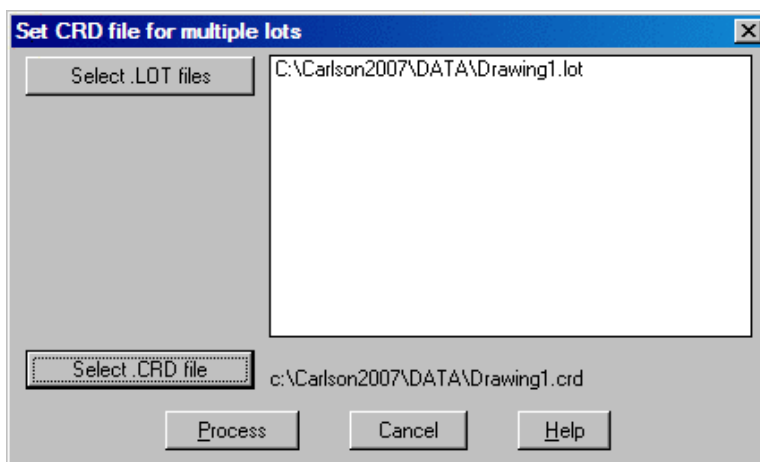
Pulldown Menu Location: Area/Layout > Lot File Utilities

Keyboard Command: lotexport2

Prerequisite: A lot (.LOT) file

Set CRD File for Lot Files

This command allows you to set the coordinate (.CRD) file that is associated with any number of lot (.LOT) files. This can be useful if the name or location of the coordinate (.CRD) file is changed. In the Set CRD for Multiple Lots dialog, press the Select .LOT files button to select any number of lot (.LOT) files. They are added to the list. Next, press the Select .CRD file button. After you have selected the files, press the Process button.



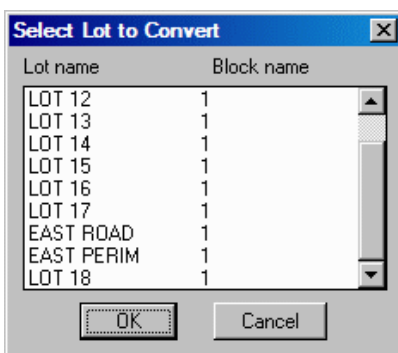
Pulldown Menu Location: Area/Layout > Lot File Utilities

Keyboard Command: lotscrd

Prerequisite: Existing lot (.LOT) file(s)

Lot File to Centerline

This command creates a centerline (.CL) file from a lot (.LOT) file. Since the lot definitions contain a series of points and a starting station, the lot (.LOT) file contains the necessary data to create a centerline. The Select Lot to Convert dialog lists the available lot names in the current lot (.LOT) file. Select a single lot to process, then specify the centerline (.CL) file name to create.



Prompts

Centerline File to Write dialog *enter new centerline (.CL) file name*

Select Lot to Convert dialog *select a lot from the list*

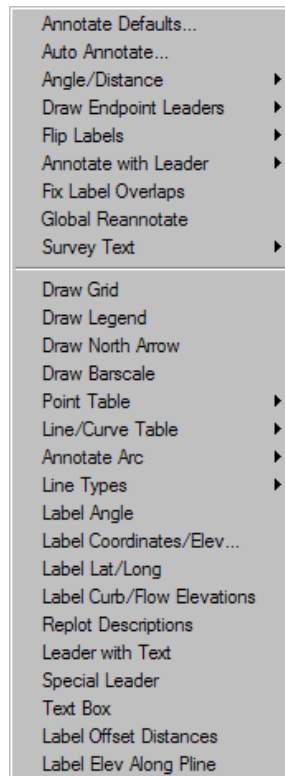
Pulldown Menu Location: Area/Layout > Lot File Utilities

Keyboard Command: lot2cl

Prerequisite: None

Annotate Menu

These menus include commands for labeling lines with bearing/azimuth and distances, special lines, coordinates, curves, curve tables and line tables. The precision of labeled distances and coordinates are set and controlled with the *Annotate Defaults* command.



Angle/Distance

The Angle/Distance sub-menu contains many commands for labeling the angle and/or distance of line segments. The line segments can be defined by picking a line, picking a polyline segment, entering two point numbers or picking two points. The angles can be labeled in bearing, azimuth or gon format. In the command names, the "_" indicates which side of the line label will appear. For example, "Bearing_" will label the bearing above the line and "_Bearing" will label below the line. There is also a Custom Label Formatter option. When this command is used and Option is chosen, there will appear a Custom Line Label dialog with various settings.

Prompts

Define bearing by, Points/type in Bearing/<select line or polyline>: P

1st Point ?

Pick point or point number: 11

PtNo. North(y) East(x) Elev(z) Description

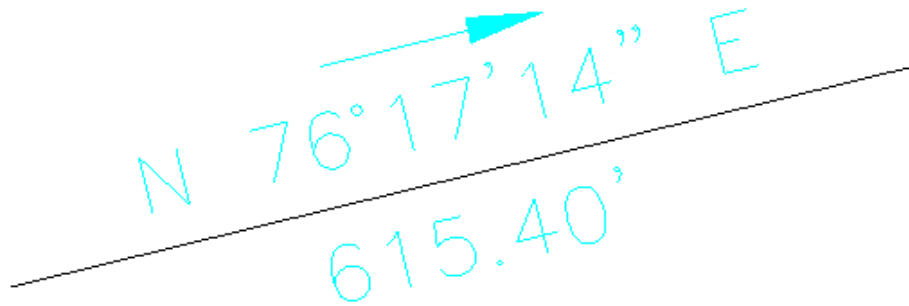
11 4869.06 4390.31 0.00

2nd Point ?

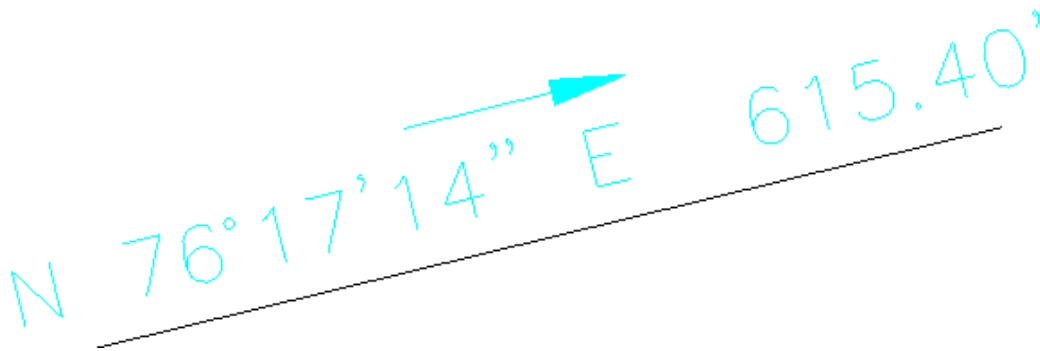
Pick point or point number: 2

PtNo. North(y) East(x) Elev(z) Description

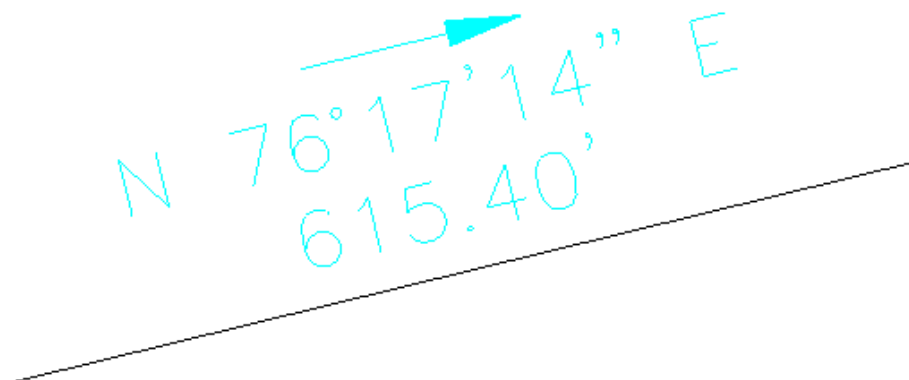
2 4610.89 4078.44 0.00



Bearing Distance



Stacked DistanceBearing



BearingDistance

Pulldown Menu Location: Annotate > Angle / Distance

Keyboard Command: Bearing and distance or bearing only or distance only: brg-, bbrg, brg_dis, dis_brg, brgdis-, bbrgdis, dis-, bdis, bbrgdis; Stacked labels: stackbd, stackdb2, stackdb3, stackdb4; Azimuth: AZI-, BAZI, AZI2-, AZI3-, azidist2, azidist3, azidist4, dist_azi; Gon: gonlab, gonlab2, gonlab3, gonlab4.

Prerequisite: None

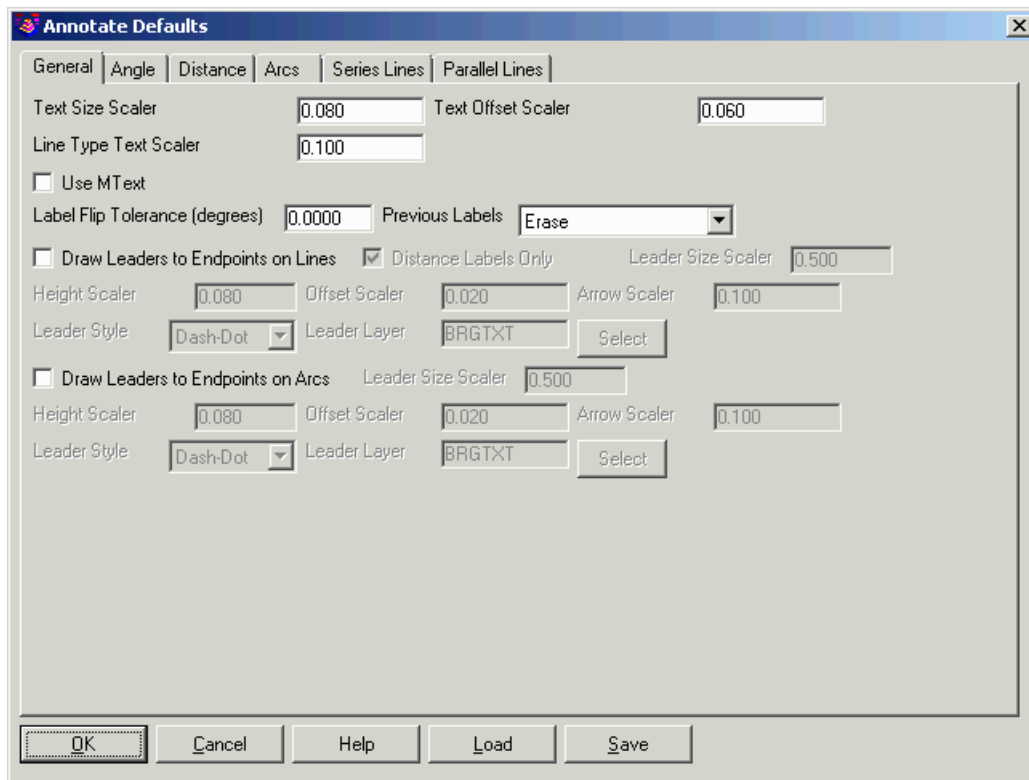
Annotation Defaults

This command sets the defaults for the annotation menus and controls the way various annotation commands work. Some of these defaults can be changed globally by running *Configure* command, which changes the file *COGO.INI* so that every time you start Carlson, the new defaults are set. When this menu option is selected the Annotate Defaults dialog appears.

This dialog is broken into 5 tabs: General, Angle, Distance, Serial Lines and Parallel Lines.

General Tab

This tab is used for settings that apply to all annotation types.



Text Size Scaler: This value is multiplied by the horizontal scale value to set the text size units. The Horizontal Scale is set in the Drawing Setup command.

Text Offset Scaler: This value multiplied by the horizontal scale defines the distance that an annotation label is placed from its defining line.

Line Type Spacing: Specifies the distance between the symbols on special line types.

Line Type Text Scaler: This value multiplied by the horizontal scale specifies the size of the symbols of special line types.

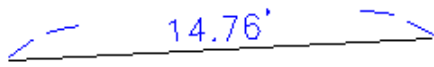
Use MText: This option creates the labels as MText instead of standard Text entities.

Label Flip Tolerance (degrees): Gives extra tolerance for label flipping for readability. Labels draw in the north-west quadrant that are within this number of degrees to due-north will be drawn upside down.

Spaces Between Bearing and Distance Labels: Controls number of spaces between labels when bearing and distance are labelled on the same row.

Previous Labels: Specifies if previous labels for the for the set of linework being annotated are kept or deleted. Setting values are Retain, Erase, Prompt Before Erasing.

Draw Leaders to Endpoints on Lines: This option creates leader lines (crow's feet) between the distance annotation and the line segment endpoints as shown below. These leaders are used to help identify the endpoints that were used to create the distance label.



Distance Labels Only: When checked, leaders will not be drawn unless the label includes a distance.

Leader Size Scaler: This option determines the maximum length for leaders. The size in drawing units will be the Leader Size Scaler multiplied by the Horizontal Scale (for example, $0.5 \times 50 = 25$). If the line segment is too short, the leader is shortened to fit.

Height Scaler: This option controls the height of the leader.

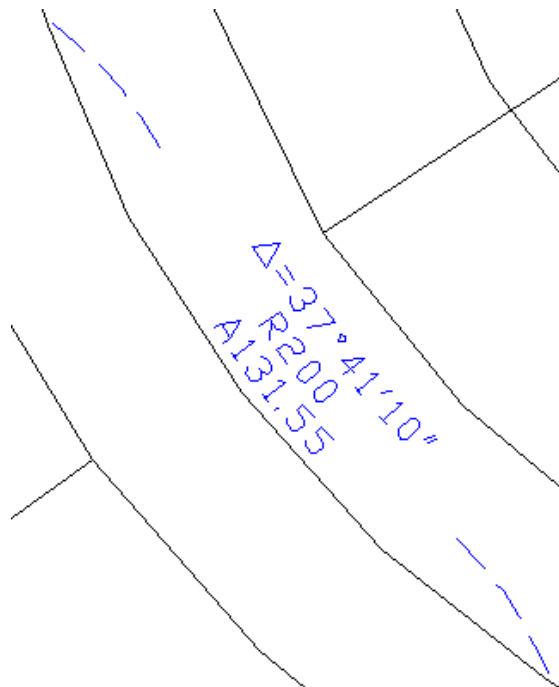
Offset Scaler: This option controls the distance between the line endpoints and the leader endpoints.

Arrow Scaler: This option controls the arrowhead size for leader styles with arrows.

Leader Style: This option determines which of the five styles of endpoint leaders to use. The styles are: Arrow-Arc, Arc-Arrow, Arc-Only, Dash-Dot, Dashed, Arrow-Line, Perpendicular and Custom. For custom, the program draws a block specified in the Block field of this dialog.

Leader Layer: This option determines the layer for drawing the leader.

Draw Leaders to Endpoints on Arcs: This option creates leader lines (crow's feet) between the arc segment endpoints as shown below. These leaders are used to help identify the endpoints that were used to create the arc label.



Leader Size Scaler: This option determines the maximum length for leaders. The size in drawing units will be the Leader Size Scaler multiplied by the Horizontal Scale (for example, $0.5 \times 50 = 25$). If the arc segment is too short, the leader is shortened to fit.

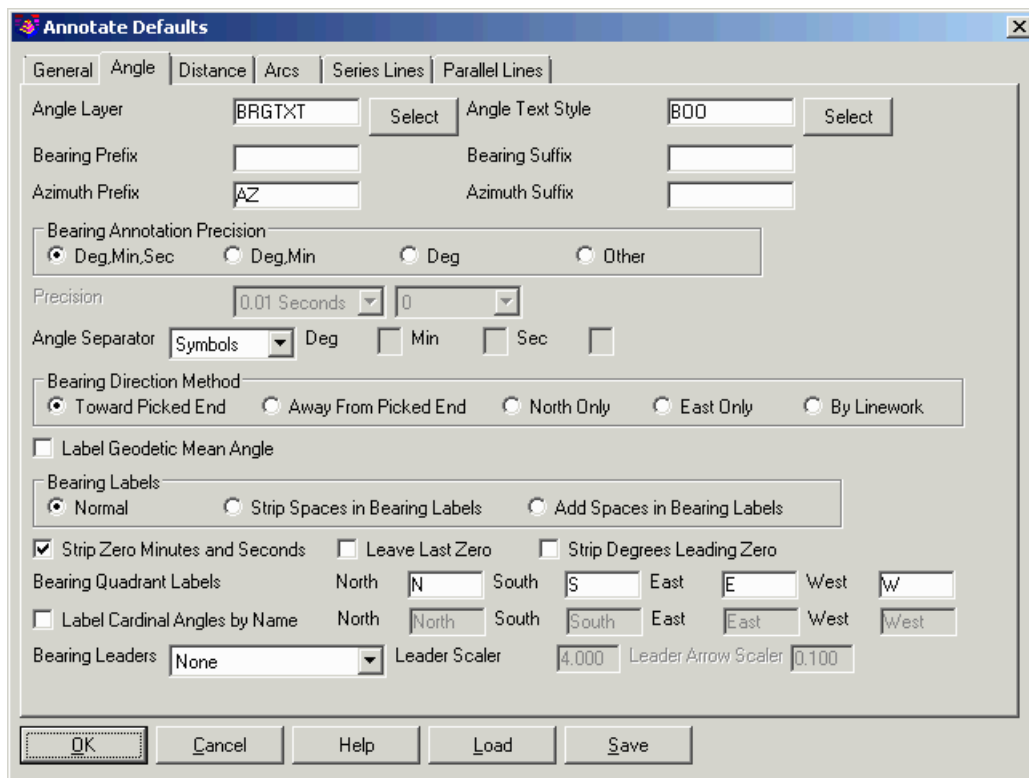
Offset Scaler: This option controls the distance between the arc endpoints and the leader endpoints.

Leader Style: This option determines which of the five styles of endpoint leaders to use. The styles are: Arrow-Arc, Arc-Arrow, Arc-Only, Dash-Dot, Dashed and Arrow-Line.

Leader Layer: This option determines the layer for drawing the leader.

Angle Tab

This tab is for settings that apply to angle labels:



Angle Layer: This specifies the layer to be used for angle labels.

Angle Text Style: This specifies the text style to be used for angle labels.

Bearing Prefix and Suffix: Specifies the prefix and suffix text for bearing labels.

Azimuth Prefix and Suffix: Specifies the prefix and suffix text for azimuth labels.

Bearing Annotation Precision: Specify the display precision for bearing labels.

Angle Separator: Choices are Symbol, Hyphen, Space, Other. When Other is chosen the Deg. Min. and Sec. fields are enable to allow the user to enter custom angle separators.

Bearing Direction Method: Choose the orientation of the bearing. This controls how lines selected for bearing or azimuth annotations will be referenced.

Toward Picked End: If this option is chosen, the line will be labeled in the direction of the endpoint that is closest to the point where you selected the line.

Away from Picked End: This labels the line in the direction away from the closest endpoint.

North Only: This option controls whether bearing annotations will always be labeled in the north quadrants (NE or NW) and never in the south quadrants.

East Only: This option controls whether bearing annotations will always be labeled in the east quadrants (NE or SE) and never in the west quadrants.

By Linework: This option labels the line in the direction that the line was drawn.

Label Geodetic Mean Angle: Instead of labeling the direct coordinate bearing between two points, this option labels the geodetic mean angle which is the average of the geodetic bearings at the two points. This method converts the drawing coordinates to lat/lon and calculates the convergence angles for both points. The projection must be defined under Settings->Drawing Setup.

Strip Spaces in Bearing Labels: This option causes the spaces in bearing labels to be removed.

Add Spaces in Bearing Labels: This option puts spaces between the degree, minutes, and seconds numbers.

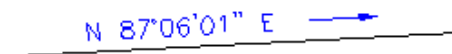
Strip Zero Minutes and Seconds: This option shortens the label by dropping either seconds and or minutes and seconds when they are equal to zero. The Leave Last Zero will keep a single zero in the label.

Strip Degrees Leading Zero: This option removes any extra leading zeros on angle degrees.

Bearing Quadrant Labels: These settings control the labels for the north/south prefix and east/west suffix for bearing labels.

Label Cardinal Angles by Name: When checked, the user is allowed to enter the labels that will be used for each of the four cardinal angles.

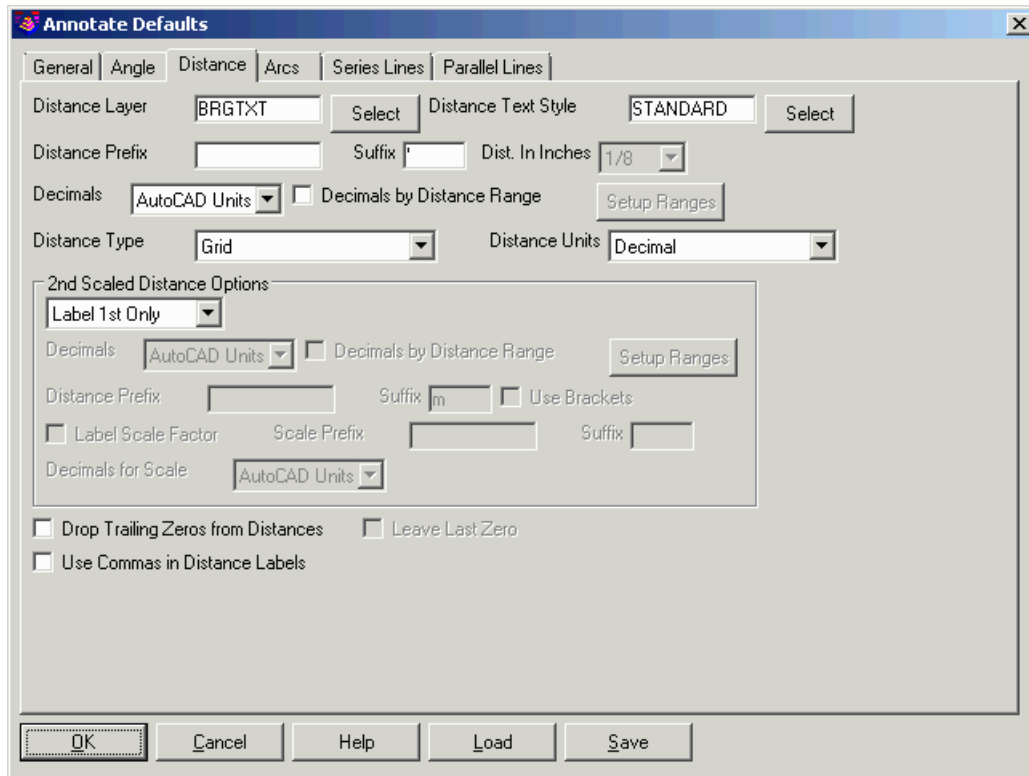
Bearing Leaders: The **Leader Scaler** controls the length for the leader. The **Offset Scaler** controls the distance between the bearing label and leader. The **Leader Arrow Scaler** controls the size of the arrowhead. The **Leaders To Side** option creates a direction arrow with the bearing annotation as shown below. The **Leaders After Bearing** option is similar to the Leaders To Side. The difference applies when having both the bearing and distance labels together. The Leaders To Side option puts the leader after the distance label. The Leaders After Bearing puts the leader between the bearing and the distance labels.



Leaders After Bearing method draws the bearing leader to the right side of the bearing label.

Distance Tab

This tab is for settings that apply to distance labels:



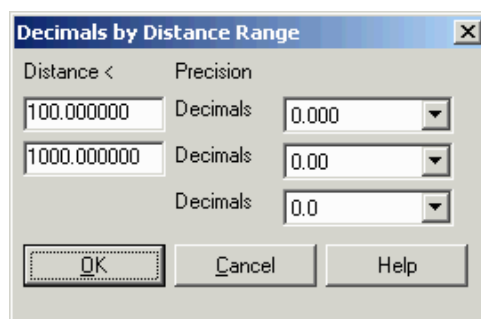
Distance Layer: This specifies the layer to be used for distance labels.

Distance Text Style: This specifies the text style to be used for distance labels.

Distance Prefix and Suffix: These specify the prefix and suffix that are added to distance annotations.

Decimals: The decimal places can be set to a specific number or set to match the CAD units which are set by the LUPREC system variable. The decimal settings with "5" round to the nearest half.

Decimals by Distance Range: This option uses different decimals depending on the distance amount. Pick the Setup button to define these decimals. In this dialog, distances less than 100 will have 3 decimals, distance between 100 and 1000 will have 2 decimals, and distance more than 1000 will have 1 decimal.



Distance In Inches: This controls the precision for inches from 1/2 to 1/256th of an inch when the Distance Units is set for inches.

Distance Type: This controls whether to label grid distances or geodetic distances at zero or mean elevation. The geodetic distances require the grid projection to be set in Drawing Setup.

Distance Units: This specifies the units used for distance labels. Choices are Decimal, Chains, Rods, Varas, "Feet and Inches" and Both.

2nd Scaled Distance Options: This option labels determines if a 2nd scaled distance is included in distance labels. This 2nd distance is scaled by the Report Scale Factor set in the Drawing Setup dialog. Choices for this option are "Label 1st Only" (label distances in current drawing units only), "Label 1st and 2nd" (label distances in both current drawing units and scaled by the Report Scale Factor) and "Label 2nd Only" (label distances scaled by the Report Scale Factor Only). There are separate settings for the 2nd Distance for the label prefix and suffix and decimal places. There is an option for labeling the scale factor used for the 2nd distance.

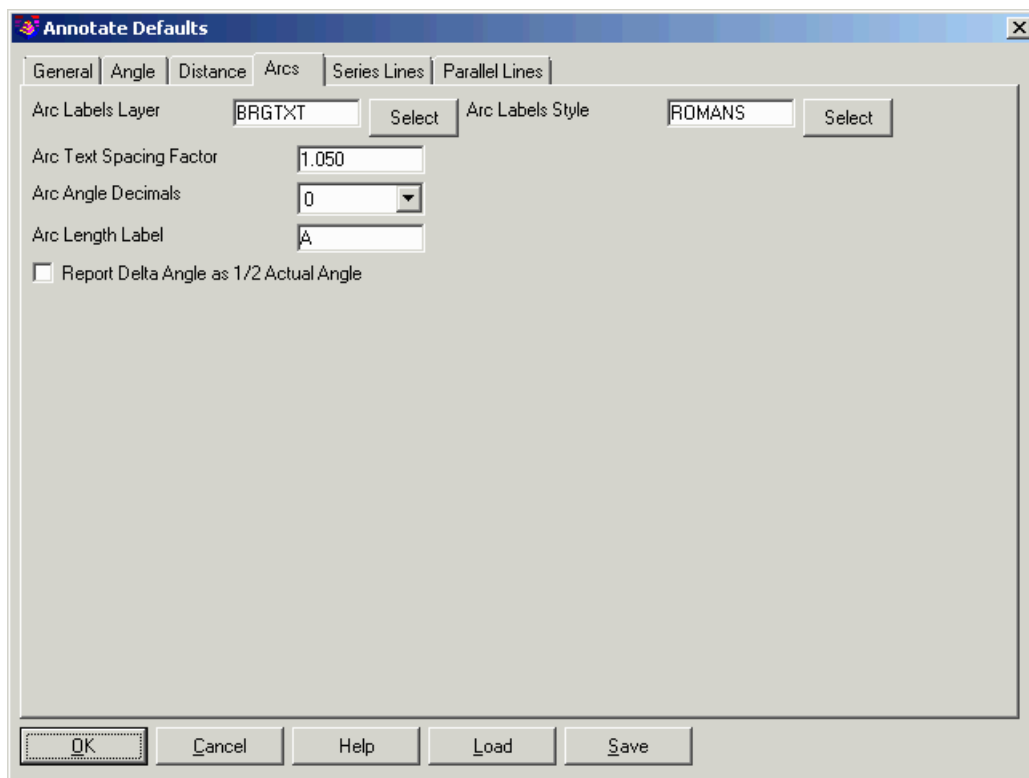
Label: This variable will be assigned as a suffix to the second scaled distance label.

Drop Trailing Zeros in Distances: This option allows you to drop trailing zeros on distance labels. **Leave Last Zero** will strip all except the decimal for tenths.

Use Commas in Distance Labels: This option formats the distances over 1000 with a commas (ie. 1,234.0).

Arcs Tab

This tab has settings for arc labels.



Arc Labels Layer: This specifies the layer to be used for arc labels.

Arc Labels Style: This specifies the text style to be used for arc labels.

Arc Angle Decimals: Controls the number of decimals for arc angle labels.

Arc Length Label: Specifies the prefix label for arc length labels.

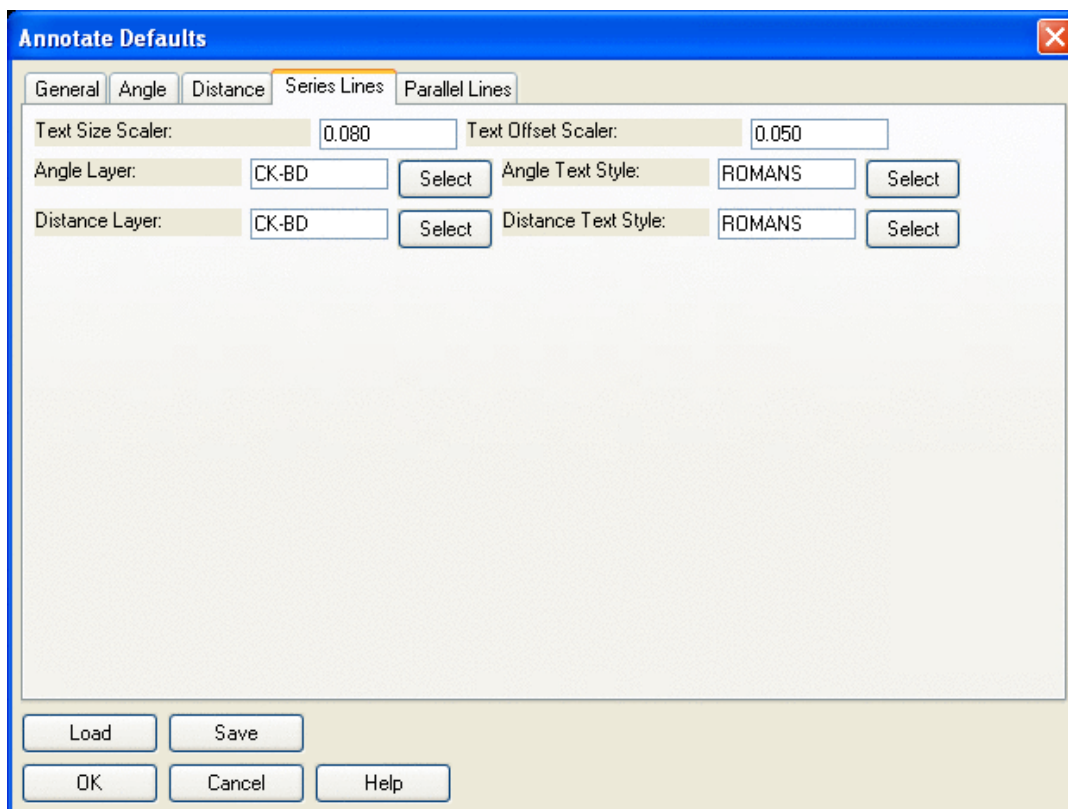
Arc Text Spacing Factor: This variable controls how close letters will be spaced when labeling arcs. The lower the number, the closer the spacing. The higher, the farther apart. (The suggested range between 0.8 and 1.5)

Report Delta Angle as 1/2 Actual Angle: The angle value in the label will be 1/2 the actual angle.

Strip Degrees Leading Zero for Delta Angle: Controls whether to have a leading zero which applies to delta angles under 100 degrees. For example, "054" verses "54".

Series Lines Tab

This tab is for settings that apply to Series Lines labels (See the section "Auto Annotate" for a detailed description of series line handling).



Text Size Scaler: This value is multiplied by the horizontal scale value to set the text size units for serial lines.

Text Offset Scaler: This value multiplied by the horizontal scale defines the distance that an annotation label is placed from its defining line for serial lines.

Angle Layer: This specifies the layer to be used for angle labels on serial lines.

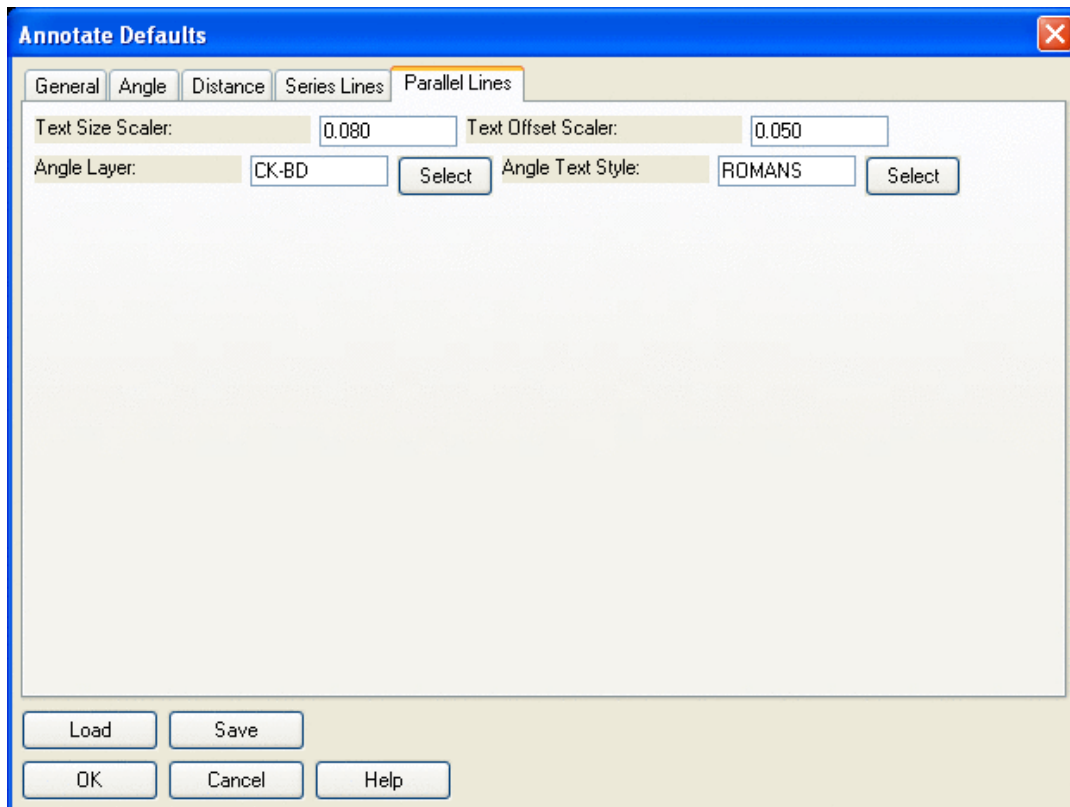
Angle Text Style: This specifies the text style to be used for angle labels on serial lines.

Distance Layer: This specifies the layer to be used for distance labels on serial lines.

Distance Text Style: This specifies the text style to be used for distance labels on serial lines.

Parallel Lines Tab

This tab is for settings that apply to Parallel Lines labels (See the section "Auto Annotate" for a detailed description of parallel line handling).:



Text Size Scaler: This value is multiplied by the horizontal scale value to set the text size units for parallel lines.

Text Offset Scaler: This value multiplied by the horizontal scale defines the distance that an annotation label is placed from its defining line for parallel lines.

Angle Layer: This specifies the layer to be used for angle labels on parallel lines.

Angle Text Style: This specifies the text style to be used for angle labels on parallel lines.

Load/Save: Choose these functions to load an existing annotation defaults file (.ADF) or save a new one, which will contain your current selections.

Pulldown Menu Location: Annotate

Keyboard Command: LDEF

Prerequisite: None

Auto Annotate

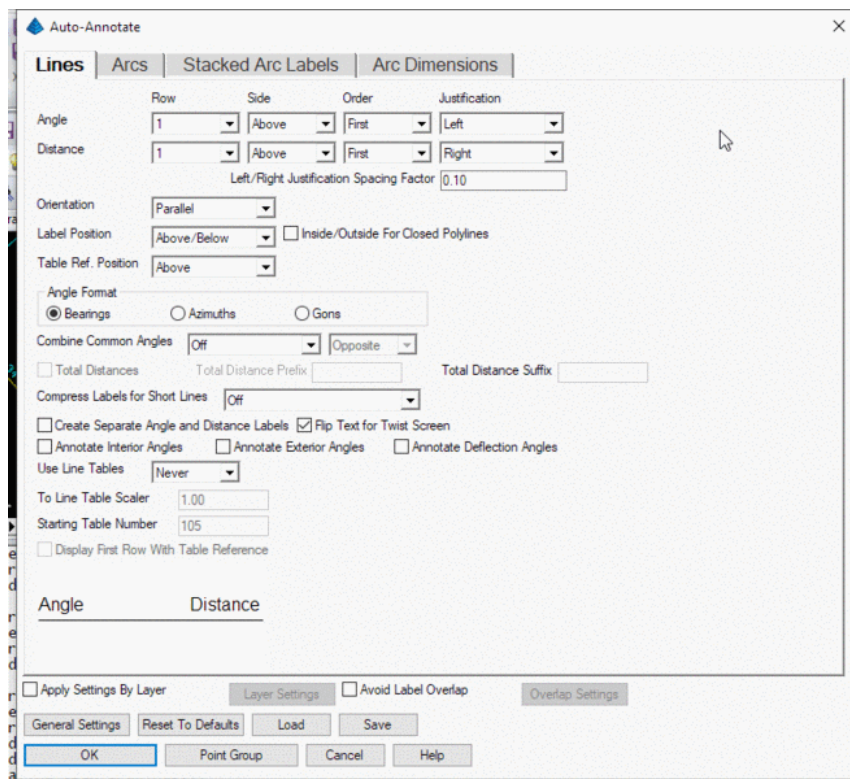
This command allows you to select a group of lines, arcs and/or polylines to be labeled. It allows for any combination of line and distance labeling, and also any combination of arc labeling.

You can position the features of the labels, once in the Auto-Annotate dialog, by using the Row, Side, Order, Orientation and Position Types options, all found under Lines tab. For Arcs, you can select the Arcs tab and determine the type of auto-annotating you would prefer for arc entities. As you select different options, you can see the changes in the preview display of the entry dialog. You will select the Angle Format in terms of Bearing, Azimuths and Gons and there is an important feature that allows you to avoid label overlaps. This is done by applying specific, user-defined settings. When labeling arcs, there are options to set the label prefixes for curve annotation. The Settings button will bring you to the Annotation Defaults dialog, as explained in a previous section. Defaults will restore the prior settings.

Apply Label Settings by Layer brings up another dialog box which allows you to import from file, or load, predetermined configurations. There is an option to have different label settings applied by layer. Apply Label Settings By Layer allows you to set, load, and save your preferred variables.

The Avoid Label Overlap option can bring up a special dialog called the Overlap Manager. This screen, which contains extra tools for, as an example, sliding or stacking the labels that are overlapping and conflicting with drawing entities, gives you the real-time ability to move along the plan and make your corrections. This also will help you to avoid overlapping with other labels, text, symbols and linework – including fence and utility lines. In this Overlap Manager, docked on the left side of the screen. it is recommended that you use the Back and Next button frequently in order to review, adjust and correct your drawing.

Auto-Annotate dialog starts with the Lines (tab).



Angle/Distance: Allows you to enter the what row the Angle label is on, what side and the order of the label on the linework. The same applies for Distance labels. Notice the preview display changing.

Row: Using numbers (1 or 2), or choosing None, you can determine the order and appearance of the descriptions.

Note the change in the preview display.

Side: Choose inside or outside of the line.

Order: If you determine that the annotations are to be on the same row and same side of the line, then you must pick the order in which they will appear, from left to right.

Justification: This option gives the ability to left or right justify labels at ends of line or center justify the labels.

Left/Right Justification Factor: This value specifies, in percent, how far right (for left justified) and how far left (for right justified) the labels will be moved along the annotated line. This gives you fine control over the justification behavior.

Orientation: This offers this choice between parallel or perpendicular with regards to the labels' orientation to the line being labeled.

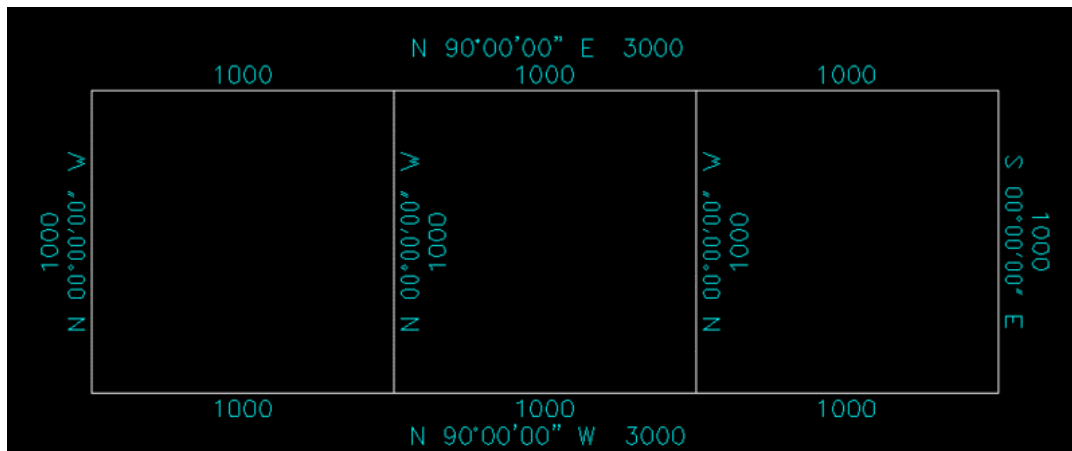
Position Types: Determined how each label is placed in relationship to the line and the other label. The **Inside/Outside For Closed Polygons** treats the first position type as the inside position and the second as outside for labeling closed polygons which applies when labeling lot polygons that are closed and you want a style like distances on the inside of the lots and bearings on the outside.

Angle Format: Bearing, azimuths or gons are the choices.

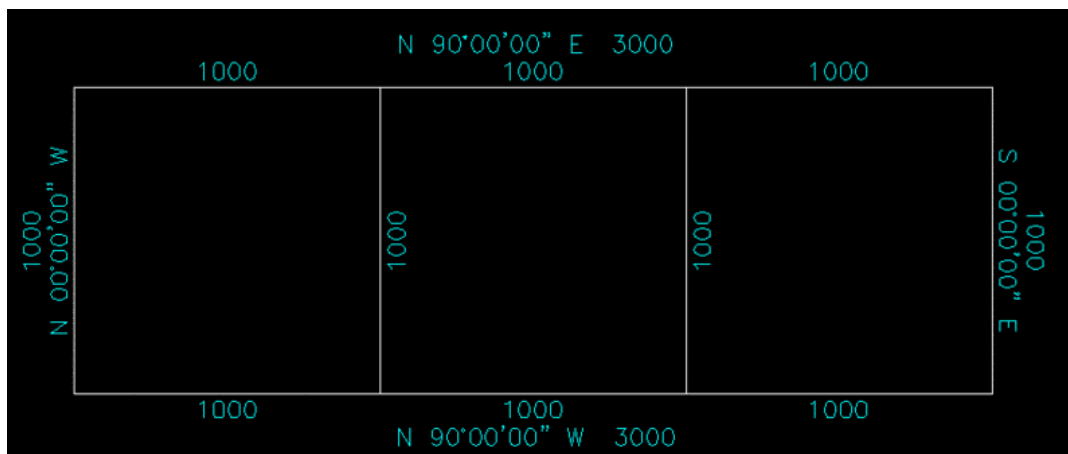
Combine Common Angles: This allows the user to reduce label clutter by minimizing labeling of serial and parallel linework. Choices are Off, Series, Parallel and "Series and Parallel". Series common angles are those where serially connected linework share the same angle. Common series angles are labeled at the mid-point of the series of connected line segments. When series common angles are selected they may be drawn stacked on the same side as the distance labels or on the opposite side from the distance labels. Also, for serial common angles the total distance may be included in the label. Parallel common angles are those where adjacent areas share parallel lines that include the line that bisects the areas. In this case, only the outer-most lines of the set of parallel lines will be labeled with the angle.

The common angle labels have separate settings for layer, style, size and offset. Please see the section "Annotate Defaults" for information on how to control these settings.

The following example shows the results of combining common serial labels, including totaling of the distances:



The following example shows the results of combining common serial and parallel labels:



Compress Labels for Short Lines: When angle and distance labels are being placed on the same side and row, this feature allows the user to place the label on different rows in the case that the label will not fit on the line otherwise. The options are Off, "Angle Above, Distance Below", "Distance Above, Angle Below", "Stacked Angle-Distance" and "Stacked Distance-Angle".

Add Space Between Angle and Distance Labels: When angle and distance labels are being placed on the same side and row, this feature allows the user to have the angle and distance labels spread apart from each other as allowed by the length of the line being annotated.

Reduce Space Between Angle and Distance Labels: When angle and distance labels are on the same row, this option puts a single space between them. Otherwise, there are two spaces.

Create Separate Angle and Distance Labels: When the angle and distance labels are on the same row, this option creates them as separate text entities. Otherwise, the labels are combined in a single text entity as long as their text styles match from Annotation Defaults.

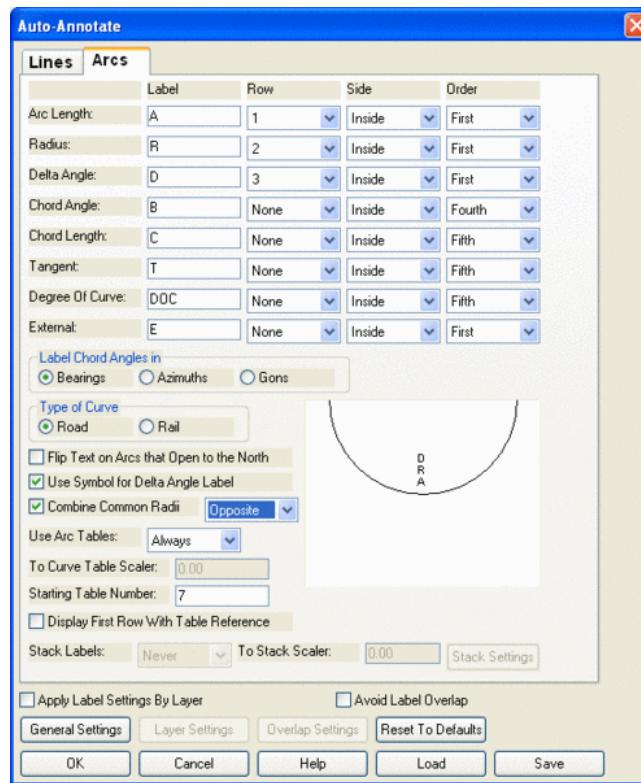
Flip Text for Twist Screen: This option automatically flips the labels when needed to make them right-side up.

Use Line Tables: Line tables are sometimes preferred as they keep the drawing linework clean and free of labeling. Choices are Always, Never or By Scaler. If By Scaler is chosen "To Line Table Scaler" is enabled.

To Line Table Scaler: If the length of the line is less than this minimum, the line is labeled as a line table entry. The To Line Table Scaler is relative to the current horizontal scale and represents the length of the line in plotted inches.

Starting Table Number: User choice. You might change this because perhaps you have another group of line labels, in table form, in the drawing. Line table entries are numbered sequentially beginning at the line Starting Table Number. The location for the line table can be picked if there is no current table. Otherwise, Auto Annotate will add to the end of the current line table. To set the location for the current line table, run the Table Header command in the Annotate > Line/Curve Table menu.

Auto-Annotate dialog box, by selecting the **Arcs tab**, displays the options for auto-annotating arcs. The columns are described, followed by the rest of the options.



Label: Here you might alter slightly the defaults by entering a letter or acronym that will represent to type of calculation. Or you could leave it alone.

Row: Using numbers, or choosing None, you can determine the order of the descriptions, and determine whether or not some might be left off altogether.

Side: Choose inside or outside of the arc.

Order: If you determine that the annotations are to be on the same row and same side of the curve, then you must pick the order in which they will appear, from left to right.

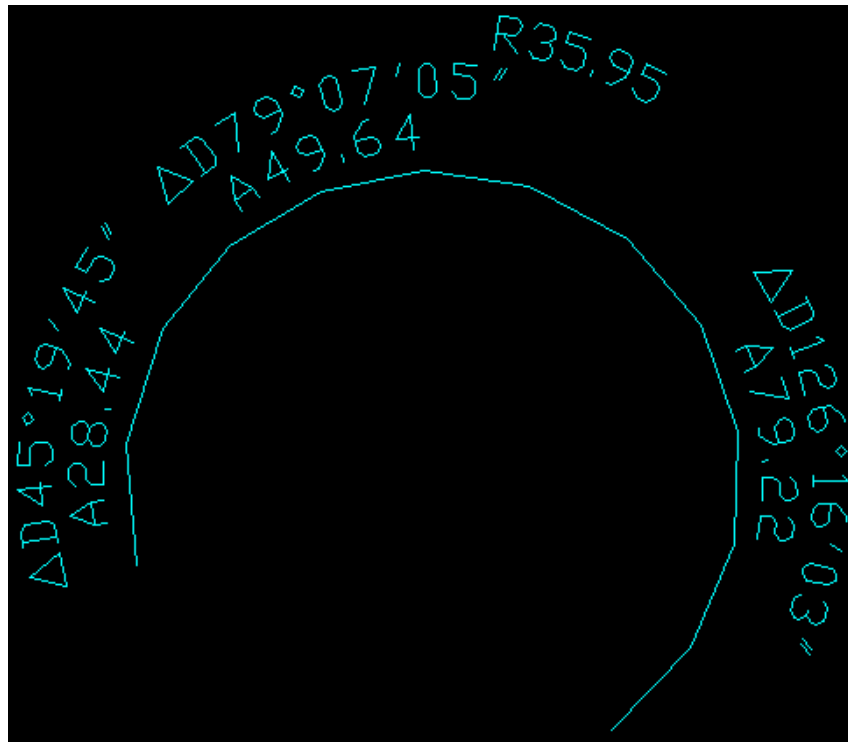
Label Chord Angles in: Bearing, azimuths or gons are the choices.

Type of Curve: Choose between Road and Rail.

Flip Text on Arcs that Open to the North: Clicking here might make for a easier to read finished plan. User preference.

Use Symbol for Delta Angle Label: The popular and traditional triangle-shaped symbol can be used, instead of the letter D, or any other letter(s).

Combine Common Radii: This allows the user to reduce label clutter by minimizing labeling of connected arc segments that share a common radius and center point. When selected, only one radius label will be generated for such arc segments. The following shows an example where a curve made of three arc segments is labeled with only one radius label. The radius label is placed offset to the mid-point of the combined arcs.



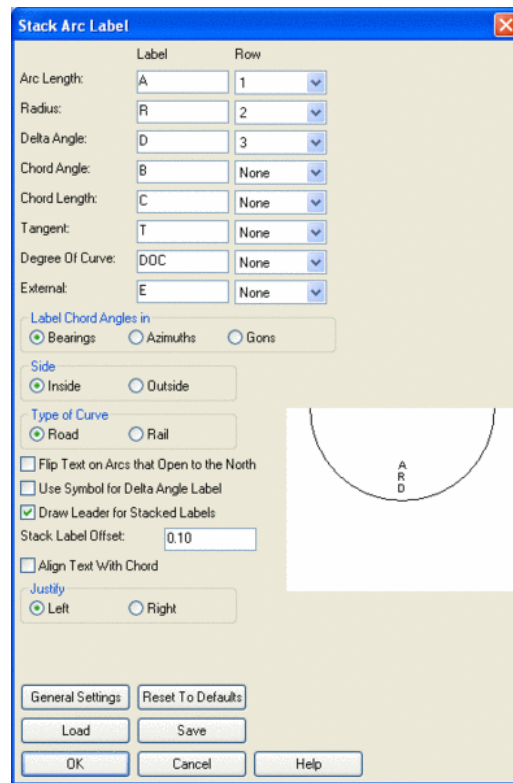
Use Arc Tables: Curve tables are sometimes preferred as they keep the drawing linework clean and free of labeling. Choices are Always, Never or By Scaler. If By Scaler is chosen "To Curve Table Scaler" is enabled.

To Curve Table Scaler: The To Curve Table Scaler applies when the Type of Arc label options is not set to Curve Table. If the length of the arc is less than this minimum, the arc is labeled as a curve table entry. The To Curve Table Scaler is relative to the current horizontal scale and represents the length of the arc in plotted inches.

Starting Table Number: The Starting Table Number is the starting number for the first line entered in the Curve Table. Curve Table entries are numbered sequentially from the curve Starting Table Number. The location for Curve Tables can be picked if there is no current table. Otherwise, Auto Annotate will add to the end of the current Curve Table. To set the location for the current Curve Table, run the Table Header command in the Annotate > Line/Curve Table menu.

Stack Labels: Stacked labels are sometimes preferred as they can help reduce label overlapping. Choices are Always, Never or By Scaler. If By Scaler is chosen "To Stack Scaler" is enabled.

To Stack Scaler: When Stack Labels is set to "To Stack Scaler" this control is enabled. If the length of the arc is less than this minimum, the arc is labeled as a stacked label. The To Stack Scaler is relative to the current horizontal scale and represents the length of the arc in plotted inches. The Stack Settings button is enabled when Stack Labels is set to Always or By Scaler. This button brings up the Stack Arc Labels which displays the options for creating stacked arcs labels. The columns are described, followed by the rest of the options.



Label: Here you might alter slightly the defaults by entering a letter or acronym that will represent to type of calculation. Or you could leave it alone.

Row: Using numbers, or choosing None, you can determine the order of the labels, and determine whether or not some might be left off altogether.

Label Chord Angles in: Bearing, azimuths or gons are the choices.

Side: Choose inside or outside of the arc.

Type of Curve: Choose between Road and Rail.

Flip Text on Arcs that Open to the North: Clicking here might make for a easier to read finished plan. User preference.

Use Symbol for Delta Angle Label: The popular and traditional triangle-shaped symbol can be used, instead of the letter D, or any other letter(s).

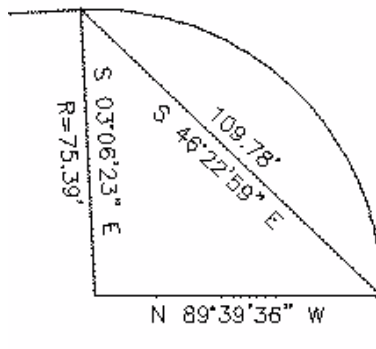
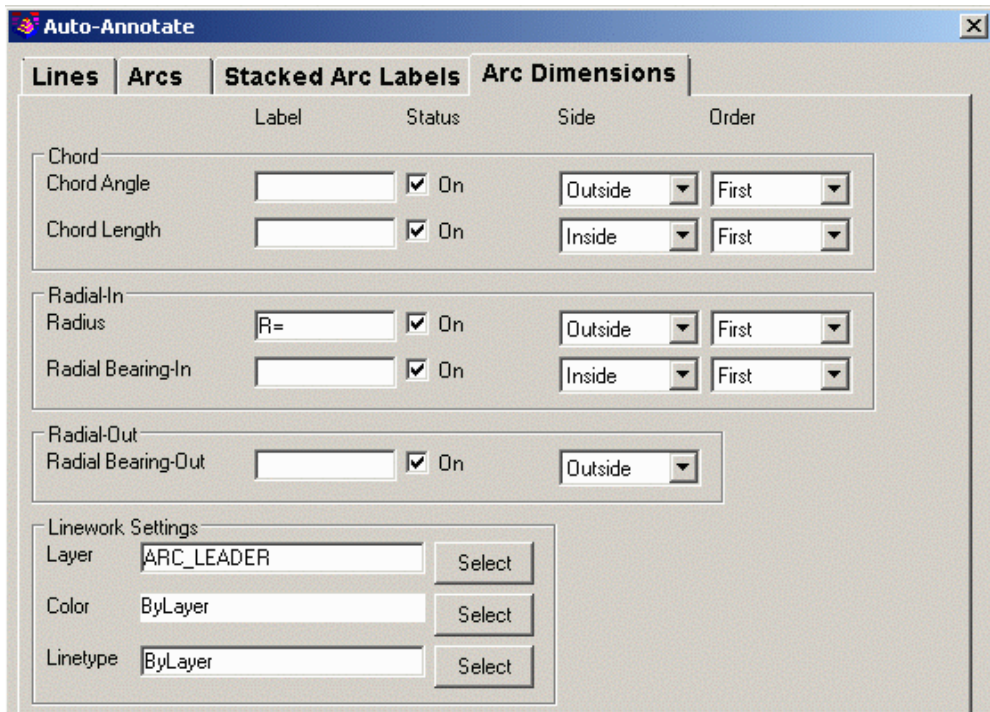
Draw Leader for Stacked Labels: When checked, a leader will be drawn from the stacked label to the mid-point of the arc.

Stack Label Offset: This value multiplied by the horizontal scale defines the distance that an annotation label is placed from its defining arc.

Align Text With Chord: Determine whether the stacked label is oriented horizontally (unchecked) or in the direction of the chord (checked).

Arc Dimensions

The Arc Dimensions style draws and annotates lines for each arc for the chord and radial lines. To use the Arc Dimensions label style, turn on **Use Arc Dimension Labeling** on the Arcs tab. On the Arc Dimensions tab, there are settings for the label prefix and position for the chord angle, chord length, radius and radial angles. You can also set the layer, color and linetype for the arc dimension lines.



Auto-Annotate dialog commands, common to both Lines and Arcs.

Apply Label Settings By Layer: See the Label By Label Settings dialog and details below.

Avoid Label Overlap: See dialog and details below.

General Settings: Brings you to the A ate Defaults dialog.

Layer Settings: Apply Label Settings By Layer option must be clicked in order to activate. You will then see the Label By Layer Settings dialog.

Overlap Settings: Avoid Label Overlap option must be clicked in order to activate. Brings up the Avoid Label Overlap dialog.

Reset to Defaults: This returns you to the default label values.

Point Group: This function prompts for a point group to use for the input data to annotate. The program uses the series of points to define the lines and arcs to annotate.

Load: You can load an existing .AAN file.

We will now say, for example, that with linework only to label in the drawing we run this routine. We first decide to go without the Avoid Label Overlap feature. This can be done by unclicking this option in the Auto-Annotate dialog. We will say that there is a fence line cutting through our property line, the property lines being the lines that we want to auto-annotate. In going without Auto Annotate's overlap protection, we perform Auto Annotate and we see that there is an overlap, with the labels running into the property lines and the fence line.

Panning and zooming the screen shows the problems we confront. Now, run Auto annotate again, but this time click ON the Avoid Label Overlap feature. Then click Overlap Settings button which brings up a dialog as shown below. This program and this specific dialog box has many different methods for fixing the overlaps. We will choose the different methods to apply.

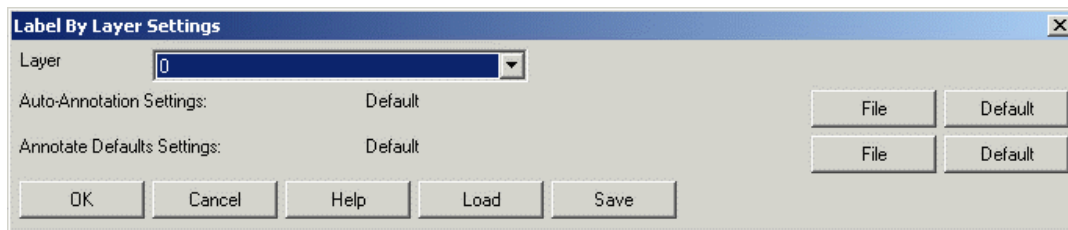
First, we will choose Slide. This slides the labels along the linework. We can even choose a maximum amount of slide and other related parameters. We will also turn on the Stack method. The Avoid Linework Conflicts feature pertains to that fence line we have. Finally, click OK. Now can pick the linework. Note that you do not need to erase the existing auto annotate labels ahead of time. This command will remember that those labels were created with this command. It will simply replace the entire group of labels with the new auto annotate labels.

The result, with overlap detection on, is that this routine fixed 7 out of 7 of the conflicts. It slid some of the labels over and stacked others. You can also run Auto Annotate Overlap with manual mode. To do this, remove the automatic options (such as Stack, Slide, etc.) and click View Remaining Overlaps After Applying Rules ON. Say OK. It docks the Overlap Manager on the left side of the screen.

You can then fix the conflicts with this Overlap Manager by using the different methods presented in this new window. This manager will highlights the conflicts, it will, for example, slide to the next conflict and allow you to pick a new position. Hit the Next several times. Again, stack one, slide another over, and perform other changes. Then choose Close.

Also, remember that depending on the linework layer, you can even have different annotation styles. There is also an option to have different label settings "by layer". These decisions are made by using the Label By Layer Settings dialog options. To get to this dialog, click on the Layer Settings button at the bottom of the Auto-Annotate dialog.

Label By Layer Settings option and dialog.



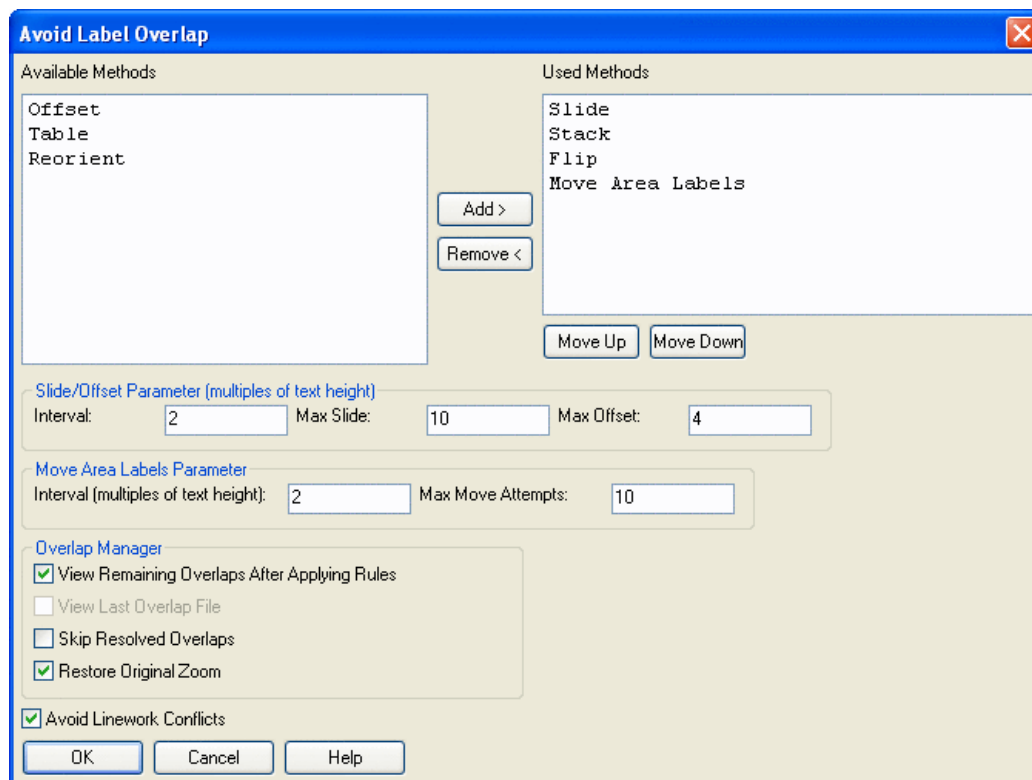
Layer: Select a layer from the existing list of layers. If the linework you select and to be labeled is on this layer, the parameters that you set in this dialog will be reflected in all labels.

Auto-Annotation Settings: Select an existing Annotation Settings file (AAN) by clicking the **File** button on the right. Or stick with the defaults.

Auto-Defaults Settings: Select an existing Default Settings File (ADF) by clicking the **File** button on the right. Or stick with the defaults.

Load: Select this option in order to load an existing layer file (LAY) to load.

Avoid Label Overlap option and dialog.



Overlap Settings dialog

Available Methods: Your choices. Pick from these.

Used Methods: Different ways in which this routine attempts to resolve the label overlaps. The overlap resolution attempt methods are applied in the order listed here.

Slide: If this is selected then the labels will be moved parallel to your linework until they do not overlap. The labels will not move past the end of the linework or the Max Slide which you determine.

Offset: will move your labels perpendicular to your linework as far as you set the Max Offset.

Table: Replaces your labels with a numbers and create a table of the numbers with the corresponding labels.

Reorient: If chosen, the labels will change orientation in the plain view to avoid overlapping.

Flip: It will flip your label onto the other side of the linework.

Stack: It will stack or unstack the text of your labels to avoid overlapping.

Move Area Labels: This method, which only applies to area labels, will attempt to move the area label to the closest place within the area that doesn't overlap with any other labels. You can control the move interval (distance between move attempts) and total number of move attempts by setting the values "**Interval (multiples of text height)**" and "**Max Move Attempts**" in the "**Move Area Labels Parameter**" section:

You can use any combination of these commands by using the add/remove button. You can also determine the order in which the command tries a method by using the **Move Up** and **Move Down** buttons. If a solution is not found by using the first method then the next method is used in descending order.

Add/Remove: Some methods you might prefer not to use.

Slide/Offset Parameter (multiples of text height): These are variable that help you to slide or offset the label(s) in question.

View Remaining Overlaps After Applying Rules: This option will help you to see what still needs treatment.

View Last Overlap File: When it is checked, the Overlap Manager will return to the previous labels that were under review.

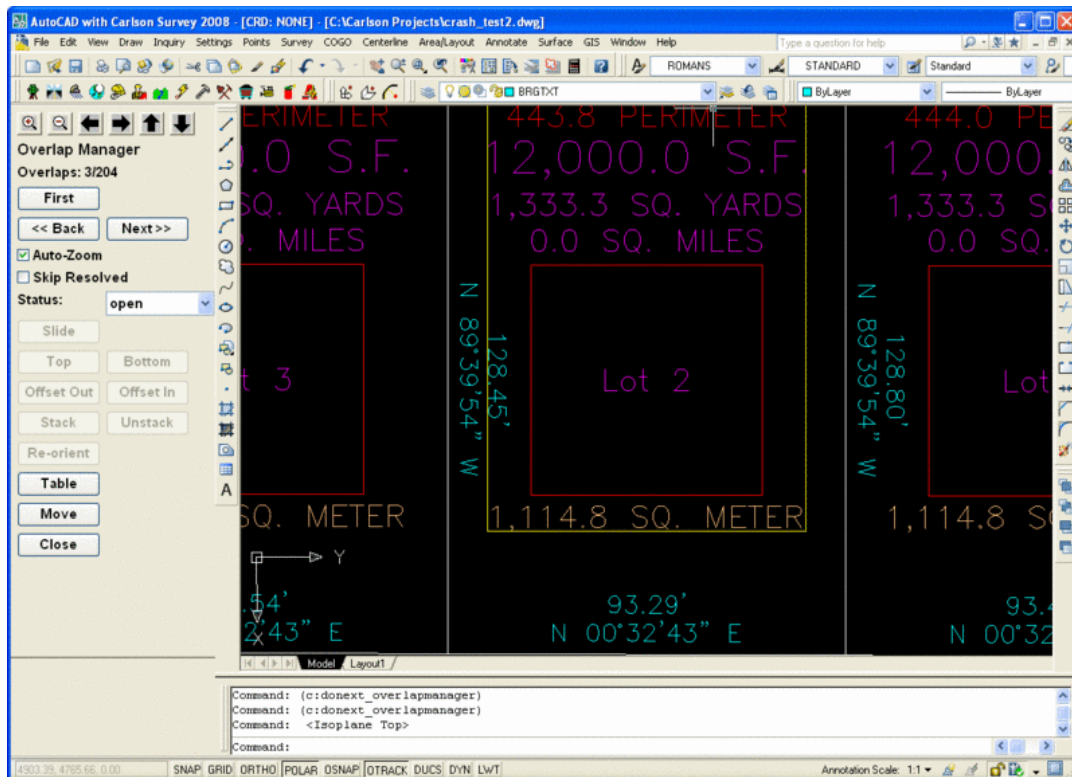
Skip Resolved Overlaps: When it is unchecked, the Overlap Manager will display all the labels that were moved by the command as a final check to you.

Restore Original Zoom: This will restore the zoom you were previously at before running the command.

Avoid Linework Conflicts: This is an extra precaution for when linework conflicts exist.

If there is a conflict, the following Overlap Manager dialog appears on the screen. It zooms to the conflict and provides you with the necessary tools to resolve the issues that need to be addressed. Many of the choices selected in the earlier dialog boxes can be modified yet again in the Overlap Manager, in your quest for a clean looking drawing. Within this special window you can zoom, pan, move to the next conflict, and perform many other tasks.

The **Overlap Manager** screen appears as a docked dialog window to the left of the main screen.



The Overlap Manager can be used to manually check and change label overlaps. The current overlap item will be have a yellow box drawn around it to help make it clear which item is the one currently being worked on. If you check on "View Remaining Overlaps After Applying Rules" then any remaining overlaps will be zoomed in on and you will have the ability with the Overlap Manager to flip through and fix or ignore the unresolved labels. When the current overlap item is an area label, only the Move and Table button will be enabled as these are the only two manual methods that can be applied to these types of labels. For line and curve labels, all methods will be enabled.

Prompts

Auto Annotate Dialog Choose settings and click OK.

Select Lines, Arcs, and/or Polylines to Annotate.

Select Objects: *pick entities*. Select the group of lines, arcs and/or polylines you want to annotate.

Pulldown Menu Location: Annotate

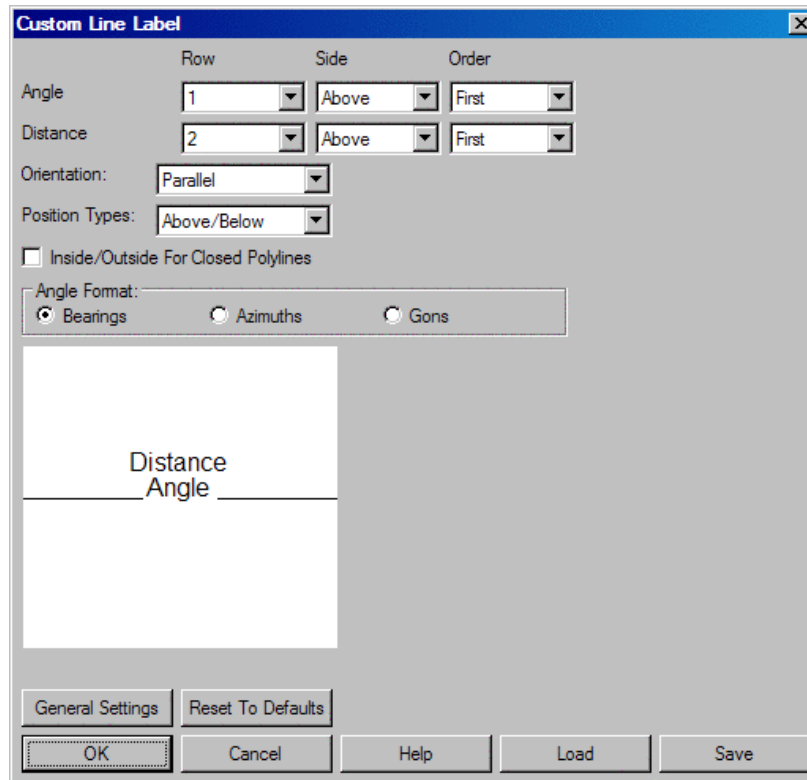
Keyboard Command: autoann

Prerequisite: Lines, arcs or polylines to annotate

Custom Label Formatter AD

This command allows you to customize the labeling for lines and polylines. You are first prompted to select a line or polyline to label, given the existing defaults currently set. The linework is shown as labeled on the screen. The

command line, shown below, also offers you an important choice called Options. When you type 'O' for options the below dialog box appears. In this dialog, there are three columns at the top of the dialog, along with other features. On the command line, there is also a choice called Format (F), which allows you to enter quick-key style keywords for quickly changing the label format. See below for these



Row: This column allows you to stack the data in different ways. You can place more than one item in the same row. If *None* is selected, then that item will not be displayed.

Side: This column allows you to place each item either inside or outside of the line or polyline.

Order: This column determines the order of items when they are placed in the same row.

General Settings: This button brings you to the Annotate Defaults dialog, see 'Annotate Defaults' for more.

Reset To Defaults: This button restores the default settings shown above.

Load/Save: You may also Load and Save different label configurations with the corresponding buttons.

Prompts

Options/Format/Points/<Select line or polyline>: *select entity*

Options/Format/Points/<Select line or polyline>: O

Custom Line Label dialog *choose your preferences and click OK*

You can decide to go into the Option dialog at the start of the command, or after your initial labeling. If you use the Format command line option, you will be asked to enter the Format command. The choices are:

B = bearing

A = azimuth

G = gon

D = distance

R = next row

_ = switch side of line

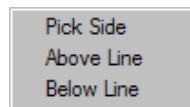
Pulldown Menu Location: Annotate > Angle/Distance

Keyboard Command: annline

Prerequisite: An arc to label

Draw End Point Leaders

These three commands draw a pair of leaders (crow's feet) at the ends of the line or polyline segment. The segment can be selected from a line, polyline or pair of points. The leaders are drawn above or below the line or polyline, or you can pick a side, depending on which Endpoint Leader command is run. The Pick Side command gives you the ability to place the crow's feet on a selected side of the line or polyline. Controls to customize the look of the endpoint leaders are accessed through the *Annotate Defaults* command in the Annotate menu. The Leader Size Scaler determines the maximum length of the leader. If the line segment is too short, the leader is shortened to fit. The actual length of the leader in drawing units is calculated by multiplying the leader scaler by the drawing horizontal scale (i.e., $0.5 * 40 = 20$). The Offset Scaler sets the distance that the leader head is off the line endpoint. There are four leader styles to choose from: Arc with Arrow, Arc Only, Dash-Dot-Dash and Dashed. Endpoint leaders can be drawn together with bearing/distance annotation by having the Draw Leaders to Endpoints option on under *Annotate Defaults*. This Draw End Point Leaders command allows you to add the leaders as another step.



Prompts

Define line by [Points/<select line or polyline>]: *Select a line or polyline.*

If you wish to define by points, enter "P" at this prompt and pick points on the screen, or type in point numbers. If a coordinate (.CRD) file has not been previously loaded, a dialog will open to allow you to select a coordinate (.CRD) file to process. While using the Point selection method, the last point picked in the selection is stored in default brackets. So if you are working around a boundary, simply press enter to accept the defaults for the first point and move ahead to the next point.



Arc with Arrow Endpoint Leader



Dashed Endpoint Leader

Pulldown Menu Location: Annotate

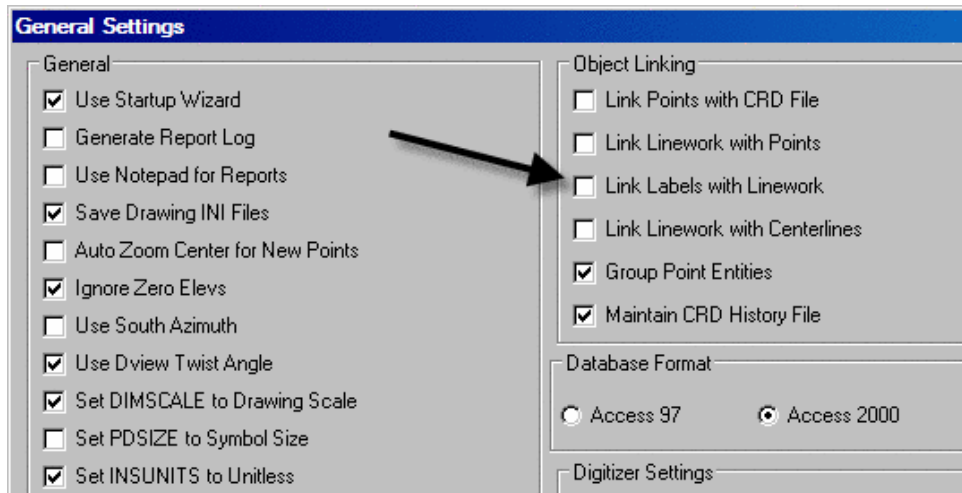
Keyboard Command: crowft

Prerequisite: None

Dynamic Annotation Note

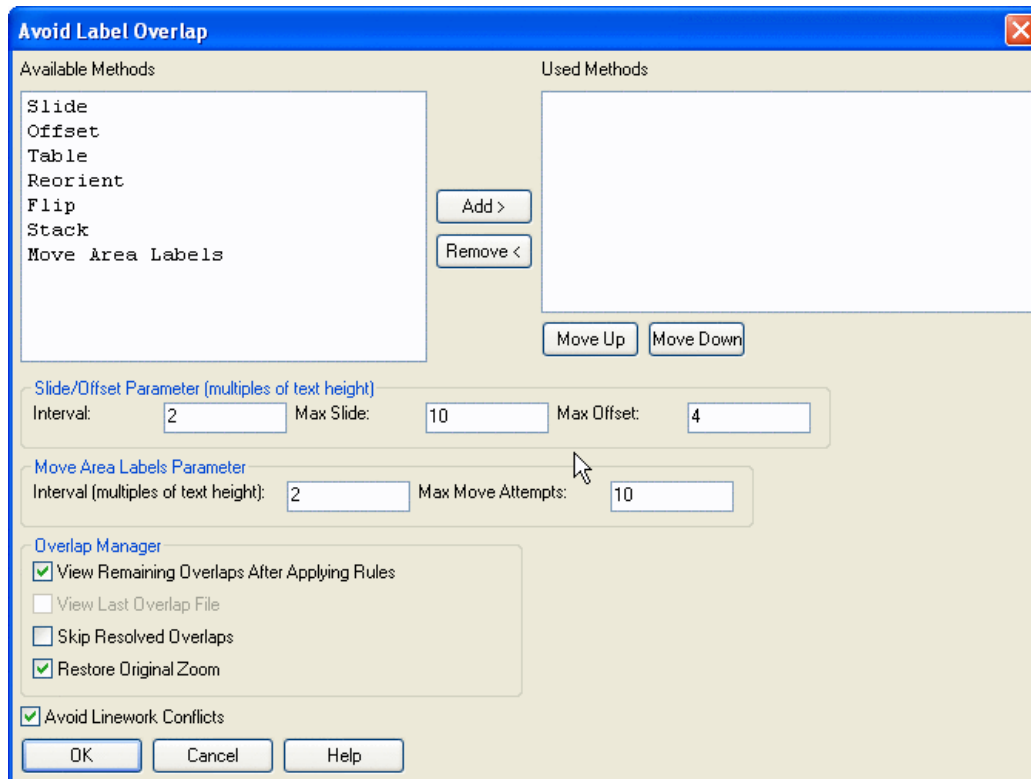
Bearing and distance annotations can be linked to the linework, such that the annotations will automatically update if the linework is changed. For example, if a line is moved with the AutoCAD *Move* command, the bearing label will update. This link can be found, and toggled on and off, under Object Linking in Configure > General Settings. Configure is in the Settings menu. The link is established between the label and the line, or polyline, when the label is created by commands such as *Auto Annotate*, *Line Table* or *Bearing Distance*. There are no links for annotation created using the Points option. To update bearing annotation without using the dynamic annotation, use the *Global Reannotate* command in the Annotate menu. To remove the links between the annotation and the linework entities,

use the *Remove Reactors* command, found under File > Drawing Utilities.



Fix Label Overlaps

This command allows you to fix label overlaps, where a conflict exists, for lines, arcs and polylines. You are immediately taken to the Avoid Label Overlap dialog. Here you can realign your labels by using a variety of optional methods. When the settings are to your liking, click OK. The command line then prompts you to select the entities for which to resolve annotation conflicts. Once you have selected your entities and hit Enter, this routine finds the conflicts and fixes the label overlaps.



If **Slide** is selected then the labels will be moved parallel to your linework until they do not overlap. The labels will not move past the end of the linework or the Max Slide which you determine.

Offset will move your labels perpendicular to your linework as far as you set the Max Offset.

Table will replace your labels with a numbers and create a table of the numbers with the corresponding labels.

If **Reorient** is selected then the labels will change orientation in the plain view to avoid overlapping.

Flip will flip your label onto the other side of the linework.

Stack will stack or unstack the text of your labels to avoid overlapping.

Move Area Labels will attempt move overlapping area labels to the closest place to the original position that does not overlap with other labels. The distance between move attempts and the number of move attempts is controlled by the **Interval** and **Max Move Attempts** settings of the **Move Area Labels Parameter** section.

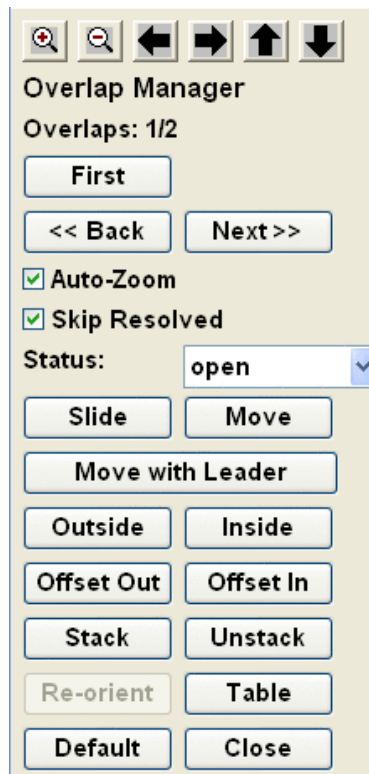
You can use any combination of these commands by using the add/remove button. You can also determine the order in which the command tries a method by using the **Move Up** and **Move Down** buttons. If a solution is not found by using the first method then the next method is used in descending order.

The **Overlap Manager** can be used to manually check and change label overlaps. If you check on "View Remaining Overlaps After Applying Rules" then any remaining overlaps will be zoomed in on and you will have the ability with the Overlap Manager to step through and fix or ignore the unresolved labels. When the current overlap item is an area label, only table and move buttons will be enabled as these are the only methods that apply. For line and curve label overlaps, the buttons for all methods will be enabled. Once a label is moved with the "Move with Leader", only Table, Default and "Move with Leader" will be enabled. The Default button can be used to restore the label back to its original state.

When **View Last Overlap File** is checked, the Overlap Manager will return to the previous labels that were under review.

When **Skip Resolved Overlaps** is unchecked, the Overlap Manager will display all the labels that were moved by the command as a final check to you.

Restore Original Zoom will restore the zoom you were previously at before running the command.



Prompts

Select Lines, Arcs, and/or Polylines for which to resolve annotation conflicts:

Select objects: *select entities*

Pulldown Menu Location: Annotate

Keyboard Command: annconf

Prerequisite: Annotation conflicts

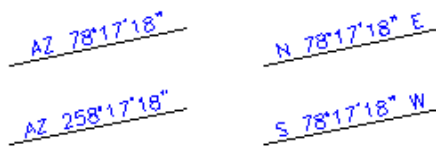
Switch Bearing/Azimuth Quadrant

This command switches the Bearing quadrant label or adds 180° to an Azimuth label. For example, N90°32'16"E would be replaced with S90°32'16"W or AZ 78°17'18" would be replaced with AZ 258°17'18". This routine changes bearing text to read as if the bearing were in the opposite direction.

Prompts

Pick Bearing or Azimuth Text: *pick text*

Pick Bearing or Azimuth Text: *press Enter to end*



Examples of switch bearing/azimuth quadrant

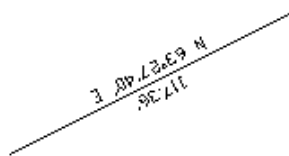
Pulldown Menu Location: Annotate > Flip Labels

Keyboard Command: brgquad

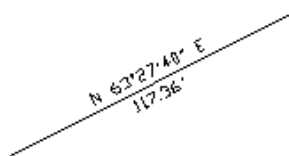
Prerequisite: bearing or azimuth label

Mirror Selected Labels

This command rotates a group of text 180 degrees and maintains the same text position. Use this command to rotate any text. Ignores all entities in the selection set except text.



Before Mirror Labels



After Mirror Labels

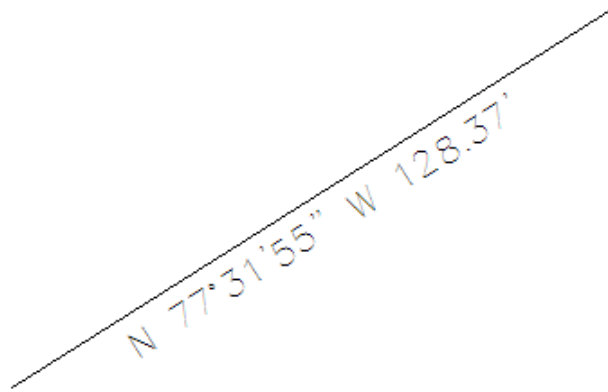
Pulldown Menu Location: Annotate > Flip Labels

Keyboard Command: flipset

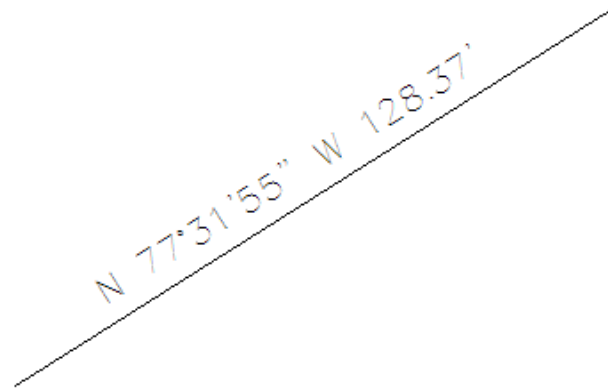
Prerequisite: Text to rotate

Mirror and Flip Selected Labels

This command mirrors the label to the other side of the labeled segment. At the new location, it then flips the label back to its original orientation. Use this command to manipulate any text. It ignores all entities in the selection set except text.



Before Mirror & Flip Labels



After Mirror & Flip Labels

Pulldown Menu Location: Annotate > Flip Labels >

Keyboard Command: MFLIP_LABELS

Prerequisite: Text to rotate

Flip Last Label

This command flips the last text drawn 180 degrees. Use this command to rotate your last annotation.

Pulldown Menu Location: Annotate > Flip Labels

Keyboard Command: flip

Prerequisite: Text to flip

Flip ON/OFF

When activated, the bearing and distance text will be rotated 180 degrees when drawn.

Pulldown Menu Location: Annotate > Flip Labels

Keyboard Command: flp

Prerequisite: None

Move Label with Leader

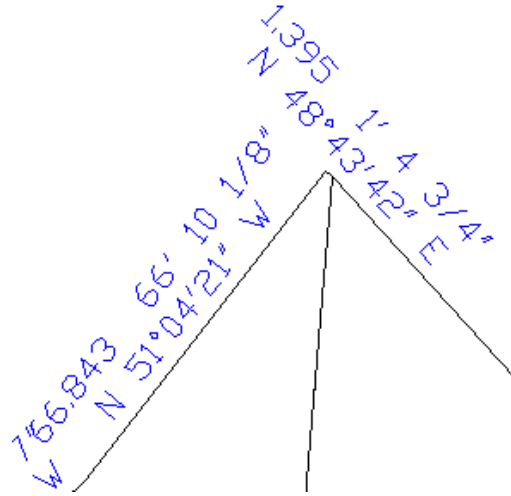
This command allows the user to make a leader label out of a selected angle/distance label.

Prompts:

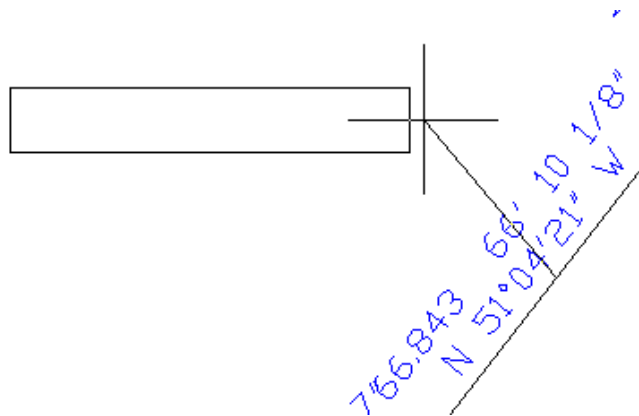
Select Label to Move (O for Options,R for Restore): pick an angle or distance label.

Pick end point for move: pick the end point of the move (end of leader).

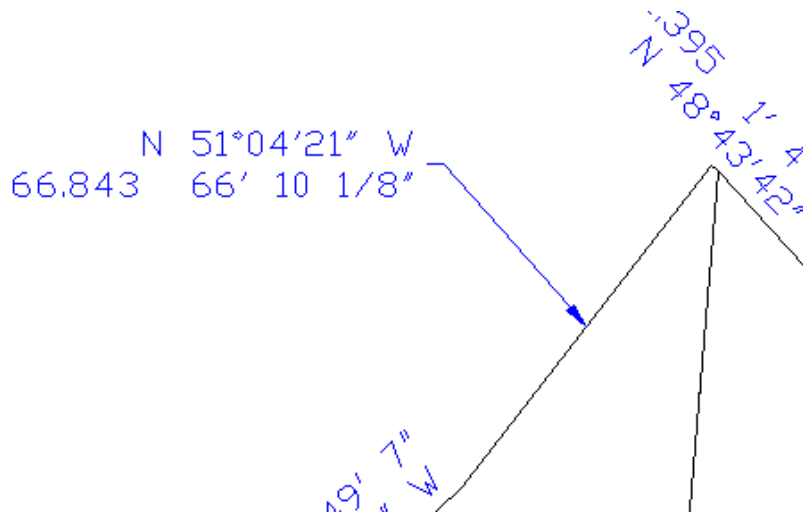
Select another Label to Move (O for Options,R for Restore,Enter to End): pick another angle or distance label if desired.



Before Move



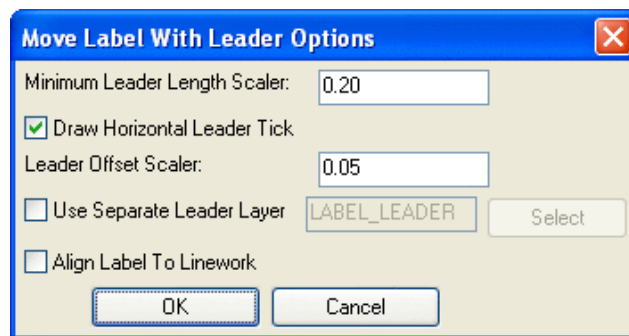
While moving the label, the user is shown where the leader and label will be drawn



After Move is Completed

Select Label to Move (O for Options,R for Restore): O

When Options is chosen the "Move Label With Leader Options" dialog allows the user to customize the leader and label drawing settings:



- Minimum Leader Length Scaler:** If the distance of the move is less than this value, a leader will not be drawn.
 - Draw Horizontal Leader Tick:** When checked, a horizontal leader tick will be drawn from the end of the leader towards the annotation.
 - Leader Offset Scaler:** This is used to set the distance from the end of the leader and the annotation.
 - Use Separate Leader Layer:** This allows the user to place the leader on a separate layer from the annotation.
 - Align Label to Linework:** When selected the orientation of the label will be parallel to the linework. Otherwise the label is orientated horizontally.
- NOTE: The leader scaler units (Minimum Leader Length Scaler and Leader Offset Scaler) are multiplied by the current horizontal scale value, which was set in the auto annotation dialog.

Select Label to Move (O for Options,R for Restore): R

Select Label to Restore: pick an angle or distance label that had been moved with the "Move with Leader" command previously.
The selected label will be restored to its previous state.

Pulldown Menu Location: Annotate > Annotate with Leader

Keyboard Command: annlead

Prerequisite: Angle or distance label to move.

Bearing with Leader

This command places the bearing of a line or polyline segment at a point, then plots a user specified leader line to point to the defining line or polyline. There is the ability for multi-segment leaders, and the option to align the label horizontal to the current view or parallel to the linework.

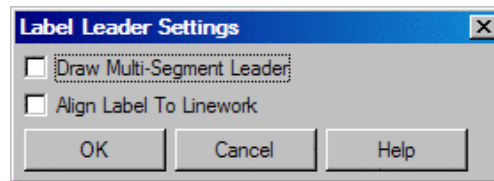
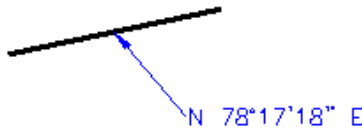
Prompts

Options/Points/⟨Select line or polyline⟩: *select entity*

Pick point to start leader: *pick a point near the entity*

Label Position: *pick a point* Select the point where to place the label.

Options/Points/⟨Select line or polyline⟩: *O*



When Options (O) is chosen

Pulldown Menu Location: Annotate > Annotate with Leader

Keyboard Command: brglead

Prerequisite: None

Distance with Leader

This command labels the distance of a line or polyline segment at a point then draws a user specified leader line to point to the defining line. There is the ability for multi-segment leaders, and the option to align the label horizontal to the current view or parallel to the linework.

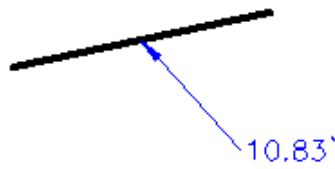
Prompts

Define distance by, Points/⟨Select line or polyline⟩: *select a line*

Pick point to start leader: *pick a point near the line*

Label Position: *pick a point*

Define distance by, Points/⟨select line or polyline⟩: *press Enter to end*



Pulldown Menu Location: Annotate > Annotate with Leader

Keyboard Command: distlead

Prerequisite: None

Bearing-Distance with Leader

This command places the bearing and distance labels of a line or polyline segment at a selected position and draws a leader line to the defining line or polyline.

At the command prompt, type O for Options to bring up the options dialog. There is the ability for multi-segment leaders and the option to align the label horizontal to the current view or parallel to the linework.

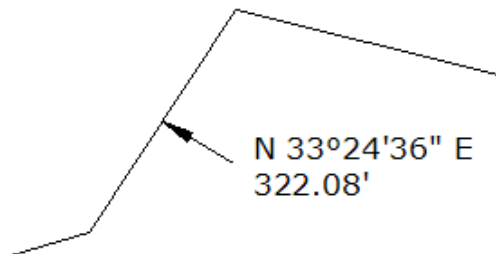
Prompts

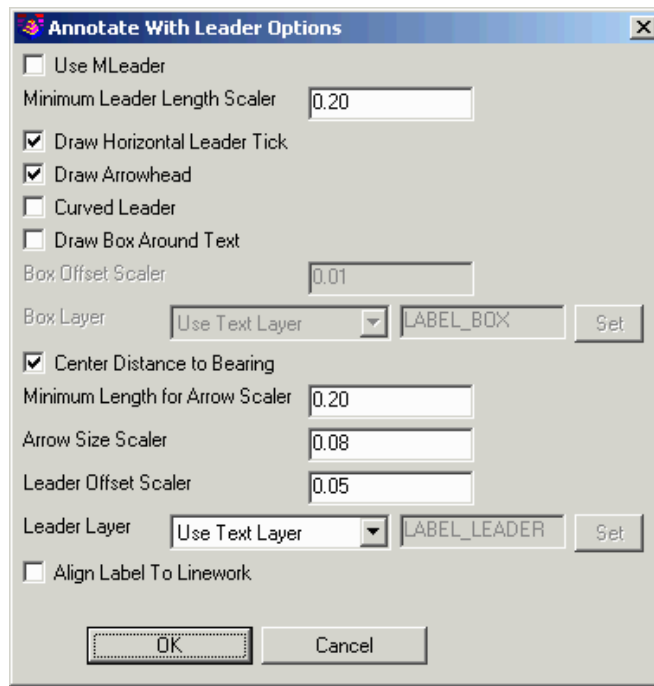
Options/Points/⟨Select line or polyline⟩: *select entity*

Pick point to start leader: *pick a point near the entity*

Label Position: *pick a point* Select the point where to place the label.

Options/Points/⟨Select line or polyline⟩: *O*





Pulldown Menu Location: Annotate > Annotate with Leader

Keyboard Command: bdlead

Prerequisite: None

Distance-Bearing with Leader

This command labels the distance and bearing of a line at the end of a user-specified leader which points to the defining line. The line can be specified by two points or by selecting a line or polyline entity. There is the ability for multi-segment leaders and the option to align the label horizontal to the current view or parallel to the linework.

Prompts

Options/Points/<Select line or polyline>: *select entity*

Pick point to start leader: *pick a point near the entity*

Label Position: *pick a point*Select the point where to place the label.

Pulldown Menu Location: Annotate > Annotate with Leader

Keyboard Command: dblead

Prerequisite: None

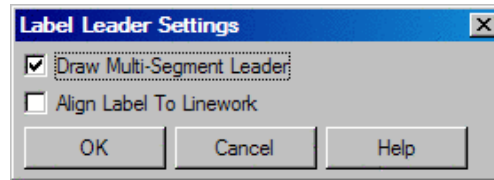
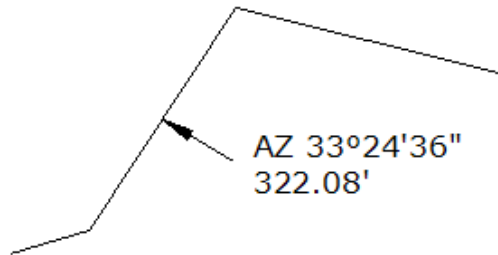
Azimuth-Distance with Leader

This command places the azimuth and distance label of a line or polyline at a point, and then plots a user specified leader line which points to the defining line or polyline. There is the ability for multi-segment leaders and the option to align the label horizontal to the current view or parallel to the linework.

Prompts

Options/Points/<Select line or polyline>: *pick entity*

Pick point to start leader: *pick point*
Label Position: *pick location*
Options/Points/<Select line or polyline>: *O*
Label Leader Settings dialog *make selection*

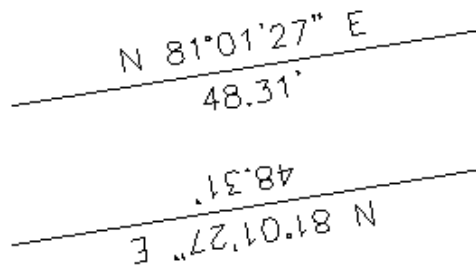


When Options (O) is chosen

Pulldown Menu Location: Annotate > Annotate with Leader
Keyboard Command: azilead
Prerequisite: None

Flip Selected Labels

This command rotates a group of text 180 degrees. Use this command to rotate any text. The command ignores all entities in the selection set except text.



Before and After Flip Labels

Pulldown Menu Location: Annotate > Flip Labels
Keyboard Command: flip_labels
Prerequisite: Text to rotate

Global Reannotate

This command updates bearing and/or azimuth labels for when the lines and polylines associated with the labels have been rotated after the bearings and/or azimuths were labeled.

Prompts

Select One Bearing/Azimuth Text Before Rotation: *pick a bearing or azimuth label*

Pick line associated with old bearing/azimuth: *pick the line or polyline for the selected label*

Select All or specific objects to reannotate (<All/Objects)? *press Enter to update all text*

Pulldown Menu Location: Annotate

Keyboard Command: globalre

Prerequisite: Bearing or azimuth labels and lines or polylines

Survey Text Defaults

This dialog box routine sets up the defaults for the Building Dimensions, Offset Dimensions and Adjoiner Text commands. The Load and Save functions allow you to store and recall the settings using a .svt settings file.

The screenshot shows the 'Survey Text Defaults' dialog box with the following settings:

- Building Dimensions:** Layer: BLDGTX, Text Style: STANDARD, Text Size Scaler: 0.080, Decimal Places: 0.0, Drop Trailing Zeros: unchecked, Use MText: unchecked, Distance in Inches: Off, Characters To Append: (empty), Offset From Line: 0.030, Auto Label Placement: None selected.
- Adjoiner Text:** Layer: ADJNTX, Text Style: Anno, Text Size Scaler: 0.100, Justification: BR, Use MText: unchecked, Label Flip Tolerance (degrees): 0.000.
- Offset Dimension Text:** Layer: DIMENTX, Text Style: STANDARD, Text Size Scaler: 0.040, Arrow Size Scaler: 0.080, Decimal Places: 0.0, Use EndPoint Snap For 1st Point: checked, Drop Trailing Zeros: unchecked, Use MText: unchecked, Distance in Inches: Off, Suppress Dim Extension Lines: unchecked, Characters To Append: (empty), Offset From Line: 0.040, Text Alignment: Horizontal selected, Position: Auto selected, Dimension Line Type: Single Arrow Line selected, Use Current Dimension Style: checked.

Building Dimensions allows you to set text specifications for building dimensions.

Layer: Allows you to set the layer for the building text.

Text Style: Allows you to set the text style for the building text.

Text Size Scaler: This value multiplied by the horizontal scale determines the actual text size.

Decimal Places: Allows you to set the display precision for the building dimensions. The AutoCAD Units option sets the decimals to match the current drawing precision (LUPREC system variable).

Drop Trailing Zeros: Allows you to truncate trailing zeros from dimensions.

Characters To Append: Allows you to set characters to add to reported dimensions.

Offset From Line: Allows you to set the offset distance from the line to the dimension text.

Auto Label Closed Plane allows you to choose between automatically labeling the Interior or Exterior or closed polylines. You may also choose none.

Offset Dimension Text allows you to set text specifications for offset dimensions.

Layer: This option allows you to set the layer for the offset text.

Text Style: This option allows you to set the text style for the offset text.

Text Size Scaler: This value multiplied by the horizontal scale determines the actual text size.

Arrow Size Scaler: This option allows you to set the arrow scaler to determine arrowhead size.

Decimal Places: This option allows you to set the precision for the offset dimensions. The AutoCAD Units option sets the decimals to match the current drawing precision (LUPREC system variable). The Annotation Defaults option uses the distance decimal settings from the Annotation Defaults command including the option to set the decimals by the distance range.

Drop Trailing Zeros: This option allows you to truncate trailing zeros from dimensions.

Distance In Inches: This option allows you to use feet and inches.

Suppress Dim Extension Lines: This option skips drawing extension lines.

Characters To Append: This option allows you to set characters to add to reported dimensions.

Offset From Line: This option allows you to set the offset distance from the line to the dimension text.

Text Alignment allows you to align text either parallel to the line or horizontally in the drawing.

Position allows you to determine if you are to pick the location of the text, or if the text is automatically positioned in the drawing.

Dimension Line Type allows you to determine the line style to use for dimensions.

Single Arrow Line: Draws a line with an arrowhead from the dimension text to the figure.

Dual Arrows Line: Draws dual arrowhead.

Standard Line: Draws a line with no arrowhead from the dimension text to the figure.

Curved Leaders: Draws a curved line with an arrowhead from the dimension text to the figure.

Dimension Only: Draws the dimension text with no line.

Use Current Dimension Style: Chooses whether to use the current dimension style set in the DIMSTYLE command or define a new dimension style using the settings in this dialog.

Adjoiner Text allows you to set text specifications for adjoiner text.

Layer: Allows you to set the layer for the adjoiner text.

Text Style: Allows you to set the text style for the adjoiner text.

Text Size Scaler: Allows you to set the text scaler to determine text size.

Justification: Allows you to set the text justification. See the AutoCAD Reference Manual for details on each justification choice.

Use MText: Chooses between creating MText and regular Text entities.

Pulldown Menu Location: Annotate > Survey Text

Keyboard Command: svtextdf

Prerequisite: None

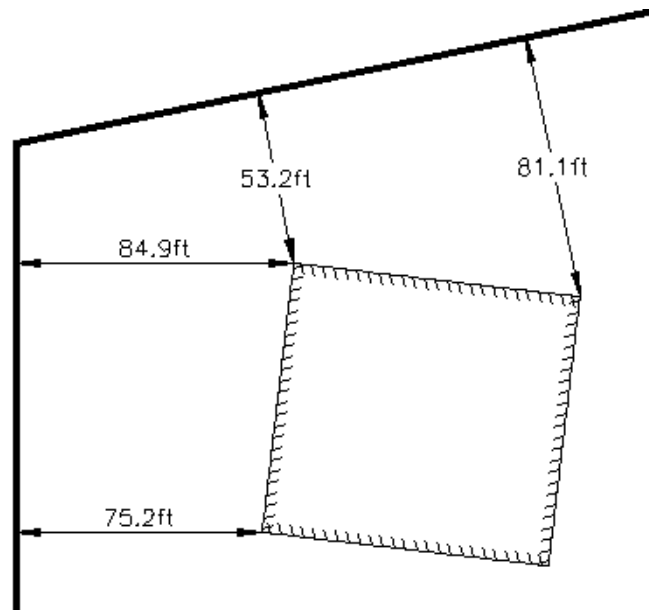
Offset Dimensions

This command labels the perpendicular distance between a point and a line or polyline. The point can be a building corner or other object. The endpoint snap is on by default for picking this point, although you may choose another snap mode manually. There is also an option for arrow only on end of line. The text layer, size, style and the dimensioning method are set in the *Survey Text Defaults* command, found in Settings > Configure > Survey Settings.

Prompts

[end on] **Pick Bldg/Object Corner:** *pick a point*

Pick Line To Offset From: *pick a line or polyline*



Offset Dimensions showing perpendicular distances from corners to property lines

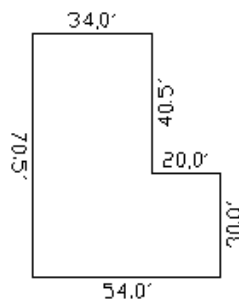
Pulldown Menu Location: Annotate > Survey Text

Keyboard Command: dimentxt

Prerequisite: Line or polyline

Building Dimensions

This command labels the length of line and polyline segments. The label is located in the middle of the line or polyline segment. The options for Building Dimensions are set in the *Survey Text Defaults* dialog. This dialog is found in Settings > Configure > Survey Settings. One option in *Survey Text Defaults* labels all the segments of a closed polyline with one pick of the polyline. Otherwise, the procedure is to pick a line or polyline segment and then choose an alignment. Depending where the alignment point is picked, the label is drawn either perpendicular or parallel, above or below the line.



Prompts

Pick Line or Polyline: *pick line or polyline segment to label*

Pick Alignment: *pick point as shown*

Pulldown Menu Location: Annotate > Survey Text

Keyboard Command: bldgtext

Prerequisite: Line or polyline

Adjoiner Text

This command draws text that is aligned with the selected line or polyline segment. The layer, style, size and justification for the text is set in the *Survey Text Defaults* command, found in Settings > Configure > Survey Settings. To align text that is already drawn, use the *Rotate Text* command found in the Edit menu.

Prompts

Pick Line or Polyline: *pick a line or polyline for alignment*

Starting point: *pick a point to start the text*

Text: *MAIN STREET*



Adjoiner Text aligns text with a line or polyline

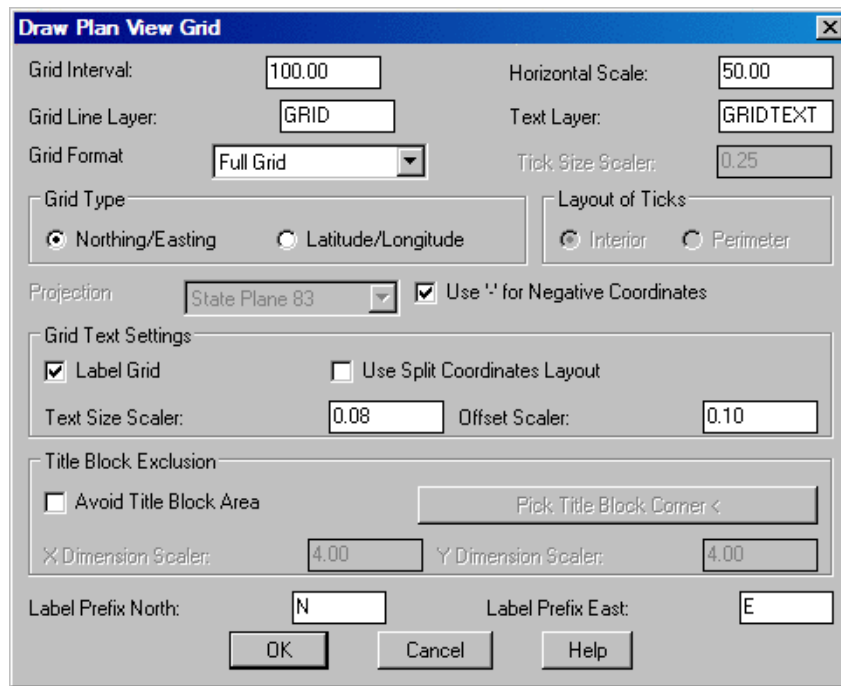
Pulldown Menu Location: Annotate > Survey Text

Keyboard Command: adjntext

Prerequisite: Line or polyline

Draw Grid

This command will plot a plan view grid at a user specified distance and optionally label the northing and easting coordinates of the grid. This command takes in consideration the current screen twist angle in which case it prompts for three corner points. After selecting the corner points the dialog below will appear. The title block is assumed right justified to the lower right corner of the grid definition points. After changing any of the settings select the *OK* button to plot the grid.



Grid Interval: The distance between each grid line.

Horizontal Scale: Reports the scale of the current drawing. This can also be set using the Drawing Setup command in the Settings menu.

Grid Format: The Ticks Only option will draw tick marks instead of grid lines. Selecting the Ticks Only option activates the Tick Size option for sizing the tick marks. There is also a Full Grid and Perimeter option.

Layout of Ticks: This option places the ticks throughout the interior of the grid work or just on the perimeter of the grid boundary.

Use '-' for Negative Coordinates: This option labels the negative grid coordinates with a '-'.

Label Grid: Selecting this Grid Text Setting option labels the grid coordinates.

Use Split Coordinates Layout: Puts the thousands digits above the grid line and the hundreds digits below the grid line

Text Size Scaler: This scaler, multiplied by the Horizontal Scale, determines text size.

Offset Scaler: This scaler, multiplied by the Horizontal Scale, determines the offset for text.

Avoid Title Block Area: This Title Block Exclusion option will allow you to not draw grid lines or tick marks in the title block area. It is for making sure that the grid does not overwrite the title block.

Pick Title Block Corner: This option prompts you to pick the corner of the title block to determine where the grid lines and ticks will be omitted.

X Dimension Scaler: This is the horizontal dimension of the title block. This option is automatically filled in when the Pick Title Block Corner option is selected.

Y Dimension Scaler: This is the vertical dimension of the title block. This option is automatically filled in when the Pick Title Block Corner option is selected.

Label Prefix North: This option is for assigning a prefix to the northing grid line and tick mark coordinates.

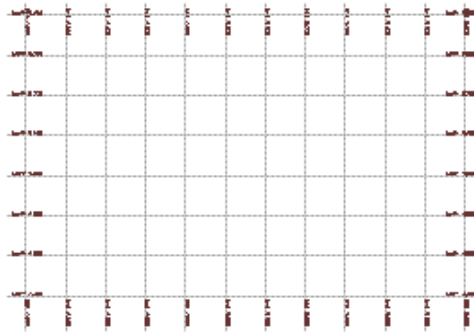
Label Prefix East: This option is for assigning a prefix to the easting grid line and tick mark coordinates.

Prompts

Pick or Type Lower Left Corner Point: *endp of (pick point)*

Pick or Type Upper Right Corner Point: *endp of (pick point)* Select the corners of your border in which you want the grid plotted.

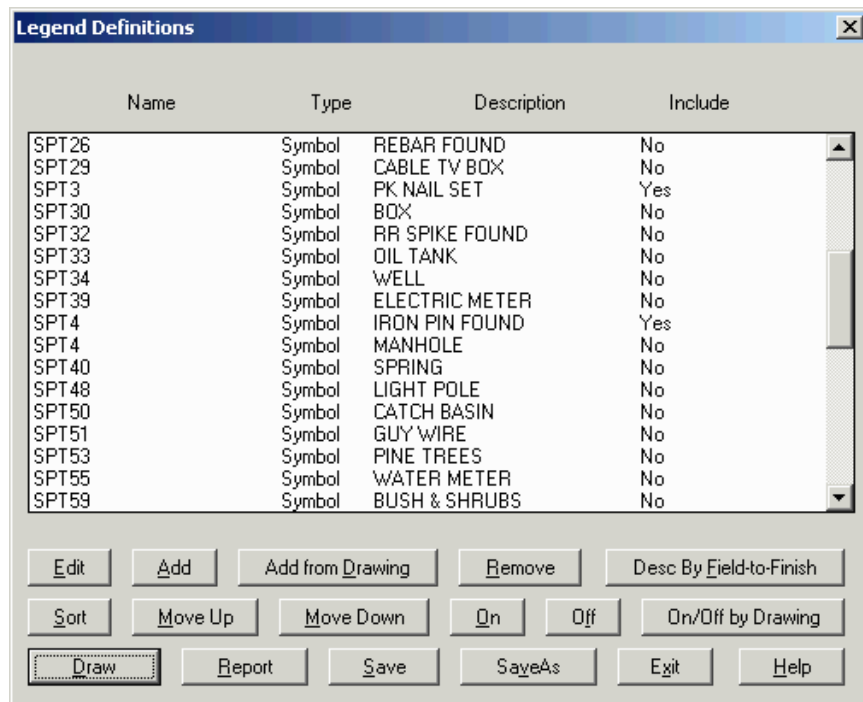
Draw Plan View Grid Dialog



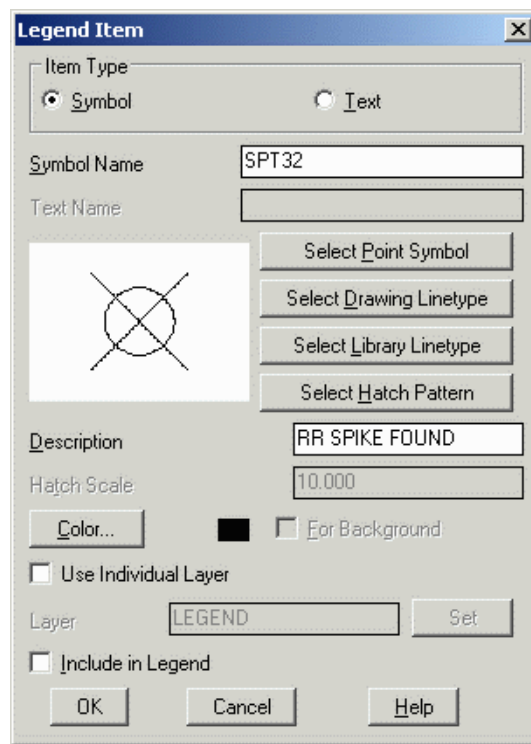
Pulldown Menu Location: Annotate
Keyboard Command: dgrid
Prerequisite: None

Draw Legend

This command draws a legend based on a legend definition file. After choosing the legend definition (.LGD) file to use, a dialog displays the current definitions. The legend definition file consists of descriptions assigned to text, symbols, linetypes and hatch patterns. The default legend that is included with Carlson is called legend.lgd.



Edit edits a definition, select it and then click on the Edit button. This brings up the Symbol Definition dialog box.



- **Item Type:** Each item can be either a simple text label or a symbol from the drawing.
- **Text Name:** This is the legend label associated with the specified Description.
- **Symbol Name** designates the symbol to draw in the legend. You can either type in the symbol name or choose it from a slide library by picking the appropriate Select button.
- **Description** is the name of the symbol.
- **2nd Description** is an optional additional name for the symbol
- The **Hatch Scale and Color** options are used if the symbol uses a hatch pattern.
- **Include in Legend:** This option corresponds to the Include column on the Legend Definitions dialog box. Not all the defined entries need to be drawn. An entry will be drawn (shown as Yes) if the Include in Legend box in the Symbol Definition dialog box is checked.
- **Use Individual Layer:** This option allows for drawing the legend item on a separate layer besides the layer from the Draw function.
- **Select Point Symbol:** This option displays a slide library of point symbols to choose from.
- **Select Drawing Linetype:** This option displays a linetype name list to choose from.
- **Select Library Linetype:** This option displays a slide library of linetypes to choose from.
- **Select Hatch Pattern:** This option displays a slide library of hatch patterns to choose from.

Add inserts a new definition to the definitions. To insert a new definition, pick an existing definition and click on the Add button. The new definition is added immediately following the existing definition.

Add from Drawing adds entries to the legend table for each different symbol that is selected from the drawing.

Remove removes the selected definition.

On switches the Include field in the selected definition to Yes.

Off switches the Include field in the selected definition to No.

On/Off by Drawing prompts you to select symbols from the drawing. Symbols found will be turned on, all others will be turned off. This helps you create a legend that includes only symbols found in the drawing.

Description by Field-to-Finish uses the description from the Field-to-Finish code definition for symbols that match

the code symbol.

Sort sorts the definitions alphabetically and numerically.

Draw draws the included definitions as a legend.

Report uses the Report Formatter to make a customized report of the names and descriptions in the legend.

Move Up: This option moves the selected definition up one row. Legend entries are drawn in the order that they are defined.

Move Down moves the selected definition down one row. Use the Move Up and Move Down buttons to change the order that the symbols will be drawn.

Save saves the legend file as its original file name.

Save As saves the legend file to a new file name.

Exit exits the command back to the drawing window.

Draw opens the Draw Legend dialog.

Draw Legend dialog box showing settings for text size, symbol size, hatch size, line size, layer name, style name, and checkboxes for drawing options like 'Draw 2nd Description', 'Draw Legend Title', 'Draw Header', and 'Draw Grid Lines'. The layout is set to 'Right Justified'.

- **Text Size** sizes the text in the legend. It defaults to the value from Drawing Setup in the Setting menu.
- **Symbol Size** defaults to the value from Drawing Setup in the Settings menu.
- **Hatch Size** sizes the hatch pattern scaler.
- **Line Size** sizes the lines in the legend.
- **Layer Name** defines the layer for the legend.
- **Style Name** sets the style for the legend labels.
- **Draw 2nd Description** creates another column with the 2nd descriptions for the symbols.

- **Draw Legend Title** draws the text from the Name, Title Line 1 and 2 fields.
- **Draw Header** adds a header row with the Symbol, Desc and 2nd Desc fields at the top of the columns.
- **Draw Grid Lines** creates a legend table with lines for the rows and columns

- **Layout Left Justified** has the symbol on the left and then the labels left-justified.
- **Layout Right Justified** has the symbols on the left and the labels right-justified.

- **Layout Right Symbol** has the labels on the left and the symbols on the right.

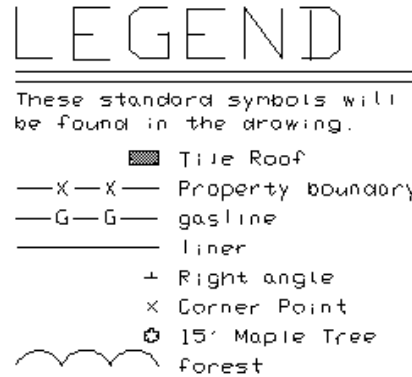
Prompts

Specify Legend Definition File Dialog *choose the file to process*

Legend Definitions Dialog

Draw Legend Dialog

Enter or pick upper left point for legend: *pick a point*



Sample legend created by Draw Legend

Pulldown Menu Location: Annotate

Keyboard Command: legend

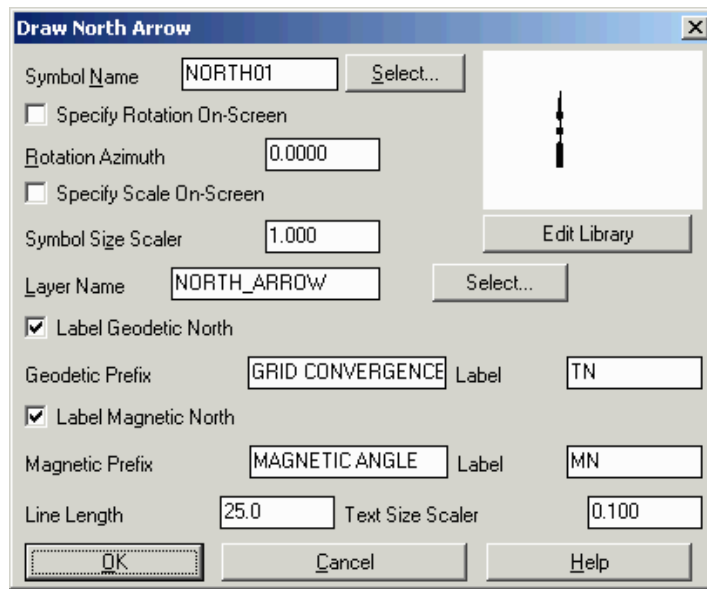
Prerequisite: None

Draw North Arrow

This command inserts a north arrow symbol. You can select from several styles of arrows, and you can add your own by using the Edit Library button which is similar to the Symbols Library command. The north arrow symbol library is stored in the narrow.dta file in the USER folder.

The Label Geodetic North option draws a line for geodetic north and labels the convergence angle between grid and geodetic north.

The Label Magnetic North option draws a line for magnetic north and labels the angle.



Prompts

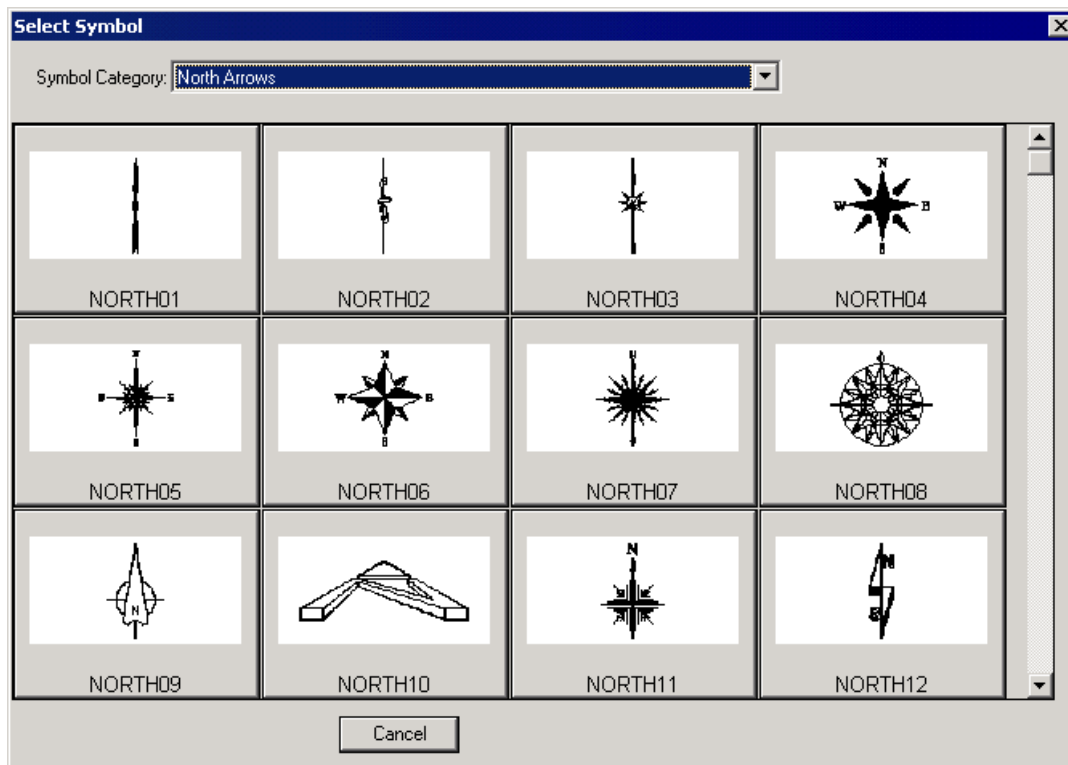
Draw North Arrow Dialog *choose an arrow symbol, layer and other variables*

Specify insertion point: *pick a point*

X scale factor <1> / Corner / XYZ: *press Enter*

Y scale factor (default=X): *press Enter*

Rotation angle <0d0'0">: *press Enter*



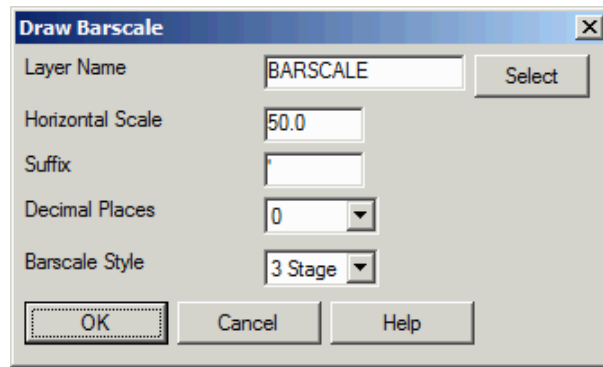
Pulldown Menu Location: Annotate

Keyboard Command: narrow

Prerequisite: None

Draw Barscale

This command draws a barscale at the user-specified scale. The command options are set in the dialog shown here. The Horizontal Scale controls the size and labels for the barscale. For example, enter 50 for 1 inch = 50 feet in English mode. The Barscale Style chooses between different barscale formats.



Prompts

Draw Barscale options dialog

Pick location for barscale: *pick a point*

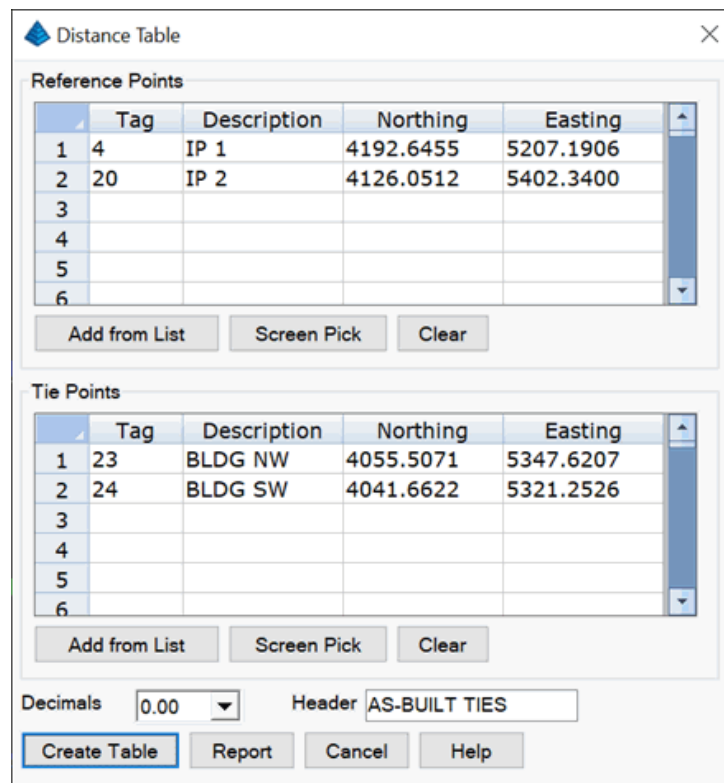
Pulldown Menu Location: Annotate

Keyboard Command: barscale

Prerequisite: None

Distance Table

This command creates a distance table for all the combinations of distances between the Reference Points and Tie Points. The points are defined in the dialog by filling out the spreadsheets. Use the Add From List to select a point from the current coordinate file. Use Screen Pick to pick a position from the drawing. The Create Table function draws the table. Use the Report to use the Report Formatter to make a custom report.



AS-BUILT TIES		IP 1	IP 2
Point	Description	4	20
23	BLDG NW	196.28	89.28
24	BLDG SW	189.22	117.03

AS-BUILT TIES		IP 1	IP 2
Point	Description	4	20
23	BLDG NW	196.28	89.28
24	BLDG SW	189.22	117.03

Pulldown Menu Location: Annotate

Keyboard Command: distable

Prerequisite: None

Create Point Table

This command draws a table of the coordinate data of the points from the current coordinate (.CRD) file using different methods displayed at the top of the dialog. The command displays the dialog shown below for setting all of the point table options. At the top of the dialog enter the range of point numbers to label, do a Screen Pick or select a Point Group(s). You can also specify the order and format of the table columns. If you do

not want to include a data type, set the Sequence number to blank. The Northing/Easting Format can be set to Degree/Minutes/Seconds for when the coordinate file contains latitude and longitude. The Max Rows Per Column setting makes the program start a new table when the specified max points is reached. The Use Table Entity option create a Carlson Table entity which has more formatting options and can be adjusted with the Edit > Table functions.

Prompts

Point Table Generator Dialog

Building Data List ... Done.

Table Upper Left Corner: *pick a point*

Generating Table... Done.

	Seq#	Heading	Width	Align	Prec
Point Num	1	Point	8	R	
Northing	2	Northing	12	R	3
Easting	3	Easting	12	R	3
Elevation	4	Elevation	8	R	2
Description	5	Description	30	L	

Northing/Easting Format: Decimals Max Rows Per Column: 1000

General Settings:

Table Layer: PNT_TABLE Select Interline Scaler: 1.71

Text Style: STANDARD Select Space Between Columns: 2

Grid Layer: PNT_TBL_GF Select Column Scale Factor: 1.0

Text Size Scaler: 0.10 Draw Column Headings Draw Grid Lines

Header: Footer:

Buttons: OK, Cancel, Help

POINT	NORTHING	EASTING	ELEVATION	DESCRIPTION
10	4837.185	4938.546	9.57	17
11	4814.573	4926.339	10.09	17
12	4768.075	4942.627	9.71	17
13	4720.630	4943.015	10.39	17
14	4672.472	4943.661	11.02	17
15	4640.627	4935.524	11.57	17
16	4608.962	4915.995	11.93	17
17	4577.455	4880.778	12.95	tp
18	4504.613	4821.334	13.78	17
19	4436.627	4755.157	15.09	17
20	4370.285	4693.938	16.34	17

Typical Point Table

Pulldown Menu Location: Annotate > Point Table

Keyboard Command: pointtbl

Prerequisite: A coordinate (.CRD) file

Update Point Table

This command prompts you to select an existing point table. The program then reads the settings from this table and displays these settings in the same dialog used in *Create Point Table*. You can change any of the table format options. The program will also update the table to reflect any changes to the coordinate (.CRD) file.

Prompts

Select existing point table: *pick anywhere on the point table or select points from the screen*

Point Table Generator Dialog

Pulldown Menu Location: Annotate > Point Table

Keyboard Command: pointtblupd

Prerequisite: An existing point table, .CRD file

Point To Point Table

This command creates a course table of the angle and distance between pairs of points. The table data is entered in a spreadsheet. Fill in the From and To point numbers. The program reads the coordinates for the points from the current coordinate file and fills out the Angle and Distance. There are settings for the Angle Format as either Bearing or Azimuth, Decimals and Suffix for the distance, and Header labels for the table.

The Add From List button selects points from the current coordinate file. The Screen Pick button prompts to pick the points from the drawing.

Once the data is entered, use either the Create Table or Report buttons. The Create Table function draws the table in the drawing. The Report function uses the Report Formatter to create a user-specified report.

	From Point	To Point	Angle	Distance
1	17	20	S 50°38'52" W	180.225
2	19	21	S 42°15'18" W	100.559
3	48	49	N 42°10'20" W	57.474
4	102	106	S 35°03'37" E	205.985
5				
6				
7				
8				
9				
10				
11				
12				
13				
14				

Angle Format: Bearing Azimuth

Decimals: 0.00

Distance Suffix:

Headers: From [From] To [To] Angle [Direction] Distance [Distance]

Buttons: Create Table, Report, Cancel, Help

Course Table			
From	To	Direction	Distance
17	20	S 50°38'52" W	180.23
19	21	S 42°15'18" W	100.56
48	49	N 42°10'20" W	57.47
102	106	S 35°03'37" E	205.99

Pulldown Menu Location: Annotate

Keyboard Command: p2ptable

Prerequisite: Points in current coordinate file

Table Defaults

This command sets the format for line and curve tables. Line and curve tables are commonly used in situations where:

1. The amount of line/curve annotation in the drawing itself makes the drawing look too "cluttered," and/or
2. The length(s) of the line(s)/curve(s) are too short for the annotation label being placed

You specify the label and table attributes in the Line/Curve Table Defaults dialog:

Combine Line and Curve Tables: This option makes a single table with both line and curve data. When this option is active, the curve settings govern and the line settings aren't used. For line segments, the chord length and chord bearing fields are used for the line length and bearing. Typically the chord length and chord bearing would be set to Sequence numbers 1 and 2. Also, under Curve Table Labels you can set the headers for curve number to "ID", chord length to "Length" and chord bearing to "Bearing" to make the headers apply for both lines and curves.

ID	LENGTH	BEARING	RADIUS	ARC LENGTH	DELTA ANGLE
C1	163.32'	N 66°32'02" E			
C2	142.12'	S 78°41'24" E	124.59'	151.24'	69°33'07"
C3	209.36'	N 77°30'20" W	189.20'	221.85'	67°11'00"
C4	120.27'	N 51°07'27" E			

Label Text Layer: Click the Set button or specify the layer of the annotation which is applied to the line/curve itself.

Label Text Style: Click the Set button or specify the text style of the annotation which is applied to the line/curve itself.

Label Text Size: Specify the text size of the annotation which is applied to the line/curve itself.

Line Label Prefix: Specify a prefix which should be inserted prior to each line number. The prefix can be an alpha-numeric string.

Line Table Title: Specify a caption for the line table.

Table Text Layer: Click the Set button or specify the layer of the annotation which is inserted to the line/curve table.

Table Text Style: Click the Set button or specify the text style of the annotation which is inserted to the line/curve table.

Table Text Size: Specify the text size of the annotation which is inserted to the line/curve table.

Row Height Factor: Indicate a positive, non-zero multiplier of the Table Text Size to help adjust spacing for each row in the table.

Curve Label Prefix: Specify a prefix which should be inserted prior to each curve number. The prefix can be an alpha-numeric string.

Curve Table Title: Specify a caption for the curve table.

Set Line Table Labels: See the expanded Set Line Table Labels section below.

Set Curve Table Labels: See the expanded Set Curve Table Labels section below.

Draw Text Only: Creates a table as MText without drawing grid lines.

Prompt for Label Location: When enabled, this option prompts you to pick the location for the label placed onto the line/curve itself. If this is disabled, the location is chosen automatically.

Label Symbol: Select a geometric shape that is placed around the label that is applied to the line/curve itself.

Distance Format: Indicate how distances for the lines are reported:

- **Horizontal:** The distance displayed is only the horizontal distance, even if the selected entity has different "Z" values at either end of the line.
- **Slope:** The distance measured is the slope distance, used mostly for 3D polylines to get their true length.
- **None:** For a table of angles only.

Total Distances: Adds a row at the bottom of the line table to label the total distance for the table.

Label Angles in: Indicate how the line direction is labeled:

- **Azimuth:** The angles are reported as azimuths.
- **Bearings:** The angles are reported as bearings.
- **Gons:** The angles are reported as gons.
- **None:** For a table of distances only.

Automatic Table Update: Indicate if labels in the table should be re-sequenced:

- **On:** This option renumbers the other table entries and the associated labels in the drawing if a new (but previously used number) is specified for the table. For example, if a line table contained lines #1-5 and a line

#4 was added, the new line #4 would be inserted into the table and the previous lines #4 and #5 would be updated to #5 and #6. The L4 and L5 labels on the lines would also be updated to L5 and L6.

- **Off:** You must manually pick the entry location and update the labels.

Label Alignment: Indicate the method by which the label is oriented on the line/curve itself:

- **Horizontal:** This option places the label horizontal to the current screen alignment, as defined by the various *Twist Screen* commands (Standard, Line, Polyline or Text, Surveyor or Restore Due North).
- **Parallel:** This option will orient the label parallel to the line or curve chord.

Use Table Entity: When enabled, Line and Curve Tables can be further manipulated with the Split Table, Merge Tables and Edit Table Values commands. The **Row Height Factor** controls the height of the table rows.

Combine Equal Rows: When enabled, lines or curves that share identical geometry with other lines and curves can assume the number of the equivalent line/curve. As an example, if a line *100 feet* long on a bearing of *N 90d00'00" E* is assigned a label of *L3* and additional lines with this geometry are labeled, you will have the option of re-using the *L3* label for these additional lines. In other words, a single label reference in the table can correlate to many identical entities in the drawing and can keep the overall length of the line/curve table to a minimum.

Append First Table Item To Line/Curve Label: When enabled, an additional reference item from the Line/Curve Table will be placed alongside the label number assigned to the line/curve itself.

Curve Options: Indicate the order in which curve labels shall be inserted into the curve table. Entries left blank (empty) will not be listed in the curve table.

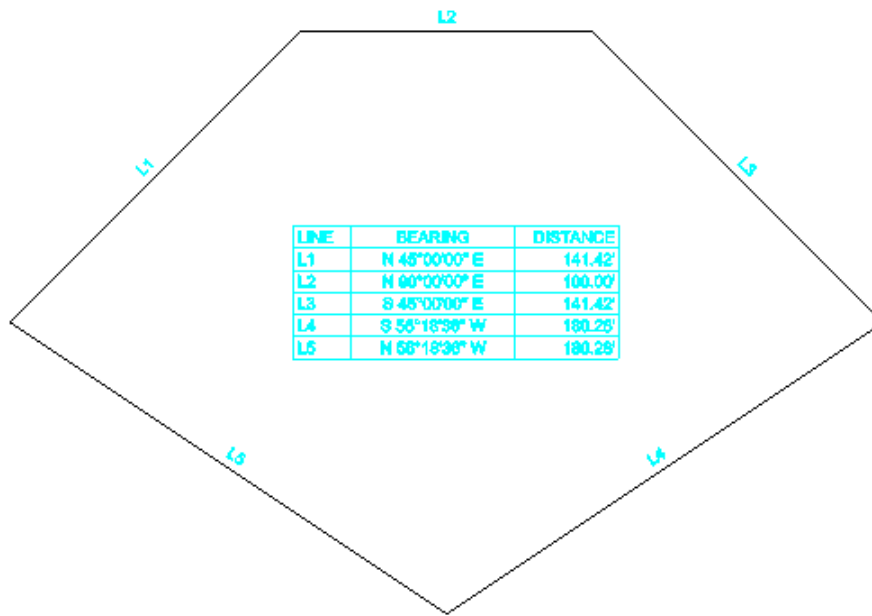
Load: Loads a previously saved collection of Line/Curve Table Default values (*.LCT) into memory.

Save: Saves the current Line/Curve Table Default values to a *.LCT file.

Selecting the **Set Line Table Labels** option allows you to control the label, column width, text justification and displayed precision for the options selected in the Line Table Distance and Label Angles In controls.

Field	Label	Scaled Label	Width	Justification	Precision
Line Number	LINE		6.250	LEFT	
Bearing	BEARING		14.000	LEFT	
Azimuth	AZIMUTH		14.000	LEFT	
GDN	GONS		14.000	LEFT	0.00
Horiz Distance	DISTANCE	DISTANCE2	11.500	RIGHT	
Slope Distance	SLOPE DIST	SLOPE DIST2	11.500	RIGHT	0.00

With the above settings, you might find the Line Table more aesthetically pleasing as it produces the following example:

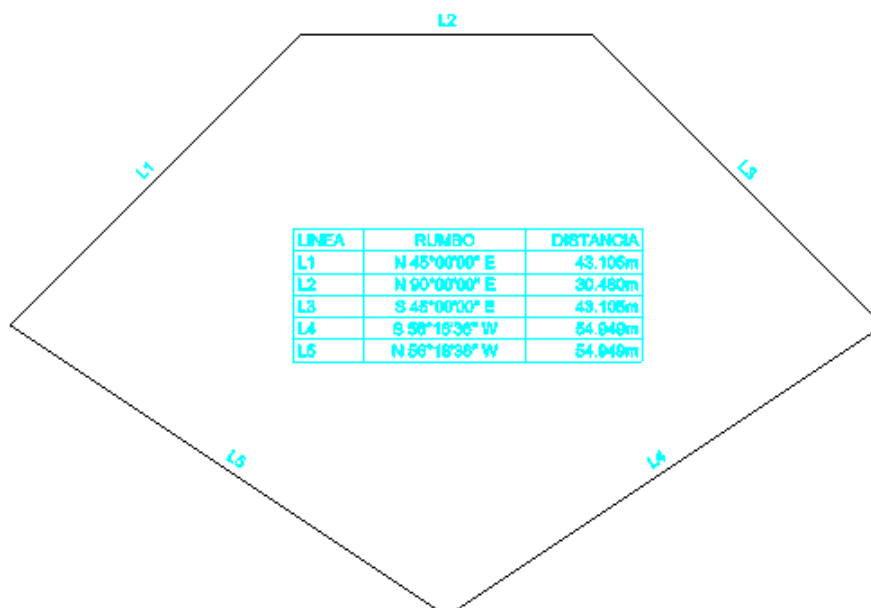


The prefix flexibility and the fact that the text used for the column header can be changed, means that line and curve tables can be plotted in any language. For example, in Puerto Rico survey plats are typically submitted in bearings, in meters and in Spanish. For that location, the table could be reconfigured as shown here:

Field	Label	Scaled Label	Width	Justification	Precision
Line Number	LINEA		6.250	LEFT	
Bearing	RUMBO		14.000	LEFT	
Azimuth	AZIMUTH		14.000	LEFT	
GON	GONS		14.000	LEFT	0.00
Horiz Distance	DISTANCIA	DISTANCE2	11.500	RIGHT	
Slope Distance	SLOPE DIST	SLOPE DIST2	11.500	RIGHT	0.00

Buttons: OK, Cancel, Help

This would lead to the following line table (see the Notes section below for additional information):



Essentially identical to the *Set Line Table Labels* command, the **Set Curve Table Labels** command allows you to control the label, column width, text justification and displayed precision for the options selected in the Curve Options control.

Field	Label	Scaled Label	Width	Justification	Precision
Curve Number	CURVE		8.000	LEFT	
Radius	RADIUS	RADIUS2	9.000	LEFT	0.00
Arc Length	ARC LENGTH	ARC LENGTH2	14.000	LEFT	0.00
Chord Length	CHORD LENGTH	CHORD LENGTH2	14.000	LEFT	0.00
Chord Bearing	CHORD BEARING		15.000	LEFT	
Chord Azimuth	CHORD AZIMUTH		15.000	LEFT	
Chord GONS	CHORD GONS		15.000	LEFT	
Delta Angle	DELTA.ANGLE	TANGENT2	13.000	LEFT	0.00
Tangent	TANGENT		10.000	LEFT	
Degree Of Curve	DEGREE OF CURVE		17.000	LEFT	
Radial Bearing-in	RADIAL BEARING-IN		19.000	LEFT	
Radial Bearing-out	RADIAL BEARING-OUT		19.000	LEFT	

Note:

- Changing the distance suffix to "m" (or omitting any suffix by making it blank) is accomplished in the more general command of Annotate Defaults.
- Reporting distance units in a unit of measure different from that of the current project is accomplished via the Drawing Setup > Report Distance Scale Factor option and the Annotate Defaults command.
- Physical changes to the lines/curves will trigger label updates if the **Link Labels with Linework** option (if available) is enabled under Carlson Configure > General Settings.

Pulldown Menu Location(s): Annotate > Line/Curve Table

Keyboard Command: tdef

Prerequisite: None

Table Header

This command draws the column header labels for the *Curve Table* and *Line Table* commands. When prompted for the starting point, the user may enter a coordinate or pick a point on the screen. The starting point location that the curve or line table command defaults to is one row below the start of the header labels.

CURVE	RADIUS	ARC LENGTH	CHORD LENGTH	CHORD AZIMUTH	DELTA ANGLE
-------	--------	------------	--------------	---------------	-------------

Curve table header (C option)

Prompts

Type of table [Line/<Curve>]? C

Starting point of Curve table text <(6585.0 -704.0 0.0)>: pick point

Pulldown Menu Location: Annotate > Line/Curve Table

Keyboard Command: tabhead

Prerequisite: None

Set Table Position

This command sets the position for adding line table entries. The next line table rows will start from this point. To add to an existing table, pick a point at the lower left of the existing table.

Pulldown Menu Location: Annotate > Line/Curve Table

Keyboard Command: set.tbl

Prerequisite: None

Curve Table

This command will compute curve data and draw it in tabular form using the settings specified in *Table Defaults*. The program computes the curve data from an arc entity, an arc segment of a polyline or from specified points on an arc. The curve data includes radius, length of curve, chord length, chord bearing, tangent and delta or included angle. The current curve table numbers are remembered between drawings. The user is prompted for curve number (default is sequential starting with 1) and the starting point of the table. The curve is labeled with a C#, where # is the sequential curve number. After picking the starting point of the table, the placement point for the other table entries will default to the next position and you can just press Enter unless you want to relocate the table. The Auto Annotate command can also create curve tables. Use the *Table Header* command to draw the column header of the curve data names.

Prompts

Define arc by, Points/<Select arc or polyline>: *pick an arc*

Enter curve number <1>: *press Enter*

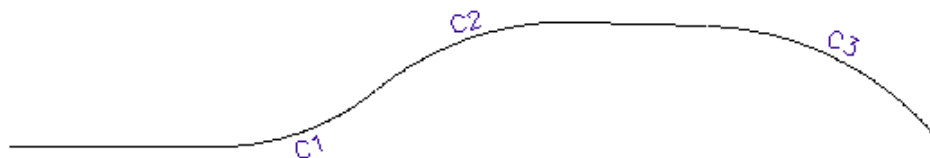
Starting point of curve table text <(5000,5000)>: *pick a point in a clear area of the drawing*

Define arc by, Points/<Select arc or polyline>: *pick another arc*

Enter curve number <2>: *press Enter*

Starting point of curve table text <(4030,4490)>: *press Enter to use next position*

Define arc by, Points/<Select arc or polyline>: *press Enter to end*



CURVE	RADIUS	ARC LENGTH	CHORD LENGTH	CHORD BEARING	DELTA ANGLE	TANGENT
C1	89.44'	60.73'	59.57'	S 70°32'53" W	38°54'14"	31.59'
C2	107.82'	76.59'	74.99'	N 71°26'44" E	40°41'57"	39.99'
C3	110.31'	94.29'	91.45'	S 63°43'00" E	48°58'35"	50.24'

Pulldown Menu Location: Annotate > Line/Curve Table

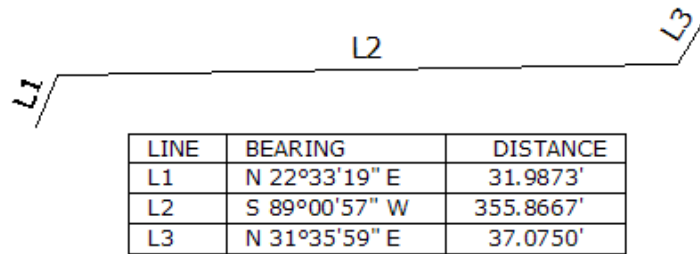
Keyboard Command: arctabl

Prerequisite: None

Line Table

This command will compute line data and draw it in tabular form, using the settings specified in *Table Defaults*. The program computes the bearing and distance from a line, polyline segment or between points. The current line table numbers are remembered between drawings. The line is labeled with a L#, where # is the sequential number of the line picked. The bearing and distance will then be drawn in tabular form similar to the previous Curve Table

command. The Auto Annotate command can also create line tables. Use the Table Header command to draw the column header of the line data names.



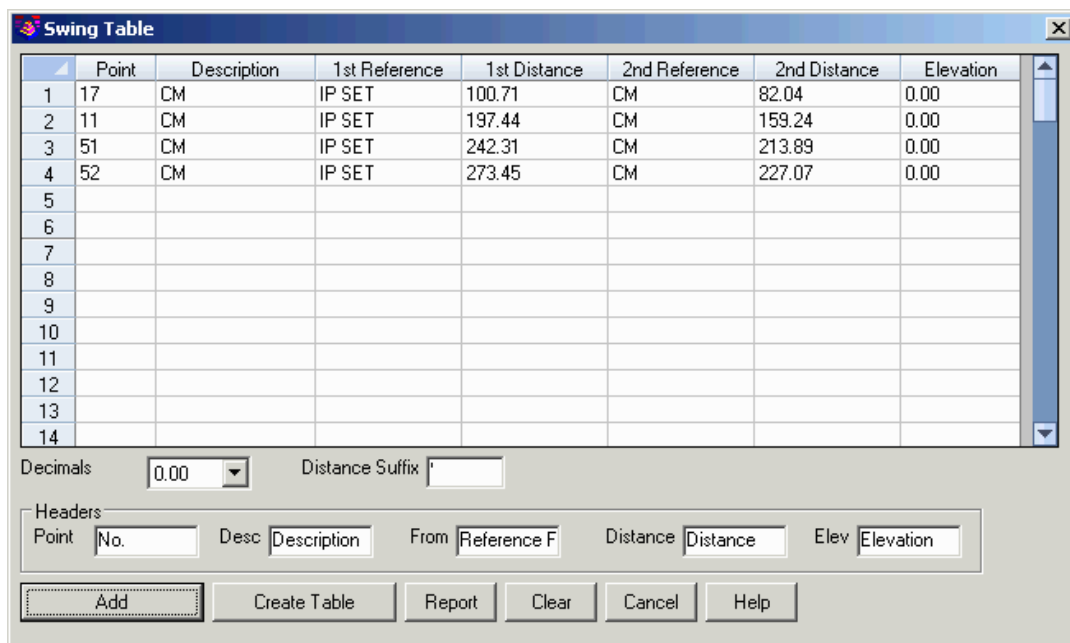
Pulldown Menu Location: Annotate > Line/Curve Table

Keyboard Command: linetabl

Prerequisite: None

Swing Table

This command creates a swing table. From two reference points, the table reports the two distances to target points. Use the Add function to fill in the spreadsheet. The Add function prompts for two reference points and then the target points. You can also edit the spreadsheet directly. When the data is ready, use the Create Table function to draw the table. Use the Report to use the Report Formatter to make a custom report.



No.	Description	Reference From	Distance	Reference From	Distance	Elevation
17	CM	IP SET	100.71	CM	82.04	0.00
11	CM	IP SET	197.44	CM	159.24	0.00
51	CM	IP SET	242.31	CM	213.89	0.00
52	CM	IP SET	273.45	CM	227.07	0.00

Prompts

Reference Point 1 (pick point or point number): *pick a point*

Reference Point 1 Name <A>: *IP SET*

Reference Point 2 (pick point or point number): *pick a point*

Reference Point 2 Name : *CM*

Target Point (pick point or point number): *pick a point*

Target Name <5>: *press Enter*

Target Description: *press Enter*

Target Point (pick point or point number or Enter to end): *press Enter*

Pulldown Menu Location: Annotate

Keyboard Command: swingtbl

Prerequisite: None

Railroad Curve Table

This command is exactly like Curve Table, except the curve data is calculated for Railroad curves. See the Curve Table command for more details.

Pulldown Menu Location: Annotate > Line/Curve Table

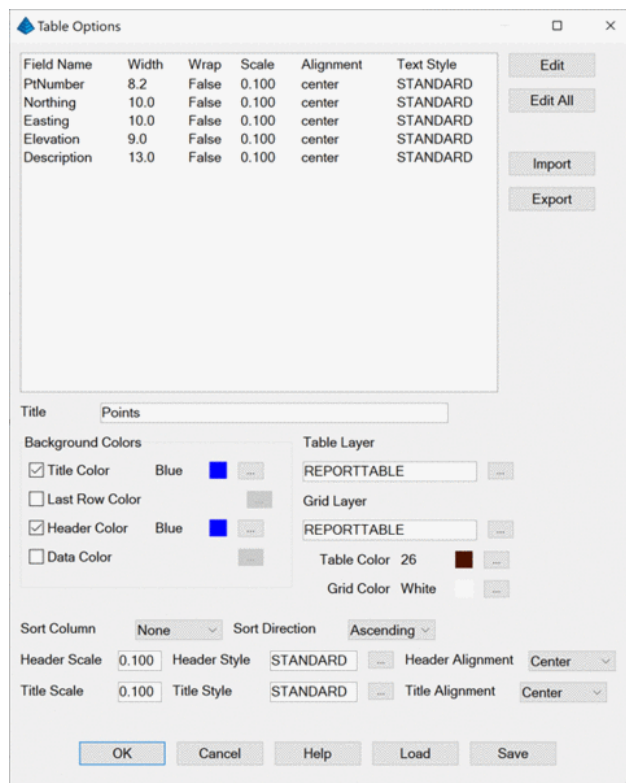
Keyboard Command: rr_curvetbl

Prerequisite: None

Edit Table Properties

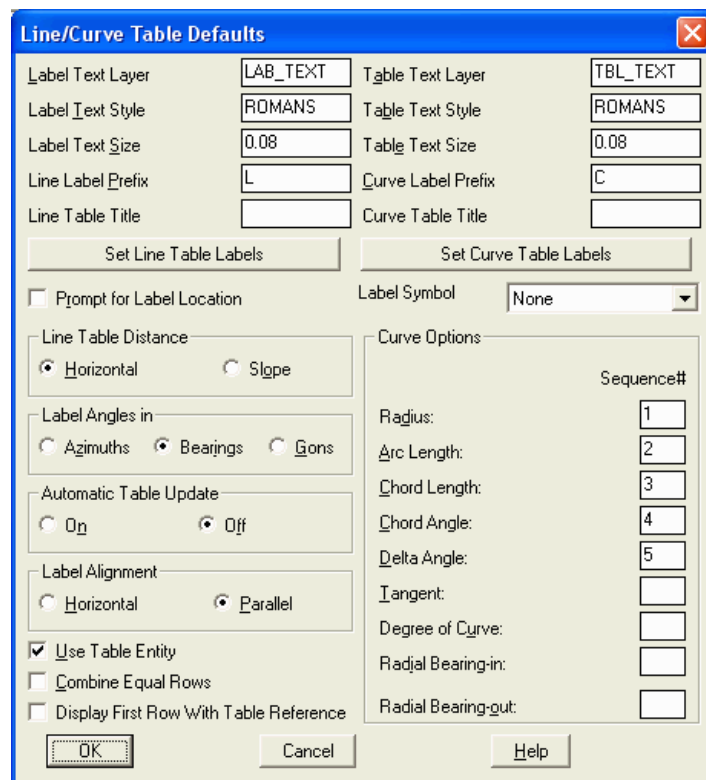
This command allows you to edit the properties of an entity based table.

For general tables, the Edit Table Options dialog is shown. In the dialog, you can change the header labels, width, word wrap, size, alignment and style. The text size is set by the Scale multiplied by the Horizontal Scale from Drawing Setup. The Wrap option automatically makes multi-line rows for long text strings. There are settings for the table colors, title and layer.



For line and curve tables, the Line/Curve Table Defaults dialog will be displayed. Here you can change the settings of the selected table. Change the settings for either line or curve tables depending upon the type of table selected. The changes will be reflected once the user selects the OK button.

Text Layer and Text Style determine the layer and style of the line/curve table text. The distance for line tables can be labeled in horizontal or slope distance. The Automatic Table Update option will automatically insert the entry into the line or curve table. The auto update will renumber the other table entries and the associated labels in the drawing. For example, if a line table had lines #1-5 and a line #4 was added, then the new line #4 would be inserted into the table and the previous lines #4 and #5 would be updated to #5 and #6. The L4 and L5 labels on the lines would also be updated to L5 and L6. Without the automatic update option, the entry location must be picked and the labels updated manually. The Label Alignment determines the orientation of the L# or C# that is labeled on the line or curve. Horizontal will make the label horizontal to the current twist screen and Parallel will draw the label parallel with the line or curve chord. The Line and Curve Label Prefix sets the text before the number that is drawn in the table and on the line or curve (i.e. "L3" or "Line3"). The Curve Options specifies which curve data to include in the table and the order. You specify the label and table attributes in the Line/Curve Table Defaults dialog.



Label Text Layer: determines the layer of the line/curve text.

Label Text Style: determines the style of the line/curve text.

Label Text Size: determines the size of the line/curve text.

Line Label Prefix: determines the prefix for each line.

Line Table Title: draws a title row at the top of the line table with this string.

Table Text Layer: determines the layer of the line/curve table text.

Table Text Style: determines the style of the line/curve table text.

Table Text Size: determines the size of the line/curve table text.

Curve Label Prefix: determines the prefix for each curve.

Curve Table Title: draws a title row at the top of the curve table with this string.

Prompt for Label Location: prompts you to pick the location to label each line or curve. If this is not selected, the location is chosen automatically.

Under **Line Table Distance**, the method for measuring distance is specified.

Horizontal: The distance measured is only horizontal, even if the line is a 3D polyline.

Slope: The distance measured is the slope distance, used mostly for 3D polylines to get their true length.

Under **Label Angles in**, the type of angle is selected.

Azimuths: The angles are reported as azimuth.

Bearings: The angles are reported as bearings.

Gons: The angles are reported as gons.

Under **Automatic Table Update**, the option automatically inserts the entry into the line or curve table. The auto update rennumbers the other table entries and the associated labels in the drawing. For example, if a line table contained lines #1-5 and a line #4 was added, then the new line #4 would be inserted into the table, and the previous lines #4 and #5 would be updated to #5 and #6. The L4 and L5 labels on the lines would also be updated to L5 and L6. If you set the Automatic Table Update to Off, you must manually pick the entry location and update the labels. If Automatic Table Update is set to On, the table is updated automatically whenever the line is modified.

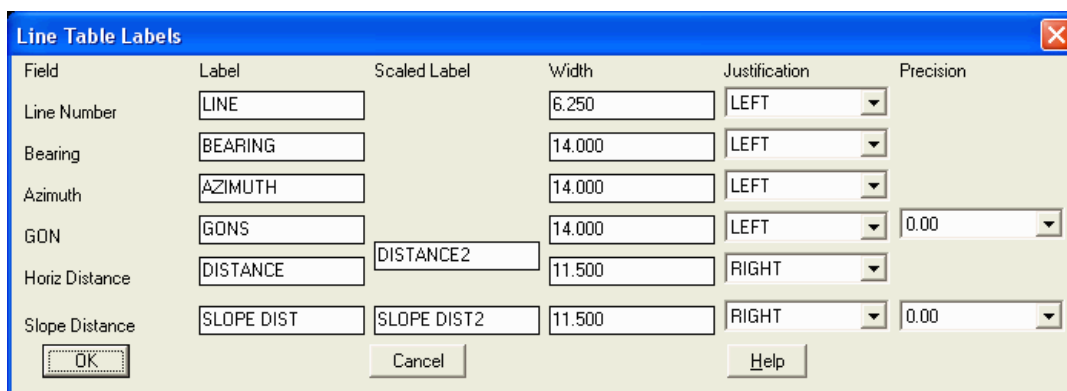
Label Alignment determines the orientation of the L# or C# that is labeled on the line or curve. Horizontal will make the label horizontal to the current screen alignment, Parallel will draw the label parallel to the line or curve chord. Under Curve Options, you specify which curve data to include in the table and the order.

Use Table Entity: will use single block for the whole table. Otherwise, each row is a separate block.

Combine Equal Rows: will use the same line or curve number when the data exactly matches an existing row in the table. For example, if two line segments have the same bearing and distance, then they would both get the same line# (ie. "L5").

Display First Row With Table Reference: When there is room on the line or arc, this option will label both the number and the first column data value from the table on the line or arc. For example, if the first curve table column is for radius and the arc length is big enough to fit the label, then the program would label both the curve # and the radius (ie. "C5 R=100.0").

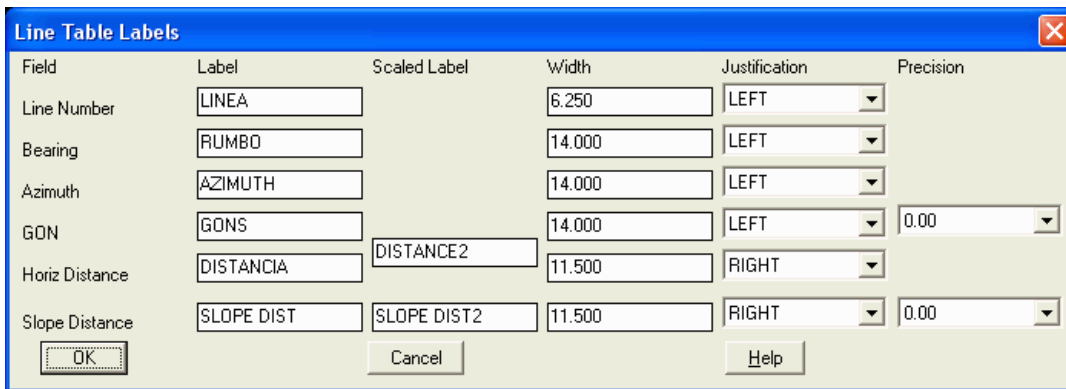
Selecting "Set Line Table Labels" will lead you to the Line Table controls, as "Set Curve Table Labels" (see graphic at end of this command page) leads to the Curve Table controls. For fields that apply to the Report Scale Factor from Drawing Setup, there is a second Scaled Label name for the table header. This scale factor can be used for reporting both grid and ground or both english and metric distances. The options in "Set Line Table Labels" are shown below:



With the above settings, for example, the Line Table appears as shown below. For improved "aesthetics", you might prefer to change the Bearing justification to "Center", for example.

LINE	BEARING	DISTANCE
L1	S 58°18'48" W	87.33'
L2	S 75°06'27" E	148.57'
L3	N 88°27'07" E	63.44'
L4	N 58°40'01" W	63.44'

To save space, you can reduce the size of the "Distance" column from 11.5 to 10. Note that using the Line Label Prefix option, L1 and L2, for example, can read Line1 and Line2, and for that, you may want to expand the "Width" setting for the Line column. The prefix flexibility, and the fact that the text used for the column header can be changed, means that line tables and curve tables can be plotted in any language. In Puerto Rico, for example, surveys are typically conducted in bearings, in meters and in Spanish. For that location, the table could be reconfigured as shown here:

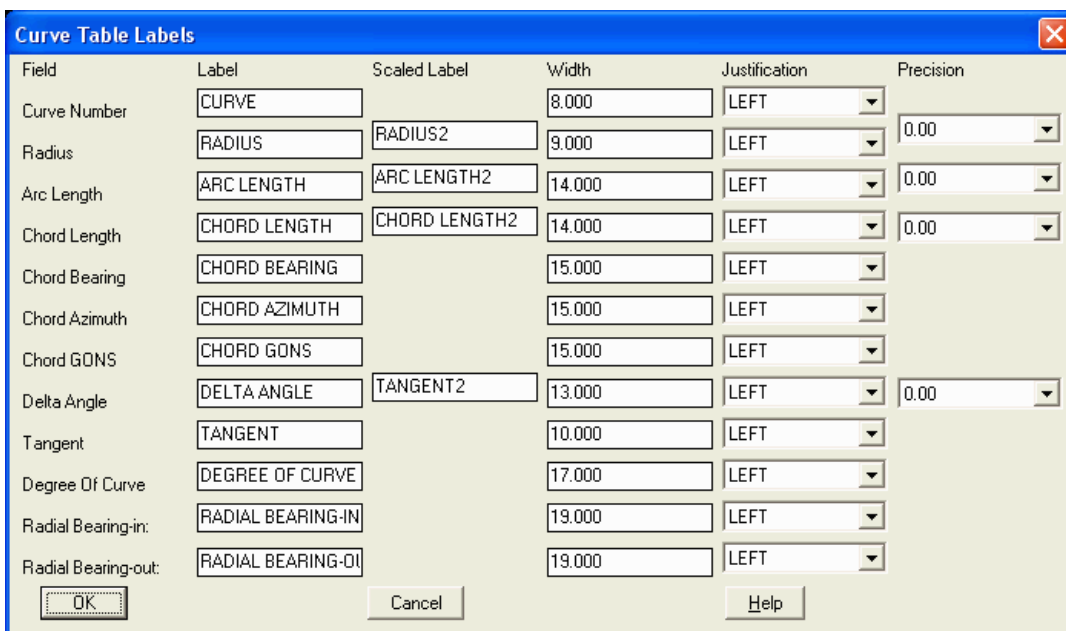


This would lead to the following line table:

LINEA	RUMBO	DISTANCIA
L1	S 58°18'48" W	87.33m
L2	N 88°27'07" E	63.44m
L3	N 58°40'01" W	63.44m

Note that changing the distance suffix to "m" (or omitting any suffix by making it blank) is accomplished in the more general command Annotation Defaults.

Finally, below we see the dialog that appears when you choose Set Curve Table Labels.



Prompts

Select a line or curve table to modify: *pick an entity based line or curve table.*

Pulldown Menu Location: Annotate > Line/Curve Table, and Edit > Table

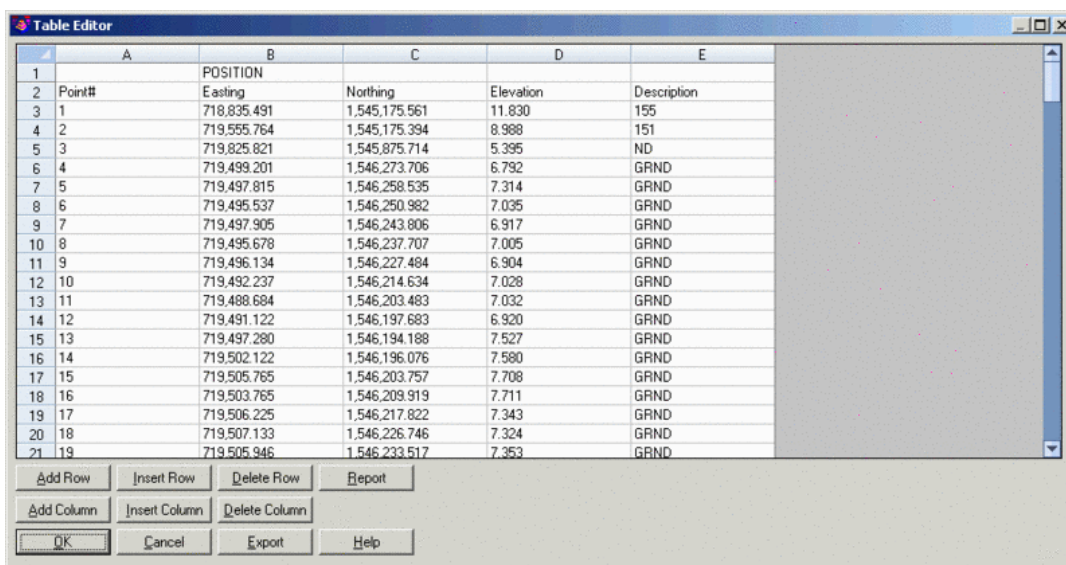
Keyboard Command: tabedit

Prerequisite: An entity based line or curve table.

Edit Table Values

The **Edit Table Values** permits the modification of any of the text labels found within a Carlson table entity. A Carlson table can be created using the Report Formatter, the Draw > Table command, or by Line or Curve table created with the Use Table Entity option enabled under Line/Curve Table Defaults.

To sort the table, you can pick on the header in the spreadsheet for the column to sort by.



	A	B	C	D	E
1		POSITION			
2	Point#	Easting	Northing	Elevation	Description
3	1	718,835.491	1,545,175.561	11.830	155
4	2	719,555.764	1,545,175.394	8.988	151
5	3	719,825.821	1,545,875.714	5.395	ND
6	4	719,499.201	1,546,273.706	6.792	GRND
7	5	719,497.815	1,546,258.535	7.314	GRND
8	6	719,495.537	1,546,250.982	7.035	GRND
9	7	719,497.905	1,546,243.806	6.917	GRND
10	8	719,495.678	1,546,237.707	7.005	GRND
11	9	719,496.134	1,546,227.484	6.904	GRND
12	10	719,492.237	1,546,214.634	7.028	GRND
13	11	719,488.684	1,546,203.483	7.032	GRND
14	12	719,491.122	1,546,197.683	6.920	GRND
15	13	719,497.280	1,546,194.188	7.527	GRND
16	14	719,502.122	1,546,196.076	7.580	GRND
17	15	719,505.765	1,546,203.757	7.708	GRND
18	16	719,503.765	1,546,209.919	7.711	GRND
19	17	719,506.225	1,546,217.822	7.343	GRND
20	18	719,507.133	1,546,226.746	7.324	GRND
21	19	719,505.946	1,546,233.517	7.353	GRND

Add Row: Adds a blank line to the end of the spreadsheet control.

Insert Row: Inserts a blank line immediately preceding the currently selected line in the spreadsheet control.

Delete Row: Removes the currently selected line from the spreadsheet control.

Add Column: Adds a blank column to the end of the spreadsheet control.

Insert Column: Inserts a blank column immediately preceding the currently selected column in the spreadsheet control.

Delete Column: Removes the currently selected column from the spreadsheet control.

Report: Sends the current content of the spreadsheet control to the Standard Report Viewer.

Export: Exports the current content of the spreadsheet control to an XLS file compatible with most spreadsheet applications, including Microsoft Excel (R).

Through the use of the Insert and Delete commands along with standard Windows Copy (Ctrl+C) and Paste (Ctrl+V) functionality, it is possible to return the list above into a normal-order list as illustrated below:

	A	B	C
1	LINE	BEARING	DISTANCE
2	L1	N 45°00'00" E	141.42'
3	L2	N 90°00'00" E	100.00'
4	L3	S 45°00'00" E	141.42'
5	L4	S 56°18'36" W	180.28'
6	L5	N 56°18'36" W	180.28'

Note:

- Changes to the direction or length values **DO NOT** change the direction or length of the corresponding line or curve entity in the drawing!

Prompts

Select an entity table to modify: Graphically select any portion of a table that is to be edited

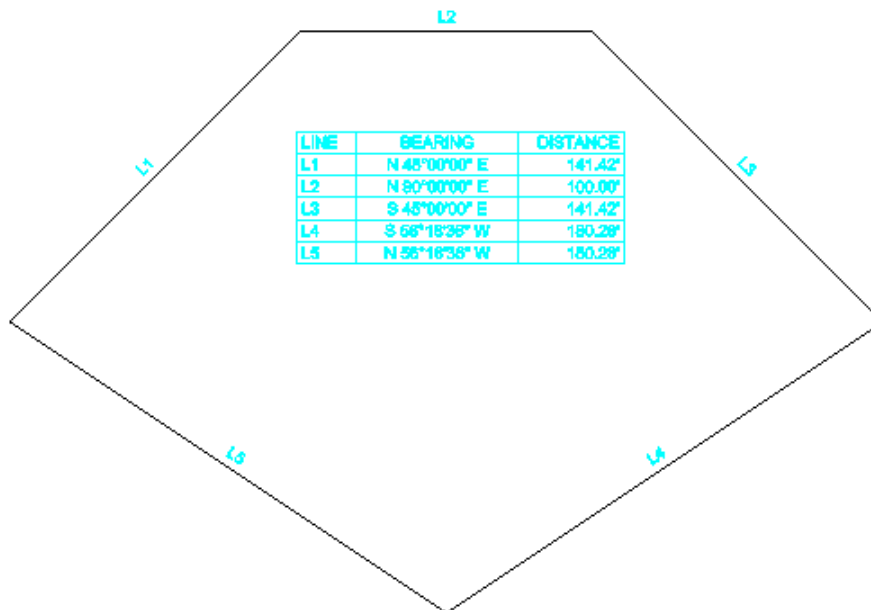
Pulldown Menu Location(s): Annotate > Line/Curve Table

Keyboard Command: edittbl

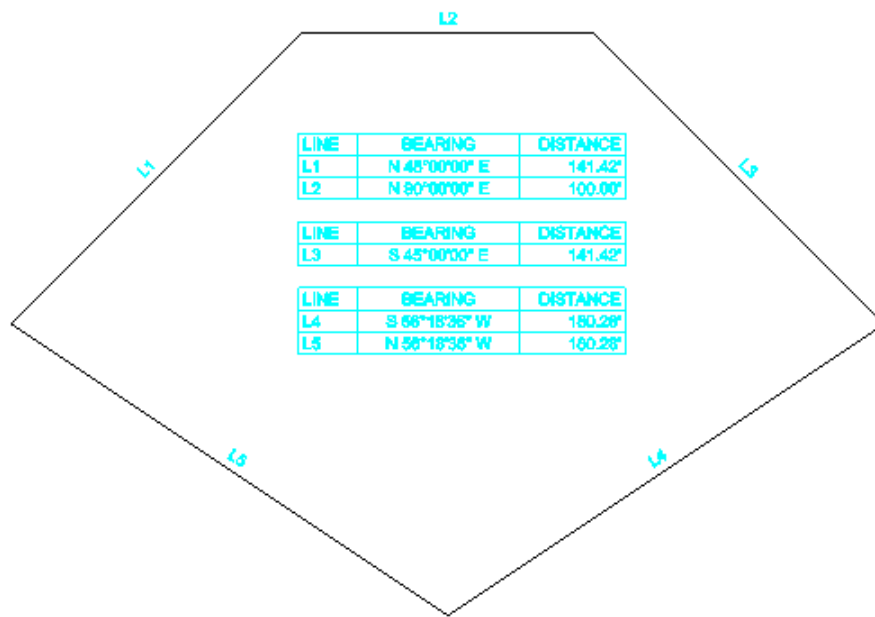
Prerequisite: A line or curve table created with the Use Table Entity option enabled under Line/Curve Table Defaults

Split Table

The **Split Table** command allows you to break a Line or Curve table created with the Use Table Entity option enabled under Line/Curve Table Defaults. This is commonly desired when a table is too lengthy to fit in its entirety on a plat. Splitting the table into two or more smaller tables allows the tables to be independently positioned on the plat. For example:



Becomes:



Note:

- In the example above, the initial table was first split at "L2" and then again at "L3".
- Split tables can be re-assembled through the use of the Merge Tables command.

Prompts

Select row of table to perform split on: *Graphically select the last row of the table that is to be retained in the original table*

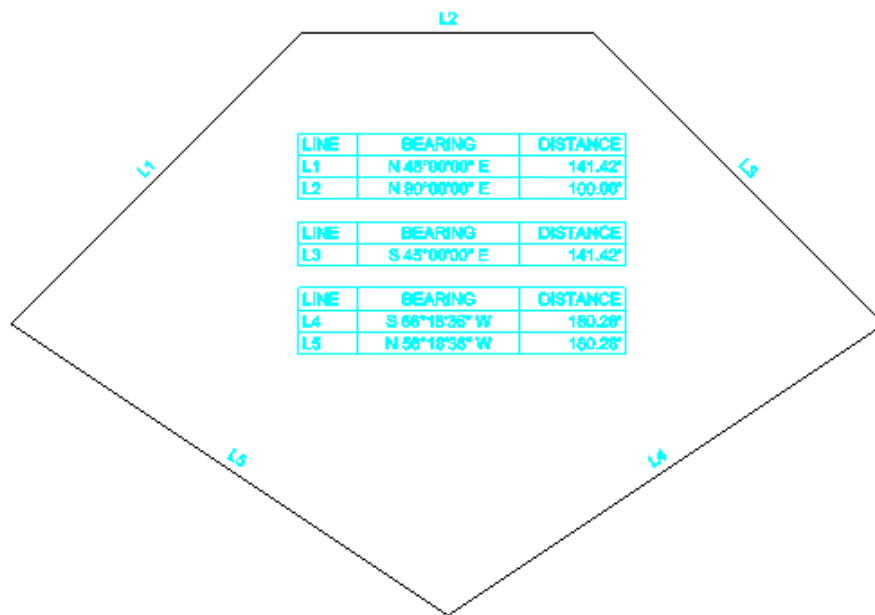
Pulldown Menu Location(s): Annotate > Line/Curve Table

Keyboard Command: splittbl

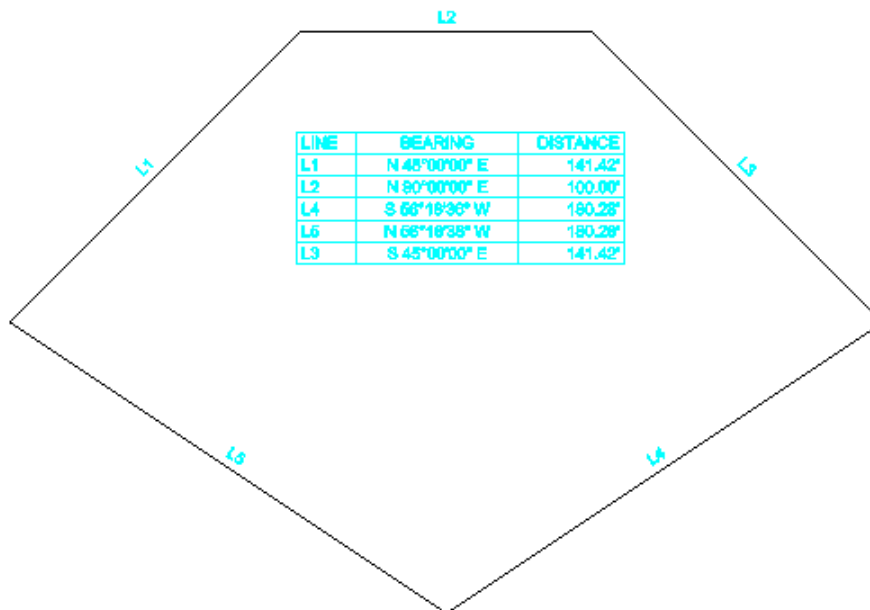
Prerequisite: A line or curve table created with the Use Table Entity option enabled under Line/Curve Table Defaults

Merge Tables

The **Merge Tables** command allows you to combine two Line tables or two Curve tables into a single table. Both tables in the merge must have been created with the Use Table Entity option enabled under Line/Curve Table Defaults. For example:



Can become:



Note:

- In the example above, table "L1" was merged with table "L4" and then the modified "L1" table was merged with table "L3".
- Table numbers can be re-ordered through the use of the Edit Table Values command.

Prompts

Select first table of merge: Graphically select the first of two tables that should be combined together

Select second table of merge: Graphically select the second of two tables that should be combined together

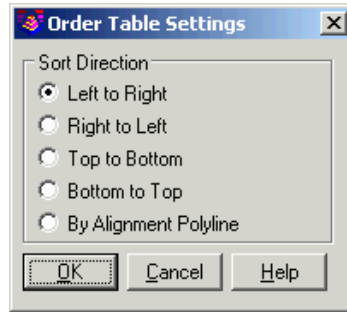
Pulldown Menu Location(s): Annotate > Line/Curve Table

Keyboard Command: splittbl

Prerequisite: Two or more line or curve table created with the Use Table Entity option enabled under Line/Curve Table Defaults. Each table in the merge must contain the same number of columns as the other table.

Order Table

This command orders the tags for line or curve tables to make the tags easier to reference in the drawing. To run this command, simply pick the line or curve table to order. At the command prompt, type O for Options to select the ordering method. The Alignment Polyline method lets you pick a polyline such as a centerline to have the tag order follow the polyline.



Prompts

Select table to order (O for Options): *pick table*

Select table to order (O for Options): *press Enter*

Pulldown Menu Location: Annotate > Line/Curve Table

Keyboard Command: order.tbl

Prerequisite: line or curve table

Delete Table Elements

This command erases rows from line or curve tables. The table entries following the removed rows are automatically repositioned and renumbered. The line or curve labels on the linework in the drawing are also updated.

Pulldown Menu Location: Annotate > Line/Curve Table

Keyboard Command: del.tbl

Prerequisite: Line or curve tables

Label Arc

This command labels the arc data along the arc between the endpoints of the arc. The curve information is also displayed. The format for the label is set in the dialog shown here. For each arc data value, you can specify the label, the row number, and the side of the arc it will appear on. If a row number is left blank, then that value is not labeled. There is a choice of labeling inside or outside of the arc. Annotation is drawn as a block. The advantage of this is that the characters, rather than being individual entities, are plotted as a single entity that can be moved and edited as a unit. You would need to explode the "blocked" text in order to edit the text. A toggle button determines whether the user wants to flip the text on arcs that open to the top of the drawing.

Prompts

Define arc by, Points/<select arc or polyline>: *select arc*

After selecting the arc or polyline arc segment the command displays the dialog below. Select the OK button and the arc is labeled with the current settings of the dialog.

Label Arc Settings ✖

Label Options

	Label	Row	Side
Arc Length:	L=	1	Inside
Radius:	R=	1	Inside
Delta Angle:	D=	None	Inside
Chord Angle:	BRG=	None	Inside
Chord Length:	C LEN=	None	Inside
Tangent:	T=	None	Inside
Other Text:	Dc=	None	Inside

Flip Text on Arcs that Open to the North

Use Symbol for Delta Angle Label

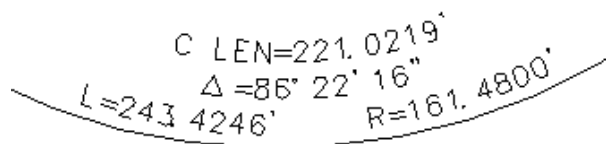
Label Chord Angles in...

Bearings Azimuths Gons

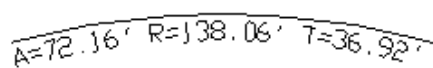
Type of Curve

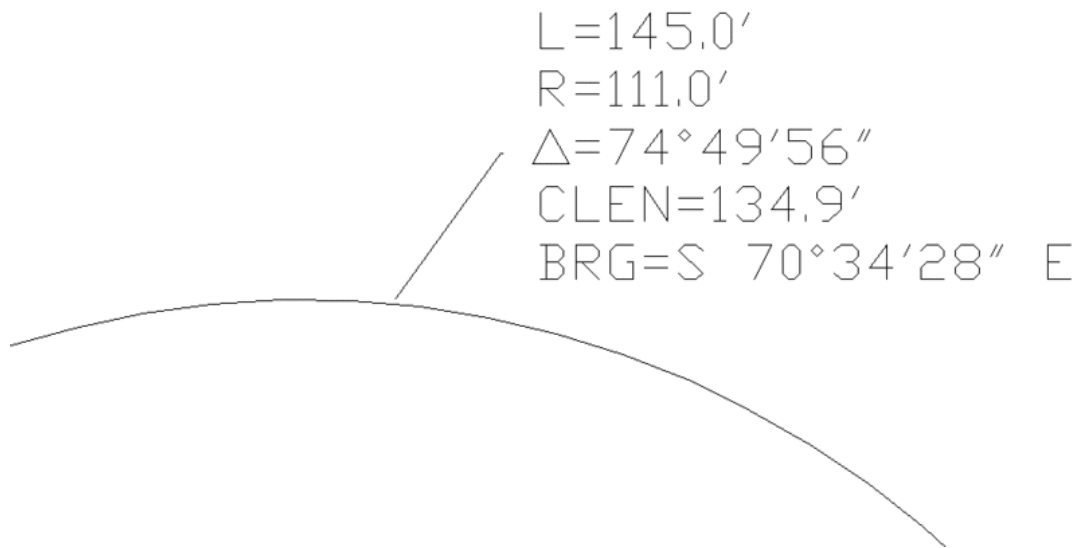
Road Rail

OK Cancel Help



Examples of Label Arc (above and below)





Example of Stack Label Arc

Pulldown Menu Location: Annotate > Annotate Arc

Keyboard Command: labarc

Prerequisite: Arc or polyline should be drawn before execution

Stack Label Arc Auto Place

This command is similar to Stack Label Arc Pick Side. The difference is the labels are automatically drawn at the middle of the arc. The command prompts to select an arc, define the arc by three points, or type O for Option to display the dialog shown here. See the Stack Label Arc Pick Side topic for a description of the options.

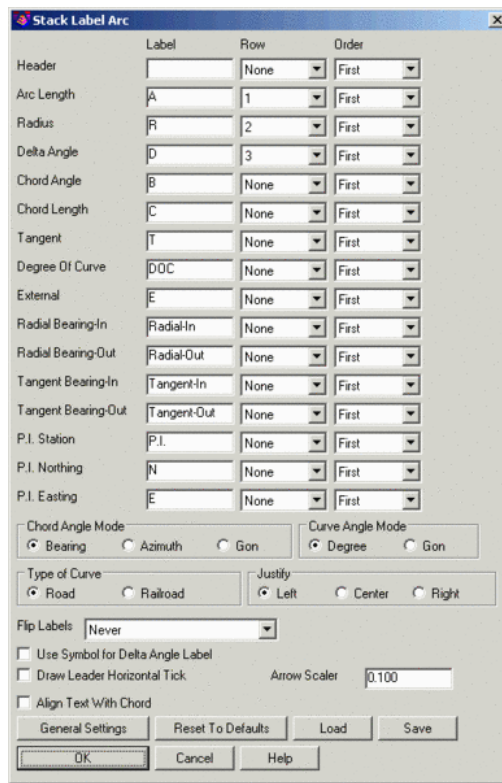
Pulldown Menu Location: Annotate > Annotate Arc

Keyboard Command: arcstack

Prerequisite: an arc entity or arc points

Stack Label Arc Pick Side

This command draws a small table of curve data. Unlike the command *Label Arc*, instead of fitting the text on the arc, this command lines the data up in rows. The command prompts to select an arc, define the arc by three points, or type O for Option to display the dialog shown here. For each type of arc value, you can specify the label and the sequence number. Under Label Options, the Stack Label Arc data table will display the values in the order by sequence number. There are also settings to justify label left or right.



Under Label Options, the data table will display the values in order based upon sequence number. For each of the arc properties, you can set the **Label** prefix, **Row** and **Order**. For multiple labels on the same row, the Order controls the label sequence for that row. The Header field is just the specified label and doesn't have an arc value. To display the special C/L label, enter \U+2104 in the label.

Chord Angle Mode allows you to set how the chord and radial angles are labeled as azimuth, bearing or gon.

Curve Angle Mode allows you to set how the delta angle and degree of curve are labeled as degree/minute/second or gon.

The **Type of Curve** option determines the type of curve.

Roadway: The length is determined as the true length of the curve.

Railroad: The length is adjusted based on 100-foot chord segments.

Justify sets the alignment for the text as left, center or right.

Flip Labels controls whether the text is drawn upside down in the current twist screen view.

The **Use symbol for Delta Angle** option uses a delta triangle symbol for the prefix.

Draw Leader Horizontal Tick draws a short horizontal line at the label end of the leader.

Align Text With Chord sets the angle of the text to match the chord angle. Otherwise, the text is draw horizontal to the current twist screen.

General Settings shows Annotation Defaults which has settings such as Text Size Scaler which apply to this routine.

Reset To Defaults puts the settings back to built-in defaults.

Load and **Save** functions store and recall the settings to an .ANS file. This is a way to share a label style with others or manage different styles.

Prompts

Options/Points/<Select arc>: P The P option causes the command to prompt for points on the arc. This can be

useful for labeling sub-arcs such as lot corners of a cul-de-sac.

Pick point or point number for Endpoint of arc: *pick a point*

Pick point or point number for Radius: *pick a point*

Pick point or point number for Other Endpoint: *pick a point*

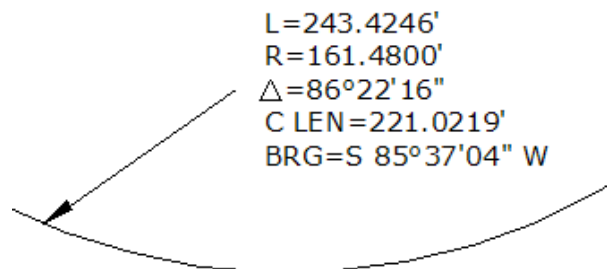
Direction of curve [Left/<Right>]? *press Enter for right*

Pick stack label point (Enter for none): *pick a point*

Pick point to start leader at ([Enter] for none): *pick a point*

To point: *pick a point*

Options/Points/<Select arc> (Enter to end): *press Enter to end*



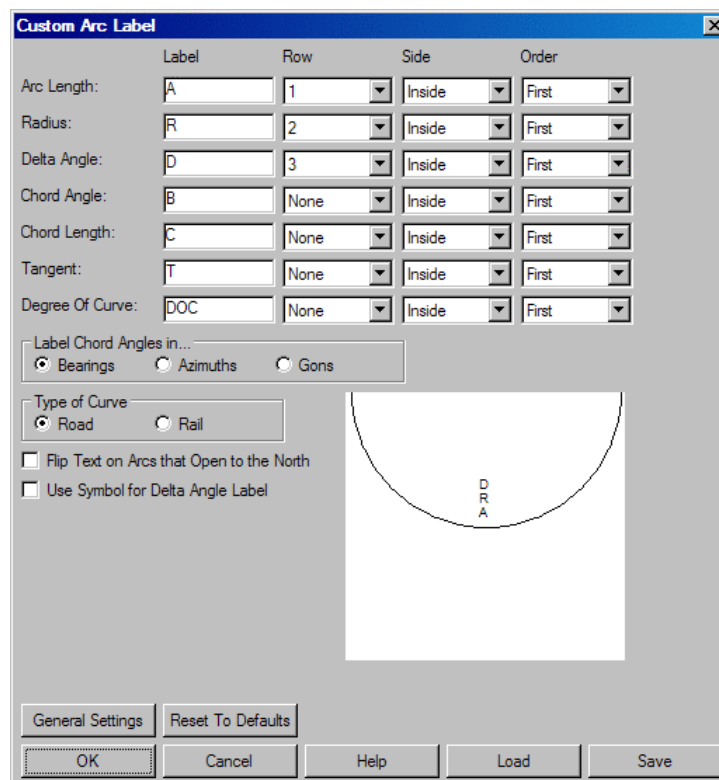
Pulldown Menu Location: Annotate > Annotate Arc

Keyboard Command: slabarc

Prerequisite: an arc entity or arc points

Custom Label Formatter

This command allows you to customize the labeling for arcs. You are first prompted to select an arc to label, given the existing defaults currently set. The arc is shown as labeled on the screen. The command line, shown below, also offers you an important choice called Options. When you type 'O' for options the below dialog box appears. There are four columns at the top of the dialog along with other features.



Label: This first column allows you to set the prefix that will go before your arc data.

Row: This column allows you to stack the data in different ways. You can place more than one item in the same row. If *None* is selected then that item will not be displayed.

Side: This column allows you to place each item either inside or outside of the arc.

Order: This column determines the order of items when they are placed in the same row.

Flip Text on Arcs that Open to the North: When this is checked text will be orientated according to the open side of your arcs instead of being orientated according to the plain view.

Use Symbol for Delta Angle Label: Allows you to use the triangle symbol for delta as the label instead of plain alphabetic or numeric representation.

General Settings: This button brings you to the Annotate Defaults dialog, see 'Annotate Defaults' for more.

Reset To Defaults: This button restores the default settings shown above.

Load/Save: You may also Load and Save different label configurations with the corresponding buttons.

Prompts

Options/⟨Select arc⟩: *select entity*

Options/⟨Select arc⟩: *O*

Custom Arc Label dialog *choose your preferences and click OK*

You can decide to go into the Option dialog at the start of the command and after your initial labeling.

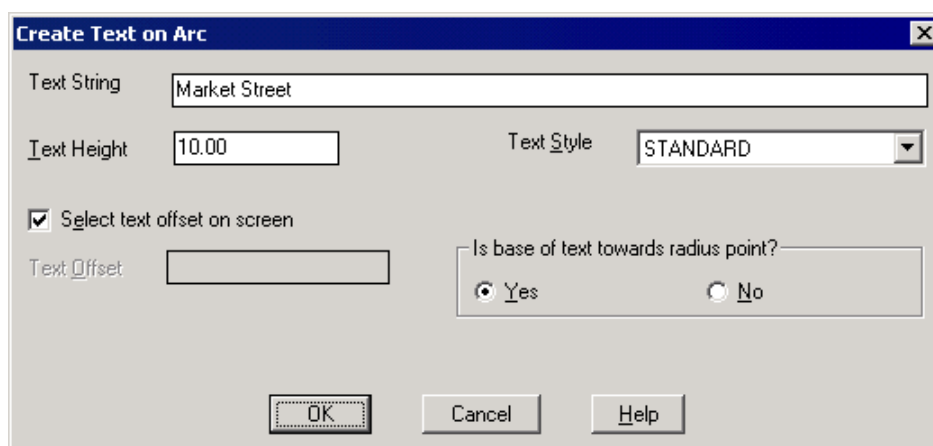
Pulldown Menu Location: Annotate > Annotate Arc

Keyboard Command: annarc

Prerequisite: An arc to label

Draw Text On Arc

This command draws text that aligns with an arc or polyline arc segment. Each letter of the text is drawn as a separate text entity that is rotated to align with the arc at that point. These text letters are automatically grouped together as an anonymous block. This command starts with the Create Text on Arc dialog. This command draws text that aligns with an arc, beginning at a picked point. Each letter of the text is drawn as a separate text entity that is rotated to align with the arc. These text letters are automatically grouped together as a block. The text string, text height, and text style are set in the Create Text on Arc dialog box.



Text String: Specify the text to label on the arc.

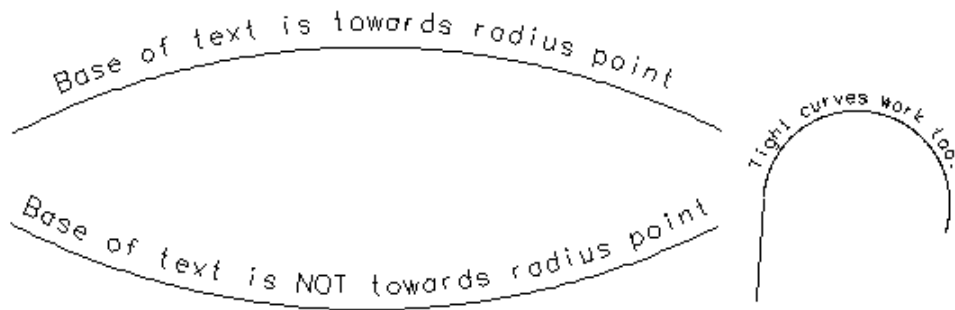
Text Height: Specify the text height. The default value is obtained from the text height specified in Drawing Setup. The value set here is retained throughout the drawing session.

Text Style: Choose an existing text style from the list of defined styles.

Select text offset on screen: When checked, the program will prompt you for offset. You can set the text offset from the arc by graphically picking the offset point on the screen. When this option is not checked, the Text Offset field described below becomes available to specify a known offset distance.

Text Offset: If the above setting is not selected, specify the Text Offset here. A positive value denotes an offset distance inside the arc, while a negative value denotes an offset distance outside the arc.

Is base of text towards radius point?: This option determines whether the base of the text should face the radius point of the arc. It orients the text to the curve. Examples showing the results of different settings follow.



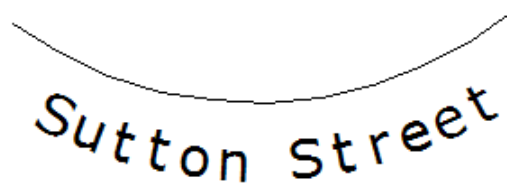
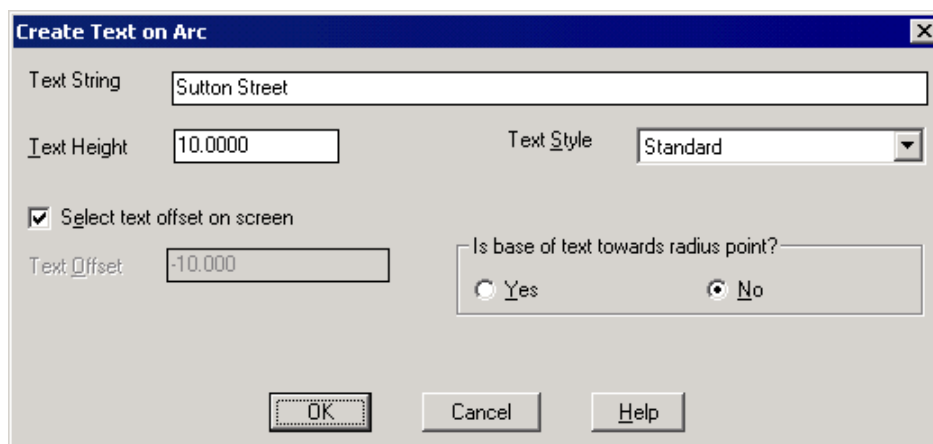
Example 1 - Offset distance specified on screen and base of text away from radius point.

Select Arc or Polyline segment: **pick Arc or Polyline segment** to place text on.

Select Text Offset: **pick the desired offset distance from arc**

Select Text Placement: **pick a point**, select the desired position to draw the text. Note that the text remains visible on the screen and attached to the "rubber banding cursor" so that various positions can be inspected before specifying the placement point. The graphic below shows this aspect of the command.

Note that the ghosted text is located along the mid point of the arc. If no offset distance is specified or picked from the screen, the text will be placed at this point. An offset of zero puts the text directly on the arc.

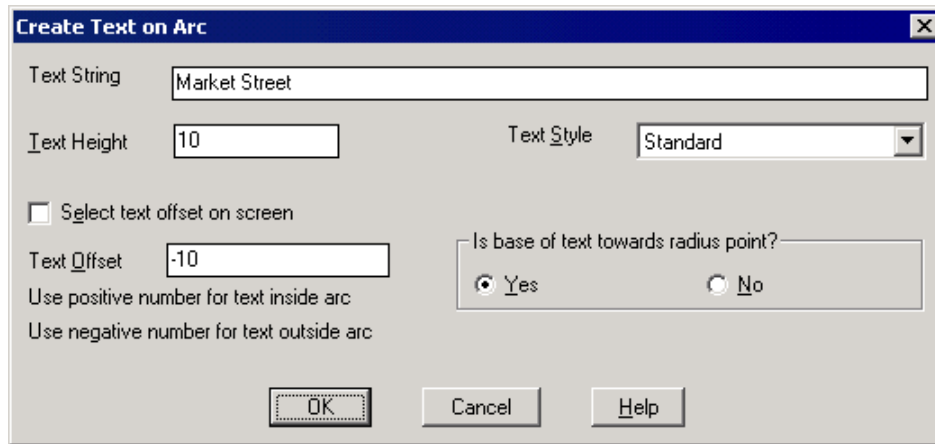


Example 2 - Offset distance specified in dialog and base of text towards radius point.

Select Arc or Polyline Arc Segment: **pick Arc**

Select Text Placement: **pick point**

Note that the prompt for offset distance was skipped because the offset distance was input on the dialog box. Simply select the text placement point resulting in the graphic below.



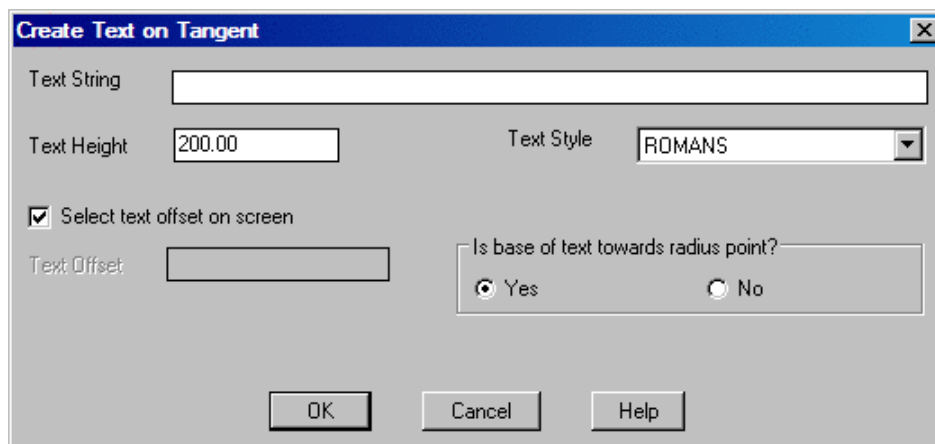
Pulldown Menu Location: Annotate > Annotate Arc

Keyboard Command: textarc

Prerequisite: An arc entity

Draw Text on Tangent

This command is identical to *Draw Text on Arc*, except that the text is not curved to fit the arc. You are presented with this dialog box. Fill in the text, decide on the other options, click *OK*, and then follow the prompts.



Pulldown Menu Location: Annotate > Annotate Arc

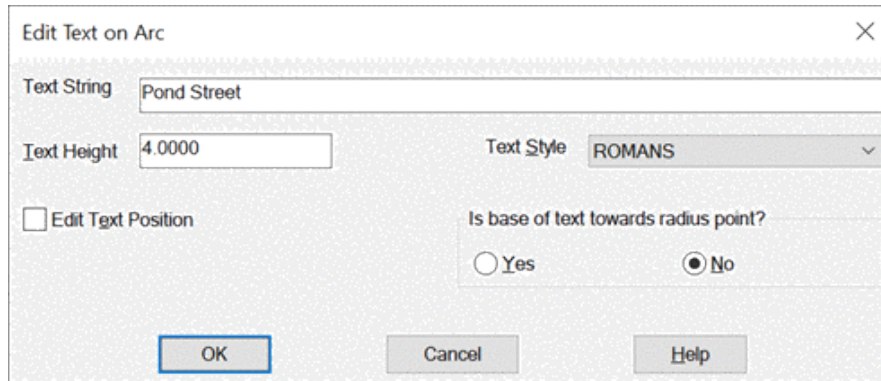
Keyboard Command: textarctan

Prerequisite: Arc or polyline arc segment

Edit Text on Arc or Tangent

This command allows you to edit text created by the *Draw Text on Arc* or *Draw Text on Tangent* command. You can change the text string, text height and text style. Use Edit Text Position to slide the text along the arc. The Towards Radius Point option can be used to flip the text.

The program prompts you to select the Text on Arc entity, then displays the same dialog used in *Draw Text on Arc*.



Pulldown Menu Location: Annotate > Annotate Arc

Keyboard Command: editarctext

Prerequisite: text entity as described above

Fit Text Inside Arc

This command fits text between two points picked along an arc. Text is curved to fit the arc using individual text entities, which can only be edited one at a time. The *Draw Text on Arc* command creates a text entity that can be edited using *Edit Text on Arc or Tangent*. It will optionally display information about the selected arc. If you choose to display the curve data, you will be prompted to pick the endpoints of the arc in a clockwise manner. When prompted, enter the text you want drawn inside the arc.

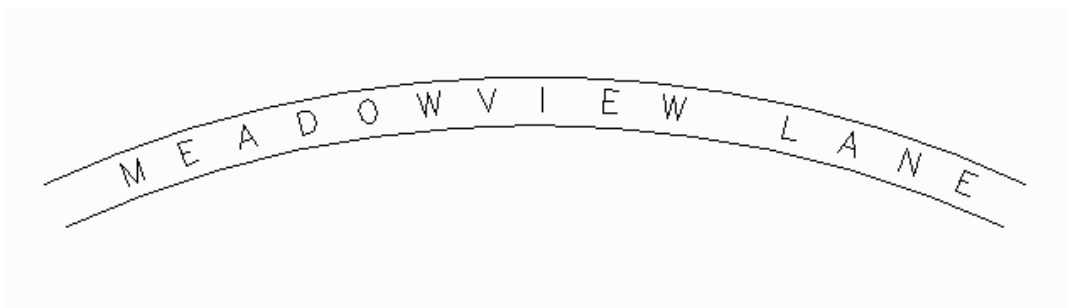
Prompts

Pick points in a clockwise direction.

[nea on] Start Point on arc for text: *pick point on arc to start text* Notice that the Nearest snap is turned on by default.

[nea on] End Point on arc for text: *pick point on arc to end text* Notice that the Nearest snap is turned on by default.

Enter text for inside of arc: MEADOWVIEW LANE



Pulldown Menu Location: Annotate > Annotate Arc

Keyboard Command: inarc

Prerequisite: An arc entity

Fit Text Outside Arc

Same as the previous command except this command fits text on the outside of the arc.

Pulldown Menu Location: Annotate > Annotate Arc

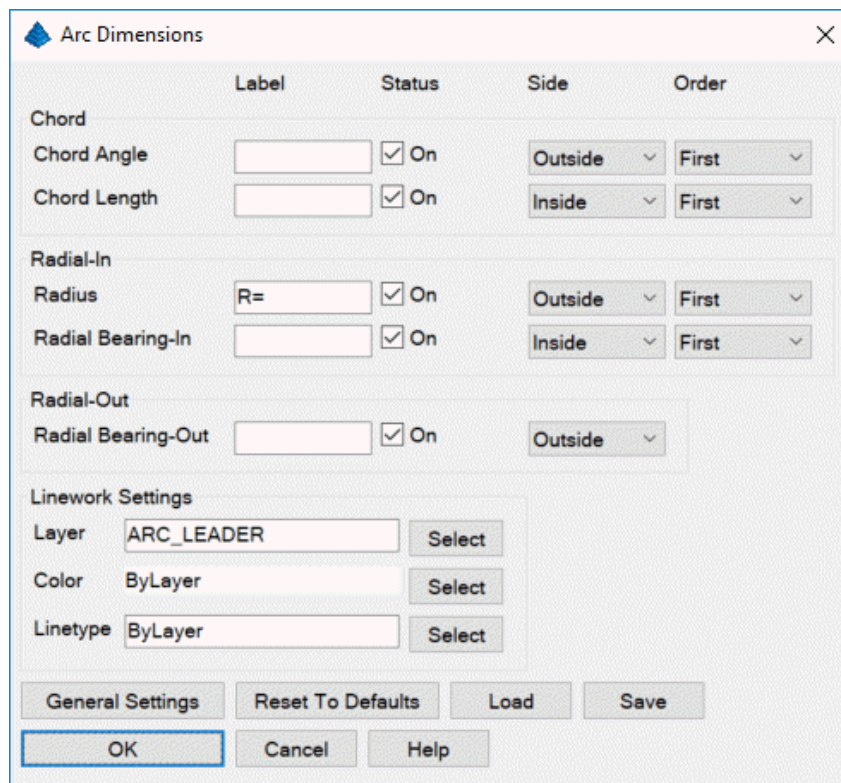
Keyboard Command: OARCT

Prerequisite: An arc entity

Arc Dimensions

This command labels dimensions of an arc including the chord angle, chord length, radius and radial bearing in and out. Besides creating the labels, this command also draws the chord line and radial lines. In the options dialog, choose which labels to draw, the label prefix and label position. The arc can be defined by selecting an arc entity, selecting an arc segment of a polylines or by entering points for the arc.





Prompts

Options/Points/⟨Select arc⟩: *pick an arc*

Pulldown Menu Location: Annotate > Annotate Arc

Keyboard Command: arcdim

Prerequisite: None

Change Polyline Linetype

This command changes the linetype of polylines or lines to the linetype selected from the dialog. True AutoCAD linetypes are created and applied to the selected entities, compared to other commands, such as *Polyline to Special Line* and *Special Line/Entity*, which break the polyline into segments. The spacing between linetype symbols and the symbols size are controlled by the Line Type Spacing and Symbol Size Scaler settings in the dialog. The Extra Dash is used for linetypes that have a symbol or letter between dashes. This setting controls how many dashes to draw between the symbol or letter. For example with the Electric2 linetype and Extra Dash set to 1, the linetype would have "- - E - - E - -". The Gap Size Scaler controls the size of the break in the line for the linetypes that have a break like UserDef and Arrow_B. The Gap Size Scaler is multiplied by the symbol size to get the gap size in drawing units. The Text Style is used for linetypes that use letters like "E" in Electric2.

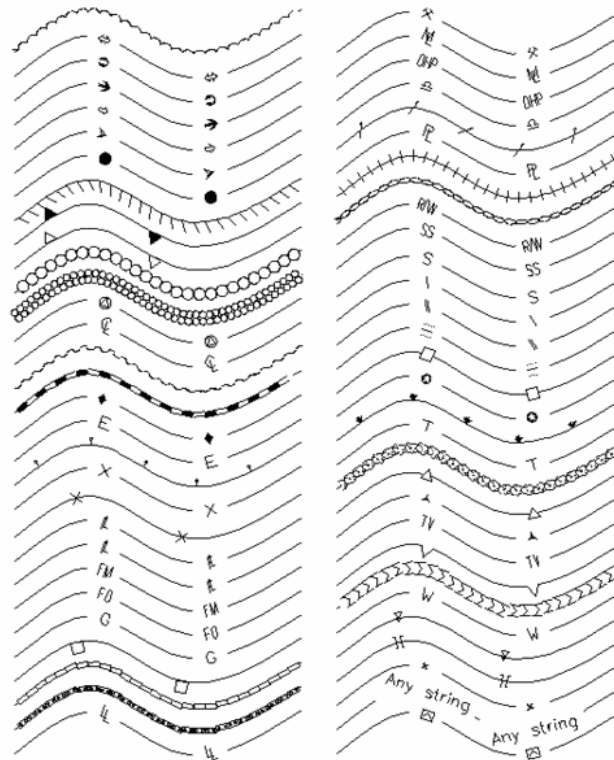
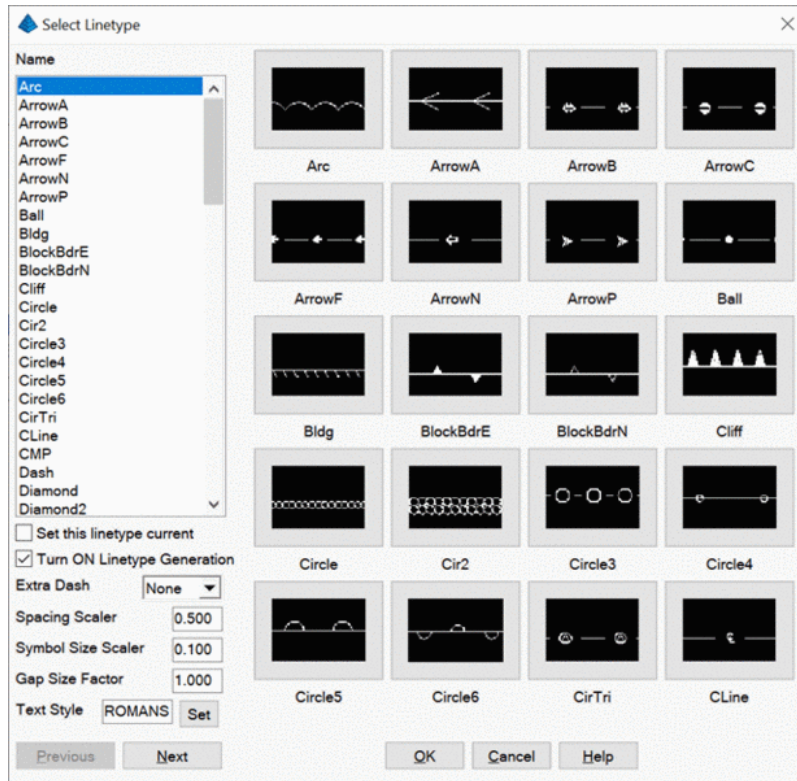
To select a linetype from the dialog, pick on the linetype image. Use the Next button to see more linetypes. At the end of the list of linetypes, there are two special choices. The UserDef choice lets you enter your own text string into a linetype, and the Wingdings choice lets you insert any Wingdings font character into a linetype. Consult Windows® documentation for a listing of Wingdings characters.

Prompts

Select Linetype dialog *select linetype and adjust other variables*

Select items to change.

Select objects: pick the polylines



Linetype styles available using Change Polyline Linetype

Pulldown Menu Location: Annotate > Line Types

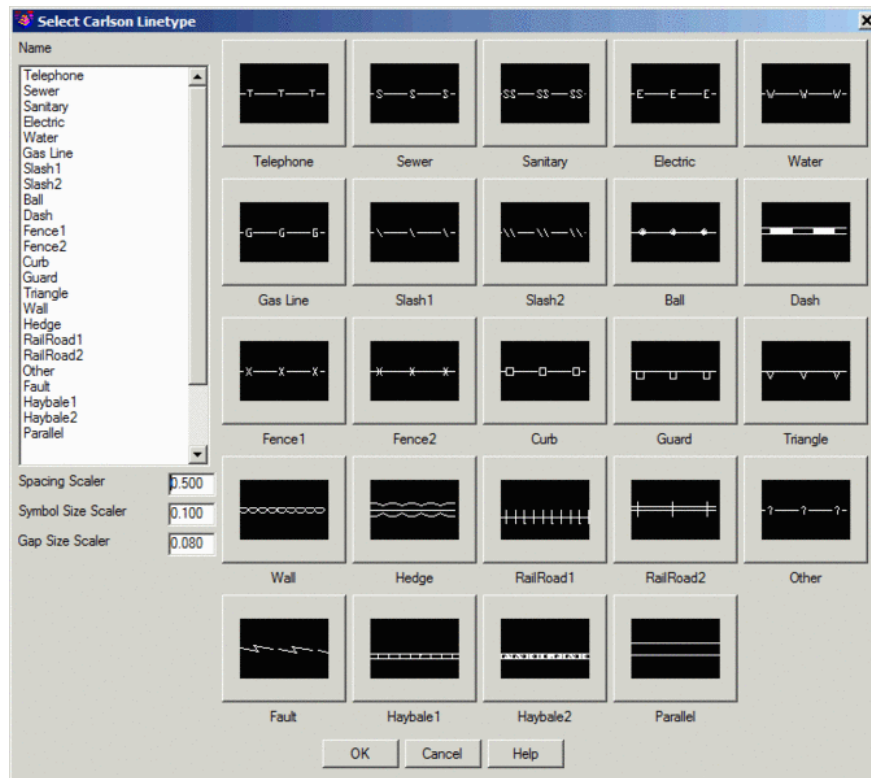
Keyboard Command: pltype2

Prerequisite: Polylines

Polyline to Special Line

This command converts polylines into special lines by adding the appropriate symbol onto the polyline, such as railroad, hedge, stonewall or telephone lines. Carlson has defined several line types as shown below. You can create custom lines by selecting the Other '?' which then prompts you for the text label to use. The size and spacing are set by the **Spacing Scaler** and **Symbol Size Scaler** settings. For some of the linetypes, this routine breaks the polyline using the **Gap Size Scaler** in order to fit in the symbol. Broken polylines cannot be used by the *Area* command, and are difficult to edit.

The *Change Polyline Linetype* command is another linetype method that creates actual AutoCAD linetypes that are applied to the selected entities.



Prompts

Select Carlson Linetype dialog

Select the polyline(s) to convert.

Select objects: *pick the polylines*

Pulldown Menu Location: Annotate > Line Types

Keyboard Command: pltype

Prerequisite: Polylines

Polyline to Tree Line

This command changes a polyline into a series of semicircles for representing a tree line.

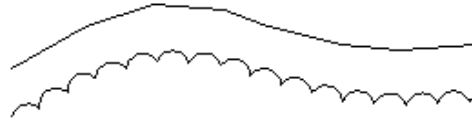
Prompts

Side for arcs on polyline direction? (<Left>/Right) *press Enter*

Enter the segment distance <10.0>: *press Enter*

Select the polylines to convert.

Select objects: *pick one or more polylines*



Before and After Polyline to Tree Line

Pulldown Menu Location: Annotate > Line Types

Keyboard Command: maketree

Prerequisite: Polyline

Add Zig to Polyline

This command draws a [not-to-scale] style zig to a polyline. First pick the polyline and then pick a position on the polyline to draw the zig.

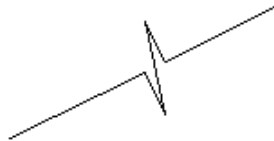
Prompts

Zig size <4.0>: *press Enter*

Select polyline to add zig: *pick a polyline*

Pick or enter point to add zig: *pick a point along the polyline*

Select polyline to add zig: *press Enter to end*



A zig in a polyline

Pulldown Menu Location: Annotate > Line Types

Keyboard Command: addzig

Prerequisite: Polyline

Add Culvert to Polyline

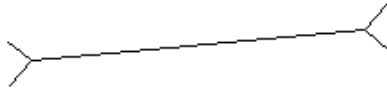
This command adds culvert style brackets to both ends of the selected polylines.

Prompts

Culvert symbol size <4.0>: *12*

Select polylines to add culvert symbols.

Select objects: *pick the polylines*



Culvert symbols added to polyline

Pulldown Menu Location: Annotate > Line Types

Keyboard Command: drwcvert

Prerequisite: Polyline

Sketch Tree Line

This command draws a tree line as you move the cursor. in the drawing editor. At the first prompt, you can type **O** for **Options** and set the Interval Scaler which controls the spacing of the arcs (bubbles) as well as the option to invert the arcs. In addition, you can select an existing polyline to convert to a tree line by typing **P** for **Polyline** at the Command Line Prompt. At the conclusion of sketching a treeline there will be a prompt and option to reverse the direction of the polyline. Choosing **Yes** results in the arcs being drawn on the opposite side of the polyline.



The resulting tree line after you pick point to end

Example of Treeline with Inverted Arcs

Prompts

Pick First Point [Options/Polyline]: *pick a point*

Sketch treeline (pick point to end): *slowly move the cursor and pick a second point to end the routine*

Reverse direction [Yes/<No>]?: *press Enter*

Options

Pick First Point [Options/Polyline]: O

Interval scaler <0.150>: *press Enter*

Invert Arcs Direction [Yes/<No>]?: N

Pulldown Menu Location: Annotate > Line Types

Keyboard Command: treeline

Prerequisite: None

Special Line/Entity

This command breaks a line, arc or polyline and inserts a string of text or a block at an interval. It can be used to draw fence lines, utility lines, tree lines or any line which can be constructed by inserting a text or block entity. The command prompts to select an entity then the distance between inserts. Next, the user selects whether to insert text or a block, and whether to enter the distance or length to be broken out of the entity. If the user enters a 0 distance for the break distance, then the entity is not broken. If a distance greater than 0 is entered, then this distance is divided in half and broken out of the entity on both sides of the point at which the insert distance measures the entity.

If the user elects to insert text, the command prompts for the text to be inserted. Next, choose whether you want the text Middle or Center aligned, and whether you want to have the text flipped so it does not appear upside down. See the AutoCAD Reference Manual for more information on justification options. The size of the text is controlled by the text size setting in *Drawing Setup*.

If the user elects to insert a block, the command prompts for the block name. The size of the block is controlled by the symbol size setting in *Drawing Setup*. Considering that almost anything can be made into a block, such as raster images, wipeout entities, etc., this is a very powerful command.

Alternatives to this command are *Polyline to Special Line* and *Change Polyline Linetype*.

Pulldown Menu Location: Annotate > Line Types

Keyboard Command: speent

Prerequisite: None

Guard Rail

This command adds box symbols along a polyline to generate a guard rail. See the command *Change Polyline Linetype* also.

Prompts

Pick Polyline/Last: *pick a polyline*

Left/Right: *L for Left*

Pulldown Menu Location: Annotate > Line Types

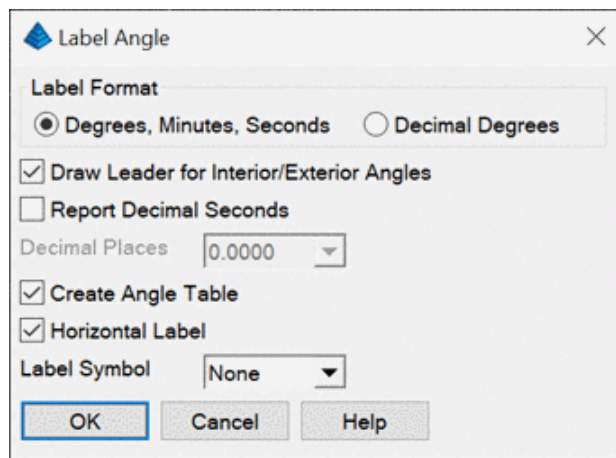
Keyboard Command: grail

Prerequisite: Polyline

Label Angle

This command will label and report the interior, exterior and deflection angles between two directions. The angles can be defined by three points, or by two line or polyline segments that have a common endpoint. An example of labeling interior and exterior angles is shown below. The Both option will label interior and exterior angles simultaneously. When the input data has different elevations, the program prompts whether to label the 2D or 3D angle. For 2D angle, the program ignores the elevations. For 3D angle, the program projects the data onto a 3D plane and calculates the angles on this plane.

At the Command line, type O for Options to bring up the options dialog. There are settings for the angle format as degrees, minutes and seconds, or as decimal degrees. You can also control the number of decimals and choose whether to draw dimension leaders for the interior angle labels. The Create Angle Table option will prompt to type in a Reference Label (ie. "A1") to draw in place of the angle value. Then at the end of the command, the program draws a table of the reference labels and angle values. The Horizontal Label option controls whether to draw the Reference Label horizontal or at the angle. The Label Symbol option sets the type of symbol to draw around the Reference Label.



Prompts

Define angle by [Options/Points/<select line or polyline>]: *pick a polyline segment*

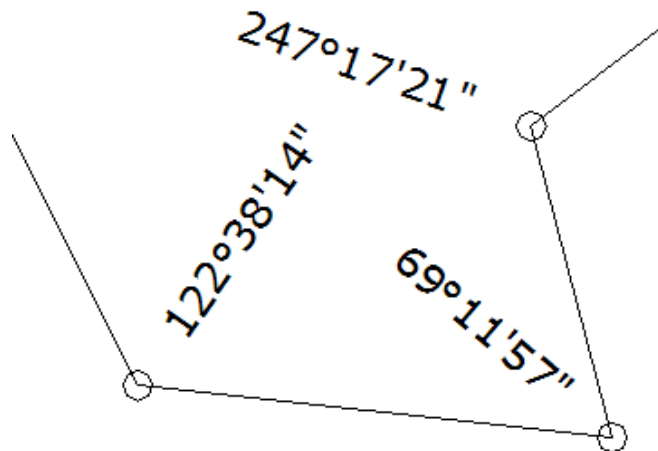
Select adjoining line or polyline: *pick another polyline segment*

Interior: 64d15'01" Exterior: 295d44'59" Deflection: 115d44'59"

Angle to label [<Interior>/Exterior/Both/Deflection/None]? *press Enter*

Typing B for Both labels both the interior and exterior angles.

Define angle by, Points/<select line or polyline>: *press Enter to end*



Pulldown Menu Location: Annotate

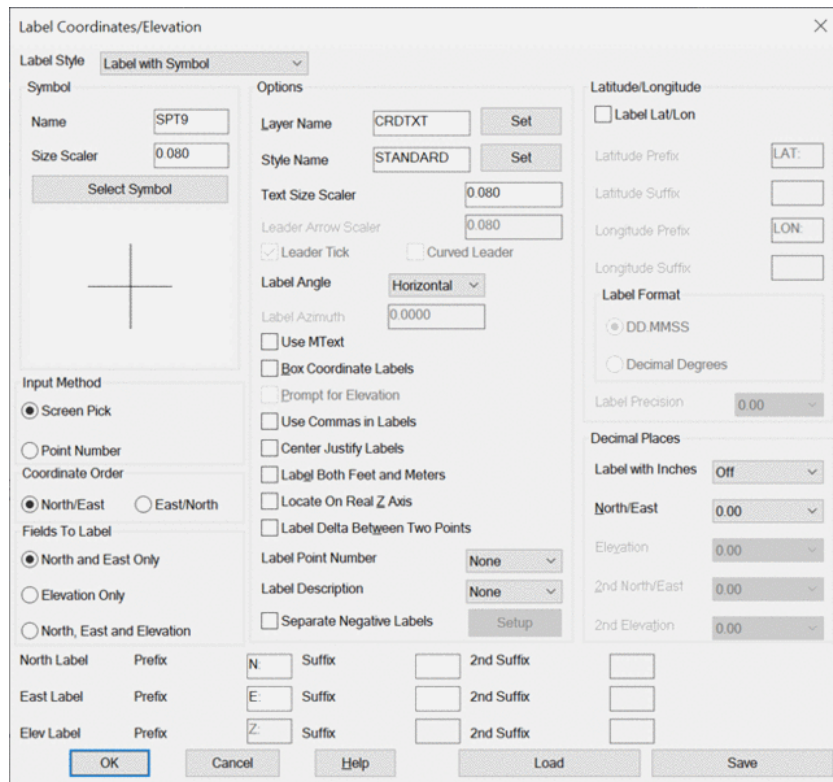
Keyboard Command: labang

Prerequisite: None

Label Coordinates/Elevation

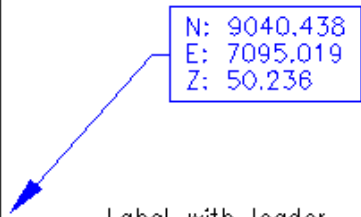
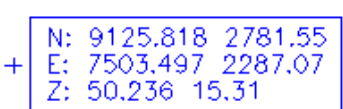
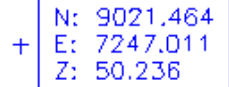

This command will label a coordinate on the screen. You can choose to label the northing and easting, or the elevation, or all three properties. The point can be picked on screen, or specified by point number or point group from the current coordinate (.CRD) file. Options include drawing a box around the label, labeling both feet and meters, setting the layer name for the label, setting the display precision, deciding whether or not to use a leader and selecting a change in the symbol used to mark the point. You can also set the text prefix and suffix for the label. Additionally, you can locate a label on Real Z Axis. The **Label With Inches** option labels with whole feet

and inches for the decimal part. The **Integers** controls how many decimals to label to the left of the decimal. The **Label Point Number** option uses the point number from the coordinate file. The **Label Description** option is for including a description with the label. There is a choice for placing this description as a header or footer. The program will prompt for the description. The **Use MText** option chooses between creating the labels as MText or regular text entities. The **Label Angle** setting chooses between having the labels horizontal, at a fixed azimuth or prompting to pick the angle for each label. The **Label Style** chooses between labeling with a leader, with a symbol, as an MLeader entity or along the x/y axis. The **Separate Negative Labels** option has prefix and suffix settings for negative coordinates that can be used for South and West labels. The **Label Lat/Lon** adds latitude and longitude labels to the coordinate labels. To use this option, the coordinate system must be defined in the Drawing Setup command.



N: 4066.66
 E: 5099.26

Label Along Axis

 <p>Label with leader</p>	 <p>Label both feet and meters</p>
 <p>Label without leader</p>	 <p>Label elevation only</p>

There is also an option to label the Delta X, Y and/or Z between two points, which is called **Label Delta Between Two Points**. When this option is clicked On, and after the prompt, you will first click two points locations. The label, with the delta value(s), will then be placed precisely in between these two pick locations. If, for example, the North, East and Elevation button is chosen, the resulting label will show the N, E and Z delta values.

The Save and Load buttons save and recall all the settings for this command to .LCE files. This is a way to manage different label styles for different mapping standards and to share between users.

Prompts

Label Coordinates/Elevation dialog

Point to Label ?

Pick point or point number: *pick a point*

Point to Label (ENTER to End)?

Pick point or point number: *press Enter*

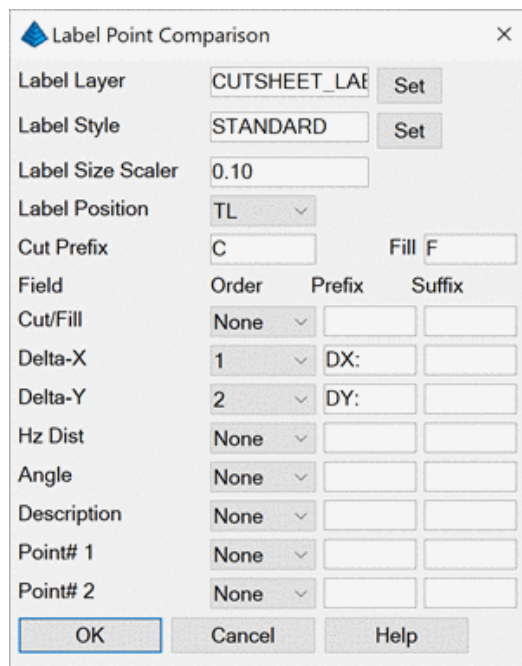
Pulldown Menu Location: Annotate

Keyboard Command: labcoor

Prerequisite: None

Label Point Comparison

This command labels the difference between two points. The points can be entered by point number or screen picked. The label is located between the points. The label layer, style, size and justification is set in the dialog along with which properties to label.



Prompts

Pick point or point number for 1st point: *pick point*

Pick point or point number for 2nd point: *pick point*

Pulldown Menu Location: Annotate

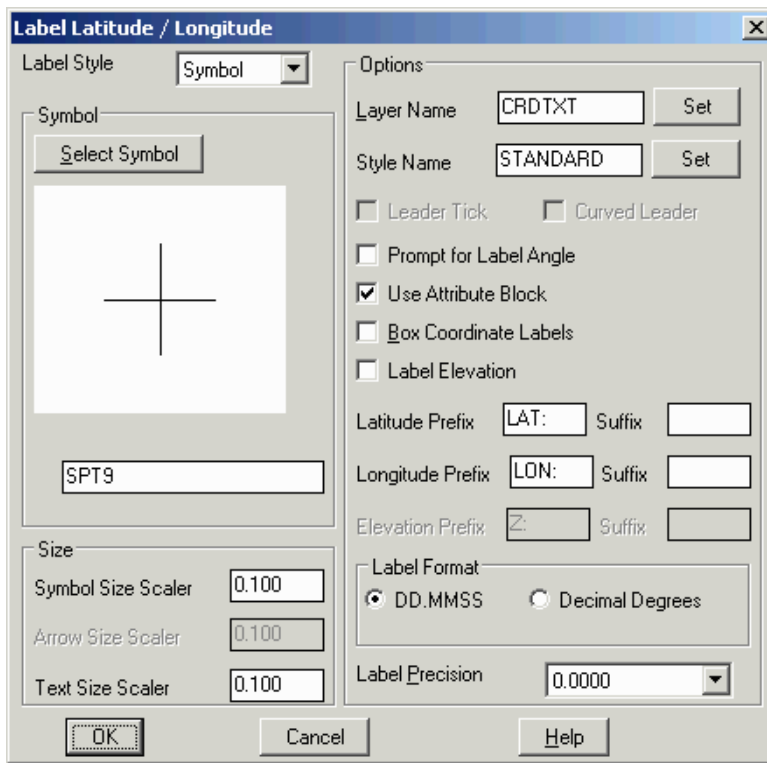
Keyboard Command: ptdiff

Prerequisite: None

Label LatLong

This command will label the latitude and longitude of a selected point. The program will convert the northing and easting of the input points to latitude and longitude. There is an option to include elevation in the labels. The coordinate system for the drawing coordinates must be defined in Drawing Setup before running this command. First, the program displays a dialog box with options to set the sizes, to set label prefixes, to set the display precision, to draw a box around the label and to change the symbol used to mark the point. The **Prompt For Label Angle** option prompts for the label rotation instead of automatically drawing the label horizontal. For the **Use Attribute Block** option, the symbol must be a block with three attributes for the latitude, longitude and elevation that the program will place and fill out the attributes instead of creating text labels. After the options dialog, the program prompts for the points to label. As you move the cursor, the program display the latitude/longitude in real-time.

For **Label Style**, the Leader and MLeader styles prompt for the point to label and then the label location and draws a leader or mleader between these points. For the Symbol style, the specified symbol is drawn at the point and the label is drawn automatically to the size.



Prompts

Label Latitude / Longitude dialog

Pick point or point number: *pick a point*

Pick point or point number: *press Enter to end*

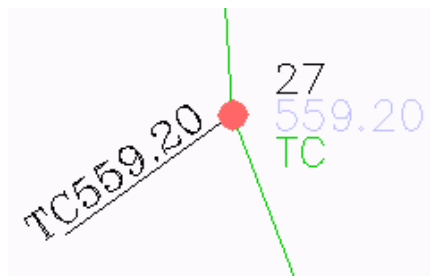
Pulldown Menu Location: Annotate

Keyboard Command: lablat

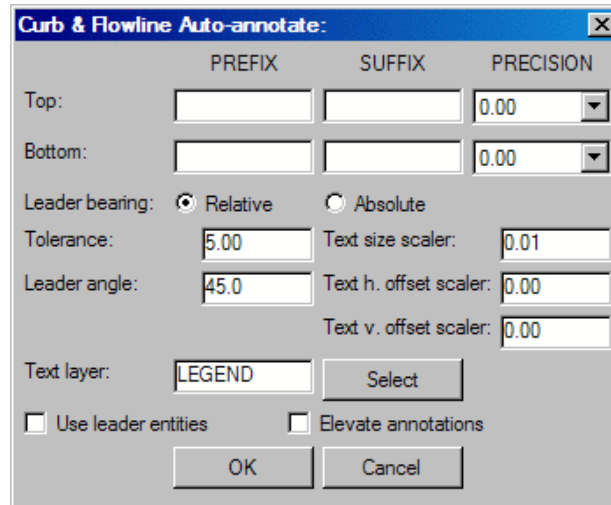
Prerequisite: Define coordinate system in Drawing Setup

Label Curb Flow Elevations

This command labels top of curb and/or bottom of curb (flowline) elevations with a leader along an alignment. The data to label comes from Carlson points and alignment is defined by a selected polyline. The program reads all the points in the drawing and then you select which descriptions to use the top of curb labels and which to use for the bottom of curb labels.



There are separate settings for the top and bottom to control the label prefix, suffix and decimals. The Tolerance setting is the maximum distance between a point to label and the polyline. The Leader Bearing determines how the Leader Angle is applied: Relative adds the angle to the alignment polyline and Absolute means based on the orientation of the screen. The Text Horizontal Offset Scaler controls the distance between the alignment polyline and the label. The Text Vertical Offset Scaler controls the buffer offset between the leader line and the label. The User Leader Entities option chooses between drawing the leaders as polylines or as leader entities. The Elevate Annotations setting controls whether the labels are created at their elevation or at zero.



Prompts

Top Curb Descriptions pick descriptions to label for top of curb

Bottom Curb Descriptions pick descriptions to label for bottom of curb

Pick a polyline to annotate (Enter to End): *pick a polyline*

Pick side for flowline (Enter to End): *pick a side*

Pulldown Menu Location: Annotate

Keyboard Command: cfelev

Prerequisite: points with elevations and descriptions, and alignment polyline

Replot Descriptions

This command will create user specified text entities at the location of selected point descriptions.

Prompts

This command will Search for a certain Point Description and plot

New text on the current layer with current style.

Attribute Text to Search for <>: *STK*

New Text to plot <>: *Stake Fnd*

Select objects: *Select Carlson points*

Select objects: *press Enter*

Number of Text Entities Plotted: 4

7
x
IP 50.24 8
x
STK 50.60 9
x
STK 50.90
10
x
STK 51.07
11
x
STK 50.92

Points with description STK

7
x
IP 50.24 8
x
Stake 50.60
Find
9
x
Stake 50.90
Find
10
x
Stake 51.07
Find
11
x
Stake 50.92
Find

Found four STK descriptions and created four text entities

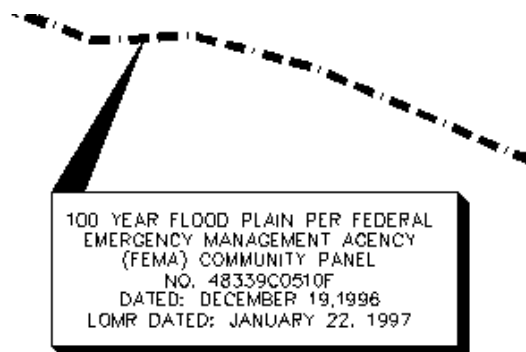
Pulldown Menu Location: Annotate

Keyboard Command: plotdesc

Prerequisite: Points with descriptions must be plotted. Set the layer and text style that you require.

Textbox

This command allows you to draw a shaded, shadow text box around a selection of Text or Mtext. The size of the shading and the optional leader are determined by the height of the text that is selected.



Pulldown Menu Location: Annotate

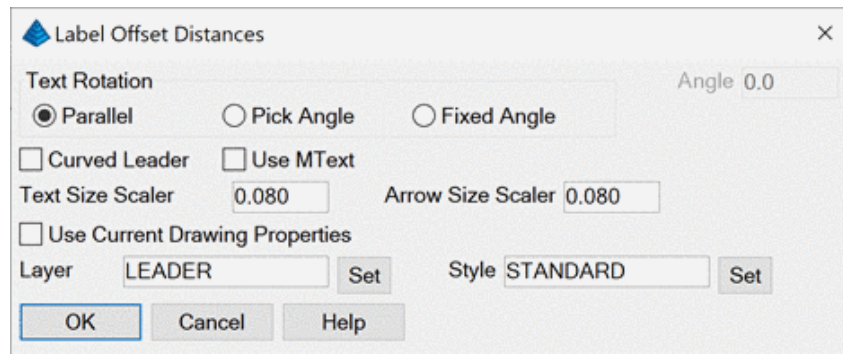
Keyboard Command: textbox

Prerequisite: Text or Mtext entities

Label Offset Distances

This command labels the distances of a point to one or two lines. The first distance is between the point and an east-west line. This distance is labeled as either north or south of the line. The second distance is between the point and a north-south line. This distance is labeled as either east or west of the line. The distances are labeled with a leader and a description of the point.

The settings dialog controls the label rotation, leader style, label size, layer and style.



Prompts

Pick 'E-W', Left to Right Property Line (if any)

Pick Line or Polyline (Enter for None): *pick the polyline*

Pick 'N-S', Top to Bottom Property Line (if any)

Pick Line or Polyline (Enter for None): *pick the polyline*

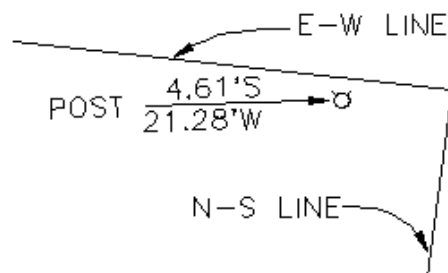
Pick Offset Point, (N) for Number, <E> to Exit: *pick a point*

Pick point to start leader at: *pick a point at or near the offset point*

To point: *pick an alignment point for the label*

To point: *press Enter*

Pick Offset Point, (N) for Number, <E> to Exit: *press Enter to End*



Pulldown Menu Location: Annotate

Keyboard Command: offlab

Prerequisite: Polyline or Line

CGSurvey Module

4

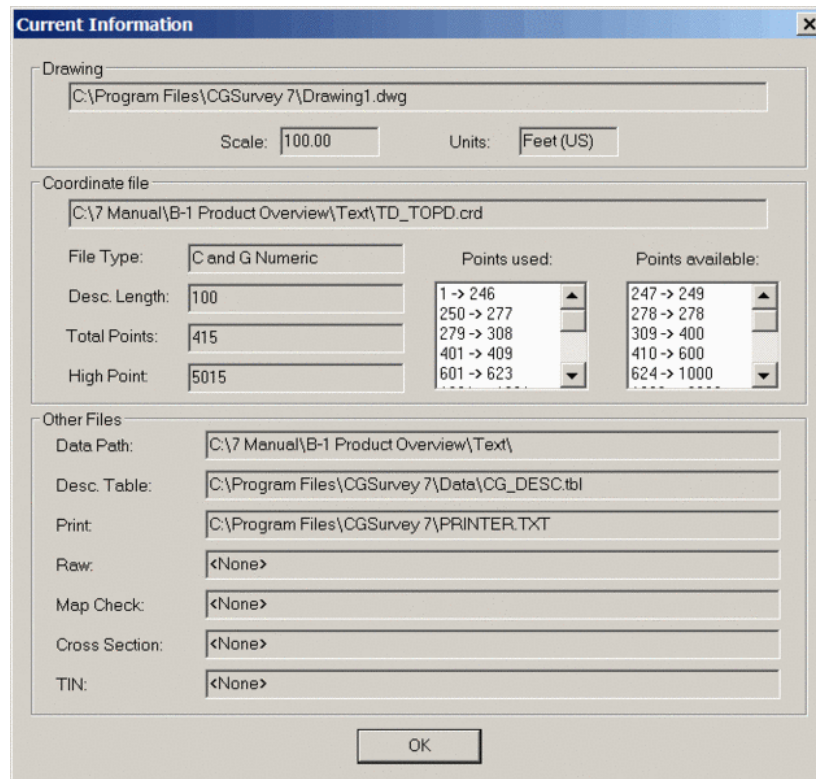
CGFile

Current Information

The Current Information Dialog Box contains information on:

Drawing: displays the current drawing file Path, Name, Scale, and Units

Coordinate File: displays the current coordinate file path and names.



File Type: This will display the current Coordinate file type.

The file types are:

C&G Numeric (.CRD) (PT #: 126)

C&G Alpha-Numeric (.CGC) (PT #: RW126)

Carlson Numeric (.CRD) (PT #: 126)

Carlson Alpha-Numeric (.CRD) (PT #: RW126)

Description Length: Numbers of character in the description

Total Points: Total number of points in the file

High Point: The highest point number stored

Points Used and Points Available: displays the block or blocks of points used or available in the coordinate file currently open.

Other Files: Displays the files that are currently open:

Data Path: displays the current default path and coordinate file name

Description Table: displays the current default path and description table

Print: displays the current default path and Print file name

Raw: displays the current default path and RAW file name

Map Check: displays the current default path and Map Check file name

Cross Section: displays the current default path and Cross Section file name

TIN: displays the current default path and TIN file name

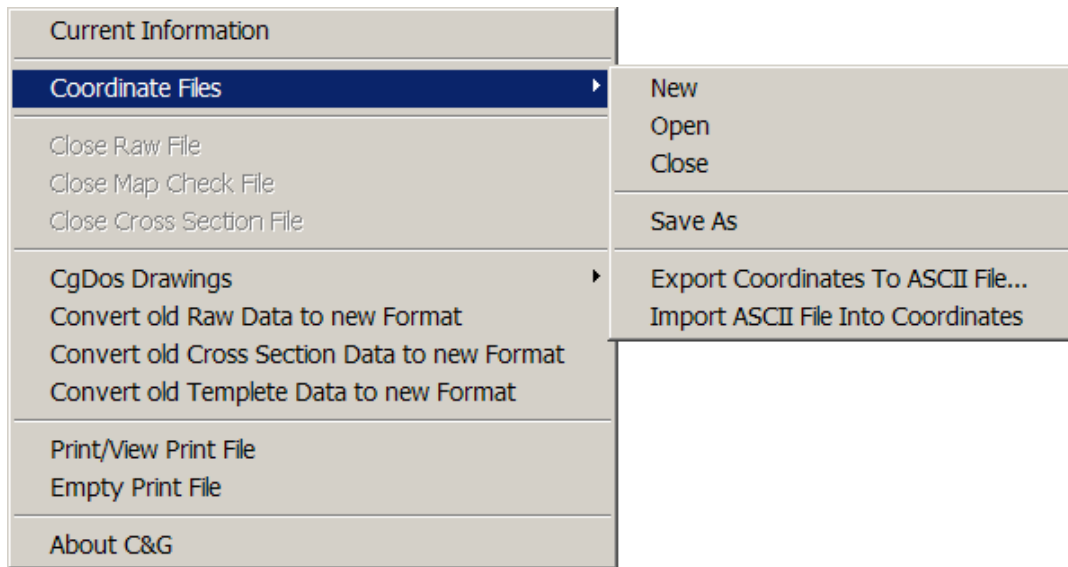
Pulldown menu Location:CG-Survey > File

Keyboard Command:INF, cg_current_info

Prerequisite: None

Coordinate Files

Choose Coordinate Files from the CGFile pull-down menu.

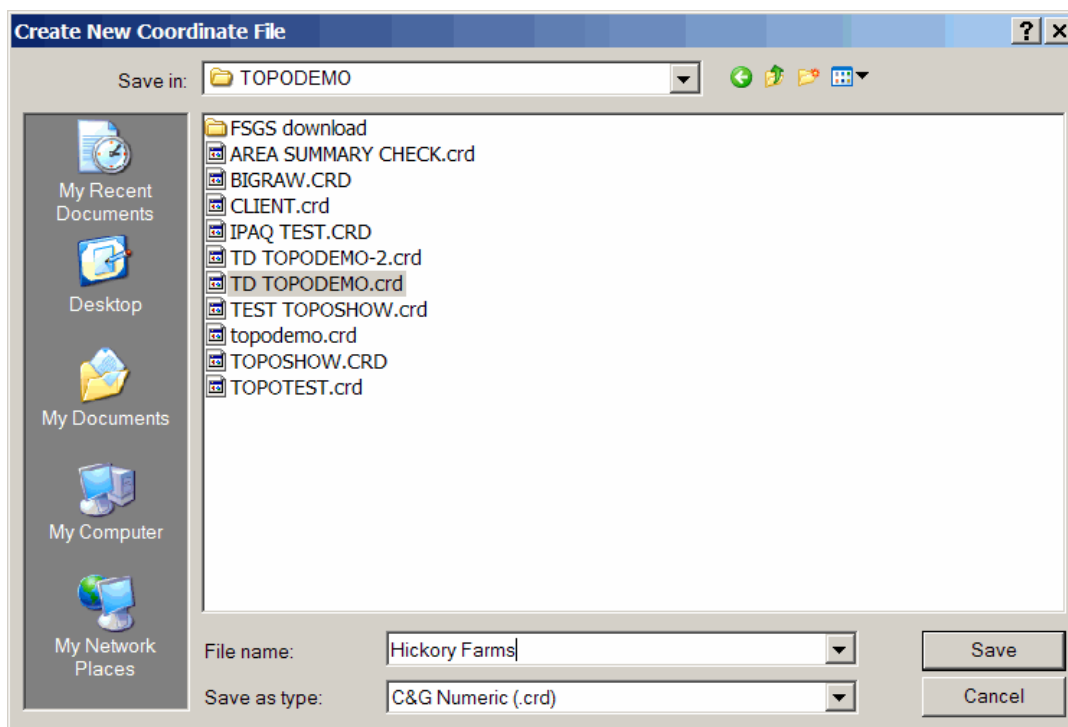


New

The New allows you to create a new coordinate file.

Prompts

Follow these steps: CGFile > Coordinate Files > New Coordinate File



Save in: Browse to folder location

Enter the name of the coordinate file you wish to create: File Name: Hickory farms

Press enter or press Save Button

NOTE: The directory displayed is the Data Path is the directory as set from the tool bar:

CG-Tools > CG Options > Data Path Options

NOTE: The description length for the new file just created will be set based on the current description length setting in the:

CG-Tools > CG Options > General

NOTE: You will not be able to change the description length once the new file is created. You must set the description length prior to creating the new file. You can however move the points to another file that has a longer description length

Pulldown Menu Location: CG-Survey > File > Coordinate Files

Keyboard Command: OPNC, CG_NEW_COORD

Prerequisite: None

Open

The Open menu item allows you to open an existing .CRD or .CGC file. Only one coordinate file can be open at a time in a given drawing.

Prompts

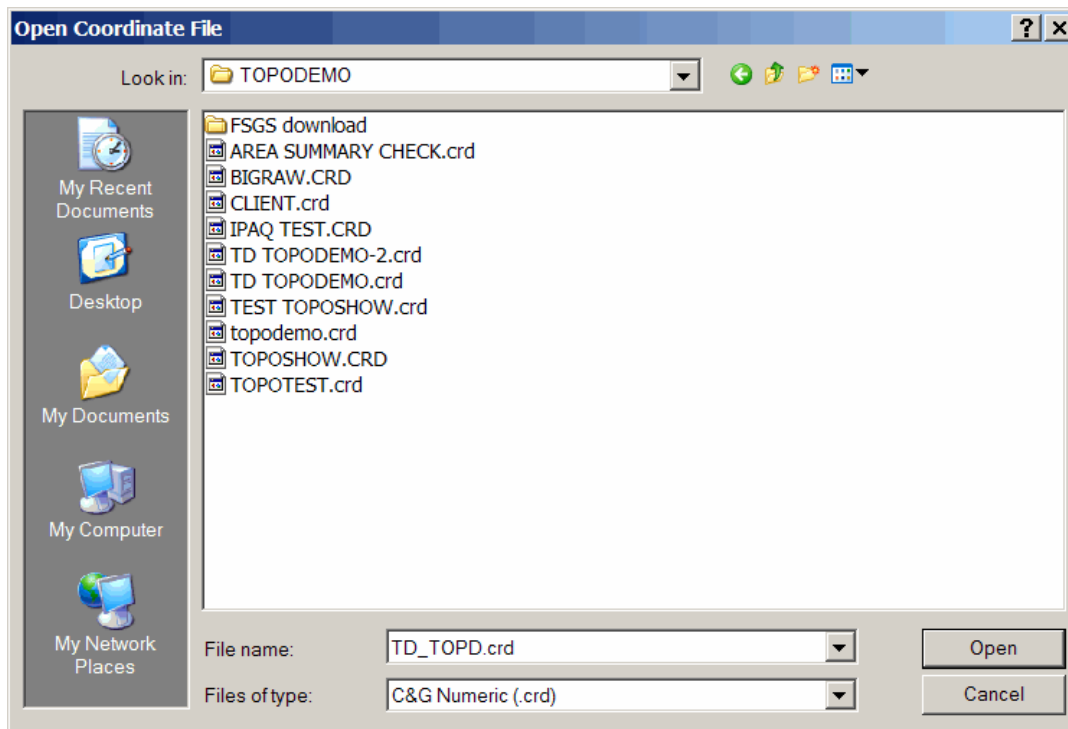
To open an existing coordinate file follow these steps:

In the file dialog box (Shown below),

Browse to folder location

select or Highlight the coordinate file you wish to open by clicking on it

Click the Open button



NOTE: The default directory is the "Data" directory below the directory where CG-SURVEY was installed. You can change the default directory by choosing:
 CG-Survey > Tools > CG Options... - Data Path tab.

Pulldown Menu Location: CG-Survey > File > Coordinate Files

Keyboard Command: OPC, CG_OPEN_COORD

Prerequisite: an existing coordinate file

Close

To close an open coordinate file

Pulldown Menu Location: CG-Survey > File > Coordinate Files

Keyboard Command: OPC, CG_OPEN_COORD

Prerequisite: Coordinate File Open

Save As

As new points are stored in a coordinate file, the file is automatically updated. If you are concerned that the changes to be made to the coordinate file may not be correct, you should use the Save As option to make an extra copy of the file before making any changes. This option allows you to save the open coordinate file under a different name. The new file becomes the current file. The original file will remain unchanged.

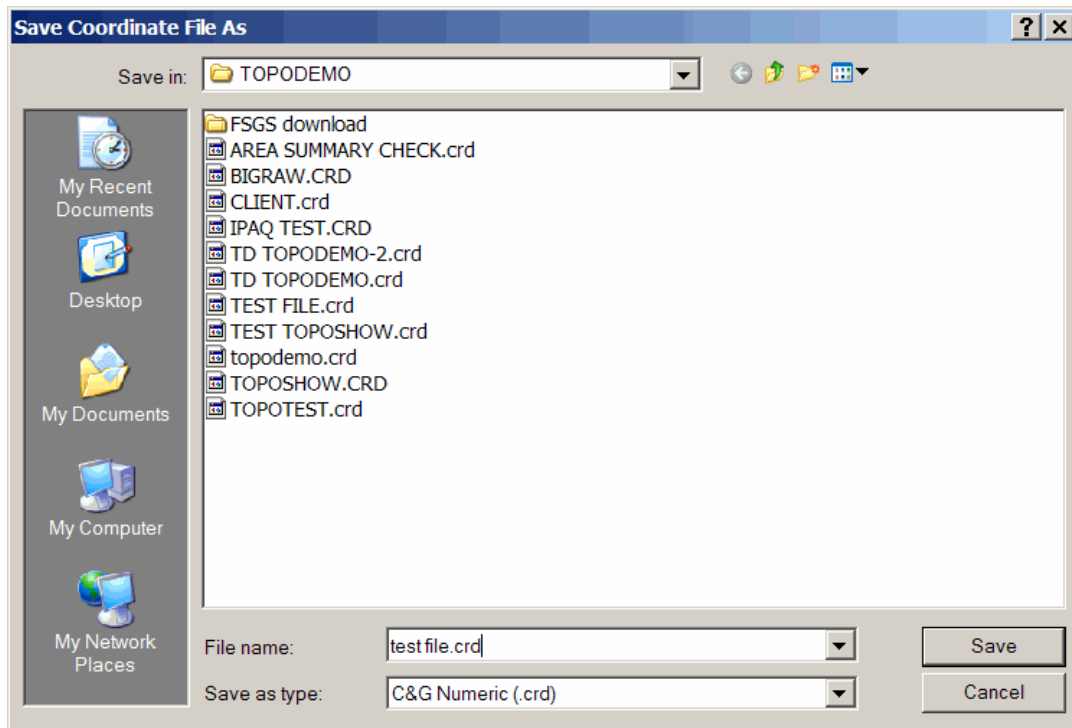
Prompts

To Save As the open coordinate file under a new name, select: > CGFile > Coordinate File > Save As

Browse to Folder Location:The Save Coordinate File As dialog box will display the default directory as set in the Data Path Options,

Enter the name : of the new file for the coordinates to be saved to.

Press Save Button: Save or Press enter



Pulldown Menu Location: CG-Survey > File > Coordinate Files

Keyboard Command: SCF, CG_FILE_SAVEAS

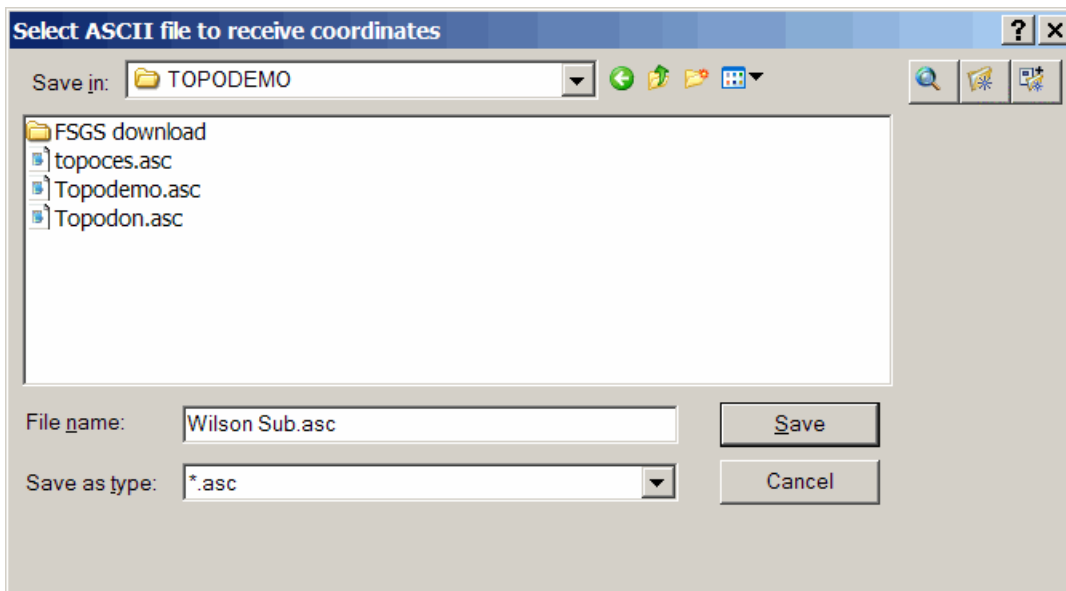
Prerequisite: None

Export Coordinates to ASCII

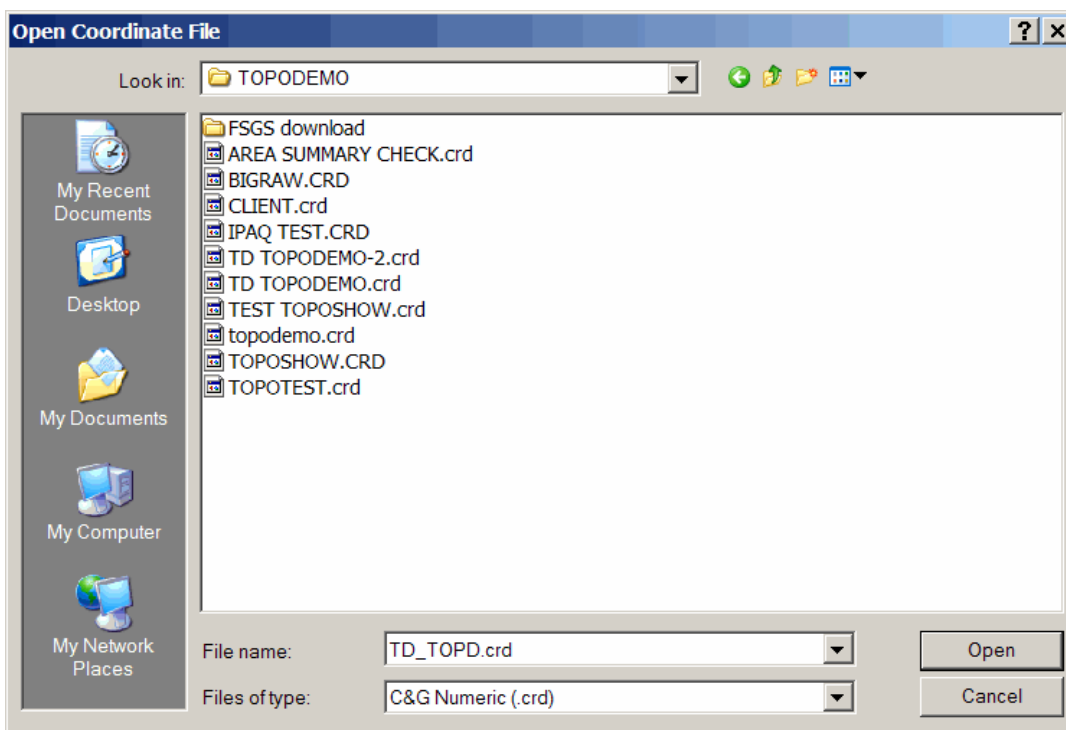
This menu item allows you to export coordinate files to an ACSII (American Standard Code for Information Interchange) file format. ASCII files are a simple text format and can be read by almost all word processors and text editors.

Prompts

To export coordinates to an ASCII file, follow these steps: > CGFile > Coordinate File > Export Coordinates to ASCII File

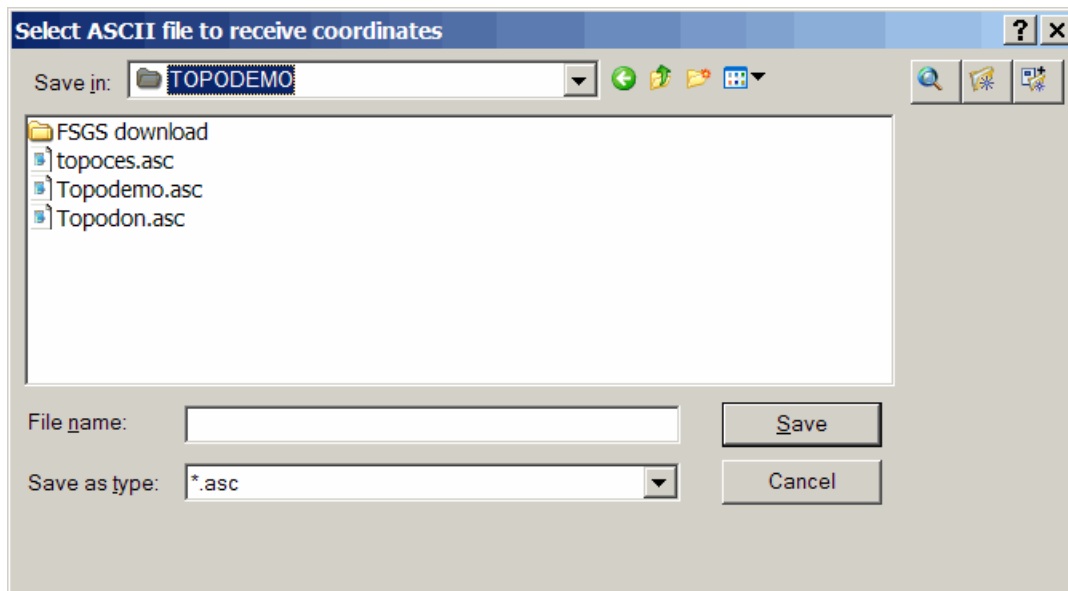


If a coordinate file is not currently open, the Open Coordinate File dialogue box will appear., select the file.

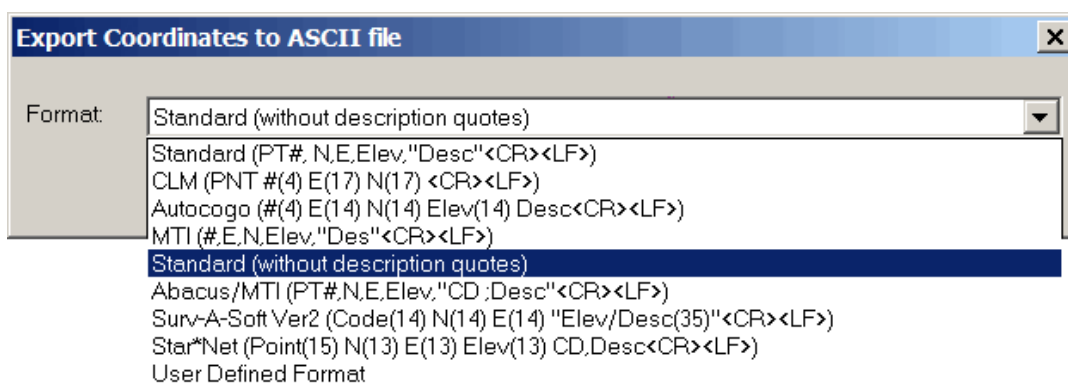


You will be prompted at the command line to select the points you wish to export:

Add points from coordinate file. (Enter When Done) (All/Block/Code/Desc/Elev/ Pt-group/Limits/Radius/Select):
 After choosing the set or sets of points you wish to export, press until the following dialogue box appears.



Enter a new file name or select an existing ASCII file and click the Save button. Next, select an ASCII file format (see the ASCII File Formats section of this chapter for an explanation of each format):



Select the OK button to export your coordinate points.

ASCII FILE CONVERSION FORMATS

NOTE: In the following formats the point code can be placed in the first two characters of the description field, followed by a semicolon. The description will follow the semicolon. You can export and import ASCII files in the following formats:

STANDARD (Point #, North, East, Elevation, "Desc")

```
2,5054.76393,9777.75761,103.70000,"gs"
3,5098.69743,9783.82411,105.20000,"gs"
4,5158.78043,9773.74111,105.67000,"gs"
5,5205.11493,9777.40661,106.25000,"gs"
```

CLM (PNT Point # Easting Northing)

```
PNT 2 9777.75761 5054.76393
PNT 3 9783.82411 5098.69743
PNT 4 9773.74111 5158.78043
PNT 5 9777.40661 5205.11493
```

Autocogo (Point # Easting Northing Elevation Desc)

2 9777.75761 5054.76393 103.70000 gs
3 9783.82411 5098.69743 105.20000 gs
4 9773.74111 5158.78043 105.67000 gs
5 9777.40661 5205.11493 106.25000 gs

MTI (Point #, Easting, Northing, Elevation,"Desc")

2,9777.75761,5054.76393,103.70000,"gs"
3,9783.82411,5098.69743,105.20000,"gs"
4,9773.74111,5158.78043,105.67000,"gs"
5,9777.40661,5205.11493,106.25000,"gs"

Standard (without description quotes) (Point #, North, East, Elevation, Desc)

2,5054.76393,9777.75761,103.70000,gs
3,5098.69743,9783.82411,105.20000,gs
4,5158.78043,9773.74111,105.67000,gs
5,5205.11493,9777.40661,106.25000,gs

Abacus/MTI (Point #, Northing, Easting, Elevation)

2,5054.76393,9777.75761,103.70000,"12;gs"
3,5098.69743,9783.82411,105.20000,"12;gs"
4,5158.78043,9773.74111,105.67000,"12;gs"
5,5205.11493,9777.40661,106.25000,"12;gs"

Surv-A-Soft (Code Northing Easting: Desc/Elevation")

6 0 "VER 2"
-1 0.00000 0.00000 " "
2 5054.76393 9777.75761 "103.70000"
2 5098.69743 9783.82411 "105.20000"
2 5158.78043 9773.74111 "105.67000"
2 5205.11493 9777.40661 "106.25000"
2 5253.39243 9779.12911 "110.47000"

The Surv-A-Soft file structure is as follows:

The first line of the file is a header line with the following information:

The total number of points is placed in the code field.

Zero (0) is placed in the northing field.

"VER 2", etc. is placed in the easting field.

After the header line each line specifies a coordinate point. The line number minus one is the point number.

The code field has three possible values:

Value Explanation

-1 no coordinate point

2 elevation (in description field)

1 description (in the description field)

Since .CRD and .CGC files can have both an elevation and description, when converting them to an ASCII Surv-A-Soft file one of the following will occur depending on the elevation value:

If the point has an elevation it will be placed in the description field. If there is no elevation, the description will be placed in the description field.

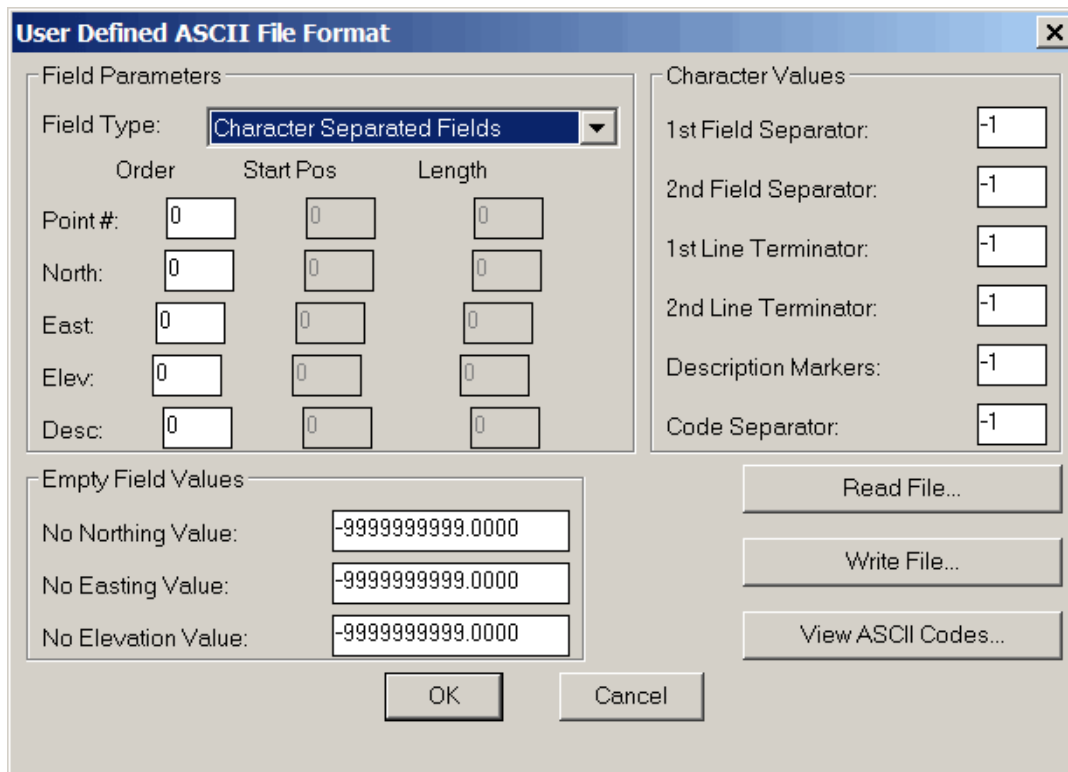
Star*Net (Point # Northing Easting Elevation Desc)

2 5054.76393 9777.75761 103.70000, gs

3 5098.69743 9783.82411 105.20000, gs
4 5158.78043 9773.74111 105.67000, gs
5 5205.11493 9777.40661 106.25000, gs

USER DEFINED

Upon selecting User Defined format, the following dialog box will appear:



The dialog box is titled "User Defined ASCII File Format" and contains the following sections:

- Field Parameters:** A dropdown menu for "Field Type" is set to "Character Separated Fields". Below it is a table with columns "Order", "Start Pos", and "Length".

	Order	Start Pos	Length
Point #:	0	0	0
North:	0	0	0
East:	0	0	0
Elev.:	0	0	0
Desc.:	0	0	0
- Character Values:** Five input fields for separators, all containing "-1":
 - 1st Field Separator: -1
 - 2nd Field Separator: -1
 - 1st Line Terminator: -1
 - 2nd Line Terminator: -1
 - Description Markers: -1
 - Code Separator: -1
- Empty Field Values:** Three input fields, all containing "-999999999.0000":
 - No Northing Value: -999999999.0000
 - No Easting Value: -999999999.0000
 - No Elevation Value: -999999999.0000
- Buttons:** "Read File...", "Write File...", "View ASCII Codes...", "OK", and "Cancel".

As the name implies you can create a format specific to your conversion needs.

Using an Existing User Defined Format To select an existing User Defined format, do the following: Press the Read File button. Select an existing user defined format from the dialog. The fields in the dialog box will automatically be updated. Select OK to accept the format.

Creating a User Defined Format: There are 5 basic pieces of information that can be defined in a user defined format. Point number Northing (required) Easting (required) Elevation Description (Code can also be part of the description field) There are two types of user defined formats " Character Separated Fields Character Separated Fields means that each field of information is separated by a character, often times a comma, but any ASCII character can be used. Fixed length Fields Fixed length fields means that you define the number characters for each field item. The fields can be in any order

Field Order

Point Number 4

North 3 (required)

East 2 (required)

Elevation 1

Description 5

NOTE: Coordinate values will be rounded based on the setting in the Rounding Options dialog box.

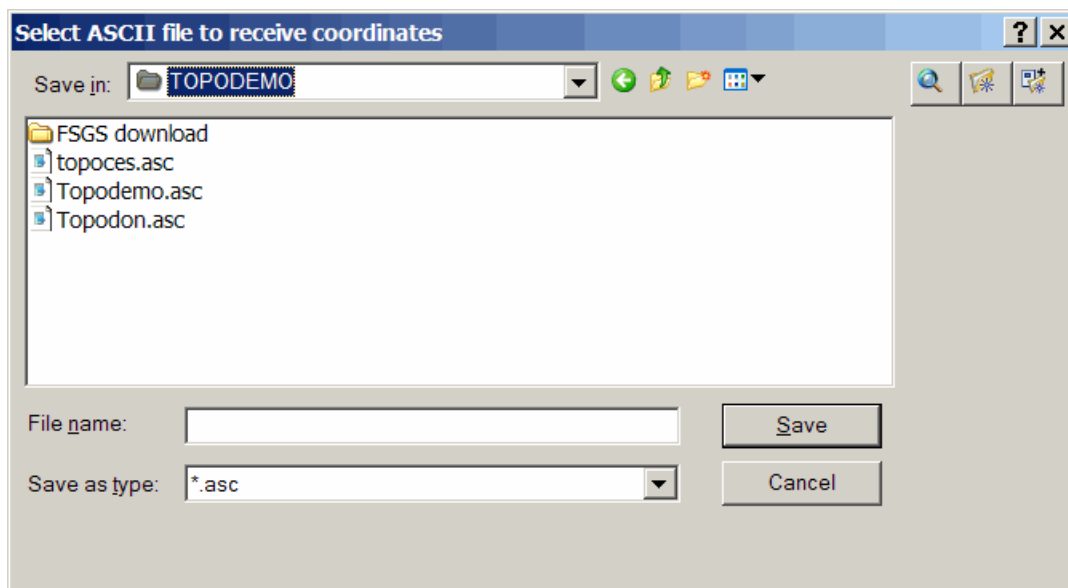
If the point number field is not assigned a value, the line number will be the point number. Select the Field Type (Character Separated or Fixed Length) and follow the appropriate instructions below:

Character Separated User Define Export File

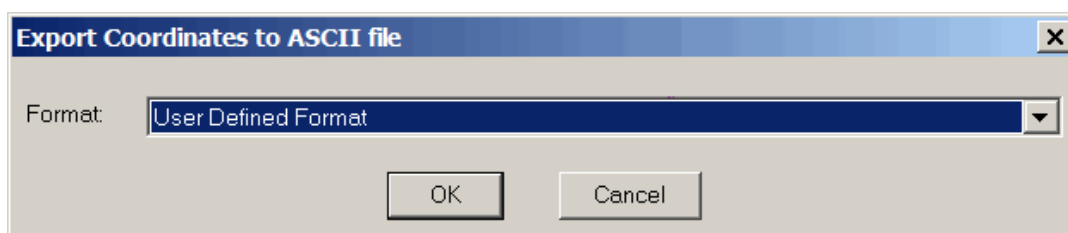
>Go to CGFile >Select Coordinate files >Select Export to ASCII At the command line you will be prompted to select points:

Add points from coordinate file. (Enter when Done) (All/Block/Code/Desc/Elev/ Pt-group/Limits/Radius/Select):

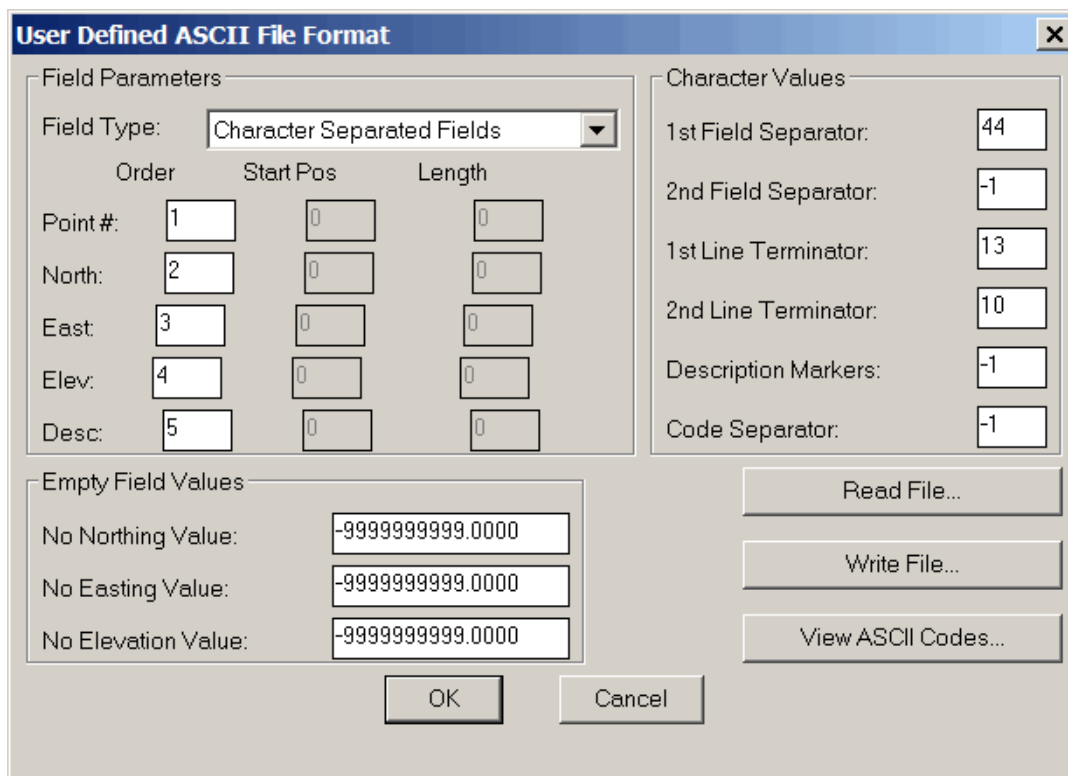
After selecting the point set or sets to export press return



Select or name the file to store the converted points.



Set the conversion format to "User Defined Format" The following dialog box allows you to define the attributes of the points being converted.



For this example Character Separated Fields has been chosen as the Field Type.

Points..... -> 1
 North..... -> 2
 East..... -> 3
 Elevation..... -> 4
 Description -> 5

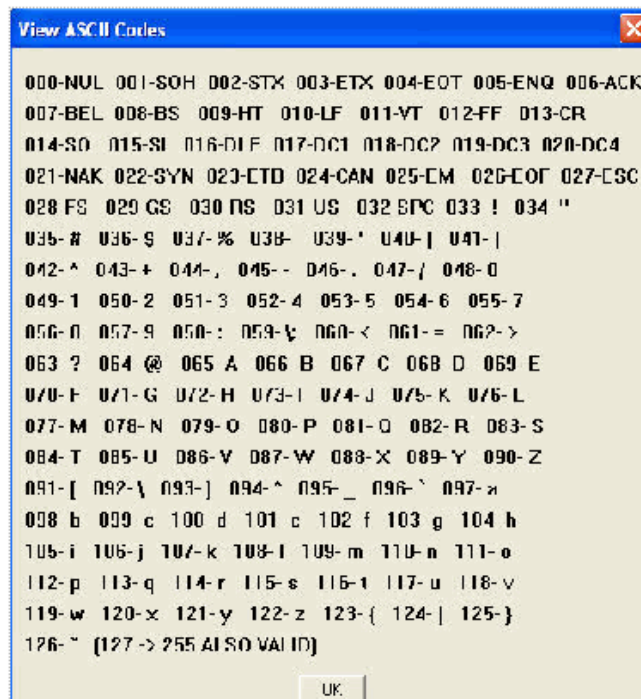
So the line data will be: Point # , North, East, Elevation, Description

Empty Field Values: It's necessary to distinguish between a field that has no value and a field that has "0" as a value. In coastal areas "0" is a valued elevation and in some cases "0" could actually be a coordinate value. By defining empty field values with a value that cannot be miss-understood for a valid value, a conversion process will not produce invalid data.

Character Values:

1st & 2nd Field Separators, these are the ASCII characters that define the fields within a line.
 1st & 2nd Line Terminator; these are the ASCII characters that define separate lines
 Description Markers; An ASCII character that surrounds the description such as quotation marks.
 Code Separator; Allows you to designate the ASCII character that separates the Code information from the Description information. In the example above the Character values are set as follows:
 1st Field Separator: 44 (which is a comma)
 2nd Field Separator: -1 (none used)
 1st Line Terminator: 13 (carriage return)
 2nd line Terminator: 10 (line feed)
 Description Marker: -1 (none used) Code Separator: -1 (none used)

This example would read as follows: 1,5000.0000,10000.0000,954.63,MH
 The following is a list of all of the ASCII codes and the respective values.



2nd Field Separator: You may however define two separators. For example, you can use a carriage return and line feed if you wish to have each field on its own line: 1 <CR> <LF> (Point number)
 1000.000 <CR> <LF> (Northing)
 1000.000 <CR> <LF> (Easting)
 954.56 <CR> <LF> (Elevation)
 MH <CR> <CF> (Description)

NOTE: Do not use a character as a separator if it appears in any of the fields. For example, if your record looks like this:

1 1000.000 1000.000 954.56 MH <CR><LF>

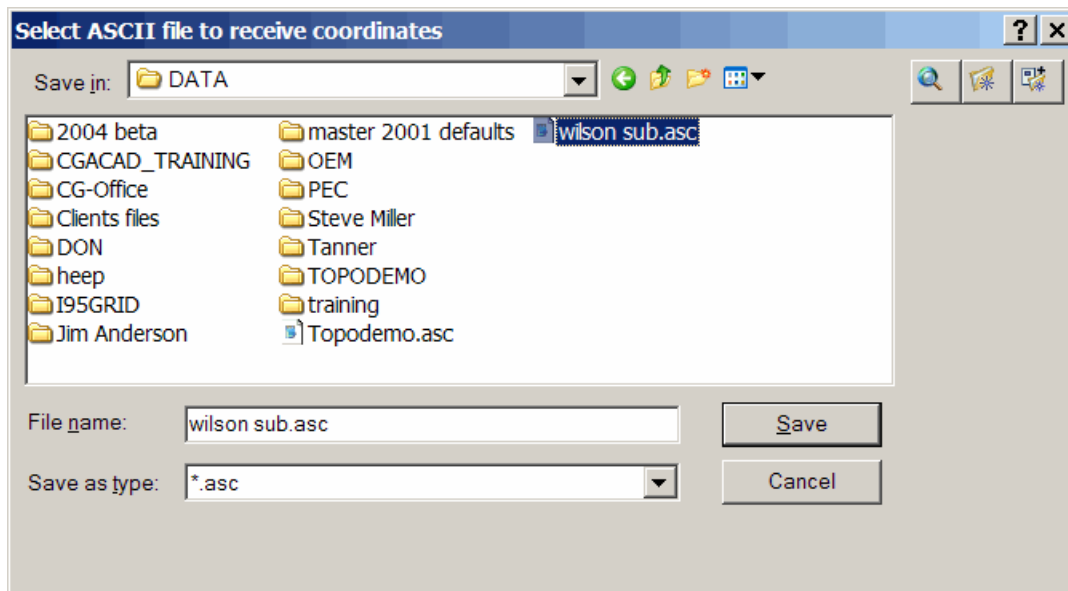
Then the period (.) character cannot be used as a separator because it is used in the northing, easting and elevation fields. Pressing the View ASCII Codes button will show you the 256 valid characters that can be used in an ASCII file. The table shows each character, with its integer value to the left of it.

NOTE: Character number 26 cannot be used as a field separate because it marks the End of File (EOF).

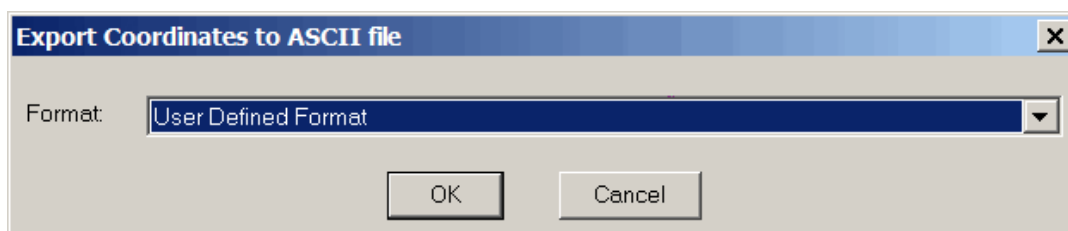
Fixed Length Field User Define Export File:

>Go to CGFile >Select Coordinate files
 >Select Export to ASCII Select points. (Enter When Done) (All/Block/Code/Desc/Elev/ Pt-group/Limits/Radius/Select):

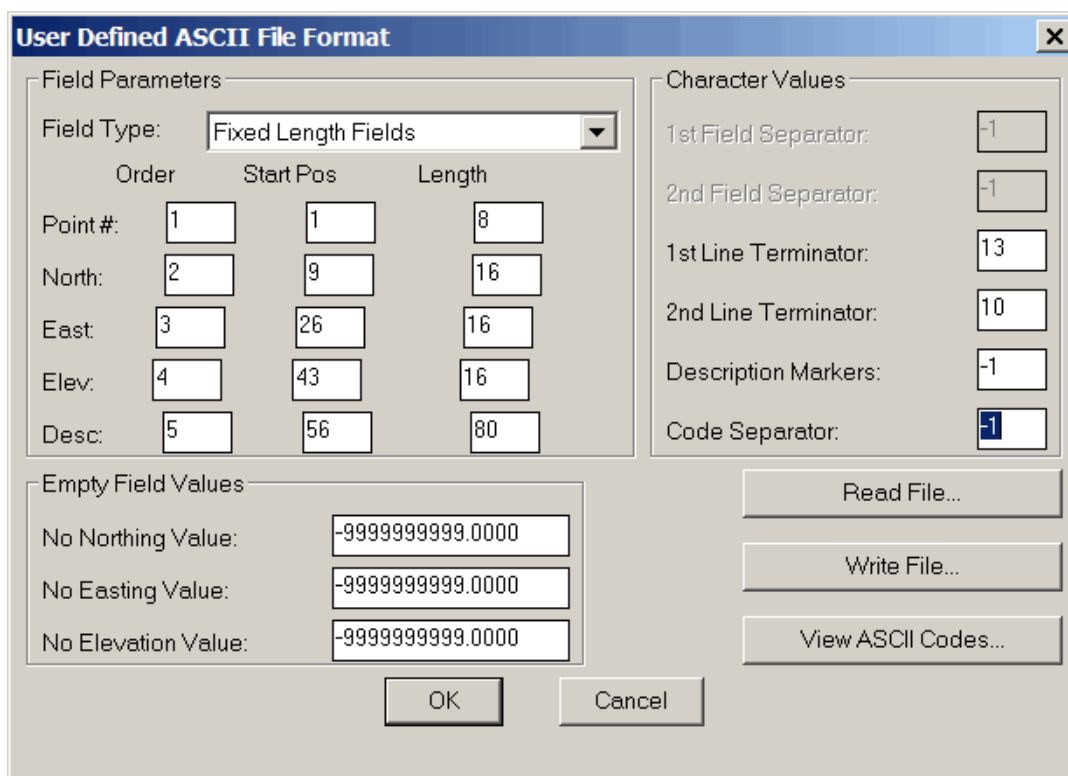
At the command line you will be prompted to Select Points After selecting the point set or sets to export press return Select or name the file to store the converted points.



Set the conversion format to "User Defined Format"



The dialog box below allows you to define the attributes of the points being converted.



For this example Fixed Length Fields has been chosen. In this case the order is set at:

Points..... -> 1
North..... -> 2
East.....-> 3
Elevation..... -> 4
Description -> 5

So the line data will be:

Point # - North-East-Elevation-Description But unlike Character Separated Fields, the information sets will be defined by their placement on the text line, rather than a separating character.

Empty Field Values: It is necessary to distinguish between a field that has no value and a field that has "0" as a value. In many cases around coast lines "0" is a contour elevation and in some cases "0" could actually be a coordinate value. By defining empty field values with a value that cannot be miss-understood for a valid value, any conversion process will not produce questionable data.

Character Values: 1st & 2nd Field Separators, do not apply, separators are defined by spacing.

1st & 2nd Line Terminator; do not apply, separators are defined by spacing.

Description Markers; An ASCII character that surrounds the description such as quotation marks.

Code Separator; Allows you to designate the ASCII character that separates the Code information from the Description information. In the example above the Character values are set as follows:

1st Field Separator: Do not apply

2nd Field Separator: Do not apply

1st Line Terminator: 13 (carriage return)

2nd line Terminator: 10 (line feed)

Description Marker: -1 (none used)

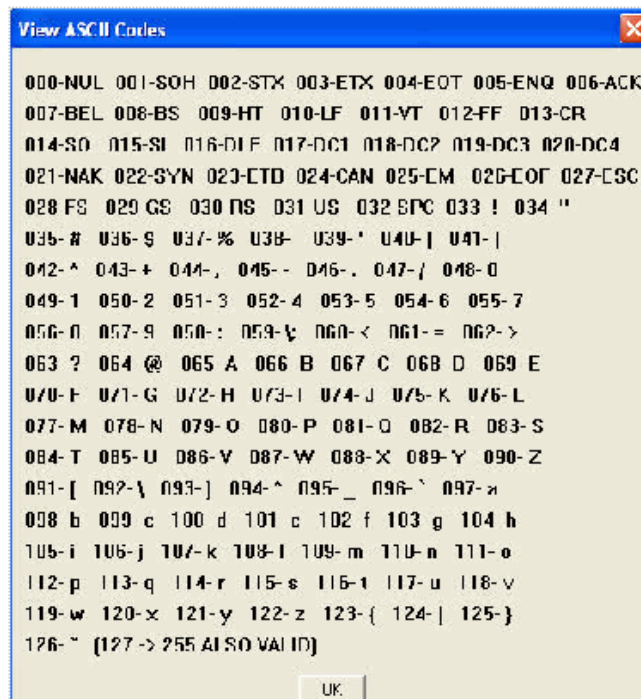
Code Separator: -1 (none used)

This example would read as follows:

1 5000.0000 10000.0000 954.63 MH <CR> <LF>

The first 8 spaces are the reserved for the point number The next 16 are reserved for the northing The next 16 are reserved for the easting The next 16 are reserved for the elevation The next 20 are reserved for the description Then a (carriage return) and a (line feed)

The following is a list of all of the ASCII codes and the respective values.



NOTE: Do not use a character as a separator if it appears in any of the fields.

Description Markers: If you have a description field, you may wish to use a Description Marker. This is a character that surrounds the description. For example, a description surrounded by quotes: 23,1056.789,2345.769,982.345,"MH" <CR> <LF>

If you are not using a description marker, enter -1 in the Description Marker box

Code Separator: If you have a description field, and want the first characters of the description field to be a C&G point code, you can enter the decimal value of the character that separates the point code from the description. This allows you to transfer both the point code and the description to an ASCII file. For example, using a semicolon as a code separator: 23,1056.789,2345.769,982.345,"MH; Inv Elev -9.23" <CR><LF>

If you are not using a code separator, enter -1 in the Code Separator box.

No Northing Value and No Easting Value

If the ASCII file does not have a point number field, the No Northing and No Easting values are mandatory. The record number will be used as the point number. This means that skipped point numbers will be filled with false northing, easting and elevation values.

Here is an example of a file with a record that has no point number field (assume you entered -999999 in the No Northing, No Easting and No Elevation boxes):

```
1056.789,2345.769,982.345,MH<CR><LF>Point 1
-999999, -999999, -999999, <CR><LF> No Point 2
2356.679,2455.645,992.678,MH<CR><LF>Point 3
2786.799,5645.789,984.234,MH<CR><LF>Point 4
```

No Elevation Value You must place a value in this box. When converting a C&G point to an ASCII point, this value will be placed in the elevation field of the ASCII point if a C&G point with "No Elevation" is encountered. When converting an ASCII point to a C&G point, if "No Elevation" is encountered in the ASCII point then "No Elevation" will be placed in the elevation field of the C&G Point.

Pulldown Menu Location: CG-Survey > File > Coordinate Files

Keyboard Command: EXC, CG_EXPORT_COORDS

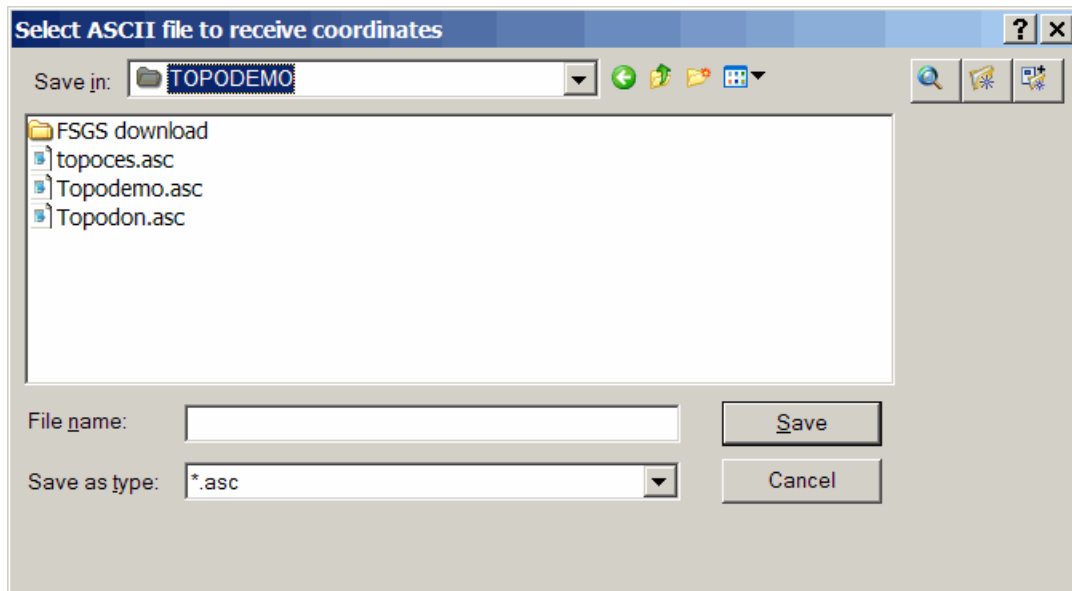
Prerequisite: None

Import ASCII File into Coordinates

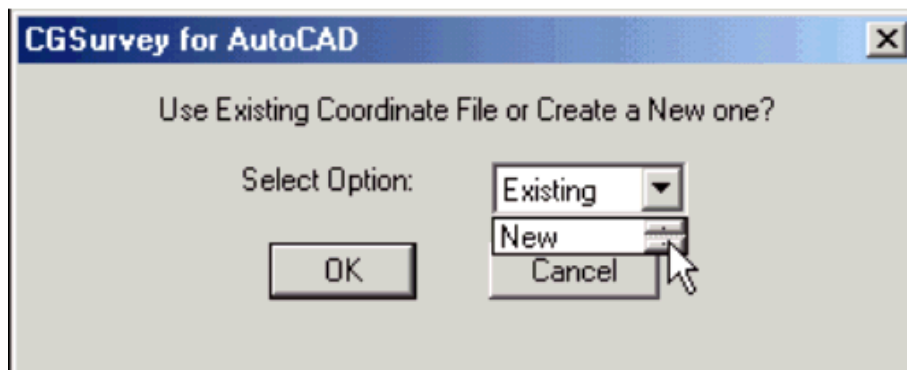
This option allows you to import the contents of an ASCII file into a coordinate file.

Prompts

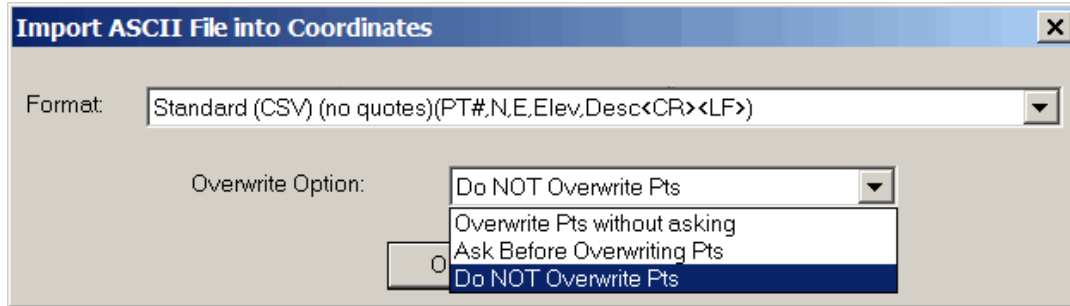
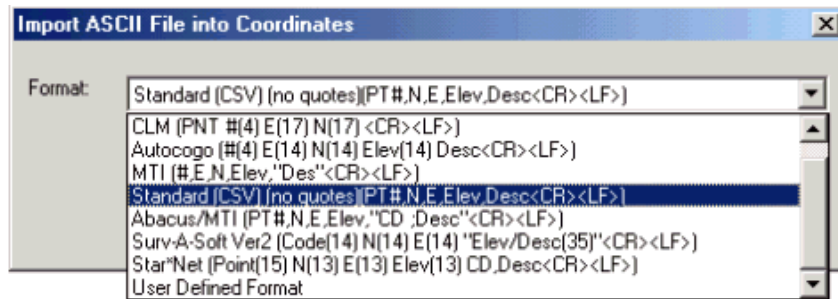
Follow these steps: > CGFile > Coordinate Files > Import ASCII File Into Coordinates



If a coordinate file is already open, the ASCII file will be imported into it, if a coordinate file is not open you will be prompted to open an existing file or create a new coordinate file.



Select the ASCII format that is being imported and how to handle duplicate points.



The points will be imported and displayed on the screen.

Pulldown Menu Location: CG-Survey > File > Coordinate Files

Keyboard Command: IMC, CG_IMPORT_COORDS

Prerequisite: None

Close Raw File

To close the current raw data file, select CGFile from the main menu and then select Close RAW File.

Pulldown Menu Location: CG-Survey > File

Keyboard Command: CLR, CG_CLOSE_RAW

Prerequisite: Raw File OPEN

Close Map Check File

To close the current map check file, select CGFile from the main menu and then select Close Map Check File.

Pulldown Menu Location: CG-Survey > File

Keyboard Command: CLM, CG_CLOSE_MAP

Prerequisite: Mapcheck file Open

CGDos Drawings

Before opening a CGDOS drawing you must choose the "setup" option to provide information needed for opening the PL1/PL2 files.



This feature allows you to import a CGDOS PL1/PL2 file and convert it to a standard CAD drawing. This is similar to a DXF conversion, but in addition to simple converting the graphics, this feature also retains the C&G data. That means that after the conversion is finished the drawing file is still referenced to the coordinate file. If you query a line it tells you what coordinate file the graphic was created from, the points that line is drawn from, the layer and line stop information just like query did in the CGDOS. This means you can continue working on the job after the conversion in a manner that is familiar to you as it was in CGDOS.

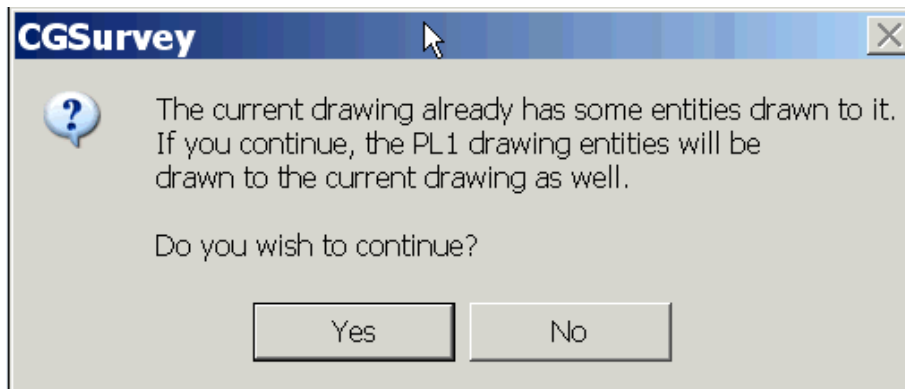
Pulldown Menu Location: CG-Survey > File

Keyboard Command: None

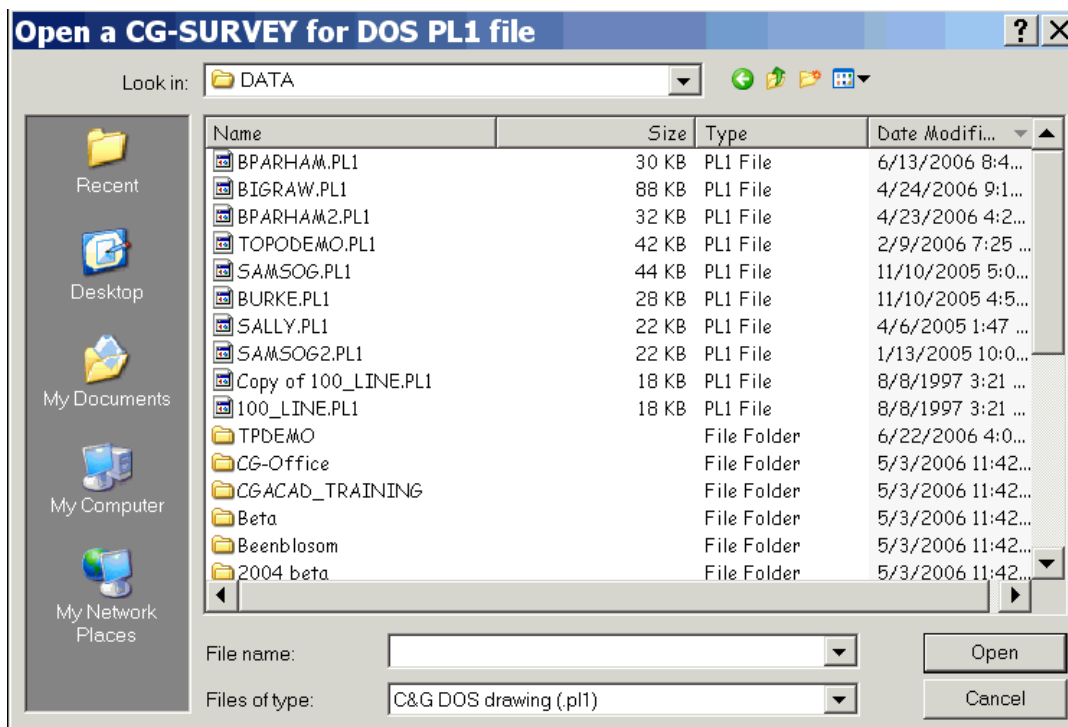
Prerequisite: CGDos Drawings>Setup

Open Dos Drawing

If the current drawing file you are in has any graphics the following dialog box will appear. This is meant to prevent you accidentally placing the PL1 drawing on top of another existing drawing file.



Selecting will bring up the following dialog box that will allow you to select the PL1 file to be converted to a standard CAD drawing.



After selecting the file to be converted, if you look at the command line you will see that the program is going through the PL1 file and converting the drawing entities one at a time to make them conform to the C&G format. This means all of the C&G data is maintained so the new drawing is still linked to the coordinate file it was created from. Also during this conversion process any of the CGDOS *.INS files (inserts) will be converted to standard CAD blocks and be added to the CG list of available inserts. Meaning all of the inserts you were accustomed to using in the CGDOS product will now be a part of the CG Survey program.

Prompts

Select a *.PL1 drawing file from browse file dialog box: Select file & click on OPEN button

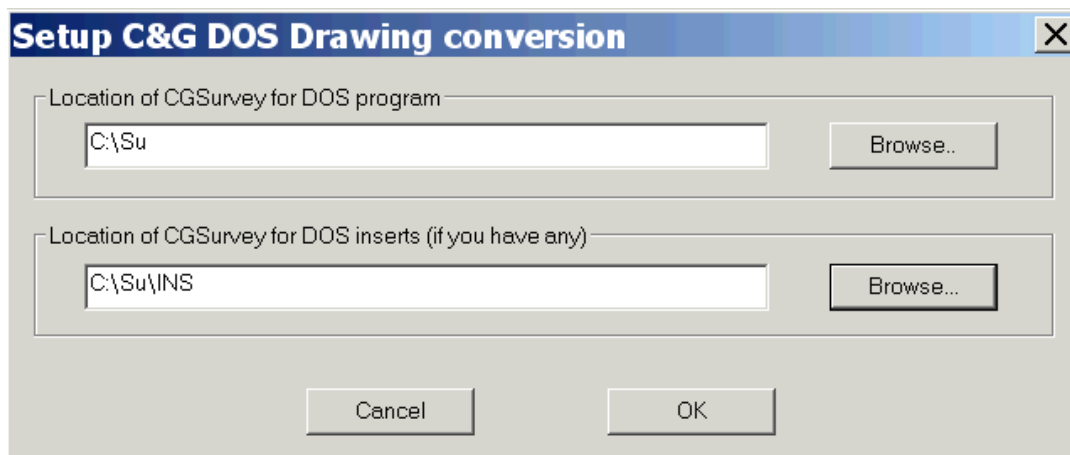
Pulldown Menu Location: CG-Survey > File

Keyboard Command: None

Prerequisite: CG-Survey > File > CGDos Drawings>Setup completed properly

Setup DOS Dwg

The first dialog asks you to give the path to the CGDOS Program Files and the path to CGDOS Inserts.



When this is set properly, any Insert used in the DOS PL1 file will be converted to a block and stored in the C&G symbols folder. These inserts will also be listed in the insert library when you go to: CGDraw > Drawing Settings > Active Symbol NOTE: Currently those inserts converted from the CGDOS PL1 files will not be shown graphically in the CGSurvey Active Point Symbol dialog box but they will appear in the symbols list and thus can be selected for use from the list.

Prompts

Select a Location for CGsurvey Program from browse file dialog box: Pick Browse button

Select a Location for Insert files from browse file dialog box: Pick Browse button

Pulldown Menu Location: CG-Survey > File

Keyboard Command: None

Prerequisite: None

Convert Old CG Dos Level File to New Format

This option converts old C&G DOS level files (files with a .LEV extension) to the new CGSurvey level file format (files with a .LEV extension).

Prompts

Select CGFile from the main menu.

Select Convert Old C&G DOS Level File to new Format from the pull-down menu.

From the file dialog box, select the file to convert:

Click the OPEN button to convert the file.

Pulldown Menu Location: CG-Survey > File

Keyboard Command: CVL, CG_CONVERT_DOS_LEVEL_FILE

Prerequisite: None

Convert Old CG Dos Raw File to New format

This option converts old C&G DOS raw files (files with a .RAW extension) to the new CGSurvey raw file format (files with a .CGR extension).

Prompts

Select CGFile from the menu bar.

Select Convert old C&G Raw File to new format from the pull-down menu.

From the file dialogue box, select the file to convert:

Click the OPEN button to convert the file.

Pulldown Menu Location: CGFILE

Keyboard Command: CVR, *CG_CONVERT_RAW

Prerequisite: None

Convert Old CG Dos Cross Section File to New Format

This option converts old C&G DOS Cross Section files (files with a .EW extension) to the new CGSurvey earthwork files format (files with a .CEW extension).

Prompts

Select CGFile from the menu bar.

Select Convert Old C&G Cross Section File to new format from the pull-down menu.

From the file dialog box, select the file to convert: Click the OPEN button to convert the file.

Pulldown Menu Location: CG-Survey > File

Keyboard Command: CVX, CG_EW_CONVERT_FILE

Prerequisite: None

Convert Old CG Dos Template File to New Format

This option converts old C&G DOS Template files (files with a .TPL extension) to the new CGSurvey earthwork files format (files with a .CTP extension).

Prompts

Select CGFile from the menu bar.

Select Convert Old C&G Cross Section File to new format from the pull-down menu.

From the file dialog box, select the file to convert:

Click the OPEN button to convert the file.

Pulldown Menu Location: CG-Survey > File

Keyboard Command: CVT, CG_EW_CONVERT_TEMPS

Prerequisite: None

Empty Print File

Choosing this menu item will remove all the text now in your print file.

You should empty the print file periodically so that it does not use too much of your disk space and become difficult to view and print.

Note: If user wishes to change the Printer.Txt file name or choose a different location. see CG-Survey > CG Options... - Output Tab

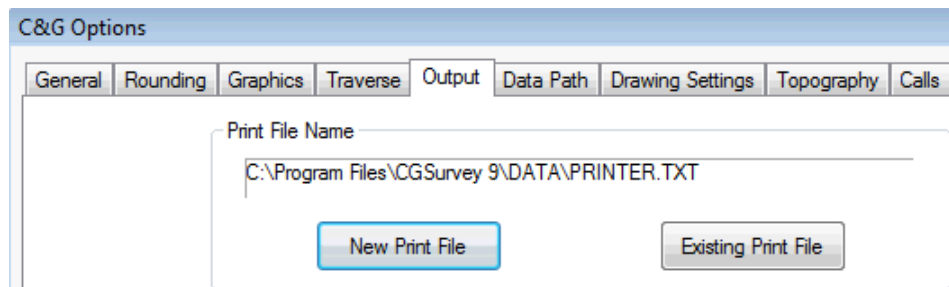
Pulldown menu Location: CG-Survey > File

Keyboard Command:EPF, cg_df

Prerequisite: Set print file name and path in CG-Survey > CG Options... - Output Tab

Print View Print File

While computations are taking place a Print File is being maintained showing all computations. This file is saved in the text file specified in the Output Options dialog box below

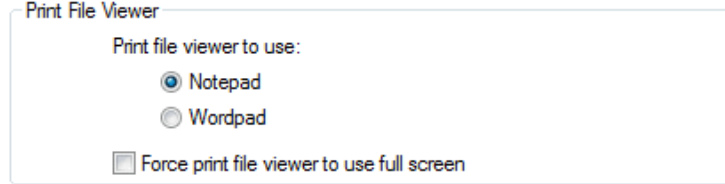


This text file may be edited, printed or viewed from any text editor or any word processor. (Note: For further explanation on Output Printing Settings please consult CGTools Menu)

After choosing the Print/View Print File menu item, the print file will be opened using the Windows text editor WordPad. To print the whole file, use the printer icon or the Print menu item on the WordPad File menu. To print a portion of the print file, you must highlight the portion you wish to print, then choose File > Print. On the General tab of the Print dialog box click the Selection radio button then click the Print button to print the highlighted text.

NOTE: Print Preview is also available on the WordPad File menu.

You can choose whether to use the Windows Notepad or Wordpad to view and print the print file by going to the CG-Survey > CG Options... menu and clicking the Output tab then clicking on either the Notepad or Wordpad radio buttons in the Print File Viewer section of the dialog (shown below).



Pulldown menu Location: CG-Survey > File

Keyboard Command:VPF, CG_VIEW_PRINT_FILE

Prerequisite: Set print file name and path in CG-Survey > CGOptions... - Output Tab

CGTrav

Quick Traverse

This feature allows you to utilize the keyboard and the mouse to perform a traverse using points and data found in the drawing and the coordinate file. There is no raw data entry associated with Quick Traverse. The Quick Traverse feature has no ability to adjust the resulting traverse. If you wish to adjust coordinates, you could create a raw data file using the CGEditor - on the CGTrav menu - then use the Reduce Traverse feature, also on the CGTrav menu.

NOTE: If you wish to check the closure of a plat from bearing and distance data, use the CGEditor to create a map check file, then use the Reduce Map Check File feature on the CGTrav menu.

Prompts

During the process of entering data for the Quick Traverse feature you will see the prompt:

[aZimuth/Bearing/Deflection/Side shot/cUrve/Closure/horiz. distaNces]

At this prompt you may:

Change the type of angular input between Horizontal Angle, Azimuth and Bearing modes at any time.

Change how distances are specified as either slope distance and vertical angle or horizontal distance and vertical distance.

Turn the vertical angle input on or off.

Traverse around tangent and non-tangent curves.

Switch from Traverse to Side shot mode.

Traverse mode: automatically occupy the foresight point.

Side shot mode: continue to occupy the current instrument point until you change to

Traverse mode: and thus occupy another point.

Note: There are several settings found in the C&G Options dialog box that should be set or checked prior using the Quick Traverse feature:

The default values for the initial traverse input modes are set in the Traverse Options.

If you wish to calculate or enter elevations, check the **Elevations** : ON checkbox and choose Enter Elev. Or Calculate Elev. as desired in the Global Options tab. If you are calculating elevations, make sure the Vertical Angles ON checkbox is checked on the Traverse Options tab.

Quick Traverse Example

In this example the mode is set to traverse and elevations are on and are to be calculated.

After choosing Quick Traverse from the CGTrav menu you will be asked to enter the following information:

Instrument point: for the example enter 1 (assuming that the currently open coordinate file has a point in it with a point ID of 1).

Backsight point: for the example enter 2.

[aZimuth/Bearing/Deflection/Side shot/cUrve/Closure/slope distaNces.]
Enter horizontal angle <0.0000>:

Since elevations are on and set to calculate so you will be prompted for the following:

If you selected H.I. as Plus-Up on the Traverse Options tab, the coordinates and elevation of the instrument point will be read from the file and you will be prompted for the instrument height (H.I).

If no elevation is found, you will be prompted to enter the ground elevation at the instrument point and then the H.I.

If you selected H.I. as Elevation in the Traverse Options dialog box, you will be asked to enter the actual elevation of the instrument scope.

Backsight Point: If you are turning angles or deflection angles instead you will be prompted for the back-sight point.

Rod Height: With Calculate Elevations on you are prompted to enter the prism height.
You will be prompted for the horizontal angle (or deflection angle)

If you need to change the prism height <esc> and you will be prompted for a new prism height, if you <esc> again you will be prompted for a new instrument point.

Angle data entry

Instrument point: 1

Back site: 2

aZimuth/Bearing/Deflection/Side shot/cUrve/Closure/slope distaNce.] S

Enter horizontal angle <0.0000>:

When you are entering Quick Traverse data you have the options to change the angular input method. To change the angular input mode, enter the upper case letter seen in the prompt for the method of entry you want to change to and press <Enter>. The prompt should then change to reflect your choice.

Note: You need not use the shift and type a capital letter to choose a command line option. For example, to change to Side shot mode you can type either s or S.

The method that is currently set will not be shown as an option in the command line prompts. For example,

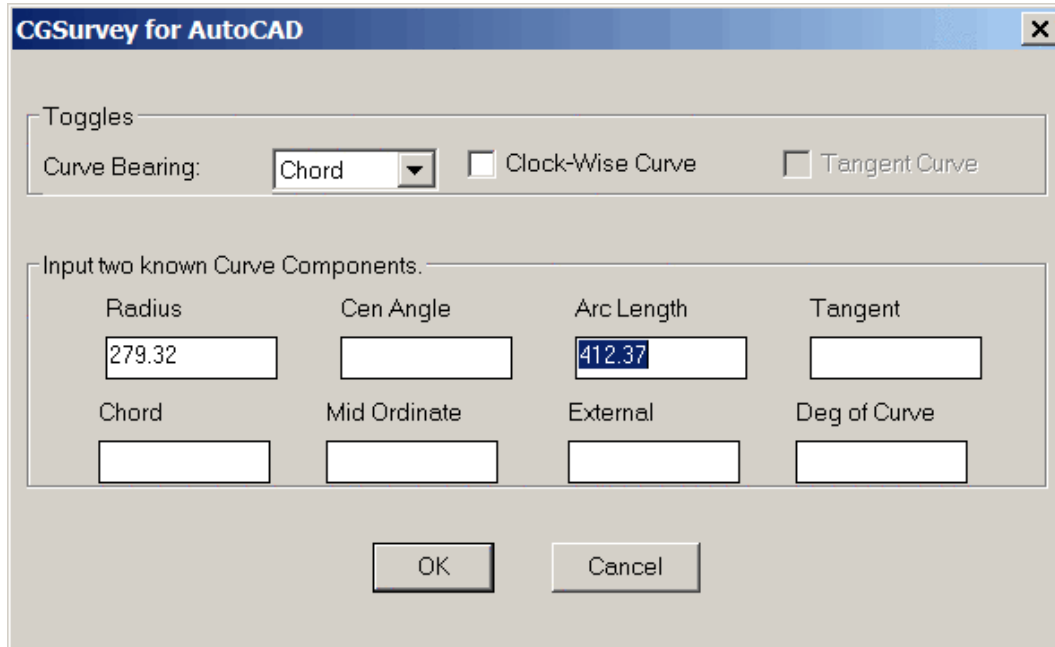
if you type s and <Enter> for Side shot mode, the prompt will change to include Traverse and Side shot will no longer be available since you have chosen it as the current mode.

Traversing a curve

The Traverse routine allows you to traverse both reverse and compound curves.

Note: You will not be allowed to traverse around a curve if calculate elevation is selected.

If you type U and <Enter> for cUrve, the following dialog box appears: U



The dialog box is titled "CGSurvey for AutoCAD" and contains the following elements:

- Toggles:**
 - Curve Bearing: Chord (dropdown menu)
 - Clock-Wise Curve
 - Tangent Curve
- Input two known Curve Components:**

Radius	Cent Angle	Arc Length	Tangent
279.32		412.37	
Chord	Mid Ordinate	External	Deg of Curve
- Buttons:** OK, Cancel

Enter any two of the curve components.

Identify the curve bearing as Chord if the angle, deflection, bearing or azimuth about to be entered is to the PT.

Identify the curve bearing as Radius if the angle, deflection, bearing or azimuth about to be entered is to the radius point.

Click the Clockwise box if the curve is clockwise. If this box is not checked, the curve is considered to be counterclockwise.

If there is a previous traverse leg, check the Tangent Curve checkbox if the curve is tangent to the previous leg. If this checkbox is not checked, the curve is assumed to be non-tangent.

When you have entered the required data: click the OK button.

The input multiplication factor is applied to the curve data you enter (radius, arc length, chord, etc.).

At the next prompt, if the curve is a non-tangent curve, enter the angle, deflection, bearing or azimuth from either the PC to the PT or the PC to the radius point (depending on whether you set Curve Bearing to Chord or Radius). If the curve is tangent to the previous traverse line you will not be asked for the angle and distance.

The curve data will be calculated and shown at the command line:

Bearing and distance from the PC point to the radius point.

Bearing and distance from the radius point to the PT point.

Bearing and distance from the PC point to the PT point.

Other curve information.

The radius and PT points will be stored in coordinate file using the STORING POINT prompt.

Closure

At the prompt: aZimuth/Bearing/Deflection/Side shot/cUrve/Closure/slope distaNce.]

Enter horizontal angle <0.0000>:

Type C and <Enter> to view closure information for the traverse to the current foresight.

Slope/Horizontal Distance Data Entry

If you have selected Slope Dist/Vert. Angle in the Traverse Options tab or switched to slope distances by typing N and <Enter> at the command line, enter the slope distance. Otherwise, enter the horizontal distance.

Note: The following steps are required only if Vertical Angles ON is checked on the Traverse Options tab or if Calculate Elev. was selected on the Global Settings tab.

For slope distance - vertical angle:

Enter the vertical angle.

Depending on the settings in the Traverse Options tab enter one of the following:

Zenith (zero up)

Nadir (zero down)

Transit (zero level)

Transit vertical angles can be full circle (0 - 360), or positive for up and negative for down.

For horizontal distance - vertical distance:

Enter the vertical distance.

Pulldown Menu Location: CGTrav

Keyboard Command: QTR, CG_QTRAV

Prerequisite: Open Coordinate File

Edit Raw File

The Edit Raw File feature allows you to use the CGEditor to create a new raw data file, append to an existing raw data file, or edit an existing raw data file. For further and complete information on using the Edit Raw File see the chapter on CGEditor in the Tools section.

CGEditor General Information

The CGEditor is an integral part of preparing files for use in C&G applications. The CGEditor is a very powerful tool. You can open multiple data files of any supported file type and edit the files as needed. The CGEditor has a full complement of tools for searching and replacing and navigating within a file. It will also allow you to cut or copy records from one file and paste them into another file in order to merge files, move data between phases of a job, etc.

The CGEditor can create and/or edit six types of data files used by C&G:

Raw Data Files

Raw data files contain information pertaining to a field traverse. Raw data files are typically downloaded from the data collector and converted to the C&G raw data file format. These files have the extension .CGR.

Map Check Files

Map Check files contain bearing, distance and curve information and are typically used to calculate the closure of a deed description. These files have the extension .CGM.

Cross Section Files

Cross Section files contain one or more cross sections identified by their station along the alignment. Each cross section record has the percent grade defined for its left and right slopes. Following the "Station" record are several "Point" records containing the elevations and offsets of the points along the cross section. Cross section files consist of a pair of files; the main data file has the extension .CEW and the index file has the extension .CEX.

Template Files

Template files are merely cross section files that represent a standard cross section and can be used to generate other cross section files. However, unlike cross section files, template files use an integer ID instead of a station to uniquely identify each template. Like cross section files, the percent grade is defined for the left and right slopes of each template and there are a set of "Point" records specifying the template elevation at a given offset. The centerline elevation at offset 0.00 is typically set to 0.00. Template files consist of a pair of files; the main data file has the extension .CTP and the index file has the extension .CTX.

Point Group Files

Point Group Files (formerly called batch point files) are simply a list of point numbers that can define a group of points, a lot/parcel of land, or an alignment. These are ASCII files and have a .PTS extension.

Coordinate Files

CGSurvey supports many different coordinate file formats:

C&G .CRD/.IDX - C&G numeric coordinate files

C&G .CGC/.CGX - C&G alpha-numeric coordinate files

Carlson .CRD - Carlson coordinate file format, numeric and alpha-numeric

Simplicity .ZAK - Simplicity coordinate file

LDT - MDB - Land Desk Top coordinate file

Pulldown Menu Location: CGTrav\Edit Raw File

Keyboard Command: ET, CG_EDIT_RAW

Prerequisite: Open Raw File

Data Collector Transfer

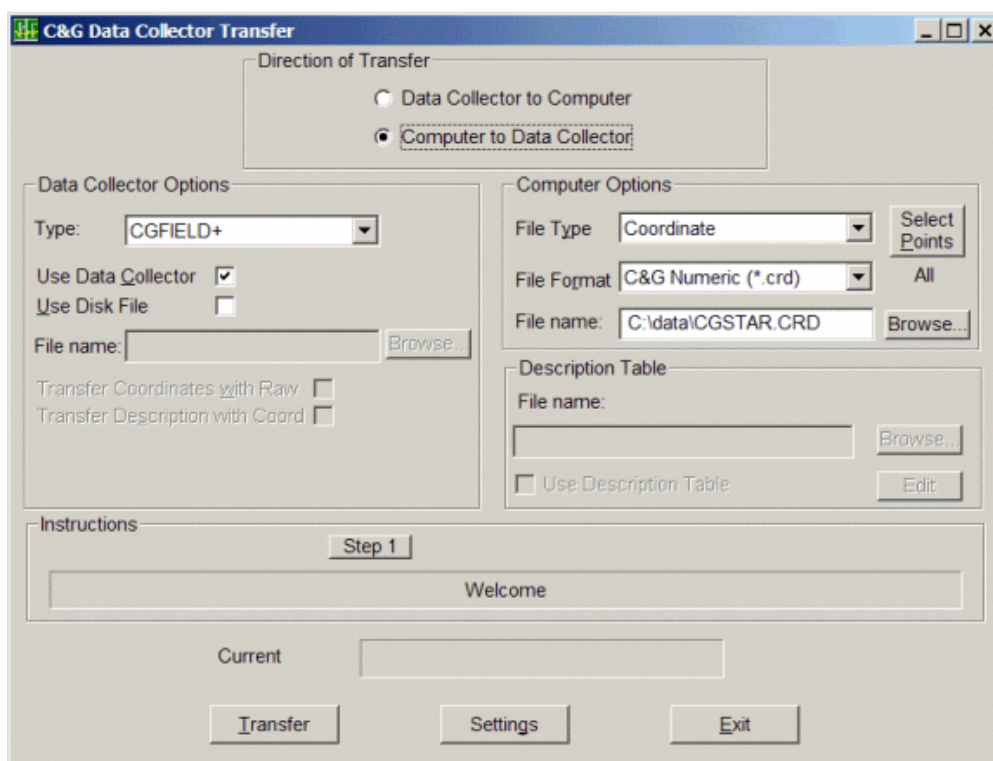
The Data Collector Transfer program allows transfer of data to and from the data collector. The program may also be used to convert raw data and coordinate files to the supported formats.

There are two variables that affect the interaction between your data collector and CGSurvey. One is the data collector itself and the other is the software you use in the data collector. This section provides information on the use of data collectors and software that will interact with CGSurvey.

NOTE: This manual is not a substitute for your data collector manual.

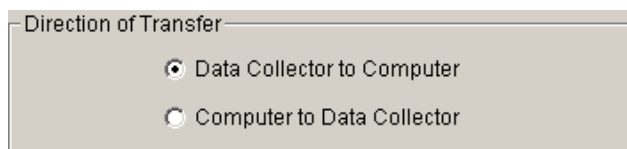
GENERAL INFORMATION ON USING DATA COLLECTORS

Before using the data collector program, make sure the correct data collector, communication port and communication parameters have been selected in the Settings dialog box.



Direction of Transfer:

Choose either "Data Collector to Computer" or "Computer to Data Collector".



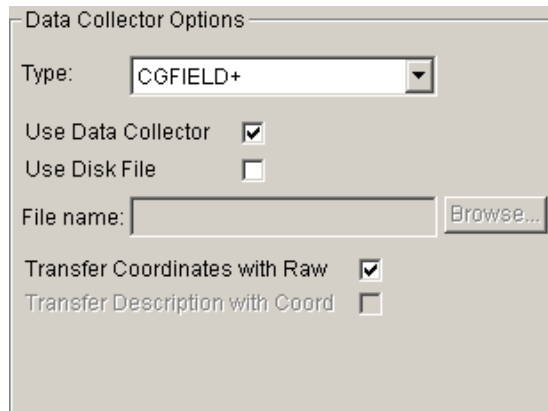
Data Collector and Computer Transfer Options

Instructions: Press the STEP 1 button. Depending on the type data collector, type file and direction of transfer, this option will give you step by step directions on how to proceed.

The Transfer dialog is divided into two sections, left and right. The left part of the dialog box pertains to "Data Collect Options" such as file source, file format and the file being transferred. The right part of the dialog box pertains to the Desk Top "Computer Options". Below are instructions for setting both.

Data Collector Options

Pressing the triangle to the right of the edit box will bring up the list of data collectors to choose. From the list select the type of data collector being used.



Use Data Collector:

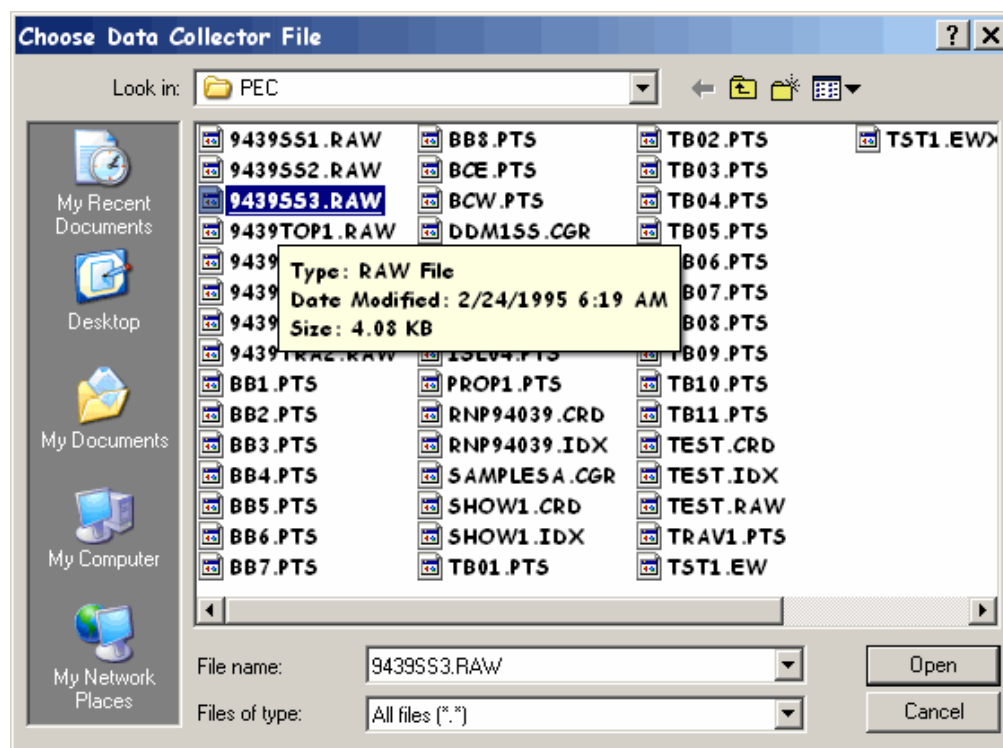
Check this box to transfer data to/from the data collector. You can also transfer to/from a file in the selected data collectors format.

Use Disk File:

Check this box if the data is in computer file. The data file must be formatted for the data collector selected.

File Name:

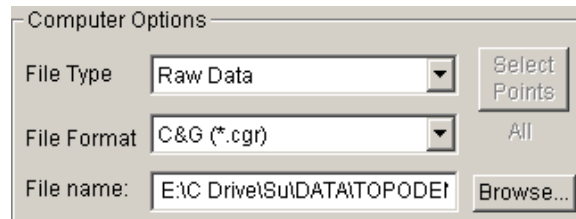
If you are importing from a file, or exporting to a file, or are connecting to a data collector that requires a file name for transfer, the File Name edit field will be active. To select the file path click on browse. In the file dialog box specify the path and file name of the file to be opened. select or enter the path and file name of the file desired file.



Transfer Coordinates with Raw:

Some field software allows unadjusted coordinates to be carried in the raw file as the field data is collected. This checkbox gives you the option to transfer this data or not. If you do not want approximate coordinates that were calculated in the field to be confused with control when processing the raw data, leave this box unchecked.

Computer Options



File type: Choose the file type you are transferring/converting. Example: Raw Data, Coordinate, ASCII, etc.

File Format: C&G will import and export several types of file formats for both Raw and Coordinate files.

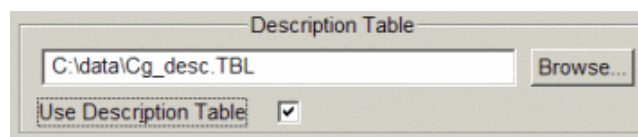
Supported Raw Data File Formats:

New CGR *.cgr
 Old C&G *.raw
 OBS.....*.obs
 Geolab *.iob
 StarNet *.dat
 SDR2x *.dat
 SDR33 *.dat

Supported Coordinate File Formats:

C&G AlphaNumeric*.cgc / cgi
 C&G *.crd / idx
 Carlson AlphaNumeric *.crd
 Carlson Numeric *.crd
 ASCII *.nez
 ASCII *.asc
 Geolab *.neo
 StarNet..... *.pts
 SDR2x *.dat
 SDR33*.dat
 Simplicity..... *.zak
 LDT..... *.mdb

Description Table:



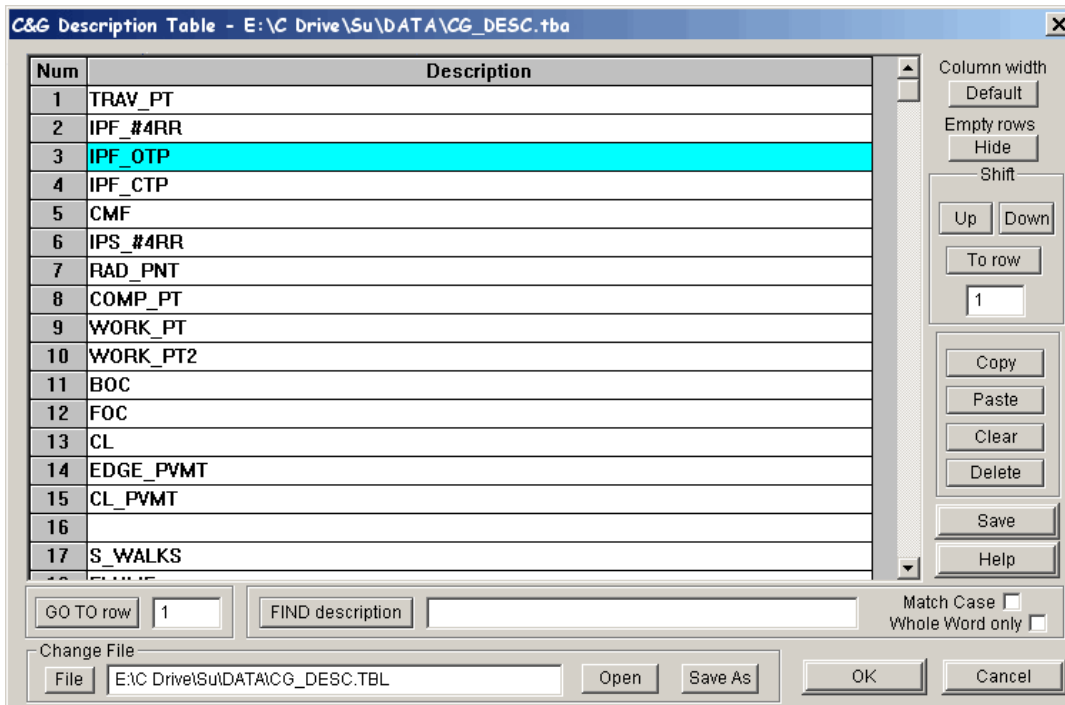
To use a description table check "Use Description Table" box.

You have the ability to use multiple description tables. Examples of that might be:

- State DOT
- Boundary
- As-Built
- Topography

Each of these could have different codes and descriptions and this option would allow you to choose which description table to use for the reduction of this file. To change the description table click the "Browse" button and select the TBL file.

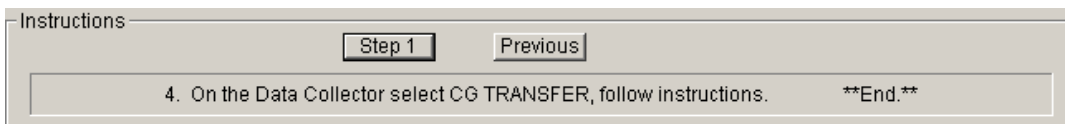
Below is an example of a description table:



When using a description table, any INTEGER numbers in the description field of the data coming from the data collector will be replaced by the description in the table. For example, if your description is "13 5", the description put in the coordinate or raw data file will be "CL CMP".

Transfer

Instructions: The instructions window will guide you step by step through the transfer routine. It will tell you what to do on the data collector, and in what order.

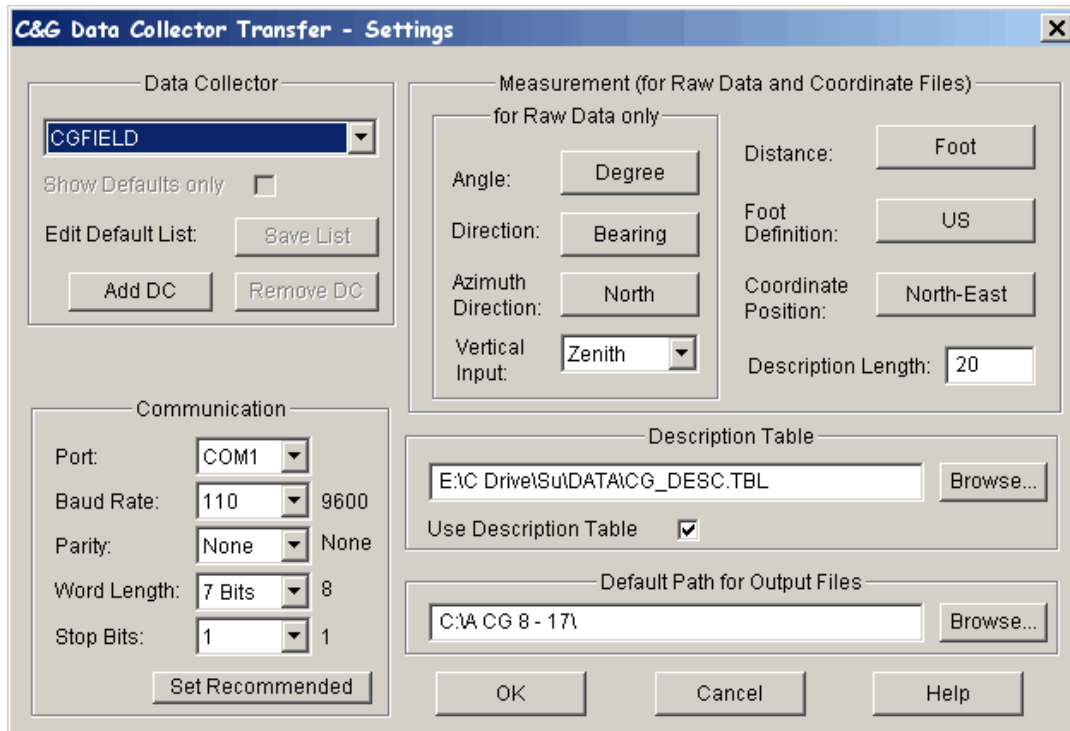


Transfer: Once all of the settings are set correct, clicking on the TRANSFER button will begin the transfer between the data collector/file and the desktop.

The **Current Status** window at the bottom of the Transfer Dialog will indicate the status of the transfer.

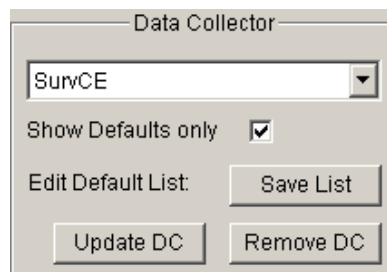
Settings

At the bottom middle of the main screen is the "Settings" button. The settings control communications, data units and output data path.

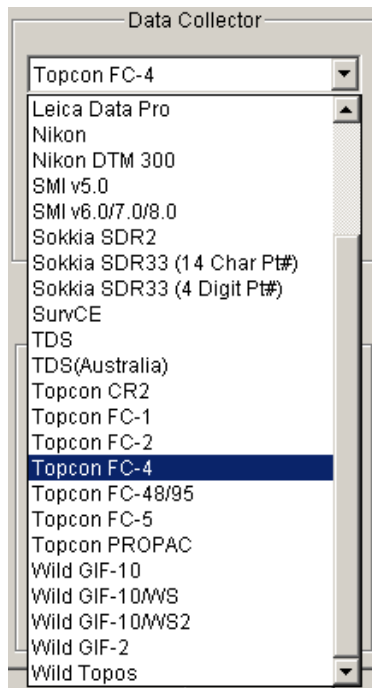


Data Collector

The Data Collector dialog box allows you to select a short-list of data collectors you are transferring to and/or from.

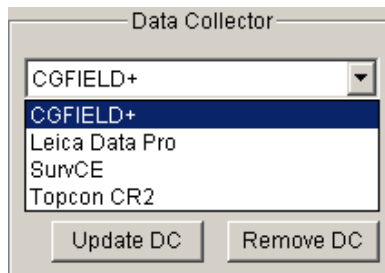


When you select the down button to the right of the data collector shown and the "Show Defaults only" box is unchecked, you will see the complete list of all the data collectors that C&G interfaces with.

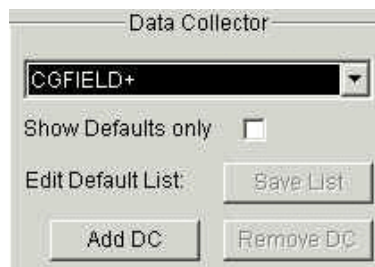


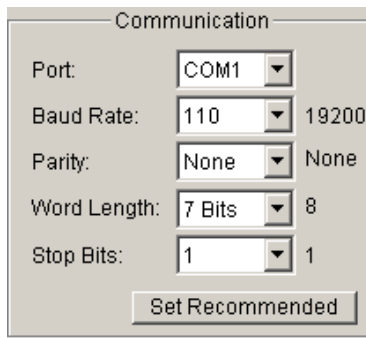
You also can create a Default List, This default list should consist of the various data collectors your company may have and/or interface with on a daily basis.

You can use the Show Defaults only check box to limit the data collectors which may be selected from the Data Collector list on the main Data Collector Transfer Screen. If this box is checked, only those data collectors you have specified for the default list will be show.



To add a data collector to the default data collector list:





First make sure the Show Defaults only box is unchecked. Next Select a data collector from the list by scrolling up and down the list using the arrow keys. When the new data collector is selected, make sure the communication parameters are correct to the data collector. Once the settings are correct, click Add DC button Now click the Save List to save the changes to the list.

When through setting all of the typical data collectors you may use, check the Show defaults only check box and only those instruments and settings will be displayed for your selection.

To remove a data collector from the default list:

Make sure the Show Defaults only is checked.

Highlight the data collector you which to remove from list

Click on the Remove DC

Click on the Save List button

Chose OK and verify that the data collector is no longer in the default listing.

Communications

The Communication box allows you to set the following parameters:

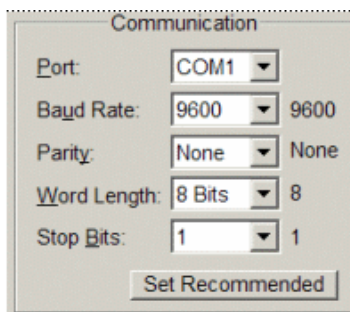
Port

Baud rate

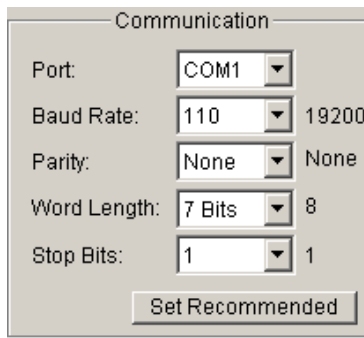
Parity

Word length

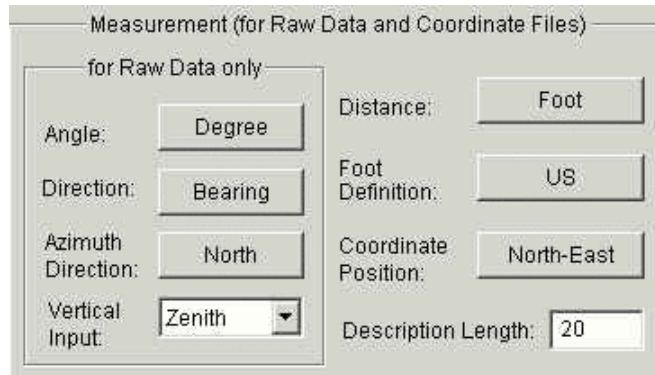
Stop bits



When a data collector is selected, C&G reads a list of default settings and compares it to the settings currently shown. If the current settings are different than those recommended the defaults will be displayed and a Set Recommended button will be displayed. This allows you to automatically set the recommended communication parameters for your data collector.

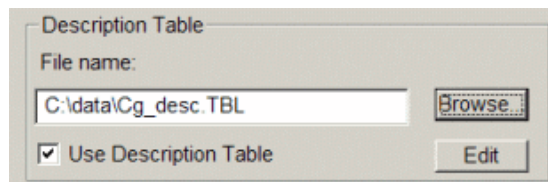


The **Measurement** portion of the Settings dialog box pertains to the units of Raw and Coordinate data input.



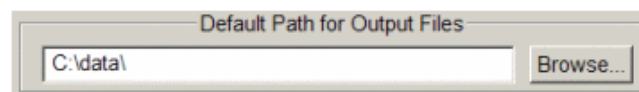
- Angle Mode..... Degrees or Grads
- Direction Mode..... Bearing or Azimuth
- Azimuth Direction:..... North or South
- Vertical Input..... Zenith, Nadir or Horizontal
- Distance.....Foot or Meter
- Foot Definition..... U.S. or International
- Coordinate Position..... North-East or East-North
- Description Length..... 1 to 100 characters

The **Description Table** portion of the Settings dialog allows you to select the default description table.



As mentioned earlier you can have multiple description tables, here is where you would select the description table to use.

The Use Description Table option, when checked , will replace any integer description found in the raw data file with corresponding description found on the description table. When this check box is not checked data will be transferred without translation.



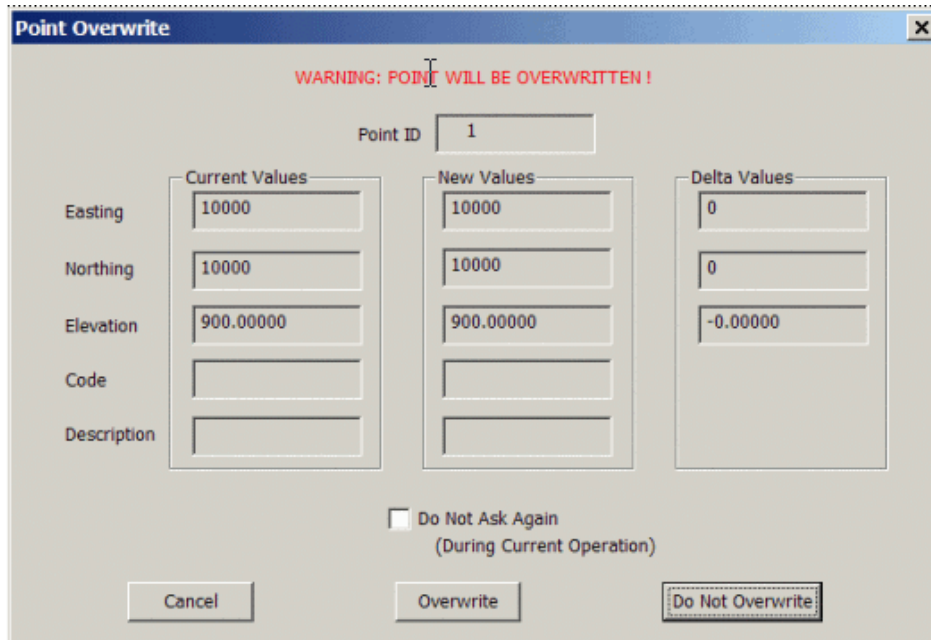
Default Path for Output Files

Allows you to set the default location for storing transferred files

Allows you to set the

Receiving Coordinates from Data Collectors:

There is a point protection feature in place when bring coordinates into an existing coordinate file from a data collector. If the point already exists, and if the coordinate values are different, you will see the following dialog box.



The image shows a 'Point Overwrite' dialog box with a warning message: 'WARNING: POINT WILL BE OVERWRITTEN!'. The dialog box contains a 'Point ID' field with the value '1'. Below this, there are three columns of input fields: 'Current Values', 'New Values', and 'Delta Values'. The 'Current Values' column has fields for Easting (10000), Northing (10000), Elevation (900.00000), Code, and Description. The 'New Values' column has fields for Easting (10000), Northing (10000), Elevation (900.00000), Code, and Description. The 'Delta Values' column has fields for Easting (0), Northing (0), and Elevation (-0.00000). At the bottom of the dialog box, there is a checkbox labeled 'Do Not Ask Again (During Current Operation)' which is currently unchecked. Below the checkbox are three buttons: 'Cancel', 'Overwrite', and 'Do Not Overwrite'.

You will have the following options:

Overwrite: overwrite existing point

Do Not Overwrite: skip point

Overwrite, Do Not Ask Again: Overwrite all existing points

Do Not Overwrite, Do Not Ask again: Bring in only new points

Transfer Options

Depending on the type of data collector that you are using, you will be able to perform some of the following functions:

- Receive raw data from the data collector or file.
- Send raw data to the data collector or file.
- Receive coordinates from the data collector or file.
- Send coordinates to the data collector or file.
- Send a program to the data collector.
- Execute a program on the data collector.
- Delete files on data collector
- View and/or Select files on data collector
- Format data area on the data collector

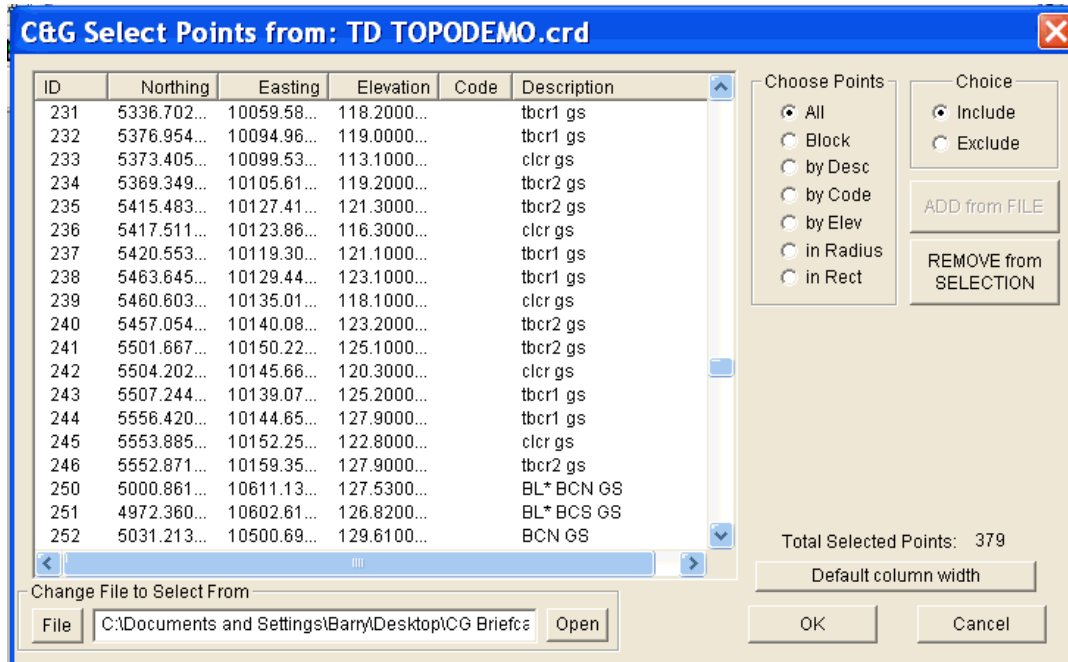
As data is received from a particular data collector or file, it is converted to a .CGR or .CRD file (or other supported format).

Data that is sent to the data collector is converted from the .CGR or .CRD format to the data collector format.

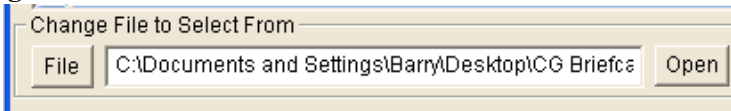
When data is received from a data collector, a read-only file in the data collectors native format is created and stored on the computer. If it is a raw data file, it has a .R\$\$ extension. If it is a coordinate file, it has a .C\$\$ extension.

Select Points

When transferring coordinates to the data collector you may choose which points are to be transferred. The default is ALL points. When you click on the Select Points button the following dialog box comes up.

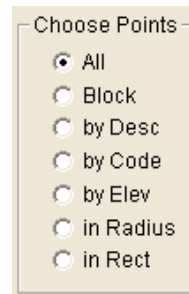


Change file select from: Click the file button to select the coordinate file that you want transfer coordinates

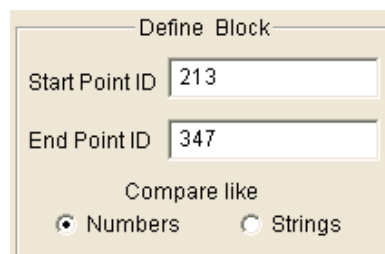


from. **Choose Points:** This option allows you to select groups of points to be included from the file you have opened, using the C&G selection options.

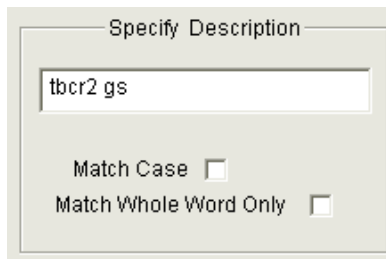
All Points: All Points in the file will be selected.



Block: select blocks of Points.



Desc: select points by their description.



Specify Description

tbcr2 gs

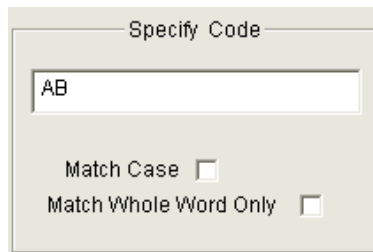
Match Case

Match Whole Word Only

Match Case: Case sensitive compare.

Match Whole Word Only: If your description is BOC this box is NOT checked, points with the descriptions BOC, BOC1, BOC2, etc. would all be included. If the box were checked, only points with the description BOC would be included.

Code: select points by Code



Specify Code

AB

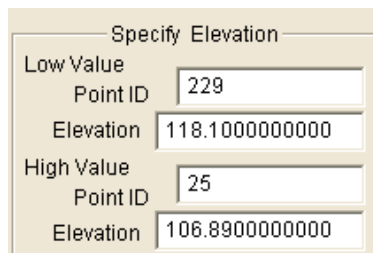
Match Case

Match Whole Word Only

Match Case: Match the case of the text

Match Whole Word Only: If checked, in the above example, only AB would be selected. Descriptions of AB1, ABC and ABB would not.

Elevation: select points by elevation



Specify Elevation

Low Value

Point ID

Elevation

High Value

Point ID

Elevation

Low Value:

Point ID: Point Number

Elevation: elevation at point

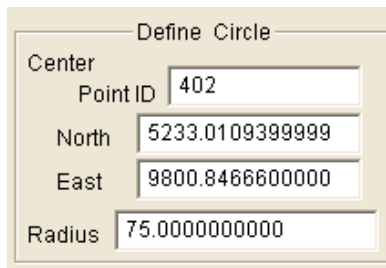
High Value:

Point ID: Point Number

Elevation: elevation at point

If a point number is entered in the point ID box the elevation for that point will be used for either the high or low elevation. You may however enter an elevation only.

In Radius: select all the points within a given radius.



Define Circle

Center
Point ID

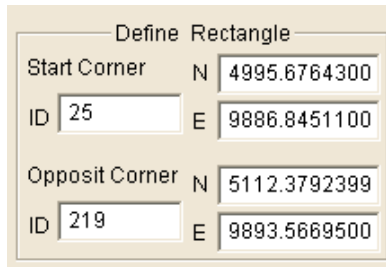
North

East

Radius

If a point number is entered in the point ID box, the northing and easting of that point will be used for the center of the search circle. To manually enter a northing and easting, leave the Point ID box empty and enter the values for the northing and easting of the circle. Enter the radius for the search circle.

In Rectangle: Select all the points within a given rectangle.



Define Rectangle

Start Corner N

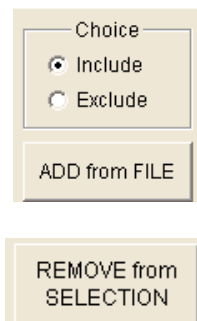
ID E

Opposit Corner N

ID E

If a point number is entered in the point ID box, the northing and easting of that point will be used for that corner of the rectangle. To manually enter a northing and easting, leave the Point ID box empty and the northing and easting values. The two points defined the diagonal corners of the rectangle.

Choice: This option allows you to choose to include or exclude points previously in the C&G select point dialog box. Example:



Choice

Include

Exclude

ADD from FILE

REMOVE from SELECTION

In the choose points dialog box: select by Desc

Then type: GS as the description

in the choice dialog box: select Exclude

Any point that has "GS" in the description field will be removed from the selection set.

Total selected Points: the total number of points selected is shown in the lower right hand corner of the dialog box.

Default Column Width: The columns have a default width. If you have changed the width of a column, say NOTHING, you may press this button to go back to the default widths.

The remainder of this section discusses specific data collectors and software.

CG-Field & FieldPlus Data Collectors

Establish a connection between the data collector and desktop computer with a standard 9-Pin serial cable, USB cable, Bluetooth, etc. Check the settings as shown above.

Download a Description Table

You can transfer the desktop description table directly to the CG-FieldPlus data collector. The table will be placed in the data collector's DC_DESC.TBL file. CG-Field will let you use codes without a description table. Simply delete the DC.CODES file from the data collector and use the code numbers to enter descriptions. When you transfer the file to the desk top, the codes will automatically be replaced with the appropriate description. (This allows you to combine codes.) For example, if you enter [1 20 30], in the description field on the data collector when the transfer takes place these numbers will be read from the desk top description table and converted to the corresponding description, such as [BL* TC SW].

Receiving Raw Data from CG-Field

NOTE: When uploading raw data from a data collector using CG-FieldPlus , a read-only file in the original CG-Field format is created on the computer (in the data directory) with a .R\$\$ extension.

On the desk top data collection transfer dialog box, set the following:

Set transfer method to Data Collector to Computer.

Data Collector Type; CGFIELD+

Use Data Collector:

Transfer Coordinates with raw: -yes or no (your choice)

File Type: Raw Data

File Format: C&G (*.cgr)

File Name: enter the path and name where the file is to be stored or click on the "Browse" button and select the path.

Description Table: enter the path where the Description Table is stored or click on the "Browse" button and select the path

Select Transfer when all settings are correct.

On the data collector Utils menu, select:

1:C&G Transfer

2:Send Raw Data

Enter or select the raw data file

The file will be transferred.

NOTE:CG-Field uses only 2-character point codes. If you have CGSurvey set for 4-character point codes, the CG-Field file will be converted to a 4-character format but it will still have the correct 2-character code. If you download the same file back to the data collector, the downloaded file will be correct if you did not add any codes that actually consist of 4 characters.

Receiving Coordinate File from CG-Field

On the desk top data collection transfer dialog box, set the following:

Transfer Data Collector to Computer

Data Collector:CGFIELD+

Use Data Collector:**File Type:** Coordinate**File Format:** C&G (*.crd) or C&G (*.cgc)**File Name:** enter the path where the file is to be stored or click on the "Browse" button and select the path.**Description Table:** enter the path where the Description Table is stored or click on the "Browse" button and select the path**Press Transfer when all settings are correct.****On the data collector Utils menu, select:****1:C&G Transfer****3:Send Coords****Select Points****All points****Blocks of points****From points file****The file will be transferred.**

NOTE: In any transfer routine it is important to prepare and have ready the device that will be receiving data first.

Sending Coordinate File to CG-Field

On the data collector Utils menu, select:

1:C&G Transfer**3:Receive Coords**

On the desk top data collection transfer dialog box, set the following:

Transfer: Computer TO Data Collector**Data Collector:** CGFIELD+**Use Data Collector:****File Type:** Coordinate**File Format:** C&G (*.crd) or C&G (*.cgc)**File Name:** enter the path where the file is to be stored or click on the "Browse" button and select the path.**Check the Select Points settings:**

This tool allows you to select what group or groups of coordinates are transferred to the data collector.

Press Transfer: when all settings are correct.

NOTE: You should not download a file containing a 4-character code to your data collector. You will be warned that the last 2 characters of the code will not be sent. This means that a code of 1584 will be received as 15.

Receiving ASCII File from CG-Field

On the desk top data collection transfer dialog box, set the following:

Transfer Data Collector to Computer**Data Collector:** CGFIELD+**Use Data Collector:****File Type:** ASCII**File Name:** enter the path where the file is to be stored or click on the "Browse" button and select the path.**Select Transfer when all settings are correct.**

On the data collector Utils menu, select:

1:C&G Transfer

6:Send ASCII
File Name With Extension
File:
Transfer Name:
*****.***
<Enter>

Sending ASCII File to CG-Field

On the data collector Utils menu, select:

1:C&G Transfer
5:Receive ASCII

On the desk top data collection transfer dialog box, set the following:

Transfer Computer to Data Collector

Data Collector: CGFIELD+

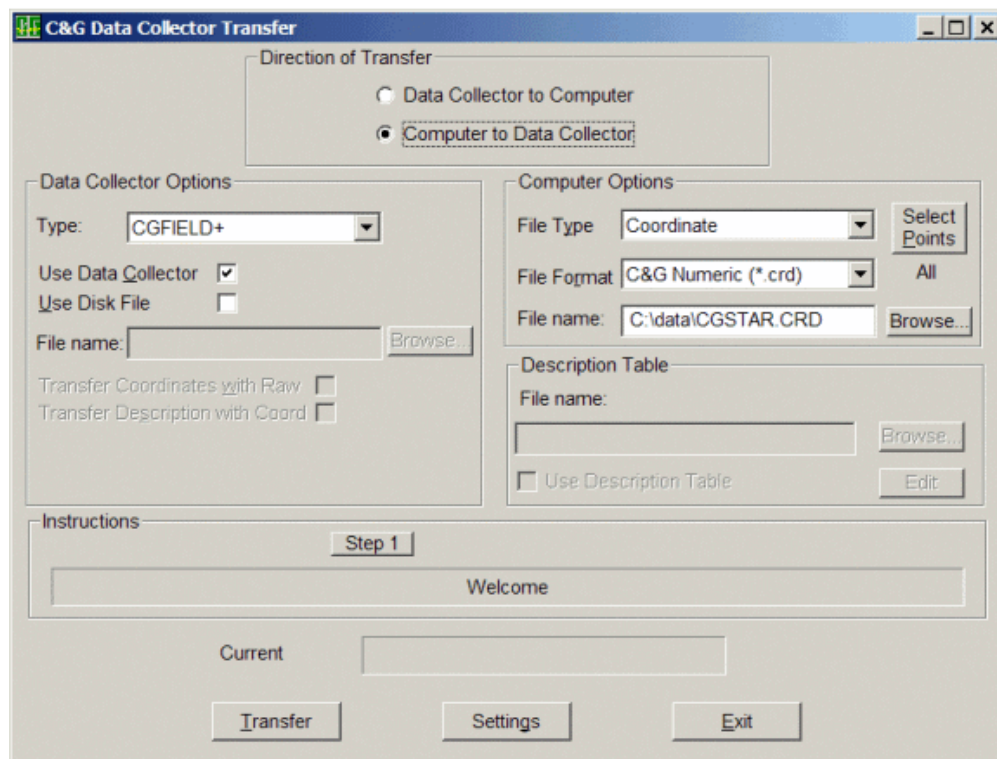
Use Data Collector:

File Type: ASCII

File Name: enter the path where the file is to be stored or click on the "Browse" button and select the path.

Select Transfer when all settings are correct.

The following are examples of typical data collection transfer settings. There will be cases with certain models or manufactures where special instructions will be required and C&G will provide those as needed.



There three different dialog boxes involved with data collection transfer:

C&G Data Collection Transfer (shown above)
Settings
Description Table Editor.

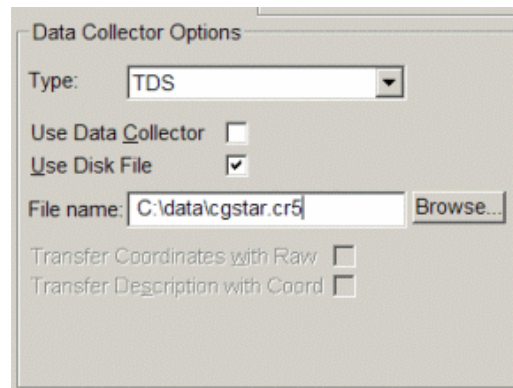
The function and settings for each of these is described in detail in the previous pages.

The examples shown on the following pages show transfers directly from and to data collectors. These same transfer routines will also work with files that have been downloaded to the desktop computer.

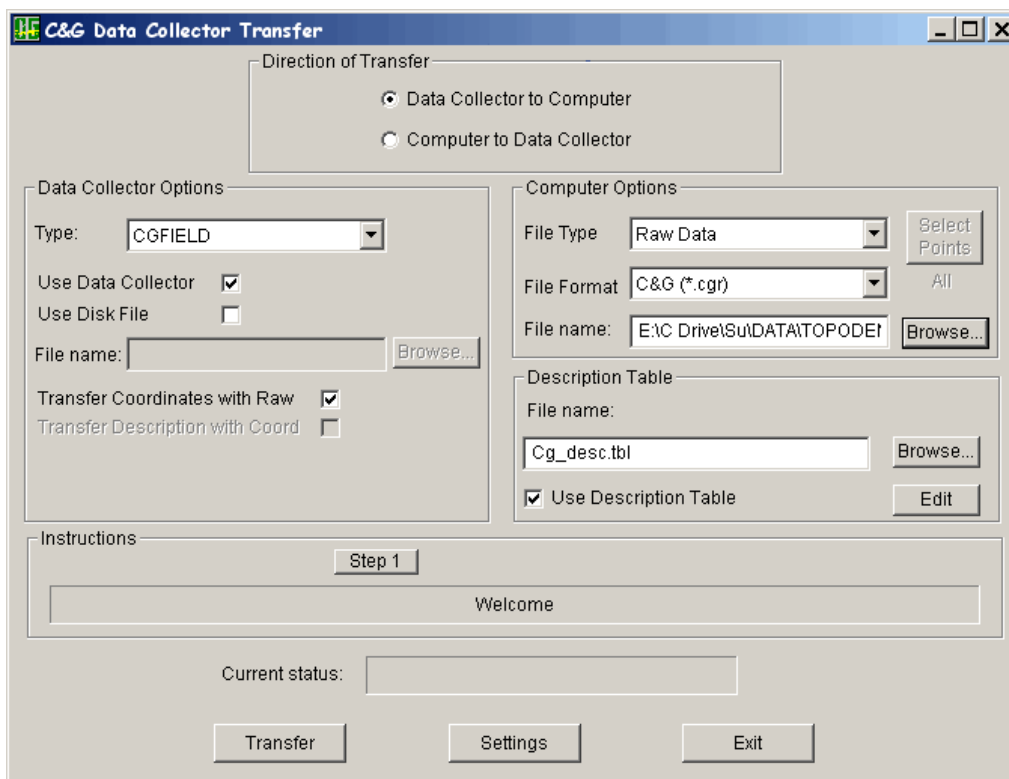
File Conversion Utility

To convert data from files check the Use Disc File box and either hand enter the path and name or click on "Browse" and search for the file location.

These files need to be in the correct data file format.



Receiving Raw Data



On the desk top data collection transfer dialog box, set the following:

Select Data Collector to Computer

Data Collector: (select data collector from list)

Check Use Data Collector:

Transfer Coordinates with raw: -yes or no (your choice)

File Type: Raw Data

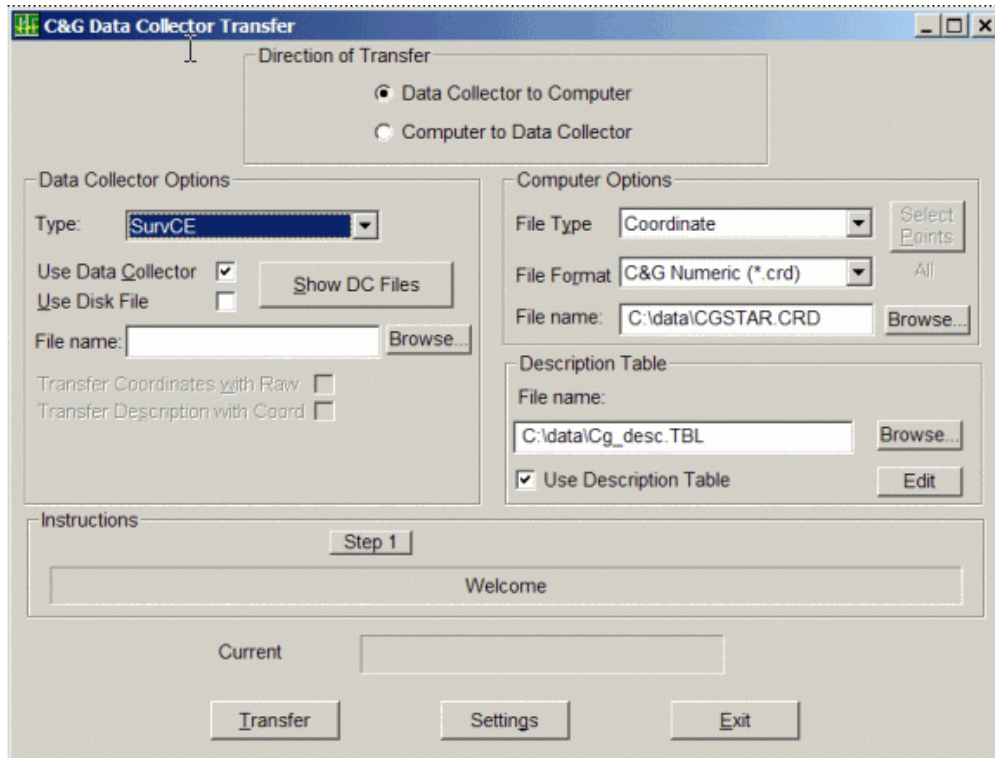
File Format: C&G (*.cgr)

File Name: enter the path where the file is to be stored or click on the "Browse" button and select the path.

Press Transfer: when all settings are correct.

Begin transfer from data collector

Receiving Coordinate Data



On the desk top data collection transfer dialog box, set the following:

Select Data Collector to Computer

Data Collector: (select data collector from list)

Use Data Collector:

File Type: Coordinate

File Format: C&G (*.crd) or C&G (*.cgc)

File: enter the path to store the file or click on the Browse button and select the path.

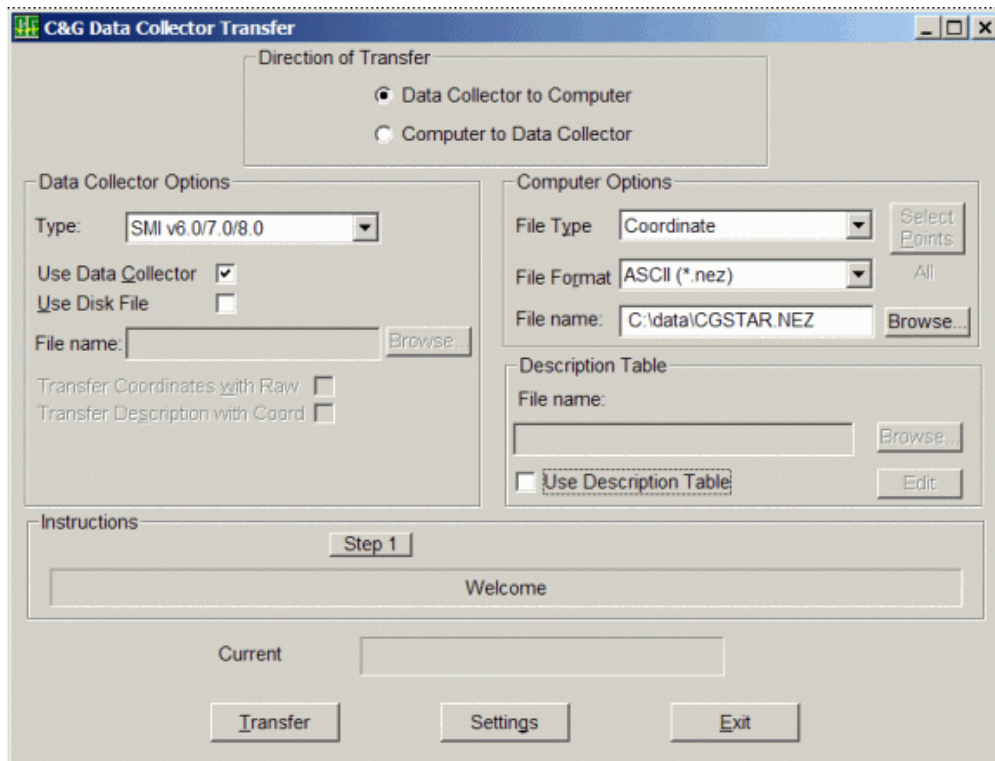
Description Table: enter the path where the Desc Table is located or click on the Browse button and select the path.

Press Transfer: when all settings are correct.

Begin transfer from data collector

NOTE: In any transfer routine it is important to prepare and have ready the device that will be receiving data first.

Receiving ASCII Data



On the desk top data collection transfer dialog box, set the following:

Select Data Collector to Computer

Data Collector: (select data collector from list)

Use Data Collector:

File Type: Coordinate

File Format: ASCII (*.nez)

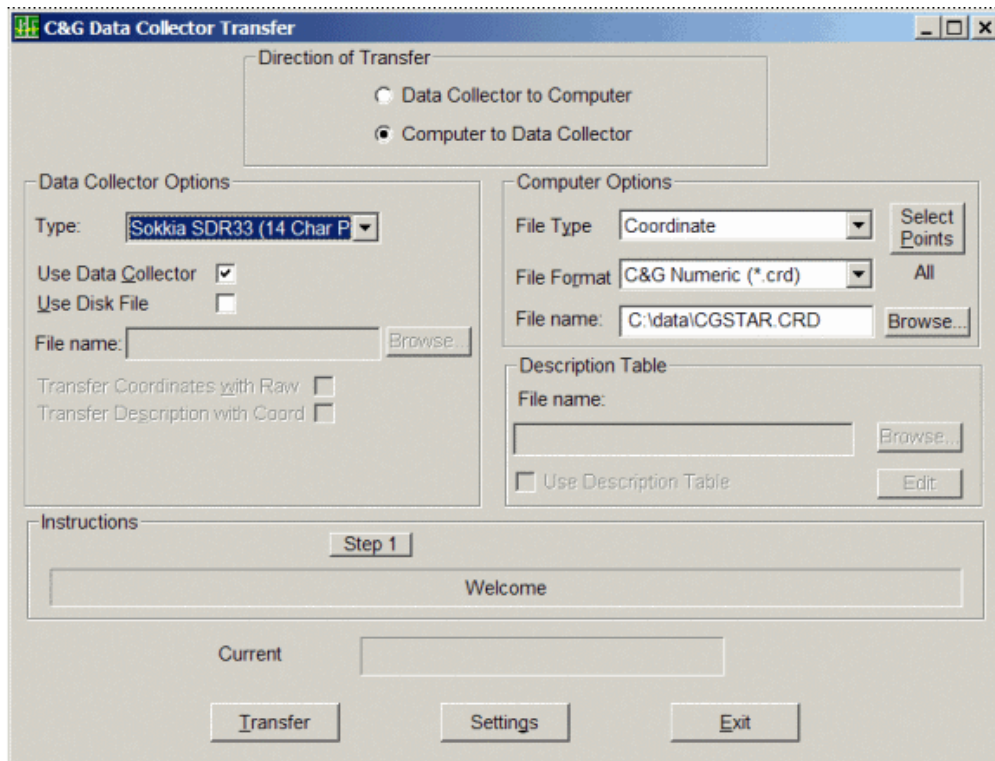
File: enter the path to store the file or click on the Browse button and select the path.

Description Table: enter the path where the Desc Table is stored or click on the File button and select the path.

Select Transfer: when all settings are correct.

Begin transfer from data collector

Sending Coordinate Data



Prepare Data collector to receive Coordinate file

On the desk top data collection transfer dialog box, set the following:

Select Computer to Data Collector

Data Collector: (select data collector from list)

Use Data Collector:

File Type: Coordinate

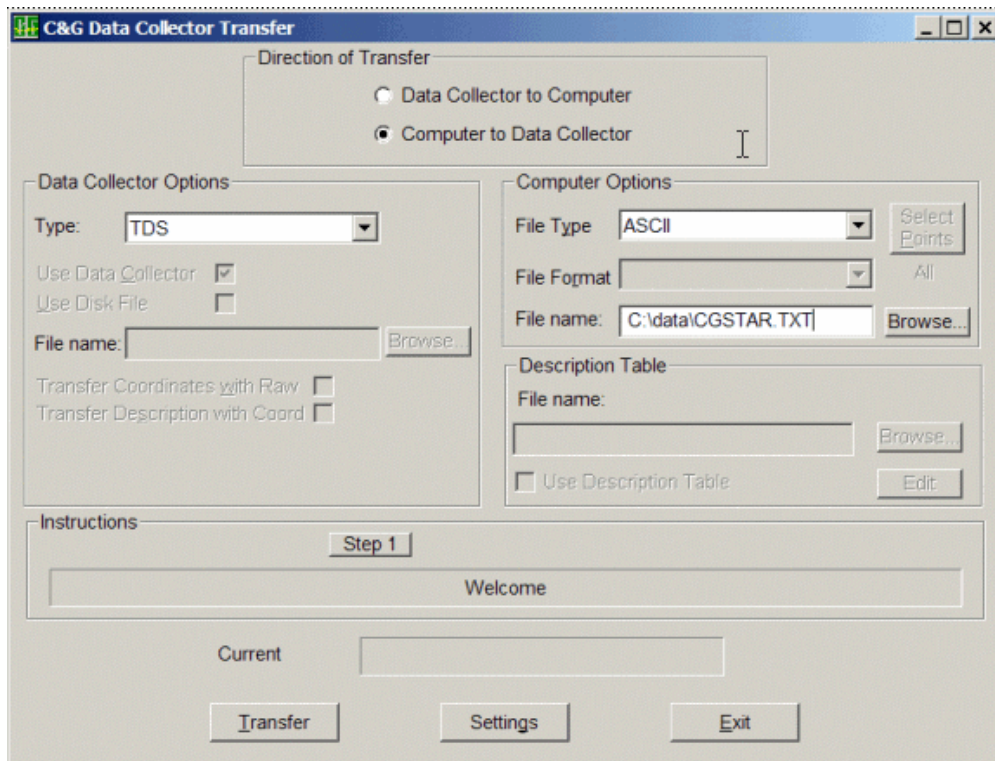
File Format: C&G (*.crd) or C&G (*.cgc)

File: enter the path to file or click on the Browse button and file

Check the Select Points setting:

Press Transfer: when all settings are correct.

Sending ASCII File



On the desk top data collection transfer dialog box, set the following:

Select Computer to Data Collector

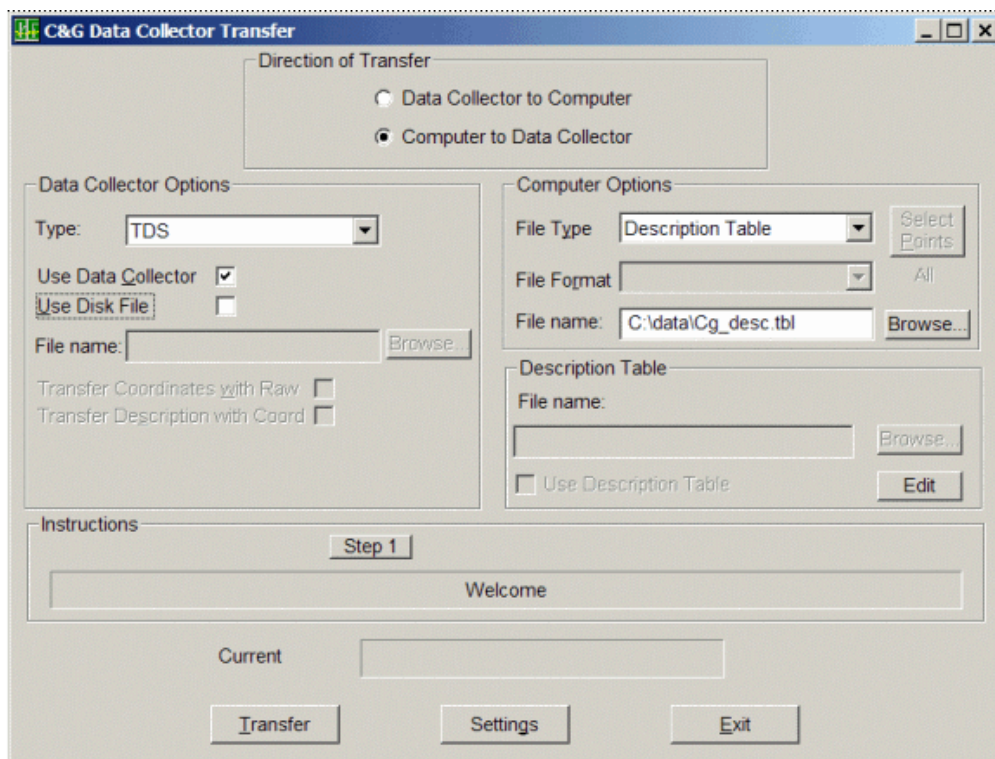
Data Collector: (select data collector from list)

File Type: ASCII

File: enter the path to store the file or click on the bROWSE button and select the path

Press Transfer: when all settings are correct.

Sending Description Table



On the desk top data collection transfer dialog box, set the following:

Select Computer to Data Collector

Data Collector: (select data collector from list)

File Type: Description Table

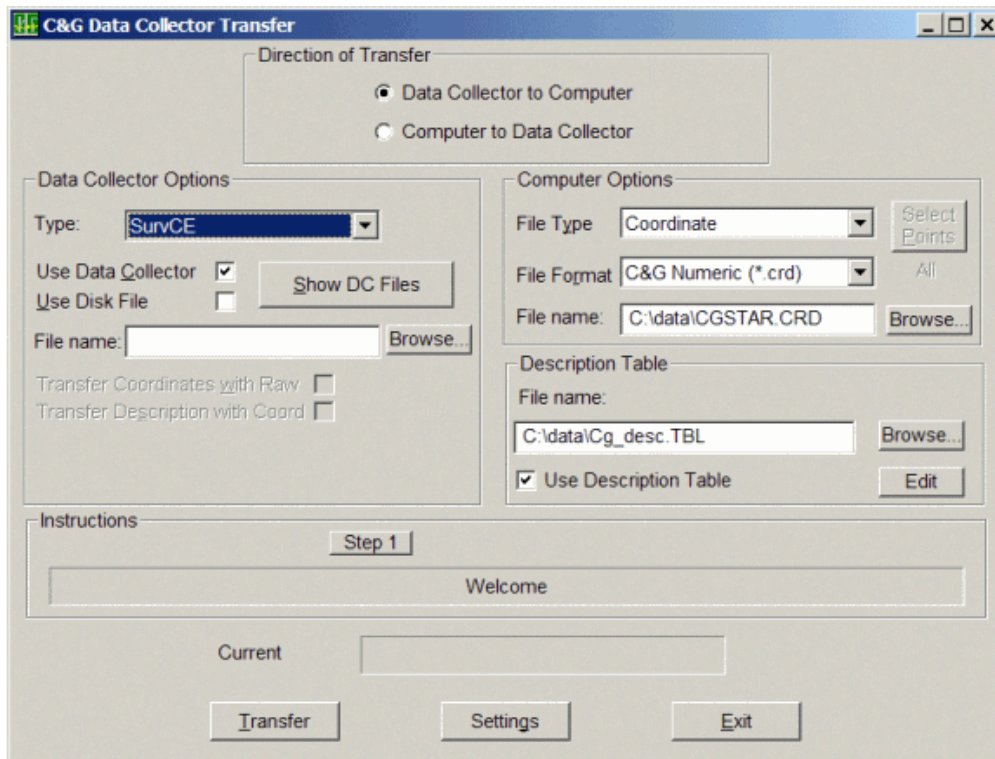
File: enter the path to store the file or click on the File button and select the path

Press Transfer: when all settings are correct.

SurvCE Data Collector

You can receive coordinates and raw data from the data collector, or send coordinates to the data collector. Make sure SurvCE is selected as the data collector.

Receive Coordinates from SurvCE



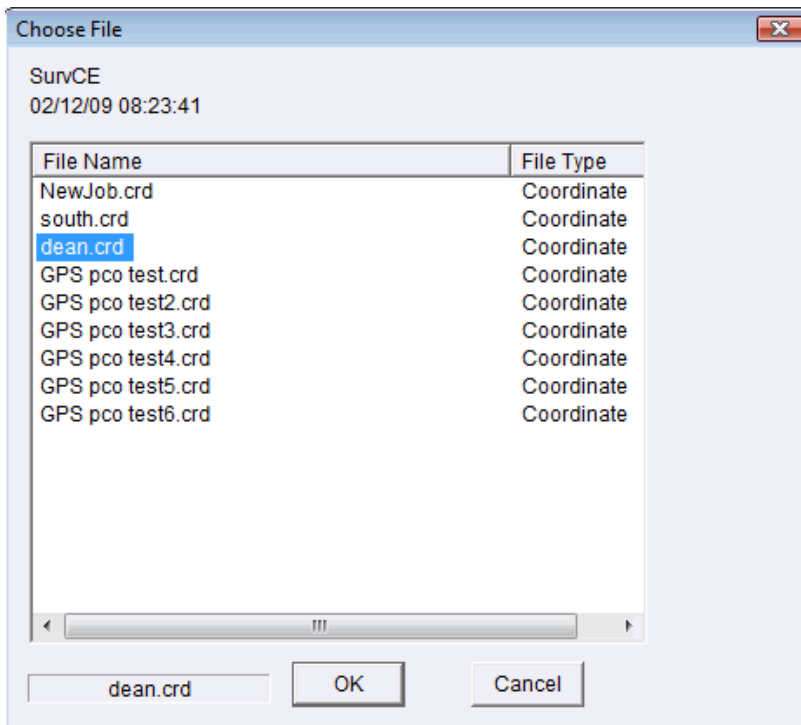
On the desktop, click on "Data collector to Computer": select SurvCE as the Type data collector.

Set FILE TYPE to Coordinate: and select the desired File Format.

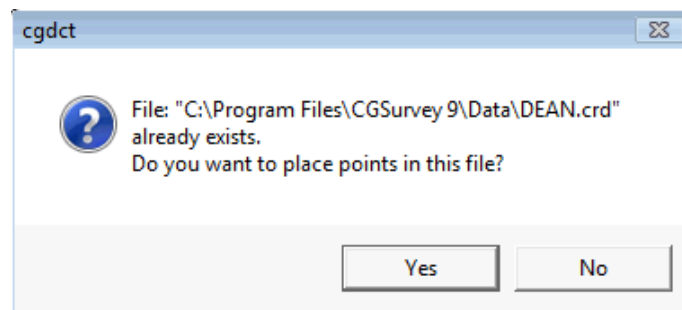
On the Data Collector, Select FILE >DATA TRANSFER: Choose Carlson/C&G Transfer.

On the desktop, select BROWSE button next to the FILENAME field: You will see the coordinate files that are on SurvCE.

Select the File you wish to download: and press OK.



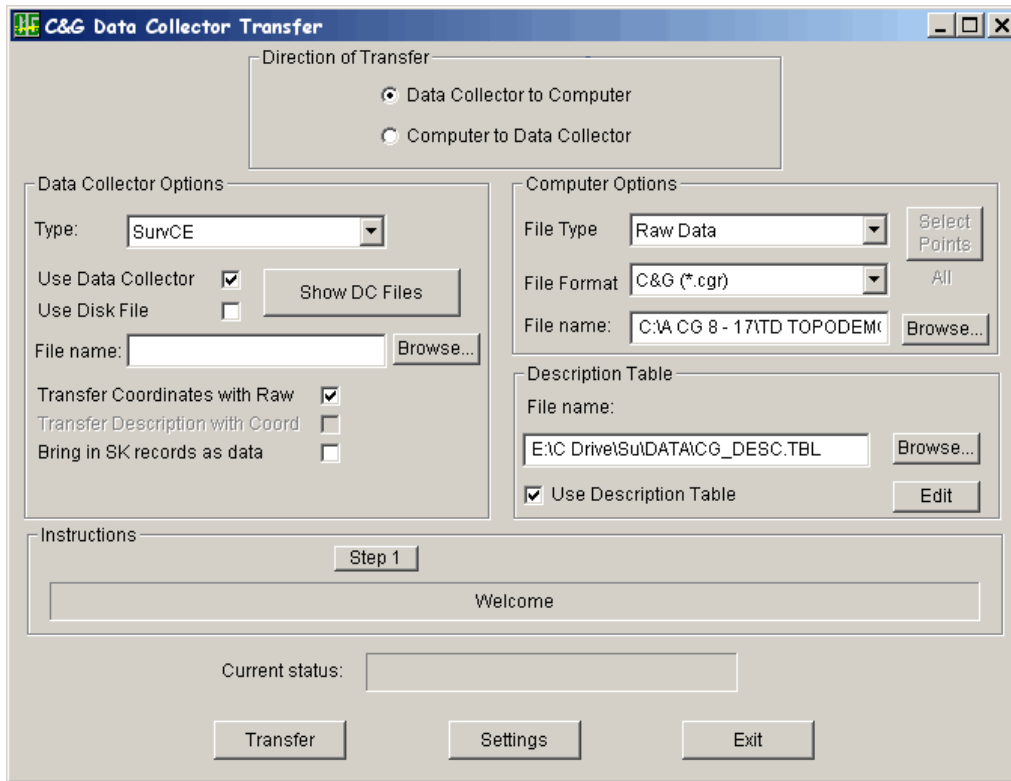
Press the Transfer button. If you do not have a destination FILE NAME selected, you will see the following dialog:



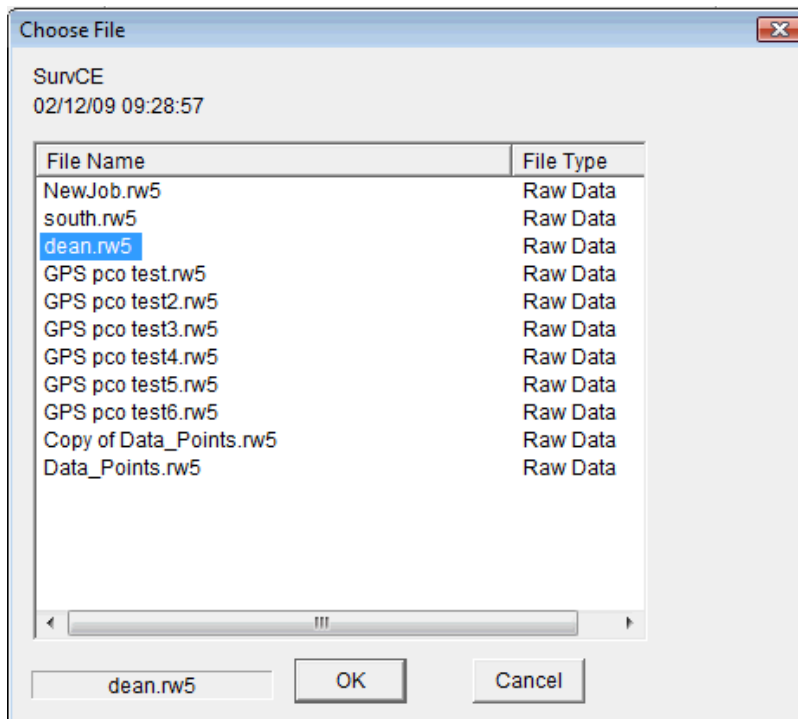
In this case, the file already exists, If you press OK the coordinates will be written to the existing file. Point Overwrite Protection will allow you to select which points you wish to bring in. You can decide individually whether you want to overwrite a point or not, or you can select overwrite ALL points, or you can select to bring in ONLY new points.

A file with the same name and a C\$\$ extension will also be created with the data that came directly from survCE in survCE's format. This file is read-only and can be archived for legal purposes.

Receive Raw Data from SurvCE



On the desktop, click on "Data collector to Computer": select SurvCE as the Type data collector.
Set FILE TYPE to Raw Data: and select the desired File Format.
On the Data Collector, Select FILE >DATA TRANSFER: Choose Carlson/C&G Transfer.
On the desktop, select BROWSE button next to the FILENAME field: You will see the raw data files that are on SurvCE.

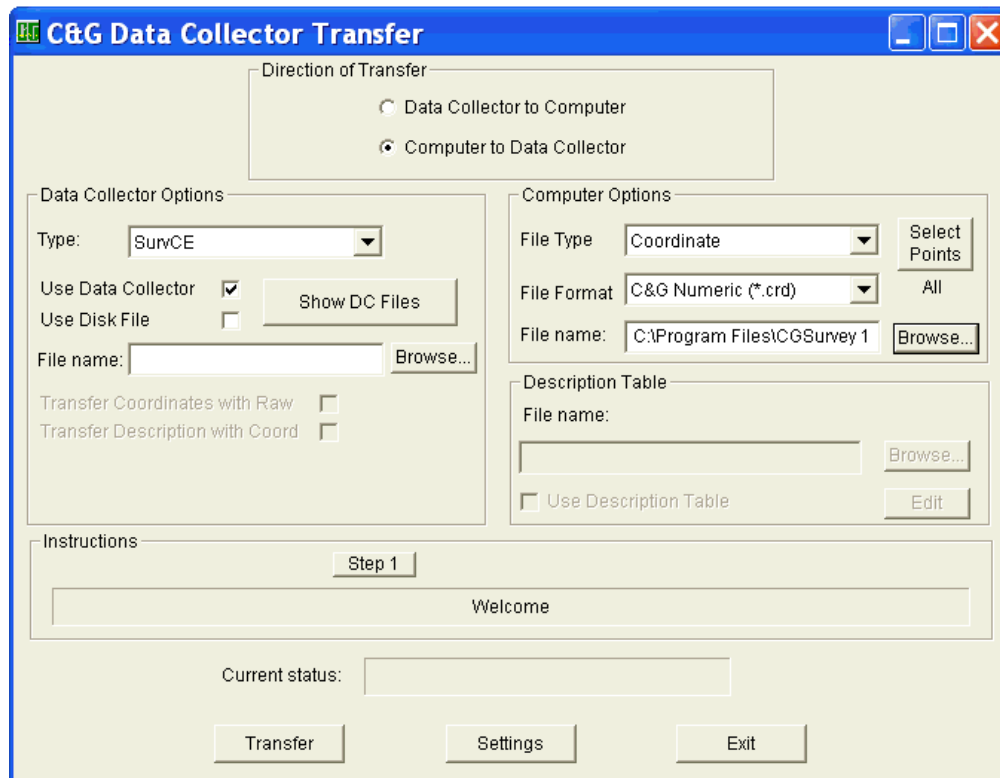


Select the File you wish to download: and press OK.

The selected raw data file will be transferred and converted to the selected format. A file with the same name and a R\$\$ extension will also be created with the data that came directly from survCE in SurvCE's format.

This file is read-only and can be archived for legal purposes.

Send Coordinates to SurvCE



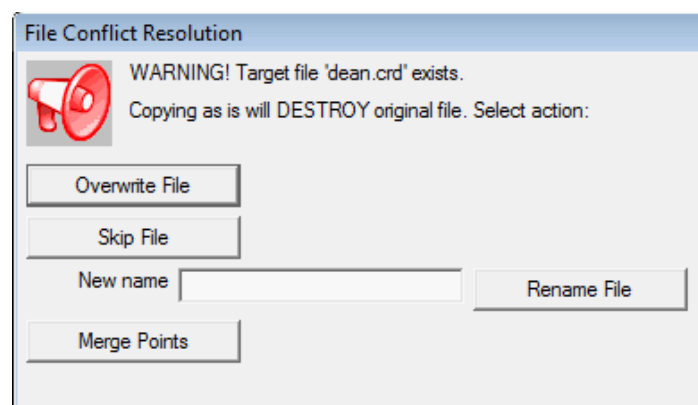
Select Computer to Data collector. Make sure the Data Collector TYPE is set to SurvCE: select the file to be set to SurvCE (N_DRUIDH.crd). If you do not select a destination name, it will be sent to the same named file as the source.

If you do not want to send ALL the points, but need to select specific point: press the SELECT POINTS button and choose the point to transfer.

On the data collector select FILE>DATA TRANSFER: Choose Carlson/C&G Transfer

On the Desktop, Press the TRANSFER BUTTON: The selected coordinates will be transferred.

If the file already exists on SurvCE, you will see the following dialog:



You have the following choices:

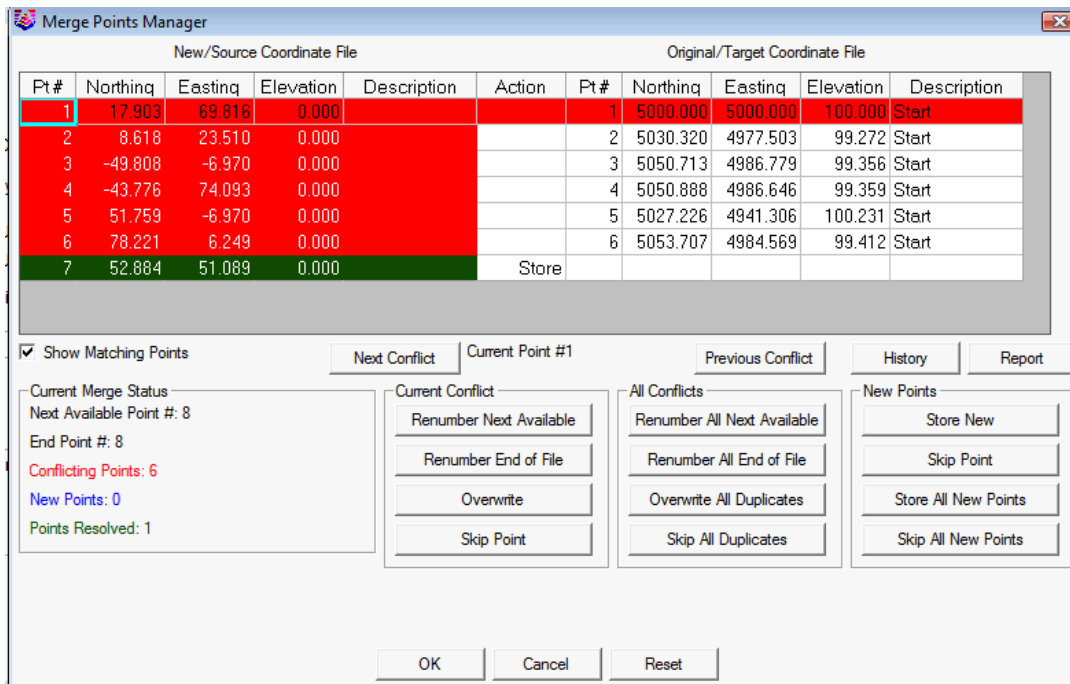
Overwrite the existing file

Skip the file (do nothing)

Rename the file

Merge the points.

If you select the Merge option, you will see Carlson's standard merge dialog:



This dialog allows you to fix all conflicts prior to transferring the points.

TOPCON DATA COLLECTORS

Use Topcon cable A-5 if your computer has a 25-pin serial port, or Topcon cable A-16 if your computer has a 9-pin serial port. When uploading raw data from a Topcon/TDS data collector, a read-only file in the original data collector format is created on the computer (in the data directory) with a .R\$\$ extension. When you send the description table to a Topcon/TDS data collector, only the first 999 descriptions will be sent.

FC1 DATA COLLECTOR

For the transfer program to be able to access any data in the FC1, it must be stored in the FC1 using the Program 2 supplied with the FC1 transfer software. First locate the necessary cables to connect the FC1 to your computer. (your dealer should be able to help you with this).

NOTE: Currently, the only programs that are supported are ET1 and GTS3, which are supplied with the system.

Follow these instructions:

Select either ET-1 or GTS-3 when prompted. Once you have loaded Program 2 into the FC1, you may enter your field data in either of two ways. The first way, is by connecting the FC1 to your total station and let the total station record angular and distance measurements for you by using the ET1 (GTS3) section of the FC1 program. The other way is to use the Manual Entry section of the program to store all of your field data directly through the FC1's keyboard.

Collecting Data using the FC1

(The following steps will be followed no matter which data entry method you use):

- 1) Set up data recording mode in the FC1.
- 2) Enter job information: job name, operator, instrument number, date, temperature, pressure.
- 3) Enter instrument point information: point number, H.I., backsight point, angle in instrument to backsight.
- 4) Enter foresight point information: point number, rod height, horizontal and vertical angles, distance.
- 5) If there are other foresights from the same instrument point, repeat step 4; or if you have another instrument setup, go to step 3; or if you are through, go to the end of the program.

NOTE: Never press the <skip> key when the FC1 is asking for data. Only use the <skip> key to by-pass "go to" options (see step 5 below).

Automatic Recording of Data

1) To set up the recording mode, have the FC1 connected to the ET1 (GTS3), and turned on. Wait until the left side of the display says READY.

If the right side of the display says PRG>2, then you are ready for step 1A.

Otherwise, press these keys: <func>, <#>, <Enter> and then go to step 1B below.

A) **Press the <F1> key.**

B) **When the display says GOTO 7 ET1-PROG? ("GOTO 7 GTS3-PRG")**, press the <Enter> key.

2) **Enter any name you want for the job-id:**

Enter the name of the operator.

Enter the instrument number.

Enter the date, temperature, and pressure.

3) **Sight the backsight point with the instrument:**

Enter the instrument point number.

Enter the instrument height (H.I.).

Enter the backsight point number.

Press <Enter> for the rod height.

Press <Enter> when the display says REC MODE 2 (REC V/H< MODE? on GTS3).

When the display says MODE>, press <2> and the ET1 should send angular information to the FC1.

4) **Turn to the foresight.**

Enter the foresight point number, and press <Enter>.

Enter a description for the foresight and press <Enter>.

Enter the foresight rod height and press <Enter>.

When the REC MD3-DR/2R (REC SD/V/HMODE?) message appears, press <Enter>, then choose mode 3 if you are entering a direct angle.

If this is your second angle to that foresight, then you may use mode 2 to record angles only.

5) **After step 4, then you should see this message in the display: GOTO 18 FS.PT#?.** If you wish to turn more angles from the current instrument point, **then press <Enter>**, and go back to step 4.

-or-

Press <Skip>. The next message will be GOTO 13 INST.PT? If you want to record another instrument set-up, then press <Enter>.

-or-

Press <Skip>. The next message is, GOTO 8 OPER.? To change operators, press <Enter>, and go to step 2.

-or-

Press <Skip>. The next message is, GOTO 53 END? To end input for this job, press <Enter>.

Steps on Manual Recording of Data

1) **To set up the recording mode, have the FC1 turned on. Wait until the left side of the display says READY. If the right side of the display, says PRG>2** then you are ready for step 1A.

Otherwise press these keys: <func>, <#>, <Enter> and go to step 1B.

A) Press the <F1> key.

B) When the display says GOTO 7 ET1-PROG? (GOTO 7 GTS3-PRG): press the <skip> key.

When the display says GOTO 30 MANUAL?: press the <Enter> key.

2) Enter any name you want for the job-id:

Enter the name of the operator:

Enter the instrument number:

Enter the date, temperature, and pressure:

3) Enter the instrument point number:

Enter the instrument height (H.I.).

Enter the backsight point number:

Enter the rod height if desired:

Enter the angle in the instrument: when the backsight was taken.

Enter the vertical angle and distance: if desired.

4) Turn to your foresight. Enter the foresight point number, and press <Enter>.

Enter a description for the foresight and press <Enter>.

Enter the foresight rod height and press <Enter>.

Enter the horizontal angle, slope distance, and vertical angle to the foresight.

5) After step 4, then you should see this message in the display: GOTO 43 FS. PT#?

If you wish to turn more angles from the current instrument point, then press <Enter>, and go back to Step 4.

-or-

Press <Skip>. The next message will be GOTO 36 INST.PT? If you want to record another instrument set-up, then press <Enter>.

-or-

Press <Skip>. The next message is GOTO 31 OPER.? To change operators, press <Enter>, and go to step 2.

-or-

Press <Skip>. The next message is GOTO 1 MENU? To go to the menu for another job, press <Enter>.

-or-

Press <Skip>. The next message is END. Press <Enter> to end entry for this session.

Receive Data from FC1

Once you have your data stored in the FC1, you must upload it to the computer.

Select the Receive Raw Data function. If you have already dumped the data stored on the FC1 to a computer file (in the Topcon format), you may choose to receive the data from the file. Enter the file name that contains the data, and the file name for the .CGR file.

NOTE: To power your FC1 while sending data to the computer, you must plug the power cable into the signal port at the top of the FC1 and toggle the FC1's power switch to EXT.

TOPCON PROPAC DATA COLLECTOR

To enable the Propac to collect raw data in a format Suitable for CG-Survey's data transfer program the CG program must be installed on the 71B. To load the software: connect the Propac to the computer com port.

Follow the directions to download.

You may store the CG program in a freeport on the 71B. This has several advantages, the main one being it will not be lost if the batteries die.

Under the Propac options choose Load CG Program Into Propac.

On the 71B :

Type Freeport(.01) and key <Endline>.
Type COPY CG TO :PORT(.01) and key <Endline>.

For ROM Versions Prior to 1.75

Type DEF KEY 'f7',"USER @ RUNCG": and key <Endline>.

For ROM Versions 1.75 or Later

Type DEF KEY 'f7',"USER @ CONT PRGM2": and key <Endline>.

This will set up the raw data collection program to run on the Propac when the yellow function key and the <7> key are pressed. Unless something happens to the 71B, you should not need to reinstall the CG-Field program again.

Now delete the CG-Field program from the main memory of the Propac by typing PURGE CG:MAIN and key <Endline>. The program is still stored in the freeport.

To use the CG-RAW data program, turn the Propac on and type RUN PRO then key <Endline>. From the KEYS prompt press the yellow function key then the <7> key. This will start the CG program, then just follow the prompts.

The CG raw data program is the only one needed on the Propac other than the Propac options already available. If you are collecting coordinates and elevations instead of raw data, simply follow the Propac instructions.

To transfer the collected data to and from the computer, choose the Propac option you wish and follow the directions on the screen.

If you have already downloaded the data stored on the Propac to a computer file (in the Propac format), you may choose to receive the data directly from the file. Enter the file name that contains the data, and the file name for the .CGR file (raw data file) or .CRD file (coordinate file).

FC-4 DATA COLLECTOR

The C&G data collector transfer program can accept data that was collected from the FC-4 in either the traverse mode or topo mode. It can receive coordinates from the FC-4 and also send coordinates to the FC-4 for stakeout. The CG data collector transfer program supports most the valid methods of collecting data in the traverse or topo mode of the FC-4 (including the ability to collect direct and reverse angles). Refer to the FC-4 users manual to learn the different methods of data collection supported by the FC-4.

Special Features - When translating the FC-4 file to a raw data file using the C&G data collector transfer program:

- 1) If an FC-4 record is not used, the record will be placed in the raw data file as a comment with the message Not Used appended. No FC-4 record will be ignored. For example, *123 Not Used
- 2) Remarks ("R" records) will be placed in the raw data file as a comment record.
- 3) Coordinates will be placed in the raw data file as a coordinate record, (C 23 10000.0000 10000.0000 923.24 "TP).
- 4) When using the FC-4 Benchmark function, the following will be placed in the raw data file:
 - A) The benchmark coordinates.
 - B) The measurements to the benchmark as a foresight point.
- 5) When using the FC-4 Angle-Offset function the following will be placed in the raw data file:
 - A) A comment line saying the next line is an angle/offset and showing the 1st and 2nd angle recorded to the point.
 - B) A foresight record combining the first distance measurement and the 2nd angle measurement.
- 6) When using the FC-4 Distance-Offset function the following will be placed in the raw data file:

A) A comment line saying the next line is a distance/offset and showing the slope distance, vertical angle and offset distance measured to the base point.

B) A foresight record with a new slope distance and vertical angle calculated from the above information.

7) When using the FC-4 Perpendicular-Offset function the following will be placed in the raw data file:

A) The foresight record to the base point.

B) A comment line saying the next line is a perpendicular offset and showing the offset forward/backward, the offset left/right and the offset up/down.

C) A foresight record with a new horizontal angle, slope distance and vertical angle calculated from the above information.

NOTE: If there is no left/right offset, data will not convert correctly to a CG-SURVEY raw data record. For example, if the Perpendicular Offset routine is used to locate a point away from the instrument but on the same line, the resulting data record will use the wrong horizontal angle.

8) Backsight azimuths are transferred to the .CGR file as reference azimuths.

When a file is first created on the FC-4 the user will be prompted for some header information. When the data is transferred to the computer, the C&G data collection transfer program will use the job-id as the file name for the raw data file created on the computer.

To prepare the data collector and computer for data transfer, connect the A-5 or A-16 cable to the serial port of the FC-4 and to the appropriate serial port of the computer. Make sure the correct data collector comm port has been chosen in Settings Dialog.

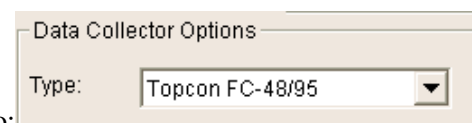
Choose the appropriate menu option on the FC-4, then follow the instructions and answer the prompts as they appear on the screen. Once the raw data has been downloaded into the computer the raw data can be edited, reduced and printed out from the Traverse/Input Edit program.

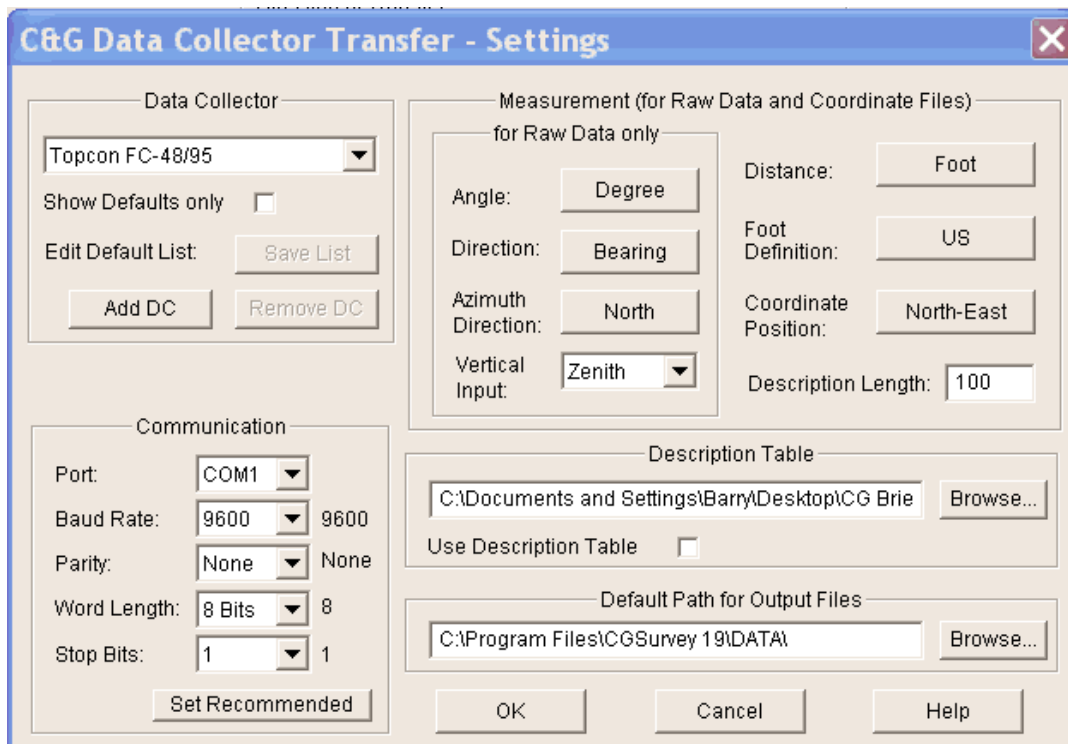
If you have already downloaded the data stored on the FC-4 to a file on the computer, you can transfer the data using the "Use Disc File" command

TOPCON and TDS

Transferring Data

On the C&G Data Collector Transfer screen set the data collector option to:
Check Settings to make sure all options are set correctly





Note: See opening section of this chapter for detailed instructions on the Settings dialog box and on sending and receiving files.

Sending Description Table to 48 When sending the description table (DC_CODES file) to the 48, the following occurs:

A new file (DESCRIPT.TXT) is created in the data directory on the computer.

The first 200 descriptions are duplicated from the DC_CODES file.

After that, the first 100 descriptions are reproduced 7 times with the following mapping codes preceding the descriptions:

- 201 - 300 BL*(DESC)
- 301 - 400 EL*(DESC)
- 401 - 500 CL*(DESC)
- 501 - 600 CF*(DESC)
- 601 - 700 OC*(DESC)
- 701 - 800 PC*(DESC)
- 801 - 900 PT*(DESC)

The DESCRIPT.TXT file is then sent to the 48.

TOPCON CR2 CARD READER

Data collected and stored using the Topcon Card Reader is in the same format as data on the FC-4. All data format rules for the FC-4 apply here.

Set-Up

Card Read Preparation. Before using the card reader it is necessary to set the two DIP switches on the bottom of the unit to the settings described below. The direction on the switch which is marked 'OFF' is really a '1'. You should read the attached label, not the switches.

The CR2 should be set as follows:

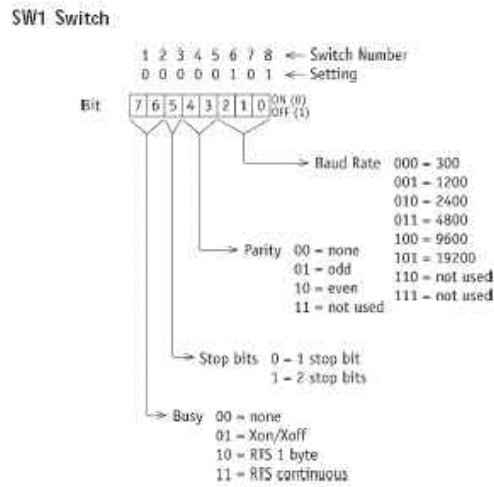
Baud: 19200

Parity: None

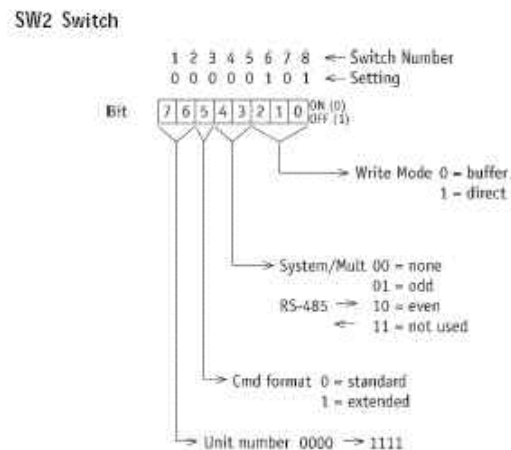
Stop Bits: 1

Word: 8

SW1 DIP Switch: Set the communication parameters



SW2 DIP Switch: Sets other parameters



Use the interface cable supplied with the CR2 unit and plug it into the comm port on the computer. Make sure you select the Topcon CR2 data collector and the correct comm port.

The Card Reader program allows:

- 1) Receive raw data from the CR2 or from a CR2 file.
- 2) Receive coordinates from the CR2 or a CR2 file.
- 3) Send coordinates to the CR2.
- 4) Send or receive a description table to CR2.
- 5) Receive description table.
- 6) Send or receive ASCII files from the CR2.
- 7) Send an executable (EXE) file to the CR2.
- 8) Catalog (or directory) of all files on CR2.
- 9) Delete files on the CR2.
- 10) Format cards for the CR2.

Receiving Data from the CR2

You may receive raw data files (.R), coordinate files (.N), ASCII files or description table from the CR2. All files on the CR2 of the type you wish to receive will be shown on the screen

Raw Data

Coordinates

ASCII

Description Table

Receiving Data from a CR2 File

If you have already downloaded the data stored on the CR2 to a file on the computer, you may choose to receive the data from the file.

Sending Data to the CR2

You may send coordinate (.CRD) files, ASCII files, EXE files or description table to the CR2. Select the file you wish to send. You may not send a file to the card reader that already exists on the CR2. You must delete the file first. The.CRD files will be converted to .N files. ASCII files will be transferred without conversion, (make sure the file you are transferring is a true ASCII file).

EXE files

Programs with .EXE extensions can be transferred to the CR2. These files will be transferred with a .X extension.

Note: See opening section of this chapter for detailed instructions on the Settings dialog box & information on sending and receiving files.

Catalog: The catalog function will show you all existing files on the Topcon Card Reader.

Deleting Files:All files on the CR2 will be shown on the screen. Select the file you wish to delete. Be careful, once the file is deleted it is gone forever.

Format: The format function will allow you to format a card, making it ready to accept (store) data. If the card is already formatted, you will be warned that all information on the card will be lost, be careful.

SOKKIA (LIETZ) SDR2 DATA COLLECTOR

General Information

Use the cable supplied with the SDR2 data collector to plug into your computers serial port.

When uploading or downloading to or from the computer, turn the switch on the cable toward the word PRINTER. If this does not work, turn the switch toward the word COMPUTER and try again. If you still have trouble please call us.

In order to use the Lietz SDR2 data collector with the transfer program, there are 4 areas that you must consider: (1) Entering data into the data collector in a format that can be sent to your computer, (2) the transfer program itself, (3) sending calculated coordinates back to the data collector, and (4) the data collector code conversion table which converts numeric codes for points into English-language descriptions as the data is sent to the computer.

Sokkia (Lietz) data collectors allow you to enter attribute data. To use attribute data with CG-SURVEY, it must be appended to the description records in the following format:

DESCRIPTION [attribute name] attribute

For example: PIPE [Diameter] 18"

Entering Data into the Data Collector

Note: that all of the following assume that you have a Getting Started book and Operator's Manual from the Lietz Company.

1) To begin a new job, press <clear> until Select operation appears in the display. Press the <Menu> key. When JOB appears in the display, press <Enter>, and enter the job name and scale factor.

2) To enter field data, press <clear> until Select operation appears in the display. Press the <Prog>. When Traverse appears in the display, press <Enter> and begin traversing.

2a) You may use the TOPO program rather than the Traverse program. If you use this program, you must use the R option when sending data to your computer. The "Transferring Field Data to the Computer" section of this contains more about this.

Note: The coordinates that you enter for the first instrument point are for the internal use of the SDR2, and can be changed when your field data gets to the computer.

Note: When recording your first backsight information, simply enter an azimuth from the instrument point to the backsight (0.0000 will do). The azimuth information is also only used internally in the SDR2. You can change all of that as you reduce your field notes on the computer.

Note: At each instrument setup, the first angle recorded must be to your backsight. Your instrument may be "zeroed" or not, but when the Traverse Reduction program runs, it will subtract the backsight angle from the foresight angles.

Transferring Field Data to the Computer

If you have already dumped the data stored on the SDR to a computer file (in the SDR format), you may choose option to receive the data from the file. Enter the file name that contains the data, and the file name for the .CGR file (raw data file) or .CRD file (coordinate file).

Before data can be transferred in either direction between the computer and the SDR2, you must set up the transfer parameters in the SDR2. Once these have been set, they will not change, until you change them again. You do not have to set them each time. (The only parameter that you may wish to change is the baud rate.)

In our tests, the computer can receive data from the SDR2 at 4800 baud, its fastest speed, but the SDR2 could only receive points at 1200 baud. For fastest transmissions to and from the SDR2, you might wish to change this parameter in the SDR2.

The SDR-22 and SDR-24 data collectors will send to the computer at 9600 baud and receive data from the computer at 4800 baud.

1) Set up parameters by pressing <clear> until the message Select operation appears in the SDR2 display. Then, press <Menu>. Press the up or down arrow until Parameters appears in the display, then press <Enter>. You can then go from one parameter to the next by pressing the up or down arrows. When a parameter you wish to change shows on the display, press the <Edit> key, and change it. (See the SDR2 operator's manual.)

These parameters must be in effect:

Baud: 4800 (or 1200 for sending to the SDR2, see above discussion.)

Parity: not set
Word length: 8
Checksum: N
Coords: N-E-Elev

2) After the parameters have been set, simply connect the SDR2 to the computer, and select the transmission option. (On the Lietz transfer cable, there is a switch that must be set to DTE.)

3) Choose the same baud rate at the computer as you selected in the SDR2 parameters. When the computer says Waiting for data..., press <clear> on the SDR2 until Select operation shows in the display window. Press the <Menu> key. Press an up or down arrow key until Comms output shows in the display, then press <Enter> on the SDR2. Answer <N> if you do not wish to send all jobs. Then enter the job that you do wish to send. (See the SDR2 manual for a complete discussion of this process.)

4) The SDR2 should then send its information to the computer.

Note: As each job record is encountered in the computer will ask you for a file name to store the data in. You may press <Enter> to use the same name as was used in the SDR2 or enter another name. You must use a valid DOS name (all numbers and letters of 8 or less characters will be fine.)

Attribute data collected by the SDR (13AT records) is appended to the descriptions as follows: DESC[Attribute Name]Attribute

Example: PIPE[diameter]18"

Transferring Coordinates to the SDR2

Be careful of the units when transferring coordinates. For example, if the SDR2 is set to Metric Mode, the SDR2 will automatically convert the coordinates from feet to meters. Before you can transfer coordinates to the SDR2, you must first set up the transfer parameters in the SDR2. Refer to the first part of the previous section for details about how to do this. Then:

- 1) Ready the SDR2 by pressing <clear> until Select operation appears on the display. Press the <Menu> key. Then press the up or down arrows until Comms input appears on the menu. Press <Enter>.**
- 2) Select the Send Coordinates option and press Transfer.**
- 3) Select the coordinate file on the computer.**
- 4) Choose which coordinates to send.**

SOKKIA (LIETZ) SDR33 DATA COLLECTOR

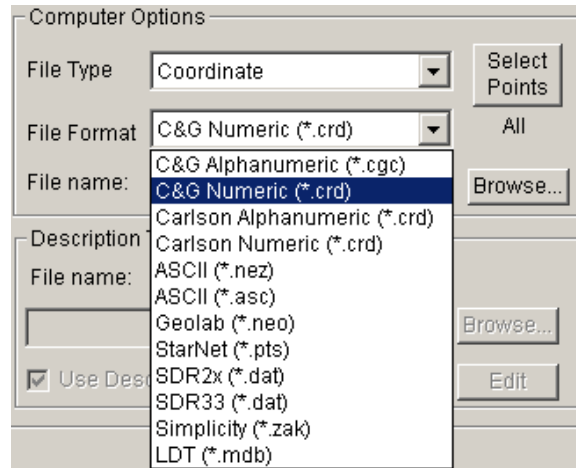
The SDR33 works the same as the SDR2. In Equipment Options there are two SDR33 choices, because when creating a new job on a SDR33 the format is determined by setting the Point ID field to Numeric (4) or Alpha (14).

SDR33 4-Pt

To transfer data to CG-SURVEY, select this setting if your SDR33 is set to Numeric (4). The highest point number allowed is 9,999.

SDR33 14-Pt

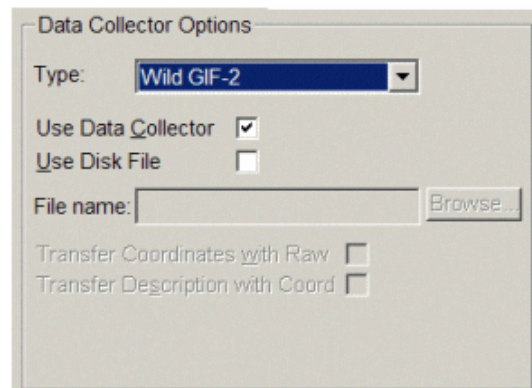
The C&G *.cgc files allow 10 character Alphanumeric point numbers. While C&G *.crd files allow only 5 digit numeric point numbers. To transfer a SDR33, set to Alpha (14), the file format setting must be set to *.cgc.



Note: See opening section of this chapter for detailed instructions on the settings dialog box and on sending and receiving files.

LEICA (WILD) GRE3/GRE4 AND GIF-2 INTERFACE

Set Up for GRE3/GRE4



Select the Wild: GIF-2 under Equipment Options. Before transferring data from the GRE3/4 to your computer, you must first set up the transfer parameters in the GRE3. To do this follow these steps on the GRE3/4:

- 1) <Set> <Mode> <7><0> <Run> <4><8><0><0> <Run> <Run>
(Sets the baud rate to 4800. If you wish, you may leave it at 2400, which is the rate the T2000 needs to communicate with the GRE3/4.)
- 2) <Set> <Mode> <7><1> <Run> <2> <Run> <Run>
(Sets even parity.)
- 3) <Set> <Mode> <7><2> <Run> <1> <Run> <Run>
(Use protocol.)
- 4) <Set> <Mode> <7><3> <Run> <0> <Run> <Run>
(<CR> only.)
- 5) <Set> <Mode> <7><4> <Run> <2> <Run> <Run>
(ACK/NAK + <CR>.)
- 6) <Set> <Mode> <4><0> <Run> <4> <Run> <Run>

(DDD.MMSS).

7) <Set> <Mode> <4><1> <Run> <1> <Run> <Run>

(Feet).

8) <Set> <Form> <.> <Run> <Rec>

(for the T2000)

-or-

<SET> <FORM> <+/-> <.> <RUN> <1> <1> <RUN> <REC>

(for the T1000)

Note: The above parameters do not "go away" when the GRE3/4 is switched off. They will stay the same until you change them or re-initialize everything.

Note: See opening section of this chapter for detailed instructions on the Settings dialog box & information on sending and receiving files.

Switch Settings/Cable

Option 1

GIF-2 Switches Cable Configuration

= X GRE-3/4 Computer

S1 < 2 < > 3

S2 < 3 < > 2

S7 < 7 < > 7

5 <

(Jump 5, 6, 8, 20) 6 <

8 <

20 <

Option 2

GIF-2 Switches Cable Configuration

= X GRE-3/4 Computer

S1 < 2 < > 3

S2 < 2 < > 2

S7 < 3 < > 7

7 <

5

(Jump 5, 6, 8, 20) 6 <

8 <

20 <

Data Collection Format for GRE3, GRE4, GIF-10

The transfer program expects your data to be in a specific format. To get your data in this format, follow these steps.

1) To begin a new job enter a "CODE 1" block into the GRE3/4. Example:

RDY [CODE]

CODE [1]

I1 ? [RUN]

your job number (Example: 87001)

[RUN]

I2 ? job date (Example: 091687 for Sept. 16, 1987)

[RUN]

I3 ? [REC]

2) Define the first automatic point number for your first foresight. Example:

RDY [SET]

SET [NR0]

S NR [2] for point number 2 as first foresight.

[RUN]

3) At each instrument point, enter a "CODE 2" block into the GRE3/4. Example:

RDY [CODE]

CODE [2]

[RUN]

I1 ? [instrument point number] EX: [1] for instrument point 1.

[RUN]

I2 ? [instrument height] EX: [550] for 5.50 feet.

[RUN]

I3 ? [REC]

Note: Each instrument point "CODE 2" block must be followed by a measurement reading to your backsight. You will probably need to change the point number for the backsight by:

RDY [NR]

NR [point number] EX: 4 for backsighting point 4.

[RUN]

THEN:

RDY [MEAS]

REC [REC]

-or-

ALL on T1000 will store in GRE3/4

4) Record your foresights. If necessary, change the rod height and/or the description of the foresight.

Use "CODE 3" or "CODE 4" to do this. Both codes are essentially the same, but one asks for the description first and the other asks for the rod height first, allowing you to skip the second entry by pressing [REC] rather than entering a value (see the third example below). This step may be skipped if you do not wish to change either rod height or description from the previous entry.

Example 1:

RDY [CODE]

CODE [3] Code 3 = description, rod height

[RUN]

I1 ? [0][1] Description = 01 (must be 2 digits)

[RUN]

I2 ? [5][5][0] Rod height = 5.50 feet

[RUN]

I3 ? [REC]

See following section on Setting-up Description Codes.

Example 2:

RDY [CODE]

CODE [4] Code 4 = rod height, description

[RUN]

I1 ? [5][5][0] Rod height = 5.50 feet

[RUN]

I2 ? [1][0] Description = 10 (must be 2 digits)

[RUN]

I3 ? [REC]

Example 3:

RDY [CODE]

CODE [3] Code 3 = description, rod height

[RUN]

I1 ? [1][2] Description = 12 (must be 2 digits)

[RUN]

I2 ? [REC] (leave the rod height the same)

5) Take your measurement (make sure that the point number is correct first):

RDY [MEAS]

REC [REC]

6) Now, go to step 4 for another foresight or to step 3 for another instrument set-up:

Uploading from Data Collector to the Computer

If you have already dumped the data stored on the GIF-2 to a computer file you may choose to receive the data from the file. Enter the file name that contains the data. After you have collected your field data, connect the GRE3/4 to your computer. Then select the receive option on the computer. The baud rate in the computer must match the baud rate in the GRE3/4. After selecting the baud rate on the computer, follow the steps on the screen to initiate transmission.

Those steps are:

- 1) **Connect the GRE-3/4 to the computer, turn it on,** and wait for RDY to show on the display.
- 2) **Put the GRE-3/4 in upload mode by pressing <Data>** on the GRE-3/4.
- 3) **Press <GoTo>, press <Run>, and wait for the GRE-3/4 to display D CD.**
- 4) **Press <Send> and press <Run>** on the GRE-3/4.

After the computer detects the end of transmission, it will begin to format the data in a usable form. When each job record is encountered (CODE 1), you will be prompted to give the computer the name that you want to enter the file name for that job.

Sending Coordinates to the GRE-3/4

From the Menu, select the send coordinates option. The baud rate in the computer must match the baud rate in the GRE3/4. Initiate transmission on the GRE-3/4 by doing the following:

Note: See opening section of this chapter for detailed instructions on the Settings dialog box & information on sending and receiving files.

- 1) **On GRE-3/4, SET MODE 80 <Run> 2 <Run> <Run>** to select file 2.
- 2) **On GRE-3/4, SET MODE 81 <Run> <#blocks> <Run> <Run>** to dimension file 2.
- 3) **Press any key to continue:**
- 4) **Next, select the points you wish to send:** The transmission will begin.

LEICA (WILD) GIF-10 INTERFACE

Note: (Refer to data collection format GRE3 within this section)

Receiving Raw Data from GIF-10 Interface:

Select the receive raw data option on the computer. If you have already dumped the data stored on the GIF-10 to a computer file (in the Wild format), you may choose receive the data from the file. Enter the file name that contains the data.

The baud rate in the computer must match the baud rate in the GIF-10. Do the following on the GIF-10:

1) Set the comm parameters as follows:

Baud: (your choice)

Parity: EVEN

Protoc: ACK/NAK

Stop Bit: 2

End Mark: CR

Connected AS: DTE

2) Put the GIF-10 in upload mode by selecting <Send> on the GIF-10.

3) Press <Run> on the GIF-10 and select the file you wish to send.

4) Press <Run> on the GIF-10.

Sending Coordinates to the GIF-10 Interface:

Select the send coordinates option. The baud rate in the computer must match the baud rate in the GIF-10. Do the following on the GIF-10:

1) Create receive file in GIF-10 now.

2) Put the GIF-10 in download mode by selecting Receive on the GIF-10.

Select the points you wish to send. The transmission will begin.

Note: See opening section of this chapter for detailed instructions on the Settings dialog box & information on sending and receiving files.

LEICA (WILD) GIF-10/WS

This is a Wild GIF-10 interface that will accept data in the same format as WildSoft. You will be asked for the Observation Pattern when the file is transferred. This pattern can be either BS-FS-FS-BS or BS-FS-BS-FS. As with WildSoft, codes 101 and above will be treated as descriptor codes. Code 100 will be subtracted from the descriptor code and that description will be read from CG-SURVEY's description table. So 101 is description 1, 102 is description 2, and so forth.

The following table shows acceptable WildSoft data collection codes:

WildSoft Data Collection Codes

Code Function

1 Start Job

2 Occupy a Point

3 FS to Traverse Point

11 Assign Coordinates

13 Target Height

14 Add to Target Height

21 Occupy Saved Point
31 FS to Single Point
32 Radial Sideshots
33 Sets of Angles
63 Remark
101+ Descriptions

LEICA (WILD) GIF-10/WS2

This is the exact same interface as the GIF-10/WS except 100 is not subtracted from the descriptor code.

LEICA (WILD) GIF-10/TOPOS

This is a Wild GIF-10 interface that will accept data in the same format as the Canadian software TOPOS. To select this format, choose GIF-10/TOP from the data collector choices in the Equipment Options. The following information explains the format.

These six Wild codes are used:

Code	Field 1	Field 2	Field 3	Field 4	Rectype
91	Job Name	Date	Temperature	Pressure	New Job
10	Label #	HI	Instrument	Pt.	
20	Label #	RH	Backsight	Pt.	
30	Label #	RH	Trav.	Pt.	
40	Label #	RH	Side Shot	Pt.	
2	Label #	RH	Offset	Angle	SS

- 1) If RH (rod height) is 999 it will be considered no value (do not calculate elevation for this point).
- 2) The Label# (point description number) can contain up to eight characters. The first four and last four will be read as separate descriptions. For example, if Label# is 00210034, then description 21 will be pulled from the description table and description 34 will be pulled from the description table. If 21 is BL* and 34 is TC, then the resulting description will be BL* TC.
- 3) Point numbers are taken from measurement records. A measurement record will follow code 10, 20, 30, and 40 records. For example: In this example, there is a side-shot record (40), a point label description (71), a rod height (2.150), and a point number (332).
- 4) If an offset distance is placed in Field 3 of a side-shot record, the measured angle will be shown in a comment line prior to the data record with the newly calculated angle.
- 5) If a comment is placed in field 4 of a side-shot record, the comment will be appended to the point's description. example: If the label# is 25 and the comment is 150, description 25 (lets say TREE) will be pulled from the description table and the comment will be appended to the description, giving TREE 150 as the description.

Leica Data Pro:

You are allowed to read and write to the Leica Data Pro formatted GSI files. There is no communication directly with the Leica Total stations.

GEODAT 122/124 DATA COLLECTOR

In order to use the Geodat 122 or 124 data collector with the transfer program, there are three areas that you

must consider: (1) entering data into the data collector in a format that can be sent to your computer, (2) the transfer program itself, and (3) the data collector code conversion table which converts numeric codes to more readable descriptions when the data is sent to the computer. The following section describes how to enter your data into the data collector. The next section will then give you some information about how to transfer the data. The data collector code conversion table can be changed with menu selection E from the program menu.

Entering Data into the Data Collector

1) Each individual job stored in the Geodat's memory should begin with a job identifier. To enter a job identifier, follow these steps:

a) Press the <Info> key.

b) At the prompt "inFo=" enter a job number, like this: inFo=87001<Ent> (<Ent> means to press the <Ent> key.)

c) At the prompt "dAtA=" enter the date like this: Example: dAtA=050187<Ent> (for 05/01/87)

2) For each instrument location, you must enter an instrument point identifier.

Follow the following steps to do this:

a) Press the <Stn> key.

b) At the "Stn=" prompt enter you instrument point number like this: Stn=1<Ent> (for instrument point 1)

c) At the "iH=" prompt enter the instrument height, like this: iH=5.5<Ent> (for 5.5 feet)

d) Next the prompt "PCod=" will appear on the display. At this time, enter your backsight point, just like the foresights in the next step. You may enter "0" (zero) for all of the fields except:

"Pno=" (enter the backsight point number)

"Hor=" (enter the angle in your instrument when you take the backsight.)

3) Recording foresights. You are now ready to record a foresight:

a) At the "PCod=" prompt enter the point code for your foresight like this: PCod=10<Ent> (for point code 10)

Note: The point code will be used to assign a description to your foresight. The description that is associated with each code is up to you. Use the "Edit data collector code table" program to set up your codes before uploading your data to the computer.

b) At the "Pno=" prompt enter your foresight point number like this: Pno=2<Ent> (for point number 2)

c) At the "SH=" prompt enter the rod height of your foresight like this: SH=5.5<Ent> (for 5.5 feet)

The next three fields may be entered manually or may be automatically stored by your instrument.

d) At the "Hor=" prompt enter the horizontal angle to your foresight like this: Hor=65.1253<Ent> (for 65 degrees, 12 minutes, and 53 seconds.)

e) At the "ELE=" prompt enter the vertical angle to your foresight like this: ELE=90.1215<Ent> (for 90 degrees, 12 minutes and 15 seconds)

f) At the "diSt=" prompt enter the slope distance to your foresight like this: diSt=100.128<Ent> (for 100.128 feet)

Now, go to step 2 for a new instrument point, or step 3 for another sight from the current instrument point.

Note: If you have already dumped the data stored on the 122/124 to a computer file (in the 122/124 format), you may choose to receive the data from the file. Enter the file name that contains the data, and the file name for the .CGR file.

Receive Data from Data Collector

Select the receive raw data option.

1) Before you upload your data, make sure that the description table is current.

2) Before you initiate the upload program, you must first define the upload parameters for the Geodat. To do this, follow the steps outlined below.

3) This should not have to be done each time. The values that you enter should stay the same until you change them.

a) Press the <f>, the <1>, the <0> and the <Ent> keys. (For function 10.)

b) Answer the baud rate question like this: bAud=1200<Ent> (for 1200 baud you can use 300 baud, but it will take longer to transfer your data.)

c) Make sure that "Eob=" looks like this: Eob=0123456789<Ent>

d) Make sure that the "StArt=" prompt looks like this: StArt=035<Ent>

e) Make sure that the "StoP=" prompt looks like this: StoP=000<Ent>

f) Make sure that the "ErrCodE=" prompt looks like this: ErrCodE=037<Ent>

g) Make sure that the "nuLLS=" prompt looks like this: nuLLS=000<Ent>

GEODAT 126, 400, 500 AND INTERNAL MEMORY THEODOLITES

These Geodat data collectors use the following data entry format:

Instrument Point Setup

Labels Explanation

*2 Instrument Point

3 Height of Instrument

*62 Backsight Point

21 Backsight Angle

6 Backsight Rod Height

7 Horizontal Angle to Backsight

8 Vertical Angle to Backsight

9 Slope Distance to Backsight

* Indicates required code

Note: Pcodes (label 4) cannot be used anywhere except in foresight records.

Note: The order of the instrument point setups is not important.

Foresight Points

Labels Explanation

*5 Foresight point

6 Backsight rod height

*7 Horizontal angle to backsight

*8 Vertical angle to backsight

*9 Slope distance to backsight

4 Pcode (Description)

* Indicates required code

Note: Foresight points must begin with either a Pcode (label) or a foresight point (code 5). The order of the remaining parameters is not important.

Label 4 (Pcodes) are placed in the description field of the raw data file. If you cannot get the entire point description into a single Pcode, we allow you to use multiple Pcodes for an individual point.

Example: 4 = Manhole,

4 = Inv. -10.23,

4 = 12" Conc.Pipe

The resulting point description is: Manhole, Inv. - 10.23, 12" Conc.Pipe If the Append Info Records to Pcode toggle is on, info records (label 0=) that directly follow a Pcode (label 4=) will be appended to the Pcode prior to being placed in the point description.

Example: 4 = Manhole,

0 = Inv. -10.23,

0 = 12" Conc.Pipe

The resulting point description is: Manhole, Inv. - 10.23, 12" Conc.Pipe

GEODAT 126 DATA COLLECTOR

I. The Cable

Your cable should be made as follows:

NC - No connection.

Geodat 126 (male)Computer 25 Pin

2 2 (TxD)
3 3 (RxD)
7 7 (S.GND)
5-NC ---5 (CTS) jumper 5-6-8-20
6-NC ---6 (DSR)
8-NC ---8 (CD)
20-NC ---20 (DTR)

Geodat 126 (male)Computer 9 Pin

2 3 (TxD)
3 2 (RxD)
7 5 (S.GND)
5-NC ---8 (CTS) jumper 8-6-1-4
6-NC ---6 (DSR)
8-NC ---1 (CD)
20-NC ---4 (DTR)

II. Set Protocol 2 and 5; Set Format 2

Be sure the INT./EXT. switch is set to INT. if you are not connected to an external power source. Be sure the on/off switch is in the on position.

Set protocol by using program 51 in the Geodat.

Protocol 2 Protocol 5

1: 9600 1: 9600
2: 2 2: 2
3: 7 3: 7
4: 2 4: 2
5: 10 5: 10
6: 0 6: 0
7: 0 7: 0
8: 0 8: 0
9: 0 9: 0
10: 0 10: 0
11: 0 11: 0

12: 0 12: 0
13: 0 13: 0
14: 1.13 14: 1.13
15: 0 15: 0
16: 3.371310 16: 3.371310

Set format by using program 50 in the Geodat.

Format 2

1: 1
2: 80
3: 13
4: 15

III. Data Storage in the Geodat 126

Raw data is gathered into job files using the pre-programmed UDS's in the Geodat 126. Call us if you wish to create others.

To use the existing programs, begin by choosing program #10. This sets up the header information and first instrument/back-sight points. For foresights, choose program #0 if you are carrying elevations or program #1 for horizontal locations only. Use program #11 to change instrument set-ups. These programs are explained on page 8:4 in the Geodat manual.

Coordinates are transferred from and into area files.

Special numeric point codes may be used. These codes are converted to alphanumeric descriptions as the data is received from the Geodat 126. The codes are defined in the description table.

IV. UDS Requirements

The initial testing of the Geodat 126 was done using the standard UDS's supplied with the Geodat 126. If you wish to try using your own, these rules apply:

- 1) A measurement must end with label 9. (See sample UDS's 0, 1, 2, 3.)
- 2) Each Job. No. file must begin (1st record) with ADM type data (see sample UDS 10). This sequence must end with label type 74 - Air Pressure.
- 3) Each instrument station sequence must end with Hz. Ref. (Label 21). See the UDS 11 in Geodat 126 manual for sample.

Select the receive/send option from the computer menu and follow the prompts. If you have already dumped the data stored on the DR-2 to a computer file (in the DR-2 format), you may choose to receive the data from the file. Enter the file name that contains the data, and the file name for the .CGR file (raw data file) or .CRD file (coordinate file).

GEODAT 400 DATA COLLECTOR

Setting up the Geodat 400 for use with the transfer program:

- 1) Set protocol 0 (as shown below).
- 2) Set format 0 (as shown below).

Be sure the on/off switch is in the on position. Set protocol by using program 51 in the Geodat field instru-

ment.

Protocol 0

1: 9600
2: 0
3: 8
4: 1
5: 10
6: 0
7: 1
8: 17
9: 19
10: 0
11: 0
12: 0
13: 0
14: 1.13
15: 0
16: 1.04

Line 7 implements software handshaking between the 400 and the MS-DOS computer by using a value of 1. When the value of item 7 is 0, then no software handshaking is done.

Line 8 is given a value of 17 which is the Xon value used for the communication handshaking.

Line 9 is given a value of 19 which is the Xoff value used for the communication handshaking.

Set format by using program 50 in the Geodat field instrument.

Format 0

1: 1
2: 80
3: 324:
4 *

Note: It is important that the values above be set as we show them or our software can not communicate with the Geodat 400 Data Recorder.

If you have already dumped the data stored on the 400 to a computer file (in the 400 format), you may choose to receive the data from the file. Enter the file name that contains the data, and the file name for the .CGR file (raw data file) or .CRD file (coordinate file).

Uploading Raw Data to the Computer

After you have collected your field data, connect the Geodat 400 to your computer. Select the receive raw data option. The baud rate in the computer must match the baud rate in the Geodat 400. Do the following:

- 1) **Connect the 400 to the computer, turn it off, then on.**
- 2) **Make certain that you have selected the correct protocol and format.**
- 3) **Enter name of Geodat job file.**

Download Coordinates into 400 Area File

Select the send coordinates option.

Select the points to send.

Ready the 400 with the following steps:

- 1) **Connect the 400 to the computer, turn it off, then on.**
- 2) **Make certain that you have selected the correct protocol and format.**
- 3) **Enter name of Geodat area file.**

The coordinates will be transferred.

Get Coordinates from 400 Area File

Receive coordinates from 400 Area File

The baud rate in the computer must match the baud rate in the 400. Do the following:

- 1) **Connect the 400 to the computer, turn it off, then on.**
- 2) **Make certain that you have selected the correct protocol and format.**
- 3) **Enter name of Geodat area file.**

The transmission will begin.

GEODAT 500 DATA COLLECTOR

Setting up the Geodat 500 for use with the transfer program:

- 1) Set protocol 0 (as shown below).
- 2) Set format 0 (as shown below).

Be sure the on/off switch is in the on position.

Set protocol by using program 51 in the Geodat field instrument.

Protocol 0

- 1: 9600
- 2: 0
- 3: 8
- 4: 1
- 5: 10
- 6: 0
- 7: 1
- 8: 17
- 9: 19
- 10: 0
- 11: 0
- 12: 0
- 13: 0
- 14: 1.13
- 15: 0
- 16: 1.04

Line 7 implements software handshaking between the 500 and the MS-DOS computer by using a value of 1. When the value of item 7 is 0, then no software handshaking is done.

Line 8 is given a value of 17 which is the Xon value used for the communication handshaking.

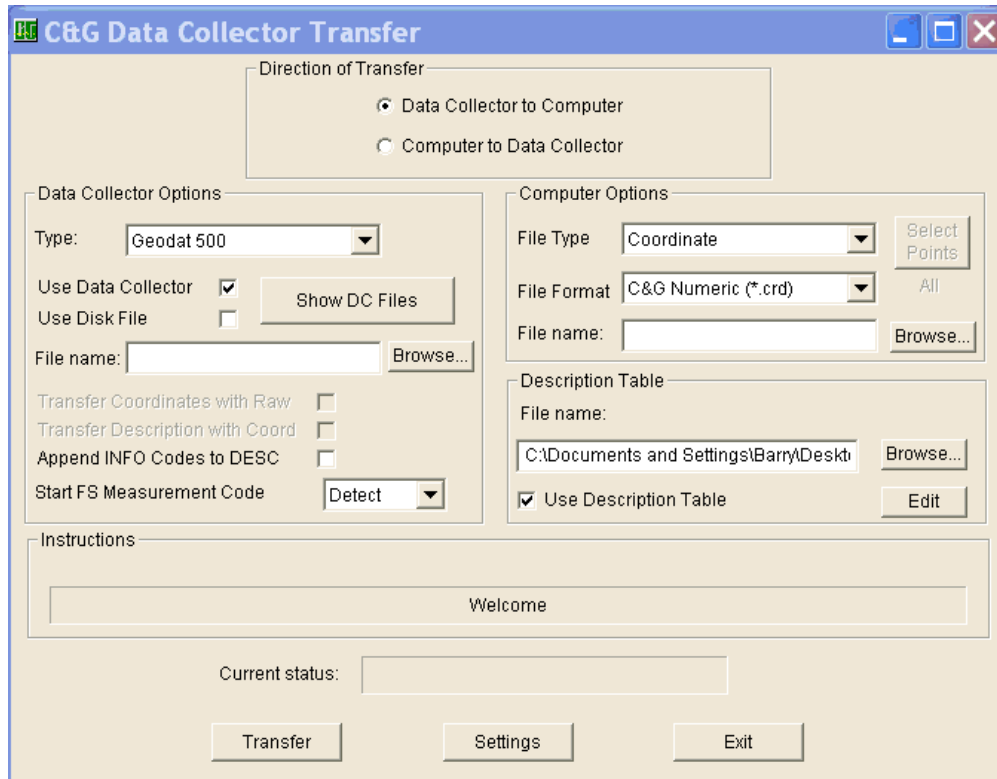
Line 9 is given a value of 19 which is the Xoff value used for the communication handshaking. Set format by using program 50 in the Geodat field instrument.

Format 0

- 1: 1
- 2: 80
- 3: 32
- 4: *

Note: It is important that the values above be set as we show them or our software can not communicate with the Geodat 500 Data Recorder.

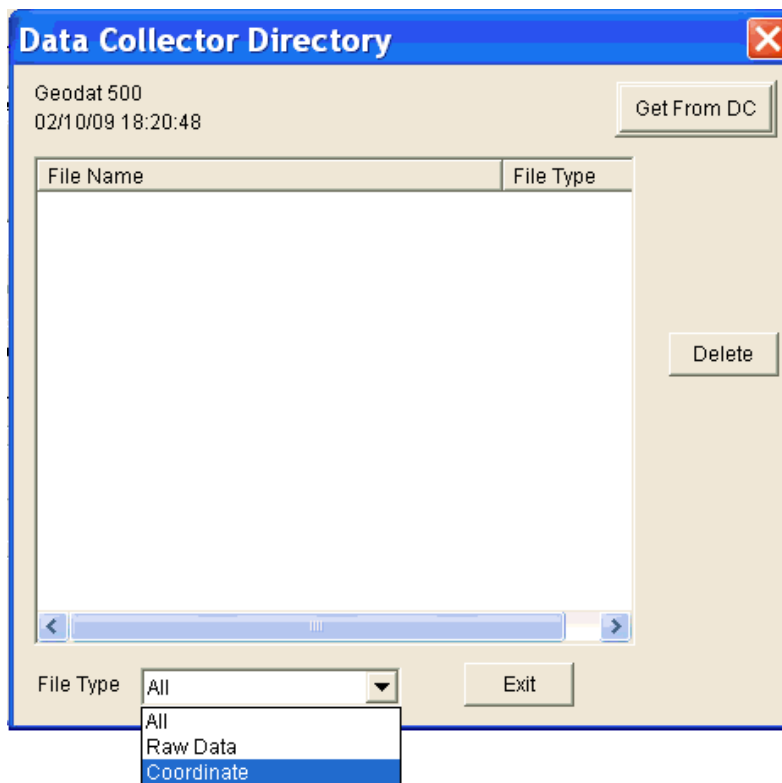
The C&G Data collector Transfer dialog box has an additional option for the GEODAT 500 collector, as shown below. the show DC files:



This Option actually reads and displays the data files on the GEODAT 500 data collector. From the display options you can select to view all files, just coordinate files or just raw files.

If you have already dumped the data stored on the 500 to a computer file (in the 500 format), you may choose to receive the data from the file. Enter the file name that contains the data, and the file name for the .CGR file (raw data file) or .CRD file (coordinate file).

You can also delete files from the GEODAT 500. Be careful, once the files is deleted it is gone forever. When you have the files selected, you want to transfer, select exit.



The Geodat 500 program allows:

- 1) Receive raw data from the Geo 500
- 2) Receive raw data from a file.
- 3) Receive coordinates from the Geo 500.
- 4) Receive coordinates from a file.
- 5) Send coordinates to the Geo 500.
- 6) Catalog (or directory) of all files on Geo 500.
- 7) Delete files on the Geo 500.

Receiving Data from the 500

You may receive raw data files (M=), or coordinate files (I=). All files on the 500 of the type you wish to receive will be shown on the screen (for example, all I= files for coordinate).

- 1) Select the file you wish to receive. Raw Data (Job Files)

- 2) After the raw data file is transferred, you will be asked to select the file name it will be stored under on the computer. The default value will be the same name with a .CGR extension. Coordinates (Area Files)

- 3) After the coordinate file is transferred, you will be asked to select the file name it will be stored under on the computer. The default value will be the same name with .CRD/.IDX extensions.

Sending Data to the 500

You may send coordinate (.CRD) files to the 500. All coordinate files on the computer will be shown on the screen. Select the file you wish to send.

- 1) You may select the only the coordinates that you wish to send (you do not have to send the entire file). Catalog
- 2) The catalog function will show you all existing files on the Geodat 500.

Deleting Files

All files on the 500 will be shown on the screen.

- 1) Select the file you wish to delete. Be careful, once the file is deleted it is gone forever.
- 2) Press <Esc> if you do not want to delete a file.

GEODIMETER TOTAL STATIONS WITH INTERNAL MEMORY

You can select Geodat 500 and interface directly with any Geodimeter that has internal memory. To transfer data from a Geodimeter Total Station with internal memory, do the following:

- 1) In Equipment Options, select Geodat 500 as the data collector and run data collection program.**
- 2) Use Geotronix cable #571136756. Connect RS232 on computer to RS232 on Geodimeter with cable.**
- 3) Power on Geodimeter and turn off compensator with Function 22 as follows:**

Key

<F> (Function)
<22>
<Ent> (Enter)
<0>
<Ent>

Then press <Ent> until P0 is displayed on Geodimeter screen.

- 4) Set the END character to 4 with Function 79 as follows:**

Key

<F> (Function)
<79>
<Ent> (Enter)
<4>
<Ent>

- 5) Initiate comm port on Geodimeter as follows:**

Key

<Mnu> (menu)
<4> (data com)
<1> (select device)
<2> (serial)
<Yes> (serial on)
<1.8.0.9600> (com=) skip if already set
<Ent> (enter)
<0> (table no=)
<Ent>
<No> (REG. key?)
<No> (Slave ?)

- 6) You may now select all options on the computer menu for data collection transfer with the Geodimeter. See Geodat 500 instructions for data transfer (disregard formatting procedures).**

SMI 48 ENHANCED DATA COLLECTOR

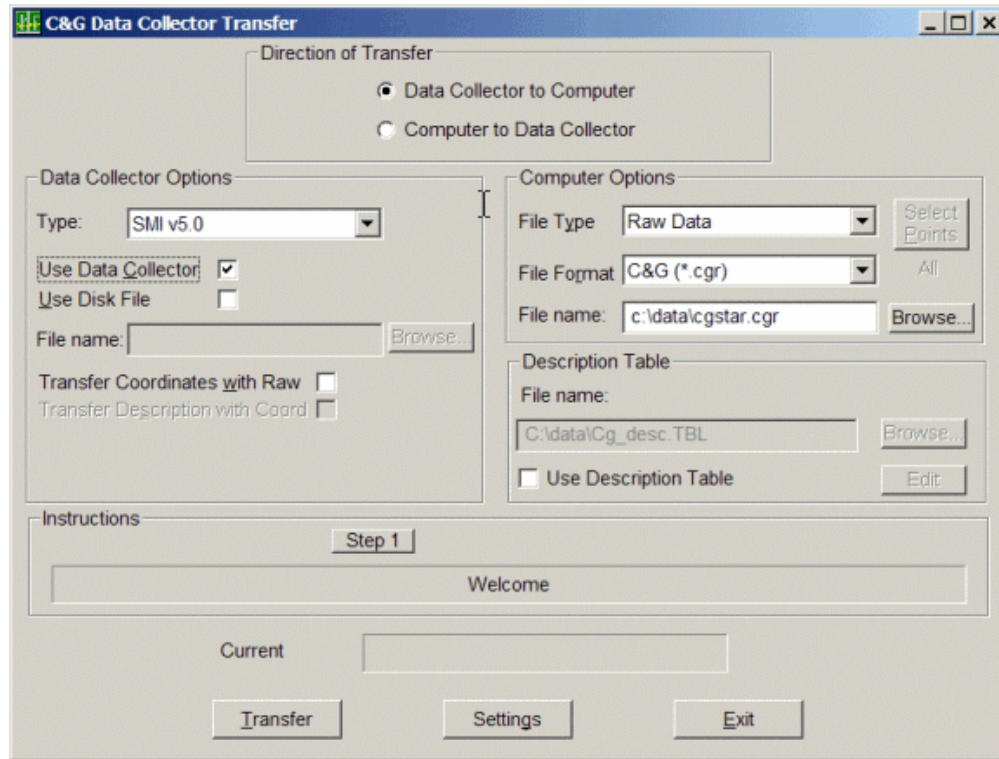
The SMI interface routine works only with SMI Enhanced Cards. Use the interface cable supplied with the SMI unit (plugs into the comm port on the computer).

SMI 48 transfer Versions 5

Receiving Data

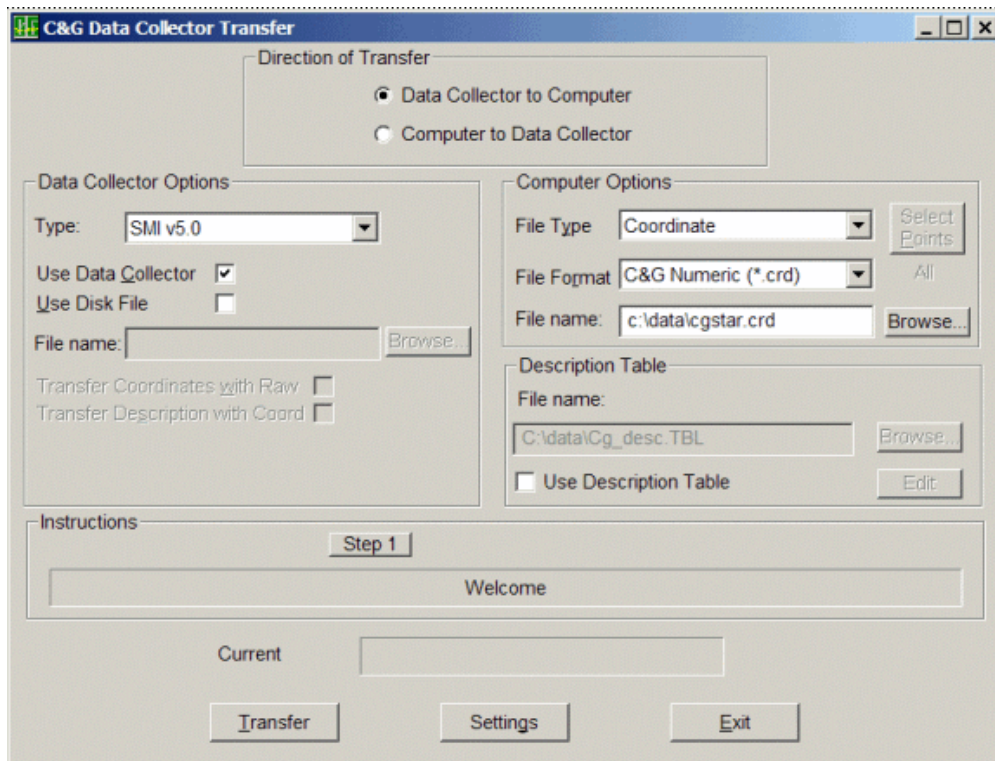
If you have already dumped the data stored on the SMI to a file on the computer, you may choose to receive the data from the file.

Receiving Raw Data from the SMI



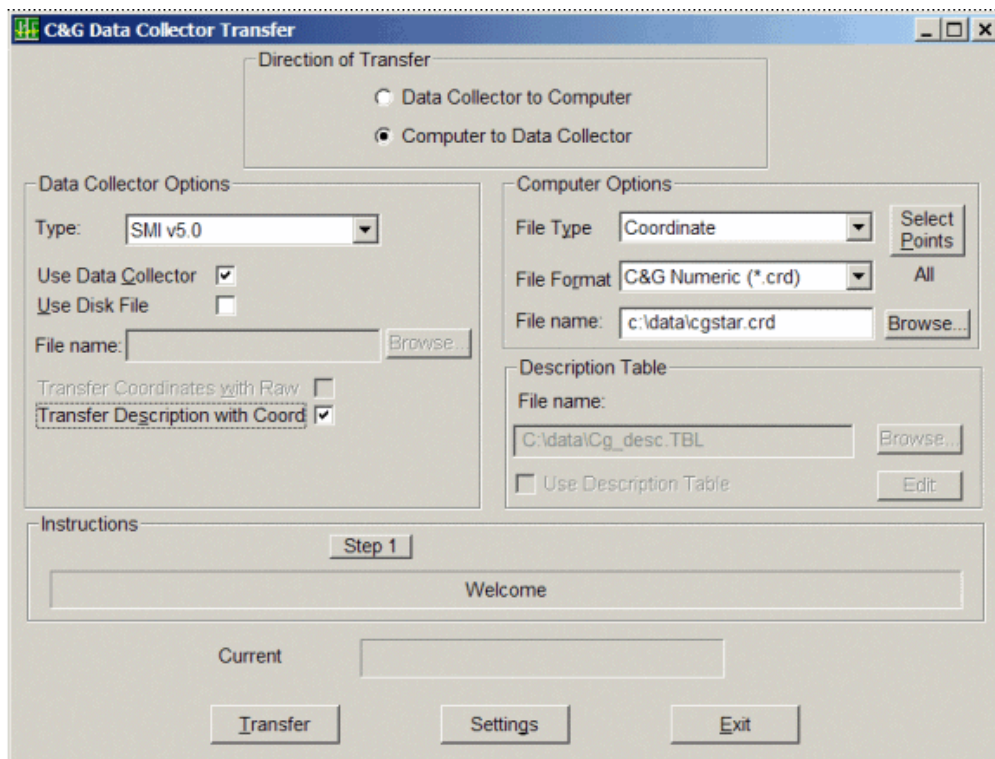
Select the Receive raw data option on the computer. On the SMI, select TOPC and then RAW. The transfer will begin. The file name will be shown on the screen after the transfer is complete. You may enter a new file name if you wish. Our reduction program does not allow a raw data file with mixed angle types (for example: azimuths, angles right, deflections, etc.). When you are collecting data on the SMI, stick to one angle type. You can mix distance types if you wish (slope/zenith, horizontal/vertical).

Receiving Coordinates from the SMI



- 1) Select the Receive Coordinates option on the computer: On the SMI select TOPC and then SMI.
- 2) On the SMI, enter the first and last point numbers you wish to send: The transfer will begin.

Sending Coordinates to the SMI



- 1) On the SMI select TD48 and the SMI.
- 2) Select the Send Coordinates option on the computer. You will be asked if you wish to send descriptions.

The answer to this question depends on whether the SMI coordinate file you are sending to is a 15 byte file (no descriptions) or a 30 byte file (descriptions).

3) **On the computer, select the points you wish to send. When the selection set is complete, press <T> for transmit.** The transfer will begin.

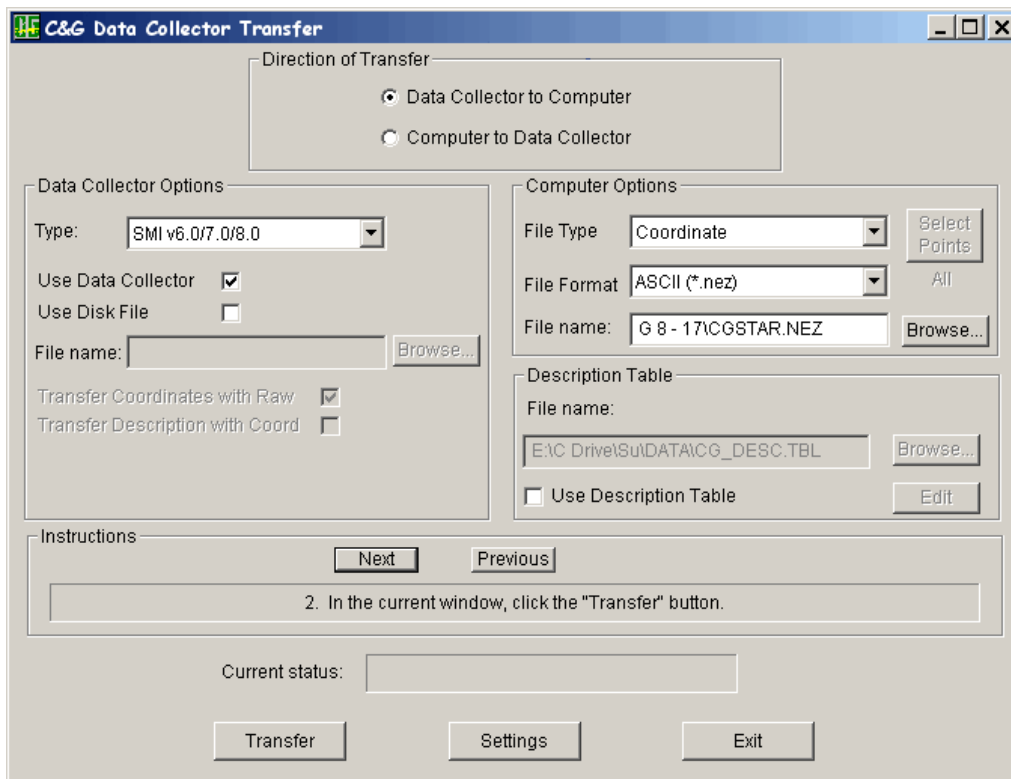
SMI 48 transfer Versions 6, 7 & 8

Receiving Raw Data from the SMI

1) **Get the C&G Transfer Program ready to receive raw data:** Press Transfer

2) **On the SMI data Collector select [PRINT]:** set the soft-key to [WIRE]

Receiving Coordinates from the SMI



1) **Get the C&G Transfer Program ready to receive raw data and Press Transfer**

2) **On the SMI data Collector select [JOB]. then [KERM]:** set the soft-key to [NE] and [COMM]. select [SEND] and then select the points to transfer.

Sending Coordinates to the SMI

1) **On the SMI data collector select [JOB] the [KERM]:** set the soft-key to [NE] and [COMM]. select [RECV].

2) **Configure the C&G Transfer Program to send Coordinates:** select the points to be sent and press TRANSFER

Nikon Data Collection Transfer

Receiving Raw Data from the Nikon Total Station:

- 1) **Get the C&G Transfer Program ready to receive raw data and Press Transfer**
- 2) **On the Nikon** select [MENU]. Select option [SET] and then option [COMM]. Set "Ext.Comm:" to Nikon. Set the communication parameters to match those in the C&G transfer program.
- 3) **From the Main Menu on the Nikon select "Comms" and "Download":** select format: NIKON and Data: RAW
- 4) Press ENTER to send.

Receiving Coordinates Data from the Nikon Total Station

- 1) **Get the C&G Transfer Program ready to receive raw data and Press Transfer**
- 2) **On the Nikon** select [MENU]. Select option [SET] and then option [COMM]. Set "Ext.Comm:" to Nikon. Set the communication parameters to match those in the C&G transfer program.
- 3) **From the Main Menu on the Nikon select "Comms" and "Download":** select format: NIKON and Data: COORD.
- 4) Press ENTER to send.

Sending Coordinates Data from the Nikon Total Station

- 1) **On the Nikon** select [MENU]. Select option [SET] and then option [COMM]. Set "Ext.Comm:" to Nikon. Set the communication parameters to match those in the C&G transfer program.
- 2) **From the Main Menu on the Nikon:** select "Comms" and "Upload Data". Press ENTER to receive.
- 3) **Configure the C&G Transfer Program to Send Coordinates:** select the points to be sent and press TRANSFER.

Pulldown Menu Location: CG-Survey>CGTrav>Data Collector Transfer

Keyboard Command:DC, CG_DATA_COLLECTOR

Prerequisite: Check Cable Connection & Communication Parameters

Reduce Traverse

The Reduce Traverse feature allows you to reduce a raw data file, with or without adjustment, and thus create a coordinate file or append to an existing coordinate file.

NOTE: Before you reduce a traverse, check the traverse settings on the Traverse Options tab of the C&G Options dialog.

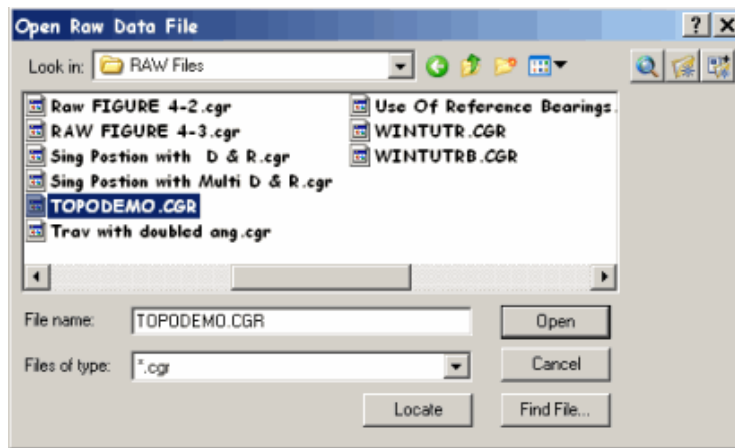
Select the type of adjustment to use: (Compass, Least Squares, etc..)

Adjust Angles: (off/on)

Balance Elevations: (off/on)

If you are adjusting a 3-D traverse, make sure Elevations are turned on: ON

Once the traverse options are set properly you can proceed with traverse reduction.



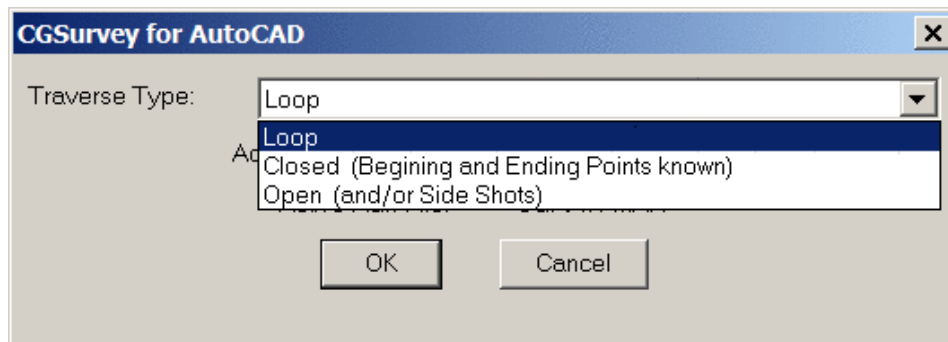
Select Reduce Traverse from the CGTrav menu.

If a raw data file is already open, it will be used. If not, a dialog box will appear prompting you to open a raw data file.

If a coordinate is already open it will be used. If one is not opened you will be prompted to open one. You can select an existing file or type in the name of a new file to create.

NOTE: One coordinate file may be used with many raw data files. For example, you may store the coordinates reduced from an initial boundary traverse (raw data file) in a newly created coordinate file. If you do additional location or traverse work with the control created by the original traverse, this additional work may be placed in new raw data files and reduced to the same coordinate file.

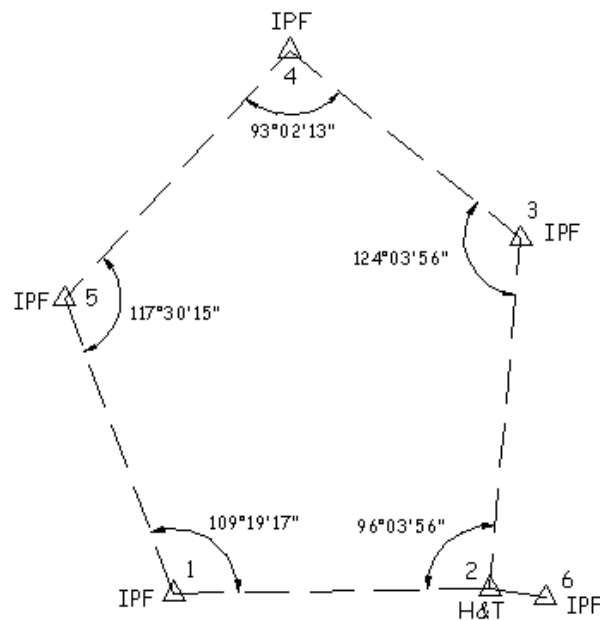
If the raw data file does not have traverse codes (*see the CGEditor chapter*) a dialog will appear asking you which type to use. There are three types of traverses that can be processed. These are shown in the following figure:



The following figures show examples of the three traverse types. The H.I. and rod height entries are optional (if Elevations are on). These are examples of a single distance/angle entry. Each type traverse may be placed in a separate raw data file and reduced into a single coordinate file. However, with the use of special codes you can combine traverses in a single raw data file (*See the CGEditor chapter*).

Traverse Reduction Types:

Closed Loop Traverse

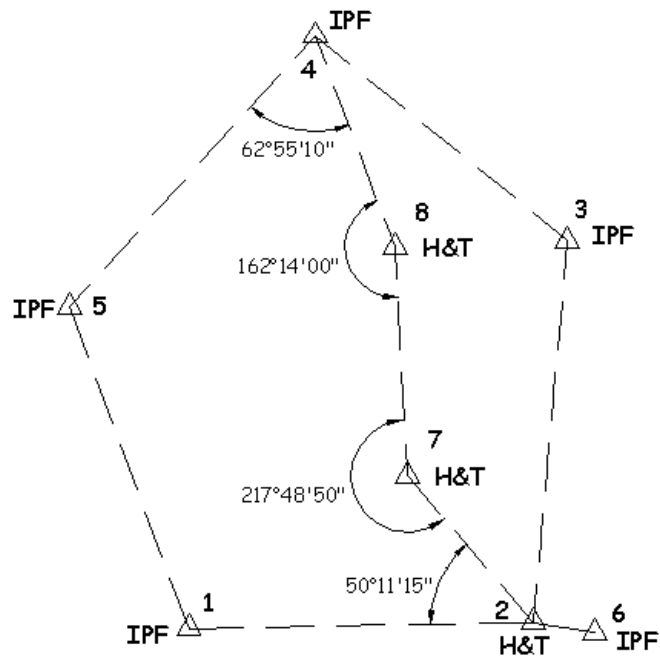


CLOSED LOOP TRAVERSE

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Horz.Angle	Slope Dist.	Vert.Angle	Foresight	Code	Description
IP	1	5.32000	5		0.00000					
FS				5.00000	109.19170	292.31000	88.35000	2	TP	H&T
IP	2	5.20000	1		0.00000					
FS				5.00000	96.03560	324.26000	88.51320	3	TP	IPF
FS				5.00000	190.32100	52.39000	90.32550	6	SS	IPF
IP	3	4.90000	2		0.00000					
FS				5.00000	124.03560	275.84000	92.22400	4	TP	IPF
IP	4	5.12000	3		0.00000					
FS				5.00000	93.02130	309.68000	89.13000	5	TP	IPF
IP	5	5.15000	4		0.00000					
FS				5.00000	117.30500	290.55000	90.00000	1	TP	IPF

Closed Loop Traverse Beginning and Ending at Ending at Known points

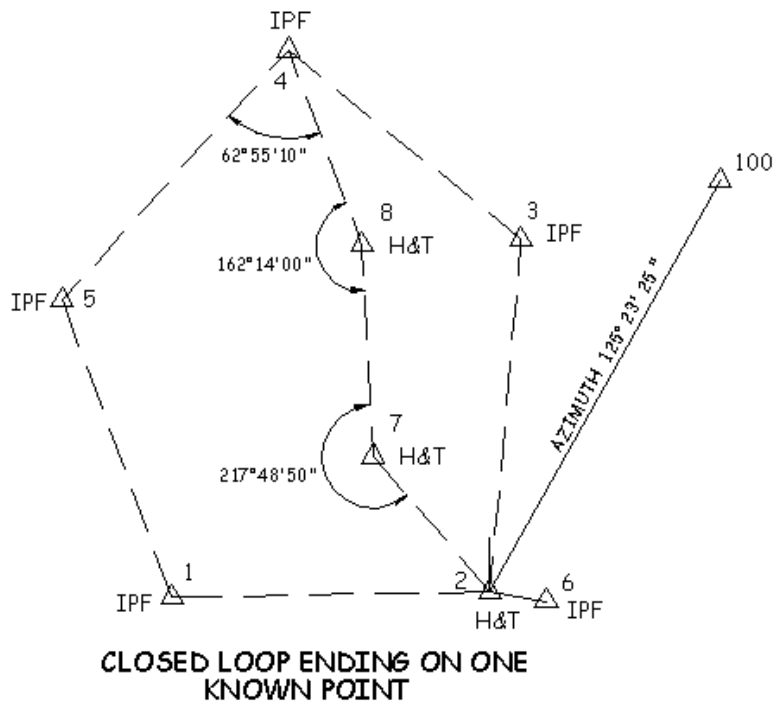
Shows above is closed traverse beginning on two known points (1 and 2) and ending on two known points (4 and 5). With this type of traverse, both a linear and angular closure can be calculated.



CLOSED LOOP TRAVERSE BEGINNING AND ENDING AT KNOWN POINTS

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Horz.Angle	Slope Dist.	Vert.Angle	Foresight	Code	Description
IP	2	9.0000	1		0.0000					
FS				5.0000	50.11150	164.12000	88.20200	7	TP	H&T
IP	7	5.2300	2		0.0000					
FS				5.0000	217.48500	193.91000	90.58300	8	TP	H&T
IP	8	5.3400	7		0.0000					
FS				5.0000	162.14080	191.57000	92.03140	4	TP	IPF
IP	4	5.3000	8		0.0000					
FS				5.0000	62.55100			5		

Closed Loop Ending on One Know Point



Set	From Point	To Point	Bearing							
DR	2	100	125.23240							
IP	2	4.90000	100		0.00000					
FS				5.00000	50.11150	164.12000	88.20200	7	TP	H&T
IP	7	5.23000	2		0.00000					
FS				5.00000	217.48500	193.91000	90.58300	8	TP	H&T
IP	8	5.34000	7		0.00000					
FS				5.00000	162.14030	191.57000	92.03140	4	TP	IPF
IP	4	5.30000	8		0.00000					
FS				5.00000	87.23150			101		

Shown above is a traverse that begins on two known points, or a single known point and a back sight azimuth, and ends on one known point.

This situation sometimes occurs when you begin on two known points (or a single known point and a back sight azimuth) and end on one known point. In this case only a linear closure is possible.

In order to reduce this type of traverse you must use the CGEditor to enter data not gathered in the field.

Points 2 and 4 are the known beginning and ending points.

Points 100 and 101 do not exist.

We have entered a back sight reference bearing (N 25° 23' 25" E) from 2 to 100.

Line 8 is a dummy setup (we never setup on point 4 and back sighted point 8).

Line 9 shows a dummy angle to the dummy point 101.

Reduce the traverse as a closed Traverse Beginning and Ending on Known Points.

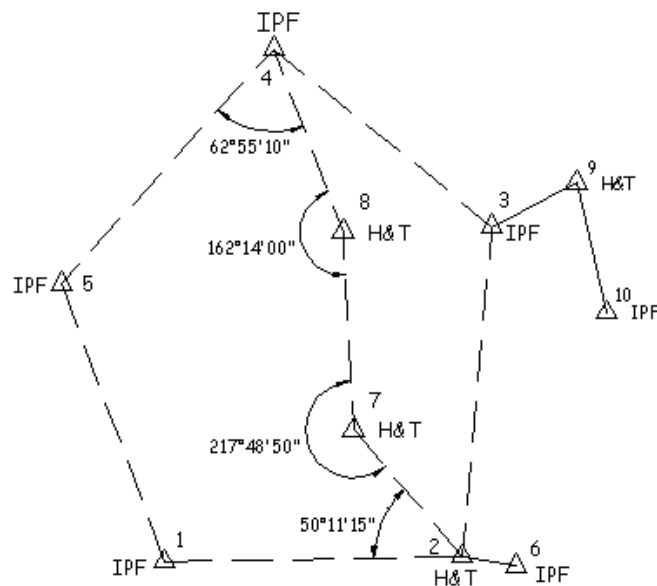
When the traverse is reduced you will have to enter one of the following:

The coordinates of point 101

The bearing from point 4 to 101. Or press <esc> for no angular closure.

If you choose no angular closure, the traverse will be reduced but will report only a linear closure. The adjustment will be made assuming no angular error.

Open Traverse



OPEN TRAVERSE - SIDESHOTS

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Horz.Angle	Slope Dist	Vert.Angle	Foresight	Code	Description
IP	3	5.12	4		0.00000					
FS				5.00	113.19430	92.91000	89.15300	9	TP	H&T
IP	9	5.26	3		0.00000					
FS				5.00	284.01250	130.44000	91.15210	10	TP	IPF

An Open Traverse is either an open ended traverse which ties into no known points or a file containing only side shots. In both cases no adjustment is possible.

Note: The data shown in the CGEditor views accompanying the four illustrations include instrument height (HI) and rod height entries. However, if you have elevations turned off, these entries are optional. Also, the examples use single distance and angle entries but multiple measurements are allowed.

In these figures each traverse has been placed in a separate raw data file. However, with the use of special codes you can combine multiple traverses in a single raw data file.

Notes on Traverse Types and Reduction

Closed and Azimuth Traverses: If you are running azimuth traverses, the angle to the side shot is calculated and stored instead of the azimuth. After the traverse has been reduced and adjusted, the angles are used to calculate the side shot coordinates. Thus the side shots are always relative to the instrument point and backsight point used in their location. The first azimuth in the raw data file will be considered a reference azimuth and will be held.

Reducing Loop Traverses:

If there is at least one reference bearing in the raw data file being reduced you will not be asked for a starting bearing. If the instrument point coordinates at the first reference bearing exists, you will not be asked to enter the starting coordinates or elevation. The traverse reduction will begin from the first reference bearing in the raw data file, not necessarily the first instrument point.

If you have more than one reference bearing in the raw data file, the angular closure and adjustments will be from one reference bearing to the next. In other words, all reference bearings will be held as correct, and any angle adjustment will be done from one to the next. This feature was designed for those surveyors who perform

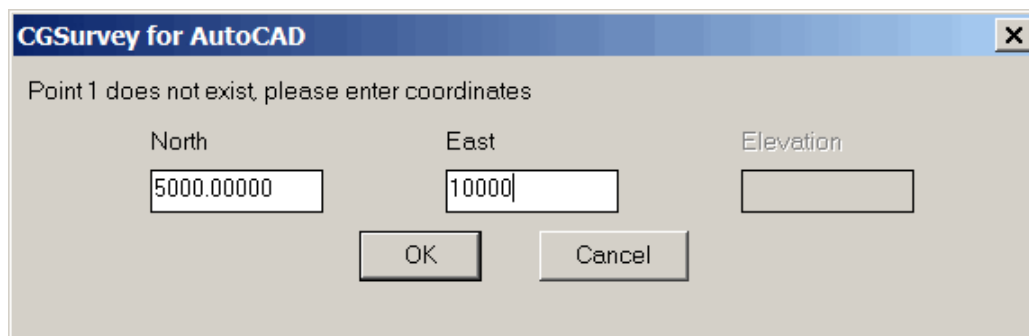
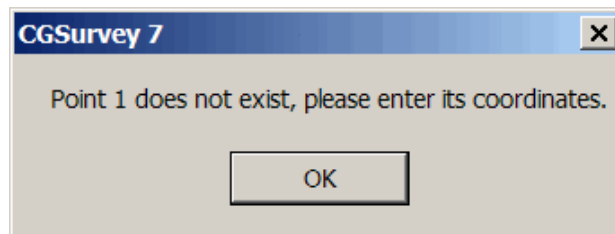
Solar or Polaris observations at intermediate traverse stations, and wish to hold the observed bearing at those stations (the bearings will of course change when the coordinates are adjusted, unless you use Crandall's Rule which does not change bearings)..

Reducing Open Traverse:

Any Reference Bearings found in the raw data file for an Open traverse will be ignored (except the starting reference bearing/azimuth to the back sight point).

Traverse Reduction: Closed Loop

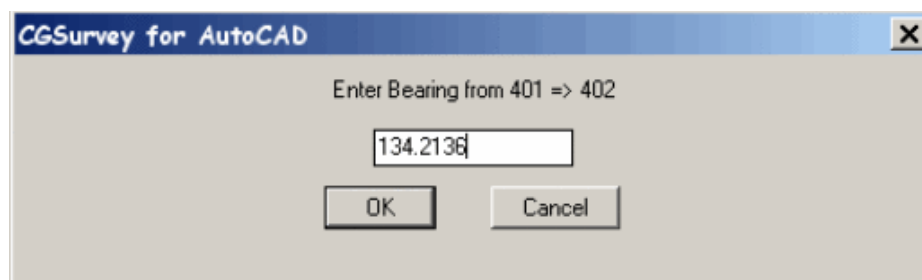
If the first instrument point in the raw data file does not exist, you will be asked to enter the coordinates for that point.



If the first back sight point in the raw data file does not exist and you do not have a reference bearing/ azimuth to the back sight point in the raw data file, you will be given the choice of entering one of the following:

Back sight point coordinates

Bearing from the first instrument point to the first back sight point



If you are processing a Closed Traverse that Begins and Ends on known points, and the last (tie) instrument point in the raw data file does not exist, you will be asked to enter the coordinates for that point. If the last foresight point in the raw data file does not exist and you do not have a reference bearing/azimuth to the foresight point in the raw data file, you will be given the choice of entering one of the following:

Foresight point coordinates

Bearing from the last instrument point to the last foresight point (the last instrument and foresight point are the tie points necessary for linear and angular closure calculations).

Note: The bearing from the first instrument point to the first back sight point, and the bearing from the last (or tie) instrument point to the last (or tie) foresight point will be treated as reference bearings (held fixed). These four points will not be adjusted. If there are any reference bearings in the raw data file, the angular closure and adjustments will be from one reference bearing to the next, just as in Loop Traverses.

Since you may have many foresights from the instrument tie point (side shots), you will be asked to enter which foresight point you will be tying into (unless there are no side shots at the last instrument point).

The traverse will begin by the coordinates found in the coordinate file for the first instrument point and backsight point (coordinate values can be placed directly into the raw data file). The traverse will then be calculated. When the traverse is finished, the coordinates for the last instrument point and foresight point in the raw data file will be read from the coordinate file (or raw data file) in order to calculate the angular, vertical and horizontal closure.

If Elevations are ON you will be shown the elevation control found in the Raw Data and Coordinate files that pertains to your traverse. If no elevation control is found none will be shown and you will have to ADD control. Your elevation control can be anywhere in the traverse. It does not have to be on the first point.

You will have the following option at the command line:

Point Elevation

1 500.00

[Add/Change/Delete/Go/aBort]: <G>g

Select Add to add points to elevation control: A

Select Change to change the elevation assigned to a point in the elevation control: C

Select Delete to remove a point from the elevation control: D

Select Go to calculate elevations: G

Select aBort to quit without calculating elevations B

Select the appropriate option and the elevations will be calculated based upon the supplied information.

At this point you will get two closure reports:

The first report is before angle adjustment:

***** Closure Report *****

Total angular error: -0°00'06"

Angular error per point: -0°00'01"

Correct Ending Coordinates, North: 5000.00000 East: 5000.00000

Ending Coordinates, North: 5000.04008 East: 5000.00421

Error, N: 0.04 E: 0.00 Total: 0.04 Brg: S 05°59'43"W

Distance Traversed: 2470.51 Closure: 61308

The Second Report is after angle adjustment:

***** Closure Report *****

Total angular error: 0°00'00"

Angular error per point: 0°00'00"
 Correct Ending Coordinates, North: 5000.00000 East: 5000.00000
 Ending Coordinates, North: 5000.04314 East: 5000.01593
 Error, N: 0.04 E: 0.02 Total: 0.05 Brg: S 20°16'08"W
 Distance Traversed: 2470.51 Closure: 53721

Following the angular adjustment the reduced traverse will be displayed:

Adjusted by Least Squares
 Bearing Distance Northing Easting Elevation Pt ID Code Description
 5000.00000 5000.00000 500.00 1 1 TP1 2
 N 00°00'00"E 242.12 5242.12397 5000.00000 496.39 2 1 tpns
 N 74°41'24"E 199.78 5294.87495 5192.69243 467.97 3 1 tpns
 N 00°22'42"W 148.48 5443.34679 5191.71202 460.90 4 1 tpns
 N 04°35'35"W 310.32 5752.67444 5166.86125 458.07 5 1 tpns
 S 83°11'32"W 300.98 5716.99780 4868.00744 473.72 6 1 tpns
 S 84°09'21"W 290.03 5687.46658 4579.48877 472.10 7 1 tp hole
 S 13°25'02"E 137.70 5553.52582 4611.44085 484.33 8 1 tpns
 S 05°29'41"E 234.70 5319.90709 4633.91387 501.54 9 1 tpns
 S 12°52'27"E 308.42 5019.23837 4702.63376 517.34 10 1 tpns
 S 86°17'54"E 297.99 5000.00000 5000.00000 500.00 1 1 TP1 NAIL

Sq. Feet: 341547 Acres: 7.8

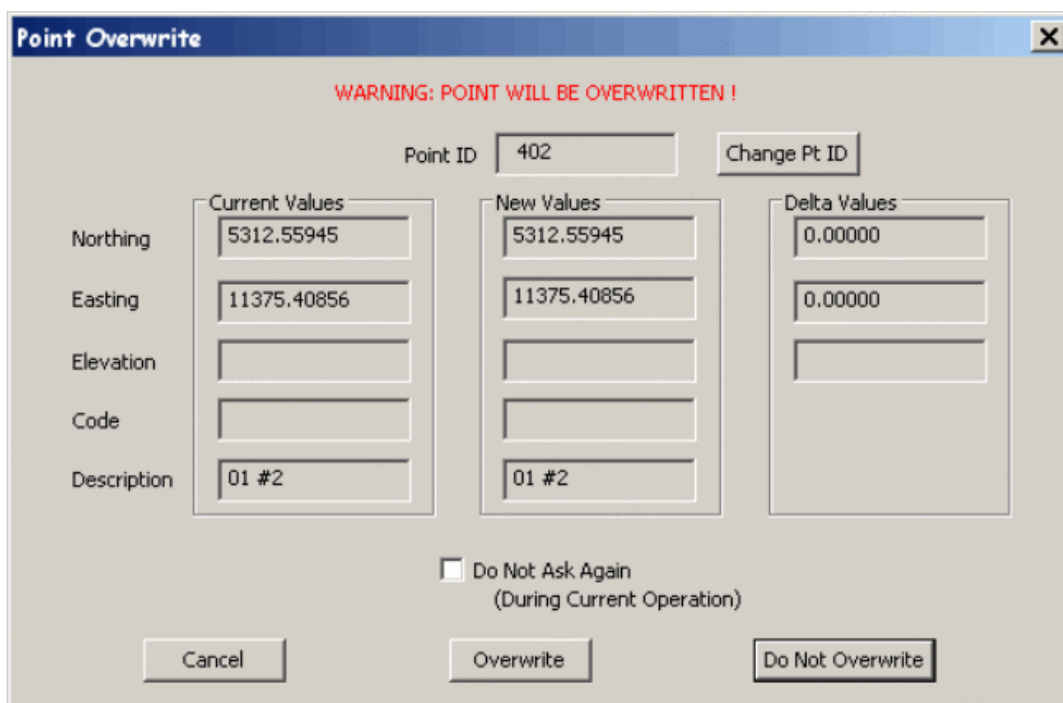
Once the traverse is reduced the side shots will be computed and displayed:

Side Shots
 Angle Distance Northing Easting Elevation Pt ID Code Description
 Inst.Pt.: 1 Bs.Pt.: 10
 148°15'53" 123.43 5058.01266 5108.95161 489.96 47 3 ipf1otp
 97°53'24" 46.81 5045.85154 5009.40500 499.25 48 2 ip4rb
 17°33'40" 96.60 5035.03240 4909.97367 506.27 49 2 ipf4rb
 Inst.Pt.: 2 Bs.Pt.: 1
 255°33'17" 93.22 5265.37939 5090.27763 480.73 25 4 ipf1\2" ctp
 146°29'54" 17.38 5256.61928 4990.40516 500.49 26 4 ipf1\2" ctp
 Inst.Pt.: 3 Bs.Pt.: 2
 297°01'47" 18.33 5276.92239 5188.96820 468.73 27 4 ipf1#ctp
 Inst.Pt.: 4 Bs.Pt.: 3
 10°21'19" 65.64 5378.69600 5180.33917 466.55 28 4 ipf1ctp
 159°23'20" 63.27 5502.41856 5169.04898 461.70 29 3 ipf1otp
 113°52'33" 138.30 5498.48975 5064.87673 483.03 30 4 ipf1\2" ctp
 113°47'52" 186.84 5517.60975 5020.26008 489.30 31 9 fly
 291°56'23" 100.21 5406.52118 5284.90634 455.81 32 9 fly
 299°04'02" 111.18 5389.97593 5289.24079 455.88 33 4 ipf1ctp
 Inst.Pt.: 5 Bs.Pt.: 4

 39°33'59" 47.28 5713.93615 5139.76338 458.30 34 4 ipf1ctp
 260°33'36" 119.08 5781.54910 5282.38627 464.12 35 2 ipf4rb
 72°51'12" 136.19 5702.23225 5040.36168 469.98 36 4 ipf1\2ctp
 Inst.Pt.: 8 Bs.Pt.: 7
 32°47'04" 103.73 5651.38227 4645.83837 475.70 37 9 nf
 150°46'50" 209.58 5399.34540 4753.39990 512.22 38 9 fly
 104°48'11" 144.87 5550.02257 4756.26507 497.59 39 9 fly
 Inst.Pt.: 10 Bs.Pt.: 9

156°21'56" 66.78 4965.56171 4742.36495 517.21 46 9 stk
***** Elevation Calculations - Elevations Adjusted *****
Elevations from Points: 1 -> 1
Vertical Err: -0.01, Distance Traversed: 2470.51

The calculate points will be stored in the coordinate file. There is an overwrite protection built into the software. If a point already exists in the coordinate file you will have the following options:



CANCEL: will terminate the process of storing coordinates.

OVERWRITE : will overwrite the existing point.

DO NO OVERWRITE: skip to the next point. If you have the "*Do Not Ask Again*" box checked, OVERWRITE will overwrite all points without asking,

DO NOT OVERWRITE: will only write NEW points to the coordinate file.

Traverse Reduction: Open Traverse/Side Shots

When reducing these types of traverses, no adjustments are possible. The coordinates for instrument points and back sight points will be pulled from the coordinate file (or raw data file) and used to calculate and store the foresights. This option allows you to occupy newly created points.

Coordinates of back sight points will be calculated only if a distance has been entered to the back sight point and the back sight point does not exist in the coordinate file.

If you are back sighting a point that does not exist in the coordinate file and the raw data file does not contain a reference bearing or azimuth to the back sight point, you will be given the choice of entering one of the following:

Coordinates of the back sight point

Bearing from the instrument point to the back sight point

If you choose to enter the bearing and there is no distance to the back sight point in the raw data file (thus making it impossible to calculate its coordinates), and you later occupy that point, you will be asked to enter the

real coordinates of the point.

If you are backsighting a point that does exist, and you have a distance measurement to the backsight point in the raw data file, we will show a warning if the inversed distance from the coordinate file does not match the measured distance within the tolerances set in the CGTools->Global Options->Traverse Options dialog.

A table will be printed containing the following:

Side Shots

Angle Distance Northing Easting Elevation Pt ID Code Description

Inst.Pt.: 1 Bs.Pt.: 10

148°15'53" 123.43 5058.01266 5108.95161 489.96 47 3 ipf1otp

97°53'24" 46.81 5045.85154 5009.40500 499.25 48 2 ip4rb

17°33'40" 96.60 5035.03240 4909.97367 506.27 49 2 ipf4rb

Angle Adjustments

If you have set Adjust Angles in the Traverse Options dialog box, all angles will receive equal adjustment. If there is more than one reference bearing, the angles will be adjusted equally between reference bearings. You will be shown the closures before and after the angle adjustment.

NOTE: If you are going to use the Least Squares Adjustment, you should not adjust the angles. Angular adjustment is part of the Least Squares Adjustment process.

Elevation Adjustment

If you have set Adjust Elevations in the Traverse Options dialog box, the elevations will be adjusted in proportion to the lengths of the lines (the longer the line, the more the adjustment).

Least Squares, Crandall's and Compass Rule

If you select any of these adjustment options the coordinates will be adjusted with the appropriate method.

Find Bad Angle

If you have a bad angular closure, select Find Bad Angle in the Traverse Options dialog box instead of an adjustment type. This function will not create or store any coordinate points.

NOTE: This option cannot be used with Azimuth Traverses.

You will see the following report:

Total angular error: 0°00'07"

Angular error per point: 0°00'01"

Correct Ending Coordinates, North: 10000.00000 East: 10000.00000

Ending Coordinates, North: 10000.05876 East: 9999.95840

Error, N: 0.05876 E: -0.04160 Total: 0.07200 Brg: S 35°17'49"E

Distance Traversed: 1492.10800 Closure: 20725

Instrument point: 1, Error: 0.07200, Closure: 20725

Instrument point: 2, Error: 0.08249, Closure: 18089

Instrument point: 3, Error: 0.08284, Closure: 18013

Instrument point: 4, Error: 0.07542, Closure: 19785

Instrument point: 5, Error: 0.06751, Closure: 22103

Worst Closure: 18013

Average Closure: 19620

Possible bad angle at instrument point: 5, Closure: 22103

In the above example, there were 5 traverse points. The traverse is reduced five times, beginning at each traverse point. The starting instrument point that produces the best closure is shown as having the bad angle. All closures are shown.

OTHER METHODS OF TRAVERSING

Every surveyor has his own unique methods when it comes to traversing. This section describes and shows examples of four additional entry methods.

Notice in the sample traverses there is a distance and vertical angle recorded for each foresight and back sight. This is optional, but you need at least one distance to each foresight.

Where both foresight and back sight distances are recorded, distances will be averaged when reduced

Side shots may be entered along with traverse information. You may turn more than one angle to side shots if you wish.

A description and/or code only needs to be entered once for a given foresight point.

Single Position with Direct and Reverse Angles

Perform this method as follows:

Shoot the back sight.

Turn to a foresight.

Record the angle and distance.

Plunge the instrument.

Take another reading (reversed) to the foresight. You may do this to traverse points and side shots.

Turn back to the back sight with the instrument reversed.

Record another angle to the back sight.

The final angle in each set for each instrument point must be a reverse reading to the back sight.

The angle in the instrument for the first back sight will be subtracted from the first angle to each foresight. The final (reverse) angle to the back sight will be subtracted from the second angle to each foresight. The two resulting angles will then be averaged to give you an angle to the foresight. All distances recorded will be averaged.

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Horz.Angle	Slope Dist.	Vert.Angle	Foresight	Code	Description
IP	1	5.32000	5	5.00000	0.00000	290.54000	88.35000			
FS				5.00000	109.19170	292.31000	88.35000	2		H&T
FS				5.00000	289.19300	292.31000	271.25100	2		
FS				5.00000	190.32100	52.39000	90.32550	6		IPF
FS				5.00000	10.32120	52.39000	269.27080	6		
FS				5.00000	180.00020	290.54000	271.25100	5		
FS				5.00000			90.00000	6		
IP	2	5.20000	1	5.00000	0.00000	292.31000	91.31300			
FS				5.00000	96.03560	324.26000	88.51300	3		IPF
FS				5.00000	276.03580	324.26000	271.08300	3		
FS				5.00000	180.00100	292.31000	268.28300	1		
IP	3	4.98000	2	5.00000	0.00000	324.26000	91.10000			
FS				5.00000	124.03560	275.84000	92.22400	4		IPF
FS				5.00000	304.03400	275.84000	267.37100	4		
FS				5.00000	179.59500	324.26000	268.50020	2		
IP	4	5.12000	3	5.00000	0.00000	275.84000	87.37000			
FS				5.00000	93.02130	309.68000	89.13000	5		IPF
FS				5.00000	273.02200	309.68000	270.47100	5		
FS				5.00000	180.00100	275.84000	272.23000	3		
IP	5	5.15000	4	5.00000	0.00000	309.68000	90.50000			
FS				5.00000	117.30500	290.55000	91.24300	1		IPF
FS				5.00000	297.30400	290.55000	268.35000	1		
FS				5.00000	180.00050	309.67000	269.10050	4		

Single Positions with Multiple Direct and Reverse Angles

Entering multiple sets of direct and reverse angles is very much like the preceding example where 1 direct and reverse set was entered. The only thing to remember is that each direct and reverse pair is a set. When another set is entered, it begins with a back sight direct angle (recorded like a foresight), has direct angles and reverse angles to the foresights, and ends with a reverse angle to the back sight. Do not begin a new instrument point for the second set, merely record a new back sight angle and continue with the procedure through each foresight, and end with another reverse angle to your back sight.

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Horz.Angle	Slope Dist.	Vert.Angle	Foresight	Code	Description
IP	1	5.32000	5	5.00000	0.00000	290.54000	88.35000			
FS				5.00000	109.19170	292.31000	88.35000	2		H&T
FS				5.00000	289.19300	292.31000	271.25100	2		
FS				5.00000	180.00200	290.54000	271.25100	5		
FS				5.00000	0.00000	290.54000	88.35020	5		
FS				5.00000	109.19150	292.31000	88.35010	2		H&T
FS				5.00000	289.19280	292.31000	271.25080	2		
FS				5.00000	180.00150	290.54000	271.25050	5		
IP	2	5.20000	1	5.00000	0.00000	292.31000	91.31300			
FS				5.00000	190.32100	52.39000	90.32550	6		IPF
FS				5.00000	10.32120	52.39000	269.27080	6		
FS				5.00000	96.03560	324.26000	88.51320	3		IPF
FS				5.00000	276.03560	324.26000	271.08300	3		
FS				5.00000	180.00100	292.31000	268.28320	1		
FS				5.00000	0.00000	292.31000	91.31280	1		
FS				5.00000	96.03580	324.26000	88.51300	3		IPF
FS				5.00000	276.03550	324.26000	271.08310	3		
FS				5.00000	180.00080	292.31000	268.28330	1		
IP	3	4.98000	2	5.00000	0.00000	324.26000	91.10000			
FS				5.00000	124.03560	275.84000	92.22400	4		IPF
FS				5.00000	304.03400	275.84000	267.37100	4		
FS				5.00000	179.59500	324.26000	268.50020	2		
FS				5.00000	0.00000	324.26000	91.10002	2		
FS				5.00000	124.03580	275.84000	92.22410	4		IPF
FS				5.00000	304.03410	275.84000	267.52250	4		
FS				5.00000	179.59550	324.26000	268.50050	2		
IP	4	5.12000	3	5.00000	0.00000	275.84000	87.37000			
FS				5.00000	93.02130	309.68000	89.13000	5		IPF
FS				5.00000	273.02201	309.68000	270.47080	5		
FS				5.00000	180.00050	275.84000	272.23000	3		
FS				5.00000	0.00000	275.84000	87.37020	3		
FS				5.00000	93.02150	309.68000	270.47080	5		IPF
FS				5.00000	273.02210	309.68000	270.47080	5		
FS				5.00000	180.00050	274.84000	272.23010	3		
IP	5	5.15000	4	5.00000	0.00000	309.68000	90.50000			
FS				5.00000	117.30500	290.55000	91.24300	1		IPF
FS				5.00000	297.30400	290.55000	268.35000	1		
FS				5.00000	180.00050	309.67000	269.10050	4		
FS				5.00000	0.00000	309.68000	90.50020	4		
FS				5.00000	117.30520	290.55000	91.24320	1		IPF
FS				5.00000	297.30410	290.55000	268.34580	1		
FS				5.00000	180.00030	309.67000	269.10040	4		

Azimuth Traverses

Azimuths are entered into a file with the azimuth to each foresight entered in the Foresight data entry line at the azimuth column.

NOTE: If you are running a Closed Loop Traverse, a reference azimuth must be placed at the last instrument point if you wish to adjust the angular error.

The reference azimuth is the correct azimuth from the last instrument point in the raw data file to the first instrument point (or last foresight).

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Azimuth	^ Slope Dist.	^ Vert.Angle	Foresight	Code	Description
IP	1	5.32000	5	5.00000	0.00000	290.54000	88.35000			
FS				5.00000	173.56140	292.31000	88.35000	2		H&T
IP	2	5.20000	1	5.00000	0.00000	292.31000	91.31300			
FS				5.00000	184.27240	52.39000	90.32550	6		IPF
FS				5.00000	90.00100	324.26000	88.51320	3		IPF
IP	3	4.98000	2	5.00000	0.00000	324.04060	89.13000			
FS				5.00000	34.04060	275.84000	92.22400	4		IPF
IP	4	5.12000	3	5.00000	0.00000	275.84000	87.37000			
FS				5.00000	307.06190	309.68000	89.13000	5		IPF
IP	5		4	5.00000	0.00000	309.68000	90.50000			
FS				5.00000	244.37260	290.55000	91.24300	1		IPF

Traverse with Doubled Angles

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Horz.Angle	^ Slope Dist.	^ Vert.Angle	Foresight	Code	Description
IP	1	5.32000	5	5.00000	0.00000	290.54000	88.35000			
FS				5.00000	109.19170	292.31000	88.35000	2		H&T
FS				5.00000	109.19170	290.54000	271.25100	5		
FS				5.00000	218.38300	292.31000	271.25100	2		
IP	2	5.20000	1	5.00000	0.00000	292.31000	91.31300			
FS				5.00000	190.32100	52.39000	90.32550	6		IPF
FS				5.00000	190.32100	292.31000	91.31300	1		
FS				5.00000	21.04200	52.39000	90.32550	6		
FS				5.00000	0.00000	292.31000	91.31300	1		
FS				5.00000	96.03560	324.26000	88.51320	3		IPF
FS				5.00000	96.03560	292.31000	268.28320	1		
FS				5.00000	192.07500	324.26000	271.08300	3		
IP	3	4.98000	2	5.00000	0.00000	324.26000	91.10000			
FS				5.00000	124.03560	275.84000	92.22400	4		IPF
FS				5.00000	124.03560	324.26000	268.50020	2		
FS				5.00000	248.07550	275.84000	267.37100	4		
IP	4	5.12000	3	5.00000	0.00000	275.84000	87.37000			
FS				5.00000	93.02130	309.68000	89.13000	5		IPF
FS				5.00000	93.02130	275.84000	272.23000	3		
FS				5.00000	186.04300	309.68000	270.47100	5		
IP	5	5.15000	4	5.00000	0.00000	309.68000	90.50000			
FS				5.00000	117.30500	290.55000	91.24300	1		IPF
FS				5.00000	117.30500	309.67000	269.10050	4		
FS				5.00000	235.01420	290.55000	268.35000	1		

Each new instrument setup requires a 0 to the back sight. The first angle to the foresight is the single angle. This angle is locked into the gun and the back sight is retaken. The second angle to the foresight is the doubled angle. You can double angles to side shots.

Loop Traverse Beginning and Ending on External Reference Azimuths

This type of traverse occurs frequently. The example below shows a Loop Traverse that begins on an external reference azimuth and ends on an external reference azimuth. Even though this traverse closes on itself, it must be reduced as a Closed Traverse Beginning and Ending at Known Points.

Point 100 is a dummy point on the azimuth line. Line 3 shows a reference bearing from point 1 to 100 (negative means from ip to bs) of S00-00-00E.

Line 16 shows the same reference bearing.

Point number 100 need not exist in the coordinate file and will not be calculated, but a dummy backsight and foresight point number must be entered into the raw data file.

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Horz.Angle	^ Slope Dist.	^ Vert.Angle	Foresight	Code	Description
CT										
IP	1	5.32000	1000		0.00000					
DR	1	100	200.00000							
FS				5.00000	125.20200	292.31000	88.35000	2		H&T
IP	2	5.20000	1	5.00000						
FS				5.00000	96.03560	324.26000	88.51320	3		IPF
FS				5.00000	190.32100	52.39000	90.32550	6		IPF
IP	3	4.98000	2		0.00000					
FS				5.00000	124.03560	275.84000	92.22400	4		IPF
IP	4	5.12000	3		0.00000					
FS				5.00000	93.02130	309.68000	89.13000	5		IPF
IP	5	5.15000	4		0.00000					
FS				5.00000	117.30500	290.55000	91.24300	1		IPF
IP	1	5.15000	5		0.00000					
DR	1	100	200.00000							
FS				125.2030				100		IPF

Use Of Reference Bearings and Azimuths

Reference Bearings and Azimuths are entered by Adding or Inserting a Reference Bearing data entry line. For example:

DR 1-2 123.4523

The direction from point 1 to point 2 is N23-45-23E.

Reference bearings and azimuths are optional (except for Closed Loop Azimuth Traverses). If a reference bearing is used, that direction will be held during the reduction process. More than one reference bearing may be used. The data below shows a raw data file using multiple reference bearings:

The previous data represents a loop traverse. If you choose to adjust angles, all angles will be adjusted from one reference bearing to the next (angles 1-5, 6-1). Angular closure information will also be shown from one reference bearing to the next. See the Reduction section of this chapter for more specific information on the use of reference bearings with different types of traverses.

Except for an initial reference bearing to the back sight point, reference bearings will be ignored for Open Traverses (no adjustments are available).

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Horz.Angle	^ Slope Dist.	^ Vert.Angle	Foresight	Code	Description
IP	1		11		0.00000		90.00000			
DR	1	11	242.35410							
FS					352.27290	1235.8400	90.00320	2		
IP	2		1		0.00000					
FS					32.21140	144.79000	89.59300	3		
IP	3		2		0.00000					
FS					195.03040	111.85000	89.58320	4		
IP	4		3		0.00000					
FS					116.22250	125.89000	90.15200	5		
IP	5		4		0.00000					
FS					190.15590	93.75000	90.23560	6		
DR	5	6	136.15590							
IP	6		5		0.00000					
FS					143.21360	75.20000	88.15260	7		
IP	7		6		0.00000					
FS					90.00200	111.85000	91.23560	8		
IP	8		7		0.00000					
FS					164.56360	144.77000	89.12320	9		
IP	9		8		0.00000					
FS					213.37500	117.95000	90.11000	10		
IP	10		9		0.00000					
FS					85.20400	116.20000	88.21100	11		
IP	11		10		0.00000					
FS					36.12470	1126.1300	90.00100	1		

Multiple Traverse Codes in a Single File

This sample is of a raw data file that contains multiple traverse codes in a single file: ET end main loop traverse
Scale factors are placed after Instrument Point data entry lines. Any text following a LT, CT, OT or ET marker is

used for comments. Notice that the codes MUST precede the first instrument setup that begins the traverse. The Foresight Tie Point in the previous example is necessary because there is a side shot (point #25) at the end of the Closed Traverse. The reduction routine does not know whether you are tying into point 25 or point 2

Point	North	East	Elev	Code	Description				
1	1200.0000	1000.0000	921.5000		control point				
2	1200.2500	1200.1000	931.0000		approx corner				
3	1122.7000	779.0500	930.0000		approx corner				
LT beginning of main loop Traverse									
IP	1	5.32000	5	5.00000	0.00000	270.54000	88.35000		
S	0.9999975								
BR	1	2	149.23560						
FS			5.00000	189.19170	292.31000	88.35000	2		H&T
FS			5.00000	209.19300	292.31000	271.25100	2		
FS			5.00000	186.00200	270.54000	271.25100	5		
IP	2	5.20000	1	5.00000	0.00000	292.31000	92.31300		
FS			5.00000	190.32100	52.39000	90.32550	6		IPF
FS			5.00000	10.32120	52.39000	269.27800	6		
FS			5.00000	96.03560	324.26000	88.51320	3		IPF
FS			5.00000	276.03500	324.26000	271.00300	3		
FS			5.00000	180.00100	292.31000	268.00000	1		
IP	3	4.90000	2	5.00000	0.00000	324.26000	91.10000		
FS			5.00000	124.03560	275.84000	92.27400	4		IPF
FS			5.00000	304.03400	275.84000	267.37100	4		
FS			5.00000	179.99500	324.26000	268.90020	2		
OT beginning of open loop traverse									
IP	3	5.03000	2	0.00000					
FS			5.00000	185.23560	135.26000	95.23150	20		Nail
IP	20	5.12000	3	0.00000					
FS			5.00000	180.15260	116.45000	85.23150	21		Nail
IP	21	5.11000	20	0.00000					
FS			5.00000	183.26450	50.13000	91.23450	22		Nail
ET end of open traverse									
IP	4	5.12000	3	5.00000	0.00000	275.84000	87.30000		
FS			5.00000	93.02130	309.60000	89.13000	5		IPF
FS			5.00000	273.02100	309.60000	270.47100	5		
FS			5.00000	180.00100	275.84000	272.29000	3		
CT beginning of closed traverse that ties into main loop									
IP	5	5.00000	4	0.00000					
FS			5.00000	39.26450	130.01000	89.00001	10		Nail
FS			5.00000	219.26400	130.00000	271.00000	10		
FS			5.00000	180.00050			4		
IP	10	5.00000	5	0.00000					
FS			5.00000	241.56350	126.01000	89.30000	11		Nail
FS			5.00000	61.56300	126.02000	270.30100	11		
FS			5.00000	180.00070			5		
IP	11	5.00000	10	0.00000					
FS			5.00000	114.56340	129.42300	89.50000	12		Nail
FS			5.00000	294.56300	129.43000	270.10030	12		
FS			5.00000	180.00020			10		
IP	12	5.00000	11	0.00000					
FS			5.00000	140.26310	144.70000	88.40000	3		
FS			5.00000	320.26200	144.60000	271.20050	3		
FS			5.00000	180.00020			11		
IP	3	5.00000	12	0.00000					
FS			5.00000	232.34560	10.23000	89.34000	25		
FS			5.00000	325.54320			2		
FS			5.00000	145.54300			2		
FS			5.00000	180.00020			12		
ET end closed loop									
IP	5	5.15000	4	5.00000	0.00000	309.60000	90.50000		
FS			5.00000	117.30500	290.55000	91.30300	1		IPF
FS			5.00000	297.30400	290.55000	268.29300	1		
FS			5.00000	180.00050	309.67000	269.10050	4		

Pull Down Menu Location:CGTrav\ Reduce Traverse
Keyboard Command:RT, CG_REDUCE_RAW
Prerequisite:Open Raw file *.CGR

Edit Map Check File

The map check program is used to enter or edit deed and map information for checking closures and to assist with evaluating data from other sources for a job you are working on.

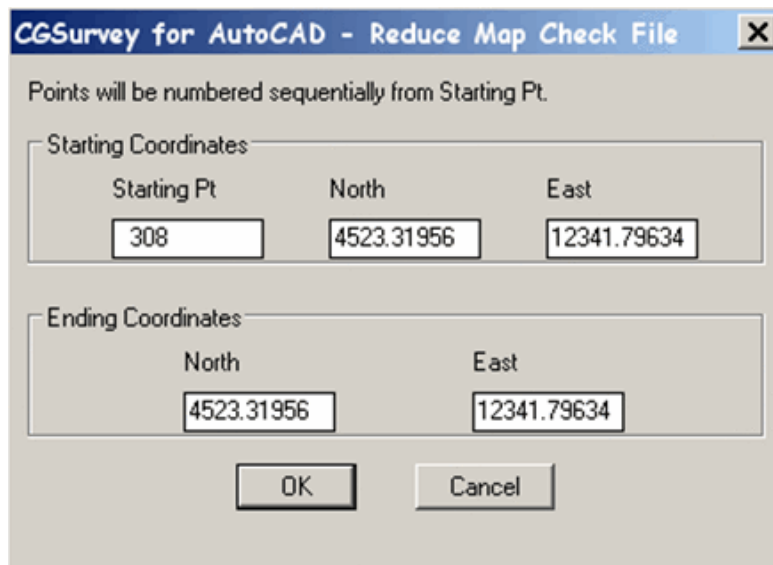
Note: for further and complete information on using the Mapcheck editor, see the chapter on CGEditor in the Tools section.

Pulldown Menu Location: CGTrav/Edit Mapcheck File
Keyboard Command: EM, CG_EDIT_MAP
Prerequisite: None

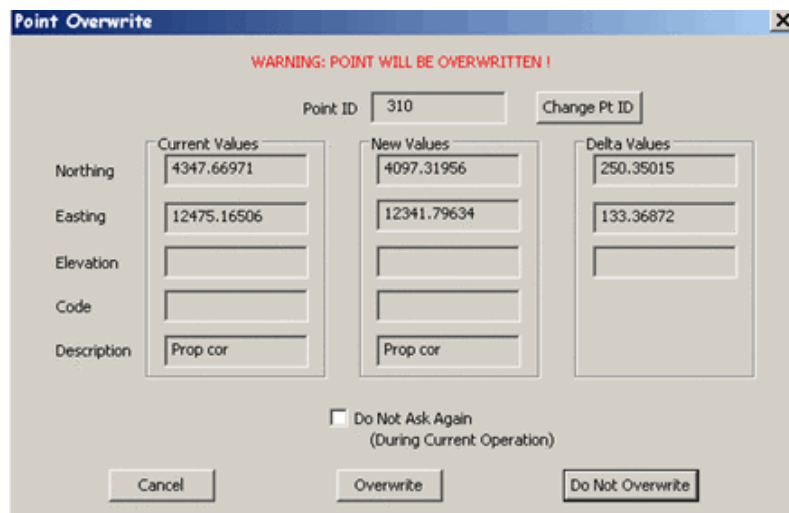
Reduce Map Check File

If a map check file is not open, a file dialogue box will appear, allowing you to open an existing map check file. If you wish the coordinates to be adjusted, select the type of adjustment in the Traverse Options dialog box. If a coordinate file is not open, a file dialog box will appear allowing you to open one. NOTE You may use the same coordinate file as often as you wish. Make sure the correct coordinate file is open.

Next Enter Point values: the starting Point number, Northing and Easting and the ending Northing and Easting:



The map data will then be reduced and the coordinates stored in the coordinate file. Overwrite protection is in place in case the points already exist in the coordinate file. If a point already exists in the coordinate file you will have the following options:



CANCEL: will terminate the process of storing coordinates.

OVERWRITE :will overwrite the existing point.

DO NO OVERWRITE: skip to the next point. If you have the "Do Not Ask Again" box checked.

Overwrite will overwrite all points without asking, and Do Not Overwrite will only write NEW points to the coordinate file.

The initial closure information will be shown. For example:

Correct Ending Coordinates, North: 5000.0000 East: 5000.0000

Ending Coordinates, North: 5071.8346 East: 4894.7441

Error, N: 71.83 E: -105.26 Total: 127.43 Brg: S 55°41'15"E

Distance Traversed: 1308.19 Closure: 10

A full report including acreage may be viewed by pressing the F2 key to view the CAD Text Window. You may also view/print the display file.


```

-----
* Prepared by: C&G Software
* Routine: Reduce Map Check File  Coord File: ddm 1.crd   8/03/07 15:53:44
* Input Scale Factor: 1.000000  Output Scale Factor: 1.000000
-----

Correct Ending Coordinates, North: 5000.00000 East: 10000.00000
Ending Coordinates, North: 4944.24098 East: 9623.70313
Error, N: -55.75902 E: -376.29687 Total: 380.40558 Brg: N 81°34'17"E
Distance Traversed: 1388.19140 Closure: 4

No Adjustment
Bearing      Distance      Northing      Easting      Elevation Pt ID

S 00°00'00"E  213.00000    5000.00000    10000.00000    319
N 74°21'18"E  138.50000    4787.00000    10000.00000    320
N 74°21'18"E  138.50000    4824.35015    10133.36872    321
N 16°42'08"W  99.95000    4920.08340    10104.64332    322
N 09°02'52"E  85.44000    5004.46032    10118.07945    323
N 49°17'58"W  289.65000    5193.34275    9898.48767    324
N 71°00'44"W  43.76000    5207.58078    9857.10874    325

CURVE DEF: Arc          CURVE DIR: CCW
RAD: 50.00000  LEN: 0.00000  TAN: 0.00000  CEN. ANG: 0°00'00"
CHORD: 169.02140  HO: 0.00000  EXT: 0.00000  DEGREE: 0°00'00"
SEG: 0.00000     TRI: 0.00000  SEC: 0.00000

S 18°59'16"W  50.00000    5160.30138    9840.84042    326 PC
->RP
N 54°31'59"W  176.26183    5262.57468    9697.28413    327 RP
->PT

N 71°00'44"W  169.02140    5262.57468    9697.28413    327 PC
->PT
S 38°57'18"W  135.85000    5156.93228    9611.87390    328
S 03°11'00"E  213.02000    4944.24098    9623.70313    329

Approx: Sq. Feet: 118144.88983 Acres: 2.71223

```

Pulldown menu Location:CGTrav
Keyboard Command:RDM, CG_REDUCE_MAP
Prerequisite: Open Coordinate File

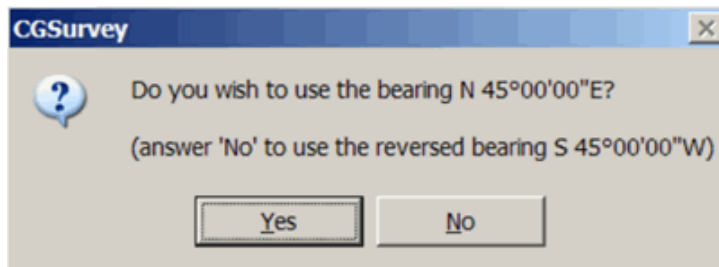
Visual Map Check

This routine allows you to graphically pick the Call Text (Bearings and Distance) from a drawing and perform a Map Check Closure.

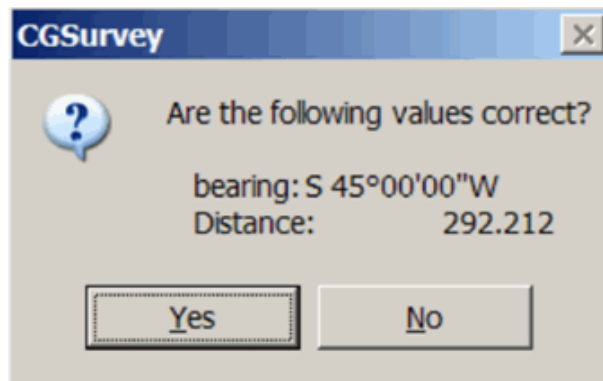
Prompts

First you will be asked: Pick Point of Beginning: You can enter the beginning point number, or graphically pick the point on the screen.

Next: Pick Bearing Text for Leg 1 (ask Reverse is ON) [Off/Done]<Done>: Graphically pick the text with the Bearing. If "ask Reverse" is turned ON, you will be allowed to reverse the direction of the bearing after it is selected:

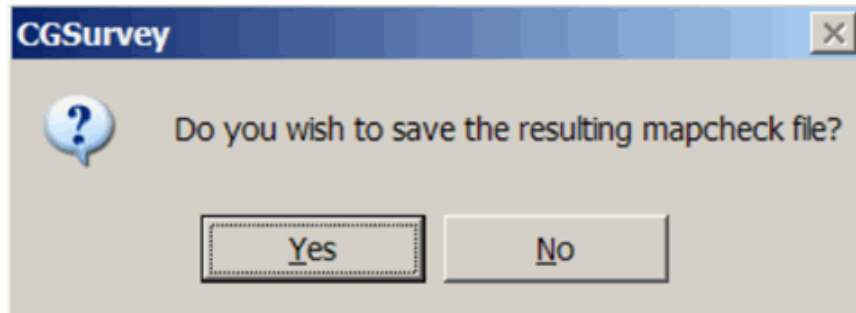


Next: Pick Distance Text for Leg 1: Graphically pick the text with the distance. You will see:

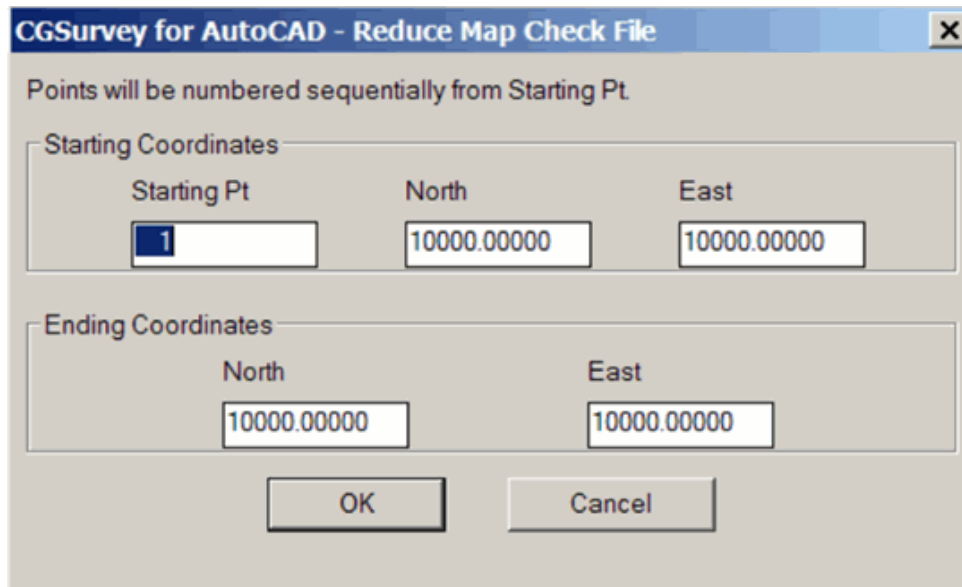


If you select YES, you will go to the next leg. If you select NO: you will be asked to pick the Bearing and distance for Leg 1 again.

After selecting all the Calls: press ENTER for DONE. You will have the option:

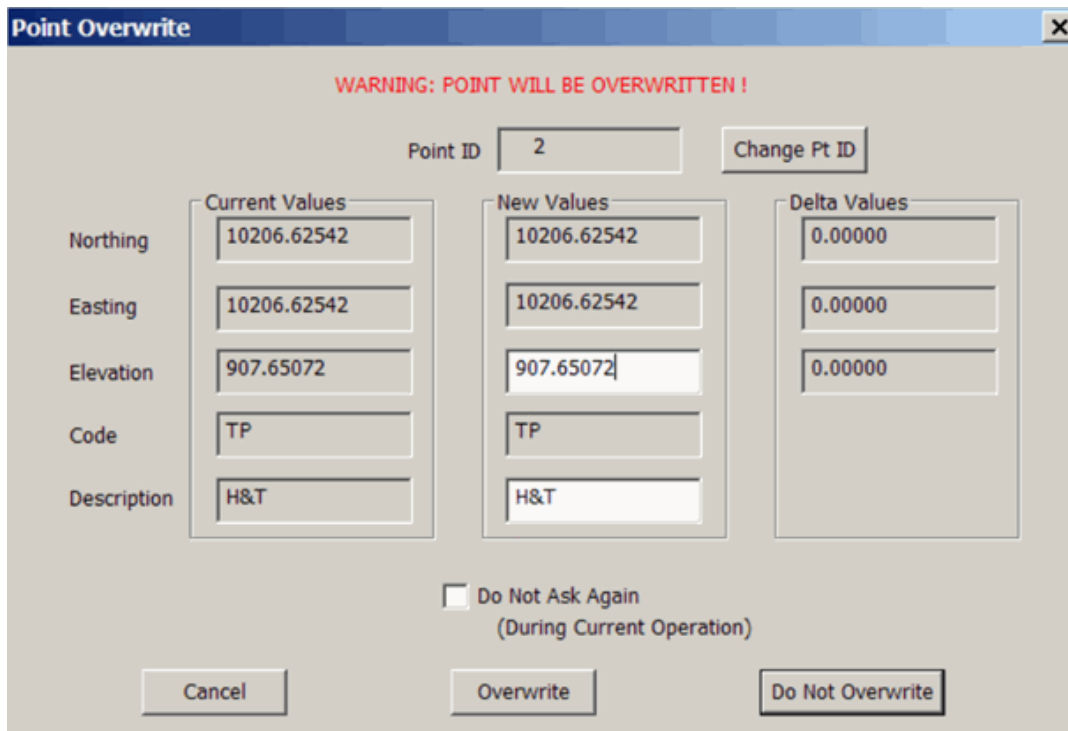


If you select YES, the information: you selected will be placed in a Map Check File. You will be asked to select the CGM file.



Next: Enter the starting and ending coordinates for the traverse.

The map data will then be reduced and the coordinates stored in the coordinate file. Overwrite protection is in place in case the points already exist in the coordinate file. If a point already exists in the coordinate file you will have the following options:



CANCEL: will terminate the process of storing coordinates.

OVERWRITE: will overwrite the existing point.

DO NO OVERWRITE: skip to the next point. If you have the "Do Not Ask Again" box checked, OVERWRITE will overwrite all points without asking, **DO NOT OVERWRITE:** will only write NEW points to the coordinate file.

Below is a sample Report:

Correct Ending Coordinates, North: 10000.00000 East: 10000.00000

Ending Coordinates, North: 9586.74896 East: 9586.74832

Error, N: -413.25104 E: -413.25168 Total: 584.42568 Brg: N 45°00'00"E

Distance Traversed: 1492.10700 Closure: 3

Adjusted by Least Squares

Bearing Distance Northing Easting Elevation Point ID

10000.00000 10000.00000 900.00000 1

S 58°19'27"W 146.64772 9922.99352 9875.19793 2

N 05°19'46"W 299.65818 10221.35627 9847.36450 3

N 73°17'06"W 156.24457 10266.29428 9697.72179 4

S 04°35'43"E 226.90862 10040.11507 9715.90113 5

S 64°19'20"E 371.14929 9879.29253 10050.39763 900.00000 1

Sq. Feet: 814183.13568 Acres: 18.69107

Pulldown menu Location: CGTrav

Keyboard Command: VM, cg_visual_mapcheck

Prerequisite: Call Text must be displayed to select

Create StarNet File

This option converts a raw data file to the Star*Net (.DAT) format. The raw data file will be preprocessed. During conversion, multiple distances and angles will be averaged and compared to the maximum ranges set in the Traverse Options dialog box. To use this option properly, you must know how Star*Net works. You should be familiar with all Star*Net codes and commands.

NOTE: This manual is not a substitute for the Star*Net manual.

Below is a sample raw data file that contains three different traverse types. This raw data file can be reduced using CG-SURVEY or written to a Star*Net file for reduction with Star*Net, both without any editing.

```

C 1 1000 1000 923.56!!! 'CM CONCRETE MONUMENT
C 2 1205.25 1208.13 931 #approx coordinates
C 5 1127.73 739.05 930 #approx coordinates
LT beginning of main loop traverse
1 5.32 5 5.00 0.00000 290.540 88.35000
#This line is a comment
S .9999978
DR 1-2 145.2356
      5.00 109.19170 292.310 88.35000 2 H&T
      5.00 289.19300 292.310 271.25100 2
      5.00 180.00200 290.540 271.25100 5
2 5.20 1 5.00 0.00000 292.310 91.31300
      5.00 190.32100 52.390 90.32550 6 IPF
      5.00 10.32120 52.390 269.27080 6
      5.00 96.03560 324.260 88.51320 3 IPF
      5.00 276.03580 324.260 271.08300 3
      5.00 180.00100 292.310 268.28320
3 4.98 2 5.00 0.00000 324.260 91.10000
      5.00 124.03560 275.840 92.22400 4 IPF
      5.00 304.03400 275.840 267.37100 4
      5.00 179.59500 324.260 268.50020 2
OT beginning of open traverse
3 5.03 2 0.00000
      5.00 185.23560 135.260 95.23150 20 Nail
20 5.12 3 0.00000
      5.00 180.15260 116.450 85.23150 21 Nail
2 5.11 20 0.00000
      5.00 183.26450 50.130 91.23450 22 IPF
ET end of open traverse
4 5.12 3 5.00 0.00000 275.840 87.38000
      5.00 93.02130 309.680 89.13000 5 IPF
      5.00 273.02200 309.680 270.47100 5
      5.00 180.00100 275.840 272.22000 3
CT Beginning of closed traverse which ties into main loop
FT 2 last tie angle is to point number 2 in main loop
5 5.00 4 0.00000
      5.00 39.26450 130.010 89.00010 10
      5.01 219.26400 130.000 271.00000 10 Nail
      5.00 180.00050 180.00050 4
10 5.00 5 0.00000
      5.00 241.56350 126.010 89.30000 11
      5.02 61.56300 126.020 270.30100 11 Nail
      5.00 56.23100 10.000 90.00000 15 IPF
      5.00 180.00070 180.00070
11 5.00 10 0.00000
      5.00 114.56340 129.423 89.50000 12 Nail
      4.99 294.56300 129.430 270.10030 12
      5.00 180.00100 180.00100 10
12 5.00 11 0.00000
      5.00 140.39310 144.700 88.40000 3
      5.02 320.39200 144.680 271.20050 3
      5.00 180.00020 180.00020 11
3 5.00 12 0.00000
      5.00 325.54320 325.54320 2
      5.01 145.54300 145.54300 2
      5.01 180.00020 180.00020 12

```

```

ET end closed traverse
5 5.15 4 5.00 0.00000 309.680 90.50000
5.00 117.30500 290.550 91.30300 1 IPF
5.00 297.30400 290.550 268.29300 1
5.00 180.00050 309.670 269.10050 4
ET end main loop traverse

```

Here is the Star Net file created using the example raw data file:

```

C 1 1000 1000 923.56 !!! 'CM, CONCRETE MONUMENT
C 2 1205.25 1208.13 931 #Elev is approx.
C 5 1127.73 739.05 930 #Elev is approx|
#BT beginning of main loop traverse
B 1-2 N45-23-56.0E
M 1-5-2 109-19-13 292.220 'TP_H&T
D 1-5 290.451 #TP_H&T
M 1-5-15 62-00-00 100.000 'TP_H&T
SS 1-5-50 52-12-35 101.778 'TP_test
D 2-1 292.207 #TP_test
M 2-1-3 96-03-52 324.196 'TP_IPF
SS 2-1-6 190-32-06 52.388 'SS_IPF
D 3-2 324.193 #SS_IPF
M 3-2-4 124-03-53 275.602 'TP_IPF
#BT beginning of open traverse
D 3-2 324.153 #TP_IPF
M 3-2-20 185-23-56 134.662 'TP_Nail
M 20-3-21 180-15-26 116.073 'TP_Nail
SS 21-20-22 183-26-45 50.115 'PL_IPF TRAVERSE
#ET end of open traverse
D 4-3 275.605 #PL_IPF
M 4-3-5 93-02-11 309.651 'TP_IPF
#BT Beginning of closed traverse which ties into main loop
M 5-4-10 39-26-40 129.985 'TP_Nail
M 10-5-11 241-56-29 126.010 'TP_Nail
M 10-5-15 56-23-10 10.000 'SS_IPF
M 11-10-12 114-56-27 129.426 'TP_Nail
M 12-11-3 140-39-24 144.651 'TP_Nail
A 3-12-2 325-54-30 #TP_H&T
.....
#ET end closed traverse
M 5-4-1 117-30-42 290.449 'TP_IPF
D 5-4 309.642 #TP_IPF
#ET end main loop traverse

```

The coordinate formats (C code) are the same for Star*Net and CG-SURVEY; no translation is necessary.

If a comment line in the raw data file uses a valid Star*Net code or command, it will be used in its original form (as with coordinates); not as a comment. These codes are #, C, A, D, V, B, M, TB, T, TE, SS (followed by a space) and all dot commands. (example: .CURVE, .SCALE, etc...)

Multiplication factors are converted to the .Scale command. The original multiplication factor set in the Global Options dialog box will be placed at the beginning of the Star*Net file. Other multiplication factors will be placed as they occur in the raw data file.

NOTE: You cannot use a multiplication factor for meter/feet conversion in a 3-D traverse in Star*Net 3.2.

Reference bearings/azimuths are converted to the B format.

All traverse points are converted to the M format and side shots to the SS format. Only points used once (as a foresight point) will be considered a side shot. If a point is located from more than one instrument setup, or is used as an instrument point or backsight point, the point will be converted to a M format

The LT, ET, OT, FT and CT codes are converted to comments. Point codes are combined with descriptions.

If Elevations are on, the Star*Net file must be adjusted as a 3-D traverse. The following could occur:

If you input slope distances and vertical angles, all distances will first be reduced to their horizontal/vertical components. Multiple distances will be averaged and then a slope distance and vertical angle will be recomputed from the averaged horizontal/ vertical components. This is done so Star*Net can compute corrections for curvature and refraction and vertical divergence (can only be done if vertical angles are used in a 3-D traverse.) A ".Delta Off" command will be placed in the Star*Net file.

If Curvature and Refraction is on in the Options/Global Options dialog box, a Curve command will be placed in the Star*Net file.

If you input horizontal and vertical distances, a ".Delta On" command will be placed in the Star*Net file. No corrections for curvature and refraction or vertical divergence will be possible.

If Elevations are off in the Options/Global Options dialog box, the Star*Net file must be reduced as a 2-D traverse. The following will occur:

If you input slope distances and vertical angles, all distances will be reduced to their horizontal/vertical components and the vertical components will be thrown away. Multiple distances will be averaged.

No corrections for curvature and refraction or vertical divergence are allowed in a 2-D traverse with Star*Net version 3.2 or earlier.

Pulldown Menu Location: CGTrav

Keyboard Command:STN, CG_REDUCE.STARNET

Prerequisite: Open CG Raw file *.CGR

CGCogo



The Command Line:

Throughout CGSurvey the user will be prompted at the command line for input. Typically the command line is at the bottom of the CAD graphic screen, although the command line can be placed above the graphics screen. To enter a command at the command line use the key board, and press the <Enter> key when finished.

The F2 hot key can be used at any time to access a full text window that displays user input and history. As point numbers are typed in, or selected on the screen using the mouse, the point number entered is displayed at the command line. When the next point ID is requested the previous point ID is used as the starting point for the command.

For example, when inverting from 5 to 7. First you type 5 at the command line and press <Enter>.

Command:

[Point group/Reset/sNap on] (last point = <none>): 5

[cLockwise curve/ccW curve/Point group/Reset/sNap on] (last point = 5):

Note that 5 is now displayed as the "Last Point". This means that when 7 is entered at the command line the inverse will be calculated from point 5 to point 7.

To clear the last point enter R or "." and <Enter> for Reset. The last point will now be shown as <none>.

Inverse

This command allows you to determine the bearing and distance between the endpoints of a line or a curve by entering the points that define the line or curve.

After choosing the Inverse menu item you are asked to "Enter point sequence". A point sequence is a series of points that define the points being used to calculate the inverse. You may enter one point ID at a time to inverse from point to point. You may enter the two points separated by a dash (5-7), etc.

Enter point sequence

[Point group/Reset/sNap on] (last point = <none>): 5

[cLockwise curve/ccW curve/Point group/Reset/sNap on] (last point = 5): 7

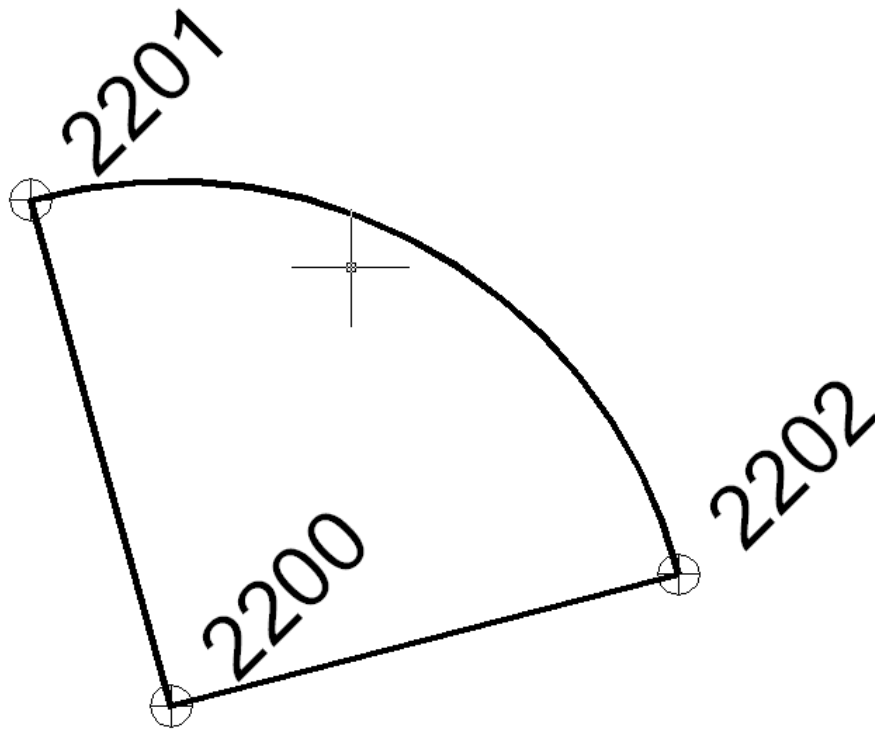
Note that 5 is now displayed as the "Last Point". This means that when 7 is entered at the command line the inverse will be calculated from point 5 to point 7.

To "Reset" the last point enter R or "." and <Enter> for Reset. The last point will now be shown as <none>.

Inversing Around Curves Clockwise

To inverse around a clockwise curve (one curving to the right), enter the PC point ID then enter "L" or "+" to indicate a clockwise curve. Next enter the point ID of the radius point of the clockwise curve and follow with the PT point ID.

For Example:



First enter the PC of the curve. In this example type or pick point ID 2201

Enter point sequence

[cg-Point-group/Reset/turn-Snap-on] (last point = <none>): 2201

Now type L or "+" and <Enter> for a clockwise curve

Enter point sequence

[clockWise curve/Ccw curve/Point group/Reset] (last point = 2201):L

Next type or pick the radius point ID 2200

Enter radius point for curve [Reset/sNap on]: 2200

You may also use the mouse to pick a C&G Arc. In this case, the arc's radius point will be used

Next type or pick the PT point ID 2202

Enter point of tangency (PT) for curve [Reset/sNap]: 2202

Inversing Around Curves Counter Clockwise

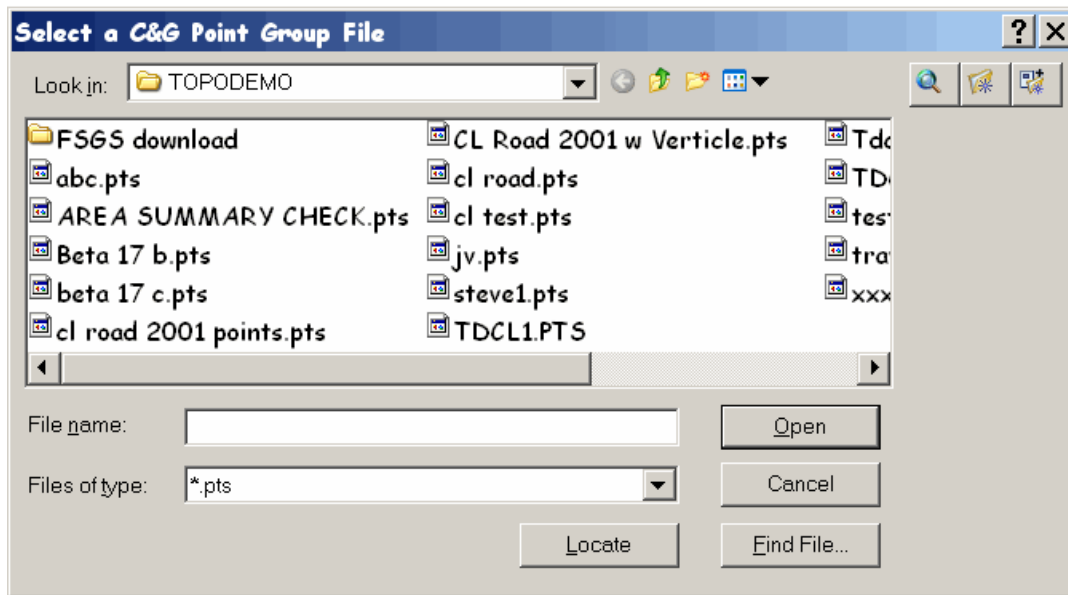
To inverse around a curve in a counter-clockwise direction (curving to the left) simply type W or "-" and <Enter> then proceed as with a clockwise curve.

Inversing between a series of points in the coordinate file

By entering 2 point numbers separated by a "+" you can inverse through successive point IDs in the order they are found in the coordinate file (either numeric or alphabetic order). For example, if you enter 3+6, inverses will be calculated and displayed from point 3 to 4, 4 to 5 and 5 to 6. You can use the <F2> key to view the information printed at the command line or you can view the print file (CGFile > Print/View Print File).

Inversing using Point Groups:

You can use a Point Group to inverse between a series of points specified by the point group. To specify a point group type a P or '*' and <Enter> at the command line. This will display a dialog box showing the Point Groups currently in the default directory.



Prompts

Enter point sequence

[Point group/Reset/sNap on] (last point = <none>): Enter or pick the first point on a line or the PC of a curve. Type "P" and Enter to use a point group to specify the inversing sequence. Type "R" and Enter to Reset the last point. Type "N" and Enter to turn on CAD snaps (these are turned off when the command starts).

[cLockwise curve/ccW curve/Point group/Reset/sNap on] (last point = 5): Enter or pick the next point ID to inverse to or type "L" and Enter or "W" and Enter to specify the radius point of a curve.

if you are entering a curve:

Enter radius point for curve [Reset/turn_Snap_on]: Enter or pick the radius point for the curve.

Enter point of tangency (PT) for curve [Reset/turn_Snap_on]: Enter or pick the PT point for the curve.

Pulldown Menu Location: CG-Survey > Cogo

Keyboard Command: cg_inverse

Prerequisite: coordinate file

Intersects

This feature allows you to calculate intersections based on one of the following methods:

Bearing-bearing

The bearing-bearing intersect is calculated based on a line passing through a point on a given bearing intersecting another line passing through a second point on another specified bearing.

Bearing-distance

This is based on a line passing through a point at a given bearing intersecting a circle at a given distance (radius) from a second point. This intersection by result in 2 points of intersection.

Distance-distance

This is based on intersecting a circle at a given distance (radius) from a point with another circle at another given distance (radius) from a second point. This may also result in 2 points of intersection.

Perpendicular

This is based on calculating the perpendicular distance from a given point to a line that passes through another point

at a specified bearing.

Tangent

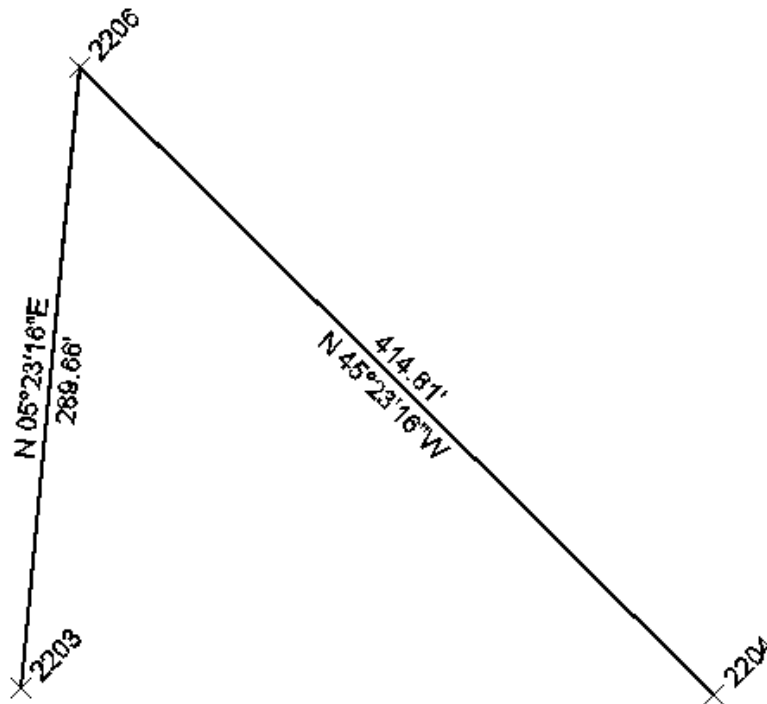
This is based on calculating the tangent points of a line drawn from a given point to a circle having a specified radius and radius point.

Command line input

After selecting the Intersects option on the CGCogo pull down menu and the Use Intersects Dialog menu item is not checked you will see the following prompt.

Intersection method: Brng-Brng/Brng-Dist/Dist-Dist/Perp/Tangent or offsets-on
[BB/BD/DD/Perp/Tangent/turn-Offsets-on]:

Bearing-Bearing Intersections:



Type "bb", then press <Enter>

At the Enter first Point: prompt type in or pick the point using the mouse

As an illustration, using the example shown in the figure: type or pick point 2203.

At the Enter first bearing: prompt there are 3 options available:

Type the bearing directly using the special C&G notation qddmss (quadrant, degrees, minutes and econds)
105.2316 (N 05° 23' 16"E)

Enter the two known C&G points that define the bearing either by typing the two points in with a dash between them or picking the two points one at a time using the mouse.

Or select a C&G line. When you select a line the bearing is computed by inverting between the two points that created the line. The bearing quadrant is based on traversing from the end point of the line farthest from the location where the line was picked to the end point of the line nearest to the point picked.

Enter second point: for the example type or pick point 2204

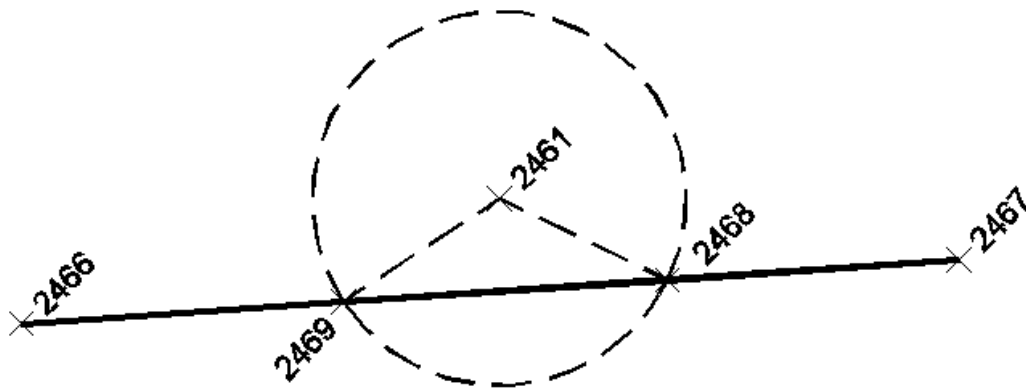
Enter second bearing: use any of the methods outlined for entering first bearing.

The intersection will then be calculated, the intersection point saved to the coordinate file and the results displayed at the command line.

When saving the intersection point, depending on your settings on the Global Settings tab of the C&G Options dialog, you may be asked to either enter or change the point ID, elevation, point code and description.

At each of the STORING POINT prompts there is an option to change settings [Settings]. Pressing S will bring up the Global Options tab of the C&G Options dialog box, allowing you to change settings prior to saving the point. (see the CGTools Chapter for a description of the CGOptions dialog box)

Bearing-Distance Intersection



Type bd to calculate the intersection of a circle with a line. Generally, the data is entered in the same fashion as for a bearing-bearing intersection.

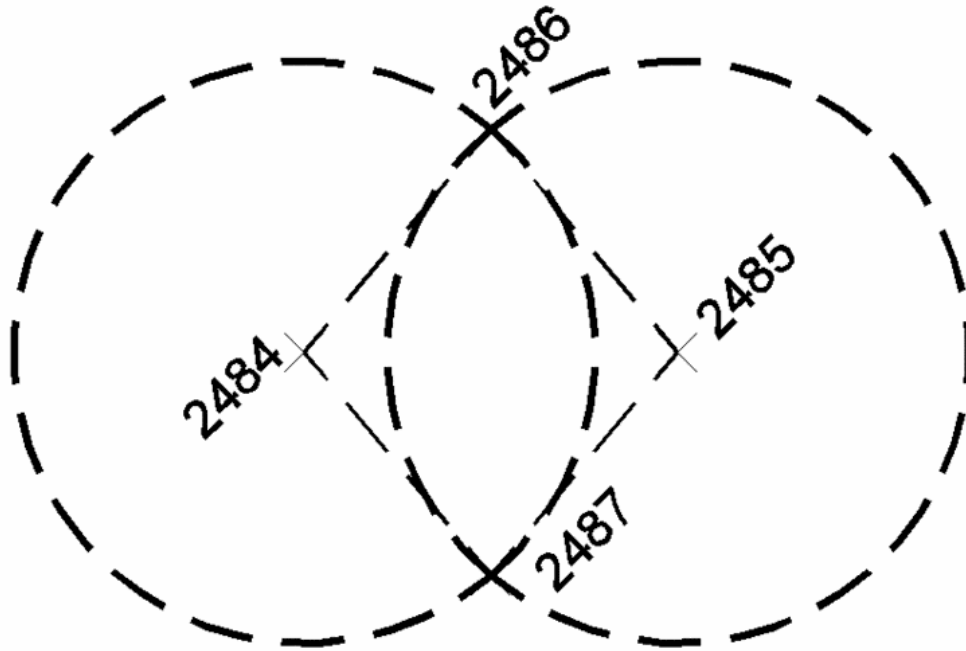
Once the data is entered each of the two solutions will be displayed one at a time.

You will be asked if the solution shown is the correct solution.

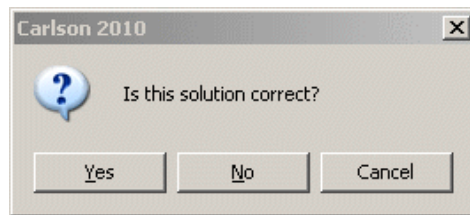
Is this the correct solution [Yes/No/ESC]:

If the solution is the correct one press Y for <Yes>. If it is not the correct solution press N for <No> and the second solution will be displayed. If neither solution is correct press <Esc> to cancel and return to the previous prompt.

Distance-Distance Intersection



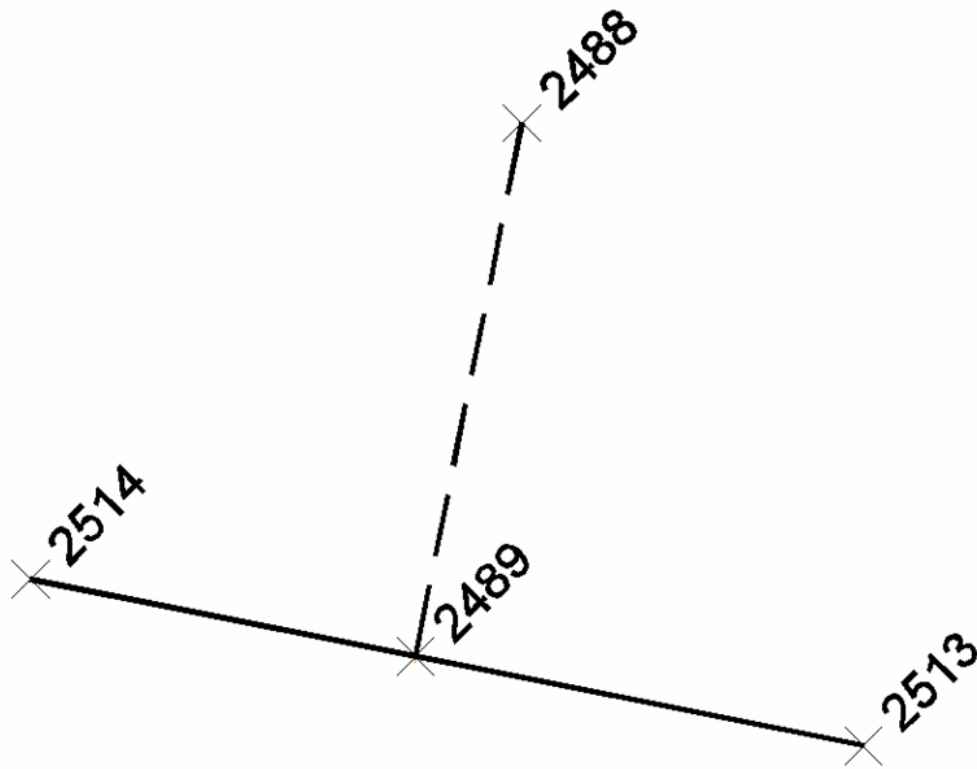
Type dd to calculate the intersection of two circles: with the distances being the radii of the circles
You will be prompted to enter the first radius point and distance (radius)
You will then be asked to enter the second radius point and distance (radius)
As with the bearing-distance intersection, the two possible solutions will be displayed and you will be asked to choose the correct one (see dialog below).



If you click the No button the other possible solution will be displayed. If you click the Yes button the intersection point will be stored. If you click Cancel the point will not be stored.

The routine will continue with additional DD Intersections prompts until you escape [ESC] the routine. The process will be repeated until the user presses <Esc> twice to end the command.

Perpendicular Intersection



Press P and <Enter> to calculate the point where the perpendicular constructed from a given point to a line intersects the line.

At the Enter first Point <>: prompt, type or pick a point on the line (in the example illustrated in the figure, type or pick point 2514)

At the Enter bearing: <>: prompt, type the bearing of the line or type or pick the two points defining the bearing (in the example, 2514-2513)

You will then be asked if you want to

Store Perp. Int. Pt. (Yes/No):

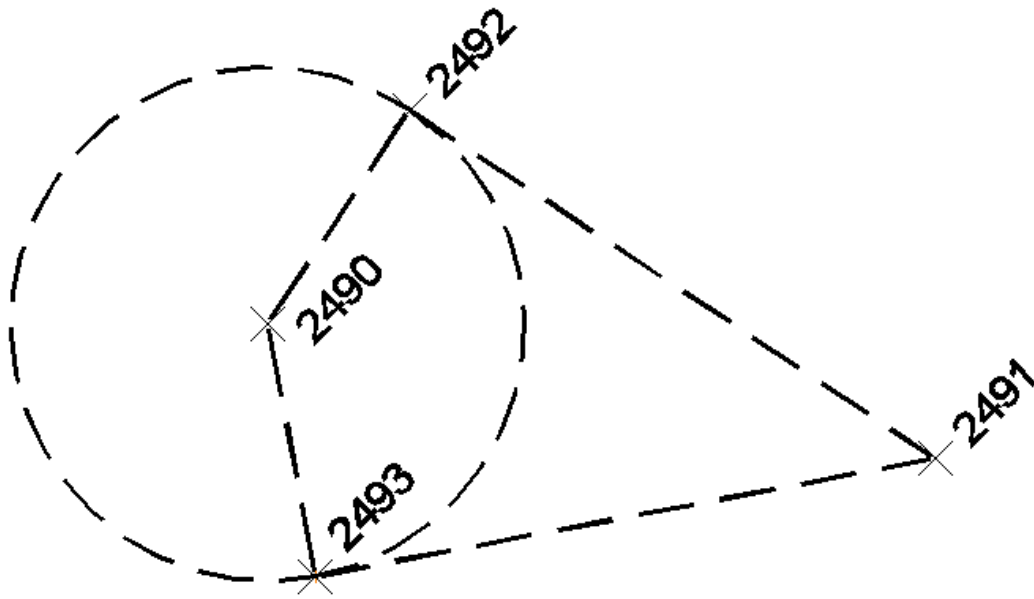
Choose whether to store the calculated point or simply view the data (you may not want to save the resulting intersection point).

Enter second point <>: Type the point ID, or use the mouse to pick the point from which the perpendicular to line is to be constructed (in the example, 2488).

The STORE POINT prompt will indicate the point being stored (in the example it will be 2489). Press <ESC> to cancel point storage.

To exit the Instrsects feature, the user must press <Esc> twice or the routine will repeat.

Tangent Intersection



Type T and <Enter> to calculate the points at which a line from a given point becomes tangent to a circle. You must choose a radius point for the circle, the radius of the circle, and the external point from which the tangent will start.

At the Enter radius point for circle: prompt, type or pick the center or radius point of the circle (in the example, 2490)

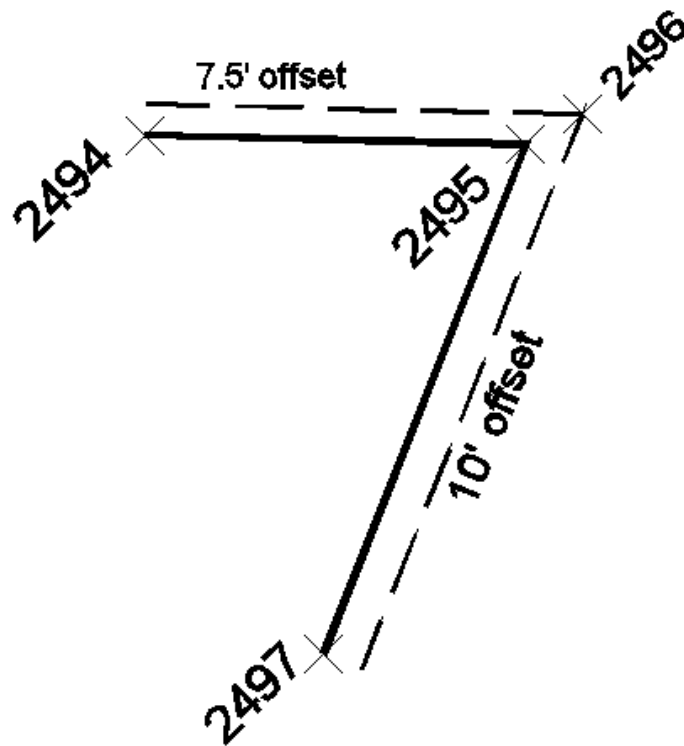
At the Enter radius of circle: prompt, type the radius or type or pick two points that define the radius distance (in this example, 2490-2492, or 69.92')

At the Second point: prompt, type or pick the point through which the tangent lines must pass (in the example, 2491)

As in some of the other intersection types, you must select the desired solution from the two possible solutions using the dialog shown for the distance-distance intersect.. If you click the Yes button, the intersection point will then be saved to the coordinate file and the results displayed at the command line.

To end the command, press <Esc> twice or the routine will repeat

Turn-Offsets-on:



Type O and <Enter> to turn the use of offsets on or off .

An example of an offset intersection would be the easement lines for a sewer line. This routine can calculate the offset intersection say for a 7.5' left offset and a 10' right offset, as shown.

For example:

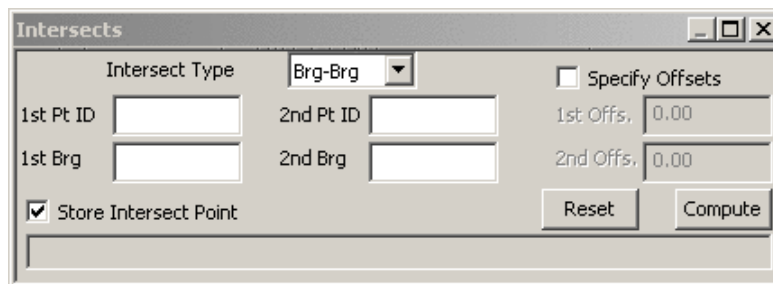
Enter first offset distance <>: -7.50

Enter Second offset distance <-7.50>: 10.00

The offset distances are positive if right of the line, as seen looking down the line in the direction of the defined bearing, negative if left of the line.

Intersects dialog

If the Use Intersects Dialog menu item is checked you will see the following dialog:



To use the Intersects dialog just set the Intersect Type drop down list to specify the type of intersect you wish to do.

Next click on the item you wish to specify. You may type in the information or you can move the cursor over the drawing area and you will be prompted for the information required for the edit box you were last in. To enter the next item, click on that edit box and type the information or, as before, move the cursor over the drawing and you

will be prompted for the necessary information for the last edit box you were in. Continue to do this until all information has been entered then click the Compute button to compute the intersection. The results will be printed

on the CAD command line and to the print file. If the Store Intersect Point check box is checked the intersection point will be stored in the coordinate file. You may specify offsets by checking the Specify Offsets checkbox and entering or picking the offset distance(s). Click the Reset button to remove all entered data from the dialog.

You may perform any other commands while the Intersects dialog is displayed - the data entered in the dialog will remain for use at any time.

Prompts

Not using Intersects Dialog:

Intersection method: Brng-Brng/Brng-Dist/Dist-Dist/Perp/Tangent or offsets-on

[BB/BD/DD/Perp/Tangent/turn-Offsets-on]: Type the 1 or 2 capitalized letters to specify the type of intersect to calculate or to turn offsets on.

point prompts

Enter point: Type a point ID or pick a point symbol on the screen.

Enter first point: Type a point ID or pick a point symbol on the screen.

Enter second point: Type a point ID or pick a point symbol on the screen.

Enter radius point for circle: Type a point ID or pick a point symbol on the screen.

Enter point on tangent line: Type a point ID or pick a point symbol on the screen.

bearing prompts:

Enter bearing <100.0000>: Type or pick the bearing.

Enter first bearing <100.0000>: Type or pick the bearing.

Enter second bearing <100.0000>: Type or pick the bearing.

distance prompts:

Enter distance <0.000000>: Type or pick the distance.

Enter 1st distance <0.000000>: Type or pick the distance.

Enter radius of circle <0.000000>: Type or pick the distance.

perpendicular intersect prompt:

Store perpendicular intersect point [Yes/No] <N>: Type "Y" or "N" and Enter.

Stopping to allow viewing of intersect point (red X) <Enter to continue> This prompt is displayed if you answered "N" to the previous prompt. Just press Enter to continue.

Intersects dialog: Type or pick the data into the appropriate edit boxes then click the Compute button to view the results and save the intersection point to the coordinate file.

Pulldown Menu Location: CG-Survey > Cogo

Keyboard Command: cg.intersects

Prerequisite: coordinate file

Station Offset

In order to use the station offset functions, you must create a Point Group (formerly called a batch point file or point file) defining a centerline alignment. The Coordinate Management > Point Groups > Create section describes how you can create a Point Group.

To illustrate the use of point groups in the various station offset features, the following point group file will be used:

7.25+ (Slope in, 1st vert. Curve)

1.75- (Slope out, 1st vert. curve & slope in, 2nd vert. Curve)

200* (Vertical Curve Length curve 1)

/123.50 (PVI Elevation curve 1)

*1300 (PVI Station curve 1)

2.00- (Slope out of curve 2)

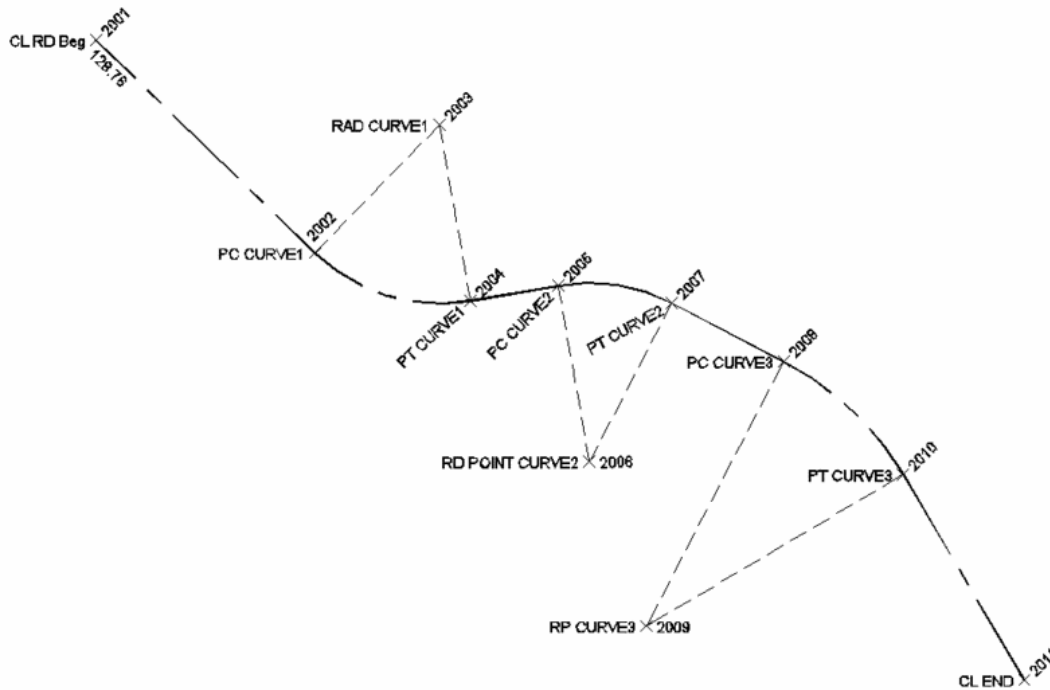
*1750 (PVI Station curve 2)

300* (Curve Length curve 2)

- 1000 (Subgroup name - also default beginning station)
- 2001 (Point ID at Sta. 10+00)
- 2002 (Point ID at PC of Curve)
- 2003 (Radius point ID, '-' indicates counter clockwise)
- 2004 (Point ID at PT of curve)
- 2005 (Point ID at PC of curve)
- +2006 (Radius point ID, '+' indicates clockwise curve)
- 2007 (Point ID of PT of curve)
- 2008 (Point ID of PC of curve)
- +2009 (Radius point ID, '+' indicates a clockwise curve)
- 2010 (Point ID of PT of curve - last entry in subgroup and in file)

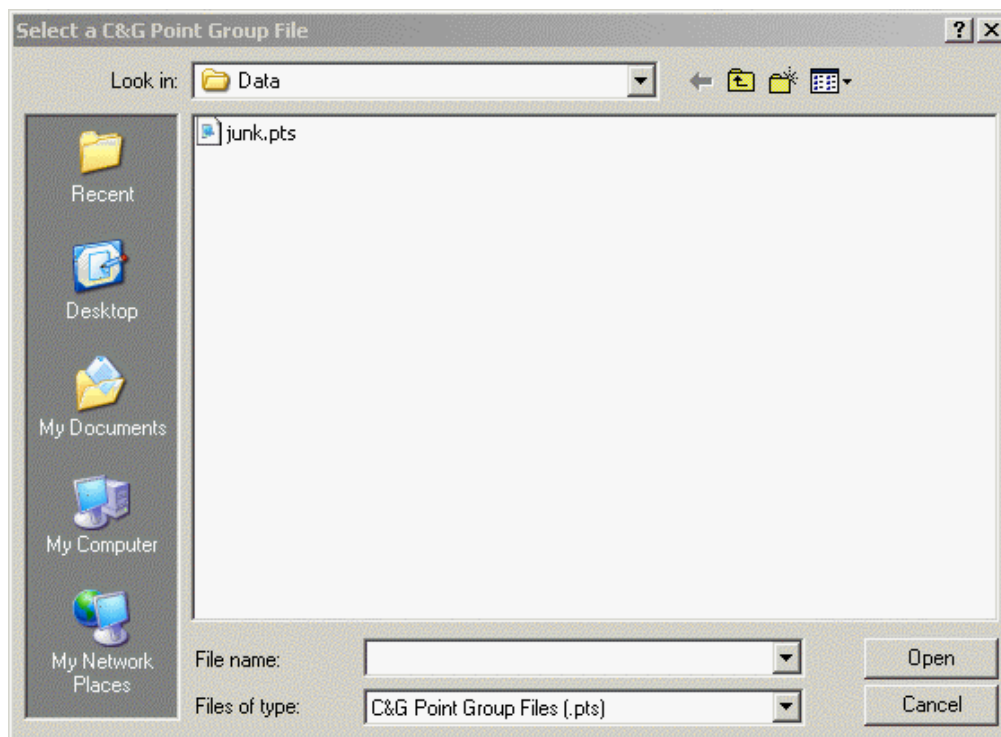
NOTE: Information shown in parenthesis are comments used here for explanation and do not appear in the point group file itself.

Alignment used for examples



Coords From Station Offset

This feature allows you to calculate and store a coordinate point for any given station and offset along an alignment defined by a point group. Use the Select a C&G Point Group File dialog to open a point group file.



Enter Starting Station [Done] <0.0000> You can Press <Enter> to use the default station shown, or you can enter a new starting station. If you enter a "+" followed by a value, ex. "+50", all stations on a 50 foot interval will be calculated automatically.

Enter Offset <0.00000>: Enter the offset distance from the alignment.

The point ID, station, offset, northing, easting, elevation and description will be printed at the command line and written to the print file.

You may repeat the process until you have calculated all the desired stations or press D then <Enter> to exit the command.

Note: If Elevations are set to "Calculate" and Elevation is on, (See General tab of the C&G Options dialog) then you will be asked to "Enter constant elevation change for offset points". The constant elevation change will be added to or subtracted from the calculated elevation of each newly created point on the alignment. If there is vertical curve information contained in the point group, this information will be used to calculate the initial elevation of each point. If there is no vertical curve information, the elevation of each new point will be calculated by interpolation between the elevations of the points contained in the point group.

Prompts

Enter Starting Station [Done] <0.0000>: You can Press <Enter> to use the default station shown, or you can enter a new starting station.

Enter station [Interval] <0.000000>: Enter a station expressed as a decimal number. Type "I" and Enter to specify an interval or, alternatively, you can precede the station number with a "+". This will cause stations to be automatically calculated based on the value you specify.

If you choose Interval in the previous prompt:

Enter interval: Enter the desired interval for automatic generation of stations.

Enter offset (+ = right, - = left) <0.000000>: Enter the offset distance from the alignment.

Pulldown Menu Location: CG-Survey > Cogo > Station Offset

Keyboard Command: cg_crds_from_staoff

Prerequisite: coordinate file, point group file defining the alignment

Create Point Group From Station Offset

This feature allows you create a point group by locating all the points along a predefined alignment at a given offset. It then sorts the points by station and saves the points to a new Point Group.

Select Create Point Group from Station-Offset from the menu.

Use the Select a C&G Point Group File: to open the point group file specifying the points in the alignment.

At the Calculate new points on the control line [Yes/No] <Y> prompt,

If you answer no to this prompt, the points chosen by you in the previous step will be saved in station order to the new point group file.

If you answer yes to the prompt, a new point will be created exactly on the offset line for each point found in the coordinate file that lies within the given range. The elevation of the new point will be set to the elevation of the nearby existing point and the new point IDs will be written to the new point group file instead of the existing point IDs.

Repeat the above steps to specify another offset.

Press D for <Done> to exit.

Prompts

At the Enter Offset <0.00000>: Type an offset if desired or just press enter for no offset. Offsets to the left should be preceded by a "-".

At the Enter Maximum Range <0.0000>: Specify the tolerance for points not exactly on the alignment.

Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt_group/Limits/Radius/Select]: Use the various methods available to choose the points to be tested for being within the tolerance from the alignment.

Calculate new points on the control line [Yes/No] <Y>: Type "Y" and Enter or just Enter to create new points on the alignment for those points found to be near the alignment and place these new point IDs in the point group being created. Type "N" and Enter to place the existing points in the point group being created.

Pulldown Menu Location: CG-Survey > Cogo > Station Offset

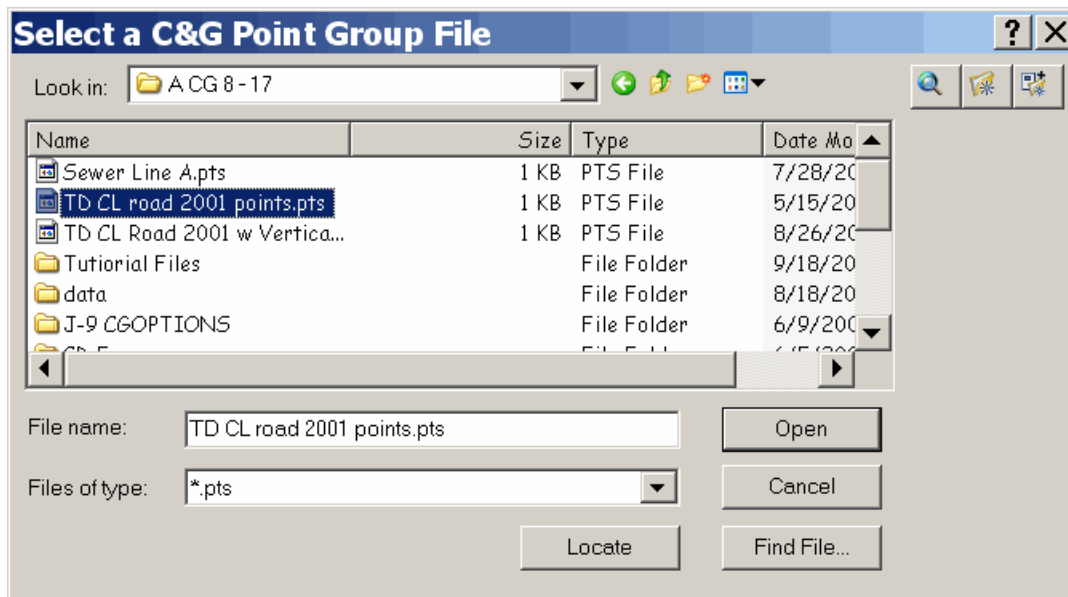
Keyboard Command: cg_bpf_from_sta

Prerequisite: coordinate file, point group containing the points in the alignment

Display Centerline Stations

This features allows you to view a list of the centerline stations for a given point group.

In the file dialog select the point group file that defines the alignment.



At the Enter Starting Station [Done] <1000>: prompt you will notice that the subgroup name is used to determine the starting station default value. To use the default value, press <Enter>. You may also enter a new starting station for the first point in the point group. For example, 24+34.12 is entered as 2434.12.

The centerline station information will be listed on the command line and written to the print file.

Point No.	Station	Offset	North	East	Elev.	Desc
2001	10+00.00	0.000	5524.47905	9768.66936	128.76	CL RD Beg
2002	12+55.95	0.000	5346.01086	9952.13553		PC CURVE1
2004	13+94.89	0.000	5305.76366	10082.50922		PT CURVE1
2005	14+69.89	0.000	5318.65419	10156.38943		PC CURVE2
2007	15+66.05	0.000	5303.94389	10251.42307		PT CURVE2
2008	16+72.04	0.000	5255.01123	10345.44075		PC CURVE3
2010	18+10.48	0.000	5160.36774	10445.29107		PT CURVE3
2011	20+10.67	0.000	4987.98790	10547.08317		CL END

The Enter Starting Station (Done) <0.00000>: prompt will again be displayed at the command line, at this point you can either enter another point group, or type in another starting elevation. When done entering data, type D and <Enter> for to end the command.

NOTE: The first line of the Point Group must be the beginning station of the alignment. In any routine that computes or requires stationing information the station numbers must be relative to the stationing in the Point Group file.

Prompts

At the Enter Starting Station [Done] <1000>: Enter the starting station for the alignment as a decimal number.

Pulldown Menu Location: CG-Survey > Cogo > Station Offset

Keyboard Command: cg_display_cl_sta

Prerequisite: coordinate file, point group file defining an alignment

Station Offset From Coords

This feature allows you to calculate the station and offset of selected points in the current coordinate file, based on the alignment as defined by the Point Group.

Select Station Offset from Coords from the menu.

You will first be asked to open the point group file that defines the alignment.

At the Enter Starting Station [Done] <0.00000>: prompt, enter the station of the first point in the point group. For example, if the first station is 24+34.12, enter it as 2434.12.

(As an example using the point group listed in the previous section, the starting station must be greater than 1000, the starting station in the point group, and less than 2016.05, the station of the last point in the point group. If not, no information will be displayed.)

Enter Maximum Range <0.00000>: The range identifies how far to look left and right of the alignment for points in the current coordinate file.

Next, you will be asked to select the points to be considered in computing the station offsets

Add points from coordinate file. (Enter when done) [All/Block/Code/Desc/Elev/Pt-group/Limit/Radius/Select]:

At the prompt type A and <Enter> for all the points in the current coordinate file or use the other options to choose a subset of the points. As indicated by the prompt, press <Enter> by itself to end point selection.

The station and offset information will be printed in order by station.

The Enter Start Station [Done] <0.0000>: prompt will appear again, you may enter the next starting station or type D and <Enter> to end the command.

Prompts

Enter Starting Station [Done] <0.00000>: Enter the starting station as a decimal number.

Enter Maximum Range <0.00000>: The range defines the distance tolerance left and right of the alignment for points selected from the current coordinate file.

Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt_group/Limits/Radius/Select]: Use the various methods to select points from the coordinate file to be searched for proximity to the alignment.

Pulldown Menu Location: CG-Survey > Cogo > Station Offset

Keyboard Command: cg_staoff_from_crds

Prerequisite: coordinate file, an existing point group file defining the alignment

Points on Line

This feature allows you to calculate and store points along a line at specified distances.

After choosing Points on Line from the menu and, if needed, opening a coordinate file, you will see following prompt:

Pts on line - specify No. of pts & dist., divide line, or place pts at interval [Number and dist/Divide/Interval]:

Choose one of the following:

Number and dist: Type N and <Enter> to specify a number of points at a given interval from the start point on a line.

Divide: Type D and <Enter> to select a line and indicate how many points you wish to create. The program then creates the specified number of points equally spaced along the length of the line.

Interval: Type I and <Enter> to create points at the specified interval along a line defined by 2 points.

Number and dist

This option allows you to calculate a given number of points at a fixed distance along a line. For example, you can set the corners for 3 lots at 150' intervals.

At the Enter start point: prompt, enter a point on the desired line by typing a point ID or picking a point with the mouse.

At the Enter bearing <100.0000>: prompt, use any one of the methods available to enter the bearing of the line on

which you wish the points to fall.

At the Enter distance <0.00000>: prompt, use any of the available methods to enter the distance between the points along the line.

Enter number of points: Enter the number of points you want created.

At the STORING POINT: prompt data required will vary depending on your current settings. You can enter a point number and its elevation, description, and code. This prompt will appear for each of the points created along the line

The Enter start point: prompt will repeat until you press <Esc>.

Pressing <Esc> again will allow you to create points on a line using one of the other methods.

Pressing <Esc> a third time to end command.

Divide

Choosing this method allows you to create points by dividing a line between two points into a specified number of divisions

Enter start point: Enter the first point defining the line by picking a point using the mouse or typing a point ID.

Enter end point: Enter the second point defining the line by picking a point using the mouse or typing a point ID.

Enter number of segments: Enter the number of points you want to create.

As the points are saved respond to the STORING POINT: prompt as required.

The Enter start point: will repeat unless you press <Esc>.

After pressing <Esc> once you may choose another method of creating points on a line or press <Esc> once more to end command.

Interval

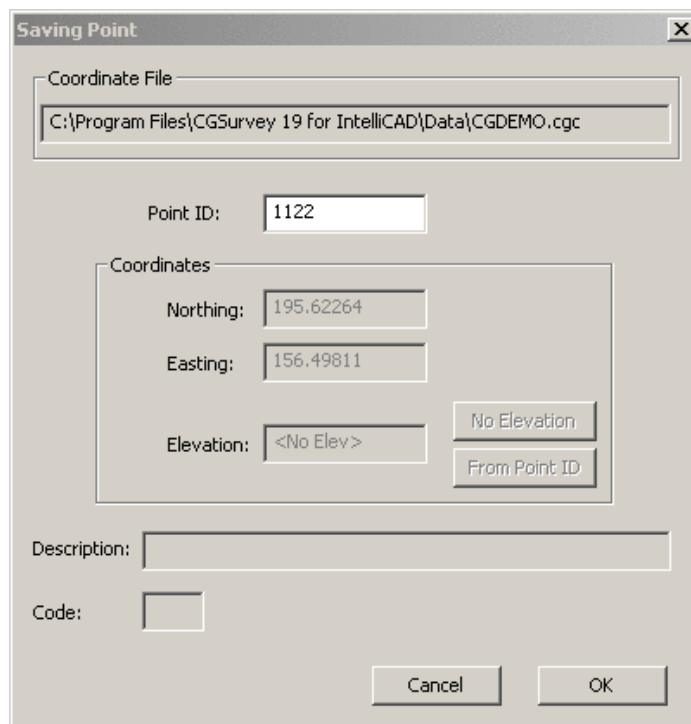
This option allows you to create as many points as possible at a specified interval on a line between two points.

Enter start point: Enter the first point defining the line by picking a point using the mouse or typing a point ID.

Enter end point: Enter the second point defining the line by picking a point using the mouse or typing a point ID.

Enter Distance <0.0000>: Enter the desired distance between the points created along the line. Points will be created along the line at the given distance. As many points as will fit between the end points at the given spacing will be created.

No matter which method is used to create points, the Saving Point dialog (see below) will appear for each of the points created.



Click OK to save the point in the coordinate file.

Repeat from the Enter start point: prompt or press <Esc> to use another method to create points along a line. Press <Esc> once more to end the command.

Prompts

Specify: Number of points and distance, divide line or points at an interval

[Number and dist/Divide/Interval]: Type "N" and Enter to create a specified number of points along a line a specified distance apart. Type "D" and Enter to create a specified number of points between 2 points. Type "I" and Enter to create a point a specified distance from the starting point of a line specified by 2 points.

Enter start point: Enter the point ID or pick the point symbol for the starting point of the line.

for Number and Dist:

Enter bearing <100.0000>: Enter the bearing of the line along which the points are to be created.

Enter distance <0.000000>: Enter the distance between the points.

Enter number of points: Enter the number of points to be created.

for Divide:

Enter end point: Enter the point ID or pick the point symbol for the ending point of the line.

Enter number of segments: Specify the number of points to be created on the line between its end points.

for Interval:

Enter end point: Enter the point ID or pick the point symbol for the ending point of the line.

Enter distance <0.000000>: Enter or pick the distance along the line for creating the new point.

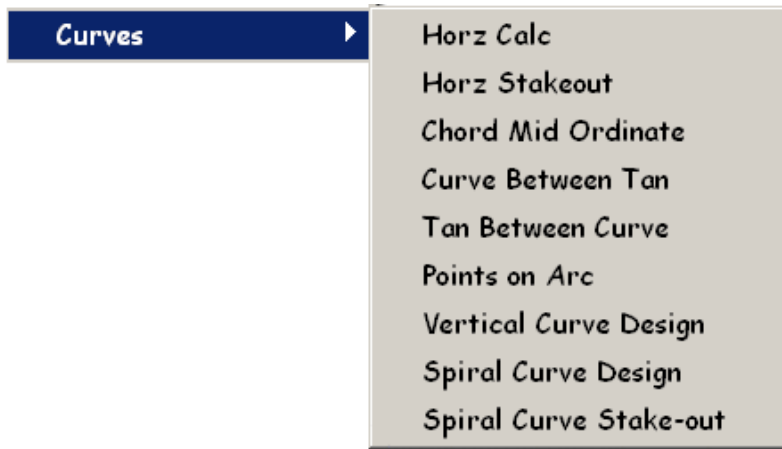
Pulldown Menu Location: CG-Survey > Cogo

Keyboard Command: cg_pol

Prerequisite: coordinate file

Curves

There are several possible curve calculations available on the Curves submenu. The available options will be described in the following sections.



Calculate Horizontal

This feature allows you to calculate the components of a horizontal curve but does not save any points to the coordinate file.

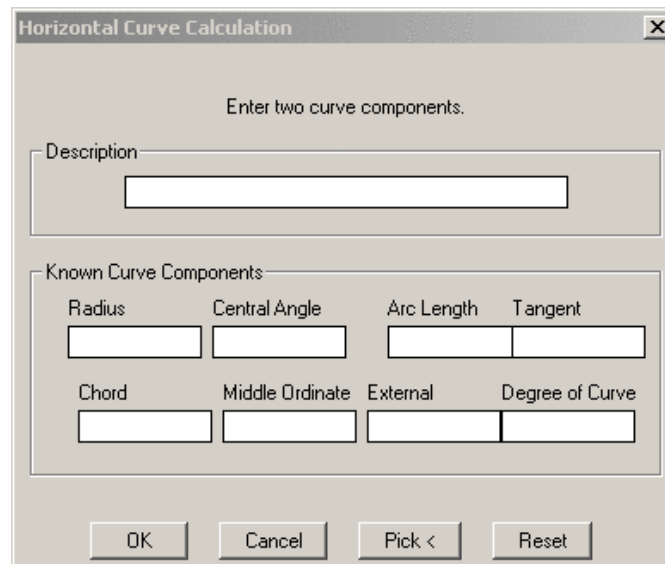
In the dialog enter any two curve components then press OK to calculate the other components.

To use the mouse to pick the two known components press the Pick button and pick the PC, PT and radius points or a C&G or non-C&G arc.

The description field is merely used to identify the curve in the printout.

The Reset button clears all fields.

When done, press the Cancel button to close the dialog.

A screenshot of a dialog box titled 'Horizontal Curve Calculation'. The dialog has a close button (X) in the top right corner. Inside, there is a text area labeled 'Description' with a text input field below it. Below that is a section titled 'Known Curve Components' containing two rows of input fields. The first row has four fields: 'Radius', 'Central Angle', 'Arc Length', and 'Tangent'. The second row has four fields: 'Chord', 'Middle Ordinate', 'External', and 'Degree of Curve'. At the bottom of the dialog are four buttons: 'OK', 'Cancel', 'Pick <', and 'Reset'.

Prompts

Horizontal Curve Calculation dialog: Enter any two curve components then press OK to calculate the other

components.

Pulldown Menu Location: CG-Survey > Cogo > Curves

Keyboard Command: cg_horz_calc

Prerequisite: None

Curve Between Tangents

This feature allows you to calculate the curve components for a curve between two tangent lines given either the radius, the length of the tangent line or a point through which the arc passes.

At the Enter first point [Done]: prompt, enter or pick a point on one of the tangent lines. The point ID of the point selected will be displayed on the command line.

Enter first Bearing <100.000000>: use any of the available methods to enter the bearing from the point you just selected going toward to the point of intersection (P.I.) of the curve. The bearing entered will be displayed on the command line.

Enter second point: type or pick a point on the other tangent line.

Enter second bearing <100.000000>: enter the bearing of the other tangent.

Offset out <0.000000>: This is an optional entry. It allows you to calculate a point outside the curve (for example, on the right-of-way). Press <Enter> to use the default value or enter another offset. The offset used will be displayed on the command line.

Offset in <0.000000>: 50 This optional entry allows you to calculate a point inside the curve.

Enter point on arc [Radius mode/Tangent-Distance mode]:

At this prompt there are three options as to how to specify the location of the desired curve:

At this prompt you can type or pick a point on the arc,

Or you can type R and <Enter> to get the prompt:

Specify radius of curve [Tangent-mode/Point-on-arc-mode]:

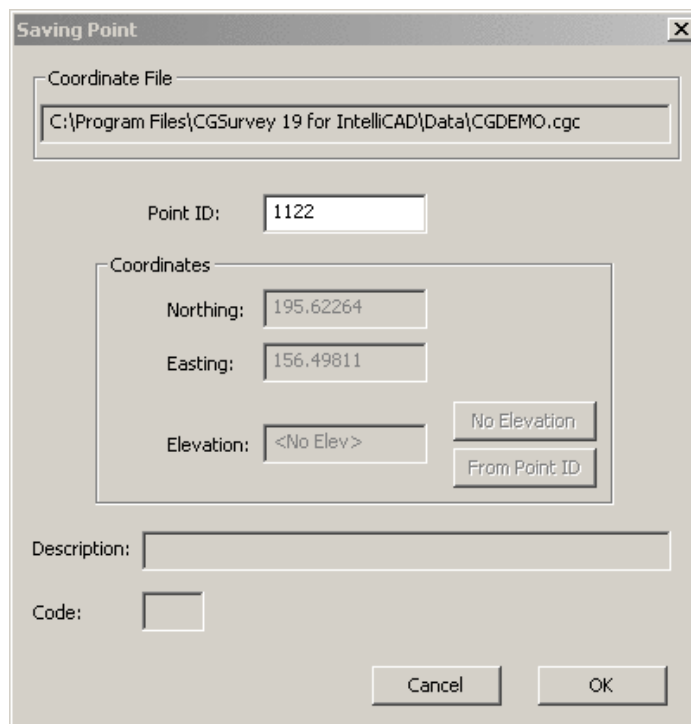
At this prompt specify the radius of the curve.

Or you can type T and <Enter> to get the prompt:

Specify tangent distance [Radius-mode/Point-on-arc-mode]:

At this prompt enter the distance from the PC or PT to the PI.

The locations of the PC, PI, PT, and radius point are calculated and the Saving Point dialog (see below) will appear once for each.



Depending on the Global Options settings, the calculated points may be drawn. If Auto Line Plot is on, the arc will be drawn as well. The coordinates of points that were created and the curve information will be displayed at the command line.

To end the command type D and <Enter> at the Specify an existing point on the first tangent line [Done]: prompt.

Prompts

Specify an existing point on the first tangent line [Done]: Enter or pick a point on one of the tangent lines.

Specify the bearing of the first tangent line <100.0000>: Enter the bearing or pick 2 points or a line to define the bearing.

Specify an existing point on the second tangent line: Enter or pick a point on the second tangent line.

Specify the bearing of the second tangent line <100.0000>: Enter or pick the bearing of the second tangent.

Offset out <0.000000>: This is an optional entry. It allows you to calculate a point outside the curve

Offset in <0.000000>: This optional entry allows you to calculate a point inside the curve.

Specify an existing point on the arc [Radius_mode/Tangent_distance_mode]: Enter or pick a point on the arc or change how you define the arc by entering "R" and Enter for the Radius method or "T" for the Tangent-distance method

Specify radius of curve [Tangent_distance_mode/Point_on_arc_mode]: Enter the radius or change the mode.

Specify tangent distance [Radius_mode/Point-on-arc-mode]: Enter the tangent distance or change the mode.

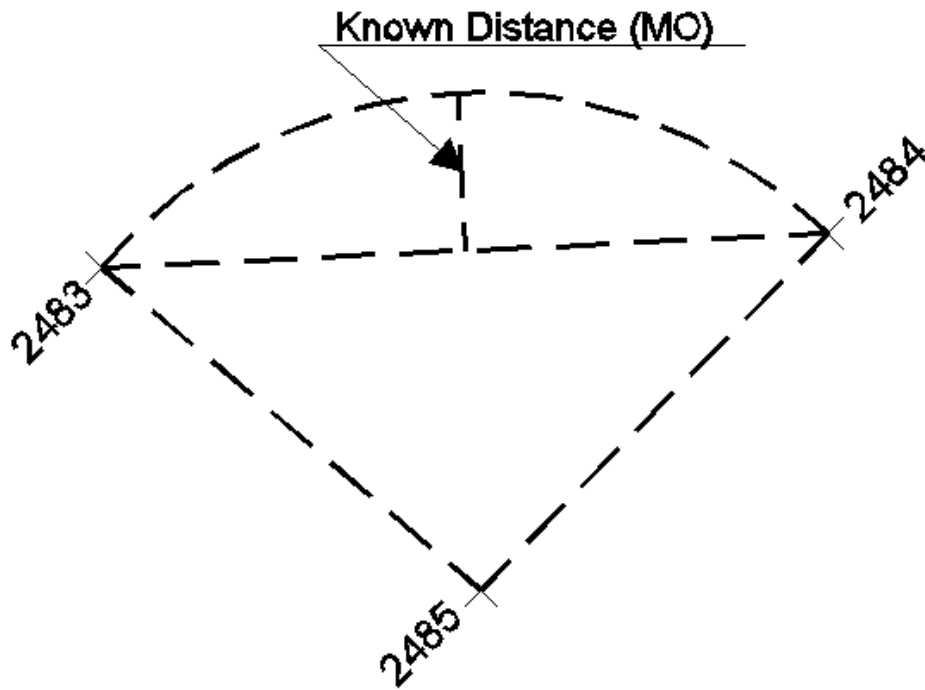
Pulldown Menu Location: CG-Survey > Cogo > Curves

Keyboard Command: cg_cbt

Prerequisite: coordinate file

Middle Ordinate Solution

Allows you to calculate the other curve elements when you can locate the chord and determine the middle ordinate distance in the field.



Prompts

Save coordinates [Yes/No] <Y>: press Enter or type "Y" and Enter if you want the calculated radius point to be stored in the coordinate file. If not type "N" and Enter. Press Esc key to end the command.

Enter P.C. point: specify the PC point by typing a point ID or picking a point on the screen.

Enter PT point: specify the PT point by typing a point ID or picking a point on the screen.

Middle ordinate: Type in the middle ordinant distance or pick it on the screen.

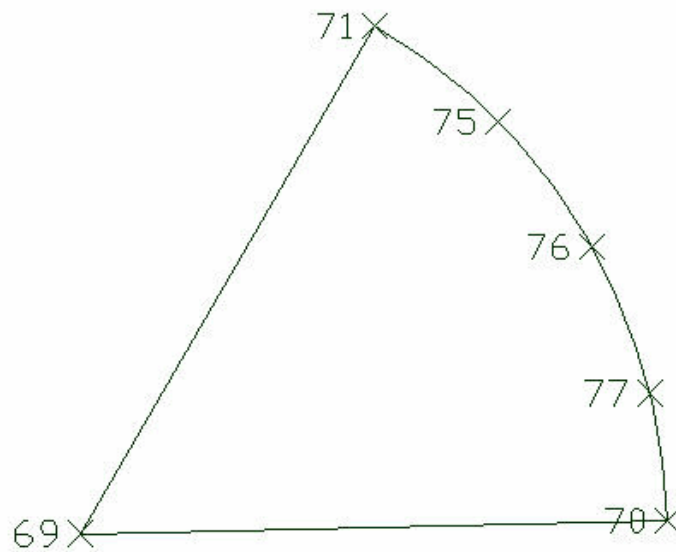
Pulldown Menu Location: CG-Survey > Cogo > Curves

Keyboard Command: cg_chd_mo

Prerequisite: coordinate file

Points on Arc

This feature allows you to create points along an arc. The first point is set at a distance measured along the arc starting at the PC.



Enter PC point or pick a C&G Curve: enter or pick the PC point or pick a C&G curve.

If you picked a C&G Curve, the PC, Radius point, Pt point and arc length will be displayed at the command line. After picking a C&G Curve, skip the next 2 steps.

Enter PT point: Enter or pick the PT point.

Enter radius point [cLockwise/ccW]: For a clockwise curve either type L or '+' and <Enter> then pick or type a point ID or type a point ID preceded by a '+'. For a counter clockwise curve either type a W or a '-' and <Enter> then pick or type a point ID or type a point ID preceded by a '-' .

Enter arc length [Occupy/Multiple points] <0.000000>:

Occupy option

In occupy mode the points are located along the arc with the arc length being measured from the previous point. Thus the occupied point moves ahead to the last computed point as calculations proceed. When you type O and <Enter> the prompt becomes:

Enter arc length [do not Occupy/Multiple points] <0.000000>:

Specify the arc length at the prompt.

A point will be created and you will be prompted for the next arc length. Continue entering arc lengths until done then press <Esc> to return to the Enter PC point or pick a C&G Curve: prompt.

Multiple points option

This option allows you to compute multiple points along the arc at a given distance. The specified distance is used to set as many points along the arc as will fit between the PC and the PT. When you type M and <Enter> the prompt becomes:

Enter arc length [do not Occupy/Single point] <0.000000>:

Specify the arc length at the prompt.

As many points as can be fit between the PC and the PT will be created. You will then be prompted for the next arc length. Continue entering arc lengths until done then press <Esc> to return to the Enter PC point or pick a C&G Curve: prompt.

At the Enter PC point or pick a C&G Curve: prompt you can continue specifying curves or you can press <Esc> to end the command.

Prompts

Enter PC point or pick a C&G Curve: Enter a point ID or pick a point symbol or a C&G Arc on the screen. Press Esc to end the command.

Enter PT point: Enter or pick the PT point.

Enter radius point [cLockwise/ccW]: For a clockwise curve either type "L" (or a plus sign '+') and <Enter> then pick or type a point ID or type a point ID preceded by a '+'. For a counter clockwise curve either type a "W" (or a minus sign '-') and <Enter> then pick or type a point ID or type a point ID preceded by a '-' .

Enter arc length [Occupy/Multiple points] <0.000000>: Enter an arc length to create a single point on the arc. Enter "O" and Enter to "occupy" the calculated point so the next arc length is calculated from that point instead of the PC. Enter "M" and Enter to calculate multiple points along the arc at a specified distance.

Enter arc length [do not Occupy/Single point] <0.000000>: For multiple points enter the desired arc length or change input mode.

Pulldown Menu Location: CG-Survey > Cogo > Curves

Keyboard Command: cg_poa

Prerequisite: coordinate file

Spiral Curve Design

This feature allows you to design a spiral curve. You will be prompted using standard spiral curve component nomenclature.

Follow the prompts described in the Prompts section below and the following Spiral curve data is calculated and displayed at the command line and written to the print file.

Point of Intersection of the spiral (PI for spiral)

Tangent point of the spiral (TS for spiral)

Point where spiral meets simple curve (SC for spiral)

Radius point of the simple curve

Point where simple curve meets outgoing spiral (CS for spiral)

Point where spiral meets tangent on outgoing side (TS for spiral)

For each of the points calculated the prompt

The Saving Point dialog (see below) will appear and allow you to specify the point ID.

The image shows a 'Saving Point' dialog box with the following fields and controls:

- Coordinate File:** C:\Program Files\CGSurvey 19 for IntelliCAD\Data\CGDEMO.cgc
- Point ID:** 1122
- Coordinates:**
 - Northing: 195.62264
 - Easting: 156.49811
 - Elevation: <No Elev>
- Buttons:** No Elevation, From Point ID
- Description:** (empty text box)
- Code:** (empty text box)
- Bottom Buttons:** Cancel, OK

Click OK to cause the point to be stored in the coordinate file.

At the Curve Description <enter if done>: prompt you can either enter the description for another curve or press <Enter> to end the command.

Prompts

Curve description <enter if none>: This description is optional but is used to identify the information in the results. Press Enter to end the command.

Enter the first point: Enter or pick a point on the tangent going into the spiral.

Enter first bearing <100.0000>: Enter or pick the bearing from first point to the P.I. of the spiral.

Enter second point: Enter or pick a point on the tangent going out of the spiral.

Enter second bearing <100.0000>: Enter the bearing from second point to the P.I.

Enter Radius [Degree.of.curve] <0.000000>: Enter or pick the radius or type "D" and Enter to change to degree of curve prompt.

or

Enter Degree of Curve [Radius]: Enter or pick the degree of curve or type "R" and Enter to change to radius prompt.

Enter spiral length in <0.00000>: Enter or pick the length of the spiral coming from the first tangent into the simple curve. Enter a zero for no spiral in.

Enter spiral length out <0.00000>: Enter the length of the spiral from the simple curve out to the second tangent. Enter a zero for no spiral out.

Pulldown Menu Location: CG-Survey > Cogo > Curves

Keyboard Command: cg_scd

Prerequisite: coordinate file

Spiral Curve Stakeout

This feature allows you to calculate points along a spiral at a given interval and offset for use in staking out the curve in the field.

Curve Description <enter if done>: This text description is only used to identify the spiral curve data printed at the command line and written to the print file.

Enter the first point: Enter or pick a point on the tangent line going into the spiral.

Enter first bearing <100.000000>: Enter the bearing going toward the P.I. of the spiral from the first point.

Enter the second point: Enter or pick a point on the tangent line going out of the spiral.

Enter second bearing <100.0000>: Enter the bearing from the tangent point just defined to the P.I. for the spiral.

Enter Radius [Degree of Curve] <0.0000>:

Radius option: Entering the radius is the default option as indicated by the wording of the prompt.

Degree of Curve option: To change to entering the degree of curve type D and <Enter>.

Once you have chosen the type of data you wish to specify, type or pick the radius of the simple curve between the two tangents or the degree of curve.

Enter spiral length in <0.000000>: enter the length of the spiral from the TS (Tangent to Spiral) to the SC (Spiral to Curve). Enter zero for no spiral in.

Enter spiral length out <0.000000>: enter the length of the spiral from the CS (Curve to Spiral) to the ST (Spiral to Tangent). Enter zero for no spiral out.

Enter P.I. station <0.0000>: Enter the station of the P.I. For example: station 460+28.52 is entered as 46028.52

Enter station interval <0.0000>: Specify the interval at which you wish to stake the spiral. For example, enter 50 to stake every 50 units.

Enter offset from centerline <0.0000>: This can be a positive or negative number depending on whether you want to set points inside or outside the spiral. If you want to place points on the centerline, simply press <Enter> to use

the 0.00 default value.

Odd stations to be staked (6+34.22 as 634.22): You can stake as many odd station locations as needed. When done press <Enter> without entering a station value.

The points will be calculated and stored in the coordinate file. The station and offset will be placed in the description field.

The results will be printed at the command line and in the print file.

Continue to stake another spiral or press <Esc> or <Enter> at the Curve Description <enter if done>: prompt to end the command.

Prompts

Curve Description <enter if done>: This text description is optional and is used to identify the spiral curve in the output

Enter the first point: Enter or pick a point on the tangent line going into the spiral.

Enter first bearing <100.00000>: Enter the bearing going toward the P.I. of the spiral from the first point.

Enter the second point: Enter or pick a point on the tangent line going out of the spiral.

Enter second bearing <100.0000>: Enter or pick the bearing from the second tangent point to the P.I. for the spiral.

Enter Radius [Degree of Curve] <0.0000>: Enter or pick the radius. or, to change to entering the degree of curve, type "D" and Enter.

Enter spiral length in <0.000000>: enter the length of the spiral from the TS (Tangent to Spiral) to the SC (Spiral to Curve). Enter zero for no spiral in.

Enter spiral length out <0.000000>: enter the length of the spiral from the CS (Curve to Spiral) to the ST (Spiral to Tangent). Enter zero for no spiral out.

Enter P.I. station <0.0000>: Enter the station of the PI. For example: station 460+28.52 is entered as 46028.52

Odd stations to be staked (6+34.22 as 634.22): Enter as many odd stations as needed. When done press Enter.

Pulldown Menu Location: CG-Survey > Cogo > Curves

Keyboard Command: cg_scs

Prerequisite: coordinate file

Stakeout Horizontal

This feature allows you to create points for field staking a horizontal curve.

After choosing the Horizontal Stakeout menu item, and opening a coordinate file you will be asked if you want to Save coordinates [Yes/No] <Y>.

If you respond Yes (or press enter), a new point will be saved to the coordinate file for each point to be staked along the curve. No matter how you answer this question, stakeout information will be generated and displayed.

Enter curve description: Enter a description that will allow you to identify the curve in the output.

P.C. station <0.000000>: Enter the station for the P.C. of the curve.

Station interval <0.000000>: Enter an interval for staking the points along the curve.

Odd stations to be staked (6+34.22 as 634.22) [Done]: Enter the station of any odd location along the curve to be staked. For example, you may wish to stake the point on the curve at which a pipe crosses or the point where the extension of a property line intersects the curve. You may enter as many odd stations as required. When done, press <Enter> at the prompt without entering a new odd station or press D and <Enter>.

Offset from C/L <0.000000>: enter a non-zero value here if you must stake points offset from the main alignment - for example: along a curb line, a barrier wall or along a property line.

If the distance is entered as a positive number, the distance will be added to the radius or staked outside the curve. If the number entered is negative, it will be subtracted from the radius or staked inside the curve. To stake the centerline, enter zero.

At the Enter PC point or pick a curve: prompt you can type a point ID for the P.C. or use the mouse to pick a point

or a C&G curve on the screen.

If you picked a C&G curve in the previous step, you need not enter the PT point or the radius point so skip the next 2 steps.

Enter PT point: type the point ID for the P.T. or click the point on the screen.

Enter radius point [cLockwise/ccW]: Use any of the available methods of specifying a radius point.

Type the radius point: If the curve is in a clockwise direction from the P.C. to the P.T., enter the point number preceded by a plus sign, e.g. +18. If the curve is in a counterclockwise direction from the P.C. to the P.T., the point ID preceded by a minus sign, e.g. -18.

Pick the radius point with the mouse: If the curve is in a clockwise direction from the P.C. to the P.T., type an L or a '+' and <Enter>, then use the mouse to pick the point on the screen. If the curve is in a counterclockwise direction from the P.C. to the P.T., type either a W or a '-' and <Enter>, then pick the point on the screen.

The report will be printed at the command line and written to the print file.

The command will repeat until you press <Esc> at the Save coordinates [Yes/No] <Y>: prompt to end command.

Prompts

Save coordinates [Yes/No] <Y>: Type "Y" and Enter or just Enter if you wish to save resulting points to coordinate file. Type "N" and Enter if not. Press Esc to end command.

Enter curve description: Enter a description that will allow you to identify the curve in the output.

P.C. station <0.000000>: Enter the station for the P.C. of the curve.

Station interval <0.000000>: Enter an interval for staking the points along the curve.

Odd stations to be staked (6+34.22 as 634.22) [Done]: Enter as many "odd" stations to be staked. Type Enter or "Done" and Enter when all odd stations have been entered.

Offset from C/L <0.000000>: If you wish to stake stations not on the centerline, enter the offset and press Enter or just press Enter to accept the default offset. Positive offset is outside the radius and negative is inside.

Enter PC point or pick a curve: Enter a point ID, pick a point symbol or pick a C&G arc.

if you did not pick a C&G arc:

Enter PT point: Enter or pick the PT point.

Enter radius point [cLockwise/ccW]: Enter "L" or "W" to choose the type of curve then Enter or pick the radius point. You may also enter "+" and Enter then enter or pick a point for the radius point of a clockwise curve or Enter a "-" and enter or pick a point for the radius point of a counter clockwise curve.

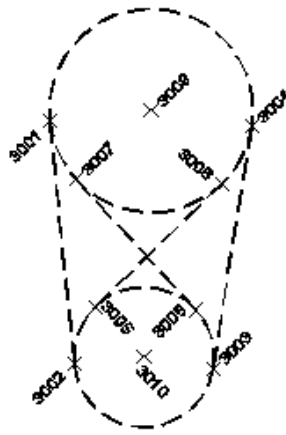
Pulldown Menu Location: CG-Survey > Cogo > Curves

Keyboard Command: cg_hcso

Prerequisite: coordinate file

Tangent Between Curves

This feature allows you to calculate the end points of a tangent line joining two curves. This may be used, for example, to layout roads which do not have curve/tangent information.



Follow the prompts noted below and, if a solution is possible, the endpoints of the tangent between the two curves will be calculated. Point IDs will be assigned and coordinates stored for the points of tangency. Repeat or enter "D" when done.

NOTE: There are 4 tangent solutions for this problem. The solutions sets differ according to the sign preceding the radius or degree of curve.

Enter first radius point [Done]: Enter or pick the center point for one of the curves. Press Enter or type "D" and Enter when done.

Enter first radius <0.00000>: Enter the radius of the first curve. Use a "+" sign before the point ID to specify a clockwise curve or a "-" sign to specify a counter clockwise curve.

Enter second radius point: Enter or pick the center point for the second the curve.

Enter second radius <0.00000>: Enter the radius for the second curve. Use a "+" sign before the point ID to specify a clockwise curve or a "-" sign to specify a counter clockwise curve.

Pulldown Menu Location: CG-Survey > Cogo > Curves

Keyboard Command: cg_tbc

Prerequisite: coordinate file

Vertical Curve Design

This feature prints a list of station and elevation information for stations along one or more vertical curves.

Use the Odd stations to be staked (6+34.22 as 634.22): prompt to enter any stations along the curve for which you wish elevation information. This permits you to calculate elevations over culverts or at other important locations.

The calculated station and elevation information . The station, tangent elevation, tangent offset and grade elevation will be printed at the command line and in the print file. The high or low point will be marked with an asterisk.

Repeat the process to design another vertical curve or press <Esc> at the Enter curve description: prompt to end the command.

Prompts

Enter curve description: Description is used to identify the curve in the output.

Enter slope in <0.00000>: The slope is entered as a percent. For example: enter -1.5 for a 1.5% downhill slope.

Enter slope out <0.00000>: Enter the slope as a percent.

Enter length of vertical curve <0.00000>: Enter the length of the vertical curve.

Enter PVI Station <0.000000>: Enter the PVI station. For example: Enter 1250.00 for station 12+50.00

Enter PVI Elevation <0.000000>: Enter the PVI elevation.

Enter station interval <0.000000>: Enter the station interval.

Odd stations to be staked (6+34.22 as 634.22): Enter any stations along the curve for which you wish elevation information.

Pulldown Menu Location: CG-Survey > Cogo > Curves

Keyboard Command: cg_vcd

Prerequisite: coordinate file

Area Summary

The Area Summary feature allows you to get information on the area and perimeter of one or more parcels and the tract that contains the parcels.

After choosing the Area Summary menu item and, if required, opening a coordinate file, you are asked to specify the type of Area Summary you want:

Type of Area Summary [Complete/Area only/Mapcheck] <C>:

Complete Area Summary

Complete summary allows you to get complete information on the area and perimeter of parcels and the tract that contains the parcels

Source of points defining area [Point group/Manual entry] <P>

If you have a Point Group, enter <P>. Enter <M> if you prefer to manually enter the points.

Once the overall area and parcels have been defined either by using a point group or manually entering the information, the Complete Area Summary is displayed at the command line.

The points used in defining the area are listed first. If there are any arcs involved in the area computation, all of the elements of the curve will be displayed as well. After listing the points defining the area, the area and perimeter summary are reported.

Area Only

The data input is the same as for the Complete Area Summary but the report produced contains only the area of each parcel and the accumulated area for the entire tract

Mapcheck Area

The data input is the same as for the Complete Area Summary as is the resulting report except that it also includes closure information. The closure information includes the correct ending coordinates; the actual ending coordinates; the northings, eastings, and bearing and distance of the error; the total distance traversed and the overall closure.

Prompts

Open Coordinate File dialog: If a coordinate file is not open, you will be asked to open one.

Type of Area Summary [Complete/Area only/Mapcheck] <C>: Press "C" and Enter or just Enter for Complete, "A" for Area only, or "M" for Mapcheck.

Source of points defining area [Point group/Manual entry] <P>: Type "P" and Enter or just Enter to use a point group to specify the points defining the tract. Type "M" and Enter to specify the points defining the tract by typing in point numbers or picking from the screen.

Pulldown Menu Location: CG-Survey > Cogo > Area

Keyboard Command: cg_asum

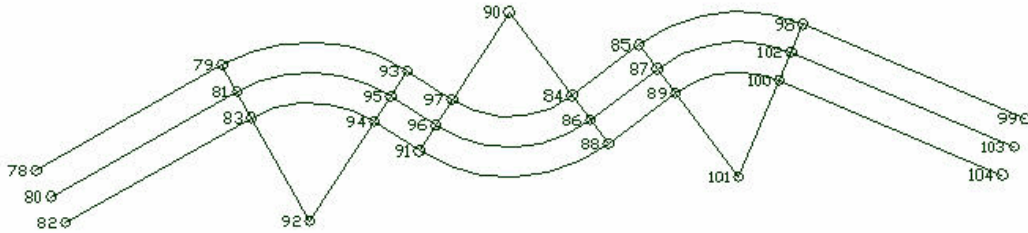
Prerequisite: coordinate file

Roadways

The roadways submenu contains 2 features: Right-of-Way/Easements and Intersections/Cul-de-Sacs and the Intersections/Cul-de-Sacs has a submenu containing several features for each type of intersection or cul-de-sac.

Right-of-Way Easements

The Right-of-Way/Easements feature allows you to compute offsets left and/or right of an alignment.



After the alignment points have been entered, offset points will be created to the left and right of each point you specified in the alignment. If Auto Point Numbering is on, the calculated points will be stored in the coordinate file. Depending on your settings for Auto Line Plot and Auto Point Plot in the Graphic tab of the C&G Options dialog, the new points and lines may also be drawn. If Auto Point Numbering is off, you will see the Saving Point dialog and can accept or change the default point number and other information associated with the point.

Prompts

Enter offset right<0.000000>: Enter the offset to the right of the alignment.

Enter offset left<0.000000>: Enter the offset to the left of the alignment.

Method for specifying center line points [Point group/Manual entry] <P>: To use a point group type "P" and Enter or just Enter and select the point group from a file dialog box. Type "M" and Enter to specify the alignment interactively.

Pulldown Menu Location: CG-Survey > Cogo > Roadways

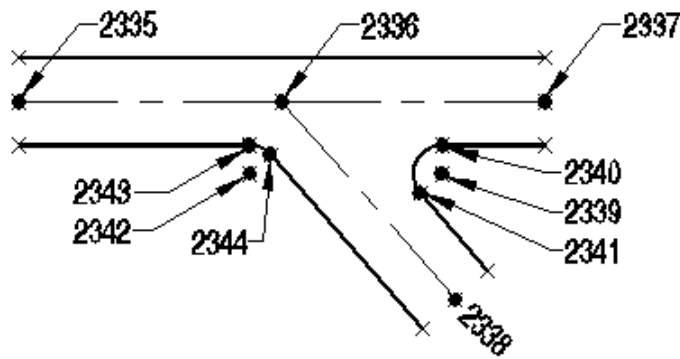
Keyboard Command: cg_rw

Prerequisite: coordinate file containing points defining the alignment

Intersections/Cul-de-sacs

T Intersections

This feature allows you to calculate the right-of-way intersection points and/or the fillet points and fillet radius points (if fillets are used) at T type intersections. One or both of the roads may have arc centerlines.



The points defining the fillet points will be calculated and stored in the coordinate file. Repeat as needed or press <Esc> or <Enter> at the Enter C/L intersection point (Enter when done): prompt to end the command.

Prompts

Enter C/L intersection point (Enter when done): enter or pick the centerline intersection point.

Enter through road C/L bearing [Arc] <0.000000>: If the through road is a straight road, enter or pick the bearing for the road. Otherwise type A and <Enter> to switch to Arc mode and enter the radius point of the through road.

Enter through road width <0.000000>: Enter the width of the through road.

Enter 2nd road C/L bearing away from intersection [Arc] <0.000>: enter the 2nd bearing or press <A> for Arc and enter the radius point. The bearing is away from the intersection.

Enter 2nd road width <0.000>: Enter the 2nd road width.

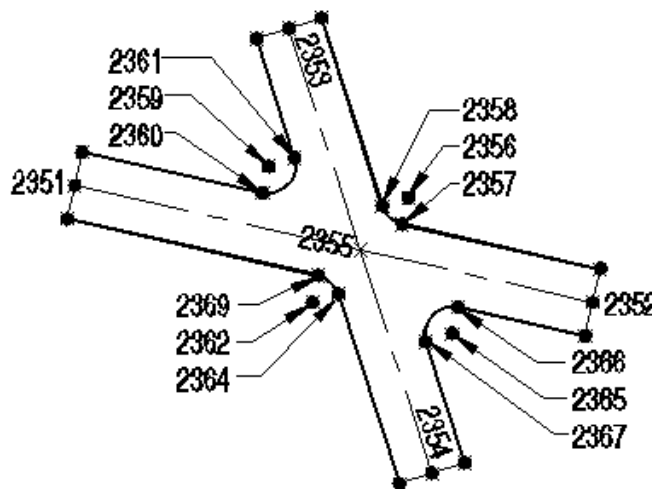
Enter fillet radius <0.000>: If you do not want to have fillets, press <Enter> to use the 0.00 default value. Otherwise, enter the fillet radius.

Pulldown Menu Location: CG-Survey > Cogo > Roadways > Intersections/Cul-de-sacs

Keyboard Command: cg_tint

Prerequisite: coordinate file

X Intersections



The radius, PC and PT points for each fillet will be calculated and stored. If Auto Line Plot is on, the fillet arcs will be drawn. Repeat as needed then press Enter or Esc to end the command.

Prompts

Enter C/L intersection point (Enter when done): enter or pick the intersection point of the two road centerlines.

Enter 1st road C/L bearing [Arc]: If first road is straight, enter the 1st bearing. If it is an Arc, enter the 1st radius point.

Enter 1st road width: Enter 2nd road C/L bearing (Arc): If the intersection point for the first road centerline is on a straight segment, enter the bearing of the centerline. If it is on an arc, type A and <Enter> then enter the first road centerline's radius point.

Enter 2nd road width: Enter the second road width.

Enter 2nd road C/L bearing [Arc]: Enter the second road centerline bearing or type A and <Enter> and specify the centerline radius point for the second road.

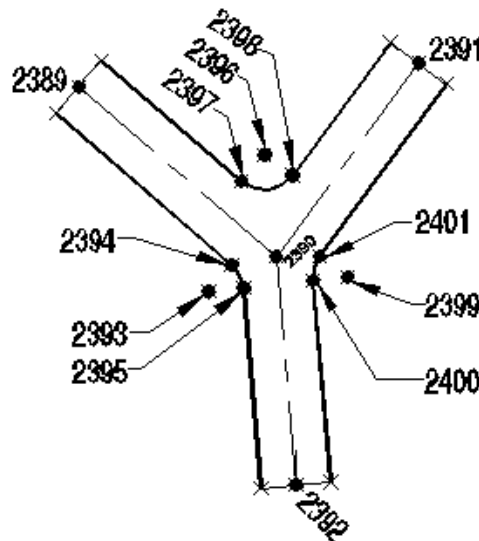
Enter fillet radius: Enter the fillet radius or zero, if there are no fillets.

Pulldown Menu Location: CG-Survey > Cogo > Roadways > Intersections/Cul-de-sacs

Keyboard Command: cg_xint

Prerequisite: coordinate file

Y Intersections



The radius, PC and PT points for each fillet will be calculated and stored and if Auto Line Plot is on, the fillet arcs will be drawn. Repeat as necessary then press <Esc> or <Enter> to end command.

Prompts

Enter C/L intersection point (Enter when done): Enter or pick the intersection point for the three road centerlines.

Enter 1st road C/L bearing away from intersection point: Enter one of the road centerline bearings (going away from the intersection point).

Enter 1st road width: enter the first road width.

Enter 2nd road C/L bearing away from intersection point: Enter another of the road centerline bearings (going away from the intersection.)

Enter 2nd road width: Enter the width of the second road.

Enter 3rd road C/L bearing away from intersection point: Enter last of the road centerline bearings (going away from the intersection.)

Enter 3rd road width: Enter width of third road.

Enter fillet radius: Enter radius of fillets or zero for none.

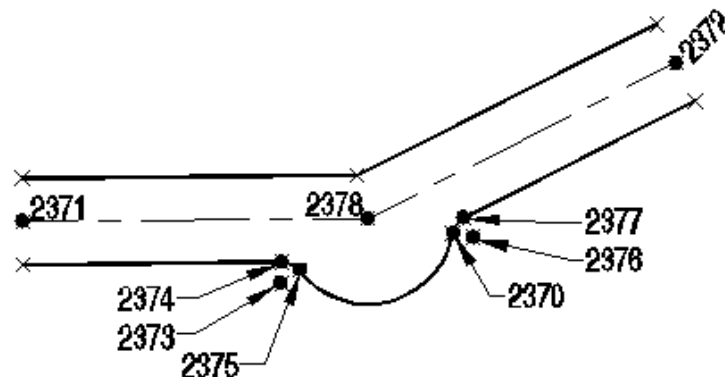
Pulldown Menu Location: CG-Survey > Cogo > Roadways > Intersections/Cul-de-sacs

Keyboard Command: cg_yint

Prerequisite: coordinate file

Bubble Cul-de-Sac

This type of cul-de-sac is also called a fish-eye cul-de-sac. It is commonly used at sharp turns in roads in subdivisions.



Follow the prompts described below. When done the fillet points and radius points will be stored in the coordinate file. These points will be plotted and the fillet arcs will be drawn if the C&G settings call for it. You may repeat the process as necessary or press <Esc> to end command:

Prompts

Enter cul-de-sac radius point (Enter when done): type or pick the radius point.

Enter cul-de-sac radius <0.000>: Enter the radius of the cul-de-sac.

Enter 1st C/L bearing away from radius point <0.000>: Enter the bearing along the first roadway centerline away from the cul-de-sac radius point.

Enter 2nd C/L bearing away from radius point <0.000>: Enter the bearing along the second roadway centerline away from the cul-de-sac radius point

Enter road width <0.000>: Enter the roadway width.

Enter fillet radius <0.000>: Enter the fillet radius or zero for no fillets.

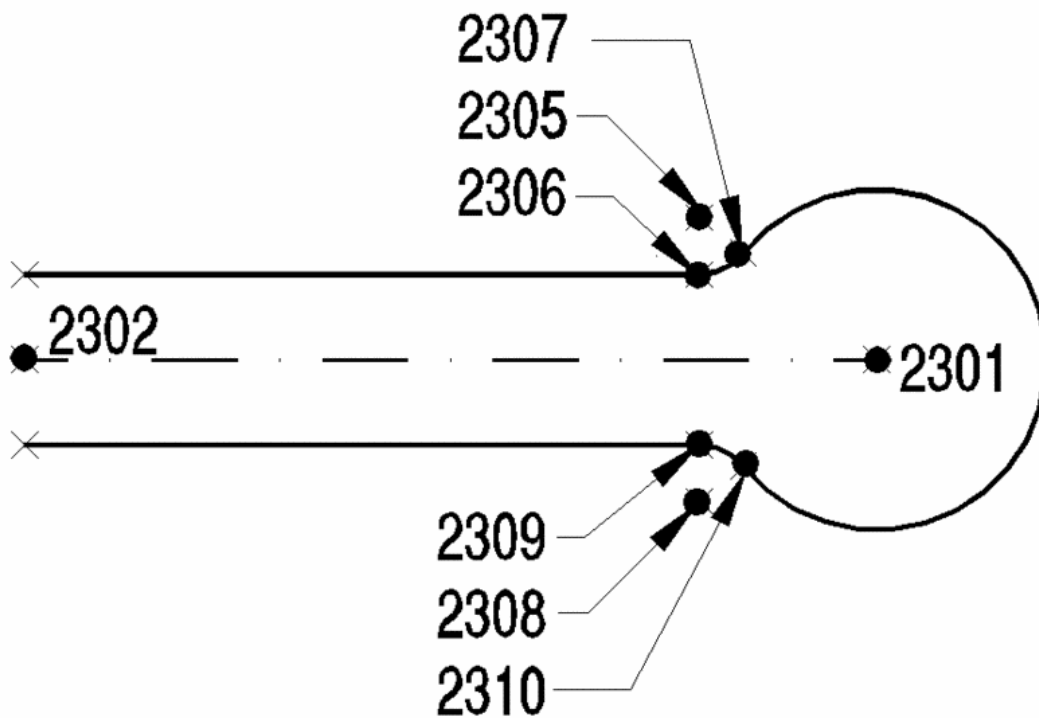
Pulldown Menu Location: CG-Survey > Cogo > Roadways > Intersections/Cul-de-sacs

Keyboard Command: cg_bcul

Prerequisite: coordinate file

Standard Cul-de-Sac

The standard cul-de-sac is a common feature of most subdivisions.



Enter cul-de-sac radius point (ENTER when done): type or pick the cul-de-sac radius point (in this example, point 2301).

Enter cul-de-sac radius <0.000000>: Enter or pick the radius of the cul-de-sac (60 units in the example).

Enter C/L bearing away from radius point [Arc] : In computing a straight cul-de-sac you must enter the bearing of the road centerline going away from the radius point. For the example the bearing is from point 2301 to point 2302.

Enter point on C/L (NOT radius point): This must be a C&G point on the centerline but cannot be the same as the cul-de-sac radius point. In this case we can use point 2302

Enter road width <0.000000>: enter the total width of the road right-of-way.

Enter fillet radius <0.000000>: Enter the fillet radius. Remember, you do not have to have a fillet radius, you may enter zero here.

The points needed to define the cul-de-sac and the fillets are calculated and stored in the coordinate file.

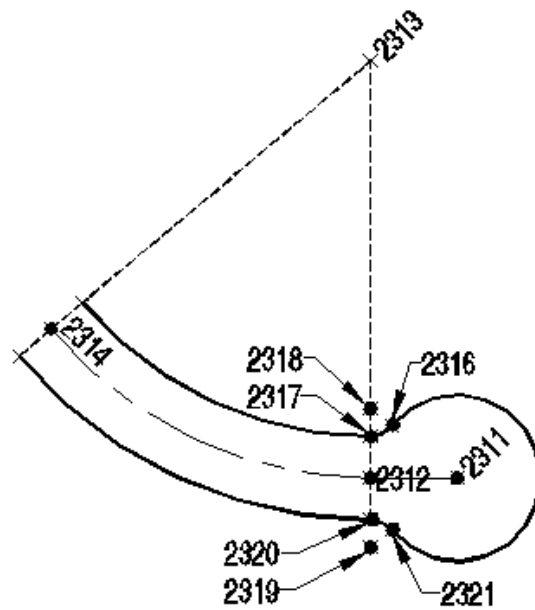
If Auto Line Plot is on the lines for the cul-de-sac and the fillets will be drawn automatically.

You may repeat the process as many times as necessary.

When done, press <Enter> at the

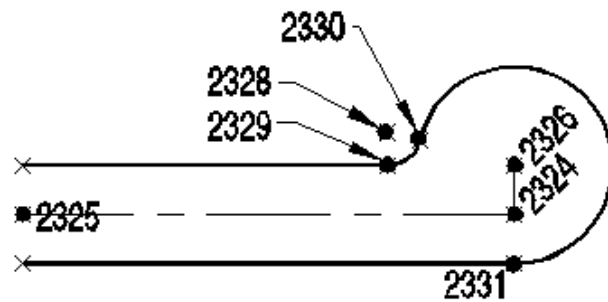
Enter cul-de-sac radius point (ENTER when done):

Standard Cul-de-Sac on ARC



The procedure for a cul-de-sac on arc is the same as it is for a straight cul-de-sac, except at the Enter C/L bearing away from radius point [Arc] : prompt, choose A for Arc, then enter the C/L radius point for the roadway, in this case point 2313.

Offset cul-de-sac:



The procedure for offset cul-de-sac is the same as a straight cul-de-sac except the radius point is the offset point. In the sketch, the point 2326 is the radius point. The bearing is from 2324 toward 2325 and the point on the C/L would be point 2325.

Prompts

Enter cul-de-sac radius point (ENTER when done): type or pick the cul-de-sac radius point. Press Enter when done.

Enter cul-de-sac radius <0.000000>: Enter or pick the radius of the cul-de-sac.

Enter C/L bearing away from radius point [Arc] : Enter or pick a point. For a straight cul-de-sac roadway, this must be a point on the centerline but cannot be the same as the cul-de-sac radius point. Type "A" and Enter to specify information for a cul-de-sac on a curved roadway.

if you chose a curved roadway:

Enter C/L radius point [Line]: Type a point ID or pick a point symbol or type "L" and Enter to switch back to a straight roadway.

Enter point on C/L (NOT radius point): Type a point ID or pick a point symbol.

Enter road width <0.000000>:

Enter fillet radius <0.000000>: Enter the fillet radius. You may enter 0.0 for no fillets.

Pulldown Menu Location: CG-Survey > Cogo > Roadways > Intersections/Cul-de-sac

Keyboard Command: cg_scul

Prerequisite: coordinate file

Stake-Out

The Stakeout feature allows you to calculate the required information for either a radial stakeout or staking out using angles right.

Angles Right

This feature is similar to the Radial Stakeout feature except it allows you select the foresight points one at a time.

If a coordinate file is not open, a file dialog will appear, allowing you to open an existing coordinate file.

After following the prompts outlined below, the angle from the backsight point, the distance, the foresight point ID, the azimuth and the description are printed at the command line and written to the print file.

You may press <F2> to view the complete listing of angles and distances.

Repeat the prompt sequence as many times as are required.

Press <Enter> or type D and <Enter> when done.

Prompts

Enter the instrument point. [Done]: Type or pick the instrument point.

Enter backsight point. [Done]: Type or pick the backsight point.

Enter foresight point. [Done]: Type or pick the foresight point.

Pulldown Menu Location: CG-Survey > Cogo > Stake-Out

Keyboard Command: cg_ar

Prerequisite: coordinate file

Radial Stake Out

This feature allows you to obtain the angles required to stakeout several foresight points from a single instrument point.

If a coordinate file is not open, a file dialog box will appear, allowing you to open an existing coordinate file.

After following the prompts outlined below, the angles right from the backsight point, the distances, the foresight point IDs, the azimuths and the descriptions for all the selected points will be printed at the command line and written to the print file.

You may press <F2> to view the complete listing of angles and distances.

Repeat as necessary for as many setups as are required.

When done, press <Enter> to end the command.

Prompts

Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt_group/Limits/Radius/Select]: Use one of several methods to specify the points in the coordinate file to be staked out.

Enter the Instrument point [Done]: Type or pick the instrument point.

Enter backsight point [Done]: Type or pick the backsight point.

Pulldown Menu Location: CG-Survey > Cogo > Stake-Out

Keyboard Command: cg_rso

Prerequisite: coordinate file

Best Fit

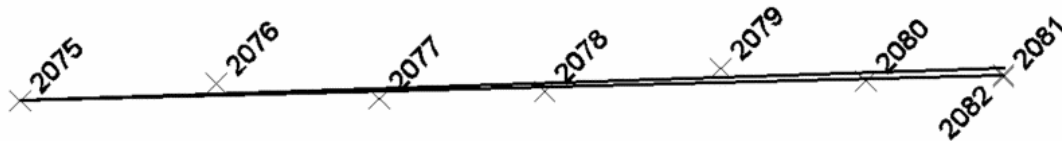
The best fit feature uses a least squares algorithm to compute the best fit line or circle for the points selected. The user can assign a weight to each point that is between 1 and 15, a point with a weight of 15 acts as if there are 15 of the points at the same location and thus skews the fit closer to that point. This is done to skew the result in favor of certain points. A weight of 0 means do not adjust this point or give it "infinite" weight.

When you choose Best Fit from the CGCogo menu you will see the following prompt:

Enter the type of best fit problem [Line/Arc/Tan-arc-tan] <L>:

Best Fit Line:

Press <Enter> for Line to calculate the best fit line through a series of points. In the example in the figure below, 2075 is an Iron Pin Found that we do not want adjusted, so the weight will be set to 0.



Enter a point ID or pick a point symbol on the line: for the example, type or pick 2075

Enter weight for point 2075 <1>: for the example type 0 (zero) and <Enter>

Enter a point ID or pick a point symbol on the line: for the example, type or pick 2076

Enter weight for point 2076 <1>: for the example, type 8 and <Enter>

Continue entering point ID - weight pairs until done then press <Enter> when asked for the next point ID.

The point locations and weights will be used to compute the best fit line .

The results, a list of the point IDs entered and their offsets from the best fit line and the bearing of the line, is printed at the command line and written to the print file.

Printed output for the line example

Pt.: 2075 Wt.: 0 Offset: 0.000 RT

Pt.: 2076 Wt.: 8 Offset: 5.360 LT

Pt.: 2077 Wt.: 3 Offset: 3.411 RT

Pt.: 2078 Wt.: 6 Offset: 1.915 RT

Pt.: 2079 Wt.: 2 Offset: 6.326 LT

Pt.: 2079 Wt.: 2 Offset: 6.326 LT

Pt.: 2080 Wt.: 4 Offset: 1.093 RT

Pt.: 2081 Wt.: 4 Offset: 0.986 RT

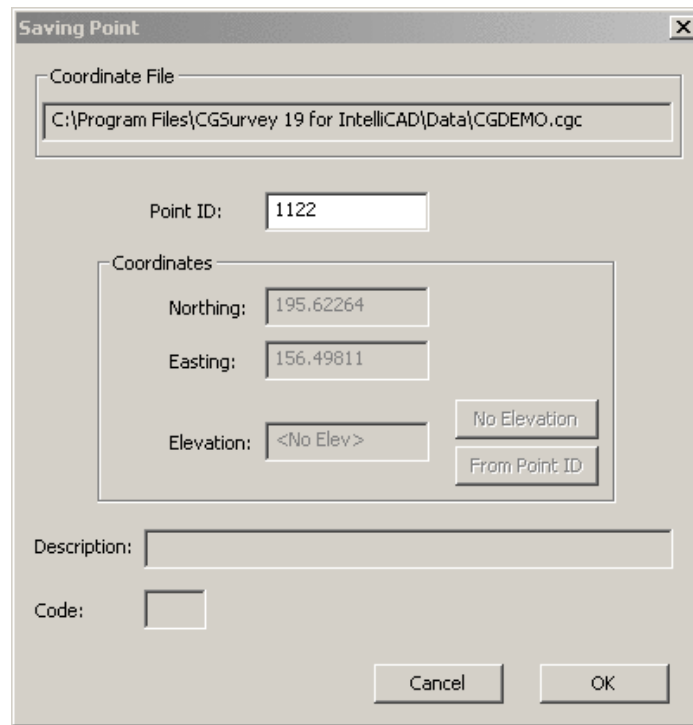
N: 6354.64727 E: 8112.07615 Dir: N 88°30'32"E

N: 6366.14982 E: 8553.96369 Dir: S 88°30'32"W

At the [Edit/Ok/Quit] <0>: prompt:

If you are satisfied with the results, press <Enter> for Ok and the endpoint coordinates will be computed and saved.

For each point saved, the Saving Point dialog (see below) will be shown.



Clicking OK will cause the point to be saved to the coordinate file.

If, on the other hand, you wish to edit the input data, type E and <Enter>. You see the following prompt:
[Add/Change/Delete/eXit]:

Add - add another point to the calculation

Change - change the weight of one of the points

Delete - remove one of the points from the calculation

eXit - when done editing.

If you wish to cancel the command without calculating the line type Q and <Enter>.

Best Fit ARC:

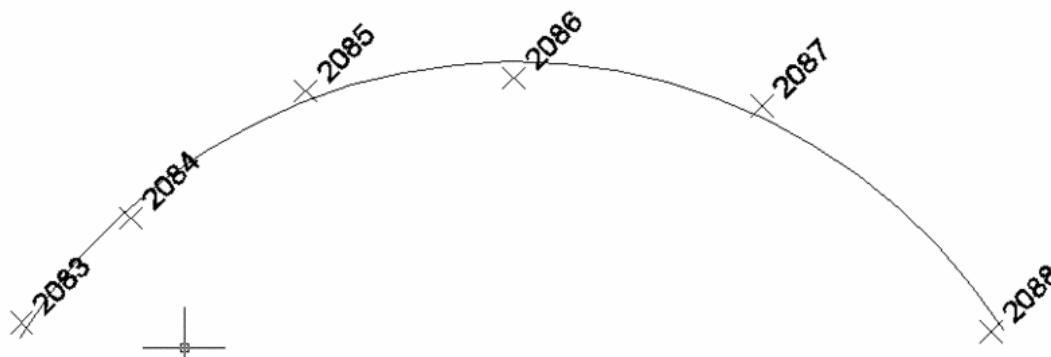
This option allows you to calculate the best fit arc through a series of points. As with the Line option, each point can be weighted from 0 (no adjustment) to 15.

At the Enter the type of best fit problem prompt, type A and <Enter> to choose the Arc option.

Enter the point ID - weight pairs as in the Line option.

When all the point ID - weight pairs have been entered press <Enter> at the Enter or Pick a C&G point on the line prompt.

A table of the results similar to that for the Line option will be displayed at the command line.



Output for the example in shown in the figure

Pt.: 2083 Offset: 2.862 OUT
Pt.: 2084 Offset: 3.699 IN
Pt.: 2085 Offset: 3.608 OUT
Pt.: 2086 Offset: 6.280 IN
Pt.: 2087 Offset: 5.584 OUT
Pt.: 2088 Offset: 1.393 IN
N: 6369.30690 E: 8269.70677 RAD: 237.282

The [Edit/Ok/Quit] <O>: prompt and its options for editing the input data are explained in the section on the Line option.

If you are satisfied with the results, press <Enter> for Ok and the PC, PT and radius point of the best fit arc will be saved to the coordinate file using the Saving Point dialog.

Tan-arc-tan:

This option allows you to calculate a combination of the best fit tangent line going into a curve, the best fit arc for the curve itself and the best fit tangent line out of the curve through a series of points defining two tangent lines and an arc.

The two tangent lines are calculated using a least squares solution and then the best fit arc is calculated. The method used to find the best fit arc is to calculate a radius and radius point for each point on the arc using a function that calculates a curve between tangents through a known point. Each radius and radius point is weighted based on the central angles between the PC, point-on-arc and PT points. The larger the central angles, the higher the resulting weight will be. All the calculated radii and radius points are then averaged. It is not necessary that you locate the actual PC or PT points in the field.

At the prompt

Enter the type of best fit problem [Line/Arc/Tan-arc-tan] <L>:

First you must enter the points on the 1st tangent line. At the series of prompts to:

Enter a point ID or pick a point symbol on 1st tangent:

and

Enter weight for point XXXXX <1>:

enter the point ID - weight pairs for the tangent going into the curve.

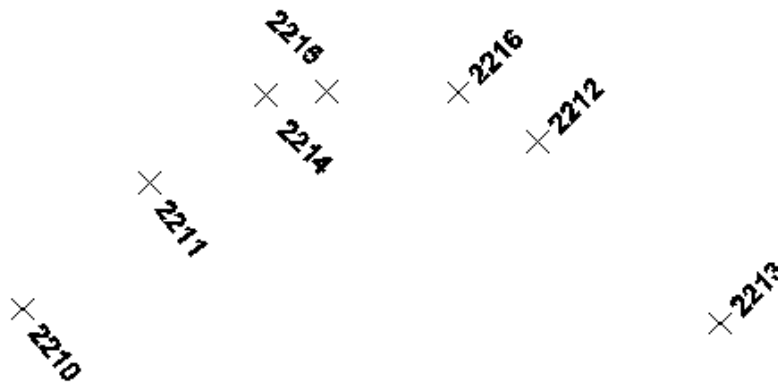
When done entering the tangent line points, press <Enter> when asked for the next point.

Next at the series of prompts:

Enter a point ID or pick a point symbol on the arc:

enter the points for the arc. Weights for these points are calculated by the program.

In the example shown in the figure, there are 2 points defining the first tangent, 3 points defining the arc and 2 points defining the tangent out.



Output for example shown in figure

Pt.: 2210 Wt.: 0 Offset: 0.000

Pt.: 2211 Wt.: 5 Offset: 0.000

N: 6512.07291 E: 8572.91824 Dir: N 45°00'00"E

N: 6567.12692 E: 8627.97226 Dir: S 45°00'00"W

Pt.: 2212 Wt.: 5 Offset: 0.000 LT

Pt.: 2213 Wt.: 1 Offset: 0.000 LT

N: 6585.04757 E: 8796.89599 Dir: S 45°00'49"E

N: 6505.88861 E: 8876.09255 Dir: N 45°00'49"W

Pt.: 2214 Offset: 1.736 OUT

Pt.: 2215 Offset: 3.232 IN

Pt.: 2216 Offset: 2.324 OUT

N: 6492.00101 E: 8721.39653 RAD: 119.183

The [Edit/Ok/Quit] <O>: prompt following the output is explained in the Line option.

If you are satisfied with the results press <Enter> for Ok.

The coordinates for the endpoints of the tangents and PC, PT and radius point of the curve are computed and saved to the coordinate file using the Saving Point dialog.

Prompts

Enter the type of best fit problem [Line/Arc/Tan-arc-tan] <L>: Type "L" and Enter or just Enter for a best fit line, "A" and Enter for a best fit arc or "T" and Enter for the best fit of a curve with two straight tangents in and out.

For a best fit line or arc:

Enter or Pick a C&G point on the line (or Arc): Type a point ID or pick a point symbol on the screen. Repeats until all points are entered and the user presses Enter at this prompt.

Enter weight for point <####> <1>: Enter a number between 0 and 15 (0 = infinite weight).

For best fit tan-arc-tan:

Enter a point ID or pick a point symbol on 1st tangent: Enter or pick as many points and weights as desired for the first tangent line.

Enter a point ID or pick a point symbol on the arc: Enter or pick as many points as desired for the arc (weights are determined by the program).

Enter a point ID or pick a point symbol on 2nd tangent: Enter or pick as many points and weights as desired for the second tangent line.

[Edit/Ok/Quit] <0>: Type "E" and Enter if you wish to change the weight of a point or add or delete points. Type "O" and Enter or just Enter to calculate the best fit line, arc or line-arc-line and store its defining points in the coordinate file. Type "Q" and Enter to quit without calculating the best fit points.

[Add/Change/Delete/eXit]: if you choose Edit then this prompt allows you to Add a point, Change a weight, or Delete a point. When done editing press "X" and Enter to return to the Edit/Ok/Quit prompt.

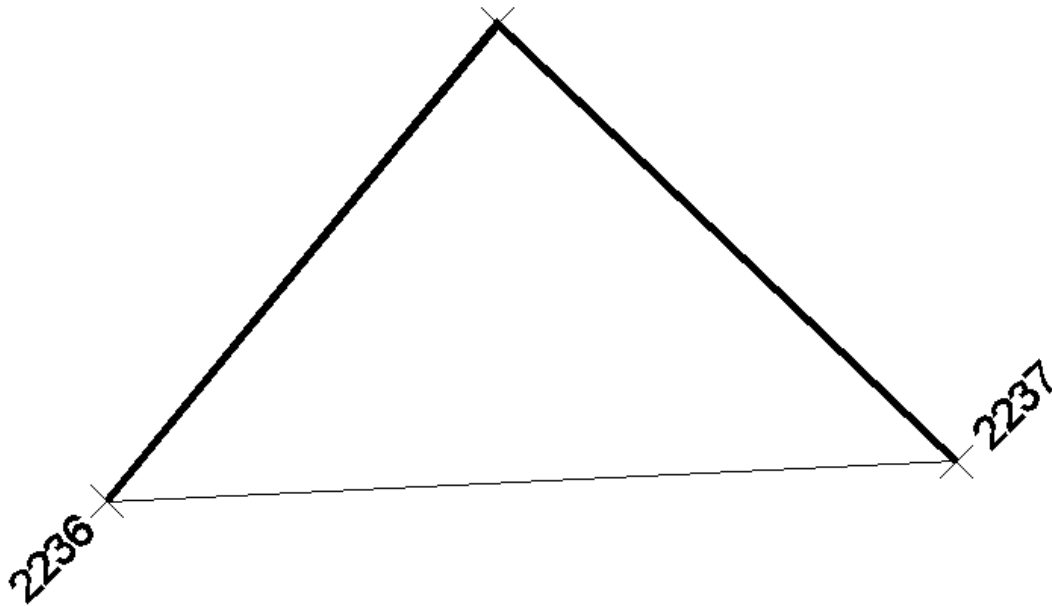
Pulldown Menu Location: CG-Survey > Cogo

Keyboard Command: cg_bfit

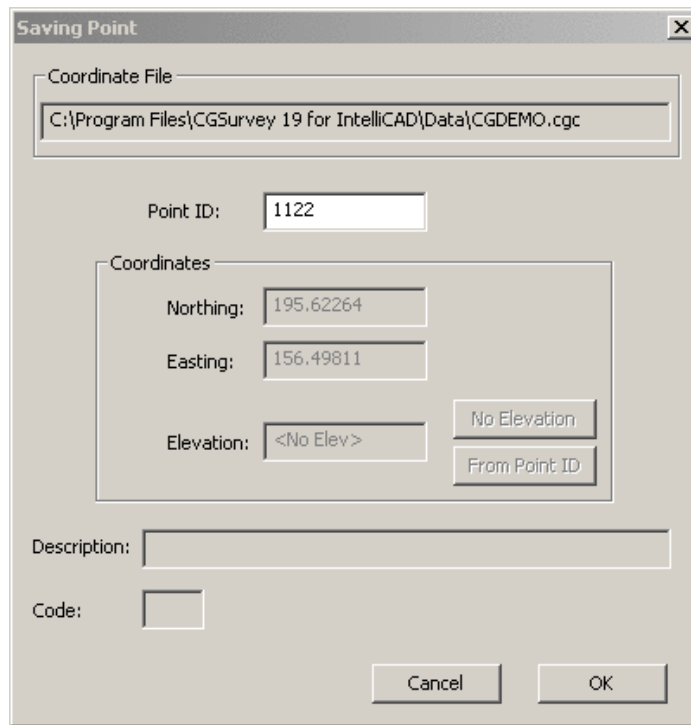
Prerequisite: Coordinate file.

Triangulation

This feature allows you to calculate the location of an unknown point given the angles at the 3 vertices of the triangle formed by the 2 known points and the unknown point.



Enter the point ID of the first known point then the point ID of the 2nd known point (the backsight) and the measured horizontal angle to the unknown point. Do the same for the 2nd known point backsighting the 1st known point. Next, if available, enter the angle between the 2 known points with the instrument at the unknown point. The standard deviation and other information for the calculation will be printed at the command line and written to the print file. The calculated point will be saved to the coordinate file using the Saving Point dialog (Shown Below).



Prompts

Enter first instrument point: Enter a point ID or pick a point symbol on the screen.

Enter first backsight point:: Enter a point ID or pick a point symbol on the screen.

Enter first horizontal angle to unknown point: Enter an angle.

Enter second instrument point: Enter a point ID or pick a point symbol on the screen.

Enter second backsight point: Enter a point ID or pick a point symbol on the screen.

Enter second horizontal angle to unknown point: Enter an angle.

Enter horizontal angle at unknown point or <skip> <0.00000>: Enter an angle if available or press Enter to skip.

Pulldown Menu Location: CG-Survey > Cogo

Keyboard Command: cg_triangulation

Prerequisite: coordinate file

NAD83

This feature allows you to convert longitude and latitude to and from NAD83 state plane coordinate systems.

NOTE: Do not use this function for any other coordinate system, i.e. NAD 1927. Make sure the correct state is selected on the General tab of the C&G Options dialog box.

After choosing the NAD83 menu item from the CGCogo menu you will be prompted for the necessary data.

At the Enter zone prompt enter the letter for the appropriate zone for the area where the survey was performed. The zones allowed may vary by state.

[Coords to longitude-latitude/Longitude-latitude to coordinates] <C>:

Coords to longitude-latitude

Pressed <Enter> (or type C and <Enter>)

you will be asked to select the points.

After selecting points a table of longitude-latitude and related data for the points will be printed at the command line

Longitude-latitude to coordinates

Type L and <Enter>.

You will be asked to enter the longitude and latitude of the points you wish to calculate

When you have entered the final longitude-latitude pair press <Enter> when asked for the next latitude.

The computed points will be stored in the coordinate file using the Saving Point dialog shown below.

The image shows a 'Saving Point' dialog box. It has a title bar with 'Saving Point' and a close button. The main area contains a 'Coordinate File' field with the path 'C:\Program Files\CGSurvey 19 for IntelliCAD\Data\CGDEMO.cgc'. Below this is a 'Point ID' field with the value '1122'. A 'Coordinates' section contains three input fields: 'Northing' with '195.62264', 'Easting' with '156.49811', and 'Elevation' with '<No Elev>'. To the right of the 'Elevation' field are two buttons: 'No Elevation' and 'From Point ID'. At the bottom of the dialog are two more input fields: 'Description' and 'Code'. At the very bottom are 'Cancel' and 'OK' buttons.

Click OK to save the point to the coordinate file.

Repeat until done or press <Enter> to end the command.

Prompts

Enter zone (E, W): enter the letter for the appropriate zone for the area where the survey was performed. The letters allowed will vary depending on the state.

[Coords_to_longitude_latitude/Longitude_latitude_to_coords] <C>: Type "C" and Enter or just Enter to calculate coordinates given longitude and latitude. Type "L" and Enter to do the reverse.

if you chose Coords to longitude-latitude:

Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt_group/Limits/Radius/Select]: Use the various selection methods to choose the points for which you wish to calculate longitude and latitude

if you chose Longitude-latitude to Coords:

Enter latitude [<Enter> when done]: Enter the latitude angle for a longitude - latitude pair.

Enter longitude: Enter the longitude angle for a longitude - latitude pair.

Pulldown Menu Location: CG-Survey > Cogo

Keyboard Command: cg_nad83

Prerequisite: coordinate file

CGDraw

Drawing Settings

See **CG Options...** menu item in the **Tools** menu.

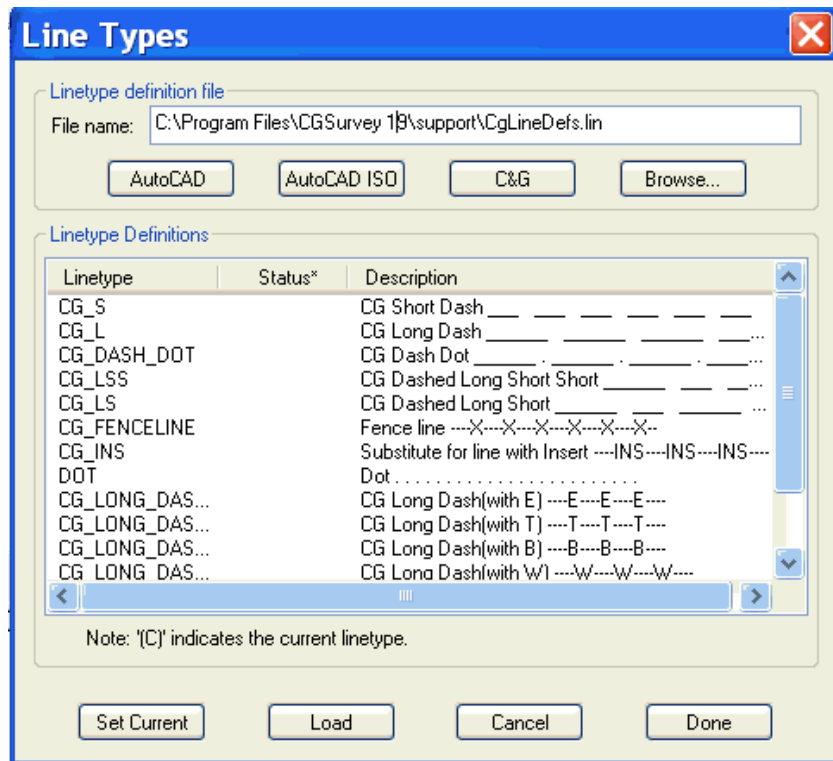
Pulldown Menu Location: CG-Survey > CGDraw>Drawing Settings

Keyboard Command: DSU, CG_DRAW_SETUP

Prerequisite: None

Set Line Type

To use a line type it must be loaded and it must be the current line type. The current line type will be used for any lines or polylines drawn. The Set Line Type feature allows you to load line types from any line type file (*.lin) and to specify the currently active line type. It gives you easy access to the most commonly use line type files while allowing you to access any line type file available to you.



Prompts

Clicking the Set Line Type menu item brings up the Line Types dialog: By default the dialog displays the acad.lin line type file contents (CgSu.lin in the standalone version of CGSurvey) .

By clicking the AutoCAD/IntelliCAD ISO button: you can view the acadiso.lin file line types (CgSu-iso.lin in the standalone version of CGSurvey).

By clicking the C&G button: you can view the custom line types created for CGSurvey (in CgLinedefs.lin). You may also use the Browse... button to view and load line types from other line type files.

To load a line type: pick the file the line type is in, highlight the desired line type, then click the Load button. Notice that the status column now indicates that the line type is "Loaded".

To make the highlighted line type current: click the Set Current button. The status column now reads "Loaded (C)", indicating that the line type is loaded and it is the currently active line type.

You can load a line type and make it current: by double-clicking it. If it is already loaded, double-clicking will make it current.

Cancel button: returns the current line type to what it was before the command was run.

Click the Done button: to close the dialog.

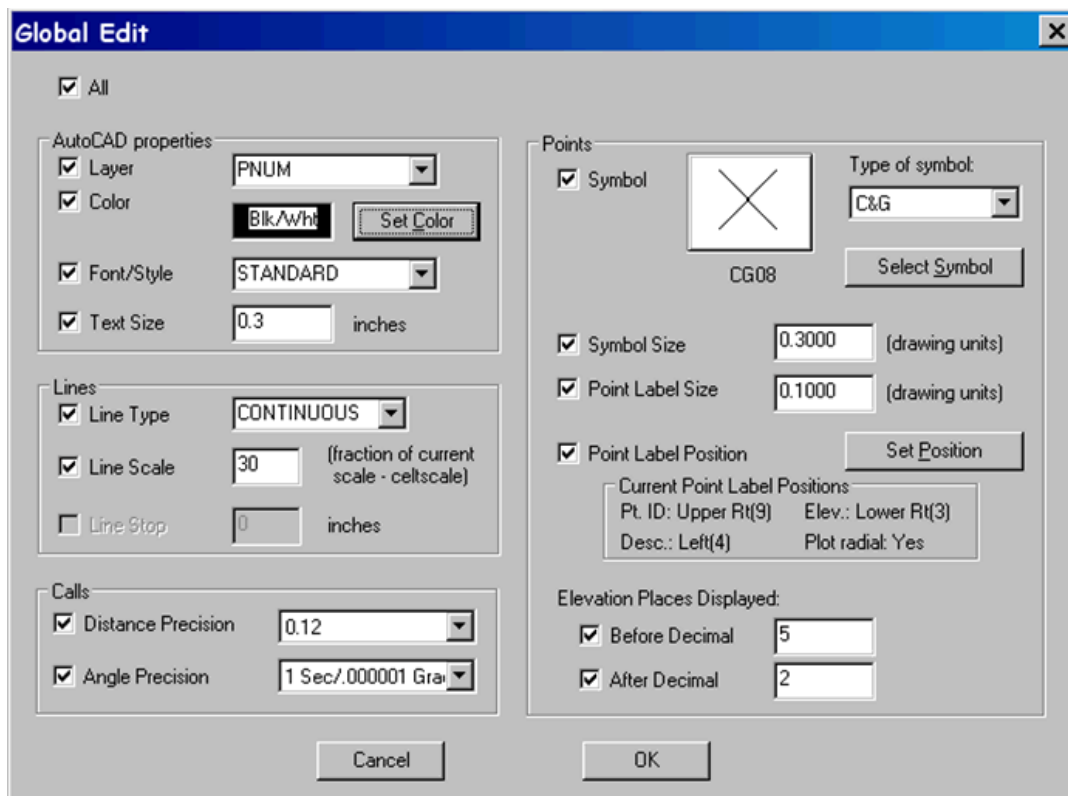
Pulldown Menu Location: CG-Survey > CGDraw>Set Line Type

Keyboard Command: SLT, CG_SET_LINE_TYPE

Prerequisite: None

Global Edit

Global Edit allows you to make several changes to one or more entities, in one operation.



Prompts

After selecting Global Edit from the CGDraw menu you will be asked to specify the method of entity selection at the command line:

Screen: This option allows you to use any of the standard mouse based CAD selection methods.

Points: Allows you to select C&G points using the standard C&G selection methods. Checking a given check box activates that section of the dialog box and allows you to make the desired changes.

Done: When finished selecting the entities enter "D" for done this will bring up the Dialog box.

There are five basic sections in this dialog.

All: checking this checkbox is just a fast way of checking all the checkboxes and thus allows you to edit all of the properties of the selected entities.

All of the editable items for the entities selected will have their checkboxes checked.

CAD properties

Layer: change the layer of the items selected

Color: Change the color of the items selected

Font/Style: Change the font style to a another existing font style.

Text Size: Change the current Text size (this setting is in inches)

Lines

Linetype: set the linetype for the lines selected. Pressing the down arrow will bring up a list of all of the available linetypes.

Line Scale: This allows you to set the length of the pattern.

Line Stop: This allows you to set the line stop. This item will only be activated if a C&G line was chosen.



Line stop is a C&G parameter that allows you to stop the line short of the point symbol plotted at the point location thus the line can be made to not go through the symbol. For example, if you were plotting 0.10 diameter circles for property corners, you could set the line stop to .10. This would cause a C&G line drawn to the property corner to end .05 plotted units short of the actual corner and thus not cross the property corner point symbol

Calls

Distance precision: From the pull down select the number of decimal places to be displayed.

Angle precision: From the pull down simple select the angle precision you need.

Points

This portion of the Global Edit dialog allows you to change various aspects of point symbols:

Symbol: from the pull down select the new symbol to use.

Symbol Size: set the symbol size (in inches or cm)

Point Label Size: Set the point label size (in inches or cm)

Point Label Position: Displays the point label configuration dialog box, set the options as needed.
Elevation Places Displayed: set the number of places to be displayed.

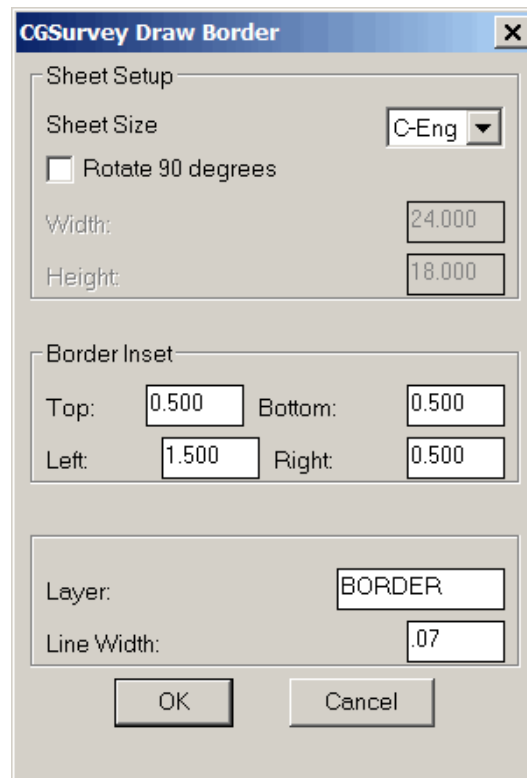
Pulldown Menu Location: CG-Survey > CGDraw>Global Edit

Keyboard Command: GE, CG_GLOBAL_EDIT

Prerequisite: Coordinate file

Border

This option allows you to place a border Polyline on your drawing with sheet sizes.



Prompts

Sheet Setup

Sheet Size: The letters A, B, C, etc. refer to ANSI sheet size standards. You also have the option of creating and naming custom sizes.

Rotate 90 Degrees: when checked will rotate the border 90 degrees.

Border Inset: specifies the inset distance for the border. Keep in mind this inset distance is measured from the edge of the plotable area of your plotter. Check the plotter manual for plotter specifications.

Layer: The layer the border will be drawn on.

Line Width: The thickness of the border line in inches (cm).

Press OK button when done and the border polyline will be drawn at the mouse cursor. You can move it to the correct location and left click to place it there.

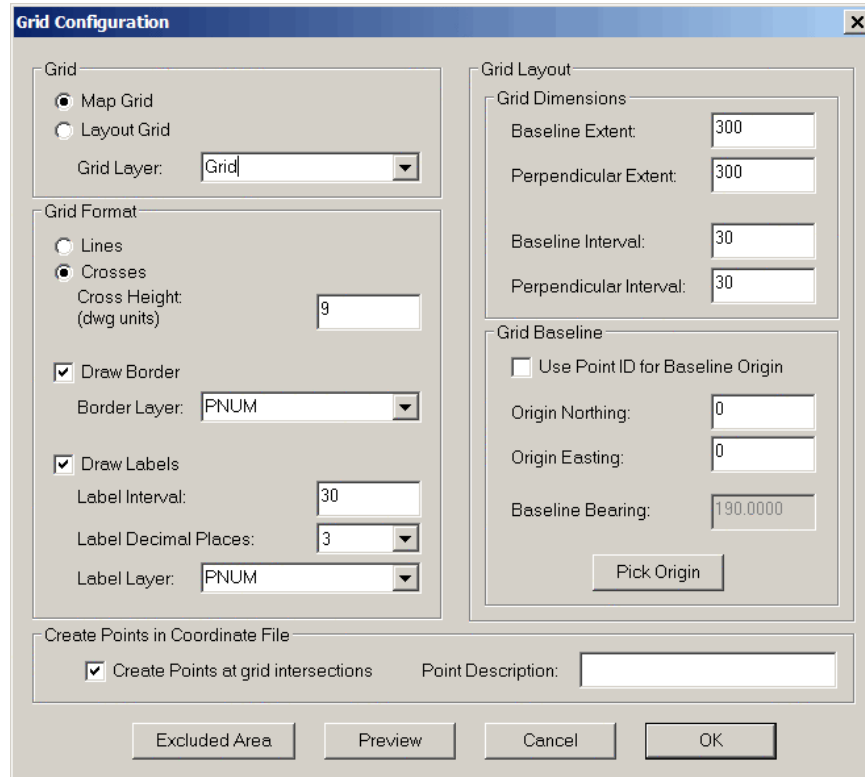
Pulldown Menu Location: CG-Survey >CGDraw>Border

Keyboard Command: DB, CG_DRAW_BORDER

Prerequisite: None

Coordinate Grid

Choosing the Coordinate Grid item in the CGDraw menu brings up the Grid Configuration dialog. The various areas of the dialog are described below:



Map Grid: is used to provide a visual reference grid to show northings and eastings on a map. A Map Grid can be labeled along its border to show the coordinate values of the grid lines. The Map Grid is oriented North-South East-West whereas a Layout Grid can be oriented at any specified bearing.

Layout Grid: is meant to be used to create points on a regular grid for laying out building columns, a topo grid, etc. A Layout Grid does not allow for a border nor for coordinate labels along the border.

Grid Layer: specify the layer the grid is to be created on. The layer does not need to exist prior to running this command.

Lines: If selected grid lines will be drawn for the full height and width of the grid dimensions.

Crosses: Only crosses will be drawn at the grid intersections for the full height and width of the grid dimensions.

Cross Height (drawing units): this defines the size of the crosses in drawing units. If your drawing scale is 40 feet and you wish to have crosses that are 0.25 inches when plotted, you must specify cross height as 10 feet.

Draw Border: If checked, a border will be drawn around the perimeter of the defined grid. You can choose a different layer for the border if you wish. This will allow you to set the color, line thickness and/or line type for the border (this option is not available for Layout Grid).

Draw Labels: Label the grid lines or crosses around the perimeter at the same interval as the Baseline and Perpendicular intervals (this option is not available for Layout Grid). If checked you must specify:

Label Interval: This number must be some even multiple of both the baseline and perpendicular intervals. The Label Interval CAN NOT be less than the base or perpendicular interval settings.

Label Decimal Places: Specify the number of decimal places used for the label text.

Label Layer: Specify the layer the labels are to be drawn on.

Grid Dimensions: Baseline Extent: This is the total width of the grid, East to West (or parallel to the baseline bearing in the case of a Layout Grid).

Perpendicular Extent: This is the total height of the grid perpendicular to the baseline.

Baseline Interval: This is the distance between the grid lines (or X's) drawn perpendicular to the baseline

Perpendicular Interval: This is the distance between the grid lines (or X's) drawn parallel to the baseline.

Grid Baseline:

Use Point ID for Baseline Origin: checking this box allows you to use an existing C&G coordinate for the **Grid Origin**. This is typically used for a Layout Grid.

Enter the point ID, or select the point from the screen.

Origin Northing/Origin Easting: manually enter the Northing and Easting value for the grid origin or pick it on the screen using the Pick Origin button.

Baseline Bearing: This is only used when you are drawing a Layout Grid. This is the bearing of the baseline. Use the standard C&G bearing input format qdd.mmss (e.g. 125.3527 for N25°35'27"E or 325.5405 for S25E54'04"W)

Pick Origin button: This option allows you to pick the origin graphically on the screen. You do not have to pick a C&G point.

Create Points at Grid Intersections:

Checking this box will cause the default C&G point to be plotted at each grid point or grid line intersection and a corresponding point to be stored in the currently open coordinate file. This is especially useful when creating a Layout Grid.

Point Description: enter a point description for the points saved to the coordinate file.

Exclude Area: This button allows you to graphically specify a horizontal window within which no grid is to be drawn. This can be used to guarantee that a title block, legend or other area is not obscured by the grid or its labels.

Preview: This button allows you to preview the grid as specified. Pressing <Enter> will return you to the Grid Configuration dialog allowing you to make changes if necessary.

Cancel : This button exits the command without drawing the grid.

OK: This button causes the grid to be drawn.

Pulldown Menu Location: CG-Survey > CGDraw>Coordinate Grid

Keyboard Command: GRD, CG_DRAW_GRID

Prerequisite: Coordinate file

Text on Arc

Create

Text on arc allows you to create text that follows an arc specified by you. Each word in the text is a separate block and can be moved later as needed.

Prompts

You will be prompted to Enter Text to place on arc:



Type the desired text then press <Enter>:

Enter center point for arc: using the mouse, select the center point of the arc the text is to follow, this can be a C&G point, or any point in the drawing. You need not actually have an arc drawn.

Enter Midpoint of text: select the midpoint of the text, this can be a C&G point, or any point in the drawing.

Move to desired location: The text will be drawn at the cursor. Move the cursor to the desired location and left click to place the text.

Pulldown Menu Location: CG-Survey > CGDraw>Text on Arc>Create

Keyboard Command: TOA, CG_TOA

Prerequisite: Coordinate file

Move

Allows you to move all the text associated with the selected text-on-arc entity.

Prompts

Select entities: select text

Entities in set: 1 item is found and selected

Select entities: press <Enter> to accept entry

Move Text to desired location: move text

Select entities: repeating the selection set

Press <ESC> or <Enter>: to end command

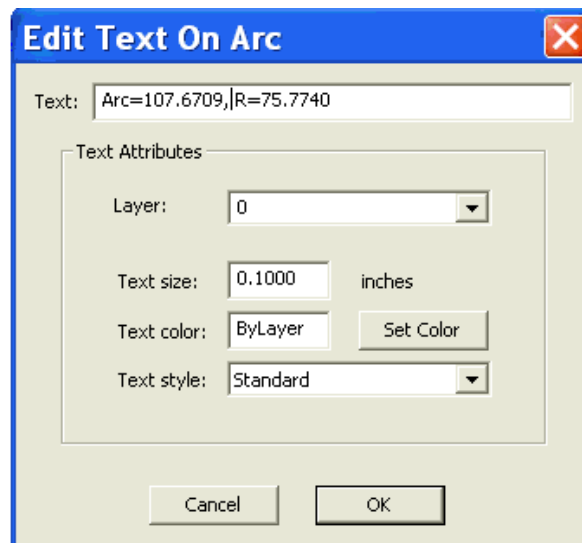
Pulldown Menu Location: CG-Survey > CGDraw>Text on Arc>Move

Keyboard Command: MTA, CG_MOVE_TOA

Prerequisite: Coordinate file

Edit

Allows you to edit the text associated with a text-on-arc entity.



Text: Arc=107.6709, R=75.7740 (Edit Text)

Text Attributes

Layer: 0 (current)

Text Size: 0.100 inches (Default)

Text color: Bylayer

Set Color Button: select a new color

Text style: Standard text style

Prompts

Select entities: select text

Entities in set: 1 item found and selected

Select entities: repeat selection set

Press <ESC> or <Enter >: to end command

Pulldown Menu Location: CG-Survey > CGDraw>Text on Arc>Edit

Keyboard Command: ETA, CG_EDIT_TOA

Prerequisite: Coordinate file

Delete

Allows you to delete all the text associated with a text-on-arc entity.

Prompts

Select entities: select text

Entities in set: 1 item found and selected

Select entities: repeat selection set

Press <Esc> or <Enter>: to end command

Pulldown Menu Location: CG-Survey > CGDraw>Text on Arc>Delete

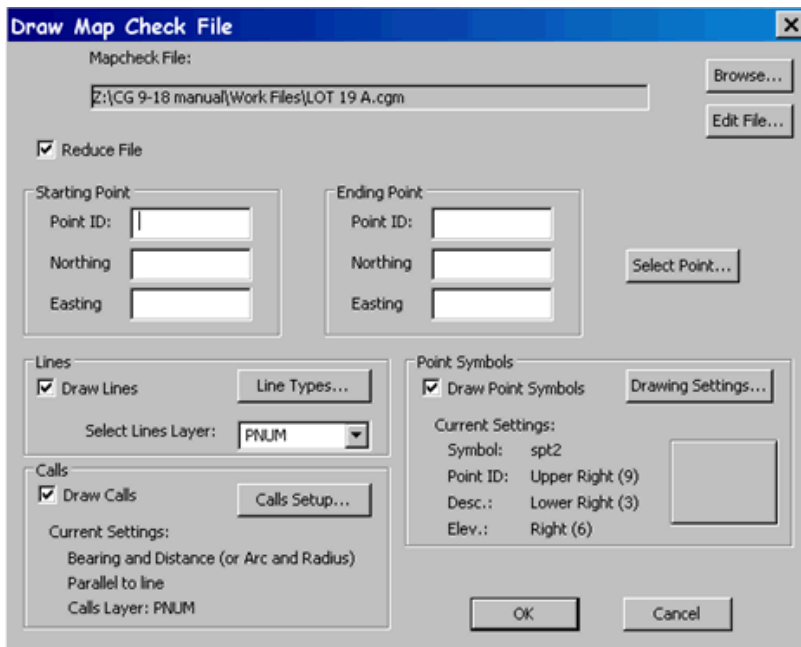
Keyboard Command: DTA, CG_DELETE_TOA

Prerequisite: Coordinate file

Draw Mapcheck

This routine will draw a mapcheck file. The settings allow you to plot points, draw lines, and place calls all at the same time.

If a mapcheck file is NOT currently open when you open the map check file routine a dialog box will prompt you to open a mapcheck file. Once a mapcheck file is open the following dialog box will open.

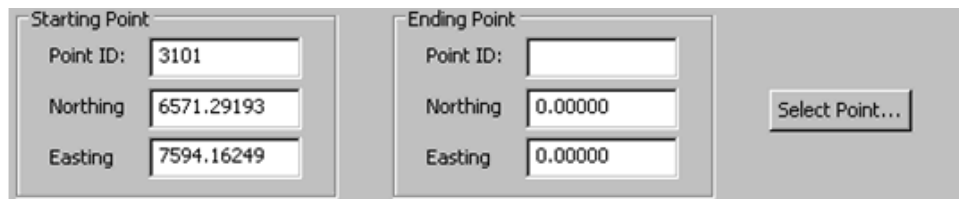


If the mapcheck file displayed at the top of the dialog box is not the file you want to draw you can use the browse button to search for another file.

You also have the option to edit the file which will take you to the mapcheck editor (CGEditor).

Reduce File: If this item is checked there will be a closure report written to the print file and the command line.

Starting/Ending Points: You have the option of entering an existing point number or manually entering the northing and easting. If you enter an existing point number the northing and easting values will be read from the coordinate file and placed in the appropriate edit boxes (see below).



If the starting and ending point are the same point, you need only enter the starting point values.

Clicking the **Select Point** button will cause the **Draw Mapcheck** dialog to be hidden thus allowing you to pick the starting and ending points from the screen or use the command prompt:

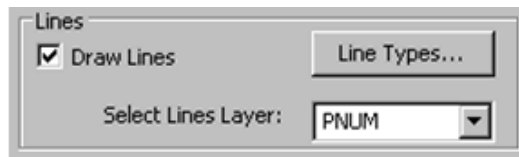
Select starting point for mapcheck:

or

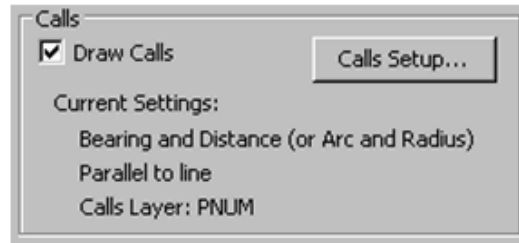
Select ending point for mapcheck:

rather than manually entering the point number or coordinate values in the edit boxes.

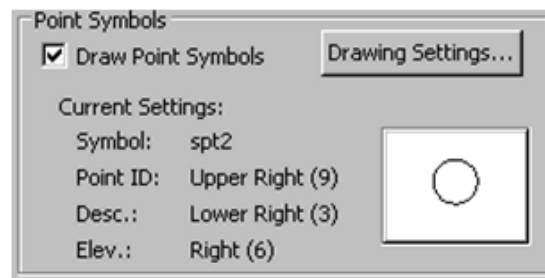
Lines: This portion of the dialog allows you to turn on or off the draw line command as well as select the linetype and layer where the line will be drawn



Calls: This section of the dialog allows you to turn on or off the draw calls command as well as edit the call setup options.



Point Symbols: This area of the dialog allows you to turn on or off the draw point symbols command as well as having a button that will take you to the drawing settings dialog. At the drawing settings dialog you can change the symbol, symbol size, label options and more.



OK: Selecting ok will cause the mapcheck file to be drawn, based on the current settings as described above.

Prompts

Select starting point for mapcheck: Select a point symbol or type a point number for the starting point.
or

Select ending point for mapcheck: Select a point symbol or type a point number for the ending point.

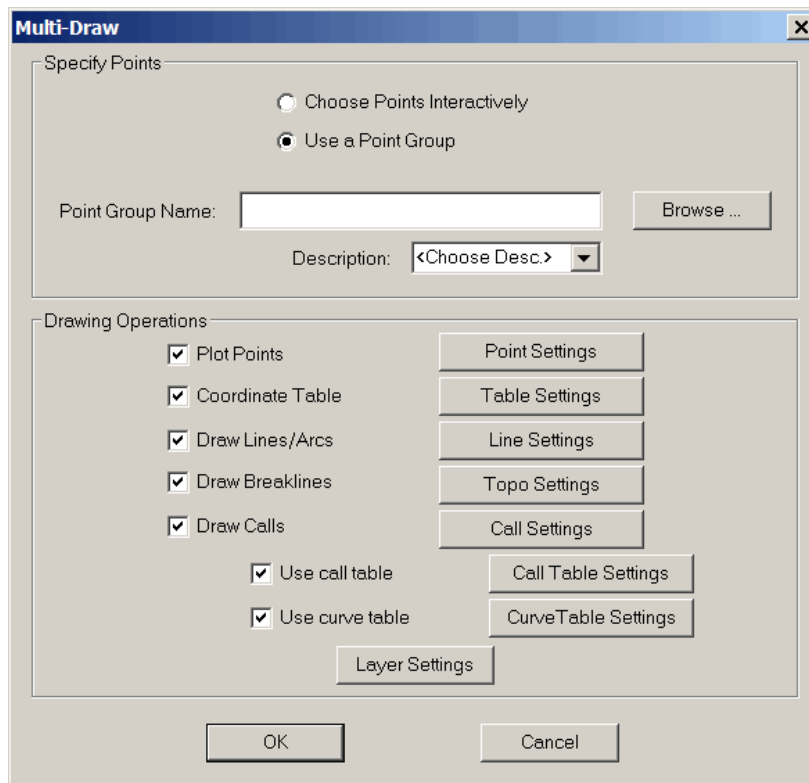
Pull-down Menu Location: CG-Survey > CGDraw>Draw Mapcheck

Keyboard Command: DMP, CG_DRAW_MAPCHECK

Prerequisite: Open Mapcheck file *.cgm

Multi-Draw

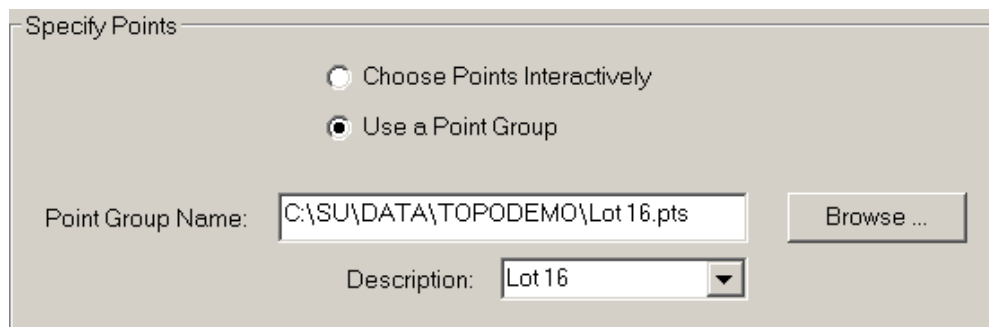
This feature allows you to complete several drawing operations at the same time. For example, in one operation you can plot points and generate a coordinate table for the points at the same time. Or you can draw lines by points and place calls on the resulting lines at the same time.



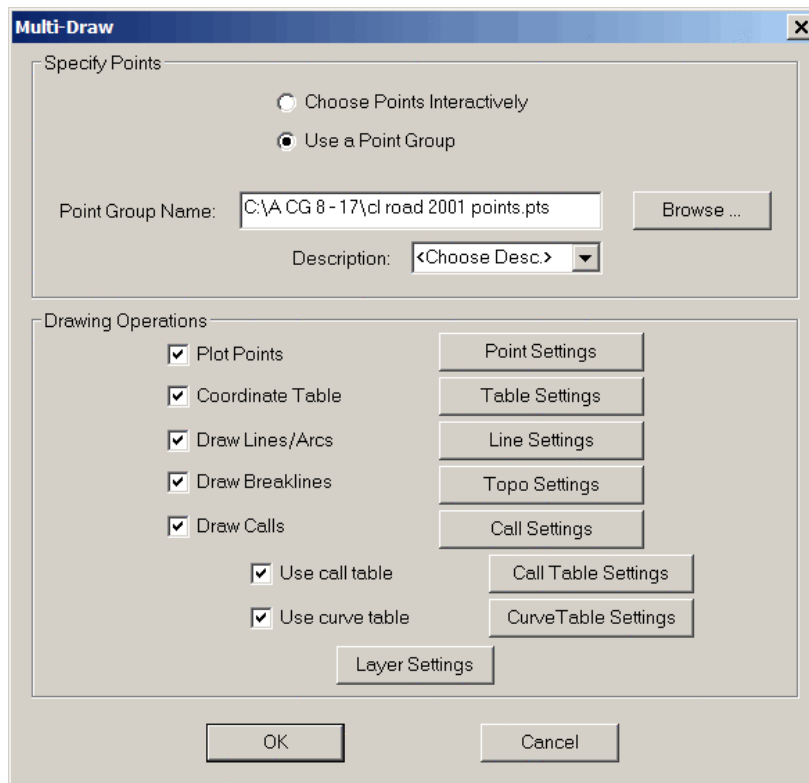
Prompts

When you choose Multi-Draw from the CGDraw menu, the Multi-Draw dialog box is displayed.

Specify Points



Use the radio buttons to indicate how you want to specify the points that will be used for the drawing operations. You can Choose Points Interactively using the standard C&G point sequence command line interface or you can Use Point Group.

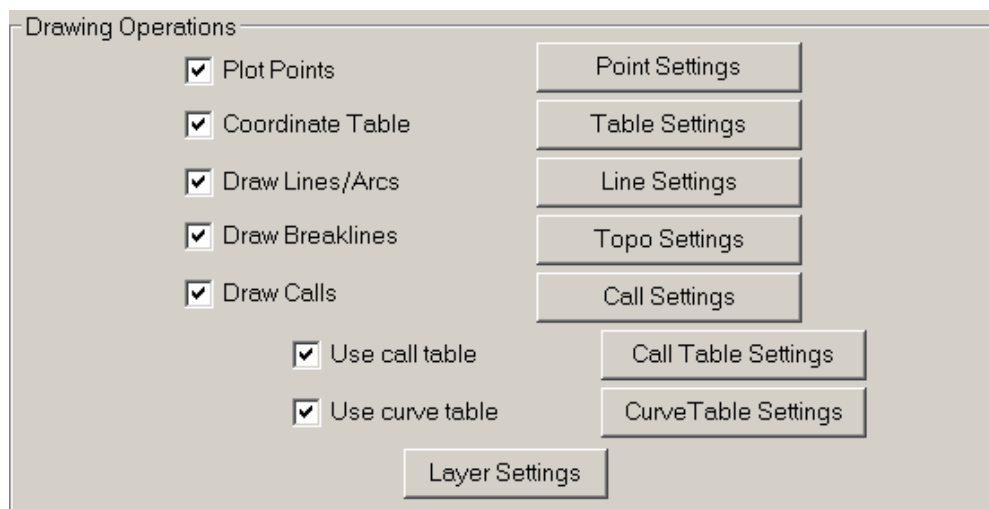


NOTE: If you Choose Points Interactively, you will not have the option of plotting points or creating a coordinate table.

If you click Use Point Group, you must specify a Point Group Name and the point group description for the points you wish to use. You can either type the full path (including the drive letter) to an existing point group file or you can click the Browse button and use the file dialog box to specify the Point Group file.

NOTE: In this context, Description refers to the Point Group description, NOT the description for the individual points as found in the coordinate file.

Drawing Operations



In this area of the dialog box you must specify which drawing operations to perform and, if needed, make the necessary settings changes required for the drawing operations. Any settings changes are saved to the drawing and thus remain in effect after this command is completed.

Plot Points: Checking this checkbox will cause the selected points to be plotted.

Point Settings button: selecting this button will bring up the Drawing Settings dialog box allowing you to make changes to the drawing settings.

Coordinate Table: Checking this checkbox causes a coordinate table to be created for the points selected.

Table Settings button: will bring up the Coordinate Table Settings dialog box allowing you to make changes to layer, text size and line spacing.

Draw Lines/Arcs: Checking this checkbox causes lines and or/arcs to be drawn between the points specified.

Line Settings button: Will bring up the Linetype Manager allowing you to select the linetype.

Draw Breaklines: Checking this checkbox causes breaklines to be drawn between the specified points. Breaklines are for use in topographic operations (see CGTopo).

Topo Settings button: displays the CGSurvey Auto Contouring settings dialog box allowing you to make changes to the current topo settings.

Draw Calls: When Draw Calls is checked, calls will be drawn between the points specified.

Call Settings button: will display the Call Settings dialog box allowing you to make changes.

If Use Call Table is checked: all call information will be placed in a call table, rather than along the line work. The Call Table Settings button displays the Call Table Settings dialog and allows you to set the layer calls will be placed on, the text size, the line spacing, the first course label, etc.

If Use Curve Table is checked: all curve information will be placed in a curve table, rather than along the line work. The Curve Table Settings button displays the Curve Table Settings dialog and allows you to specify: which components will be shown in the curve table, which layer the table will go on, the text size, the line spacing, the first course label, etc.

The Layer Settings button: displays the Layer Manager dialog. Here you can create any layers you need or set the current layer as required. When you are satisfied click OK.

If you choose Use Point Group: the drawing operations you have selected will be completed immediately.

If you have selected Choose Points Interactively: enter or pick the desired points at the following standard C&G point sequence command line prompt:

Note: When entering the points at the command line, some drawing operations may occur as you enter the points.

Placement of coordinate, call and or curve tables

No matter what point selection method you use, if you specified that any tables be drawn, the coordinate, call or curve table will be placed at the cursor and you will be asked to move the cursor to the desired location. Clicking the left mouse button will place the table in the drawing at the cursor location.

Pulldown Menu Location: CG-Survey > CGDraw>Mult-Draw

Keyboard Command: MD, CG_MULTIDRAW

Prerequisite: Coordinate File and/or Point Group

Plot Points and Symbols

Plot Points on Screen

Plot Points on Screen: This feature plots the selected points from a coordinate file on the screen.

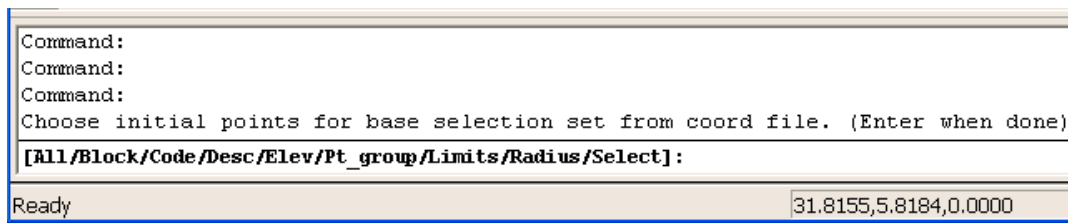
Prompts

If a coordinate file is not open, a file dialog box will appear allowing you to open one.

You will then be asked to select the points to be plotted (for additional information, see Getting Started: Coordinate point selection sets).

Type the Capped Letter to initialize the selection Set

[All/Block/Code/Desc/Elev/Pt_group/Limits/radius/select]: A



Press <Enter>3x when done: The points will be plotted on the screen.

In which layer will the points be plotted?

If the Use Description Table for point plotting parameters checkbox is not checked in the Graphic Options tab in C&G Options dialog: then all points will be plotted on the current layer according to the Drawing Settings dialog.

If the Use Description Table for point plotting parameters checkbox is checked and the Default layer for codes or descriptions not found in description table is specified: point's will be plotted to the layers specified by the description table (for a discussion of description tables see the CGMngmt chapter).

Description matches a description found in the description table: the point and its labels will be plotted as specified in the description table. For a description to match it must be a whole word match, disregarding numbers.

For example:

Table Description Point Description Match

TC TC-.5 to Bc yes

SW SW1 yes

FH TOPFH no

If a point has several different descriptions found in the description table: then that point can be plotted in more than one layer. For example: If the point's description is TC WV, it will be plotted in the layer assigned to the description TC as well as the layer assigned to the description WV.

No match is found in description table: the point will be plotted in the default layer with point labels as specified for the active point symbol.

Pulldown Menu Location: CG-Survey > CGDraw>Plot Points and Symbols>Plot Points on Screen

Keyboard Command: PP, CG_PLOT_POINTS

Prerequisite: Coordinate file

Remove Points from Screen

This feature allows you to remove/erase specified points from the drawing.

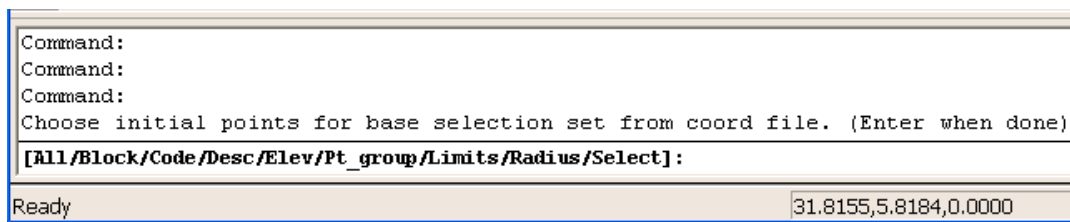
Prompts

If a coordinate file is not open, a file dialog box will appear and allow you to open an existing coordinate file.

Using the standard C&G Select Points commands, select the points to be removed.

Type the Capped Letter to initialize the selection Set

[All/Block/Code/Desc/Elev/Pt_group/Limits/radius/select]: A



Press <Enter> 3X when done: The points will be removed from the screen.

Note: Remove Points from Screen does NOT delete points from the coordinate file.

Pulldown Menu Location: CG-Survey > CGDraw>Plot Points and Symbols

Keyboard Command: RP, CG_REMOVE_POINTS

Prerequisite: Coordinate file

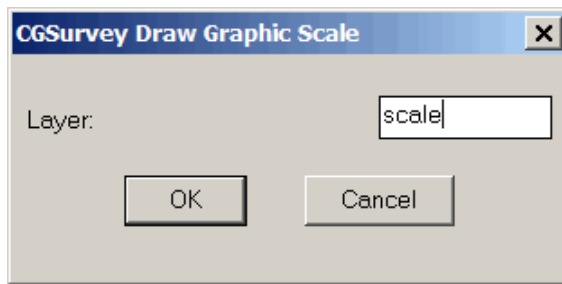
Graphic Scale

This feature allows you to draw a graphic scale. Make sure the correct scale has been specified in the Drawing Settings dialog box.

Prompts

In the CGSurvey Draw Graphic Scale dialog specify the layer for the graphic scale.

Layer: Scale



Once you have specified the layer: press the OK button.

The Graphic Scale symbol will then be drawn at the mouse cursor: You can move the cursor to position the graphic scale then press the left mouse button to place it at the cursor location.

Pulldown Menu Location: CG-Survey > CGDraw>Plot Points and symbols>Graphic Scale

Keyboard Command: GSC, CG_DRAW_GSCALE

Prerequisite: Coordinate file

Lines and Polylines

Lines by Point Number

This feature allows you to draw lines and/or arcs based on the points in the coordinate file.

Prompts

If you choose this option and a coordinate file is not open, you will be prompted to open one.

You will be prompted at the command line to enter the coordinate point IDs that define a line/arc or a series of lines/arcs. You may type the point IDs at the command line or pick points on the screen.

Enter Point Sequence

(point group/Reset/sNap on) (last point = <72>):

Once a point is selected the command line will change to a new group of options

(cLockwise curve/ccW curve/Point group/Reset/sNap on) (last point = <72>):

Once finished plotting points simply hit<Enter> and the command line will clear.

Point Input: When entering a point sequence specifying a line/arc the following input forms are acceptable:

34: Either specifies the starting point of a line or, if this is a continuing series of lines, draw a line from the previous end point to point 34 and occupy point 34.

6-9: Draw a line from point 6 to 9 and occupy point 9.

-4 : Draw a line from the previous end point to point 4 but remain at previous end point.

L: Specify a cLockwise curve. The previous end point is assumed to be the PC of the curve. The next point specified is the center or radius point of the curve and the next point entered is the PT of the curve.

W: Specify a counter clockWise curve. The data entry sequence is similar to a clockwise curve.

P: Use a Point group to specify the lines/arcs. You will be asked to pick the point group file using a file dialog box.

R: Reset the "last point" to none

N: Toggles the CAD snap command on or off

If you choose to use a point group file, lines will be drawn from point to point in the order specified in the point group file (see the CGMngmt chapter for information on point group files).

If you do not want to type in the point IDs to define the lines/arcs, you can select the points from points that are plotted on the screen using your mouse. (see Plot Points and Symbols).

Note: To specify an arc to be drawn pick the PC point of the curve then enter either **L** <Enter> for clockwise or **W** <Enter> for counter clockwise curve. When prompted, pick the radius point and then pick the PT.

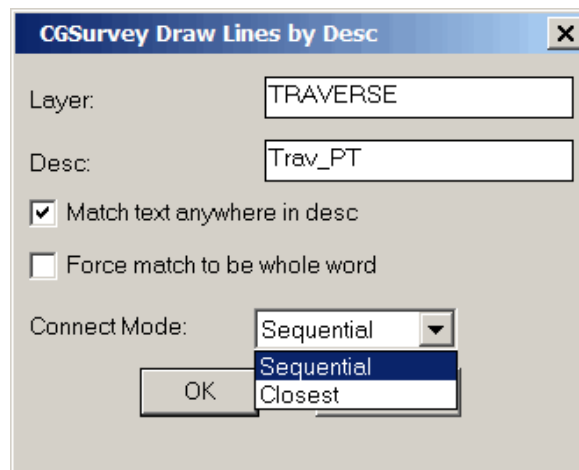
Pulldown Menu Location: CG-Survey > CGDraw>Lines and Polylines>Lines by Point Number

Keyboard Command: LBP, CG_LBP

Prerequisite: Coordinate file

Lines by Description

This feature allows you to connect lines between all the points in a coordinate file having a common description.



Prompts

After choosing the Lines by Description menu item you will see the CGSurvey Draw Lines by Desc dialog.

Layer: Specify the name of the layer the lines are to be drawn on.

Desc: Specify the description of the points you want to connect. Case is ignored and, unless the checkboxes described below are checked, only the leading characters of the point description are considered for a match.

Match text anywhere in desc: If this checkbox is checked, the entire point description field will be searched for the characters specified in the Desc: edit box. For example:

Input Description Point Description Match
MH SanMH Yes

Force match to be whole word: If this box is selected, the match must be a complete word in the point description, not just a portion of a word. For example:

Input Description Point Description Match
RD CL RD Yes (whole word)

RD CLRD No (not whole word)

Connect Mode

Sequential: Connects the points in point ID order.

Closest: Ignores point ID and connects to the closest point with named description.

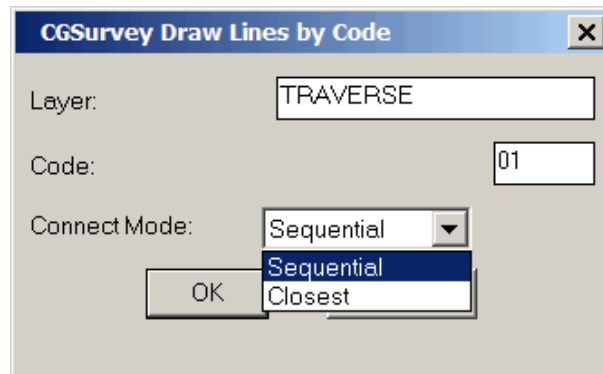
Pulldown Menu Location: CG-Survey >CGDraw>Lines and Polylines>Lines by Description

Keyboard Command: LBD, CG_LBD

Prerequisite: Coordinate file

Lines by Codes

This feature allows you to draw lines between the points in the coordinate file having a common point code. The point code is a two to four character field (depending on the type of coordinate file).



Prompts

Choosing the Lines by Code menu item brings up the CGSurvey Draw Lines by Code dialog.

With the exception of the Code: field, the items in this dialog are identical to those in the CGSurvey Draw Lines by Desc dialog.

Code: field specifies the code for the points you want to connect. Case is ignored.

Connect Mode

Sequential: connects line in point ID order

Closest: connect lines in Closet point with named description

Pulldown Menu Location: CG-Survey > CGDraw>Lines and Polylines

Keyboard Command: LBC, CG_LBC

Prerequisite: Coordinate file

Polylines by Point

This feature works very similar to the Lines by Point feature described in the previous pages. In Polyline by Points data entry is similar to Lines by Point except Reset does not apply to a polyline. The C&G Polyline allows you to treat road centerlines and other similar things that would normally be made up of several line segments, as one entity. You can use a C&G polyline to create a point group or you can place calls along it. You could also use a C&G polyline as the bounding polygon in the Fit Structure feature.

Pulldown Menu Location: CG-Survey > CGDraw>Lines and Polylines>Polylines by Point

Fit Polylines

This feature allows you to use a variety of best fit methods to smooth an existing polyline.

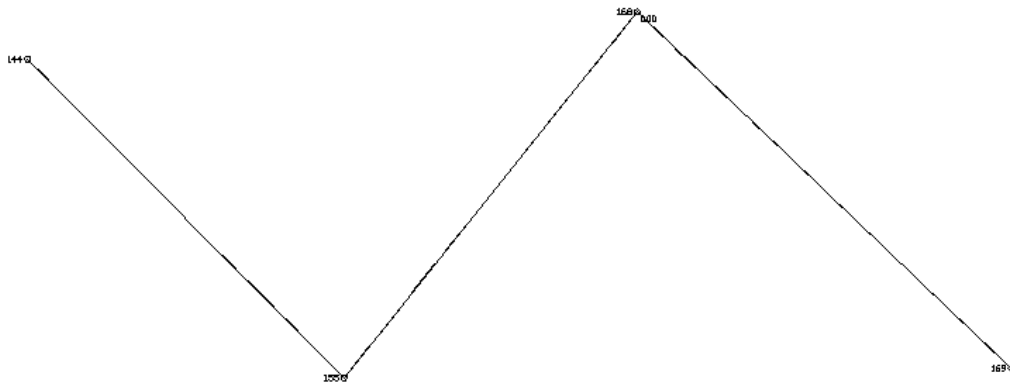
Prompts

First you must choose the method to use in fitting the selected polylines at the following prompt:

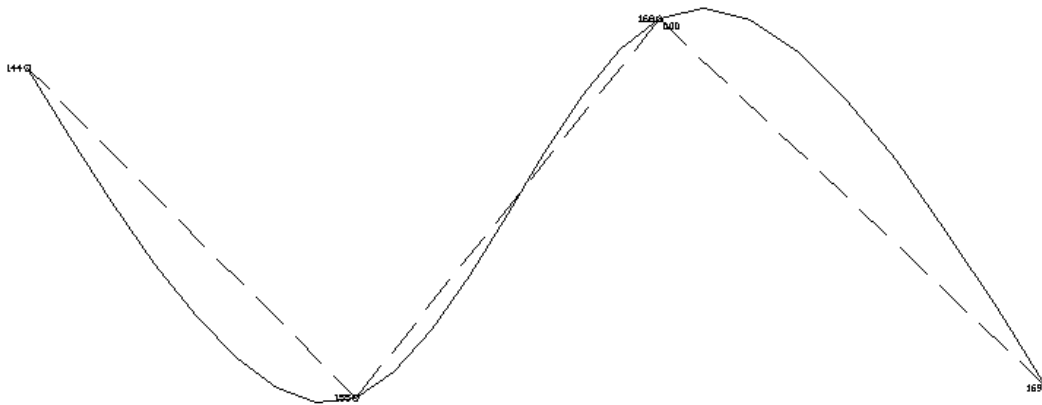
Type of fit to apply: [Decurve/Fit/Quadratic_spline/cUbic_spline/Cg_spline]<C>:

Next, select the polylines you wish to fit then press the <Enter> key or right mouse button to apply the fit to the selected polylines.

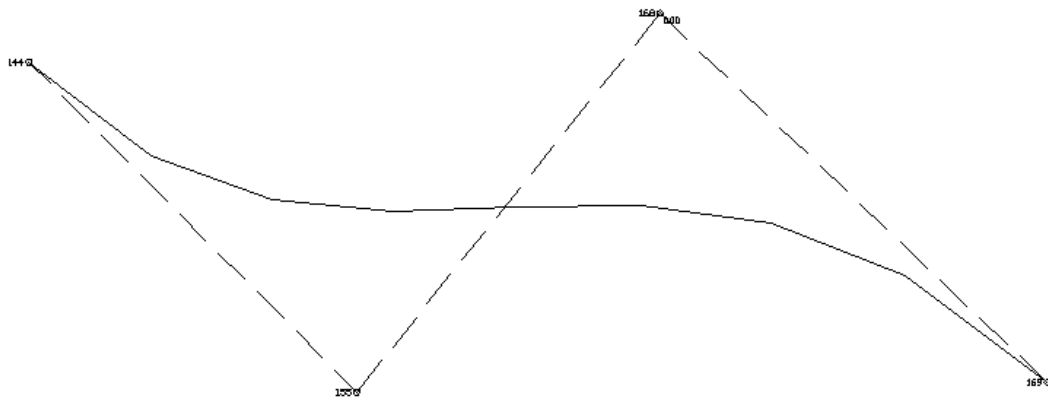
Decurve: This will decurve a previously smoothed polyline.



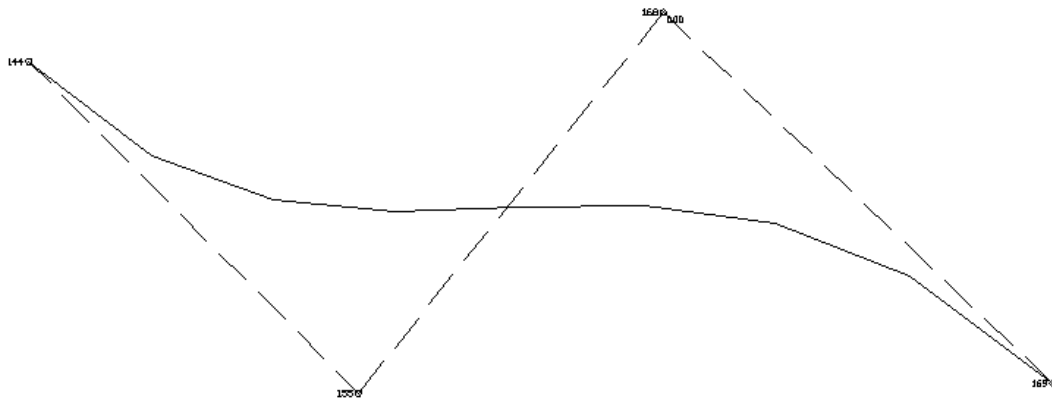
Fit: uses CAD fit - a series of interconnected circular arcs.



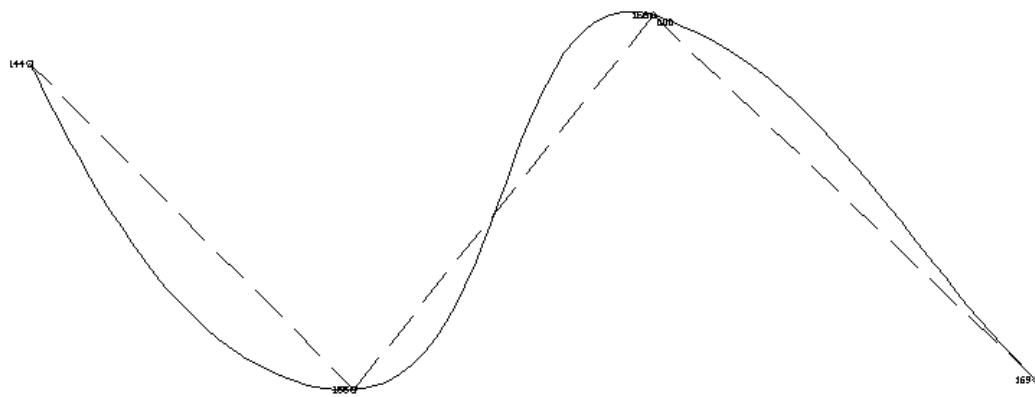
Quadratic spline: Uses a quadratic spline curve fitting algorithm.



Cubic spline: Uses a cubic spline curve fitting algorithm.



C&G Spline: Creates a smooth curve that passes through all vertices.



Pulldown Menu Location: CG-Survey >CGDraw>Lines and Polylines>Fit Polyline

Keyboard Command: FITP, CG_FIT_POLY

Prerequisite: Coordinate file

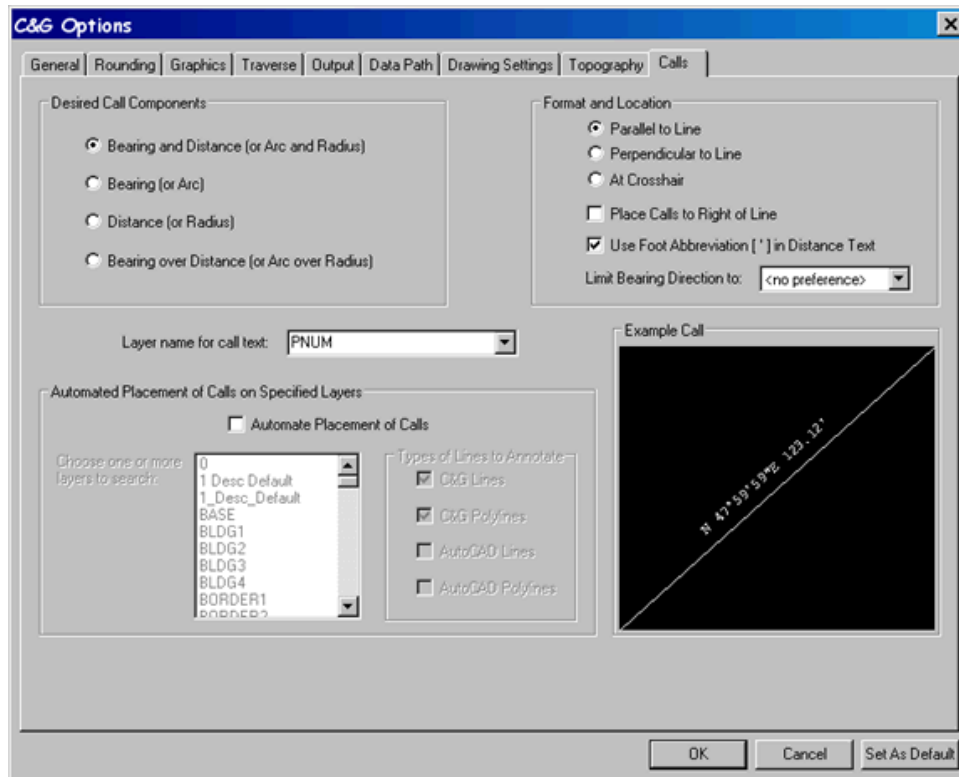
Calls

Place Calls

This feature allows you to annotate C&G and CAD lines, arcs and polylines.

Call Setup

Selecting Calls Setup will bring up the Call settings dialog box.



Desired Call Components: Specify the desired components for the call.

Bearing and Distance (or Arc and Radius)

Bearing (Arc)

Distance (Radius)

Bearing over Distance (Or Arc over Radius)

If you specify points to form a curve then the components shown in parentheses will be used to form the call text.

Format and location: Specify how you want the call placed relative to the line or arc:

Parallel: to the line or Arc

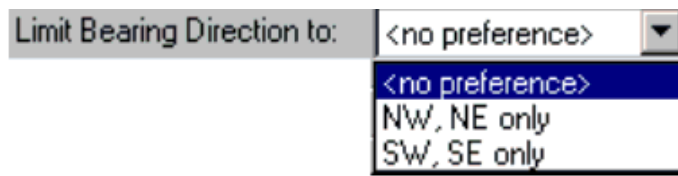
Perpendicular: to the line or arc

At Cursor: means the call text will be drawn horizontally at the cursor and you must move it to the desired location then left click to place it.

Place Call to Right of Line: If you are placing a call either parallel or perpendicular to a line or arc, select this box if you want the call placed to the right of the line or arc, assuming you are standing on the line and facing in the direction of the bearing. The call will be centered along the line or arc.

Use the Foot Abbreviation ['] in Distance Text: Checking this box will place the ['] mark after the distance (125.36'). Un-checking the box will remove the ['] mark (125.36).

Line Bearing Direction to:



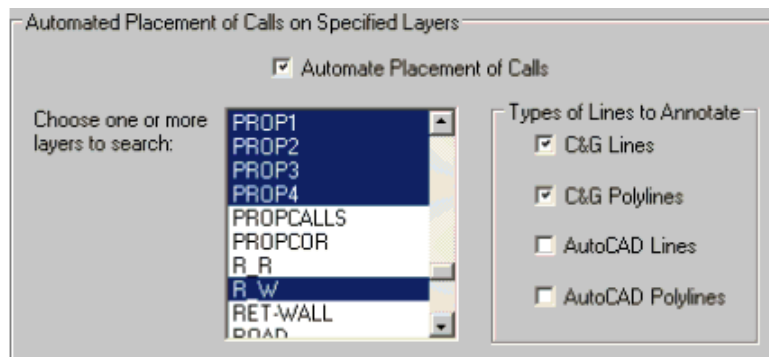
Selecting NW,NE: will force all calls to be shown only with NE and NW notation (N 428 35' 12" E or N 168 25' 31" W)

Selecting SW,SE :will force all calls to be shown only with NE and NW notation (S 428 35' 12" E or S 168 25' 31" W)

If < no preference >: is selected the software will define the bearing based on the direction of the points selected.

Layer Name for Call Text: Specify the layer where you want the calls placed.

Automated Placement of Calls on Specified Layers



Check the Automate Placement of Calls check box making the options in the dialog active. This routine allows you to select one or more layers to scan for the placement of calls. The scan will look for lines only in the layers you specify even though other layers may be currently displayed.

Choose one or more layers to search: this dialog will display the complete list of layers in the drawing file. You can scroll up and down the list and simple click with the mouse those layers you want to search for lines/polylines.

Types of Lines to Annotate:

C&G Lines and C&G Polylines: refer to lines that have been drawn using the CGDraw command, thus being based on the C&G coordinate file.

CAD lines and CAD Polylines: refer to lines that have been drawn using the CAD Draw command and are not based on the C&G coordinate files.

Example Cell: this display shows you the actual layout as it will appear on you drawing.

Prompts

When you choose the Place Calls menu item and a coordinate file is not already open, you will be asked to open a coordinate file. You will then see the following prompt at the command line:

Enter point sequence: [Point group/Reset/turn.sNap on/Setup/poLYline] (last point = <none>):

Point Group: If you press P and <Enter> you will be asked to enter a point group and it will be used to place calls automatically.

Reset: Press "R" resets the last point ID to <none>

sNap on or sNap off: Press "N" turns the CAD snaps on or off. When the command starts the AutoCAD snaps are off by default.

The Setup: Press "S" option brings up the Calls Setup dialog box.

polyline: if you Press "Y" and <Enter> you can then pick a C&G polyline and it will be annotated in the order that the vertices were specified when it was drawn.

Pulldown Menu Location: CG-Survey > CGDraw>Calls>Place Calls

Keyboard Command: CALL, CG_CALLS or CALS, CG_CALLS_SETUP

Prerequisite: Coordinate file

Move Calls

Allows you to move call text and once moved it will not go back to its original location when you use Refresh Screen to refresh calls. The calls will move or change if the point numbers that generated the call change but the position of the call relative to the end points will remain approximately the same.

Prompts

Select entities: Pick call on screen

Entities in set: 1

Select entities: Pick another call on screen

Move call to desired location.

Pulldown Menu Location: CG-Survey > CGDraw>Calls>Move Calls

Keyboard Command: MCL, CG_MOVE_CALLS

Prerequisite: Coordinate file

Reverse Calls

This feature allows you to reverse the bearing of the call.

Prompts

Pick a call to reverse: select call bearing on screen

Pick a call to reverse: select another call bearing on screen

Pick a call to reverse: select again if wish to continue or

Press <Esc>: to quit command

Pulldown Menu Location: CG-Survey > CGDraw>Calls>Reverse Calls

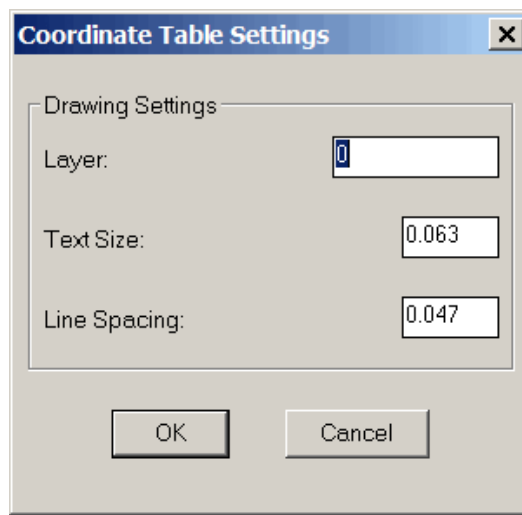
Keyboard Command: RCL, CG_REVERSE_CALLS

Prerequisite: Coordinate file

Tables

Coordinates

This feature allows you to draw a table containing information related to specified points in the coordinate file then place the table in the drawing by picking the desired location.



When you pick the Table > Coordinate menu item and a coordinate file is not already open, you will be asked to open one. Once a coordinate file is open, then the Coordinate Table Settings dialog will appear. Using this dialog you can configure the following settings:

Layer: Specify the layer on which you want the table drawn.

Text Size: Enter the text size in inches or centimeters. The text size is the size the text will appear when printed on a page.

Line Spacing: Enter the space you want between lines in inches or centimeters. The line spacing is the height of the spacing when the table is printed on a page.

Note Northings and Eastings will be rounded based on the values specified in the Rounding Options tab of the C&G Options dialog box.

Note The point ID, northing, and easting will always be part of the coordinate table. If you want elevations, codes and descriptions shown, make sure they are set to "On" on the Global Settings tab of the C&G Options dialog. Click OK to save the settings and continue the command, this will return the action to the command line. If you click Cancel the command will be canceled.

Selecting Points for the coordinate table:

Select the points that will be included in the table using the familiar C&G prompt.

Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt_group/Limits/Radius/Select]: A

Prompts

Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt_group/Limits/Radius/Select]: A

Expand base selection set: Choose more points from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt_group/Limits/Radius/Select/Include/eXclude/View]:

Building Point Selection Set...

Press Enter 2 more times to end selection set: <Enter>

When done selecting points just press <Enter>:The table will be drawn at the cursor.

Move Coord Table to desired location: Drag the table to the desired location on the drawing and press the left mouse button to place the table.

Point	Northing	Easting	Elev.
1	311.54700	156.49600	806.000
2	195.62300	178.23400	800.000
3	179.32153	349.22200	804.000
4	88.75616	482.53484	784.000
5	89.62426	592.66286	774.000
6	278.84438	641.83001	806.000
7	384.00033	644.84340	797.000
8	451.01870	439.06257	792.000
9	369.50846	370.95736	792.000
10	431.09387	252.13509	812.000
11	398.49010	186.92788	810.000
12	277.07728	299.76590	796.000
13	301.70887	505.67938	804.000
14	221.32480	513.17644	780.000
15	273.50991	391.24427	803.000

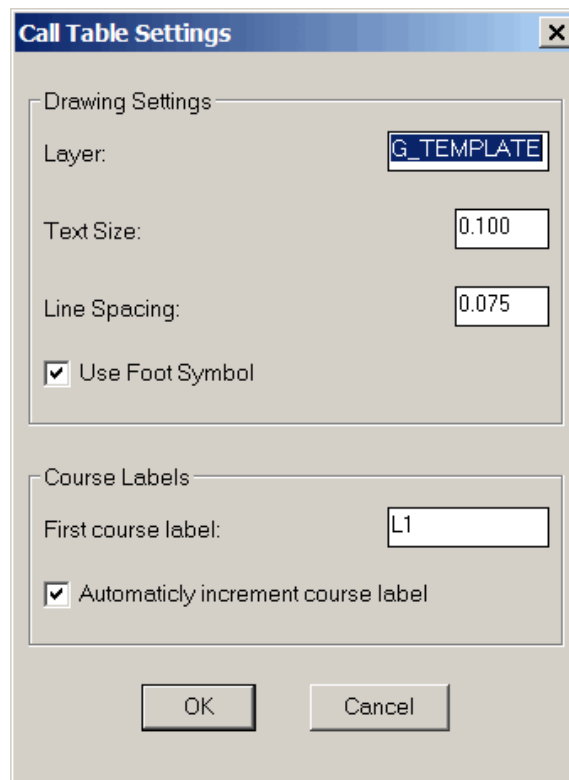
Pulldown Menu Location: CG-Survey > CGDraw>Tables>Coordinate

Keyboard Command: CRDT, CG_COORD_TABLE

Prerequisite: Coordinate file

Call Table

This feature allows you to place the bearings, distances, etc. in a table instead of along the lines and curves in the drawing. This is especially useful when space along the lines or curves is limited. When you use a call table only the course labels are placed along the line or curve to identify it in the table.



When you choose the Table > Call item from the menu the Call Table Settings dialog appears. As with the coordinate table, this dialog box allows you to enter: the layer, text size and line spacing for the call table.

Drawing Settings

Layer: CG_Template

Text size: 0.100

Line Spacing: 0.075

Use Foot symbol: Checked

This setting will place a Foot symbol < " > if desired.

Course labels

First course label: The course labels will be based on the First course label setting in the Call Table Settings dialog box. The course label will then be determined by incrementing the last character in the previous course label starting with the first course label.

For example: line1 increments to line2, line3, etc. whereas line_a increments to line_b, line_c, or as in the example above L1, L2 L3, etc.

Automatically increment course label: Check or Uncheck box

This setting will automate the process with selecting point sequence

After configuring the settings in the Call Table Settings dialog:

Ok Button : select OK

Prompts

You will be prompted at the command line to enter the point sequence.

Enter the point sequence by typing point IDs or by selecting C&G points and/or lines on the screen.

Enter point sequence: [cLockwise_curve/ccW_curve/Point_group/Reset/turn_Snap_on]

The Reset button: The ' R' resets the last point to <None>

When you are done entering calls: press <Enter>

This will end the input process and the call table will be drawn at the cursor.

Move Call Table to desired location: Drag the table and left-click the mouse button to place table on screen. The course description will be placed in the table and on the line or arc in the drawing.

Course	Bearing	Distance
L1	N 15°04'36" E	183.84'
L2	S 85°52'05" E	170.10'
L3	S 11°44'15" E	163.23'
L4	S 65°53'25" W	141.06'
L5	N 66°49'49" W	132.63'

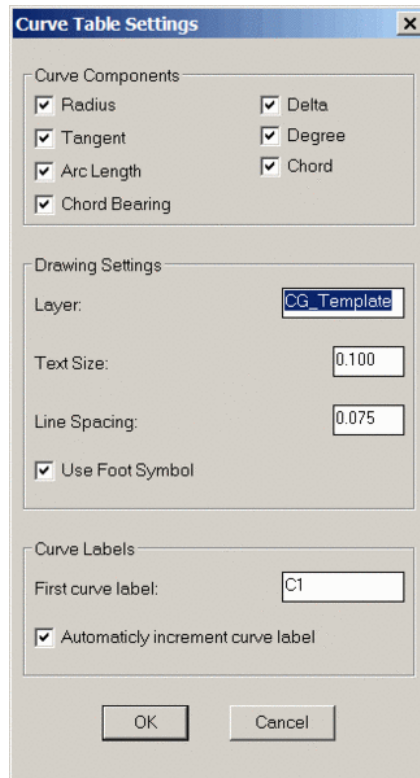
Pulldown Menu Location: CG-Survey > CGDraw>Tables>Call

Keyboard Command: CALT, CG_CALL_TABLE

Prerequisite: Coordinate file

Curve

This feature allows you to Draw a table containing curve information for specified curves.



You will be prompted to open a coordinate file if one is not already open. Once the coordinate file is open, the Curve Table Settings dialog box will appear. The Curve Table Settings dialog allows you to configure the following settings:

Curve Components

Check the checkboxes for the curve components that you wish to appear in the table, Radius, Tangent, Arc Length, Chord Bearing, Delta, Degree and Chord.

Drawing Settings

Enter the layer, text size and line spacing, and check or uncheck the Use Foot Symbol checkbox.

Layer: CG_Template

Text Size: 0.100

Line spacing: 0.075

Curve Labels

Enter the First Curve Label for the first curve

First Curve Label: C1

Automatically increment curve label: check or uncheck the Automatically increment curve label checkbox.

OK Button: When done, click OK to begin entering the curve data.

Prompts

You will be prompted at the command line to Enter point sequence

Picked C&G Point [1444]

Enter point sequence

[cLockwise_curve/ccW_curve/Reset/turn_Snap_on] (last point = 1444): L

Enter radius point for curve [Reset/turn_Snap_on]:

Picked C&G Point [1449]

Enter point of tangency (PT) for curve [Reset/turn_Snap_on]:

Picked C&G Point [1448]

Move Curve Table to desired location: Drag Table to desired location and Left-mouse click to place on the

Curve	Radius	Tangent	Length	Delta	Degree	Chord	Chord Bear.
C1	42.90'	48.07'	72.28'	98°30'20"	133°33'43"	64.01'	N 73°14'52" E

drawing.

Note: After entering the PT point for any curve, you can continue entering curve data. However, you should be aware that the PT point is shown as the last point.

If the PT point is not the PC of the next curve then you need to enter "R" for Reset. This allows you to begin the next curve at a new PC, then continue on to enter its radius point and PT.

Pulldown Menu Location: CG-Survey > CGDraw>Tables>Curves

Keyboard Command: CURT, CG_CURVE_TABLE

Prerequisite: Coordinate file

Auto Map

Map allows the user to automate the production of a drawing based on special "mapping codes" included in the descriptions found in the coordinate file. Using this feature can save a great deal of time. This allows the lines and points to be placed in the drawing based on mapping codes without user intervention.

Pulldown Menu Location: CG-Survey > CGDraw>Auto Map

Keyboard Command: MAP, CG_MAP_DRAW

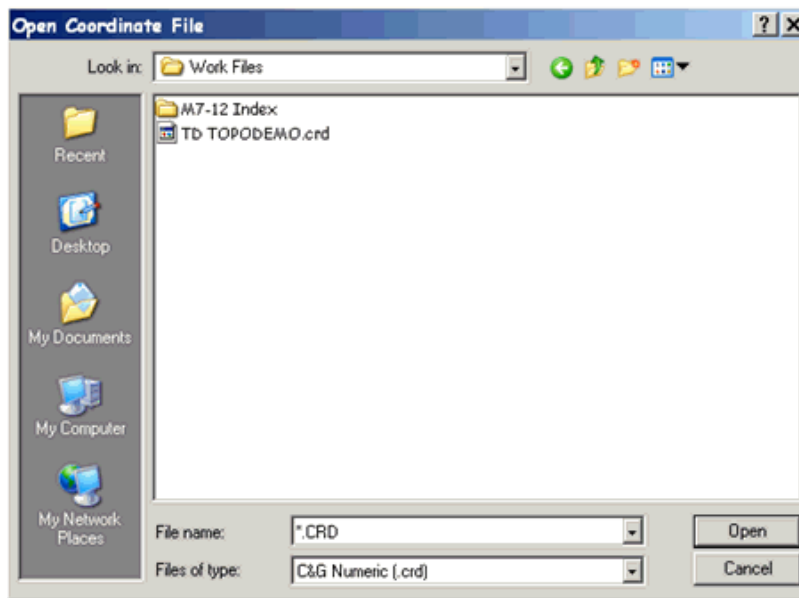
Prerequisite: Coordinate file

Draw

This feature automates the production of a drawing that can contain specific points, lines, arcs and curve fit lines. The draw option also acts as a Cogo function in that it will calculate the PC, PT and radius points of curves and has the ability to calculate points by traversing and intersection.

Prompts

After choosing the Draw command, if a coordinate file is not open, you will be asked to open one.



After opening the coordinate file, you will be asked to select the points you want to map:

Choose initial points for base selection set from coord. file: (Enter when done) [All/Block/Code/Desc/Elev/Pt-group/Limits/Radius/Select]:

Next, you will be asked whether you want to store elevations at calculated PC, PT, and radius points: When locating items like back of curb you may need to note the beginning and ending of curves. The points located are never exact as far as the beginning and ending of the curve, but when noted in the mapping routine the application will compute a PC, PT and Radius using the best fit routine and you can choose to store these points or not.

Note: If Auto Point Plot is ON as specified in the Graphic Options tab of the C&G Options dialog, points will be plotted and lines, arcs and/or curve fit lines are drawn when indicated by Mapping Codes found in the point descriptions.

Mapping Codes Used by the Draw feature

The map codes used by the Draw feature must be placed in the description field for each point in the coordinate file that is to be "Mapped".

Below is the list of map codes:

- BL - Begin Line
- EL - End Line (optional)
- CL - Close Figure
- PC - Begin Curve (tangent to previous line)
- OC - Point on Curve (begin/end non-tangent curve)
- PT - End Curve (tangent to next line)
- RP - Radius Point
- CF - Curve Fit (spline fit to irregular curves)
- CC - Compound Curve
- RC - Reverse Curve

Mapping Codes can be upper or lower case. The map code **MUST** be followed by an asterisk and a line description for the line that is being drawn. For example: BL*CURB1, where CURB1 is the line description for the line you are beginning. It is OK to have spaces between the code, asterisk and line description, but it is not necessary.

For example:

Point ID Description**5 BL* CURB1 BL*SW1 WV****6 CURB1****7 SW1 PP****8 PC* CURB1****9 FH****10 PT* CURB1****11 CURB1 SW1****12 OC* SW1****13 SW1****14 SW1****15 SW1****16 OC* SW1****17 CL* SW1****18 BL* CURB1**

Important Note: Mapped lines are connected in ascending order by point ID. The point ID's are always saved in the coordinate file in increasing order. Since the coordinate file is used to perform the Map Drawing and the point ID sequence is produced when the raw data is reduced, it follows that the order of field location of the points will determine point ID sequence order when the lines are mapped.

In the sample sequence above:

Point 5 begins two lines, **Curb1** and **SW1**. Curb1 and SW1 are line descriptions. A line description must be a whole word (no spaces). **WV** (water valve) is not the beginning of a line because an asterisk does not precede it.

For example:

5 BL*CURB1 BL*SW1 WV

The Curb1 line will be drawn from point 5 to point 6 to point 8. This begins a curve tangent to the line from 6 to 8 continuing to point 10. The curve is tangent to the line from 10 to 11. Since point 18 begins a new Curb1, point 11 is the end of the first Curb1 line (the EL code is not required in order to end a line).

A second line (SW1) will be drawn from 6 to 7 to 11 to 12. At point 12 a non-tangent circular curve begins and continues through points on the curve at 13, 14, and 15. The non-tangent curve ends at point 16 and lines continue from 16 to 17 to 1 (the CL code closes the figure). In creating the non-tangent curve from point 12 through point 16, points 13-15 are used by the Map Draw feature in the calculation of the best fit circular curve.

In addition to the lines drawn, the symbol specified for the WV description in the description table (see CGMngmt) will be placed at point 5 and, at point 7, the symbol specified in the description table for the description PP will be drawn.

As demonstrated in the above example, you may combine multiple codes and line descriptions within a single point description.

For example:

Point ID Description**20 BL*SW1 BL*CURB1 CURB2 EL*CURB3 CL*CURB4**

In this example point 20 begins the **SW1** line and the **CURB1** line. It continues the **CURB2** line. It ends the **CURB3** line and it closes the **CURB4** line.

The Begin Line Code:

All lines must start with a BL code. No lines will be connected to a point unless a word in the point description matches a BL* line name.

The Close Line Code: The close line code (CL) causes the Draw Map feature to connect the CL point to the BL point. You can also use the CL command to traverse. Thus you may place dimensions after a CL command. For example:

Point ID Description

20 *BL*BLD1

21 CL*BLD1+10.1+10.2-20.3+50.6 EL*SW1

Note: The FC-48 data collector does not allow '+' characters in description field. Because of this, the '/' character can be used instead of the '+' character in all the CL examples.

In the above example a line will be drawn from point 20 to point 21. The following points will then be calculated through a traverse sequence (assume the next point available is 100):

Occupied Pt BS Pt Angle Distance New Point

21 20 90 10.1 100

100 21 90 10.2 101

101 100 270 20.3 102

102 101 90 50.6 103

Point 103 will then be connected to point 20 to close the BLD1 line. Please note that point 21 is also the end of the SW1 line.

In a CL mapping code sequence, a negative dimension turns -90 degrees from the back azimuth and a positive dimension turns +90 degrees from the back azimuth. Both the '+' and '-' symbols are required but, as noted above, the '/' symbol can be substituted for the '+' where necessary.

This same figure could also be drawn using the following sequence:

Point ID Description

20 BL*BLD1

21 CL*BLD1+10.1+10.2-20.3+

Note that the closing distance was not included in the description sequence. See the following examples.

If you have located two corners of a rectangle, you may use the following short cut:

Point ID Description

20 BL*BLD1

21 CL*BLD1+50.6+

In the above example a line will be drawn from point 20 to point 21. The following points will then be calculated through a traverse sequence (assume the next point available is 100):

Occupied Pt BS Pt Angle Distance New Point

21 20 90 50.6 100

Point 101 will be calculated by a bearing-bearing intersection. Then point 101 will be connected to point 20. The first '+' sign determines the direction used to calculate point 100. The description ending in a '+' sign has the same effect as ending in a '-' sign: if there is no dimension after the last '+' or '-' sign, the last point will be calculated by a bearing-bearing intersect.

If you have located three corners of a rectangle, you may use the following short cut to define the lines to be drawn:

Point ID Description

20 BL*BLD1

21 BLD1

22 CL*BLD1+

In the above example lines will be drawn from point 20 to 21 to 22. The missing corner will be calculated using a bearing-bearing intersect and stored. As noted earlier, ending in a '-' sign instead of a '+' sign has the same end result.

Curve Codes

Anytime a circular curve is encountered, 3 new points may be calculated and stored in the coordinate file. These points are the PC, PT and radius point of the curve. It is necessary to calculate these points during automated mapping since the field points are only approximations of a perfect curve. They will automatically be assigned point numbers (regardless of the Auto Point Numbering setting). The points calculated during automated mapping of curves will begin with the coordinate files current high point number plus 1.

If the beginning of a line is also the beginning of a curve, one of the following formats must be used:

Point ID Description

10 BL*SW1 CF*SW1 (begin a curve-fit line)
or 10 BL*SW1 OC*SW1 (begin a non-tangent circular curve)
or 10 BL*SW1 PC*SW1 (begin a tangent circular curve)

Once a curve has begun, all matching line descriptions will be considered points on the curve until the curve is ended. A curve is ended with either a PT*, OC*, or CF* code.

For Example:

Point ID Correct Sequence Incorrect Sequence

10 OC*SW1 (Begin SW1)
11 SW1 OC*SW1 (will end SW1)
12 SW1 OC*SW1 (will end SW1)
13 SW1 OC*SW1 (will end SW1)
14 OC*SW1 (End SW1)

The first OC begins the curve. The next OC ends the curve. All the points between them are on the curve. The same is true for curve fit (CF*).

If a curve is either tangent (in), tangent (out) or tangent (in) & tangent (out), you only need two points to define the curve:

Point ID Sample 1 Sample 2 Sample 3

10 PC*CURB1 PC*CURB1 OC*CURB1
11 PT*CURB1 OC*CURB1 PT*CURB1

Otherwise you will need at least three points on a curve:

Point ID Description

12 CF*CURB1
13 CURB1
14 CF*CURB1

The RP Mapping Code

If you use the RP code (radius point), it will be used regardless of the number of points on the curve. The radius will be calculated by averaging all the distances from the radius point to the points on the curve.

Best Fit Circular Curve Calculations

If you have three or more points on a non-tangent curve, the best-fit curve solution is used to find the radius point.

If you have three or more points on a tangent curve (either tangent in, tangent out, or tangent in and out), the best-fit

curve solution is used to determine an approximate radius and radius point. A dummy point is then calculated on the curve and a curve is drawn that goes through the dummy point and meets the tangent criteria (the PC and PT points are shifted up/down the tangent lines as necessary). If only three points are located, PC, POC and PT, the curve will always go through the POC point.

If you have only two points (PC and PT) on a tangent curve, the tangent lines from the PC and PT will be intersected to find the PI of the curve. The distance from the PI to the PC and the distance from the PI to the PT will be averaged to obtain a tangent distance. A new PC and PT point will be calculated on the tangent line and the radius point will be calculated based on the tangent and central angle.

Non-Circular Curves

You may use the CF* code for a non-circular curve fit (splines). The CF code will start a curve fit line. The curve will continue until a second CF* code is encountered, example:

Point ID Description

11 CF*SW1

12 SW1

13 SW1

14 CF*SW1

Only use CF to start or end a curve. Notice points 12 and 13 do not have automated mapping codes.

A smooth curve will be drawn through points 11, 12, 13 and 14. No new coordinate points are generated with the CF code.

Layers and linetypes for mapped lines and curves

The description table determines the layer in which a mapped line will be drawn. For mapped lines and curves, only the description and layer fields in the description table are used. However, if the default layer is not set, no description table lookup is performed and the line is drawn on the current layer.

For example, assume that the default layer has been set and that the description table contains the following entry:

Desc. No. Description Layer Name

5 CURB Road

Since layer "Road" is specified for description "Curb", all lines with descriptions "Curb" will be placed in layer "Road". Numbers are not used in the comparisons: Curb1, Curb2, Curb10, etc. are considered a match for the description "Curb" and will therefore be placed in layer "Road".

If a matching description is not found in the description table, the line is drawn on the default layer (as set in the Graphic Options tab of the C&G Options dialog box).

Calculated Points

All coordinate points that are automatically calculated and stored during automated mapping are given a MP point code.

Note: Even though the point description field can contain Mapping Codes, the point code found in C&G coordinate files is separate and distinct from the point description field. All points already having an MP Code are ignored by automated mapping. This avoids re-mapping points that were generated during automated mapping and thus were not points actually located in the field.

Important Note: Consider the MP point code as a reserved code and do not use it for field data collection.

The description (e.g., CURB) used for calculated points is the same as the line description of the points the calculated point is associated with and reflects the type of calculated point that it is.

For Example:

Assuming the line description for the following points is "CURB1" and the points are the PC, PT and radius point of a curve, then the line descriptions will be:

New Point ID Point Code Description

100 MP PC CURB1

101 MP RAD.PT CURB1

102 MP PT CURB1

Plotting of Points

If Auto Plot Points is "On", all the selected points in the coordinate file will be plotted on the screen during the mapping process. If a default layer is set, Each point will be drawn on the layer specified in the description table. The points labels will be configured as specified in the description table. Any point that does not have a description match in the description table will either be drawn on the default layer.

Pulldown Menu Location: CG-Survey >Auto Map>Draw

Keyboard Command: MAP, CG_MAP_DRAW

Prerequisite: Coordinate file

Erase

The Map Erase feature will find all line, arc, curve fit and point entities created Using the Map Draw feature and delete them from the drawing. It will also delete any coordinate points from the coordinate file that where created with the using Map Draw (PC, PT, radius points and close line (CL) points or those points having the point code MP)

Pulldown Menu Location: CG-Survey >CGDraw>Auto Map>Erase

Keyboard Command: EMAP, CG_MAP_ERASE

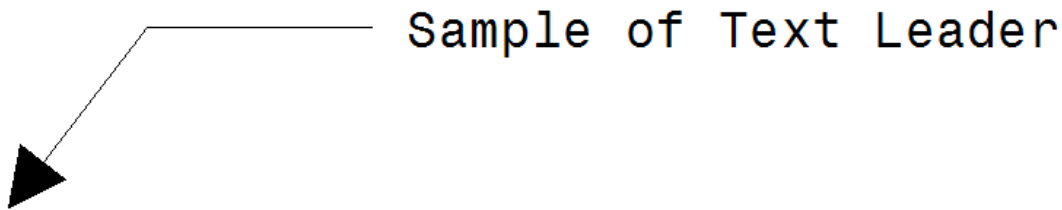
Prerequisite: Coordinate file

Leaders

Leaders are used to label features in the drawing. They consist of a line or series of connected line segments with an arrow at one end and a text label at the other end. The arrow size is determined by the symbol size as set in the Drawing Settings - Active Point Symbol dialog.

Text

A text leader allows you to draw a series of lines with an arrow at the starting point then specify the text that is to be drawn at the final endpoint of the leader.



Prompts

To draw a text leader:

Pick start of leader: Pick the starting position of the leader with the left mouse button:

Picked C&G Point [4]

To point (Enter to end): Move the cursor to the next point on the leader and press the left mouse button. The first segment of the leader will be drawn with an arrow placed at the first point picked

To point (Enter to end): As you pick succeeding points, lines will be drawn from the previous point to the current point.

To point (Enter to end): When you have picked the final point, press <Enter> or the right mouse button.

Enter Text for leader: At the command line, type the text to be placed on the leader and <Enter>.

Sample of Text Leader

Pick start of leader: repeats the command

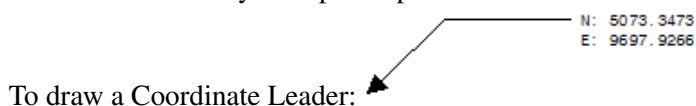
Pulldown Menu Location: CG-Survey > CGDraw>Leaders>Text

Keyboard Command: TXTL, CG.TEXT_LEADER

Prerequisite: Coordinate file.

Coordinate Leader

This feature allows you to pick a point then draw a leader that is labeled with the coordinates of the point picked.



To draw a Coordinate Leader:

Prompts

Pick the starting point of the leader with the left mouse button: If you pick a C&G point, the coordinates will be read from the coordinate file, otherwise the graphic coordinates will be used.

Move the cursor to the next point for this segment of the leader and click the left mouse button: Repeat until all desired leader segments are drawn.

The coordinates of the first point picked will be drawn near the final point on the leader. The coordinates are rounded based on the rounding specifications in the Rounding Options Tab of the C&G Options dialog box.

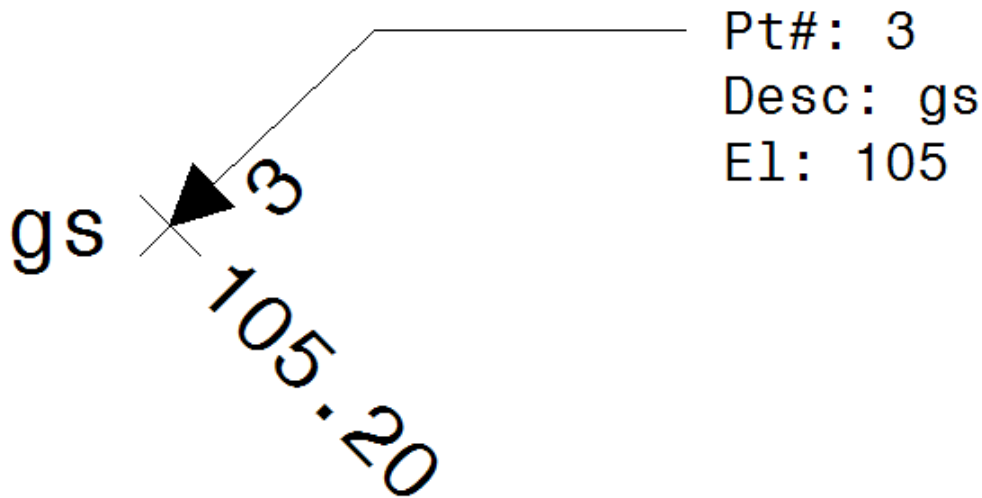
Pulldown Menu Location: CG-Survey > CGDraw>Leaders>Coordinate Leader

Keyboard Command: CRDL, CG.COORD_LEADER

Prerequisite: Coordinate file

Point Label

This feature allows you to label a point using a leader instead of the normal point labels. This feature can only be used with C&G points. The leader will display the point ID and, if Elevation and/or Descriptions are "On", the elevation and description will be displayed as well.



Prompts

It is suggested that you first plot the points on the screen with the point labels turned off by setting their point label positions to 0 on the Drawing Settings - Point Label Position dialog. Thus only the symbols will be plotted. Next, in the Drawing Settings tab of the CGOptions dialog box: turn on the items you want to be displayed on the leader.

Now choose the Point Label Leader menu item:

Pick start of leader: Move the cursor to a C&G point and press the left mouse button. Picked C&G Point [3]

To point (Enter to end): Move the cursor to the end point of the next leader segment and press the left mouse button. An arrow will be drawn at the first point picked.

To point (Enter to end): When you have picked the end of the last segment of the leader, press <Enter> or the right mouse button.

To point (Enter to end): The point attributes will be placed near the last point picked for the leader.

(You may repeat the previous step as many times as is necessary)

When done press <Enter> when asked to pick the next C&G point, Press Enter

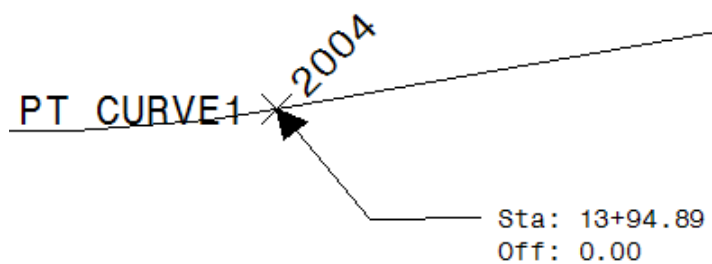
Pulldown Menu Location: CG-Survey > CGDraw>Leaders>Point Label

Keyboard Command: PTL, CG_POINT_LEADER

Prerequisite: Coordinate file

Station-Offset

This Leader feature allows you to label points along a predefined alignment with their station and offset.



Prompts

Prior to using this feature you must create a point group that defines the alignment.

Next you will be asked to open a point group file. In the Select a C&G Point Group File dialog box: select the point group file that defines the horizontal alignment you wish to use.

Enter starting station <0.00000>: 1000
10+00.00

Enter Starting station for the alignment as defined by the point group. If a station is specified for the first subgroup name in the point group file, it will be used as the default station (for more details on this, see the section on point group files in CGMngmt).

Pick the starting location of the leader: Picked C&G Point [3]

If a C&G point is not found at this location, the station and offset will be calculated using the drawing coordinates of the picked point. If a C&G point is found, the station and offset will be calculated from the coordinates read from the coordinate file. If a C&G point is found, the point ID will be printed at the command line.

To point (Enter to end): Move the cursor to the end point for this segment of the leader and press the left mouse button, An arrow will be placed at the first point picked. Repeat until all the segments of the leader have been specified.

To point (Enter to end): When you have picked the end point of the last segment of the leader, press the <Enter> key or right mouse button.

The station and offset label will be placed next to the end point of the leader.

To point (Enter to end): Enter

Pick start of leader: Repeats the command

Note: The station and offset values are rounded based on the values specified in the Rounding Options tab of the CGTools > CGOptions dialog box

Pulldown Menu Location: CG-Survey > CGDraw>Leaders>Station-Offset

Keyboard Command: STOL, CG_STA_OFF_LEADER

Prerequisite: Point group must be created

Query

Selecting Query and then selecting a drawing object will display information related to the following C&G entities:

Point symbols and labels
Lines

Arcs
Polylines
Calls
Structure Footprints
Coordinate, Call and Curve Tables

Prompts

Select entities: (Pick entity on screen) Entities in set: 1

Select entities: (item selected) C&G POLYLINE

Below is an example of a Query listing of a C&G Polyline:

Coordinate File: CGDEMO.CRD (C&G Numeric)

Plotted from Auto Mapping: No

Layer: Boundary

Points defining C&G Polyline: 11->9

Polyline is NOT CLOSED

Pulldown Menu Location: CG-Survey > CGDraw>Query

Keyboard Command: Q, CG_QUERY

Prerequisite: Coordinate file

Drop C&G Attributes

This feature allows you to strip the C&G attribute from any C&G entity. When the C&G attribute is dropped, the graphic entity becomes standard CAD entity and will no longer be affected by the Refresh Screen feature nor can they be used by C&G commands requiring C&G entities as input.

Prompts

Select entities: pick on graphic screen

Entities in set: 1

Select entities: pick again on screen

Entities in set: 4

Select entities: select another set of entities by window

Entities in set: 7

Select entities: Specify opposite corner: 8 total found,

Entities in set: 8

Pulldown Menu Location: CG-Survey > CGDraw>Drop C&G Attributes

Keyboard Command: DROP, CG_DROP

Prerequisite: Coordinate file

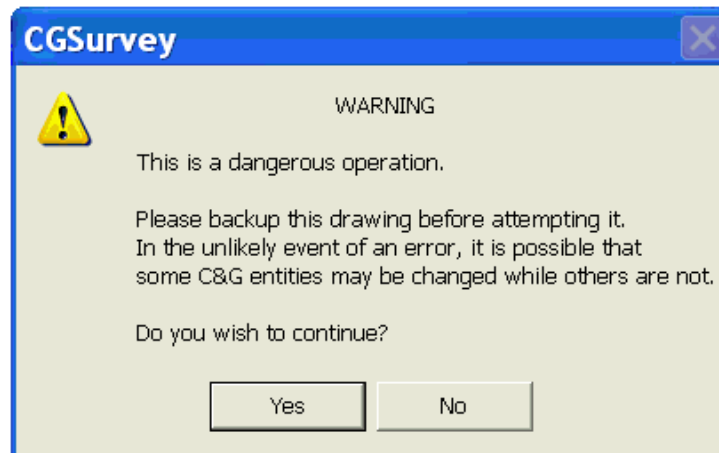
re-Associate Coord. file

This routine allows you to associate the current drawing file with different coordinate file than created the drawing file. An example of this could be a phase of a project. The overall project coordinate file might contain 10,000 to 15,000 coordinates, While working on a phase of the overall project a separate, smaller, coordinate file was created, easier to work with a 1000 points rather than 15,000. Now you want to re-associate this new drawing file with the

overall project coordinate file.

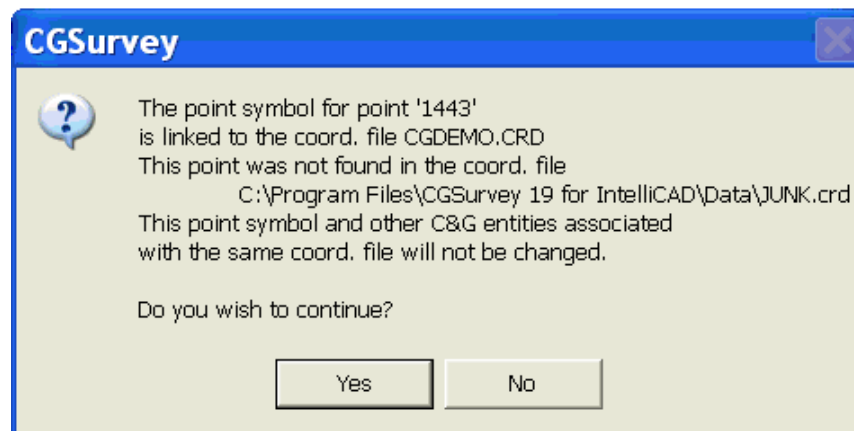
Prompts

After selecting the re-Associate command there will be displayed a Warning dialog box. This box recommends that you create a backup of your drawing file. The danger with using this application is if the coordinates are not managed carefully and the same point ID's were used in both the overall project file and the out parcel then the graphics will be incorrect. C&G graphics are based on the coordinate file and if the X/Y/Z values change so does the graphics.



Do you wish to Continue? Press <Y> button: Y

Re-associate only those C&G entities plotted using which coord. file [Any_file] <A>: A



An Additional Warning message may also appear indicating conflicts in linked crd files

Do you wish to Continue? Press <Y> button: Y

Pulldown Menu Location: CG-Survey > CGDraw>re-Associate Coord. file

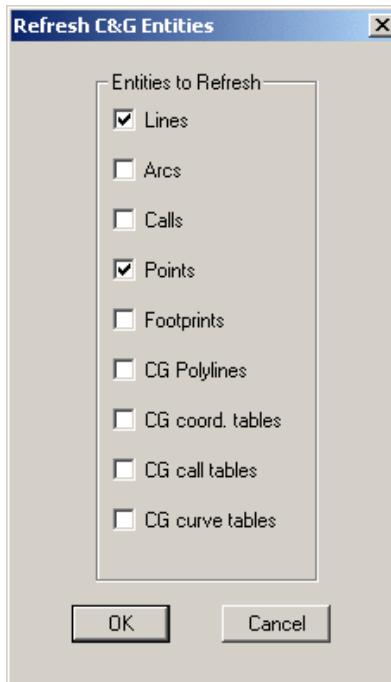
Keyboard Command: Not available

Prerequisite: Coordinate file

Refresh Screen

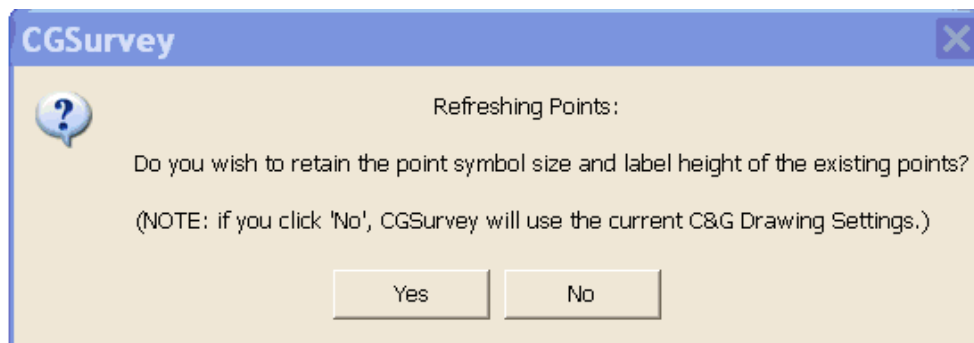
Many graphic entities created by CGSurvey contain attributes that tie them to the coordinate file (C&G points, lines, arcs, calls, etc.). Examples would be point numbers, elevations, and descriptions that are plotted with the node when you plot points. However, once an entity is drawn the user is free to move or edit it. Also, it may be necessary to change the coordinates of the point or points used to create the entity.

If C&G entities are edited or the coordinate values change, you refresh the drawing so that it reflects the current coordinate file values. You can use the Refresh Screen feature to find all C&G entities tied to the coordinate file and read the points from the coordinate file and redraw the entities based on the current coordinate values.



Prompts

Check the appropriate boxes in the list to refresh: Press Ok to continue



Do you wish to retain the point symbol size and Label height of the existing points?: Press <Y> button

Below is example of Refreshed screen entities:

Command: cg_refresh

24 Lines refreshed.

2 splines refreshed.
24 Calls refreshed.
24 Points refreshed.
There were 2 C&G polylines refreshed.

Pulldown Menu Location: CG-Survey > CGDraw>Refresh Screen

Keyboard Command: REF, CG_REFRESH

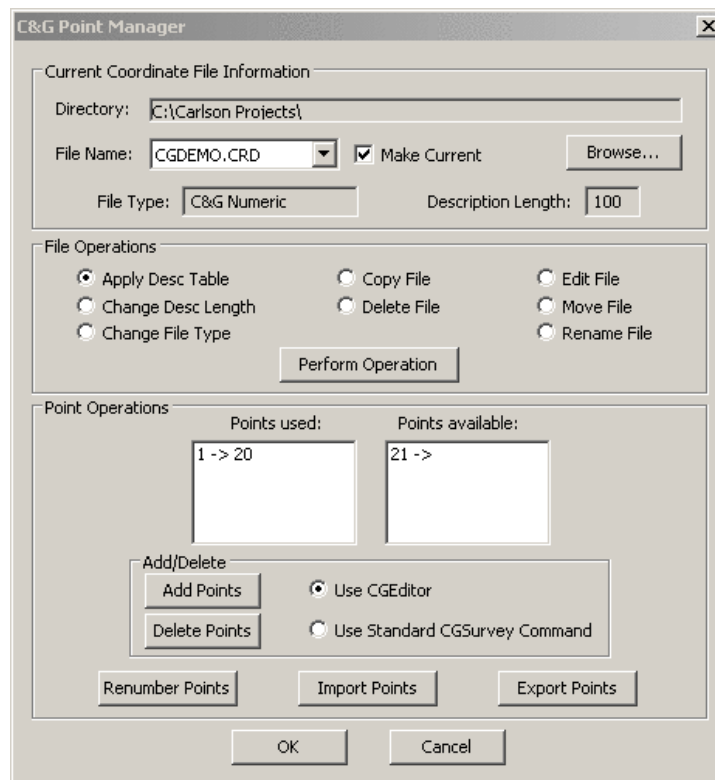
Prerequisite: Coordinate file

CGMngmt

Point Manager

The point manager allows the user to perform most of the normal coordinate file management functions. You can perform whole file operations such as renaming the file, copying or moving the file, etc. There are also point operations which allow the user to add, delete, or change individual points or groups of points in a coordinate file.

The **C&G Point Manager** dialog (shown below) is divided into three sections. These sections are described below.



Current Coordinate File Information

This section gives you basic information on the currently selected coordinate file. The **Directory** and **File Name** defaults to the currently active coordinate file but you can choose to perform operations on any one of the supported types of coordinate files by clicking the **Browse...** button. When you click the **Browse...** button you will see a file dialog allowing you to choose the coordinate file you wish to work on. The **Make Current** checkbox allows the user to make the specified file the current file. Thus, when the dialog closes, the file will be used for future commands requiring a coordinate file.

File Operations

This section of the dialog allows you to perform operations that effect the entire coordinate file.

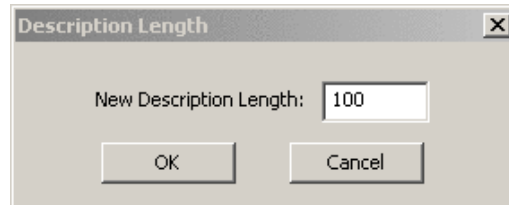
There eight operations that can be performed using this section of the dialog (see descriptions listed below). To perform one of the operations on the file shown in the **Current Coordinate File Information** section, click on the radio button for the desired operation then click the **Perform Operation** button.

Apply Desc Table

This operation only applies to C&G coordinate files and will not be available for other file types. When you apply a description table to a coordinate file it translates the numeric codes found in the description field using a C&G description table. For each point in the coordinate file having an integer in the description field the program looks for that integer description number in the description table. If a matching description number is found in the description table, the description found in the description table is placed in the description field for that point and the description number is placed in the code field for that point. The point is then stored back to the coordinate file with the changed field values. If no match is found the point is not changed in any way.

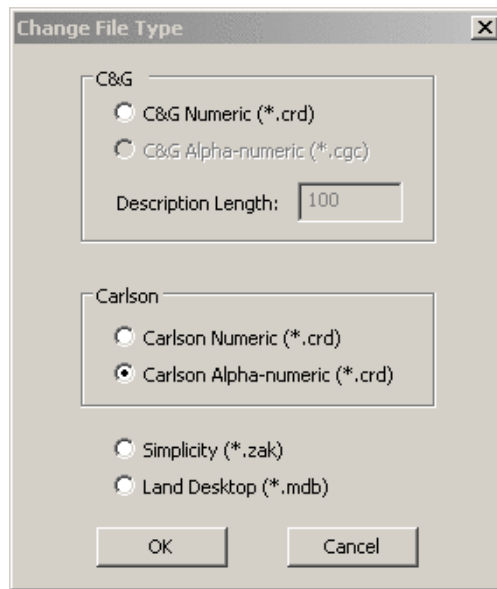
Change Desc Length

This operation only applies to C&G coordinate files and will not be available for other file types. When a C&G coordinate file is created, the user is allowed to specify the length of the description for a given point in the file. The description length may be between 1 and 100 characters. This operation allows the description length to be changed. It can be made smaller or larger. If an existing point in a coordinate file has a description that is longer than the new description length, the description will be truncated. When you click the **Perform Operation** button you will be asked to enter the desired description length (see dialog below).



Change File Type

This operation allows the use to convert among the supported types of coordinate files. The types supported are C&G numeric (*.crd) and alphanumeric (*.cgc), Carlson numeric (*.crd) and alphanumeric (*.crd), Simplicity (*.zak) and AutoCAD Land Desktop (*.mdb). When you select this operation and click the **Perform Operation** button you will see the **Change File Type** dialog:

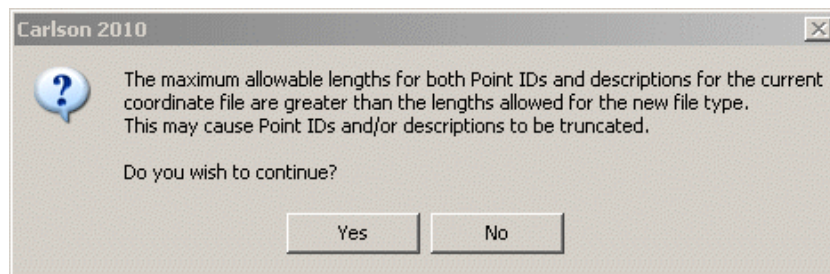


In the **Change File Type** dialog choose the type of file you want the current file to be converted into by clicking on the appropriate radio button.

Note: the radio button for the current file type is greyed out.

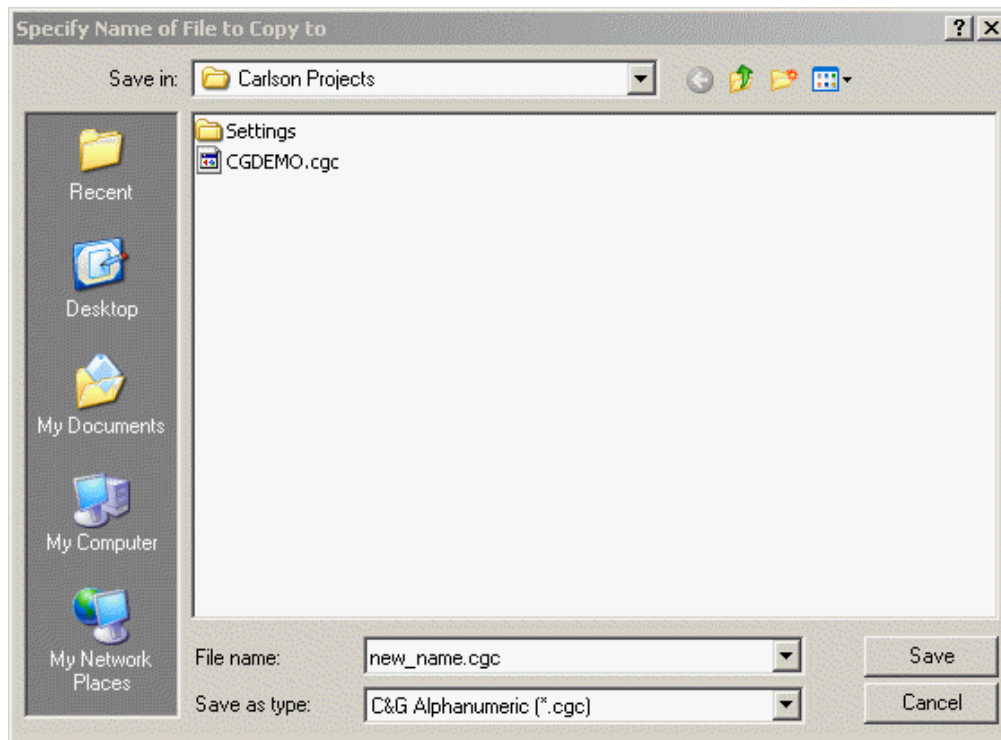
After choosing the file type click the **OK** button. Click **Cancel** to cancel the operation.

If you attempt to convert to a file type having point ID length or description length limits that are less than the limits for the file being converted, you will get the following warning:



Copy File

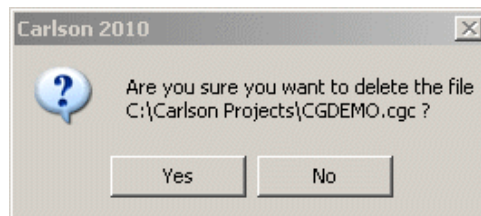
Performs a basic file copy. Must be to another directory and/or file name. When you click the **Perform Operation** button you will be asked to specify the copied file name and directory using a file dialog (see below).



Note: by changing the **Save as type:** this command can change the file type when it copies the file. However, if the type of file being copied has maximum allowable point IDs or descriptions that are greater than one or both of those for the file being copied to, you will receive a warning that point IDs and/or descriptions may be truncated (see Change File Type section above).

Delete File

Deletes the file listed in the **Current Coordinate File Information** area of the **C&G Point Manager** dialog along with any of its associated files. Before actually deleting the file you must click the **Yes** button in the following dialog.



Edit File

Allows the user to use the CGEditor to edit the coordinate file. You may add and delete points or edit any of the fields for an existing point (see the CGEditor section for more information on using the CGEditor)

CGEditor - CGDEMO.cgc

File Edit Add/Insert View Settings Tools Window Help

Pt Pt_v Pt_v Pt_v

Row #	Point ID	Northing	Easting	Elev	Code	Descripti
1	1	1311.54717	1156.49811	1808.00000		GND_SHOT
2	2	1195.62264	1178.23396	1800.00000		GND_SHOT
3	3	1179.32075	1349.22264	1804.00000		GND_SHOT
4	4	1088.75472	1482.53585	1784.00000		GND_SHOT
5	5	1099.62264	1592.66415	1774.00000		GND_SHOT
6	6	1278.94340	1641.93208	1806.00000		GND_SHOT
7	7	1384.00000	1544.84528	1797.00000		GND_SHOT
8	8	1451.01887	1439.06415	1792.00000		GND_SHOT
9	9	1369.50943	1370.95849	1792.00000		CL_CRK
10	10	1431.09434	1252.13585	1812.00000		GND_SHOT
11	11	1398.49057	1186.92830	1810.00000		GND_SHOT

For Help, press F1

Row: 1

Move File

Moves the current file to a new location. You will use a file dialog to specify the new location of the file. When moving a coordinate file you may also change the file type by changing the **Save as type:** The same cautions with regard to possible point ID and description truncation apply here as they do any time you change the file type (see Change File Type section above).

Rename File

Simply renames the file to whatever name the user specifies. You will use a file dialog to specify the new name and location of the file. Thus this command may be used to change the file type and/or move the file to a different directory. To change the file type change the **Save as type:** in the file dialog when you specify its new name. The same cautions with regard to possible point ID and description truncation apply here as they do any time you change the file type (see Change File Type section above).

Point Operations

You may perform several operations that effect one or more of the points in the current coordinate file in this section of the **C&G Point Manager** dialog. Use the **Points used:** and **Points Available:** lists to help you determine which points or ranges of points you wish to work on.

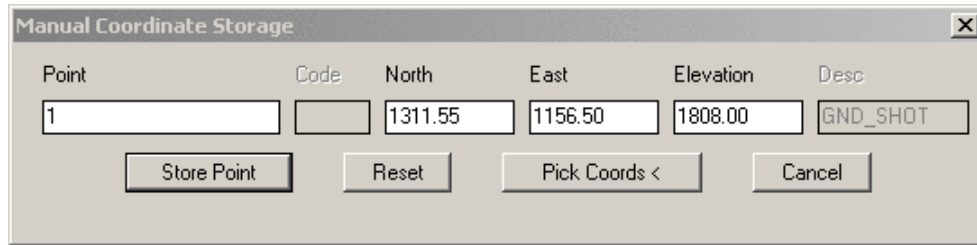
Add/Delete section

You may use the standard CGSurvey interface or the CGEditor to add or delete points. Choose which one to use using the radio buttons on the right side of the **Add/Delete** portion of the **Point Operations** area.

Add Points

If you chose to use the CGEditor, the CGEditor will come up (as shown above - see the CGEditor section for more information on using the CGEditor).

If you chose to Use the Standard CGSurvey Command, you will see the Manual Coordinate Storage dialog (see below). Fill in the edit boxes as described in **Management > Manual Storage**



The dialog box titled "Manual Coordinate Storage" contains a table with the following data:

Point	Code	North	East	Elevation	Desc
1		1311.55	1156.50	1808.00	GND_SHOT

Below the table are four buttons: "Store Point", "Reset", "Pick Coords <", and "Cancel".

Delete Points

If you chose to use the **CGEditor**, the **CGEditor** will come up (as shown above - see the **CGEditor** section for more information on using the **CGEditor**).

If you chose to **Use the Standard CGSurvey Command**, you will see the following prompt at the **Command:** line:

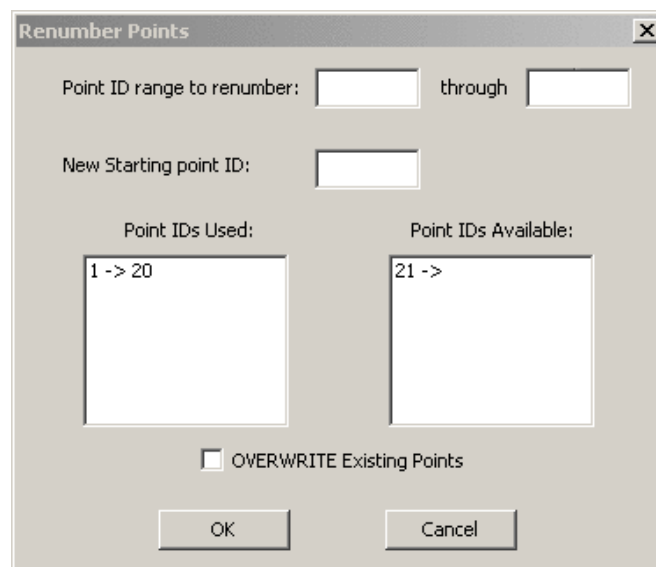
Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt-group/Limits/Radius/Select]: use one or more of the available methods to specify which points are to be deleted from the current coordinate file.

Buttons section

Renumber Points

If you click on the **Renumber Points** button you will see the **Renumber Points** dialog:



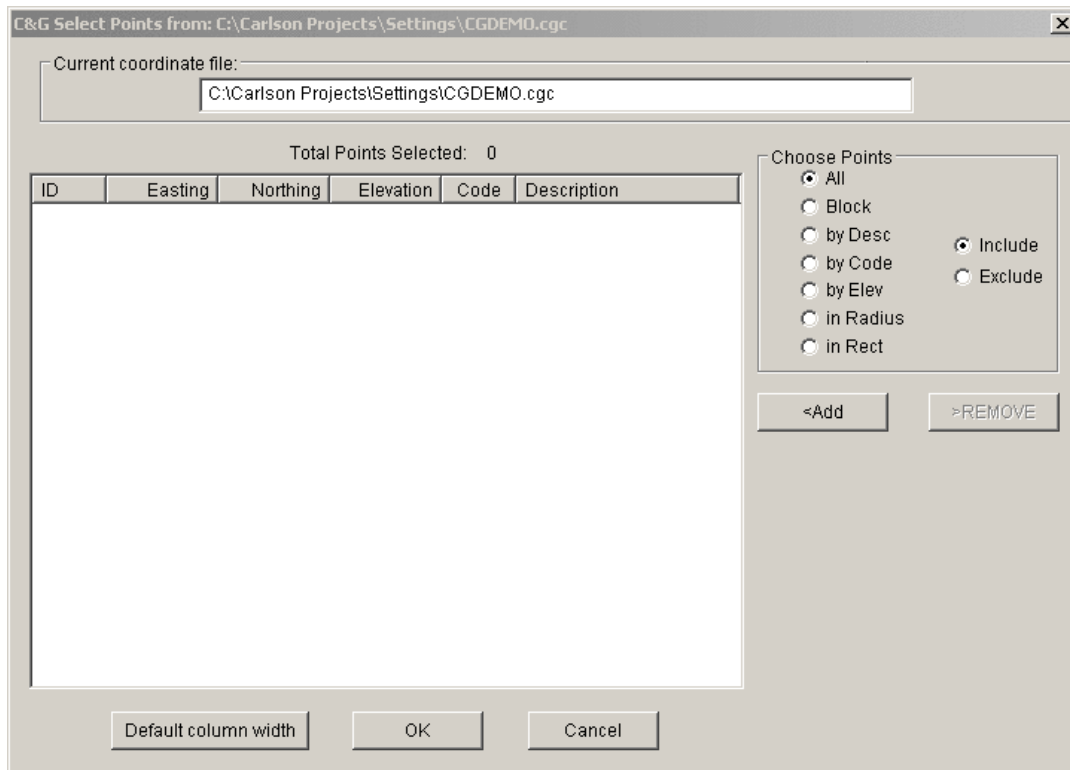
The dialog box titled "Renumber Points" contains the following fields and controls:

- Point ID range to renumber: [] through []
- New Starting point ID: []
- Point IDs Used: [1 -> 20]
- Point IDs Available: [21 ->]
- OVERWRITE Existing Points
- OK button
- Cancel button

Fill in the dialog (see the **Management > Renumber Points** section for more details) and click **OK** to renumber the specified range of points. If you check the **OVERWRITE Existing Points** checkbox, you will not be warned of any points that are overwritten during the renumbering process.

Import Points

You can use this to copy points from another coordinate file into the current coordinate file. If you click the **Import Points** button you will be asked to specify the coordinate file from which the points are to be imported. After specifying the import file name, you will use the **C&G Select Points from:** dialog to select which points are to be imported:



C&G Select Points from: <file name> dialog

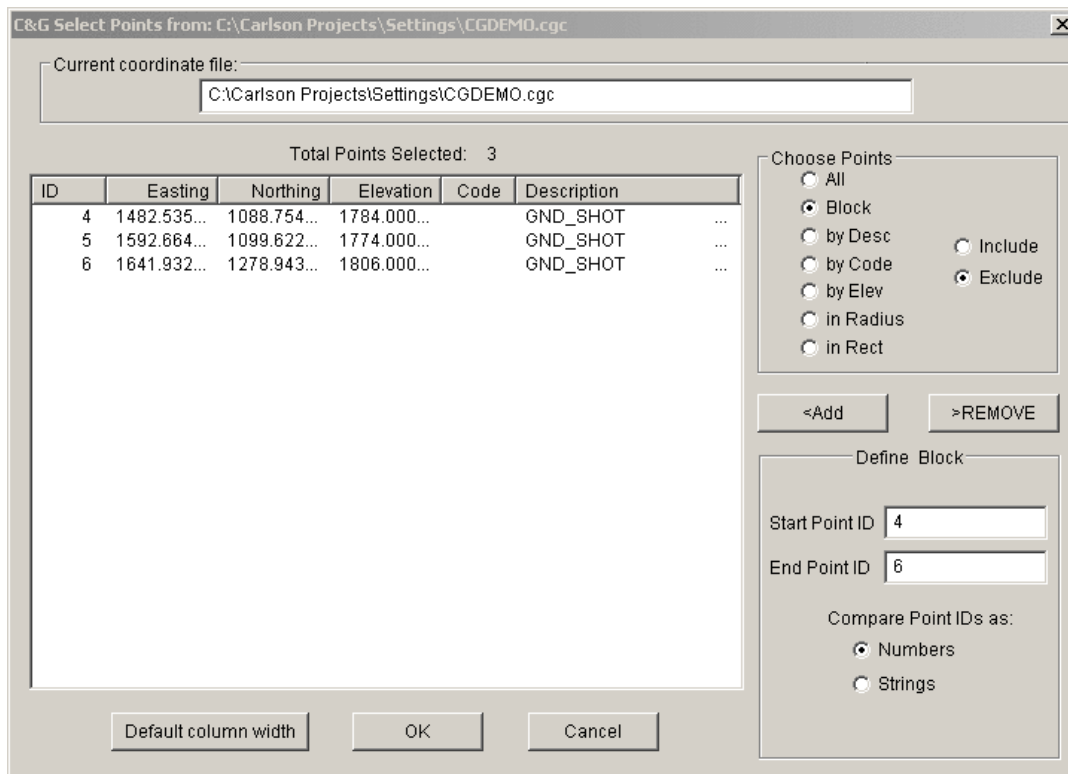
Choose Points section:

You can select any one of the methods you wish to use to choose the points by clicking one of the methods of point selection in the **Choose Points** section of the dialog. You may also specify whether you wish to **Include** or **Exclude** the points chosen. If you include the points, they will be added to the list from the coordinate file. If you exclude the points, the points chosen will be removed from the list of points previously **Included**. The method of choosing the points is very much like using the

Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt-group/Limits/Radius/Select]:

prompt. For example, if you choose the **Include** radio button and the **All** radio button then click on the **<Add** button, all the points in the coordinate file will be shown in the list on the left side of the dialog. If you then choose the **Exclude** radio button and the **Block** radio button, fill in the block of points you wish to remain in the list, then all but these points will be excluded from the list when you click the **>Remove** button (see the example dialog below).



When the points you wish to import are all in the list on the left, click OK.

CAUTION: *If the points that are being imported exist in the current coordinate file, they will be overwritten without warning!*

Export Points - click the **Export Points** button to copy points from the current coordinate file into another coordinate file. You will be asked to specify the file to export the points into then, similar to importing points, use the **C&G Select Points from:** dialog to select the points to be exported. Click OK in the **C&G Select Points from:** dialog to export the points.

CAUTION: *If the points that are being exported exist in the file they are being exported to, they will be overwritten without warning!*

Prompts

Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt-group/Limits/Radius/Select]: use one or more of the available methods to specify which points are to be deleted from the current coordinate file. Note: this prompt is only used if you have the **Use the Standard CGSurvey Command** radio button set.

Pulldown Menu Location: CG-Survey > Management > Point Manager

Keyboard Command: cg_pt_mngr

Prerequisite: One or more Coordinate files

Edit Coordinates (CGEditor)

The user can use the CGEditor to edit an existing coordinate file or create a new one. You can add and/or delete points and edit existing points. The CGEditor has many sophisticated editing tools that make editing fast and relatively easy. Please refer to the CGEditor section of the Tools menu for a detailed explanation of how to use the CGEditor.

Pulldown Menu Location: CG-Survey > Management > Edit Coordinates (CGEditor)

Keyboard Command: EDC, CG_EDIT_COORDS

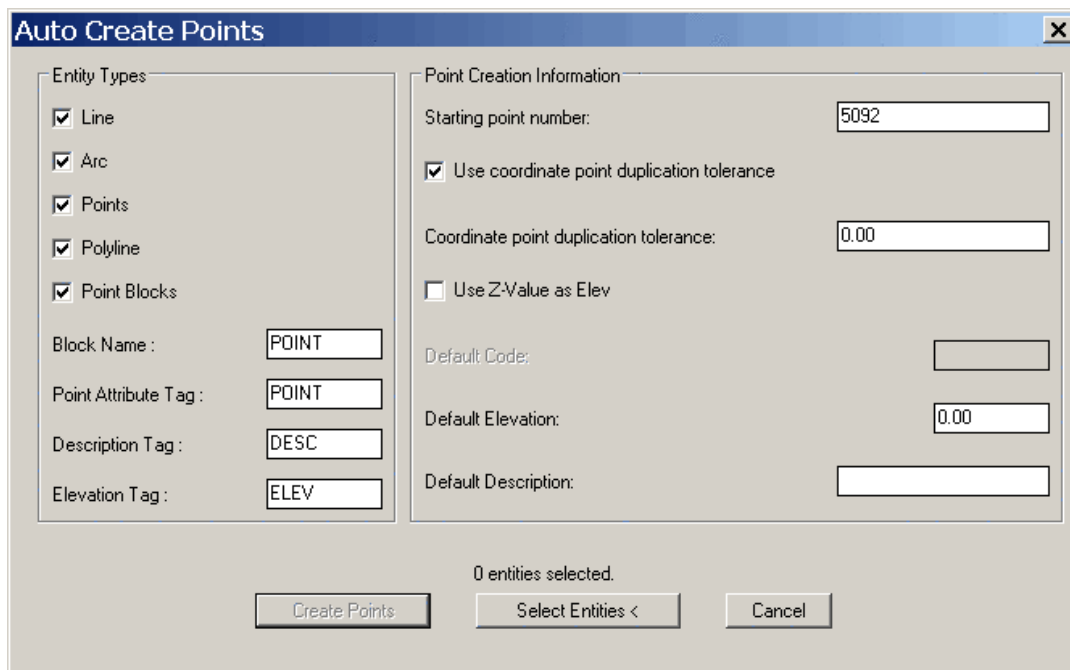
Prerequisite: None

Auto Create Points

The purpose of this feature is to create points in the current coordinate file and draw the associated point symbols using coordinate values extracted from existing drawing entities. These drawing entities may or may not have been created with CGSurvey. The user can automatically place C&G point symbols at the vertices, radius points, insertion points, etc. of selected lines, arcs, points, polylines, and point blocks. The coordinates of the newly created points are then saved in the currently open coordinate file.

Note: Unless point symbols are picked, the coordinates that are stored will be the coordinates of the CAD entity. In the case of point symbols, the point ID will be read and used to look up the proper coordinates in the current coordinate file.

After picking the Auto Create Points menu item, the Auto Create Points dialog box will appear:



Entity Types section

Select the entity types for which you wish to create C&G points. You can check any combination of the available entity types.

When you click the Select Entities button, specifying only certain entities allows you to window a large area but only have points created for the specified types of entities. You may also select individual entities or several groups of entities. After selecting the entities, click the Create Points button to create the points and save them to the current coordinate file.

Point Blocks

If you wish to have coordinates created for point blocks (or inserts) and you want the point ID, description and elevation to be set from information contained in the block, the block *must* have attributes that can be used to obtain

these values. When you choose Point Blocks, the following edit boxes in the dialog are activated and must be filled out:

Block Name: Specify the name of the blocks you wish to have points created for.

Point Attribute Tag: For the block entities chosen, specify the tag name for the attribute of the block contains the point ID. If no point ID attribute is found then the next sequential point ID will be used.

Description Tag: For the block entities chosen, specify which attribute of the block contains the description. If descriptions are ON and no description attribute is found then the default description will be used.

Elevation Tag: For the block entities chosen, specify which attribute of the block contains the elevation.

If elevations are ON and no elevation attribute is found: if the **Use Z-Value as Elev** is checked, then the Z value of the block insertion point will be used for the elevation of the newly created point; otherwise the specified default elevation will be used.

Point Creation Information section

Starting point number:

Use this to specify the starting point number. Specifying anything other than the next available point in the coordinate file as the starting point makes it possible that one or more existing points could be overwritten. However, whenever a situation arises that a point in the coordinate file may be overwritten, a dialog box will appear warning you of this and allowing you to decide whether to proceed with the overwrite or not.

Use coordinate point duplication factor:

If this box is checkbox Coordinate point duplication tolerance edit box is activated and you must enter a tolerance for determining coordinate point duplication. This is used to test if a new point that is about to be created is the same as a point already in the coordinate file. If the new point coordinates are within this tolerance the new point will not be created.

Use Z-Value as Elev

Select this box if the entities you select may have a Z value and you want that value used as the point's elevation.

Default Code

If point codes are turned ON, then this value is used as the default Code for all newly created points.

Default Elevation

If elevations are turned ON, then this value is used as the default Elevation for all newly created points.

Default Description

If descriptions are turned ON, then this value is used as the default Description for all newly created points.

Buttons

Select Entities <

Press this button to begin selecting the entities for which you wish to create coordinate points. The dialog box will disappear and you will be asked to use the normal entity selection methods to choose the entities to be used for point creation. Just press Enter at the Select Entities prompt when you are done. You will then be returned to the **Auto Create Points** dialog.

Create Points

After the entities have been selected, press this button to create coordinate points using the entities. Any existing C&G lines, arcs or polylines will be ignored since they already have points associated with them. Non-C&G lines, arcs and polylines will be converted to C&G lines, arcs and polylines.

Prompts

Fill in the dialog box as required (see above explanation).

Select Entities: use the normal entity selection methods to select the entities to use for creating points.

Pulldown Menu Location: CG-Survey > Mngmt

Keyboard Command: cg_acp

Prerequisite: Coordinate file.

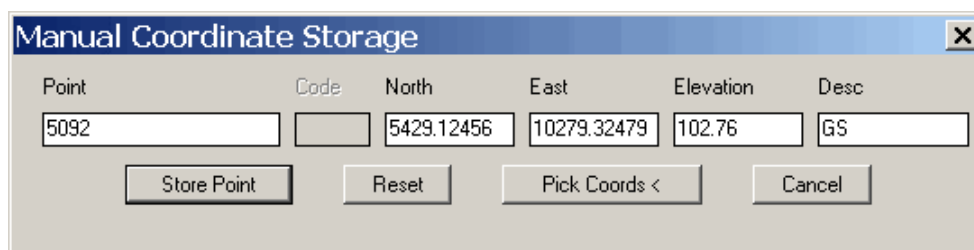
Manual Storage

This feature allows you to store points in a coordinate file by typing in the values for the point ID, code, northing, easting, elevation and/or description. You also have the option of using the mouse to pick the location coordinates on the screen.

When you choose the Manual Storage menu item:

If a coordinate file is not currently open, you will be prompted to open one.

Next, the following dialog box will appear:



Point	Code	North	East	Elevation	Desc
5092		5429.12456	10279.32479	102.76	GS

Buttons: Store Point, Reset, Pick Coords <, Cancel

Point

When the dialog box first appears, the point ID (**Point**) field defaults to the next available point ID as set on the General tab of the C&G Options dialog box. If you enter an existing point number in the point field and click on one of the other fields, the values associated with that point number will be retrieved from the current coordinate file and placed in the other fields. You may edit them if you wish.

Note: If you enter an existing point ID and alter any of the other fields associated with that point then save the point, **NO POINT OVERWRITE WARNING WILL BE GIVEN!**

North, East and Elevation

There are three different ways to enter coordinate values:

1. You can type in the coordinate values and elevation in the appropriate edit boxes.
2. You can duplicate a points values by entering a '+' sign and a point ID (example: +25) in the North field. When you click on another field, the coordinate values for the specified point will be automatically entered in the North, East and Elevation fields.
3. Or you can press the Pick Coords button. When you do this the dialog box disappears and you are prompted to pick a point on the screen. Once you have picked the desired point on the screen, the dialog box reappears with the coordinates of the selected point entered in the North and East fields.

Note: If you pick a C&G point, the coordinate values will be read from the coordinate file.

Note: The elevation field is only activated if Elevation is ON.

Code and Description

Enter the desired description in the description edit box.

Enter the point code in the edit box. The point code field is a 4 digit alpha or numeric code only used by

C&G. When present it can be used as a sorting tool in addition to the description table.

As an example: say that the description table number 25 is defined as 'Sanitary Manhole'. In addition to using 25 from the description table you also have used the code 'AB' for As-Built and 'DS' for design. Now you can build a selection set of all the description 25's, excluding all of the 'AB' codes and the selection set will contain only those points that are Design Sanitary Manholes.

Note: If descriptions are ON and **Get Description From Table** is checked, then if you enter an integer code in either the Code or Description field, that number will be used to lookup a description in the current description table. If a matching number is found it will be used for the code and the associated description will be used for the point's description. If there is no matching number found in the description table all fields will remain as entered.

Buttons

Store Point: When all the fields are entered, press the **Store Point** button to store the point in the coordinate file. If Auto Point Plot ON is checked on the Graphics tab in the C&G Options dialog, the point will be plotted as well.

Reset: This button clears all the fields in the dialog box and sets the point number to the next available point number.

Pick Coords <: click this button to use the mouse cursor to pick the point's coordinates on the screen.

Cancel: Press this button when done.

Prompts

Pick coordinates for point '<point ID>': use the mouse cursor to pick the coordinates for the point.

Pulldown Menu Location: CG-Survey > Mngmt

Keyboard Command: cg_man_coord_store

Prerequisite: Coordinate file

Delete

This feature allows the user to delete selected points from a coordinate file.

If a coordinate file is not open, you will be prompted to open one.

Select the points you wish to delete, either by picking with the mouse or entering the point sequence at the command line.

When point selection is complete, press Enter.

A dialog box will come up asking if you are sure you want to delete the points.

If you click OK, the points are deleted.

Note: Deleted points CANNOT BE RECOVERED unless point history is turned on. (See Carlson Configure)

Prompts

Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt_group/Limits/Radius/Select]: Use any of the point ID selection methods to specify which points you wish to delete from the current coordinate file.

Pulldown Menu Location: CG-Survey > Mngmt

Keyboard Command: cg_delete_coords

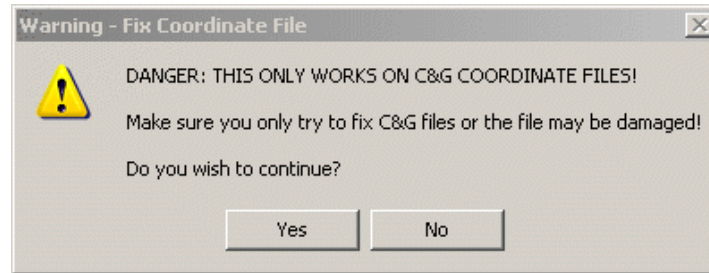
Prerequisite: Coordinate file.

Fix Coords

This feature should only be used if you wish to attempt to repair a damaged C&G coordinate file. A file can sometimes become damaged. when the computer is shut down prior to closing the file. It is wise to always keep a backup copy of your coordinate files in case a damaged one cannot be fixed.

Before running this command, try to open the file after closing any other C&G softer that may be running.

When you run this command you will first be warned that only C&G files should be fixed:



When the file dialog comes up, browse to the file you wish to attempt to repair.

If the fix is unsuccessful, you will get an error message otherwise the fix was successful.

Prompts

Use the file dialog to choose the file to be fixed.

Pulldown Menu Location: CG-Survey > Mngmt

Keyboard Command: cg_fix_coords

Prerequisite: Possibly damaged C&G Coordinate file

List

This feature allows the user to list all the information associated with selected points in the current coordinate file. If **Display at command line On** is checked on the Output tab of the C&G Options dialog box, the information for the selected points is displayed at the command line (press <F2> to view it). Otherwise the output is sent to the print file. If a coordinate file is not open, you will see a file dialog allowing you to open one.

Note: To print and/or view the print file after listing the coordinates, choose **Print/View Print File** from the CG-Survey > File menu. This will open a text editor with the print file as the current file. You may view the file or print it using the text editor.

Pulldown Menu Location: CG-Survey > Mngmt

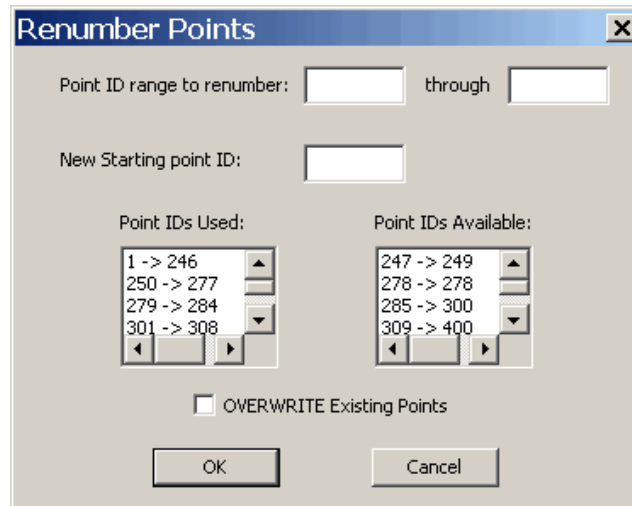
Keyboard Command: cg_list_coords

Prerequisite: Coordinate file.

Renumber Points

This feature allows you to renumber the point IDs of a range of points in the current coordinate file. The point IDs will be renumbered sequentially. If the renumbered points have been previously plotted to the drawing, the points will be redrawn to reflect the changed point IDs.

When you choose this menu item, the following dialog box appears:



Point ID range to renumber:

Specify the range of point IDs you wish to renumber. Use the Point IDs Used and Point IDs Available lists to help you determine the appropriate range.

New Starting point ID:

Specify the new starting point ID for the range specified. The specified range of points will be renumbered sequentially starting with the New Starting point ID.

OVERWRITE Existing Points

If you check this check box and, in the process of renumbering the points, the new point ID is the same as an existing point in the coordinate file, the point will be overwritten. However, if you do not check this checkbox and a possible overwrite is detected, you will be informed of the possible overwrite and required to check this checkbox before proceeding. If you do not wish to overwrite existing points, either re-specify the New Starting point ID or click the Cancel button.

To proceed with the renumbering click the **OK** button.

Note: this feature will only renumber points in a coordinate file in which all point IDs are numeric. Thus you can renumber the points in Carlson and C&G alphanumeric coordinate files only if all the point IDs are numeric.

Prompts

Fill in the dialog box as specified above.

Pulldown Menu Location: CG-Survey > Mngmt

Keyboard Command: cg_renumber_pts

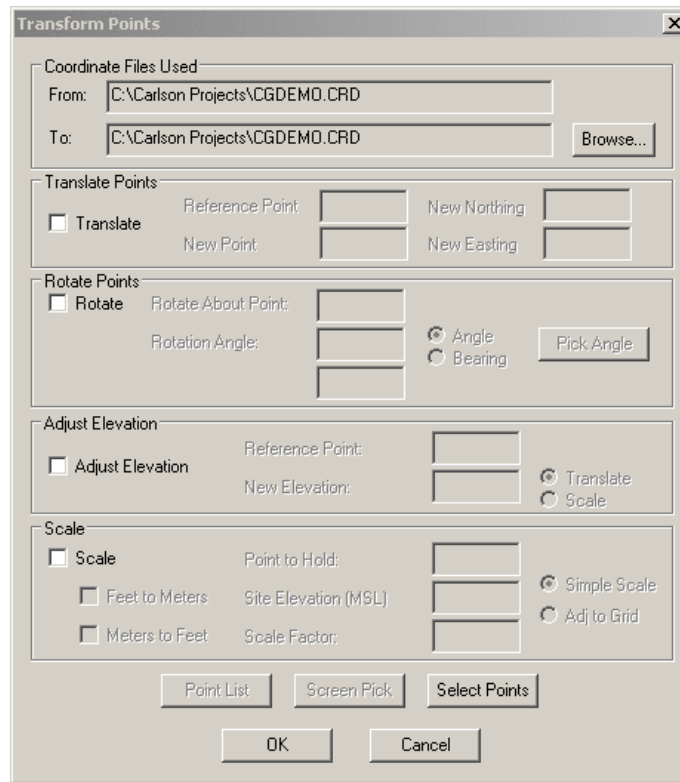
Prerequisite: Coordinate file

Transformations

Combined Transformations

The Combined Transformations feature allows the user to translate, rotate, adjust elevation and/or scale the selected points in a specified coordinate file. The user may also specify whether the transformed coordinates replace the values in the current coordinate file or are saved to another coordinate file.

The **Combined Transformations** menu item brings up the **Transform Points** dialog box. This dialog is used to configure the transformations that will be applied.

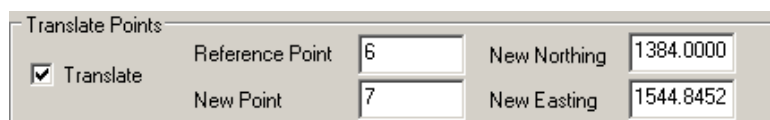


To begin the process, in the **Coordinate Files Used** area, choose the coordinate file into which the transformed points are to be stored. If you wish to store them in the current coordinate file, you can go on to the next step. However, if you wish to have the transformed points stored to a coordinate file other than current coordinate file, click the **Browse...** button and use the file dialog to choose the desired destination file.

Next, check the checkboxes for each of the transformations you wish to apply. Then, for each type of transformation to be applied, fill in each item of data in that area of the dialog.

Translate Points

To translate the points check the **Translate** checkbox, then fill in the data in the edit boxes in this section of the dialog.



The amount that the selected points are translated in the North and East directions is determined by the difference between the northing and easting of the Reference Point and the coordinates specified in the New Northing and

New Easting edit boxes.

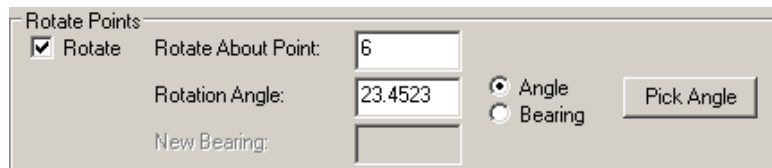
Reference Point specifies the point in the current coordinate file that is to be used as the reference point for the translation. All the selected points are translated in the same manner as the **Reference Point**. The reference point will be translated by the difference between its current coordinates and those specified in the **New Northing** and **New Easting** edit boxes. You can fill in the **New Northing** and **New Easting** edit boxes directly or you can enter a point ID in the **New Point** edit box. Assuming the point ID entered is found in the coordinate file, the coordinates read from the coordinate file will be placed in the **New Northing** and **New Easting** edit boxes. You can edit these coordinates or leave them as they are.

Rotate Points

If the **Rotate** check box is checked, the selected points will be rotated according to the specifications in the **Rotate Points** area. Rotation defaults to rotation by an angle but can be changed by merely clicking on the **Bearing** or **Angle** radio button.

Use the **Rotate About Point** edit box to specify the point in the current coordinate file about which the selected points will be rotated.

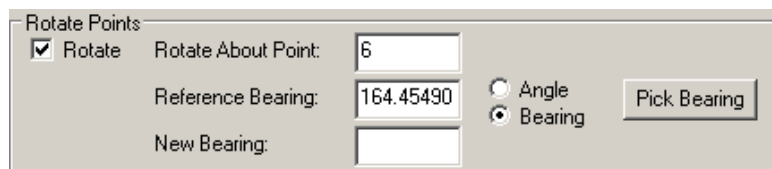
Rotating by an angle



The screenshot shows the 'Rotate Points' dialog box. The 'Rotate' checkbox is checked. The 'Rotate About Point' field contains the value '6'. The 'Rotation Angle' field contains '23.4523'. The 'Angle' radio button is selected, and the 'Bearing' radio button is unselected. A 'Pick Angle' button is visible on the right side of the dialog.

To rotate by an angle click the **Angle** radio button. Next, type in the appropriate angle in the **Rotation Angle** edit box or click the **Pick Angle** button and pick the desired angle on the screen.

Rotating by Bearing



The screenshot shows the 'Rotate Points' dialog box. The 'Rotate' checkbox is checked. The 'Rotate About Point' field contains the value '6'. The 'Reference Bearing' field contains '164.45490'. The 'Bearing' radio button is selected, and the 'Angle' radio button is unselected. A 'Pick Bearing' button is visible on the right side of the dialog.

To rotate by bearing, click the **Bearing** radio button then type in the appropriate values in the **Current Bearing** and **New Bearing** edit boxes or click the **Pick Bearing** button and pick the desired bearing on the screen.

Note: Bearings must be specified using qdd.mm.sss notation, where q is the quadrant (1 = NE, 2 = SE, 3 = SW, 4 = NW), dd is degrees, mm is minutes and sss is seconds. Seconds can be specified to 0.1 seconds if desired.

Adjust Elevation

If the **Adjust Elevation** box is checked, the elevations of the points will be adjusted according to the specifications in the **Adjust Elevation** area. The type of elevation adjustment can be specified by clicking on the **Translate** or **Scale** radio button.

To Translate elevations

The elevations of the selected points will be translated by the difference between the **Reference Point** elevation and the value entered in the **New Elevation** edit box. When you enter a point ID in the **Reference Point** edit box and click on another edit box the **New Elevation** edit box will be filled out with the current elevation of the reference point.

To Scale elevations

The elevations of the selected points will be scaled by the value entered in the **Multiplication Factor** edit box.

Scale

If the Scale check box is checked the northings and eastings of the selected points will be scaled according to the specifications in the Scale area of the dialog.

Meters to Feet and Feet to Meters

You can scale the coordinates to convert feet to meters or meters to feet by checking the appropriate check boxes. Using this form of scaling disables the other items in this section of the dialog box.

Other types of scaling:

The **Point to Hold** is a point in the current coordinate file that will be used to obtain the reference coordinates for the application of the specified scaling factor to the selected points.

Simple Scale - if you choose simple scaling it will calculate the scaled differences in northing and easting between the **Point to Hold** and each of the points selected for scaling. This scaled difference is found by calculating the difference between the coordinates of the **Point to Hold** and those of a given selected point and multiplying that times the specified **Scale Factor**. This scaled difference is then added back to the northing (easting) of the given selected point.

Adjust to Grid - this scaling method uses the **Site Elevation (MSL)** and the **Projection Table Factor** to adjust the northings and eastings of the selected points to grid coordinates

Scale

Scale Point to Hold:

Feet to Meters Site Elevation (MSL) Simple Scale

Meters to Feet Projection Table Factor: Adj to Grid

Methods of Specifying Point IDs for the Various Transformations

When specifying a point ID in the transformation data (for example to specify the **Reference Point** when the **Translate** checkbox is checked), you may select points using any one of the three options listed below:

1. Type the point ID directly into the edit box provided.
2. Point List: click the **Point List** button to bring up the **Choose Point Blocks** dialog. The left pane shows a listing of all the points found in the current coordinate file. Highlight the desired point in the **Points Available** list then click the **Add >** button and the point selected will be displayed in the **Points Chosen** list. In every case you are only allowed to choose a single point. Once you are satisfied with the point chosen click the **OK** button.

Choose Point Blocks

Coordinate file: C:\Carlson Projects\CGDEMO.CRD

Points Available:

Point Id	Northing	Easting
1	1311.55	1156.50
2	1195.62	1178.23
3	1179.32	1349.22
4	1088.75	1482.54
5	1099.62	1592.66
6	1278.94	1641.93
7	1384.00	1544.85
8	1451.02	1439.06
9	1369.51	1370.96
10	1431.09	1252.14
11	1398.49	1186.93
12	1277.08	1299.77
13	1301.71	1505.68
14	1221.32	1513.18

Add >

< Remove

Points Chosen:

Point Id	Rad	Seq ID	Northing
6		END	1278.94

OK Cancel

3. Screen Pick: when you click the **Screen Pick** button the **Transform Points** dialog disappears and **Choose a point:** prompt is displayed at the command line. You may type a point ID or pick a point symbol from the drawing.

Selecting Which Points Will be Transformed

At any time prior to clicking the **OK** button you may choose the points to be transformed. To do this click the **Select Points** button. The Transform Points dialog will disappear and you will be prompted to choose the points:

Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt-group/Limits/Radius/Select]: - use any of the available methods to specify the

points to be transformed. When done specifying the points press Enter until the Transform Points dialog reappears.

Transforming the points

To transform the selected points, click the OK button. The points will be transformed and saved to the specified coordinate file.

Prompts

Fill out the **Transform Points** as described above.

When the Select Points button is clicked the following prompt appears:

Choose initial points for base selection set from coord file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt-group/Limits/Radius/Select]: - use any of the available methods to specify the points to be transformed. When done specifying the points press Enter until the Transform Points dialog reappears.

Pulldown Menu Location: CG-Survey > Management > Transformations

Keyboard Command: cg_transformations

Prerequisite: Coordinate file

Best Fit Transformation

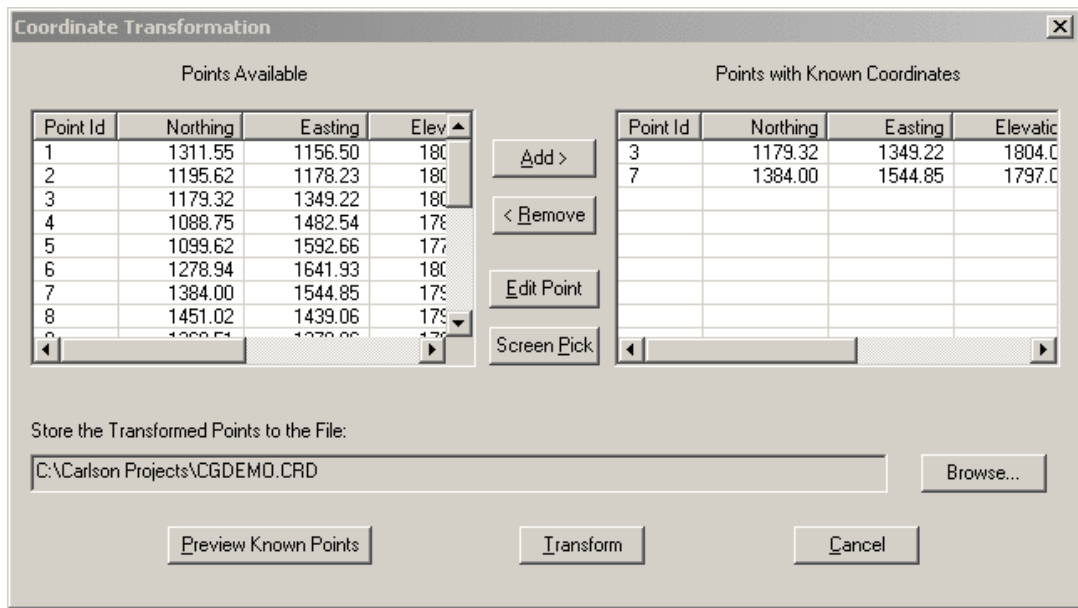
Best Fit Transformation is used to transform the coordinates in the current coordinate file using a "rubber sheet" method of transformation.

First the user must .

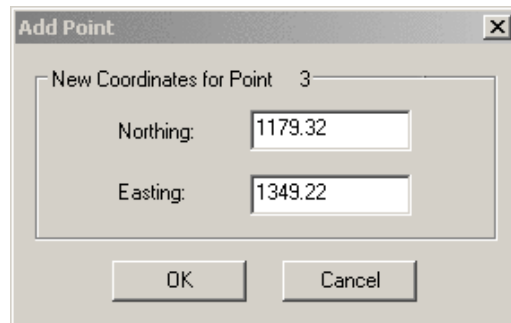
Add points from the coordinate file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt-group/Limits/Radius/Select]: specify which points are to be transformed using one or more of the available selection methods.

Next, in the **Coordinate Transformation** dialog the user must choose at least 2 points in the coordinate file that are known (or "fixed") points



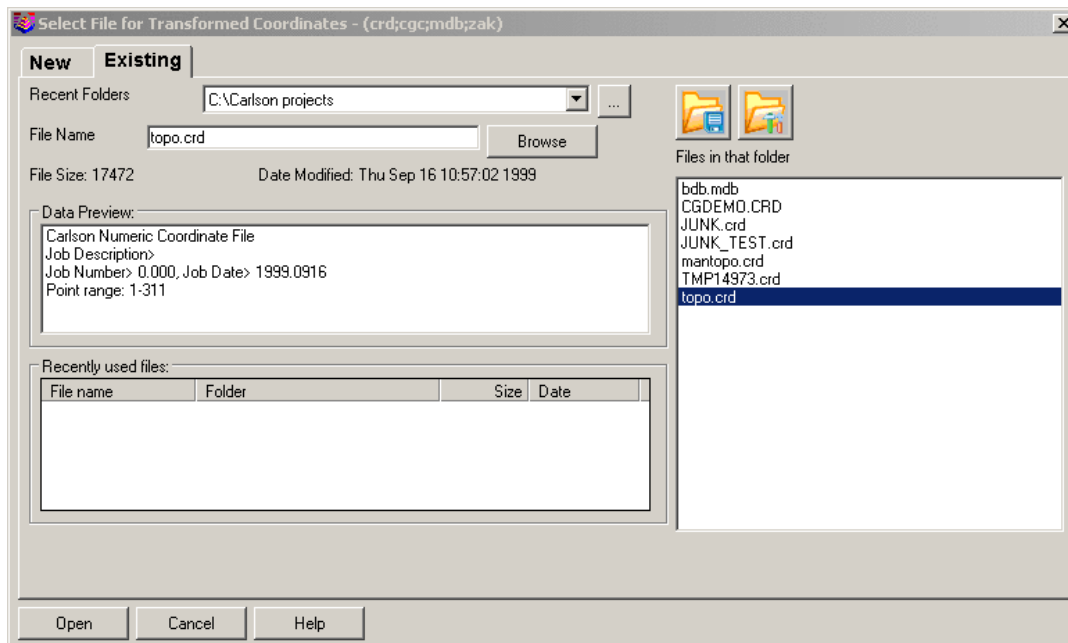
To specify a fixed point, highlight it in the **Points Available** list then press the **Add >** button to copy it to the **Points with Known Coordinates** list on the right. Or, if you wish, you can also specify a known point from the drawing by clicking the **Screen Pick** button and picking a point from the screen or typing a point ID at the command line. After choosing a known point, the following dialog will appear:



The **Add Point** dialog allows you to change the current coordinates of the known point or accept the current coordinates. When done specifying the coordinates of the known point, click the **OK** button

After a point has been placed in the **Points with Known Coordinates** list on the right, you can edit the values you entered by highlighting the incorrect point and clicking the **Edit Point** button. Or, if you wish, you can remove an incorrect point from the right hand **Points with Known Coordinates** list by highlighting it and clicking the **< Remove** button.

After specifying all the known points, you must specify which coordinate file will be used to store the transformed points. If you wish to use the current coordinate file you need do nothing. If you wish to write the transformed points to a different coordinate file than the one listed in the **Store the Transformed Points in the File:** edit box, click the **Browse...** button and use the file dialog (see below) to specify a new or existing coordinate file. When done choosing a coordinate file, click the **Open** button in the file dialog.



Click the **Transform** button in the **Coordinate Transformation** dialog to cause the transformed coordinates to be calculated and stored in the specified file

Prompts

Add points from the coordinate file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt-group/Limits/Radius/Select]: specify which points are to be transformed using the typical C&G Selection method.

Follow the instructions above to fill out the **Coordinate Transformation** dialog box.

Pulldown Menu Location: CG-Survey > Mngmt > Transformations

Keyboard Command: cg_crd_trns

Prerequisite: Coordinate file

Copy Coordinates

This feature allows the user to copy a selected set of points from the current coordinate file to itself or to another coordinate file and, optionally, increase or decrease the point ID by a specified number.

First you must choose the points to be copied:

Add points from coordinate file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt-group/Limits/Radius/Select]: use one of the available methods to specify the set of points to be copied.

Next, you must specify what value to add to or subtract from the point IDs to form the new point IDs. If you press <Enter> the default is to leave the IDs the same.

Value to add or subtract from point numbers <0>: press Enter to leave the point IDs as they are or enter a positive or negative number.

Next you must specify the coordinate file to which the points are to be copied.

Store coordinates in [Current-file/Existing-file/New-file] <C>: type "C" and Enter or just Enter to copy the specified points into the current coordinate file. Type "E" and Enter to choose an existing coordinate file into which to copy the points or Type "N" and Enter the copy the points to a new file. If you choose either an existing or new file you will use a file dialog to specify the file to use.

The selected points will then be copied to the specified file.

Prompts

Add points from coordinate file. (Enter when done)

[All/Block/Code/Desc/Elev/Pt-group/Limits/Radius/Select]: use one of the available methods to specify the set of points to be copied.

Value to add or subtract from point numbers <0>: press Enter to leave the point IDs as they are or enter a positive or negative number.

Store coordinates in [Current-file/Existing-file/New-file] <C>: type "C" and Enter or just Enter to copy the specified points into the current coordinate file. Type "E" and Enter to choose an existing coordinate file into which to copy the points or Type "N" and Enter the copy the points to a new file.

Pulldown Menu Location: CG-Survey > Management > Transformations

Keyboard Command: cg_move_coords

Prerequisite: Coordinate file

Description Tables

Description tables have several purposes. They can be used as a shortcut method of using point codes or numeric descriptions to automatically set point descriptions when points are saved to a coordinate file. (Descriptions have a similar functionality when used in conjunction with the CGEditor)

Note: the description table is only used in conjunction with saving points to the coordinate file if Descriptions are ON and **Get Description From Table** is checked on the General tab of the C&G Options dialog.

Description tables allow the user to specify many details regarding the appearance of a plotted point when using **Auto Map > Draw** or **Trav > Reduce**, or anytime points are being plotted.

Note: the description table is only used in conjunction with drawing points if Descriptions are ON and **Use description table for point plotting parameters** is checked on the Graphics tab of the C&G Options dialog.

A description number acts as an index into the description table.

When saving a point to the coordinate file and a number is used in a description field, the description table will be searched for that number. If that description number is found, the number in the description field will be replaced with the description in the description table. In the case of C&G coordinate files, the number will be moved to the point code field. If no match is found, the number will remain in the description field.

When plotting points, the description for the point being drawn is compared to the descriptions found in the description table. If a match is found then the point is plotted using the parameters specified in that description table entry.

The items in this menu allow you to create and manipulate description tables. You can create a new empty

description table and edit it. You can edit an existing description table. You can set or close the current description table and set a different default description table to be used in new drawings.

Pulldown Menu Location: CG-Survey > Mngmt

Keyboard Command: None

Prerequisite: None

Create

This feature allows you to create a new description table. A dialog box will prompt you to name the description table to be created.

Once you have specified the description table to create, you will be allowed to edit the new, empty description table. See Description Table > Edit for a detailed description of how to edit a description table.

Prompts

Use a file dialog to specify the name and location of the new description table and the Edit Description Table dialog to create the table entries.

Pulldown Menu Location: CG-Survey > Mngmt > Description Tables

Keyboard Command: cg_create_desc_table

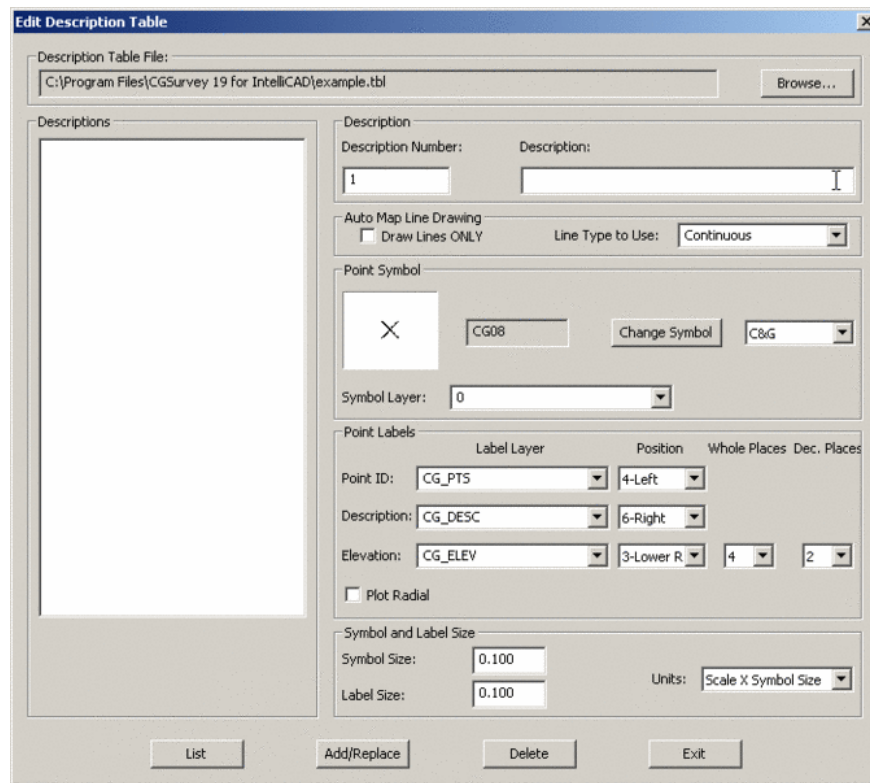
Prerequisite: None

Edit

This feature allows you to edit an existing description table. The following edit dialog box will appear:

Note: Editing a description table does not make it the current description table for the current drawing. To make a description table the current one you must choose the **Set Current** menu item in the **Description Table**.

Caution for CG-SURVEY for DOS users: You may open the older format CG-DOS Description Table. However, when opened, the CG-DOS description table will be converted to the current format and once converted, the description table cannot be converted back to or used by the CG-SURVEY for DOS program.



Description Table File:

Displays the name and location of the description table currently being edited. Pressing the **Browse** button brings up the **Choose a Description Table to Edit** file dialog box allowing you to select a description table to edit.

Note about the Browse button: If you are in the editor just after using the create menu item to create a new description table, you should not use the Browse button.

Descriptions list

This list contains a list of all the descriptions in the current description table. You may scroll through the list using the scroll bar on the right or use the scroll bar on the bottom to view the complete description. The list contains one description per line. By clicking on a description in the list its properties are displayed to the right of the list. The list itself contains some of the description's properties but you must click on the specific description you are interested in to view all its specifications. Each row in the list consists of first the description number then the description itself then the symbol and symbol and label height.

Description section

Description Number: This number acts as an index to the description table.

Description: The description you wish to set parameters for.

Auto Map Line Drawing section

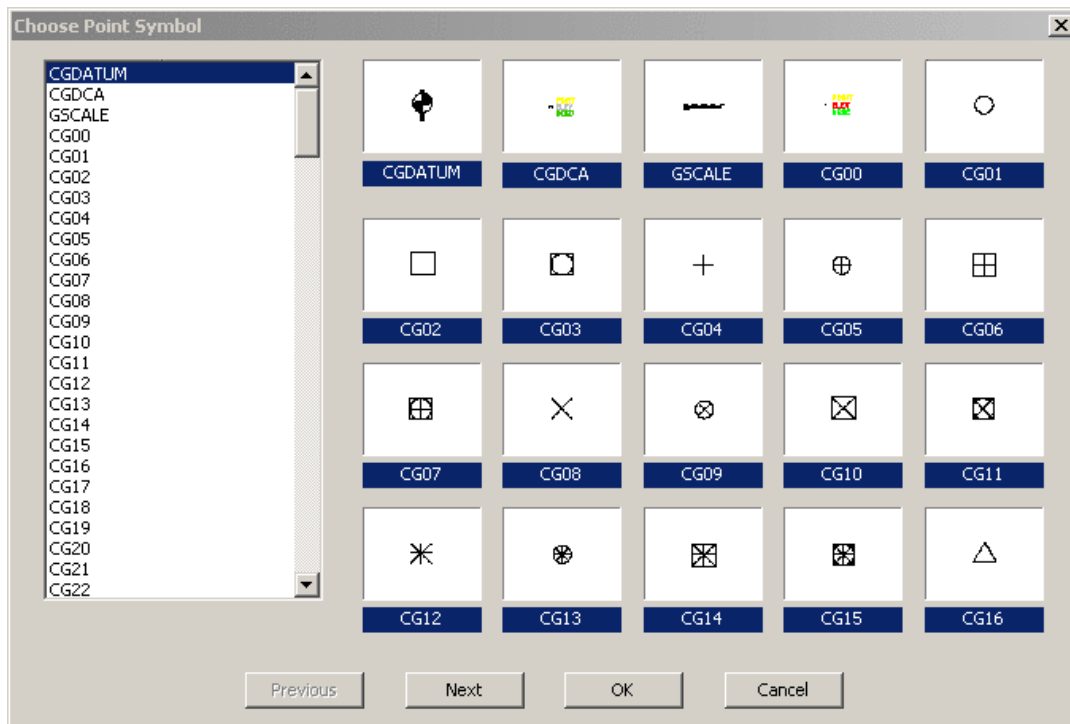
If you check the **Draw Lines ONLY** checkbox then during Auto Mapping (see Draw > Auto Map > Draw) points with this description will not have the point symbol drawn but will be connected by lines if the line drawing codes are used. The **Line Type to Use** drop down box allows you to choose one of the currently loaded line types for the

lines drawn between points having this description.

Point Symbol section

Symbol: The drawing in the box on the left side indicates which point symbol is currently specified for this description. The text next to the symbol drawing is the name of the symbol minus the .dwg ending. This is the symbol to be used when plotting a point having a description matching that specified in this description entry. You may specify any valid block name by first selecting the type of symbol you wish to use: either Carlson or C&G. You can specify any standard C&G or Carlson symbol supplied with the software or create your own custom point symbols. Custom point symbols must be located in either the users Sup directory with the Carlson symbols or with the C&G symbols in Sup\CGPTSYM generally found under C:\Documents and Settings\\Application Data\Carlson Software\\<CAD version> directory.

You may press the Change Symbol button to view and/or select the desired symbol.



Symbol Layer: The drawing layer that the point symbol will be plotted on for any point having a description matching that specified in this description table entry.

Point Labels section

Point Label Positions

The entries in this area refer to the plotting of Point ID, Description and Elevation labels around a point symbol.

Layer

The Point ID, Description and Elevation layer names can be specified. If none is specified then the current layer will be used.

Position

You can assign the Position of the point number, elevation and description labels in relation to the point symbol.

Valid positions are 0 through 9 based on the numeric keypad on your keyboard. Consider 5 as the location of the center of a point symbol. Labels can be placed around the point symbol just as the other numbers surround 5. You identify the location where you want the label to be placed around the point symbol by selecting the number on the numeric keypad that corresponds to that relative location. The relative positions are also indicated in the list and can be picked directly from the list.

Position 0 - indicates that you do not want the label to be plotted.

Position 5 - can only be used for the elevation label. (If you select position 5 for any other label it will be treated as Position 0). If you select position 5 for the elevation label the whole number portion of the elevation will be plotted to the left of the center of the symbol and the decimal portion of the elevation will be plotted to the right.

Whole Places - can be specified for the elevation label only.

Decimal Places - can be specified for the elevation label only.

Plot Radial

If Plot Radial is checked, the point labels will be plotted radially from the symbol's center at the Position specified.

If plot radial is not selected, point labels will be plotted horizontally.

Symbol and Label Size section

Symbol Size: The size of the symbol to be plotted for a point having a description matching that specified for this description table entry.

Label Size: The height of the point label text. This refers to labeling symbols with point number, elevation and description.

Units: Units can be set to **Scale X Symbol Size** or **Literal**.

Scale X Symbol Size indicates that the values for **Symbol Size** and **Label Size** will be multiplied by the drawing scale (Specified on the Drawing Settings tab of the C&G Options dialog) to determine the size of the symbol and/or label in actual drawing units.

Literal indicates that the specified **Symbol Size** is in actual drawing units and should not be scaled.

Note: If the **Units** for a given description is set to **Scale X Symbol Size**, then the symbol size and label height on the printed drawing are interpreted to be in inches if drawing units are set to FEET on the General tab of the C&G Options dialog and centimeters if drawing units are set to METERS or METRES.

Buttons

List: clicking this button causes the description table to be printed as a report on the command line and to the print file.

Add/Replace: click this button to save your changes when you complete a new description or edit an existing description.

Delete: click this button to delete the highlighted description. If you delete a description, it cannot be recovered and you will not be allowed to cancel the deletion.

Exit: When finished editing, click this button.

Prompts

Use the Edit Description Table dialog to enter or modify descriptions and their drawing parameters.

Pulldown Menu Location: CG-Survey > Mngmt > Description Tables

Keyboard Command: cg_edit_desc_codes

Prerequisite: An existing C&G description table or a newly created one (see Create)

Set Current

Allows the user to select the current description table using a file dialog. The selected description table will be active for the current drawing. When Set Current is used the description table name and location are saved with the drawing settings.

Prompts

Use a file dialog to choose the current file.

Pulldown Menu Location: CG-Survey > Description Tables

Keyboard Command: cg_set_desc_table

Prerequisite: Existing C&G Description table

Close Current

Closes the current description table. This is saved with the drawing as part of the CG_SETTINGS.

Pulldown Menu Location: CG-Survey > Mngmt

Keyboard Command: cg_close_desc_table

Prerequisite: None

Set Default

Allows the user to select the default description table using a file dialog. The selected description table will be saved to the CGSURVEY.OPT file for the current user and used to set the default settings for any new C&G drawings created by the current user.

Prompts

Use a file dialog to choose the default file.

Pulldown Menu Location: CG-Survey > Description Tables

Keyboard Command: cg_set_default_desc_table

Prerequisite: Existing C&G Description table

Convert to SurvCE FCL file

This feature allows you to convert a C&G description table to a SurvCE feature code list or FCL file.

If you are using SurvCE and wish to import your C&G description table for use with SurvCE use this feature to create the FCL file then upload it to the SurvCE data collector FCL directory.

If a C&G description table is not currently open you will be asked to choose which file you wish to convert using a file dialog box.

Next, using file dialog, you will be asked to specify the name of the FCL file to create.

When you click OK after specifying the FCL file name, the conversion will take place.

Prompts

Use file dialog boxes to pick the C&G description table to convert and to specify the name of the FCL file to create.

Pulldown Menu Location: CG-Survey > Mngmt > Description Tables

Keyboard Command: CG_CONVERT_DESC_TO_SURVCE_FCL

Prerequisite: Existing C&G description table (*.tbl) and its index file (*.tbx).

Point Groups

Point groups were formerly called Batch Point Files or point files. These files are text files with the extension pts.

These features allow the user to create or edit a point group. A point group is simply an ASCII text file that contains a list of point IDs that are in a specific sequence. The points in a point group can describe a tract of land, a road centerline, a utility line, a group of lots in a subdivision, etc. - anything that can be defined by a series of points. Point groups can also contain the PC radius point and PT for horizontal curves as well as vertical curve information.

An example of a point group file:

If you view a point group file in a text editor like notepad you will see something like this example

Note: the text in square brackets does not appear in the file itself - it is only used to clarify this example:

```
LOT 1 [Subgroup description]
1 [Point 1]
23 [Point 23, PC]
+48 [Clockwise radius point 48]
49 [Point 49, PT]
50 [Point 52]
1 [Point 1, back to starting point]
LOT 2 [Subgroup description]
12 [Point 12]
24 [Point 24]
65 [Point 65]
70 [Point 70]
12 [Point 12, starting point]
```

The above example illustrates a point group with two subgroups. Each subgroup defines a lot. The last point in each subgroup is optional - you don't need to close the lot boundary by entering the starting point twice.

Pulldown Menu Location: CG-Survey > Mngmt

Keyboard Command: None

Prerequisite: Coordinate file

Create

There are two ways to create a new point group: you can use this command or you can use the CGEditor. Previous C&G users may prefer to use this command but the CGEditor allows the user the ability to view and edit the point group as it is being created.

If a coordinate file is not open, you will be prompted to open one using a file dialog.

Once the coordinate file is open, the point group file dialog can be used to specify the name of the point group file you wish to create.

Subgroup description <Enter when done>: Enter the subgroup description.

Specify points for subgroup <filename>:

[Block/Code/Desc/Elev/Indiv/liMits/Radius/Vertical_curve] <pick polyline>:

Use any of the available methods, including picking a polyline in the drawing, to specify the point IDs of the points in the subgroup. Remember that, within a given subgroup, you are defining a specific shape or line and thus the points need to be entered as an ordered sequence that properly defines the lot, alignment, etc.

Repeat the steps outlined above until all subgroups and their points have been entered.

To end the command and create the point group file, press Enter twice after specifying the last point in the last subgroup.

Using a Polyline to specify a group of points

If you pick a polyline, the coordinate file is searched for points having northings and eastings that match the x and y coordinates of the vertices of the polyline. If none of the points in the coordinate file match the polyline vertices, then no points are added to the points in the current subgroup. Any points that match are added to the subgroup points and you are prompted for the next point in the subgroup. You may continue using any of the methods of specifying points, including picking other polylines.

Entering a Curve

First type "I" and enter to enter individual points. As you are specifying the individual points in the subgroup you can specify a curve by entering the radius point ID immediately after entering the ID of the PC. The radius point must be indicated by preceding its point ID with a plus sign for a clockwise curve or a minus sign for a counterclockwise curve. The next point ID you enter is assumed to be the PT.

Vertical Curves

If you are creating a point group to define a road alignment, you may wish to enter vertical curve information so that both the horizontal and vertical alignments are defined.

Note: You may find it more convenient to use the CGEditor to enter vertical curve information.

To do this type "V" and Enter at the command prompt. You will see the following prompt:

Vertical Curve 1

[Next/slope-In/slope-Out/Length/pvi-Station/pvi-Elevation]:

For the first vertical curve you must enter five curve components. Enter these five components by typing the capitalized letter representing the component that you wish to specify, then press Enter. You will be prompted to

enter the component specified.

You must enter a value for each of the following five required fields:

The slope-In

slope-Out

Length of the vertical curve

pvi-Station

pvi-Elevation.

Entering succeeding vertical curves

After entering the information for the first vertical curve, enter <N> for Next. Since the slope in and PVI elevation are determined by the previous vertical curve information, so you need only specify three fields for any additional curves:

slope-Out

Length

pvi-Station

Use Previous and Next to enter and/or change the vertical curve information. You may enter as many as fifty vertical curves. You can press the F2 key at any time to view the prompt history screen, then use the scroll bar on the right to view the entire data entry sequence.

Multiple Subgroups

To place more than one subgroup in a single point group, press Enter when asked to select another point for the current subgroup. This ends input for the current subgroup.

At the **Subgroup description <Enter when done>**: prompt, enter the name of the next subgroup and go on to enter a new series of points, including both horizontal and vertical curve information as needed.

Continue to enter subgroups of points by repeating these steps until all subgroups have been entered.

When you have entered all the subgroups, press Enter until you get the Subgroup Description prompt.

Press Enter at the Subgroup Description prompt to end the command and create the point group file.

Prompts

Subgroup description <Enter when done>: Enter the subgroup description.

Specify points for subgroup <filename>:

[Block/Code/Desc/Elev/Indiv/liMits/Radius/Vertical_curve] <pick polyline>: use one or more of the available methods to specify points in the current subgroup.

Vertical Curve 1

[Next/slope-In/slope-Out/Length/pvi-Station/pvi-Elevation]: specify which element of the vertical curve you wish to enter or type "N" and Enter to begin entering the next vertical curve.

Vertical Curve ##

[Next/slope-Out/Length/pvi-Station]: after entering the data for the first vertical curve the following curves are controlled by the initial curve. Thus the prompt changes.

Pulldown Menu Location: CG-Survey > Mngmt > Point Groups

Keyboard Command: cg_create_bpf

Prerequisite: Coordinate file

Edit

The Edit Point Groups feature allows you to use the CGEditor to edit/create an existing point group file.

CGEditor General Information

The CGEditor is an integral part of preparing files for use in C&G applications. The CGEditor is a very powerful tool. You can open multiple data files of any supported file type and edit the files as needed. The CGEditor has a full complement of tools for searching and replacing and navigating within a file. It will also allow you to cut or copy records from one file and paste them into another file in order to merge files, move data between phases of a job, etc.

The CGEditor can create and/or edit six types of data files used by C&G:

Raw Data Files

Raw data files contain information pertaining to a field traverse. Raw data files are typically downloaded from the data collector and converted to the C&G raw data file format. These files have the extension .CGR.

Map Check Files

Map Check files contain bearing, distance and curve information and are typically used to calculate the closure of a deed description. These files have the extension .CGM.

Cross Section Files

Cross Section files contain one or more cross sections identified by their station along the alignment. Each cross section record has the percent grade defined for its left and right slopes. Following the "Station" record are several "Point" records containing the elevations and offsets of the points along the cross section. Cross section files consist of a pair of files; the main data file has the extension .CEW and the index file has the extension .CEX.

Template Files

Template files are merely cross section files that represent a standard cross section and can be used to generate other cross section files. However, unlike cross section files, template files use an integer ID instead of a station to uniquely identify each template. Like cross section files, the percent grade is defined for the left and right slopes of each template and there are a set of "Point" records specifying the template elevation at a given offset. The centerline elevation at offset 0.00 is typically set to 0.00. Template files consist of a pair of files; the main data file has the extension .CTP and the index file has the extension .CTX.

Point Group Files

Point Group Files are simply a list of point numbers that can define a group of points, a lot or parcel, or an alignment. These are ASCII files and have a .PTS extension.

Coordinate Files

CGSurvey supports many different coordinate file formats:

C&G .CRD/.IDX - C&G numeric coordinate files

C&G .CGC/.CGX - C&G alpha-numeric coordinate files
Carlson .CRD - Carlson coordinate file format, numeric and alpha-numeric
Simplicity .ZAK - Simplicity coordinate file
LDT - MDB - Land Desk Top coordinate file

Note: for further and complete information on using the Edit Raw File see the chapter on CGEditor in the Tools section.

Pulldown Menu Location: Management\Point Groups\Edit

Keyboard Command: BPF, CG_EDIT_BPF

Prerequisite: Open Raw File

CGTopo

Topographic Settings

Allows you to view or change the **Topographic Settings**. See the **Topography** tab section of the **CG Options..** menu item of the **Tools** menu.

Pulldown Menu Location: CG-Survey > Topo

Keyboard Command: cg_cont_setup

Prerequisite: None

Erase Surface from DWG

When you open a C&G surface (or TIN file, *.tin) it is shown on the screen as a graphic image overlaid on your drawing. You must use the **Write Surface to DWG** feature to actually create contour polylines, TIN lines, etc. If the surface changes due to changes in elevation or location of points you will want to erase the old surface and write the new surface to the drawing. However, once a surface is written to the drawing, it can be a difficult process to pick out all the surface entities in order to erase them from the drawing. This feature makes this an easy, one step operation.

You can use the items in this menu to erase the various topographic features: the TIN, Main Contours, Intermediate Contours, Break Lines, Include Boundaries, and/or Exclude Boundaries or All topo items.

Note: To erase contour labels use **CG-Survey > Topo > Label Contours > Delete Labels**

Pulldown Menu Location: CG-Survey > Topo

Tin

This feature erases all the C&G TIN entities found on the TIN layer specified on the **Topography** tab of the **C&G Options** dialog.

Pulldown Menu Location: CG-Survey > Topo > Erase Surface from DWG

Keyboard Command: cg_erase_tin

Prerequisite: None

Main Contours

This feature erases all the C&G main contour polyline entities found on the main contour layer specified on the **Topography** tab of the **C&G Options** dialog.

Pulldown Menu Location: CG-Survey > Topo > Erase Surface from DWG

Keyboard Command: cg_erase_main_cont

Prerequisite: None

Intermediate Contours

This feature erases all the C&G intermediate contour polyline entities found on the intermediate contour layer specified on the **Topography** tab of the **C&G Options** dialog.

Pulldown Menu Location: CG-Survey > Topo > Erase Surface from DWG

Keyboard Command: cg_erase_interm_cont

Prerequisite: None

All topo items

This feature erases all the C&G topo entities found on any of the layers specified on the **Topography** tab of the **C&G Options** dialog.

Pulldown Menu Location: CG-Survey > Topo > Erase Surface from DWG

Keyboard Command: cg_erase_all_topo

Prerequisite: None

Label Contours

The items in this submenu allow you to either label contour lines with appropriate elevations or remove previously placed contour labels.

Pulldown Menu Location: CG-Survey > Topo

Place Labels

This feature allows you to place the appropriate elevation labels at selected locations on C&G contour polylines. The label is a **TEXT** entity overlaying a **WIPEOUT** entity. The **WIPEOUT** entity serves to create a space between the label text and the contour line and to keep the contour line from showing through the text and obscuring it.

If you have not already done so, please review the contour labeling settings by choosing the **CG-Survey > Topo > Topographic Settings** menu item. The labels will be created on the **Main** and **Intermediate Contour Label Layers**. The **Labeling Interval** determines which contours are labeled. For example, if the **Labeling Interval** is set to 2.00, then every C&G contour polyline that you choose having an elevation evenly divisible by 2.00 will be labeled. The **Label-Contour Separation Distance:** is the space separating the contour line and the start and end of the label text.

Once you have verified the correct settings, choose the **Label Contours > Place Label** menu item.

At the prompt (see below) use the left mouse button to pick any point that is on one side of the contour line you wish to label.

Pass a line thru the contours to be labeled.

Pick first point on line. [<ENTER> to quit]:

Now, at the next prompt (see below), drag the rubber band line through one or more of the contours you wish to label and click the left mouse button a second time.

Pick second point [<ENTER> to quit]:

The labeling operation can be repeated as many times as needed, then press Enter to end the command.

Some Trouble Shooting Tips for Labeling Contours:

When a C&G surface is opened it is shown only as a graphic image overlying the drawing. Therefore, before you can place labels on the contour lines, the surface must be written to the drawing using the **CG-Survey > Topo > Write Surface to DWG** menu item..

If the labels do not appear on the contour lines you chose, verify the elevation on the contour using the **CAD LIST** command.

Also, try changing the **Labeling Interval** setting on the **Topography** tab of the **C&G Options** dialog.

If the labels still do not appear on the contour lines, look at **Drawing Settings** tab of the **C&G Options** dialog and verify that the **Text Size** is set to a value that is large enough to be seen when viewing the contours.

If the elevation labels are created with an incorrect number of decimal places, check the **Elevation Precision** on the **Rounding** tab of the **C&G Options** dialog under **Text in Drawing**. Use the **CAD UNDO** command to undo the previously placed labels and try again.

Prompts

Pass a line thru the contours to be labeled.

Pick first point on line. [<ENTER> to quit]: use the left mouse button to pick any point that is on one side of the contour line you wish to label.

Pick second point [<ENTER> to quit]: drag the rubber band line through one or more of the contours you wish to label and click the left mouse button a second time.

Pulldown Menu Location: CG-Survey > Topo > Label Contours

Keyboard Command: cg_label_contours

Prerequisite: C&G contour polyline entities in the drawing

Delete Labels

This feature allows you to delete previously placed C&G contour labels. C&G contour labels consist of two entities: a TEXT entity containing the elevation text and a WIPEOUT entity used to hide the contour polyline under the elevation text. While you can delete these using standard CAD commands, it requires several steps and can be tricky. This feature makes deleting these labels a one step operation.

After choosing the **CG-Survey > Topo > Label Contours > Delete Labels** menu item you will see the following prompt at the command line:

Select contour labels to delete:

Select objects: use the mouse to pick the label text for the labels to be deleted. Press Enter when done and the labels and their accompanying WIPOUT entities will be deleted.

Prompts

Select contour labels to delete:

Select objects: use the mouse to pick the label text for the labels to be deleted.

Pulldown Menu Location: CG-Survey > Topo > Label Contours

Keyboard Command: cg_del_cont_labels

Prerequisite: None

CGTools

CG Options

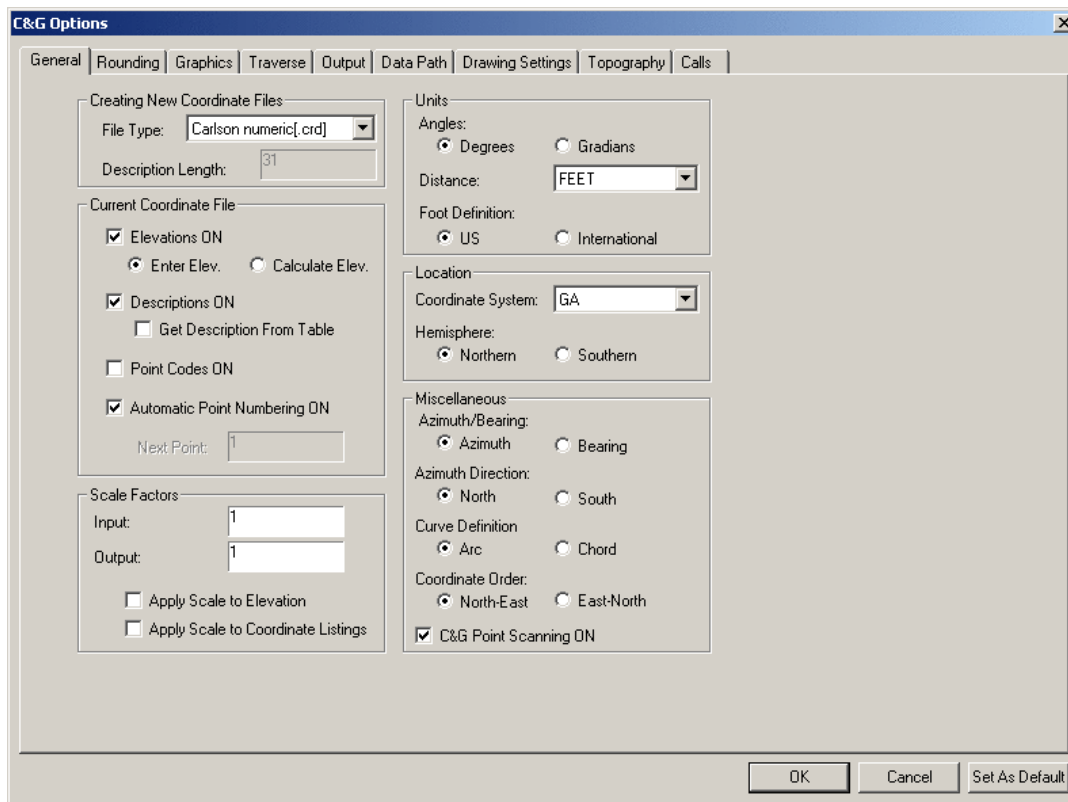
The **CG Options** menu item brings up the **C&G Options dialog**, allowing you to view or change various CGSurvey settings or save the currently configured settings to be used as the default settings for a newly created drawings.

There are nine tabs on the C&G Options dialog. Each tab pertains to a category of settings:

1. **General** tab - settings regarding the coordinate file type for new files, units, scale factors, and other general settings.
2. **Rounding** tab - number rounding settings used for the print file and for text placed in the drawing.
3. **Graphics** tab - specify when CGSurvey draws points and lines, format of bearings and other graphics related settings.
4. **Traverse** tab - settings used by all traverse related features.
5. **Output** tab - specify the name and layout of the print file and how the results of C&G features are displayed.
6. **Data Path** tab - specify the default path to your data files
7. **Drawing Settings** tab - specify drawing scale, text size, and details of how point symbols and their labels are to be drawn.
8. **Topography** tab - specify contouring parameters along with the layers used for the TIN, contour and other topographic entities.
9. **Calls** tab - specify the components, format and layer for calls (annotations).

Each of these tabs will be covered in the following sections.

This tab contains a wide variety of settings that apply to almost all of the features found in the CG-Survey menus. These are settings such as Next Point ID, Elevations, State, Arc Definition, Bearings/Azimuths, Coordinate order and more.



Creating New Coordinate Files section

File Type: You may select one of the following coordinate file types:

C&G numeric (*.crd)

- point ID can be an integer between 1 and 65,536
- description from 1 to 100 characters

C&G alphanumeric (*.cgc)

- point ID can contain up to 10 characters using any combination of letters and numbers.
- description from 1 to 100 characters

Carlson numeric (*.crd)

- point ID can be any integer containing up to 9 digits.
- description from 1 to 31 characters

Carlson alphanumeric (*.crd)

- point ID can contain up to 9 characters using any combination of letters and numbers.
- description from 1 to 31 characters

Simplicity (*.Zak)

- point ID can contain up to 8 characters using any combination of letters and numbers.
- description from 1 to 28 characters

Land Desktop Format (*.mdb)

- point ID can contain up to 255 characters using any combination of letters and numbers.
- description from 1 to 255 characters

Description Length: This value can only be set for C&G coordinate files. It becomes the default description length for new C&G coordinate and C&G raw data files. It can be set to from 1 to 100 characters.

Current Coordinate File section

Elevations ON If this checkbox is checked, elevations will be carried on all points computed and/or you will be able to enter an elevation when saving a point.

Enter Elev.: If this checkbox is checked, you will be prompted to manually enter elevations.

Calculate Elev.: If this checkbox is checked and an elevation can be computed from the data that has been entered during the command, it will be. Otherwise you will be asked.

Descriptions ON If the **Descriptions ON** checkbox is not checked, you will not be prompted to enter a description as points are created or edited.

If descriptions are ON, and **Get Description From Table** *IS NOT* checked, you will be prompted to manually enter a description for each coordinate point created. However, if Descriptions are on and **Get Description from Table** *IS* checked, when a point is stored and a description table *IS NOT* open, you will be prompted to select a description table. The description table will then be used to look up any integer number in the description in order to substitute the description in the table for the integer and move the integer to the Code files. (see help under CG-Survey > Management > Description Tables)

Point Codes ON If the **Point Codes ON** checkbox is checked, you will be allowed to enter a two to four character code depending on the number of characters in the code type you are using. This code can be used later to group points with the same code for plotting or listing points. When Point Codes are off, you will not be prompted to enter the point codes.

Automatic Point Numbering ON If the **Automatic Point Numbering ON** checkbox is checked, as points are created they will automatically be assigned the next available point ID in the current coordinate file. If Automatic Point Numbering is OFF, as points are created you will be prompted to enter their ID. If you enter a point number that already exists in the coordinate file, you will be asked if you want to overwrite the existing point or enter a new point ID.

Scale Factors section

Input: This allows you to set a scale factor that will be applied to all entered distances and coordinate values during any C&G feature.

Output: This allows you set a scale factor that will be applied to all output. For example, if this factor is set to 2.0 and the inversed distance between two points is 100.00, the output will show the distance as 200.00.

Apply Scale to Elevation If the **Apply Scale to Elevation** checkbox is checked, the Input and Output Scale Factors will be applied to elevation values.

Apply Scale to Coordinate Listings If the **Apply Scale to Coordinate Listings** checkbox is checked, the Input and Output Scale Factors will be applied to coordinates listed at the command line and in the print file using the C&G feature in menu item CG-Survey > Management > List.

Units section

Angles: Choose either the **Degrees** or **Gradians** radio buttons.

Distance: Choose Feet, Meters or Metres from the list.

Note: The only difference in the two metric choices is the spelling used for output.

Foot Definition: Choose either the **US** or **International** radio button.

Location section

State: specify the state in which the current survey was done.

This is only used in the following features:

Solar Observation

NAD83 (to and from longitude and latitude)

Hemisphere: Hemisphere can be set to **Northern** or **Southern**.

This is only used in the following features:

Solar Observation (Calculating the Convergency Angle)

NAD83 (to and from Longitude and Latitude - UTM only)

Miscellaneous section

Azimuth/Bearing: Allows you choose between **Bearing** and **Azimuth** for all direction input and output.

Azimuth Direction: This sets all azimuth input and output to either **North** or **South** azimuth.

Curve Definition The Curve Definition can be set to Arc or Chord.

Arc: the most commonly used definition in roadway design. When units are set to Feet, the degree of curve is the central angle of a 100 foot arc length.

Chord: is most commonly used in railroad work. When units are set to Feet, the degree of curve is the central angle of a 100 foot chord.

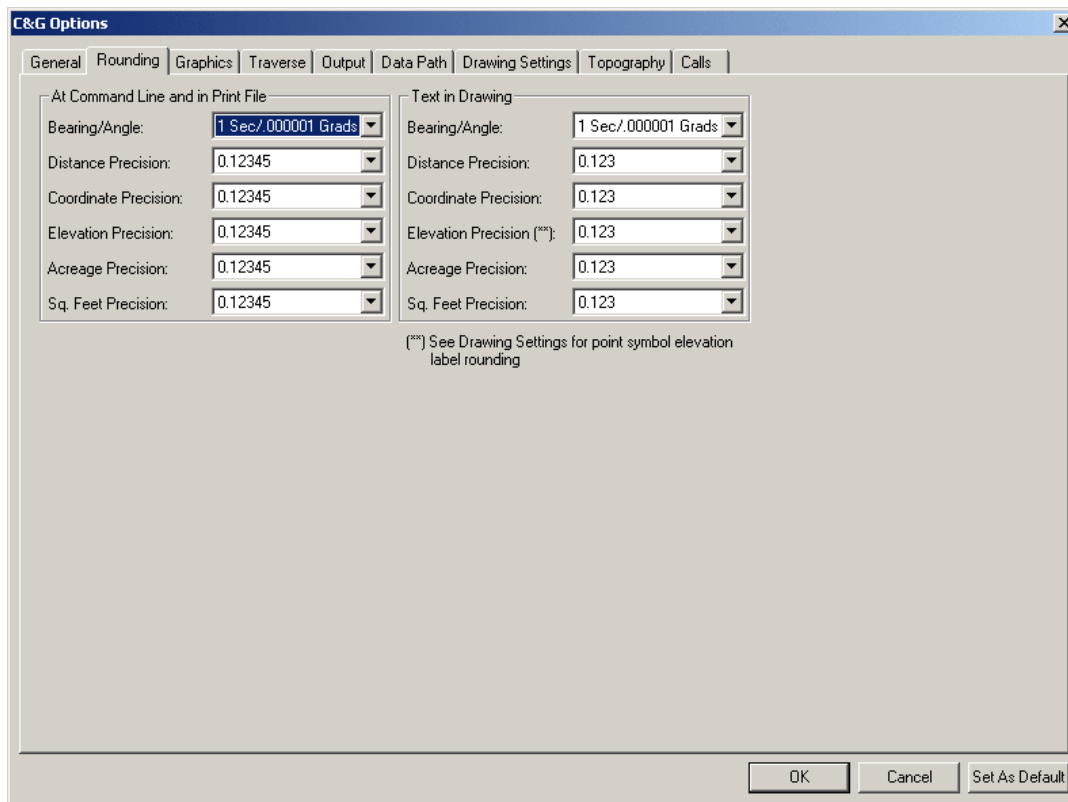
When a curve is added to a Curve Table or the results of calculations are listed at the command line and in the print file, the displayed information will reflect the **Curve Definition** setting.

Coordinate Order: Can be set to **North-East** or **East-North**. This sets the order in which coordinates are displayed and input.

Allows you to specify the rounding settings for various types of numbers for the print file text and for the drawing text.

Note: All internal calculations are performed with double precision accuracy. Only the output is rounded.

When you select the **Rounding** tab , you will see the following dialog:



The **Rounding** dialog has a section for **At Command Line and in Print File** rounding settings and a section for **Text in Drawing** rounding settings. Both sections have similar settings but they apply to different output. **At Command Line and in Print File** rounding settings effect all output to the command line and the print file. **Text in Drawing** rounding settings effect numeric text placed in the drawing.

Angular precision can be specified to the nearest:

Angles in Degrees or Angles in Grads

0.1 Second 0.000001 Grad

Second 0.00001 Grad

5 Seconds 0.0001 Grad

15 Seconds 0.001 Grad

30 Seconds 0.01 Grad

Minute 0.1 Grad

Distance precision can be specified to the nearest:

Foot (or Meter) 0 (no decimal places)

Tenth of Foot (or Meter) 0.1

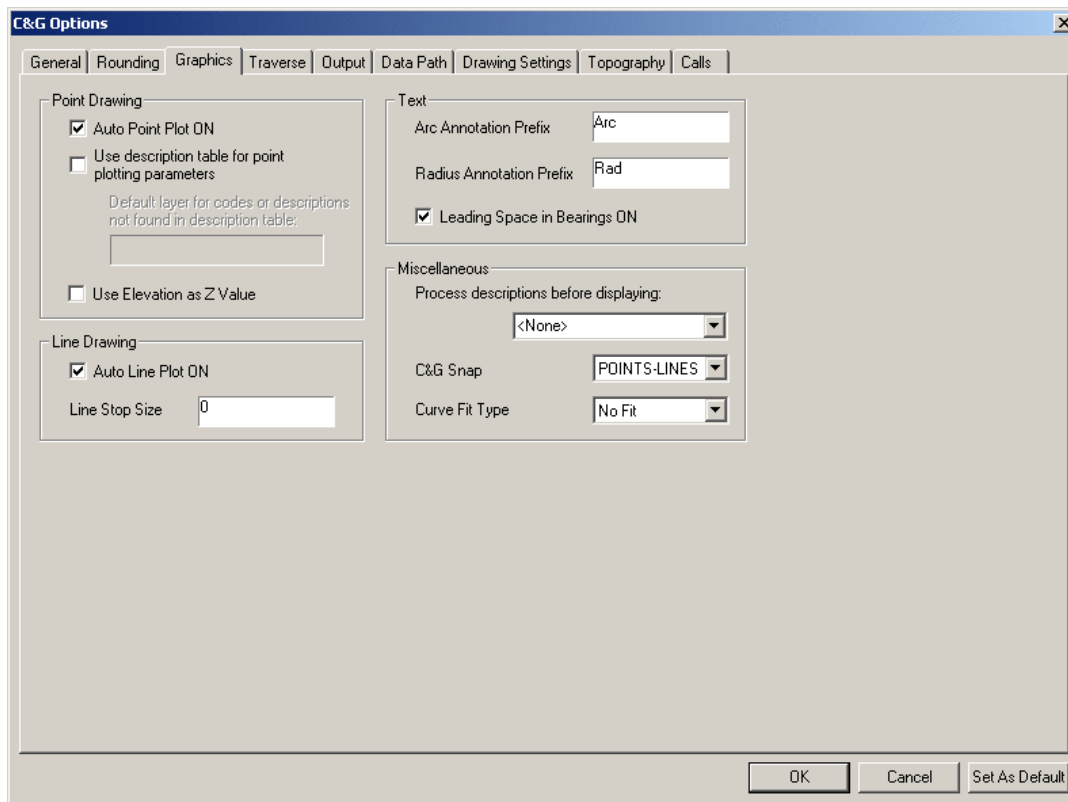
Hundredth Foot (or Meter) 0.12

Thousandth Foot (or Meter) 0.123

Ten Thousandth Foot (or Meter) 0.1234

Hundred Thousandth Foot (or Meter) 0.12345

The **Graphics** tab settings apply only to CGSurvey features that draw points, lines, etc. to the drawing. When you select the **Graphics** tab, the following dialog will appear:



Point Drawing section

Auto Point Plot ON If the **Auto Point Plot ON** checkbox is checked, points symbols will be drawn as they are calculated and saved to the coordinate file by the various C&G features.

Use Description table for point plotting parameters When this checkbox is checked the description(s) for a given point in the coordinate file will be matched with the descriptions in the description table. If a match is found then the description table information will be used to set the layer, symbol type, symbol size, and label positions of each point plotted. If no descriptions in the description table match then the layer will be set to the layer specified in the **Default layer for codes or descriptions not found in description table** edit box and the other settings specified in the **Drawing Settings** tab will be used (see below).

If the **Use Description table for point plotting parameters** checkbox is not checked, the points, symbols and labels will be plotted on the Current Layer as set in the CAD layer manager.

Default layer for codes or descriptions not found in description table: When the **Use description table for point plotting parameters** checkbox is checked, any points plotted that do not have a description or having a description that does not match any of those in the description table, will be plotted on the layer you have specified as the default layer in this edit box.

Use Elevation as Z Value: If this checkbox is checked, objects (lines/arcs/points) will be placed in 3-D space with the point elevation serving as the Z-value. C&G features, such as intersects and inverse, ignore the Z-value of lines and arcs. If you inverse a 3-D line, the 2-D distance between the points will be shown.

If the **Use Elevation as Z Value** checkbox is not checked, all objects will be placed at zero elevation.

Note: 3D lines can cause problems in trimming or editing using CAD functions. 3D lines do not intersect if their elevations are different. Thus two lines may appear to intersect in plan view but do not actually intersect in 3D space.

Line Drawing section

Auto Line Plot ON If the Auto Line Plot ON checkbox is checked, those features that create points that can be interpreted as a line will draw C&G lines.

The following features can draw lines and curves as the points are calculated:

Quick Traverse (not to side shots)

Curve Between Tangents and **Tangent Between Curves**

Bearing and **Hinge/Radial Area-Cut-Off**

Roadways (Right of Way/Easements and **Intersections/Cul-de-Sacs)**

Middle Ordinate Solution

Best Fit

Line Stop Size This allows you to terminate C&G lines at the edge of the point symbols plotted. If you are drawing lines and/or arcs with a C&G feature that draws lines and you want the line to end before crossing into the symbol, then set the **Line Stop Size** to the symbol size.

Note: If you set the line stop to something other than 0.0, the line that is drawn is shorter than the actual distance between the coordinate points. So if you wish to check the true distance of that line, use the **Query** command (on the Draw menu) rather than the CAD LIST command.

Text section

Arc Annotation Prefix

This is used when annotating arcs when drawing calls. This should be set to the desired prefix for arc length annotation.

Example:

"Arc =" annotation prefix results in the annotation being

Arc = 256.32

"A =" annotation prefix results in the annotation being

A = 256.32

Radius Annotation Prefix

This is used when annotating arcs when drawing calls. Similar to **Arc Annotation Prefix**, This should be set to the desired prefix for radius annotation.

Leading Space in Bearing

When the **Leading Space in Bearing** checkbox is checked the bearing text has a space between the N or S and the degrees text (eg, N 85°15'30"E). When left unchecked there is not space (eg, N85°15'30"E).

Miscellaneous section

Process Descriptions before Displaying:

This setting will allow you to specify how descriptions are processed prior to being displayed. It allows the removal of all underscores (_) and/or mapping codes. No change is made to the data in the coordinate file.

C&G Snap can be set to:

Off: No snap.

POINTS - Snap to C&G point symbols and labels.

LINES - Snap to C&G lines.

POINTS-LINES - Snap to C&G points and lines.

All C&G functions will use this setting when you are picking point symbols, point labels, lines, and arcs on the screen.

Curve Fit Type

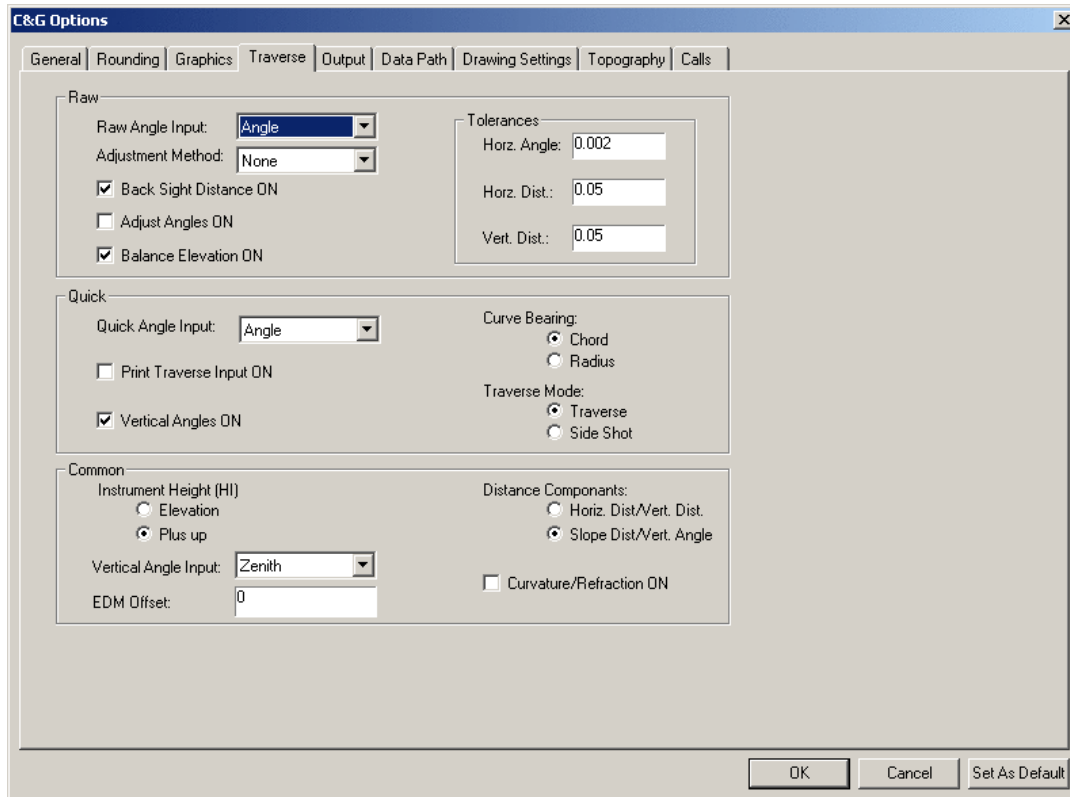
When contouring, the contour lines that are created can be smoothed using one of the following methods:

No Fit - Straight line segments between the points.

Fit - Use the CAD program's standard fit method. Contours may not pass through point symbols having the same elevation as the contour.

C&G Spline - Use the C&G Spline Fit algorithm. Contours are guaranteed to pass through point symbols having the same elevation as the contour.

These settings are specific to traverse raw data entry using the CGEditor and the traverse reduction and quick traverse features.



Raw section

Raw Angle Input

This allows you to specify how you want to specify angles when inputting raw traverse data. The options are: **Angle**, **Azimuth** or **Deflection** Angle.

Adjustment Method

You have the following choices for traverse adjustment:

None

Least Squares (NOT network least squares - see **SurvNET** for that)

Find Bad Angle

Compass

Transit

Crandall

Note: See the **Reduce Traverse** feature help section for more details on these methods.

If the **Backsight Distance ON** checkbox is checked and you entering raw traverse data, you must specify the distance to the backsight at each instrument point. These distances will then be used during the reduction process.

If the **Adjust Angles ON** checkbox is checked, angles will automatically be balanced prior to traverse adjustment. Angular error will be spread equally between all points. Closure information prior to and after balancing will be displayed at the command line.

If the **Balance Elevations ON** checkbox is checked, the elevations in a 3-D traverse will automatically be balanced during traverse adjustment. The elevations are adjusted proportional to the length of the traverse legs.

Tolerances section

Horz. Angle.

When comparing multiple angles for a given foresight point from a given instrument point and backsight point, this value will be used as the maximum acceptable angular error. If the difference between any two angles is greater than the acceptable limit, the reduction process will pause and showing the instrument point ID and angle measurements will be displayed at the command line.

Horz. Dist.

When comparing multiple horizontal distance components or measurements to a single foresight point, this value will be used as the maximum acceptable distance difference. If the difference between any two distances is greater than this limit, the reduction process will pause and the instrument point ID and the involved distances will be displayed at the command line.

The horizontal distance tolerance is also used as the maximum allowable difference between the two calculated curve radii at the curve end points. If the difference between the distances from the radius point to the PC point and the radius point to the PT point is greater than this value, the calculations will be terminated with an appropriate error message.

Note: for curves, if this value is set unreasonably low, many curves will produce this error message. If you change the setting to a larger, more reasonable value, the curve can be recalculated and generated without error.

Vert. Dist.

This value is the maximum acceptable elevation difference. It is used when comparing multiple vertical distance components/measurements to a given foresight point from a given instrument point. If the difference between the distances is greater than this limit, the reduction process will pause, showing you the instrument point ID and the involved distances. This only applies to the reduction of a 3-D traverse.

Quick section

Quick Angle Input

This specifies the default angle input mode for the Quick Traverse Feature. This can be changed when using the **Quick Traverse** feature.

The angle input modes are:

Angle

Deflection Angle

Azimuth

Bearing

If the **Print Traverse Input ON** checkbox is checked, all raw input data will be displayed along with the traverse output. If this checkbox is not checked, only the traverse output will be printed.

If the **Vertical Angles ON** checkbox is checked you will be asked to enter vertical angles with the traverse distances. This can be changed when using the **Quick Traverse** feature.

Curve Bearing

This defines how non-tangent curve bearings will be input and can be set to either **Chord** or **Radius** depending on how you wish to define the orientation of non-tangent curves.

When set to **Chord** and you are traversing around a non-tangent curve, you must enter the bearing or angle from the PC to the PT.

When set to **Radius** and you are traversing around a non-tangent curve, you must enter the bearing or angle from the PC to the radius point.

Curve Tables and printed calculations will reflect this setting.

Traverse Mode

Sets the default traverse mode for the **Quick Traverse** feature.

It can be set to **Traverse** or **Side Shot** mode .

Traverse mode: as a point is created the new point is occupied and backsight the previously occupied point.

Side Shot mode: as a point is created the currently occupied point and backsight will be held.

Common section

Instrument Height (HI)

The value entered for the HI can be either the actual instrument **Elevation** or the distance from the ground to the instrument (**Plus up**). In the latter case the elevation of the point the instrument is over is read from the coordinate file and the instrument height is added to it to determine the instrument elevation.

Vertical Angle Input - can be set to one of the following, depending on the type of instrument used:

Zenith: Zero angle up

Nadir: Zero angle down

Transit: Zero angle level

Note: If set to Transit, vertical can either be full circle (0 to 360 degrees; 0 to 400 grads) or positive angle up and negative angle down.

EDM Offset

Depending on where your EDM is mounted, enter the vertical difference between the center of the scope of the instrument and the center of the beam of the EDM (+ if EDM is above; - if EDM is below). Do not use an EDM Offset for scope mounted EDM's. This offset should only be applied to yoke or azimuth base mounted EDM's.

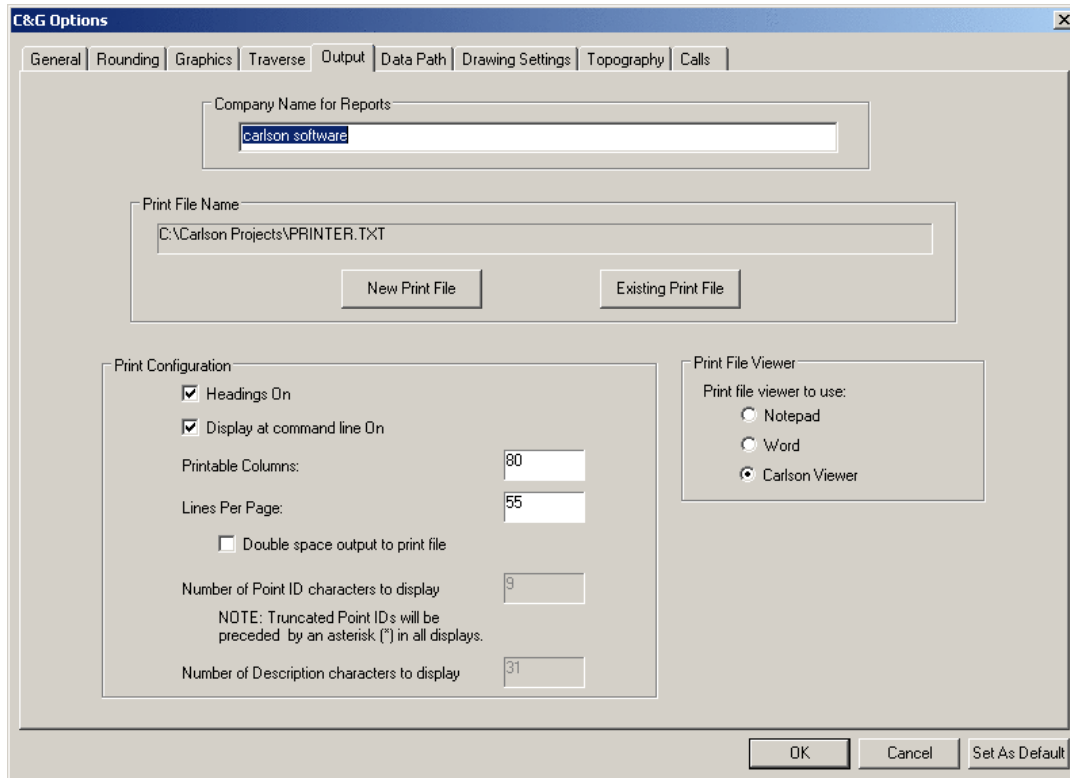
Note: Use of the EDM offset allows you to turn your vertical angles directly to the target. A correction will be applied to all distances and elevations computed from field entries in the Traverse and Quick Traverse routines. Most total stations today have the EDM coincident with the center line of instrument scope. In this case the EDM Offset should be set to zero.

Note: When an offset is entered, it is used on all distances in the traverse. If some distances are chained, the correction will also be applied. These shots should be reduced separately with no EDM Offset.

Distance Components - This option can be set to allow either **Slope Distance/Vertical Angle** or **Horizontal Distance/Vertical Distance** data entry.

If the **Curvature and Refraction ON** check box is checked, the horizontal and vertical components of all

slope distances are corrected for curvature and refraction. If your EDM does not already make this correction, it is recommended that this correction be used when carrying elevations using vertical angles and distances. This tab allows you to specify the name and format of the print file and how it is viewed.



Print File Name section

The final results of calculations and other actions performed during C&G command execution will always be printed to this ASCII text file. New information is always appended to this file and never overwritten. The default file name is PRINTER.TXT. It is recommended that you use a name that corresponds with the project you are working on. This way you will have a record of all calculations throughout the project. Use the **New Print File** button to specify a new print file to create. Use the **Existing Print File** button to specify an existing file.

Print File Viewer section

You can choose to use Microsoft Notepad or Wordpad when viewing or printing the print file. If you want the viewer to always come up full screen, check the **Force print file viewer to use full screen** checkbox.

Point Configuration section

If the **Headings On** checkbox is checked, a heading is printed to the command line and/or the print file any time multi-line output is generated by a C&G feature. The heading information contains date, time, feature name, coordinate file name and input and output scale factors. The header is repeated when the number of lines output by a function exceed the value set for **Lines Per Page**.

If the **Display On** checkbox is checked, the output from CGSurvey features is printed at the command line. Regardless of this setting, output is always sent to the print file.

Printable Columns

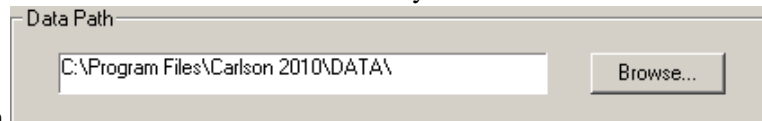
Use the edit box to specify the maximum number of characters per line to be written to the print file. This allows you to fit the text to the printed page given the font and paper you use. The acceptable values are 80 through 255.

Lines Per Page

This allows you to set the number of lines that will be placed on a page. If headings are on, a header will be printed to the print file and the command line each time this number of lines is exceeded.

On this tab you can specify the path to your data files. The data path is the default directory for file dialogs used in various C&G commands that open or save files.

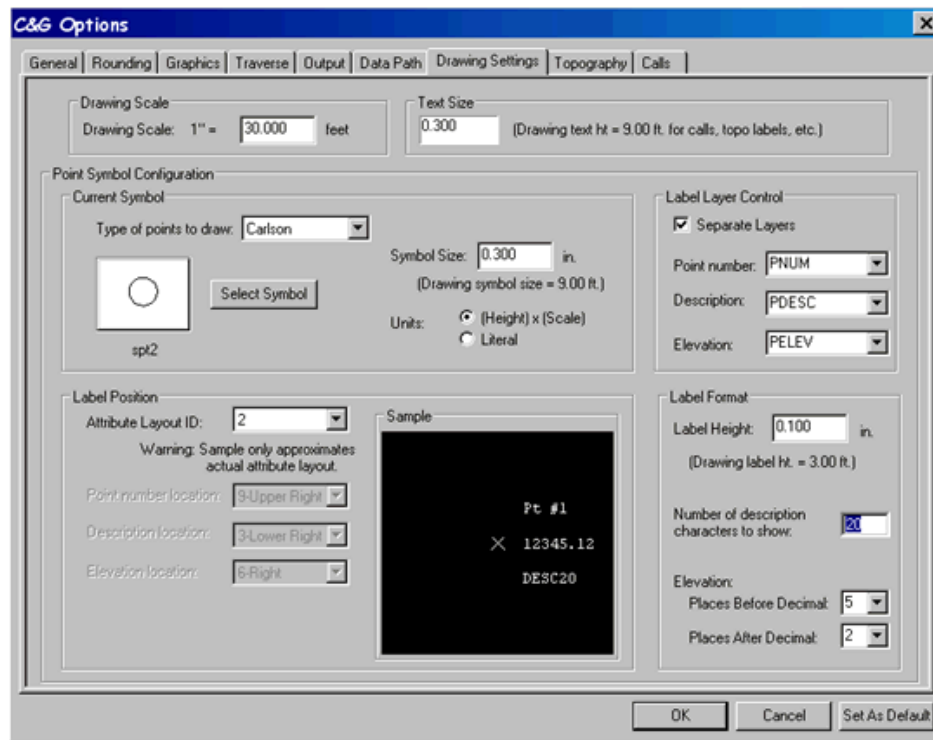
You can type the path in the **Data Path** edit box or you can use the **Browse...** button to use a file dialog to



specify the data path.

On this tab you can specify

drawing scale, text size, point symbol type and its format, and point symbol layers.



Drawing Scale section

This sets the horizontal scale. For example, if units are set to feet and you want a horizontal scale of 1" = 20' then type 20 in the Horizontal (ft/in) edit box. For metric units, if you want a scale of 1m = 500m then enter 500 in the Horizontal (m/m) edit box

Text Size section

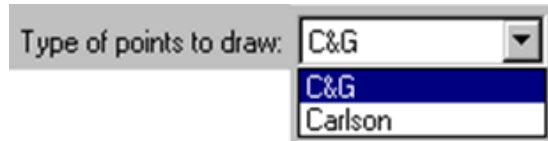
Allows you to set the text size for any text drawn using a CGSurvey feature. The text size is the size of the text as measured on the plotted or printed page. It must be specified in inches if using feet or centimeters if using meters.

Point Symbol Configuration section

Current Symbol section

This section allows you to control the symbol, its size and how it is scaled (called units here).

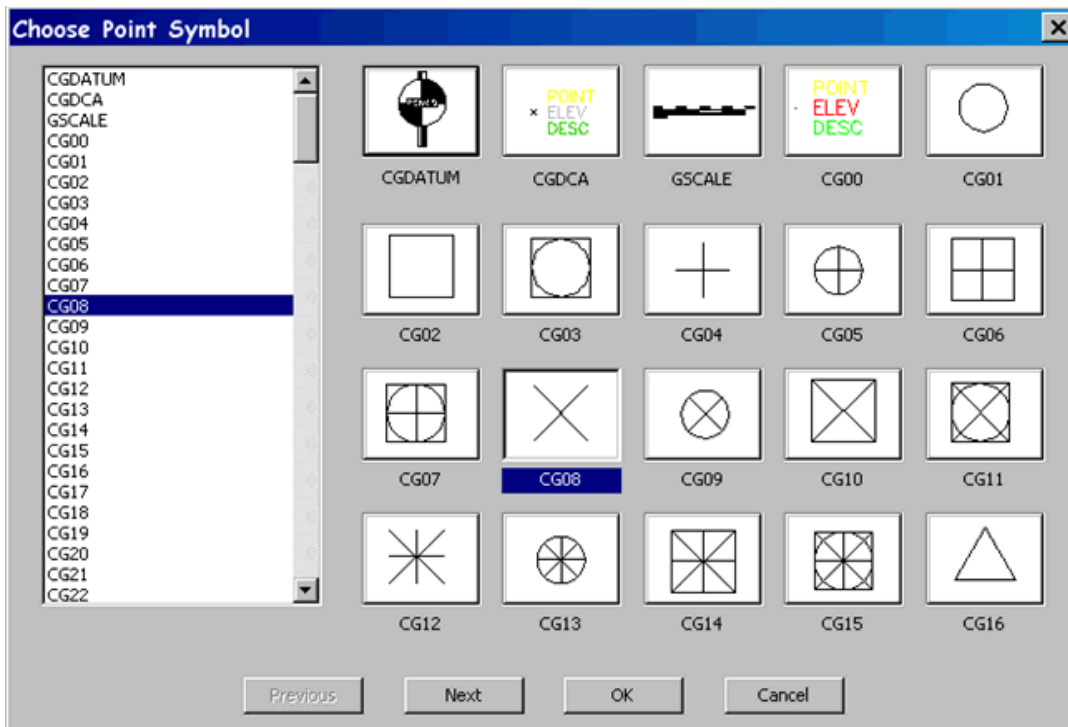
Type of Point to be Drawn: There are two point symbol libraries to select symbols from, the C&G and the Carlson symbol libraries.



Using symbols from either the C&G or Carlson symbol library both allow you to use all of the associated C&G features for plotting, sorting, line stops, attribute information, selection, etc. If you choose to use Carlson symbols the Label Position section of the dialog changes somewhat. This will be discussed later in this section.

Select Symbol button

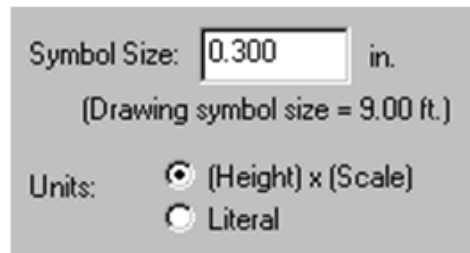
Choosing **Select Symbol** button will bring up the **Choose Point Symbol** dialog:



Use this dialog to choose the active point symbol. You do this by highlighting the symbol name in the list on the left or by clicking the symbol image on the right. Symbols CG00 and CGDCA are compatible with LDT/LDD points. The CGDCA symbol is the correct size for a true LDT/LDD point, and should be used if you are also using LDT/LDD.

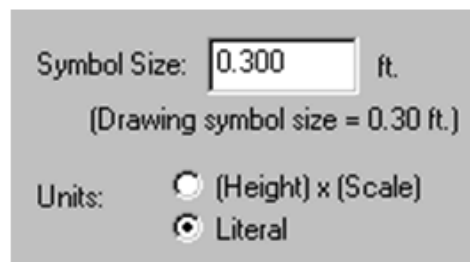
Symbol Size and Units

There are two options available for specifying symbol size: **(Height) X (Scale)** and **Literal**



Symbol Size: in.
(Drawing symbol size = 9.00 ft.)
Units: (Height) x (Scale)
 Literal

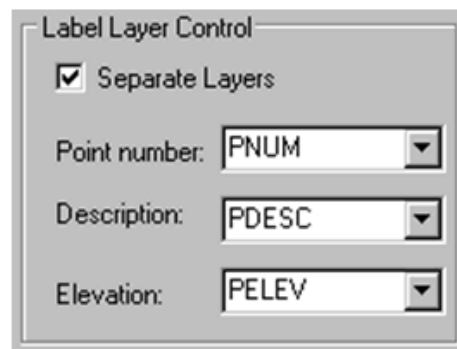
If **Units** are set to **(Height)x(Scale)**, then the symbol size entered here is specified as *plotted page units* (inches or centimeters - depending on whether feet or meters are being used). In this case, regardless of scale, the symbol will always be the same size when plotted. In example above, the symbol is set to .300". At 30 scale the symbol height will be 9 feet in the drawing itself, at 40 scale it would be 12 feet. Thus, in either case, its plotted size will be 0.3 inches.



Symbol Size: ft.
(Drawing symbol size = 0.30 ft.)
Units: (Height) x (Scale)
 Literal

If **Units** are set to **Literal** then the symbol will be drawn *in the drawing* at the size specified. This setting is often used for inserts such as title blocks, north arrows, company logos, standard notes, etc.

Label Layer Control section



Label Layer Control
 Separate Layers
Point number:
Description:
Elevation:

If you check the Separate Layers check box, you can assign each point label to a specific layer. This allows you to see only the labels you want by turning different layers on or off. If this checkbox is not checked, all the point labels will be drawn on the current layer.

Label Position section



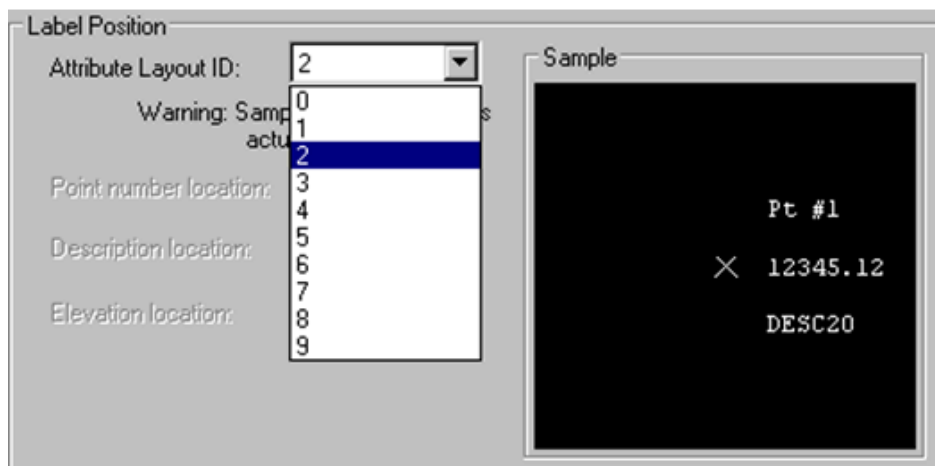
C&G Labels:

If the label location is set to **0 <Off>** that label will not be displayed when a point is plotted. Only the elevation is allowed to be at the **Center** position. If you select Center for the elevation label, the whole number portion of the elevation will be on the left side of the insertion point of the symbol and the decimal portion on the right side (example: the elevation 987.23 will be drawn as 987+23, where the plus sign represents the symbol).

If **Plot Radial** is checked, the point labels will be plotted radially from the symbol's center. If not selected, point labels will be plotted horizontally.

Label Position for Carlson Symbols

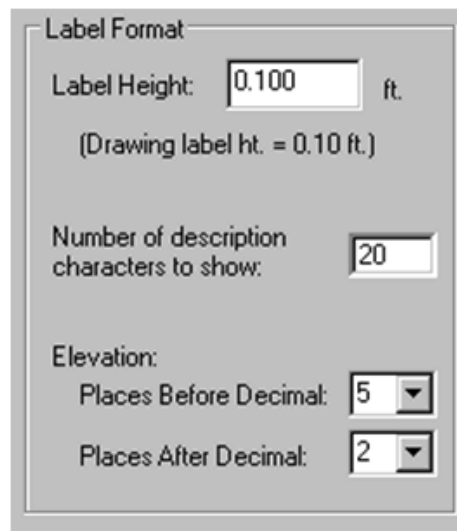
In the **Point Symbol Configuration** section of the dialog you have the option to plot C&G symbols or Carlson symbols. When the Carlson symbols are used, the **Label Position** portion of the dialog box changes to display the Carlson method for defining label positions (see below).



These "label positions" are actually pre-defined blocks with a predefined location and orientation for the attributes (or labels). There are ten blocks available. The available blocks are identified by the numbers 0 through 9.

Note: when Carlson point symbols are used, the Sample drawing is only approximate - the actual layout will look slightly different when drawn.

Label Format section



Label Height: this is the text size in inches/centimeters when **Units** are set to **(Height) x (Scale)** or feet/meters when **Units** are **Literal**. The **Label Height** is used for all three labels: point number, point description, and point elevation.

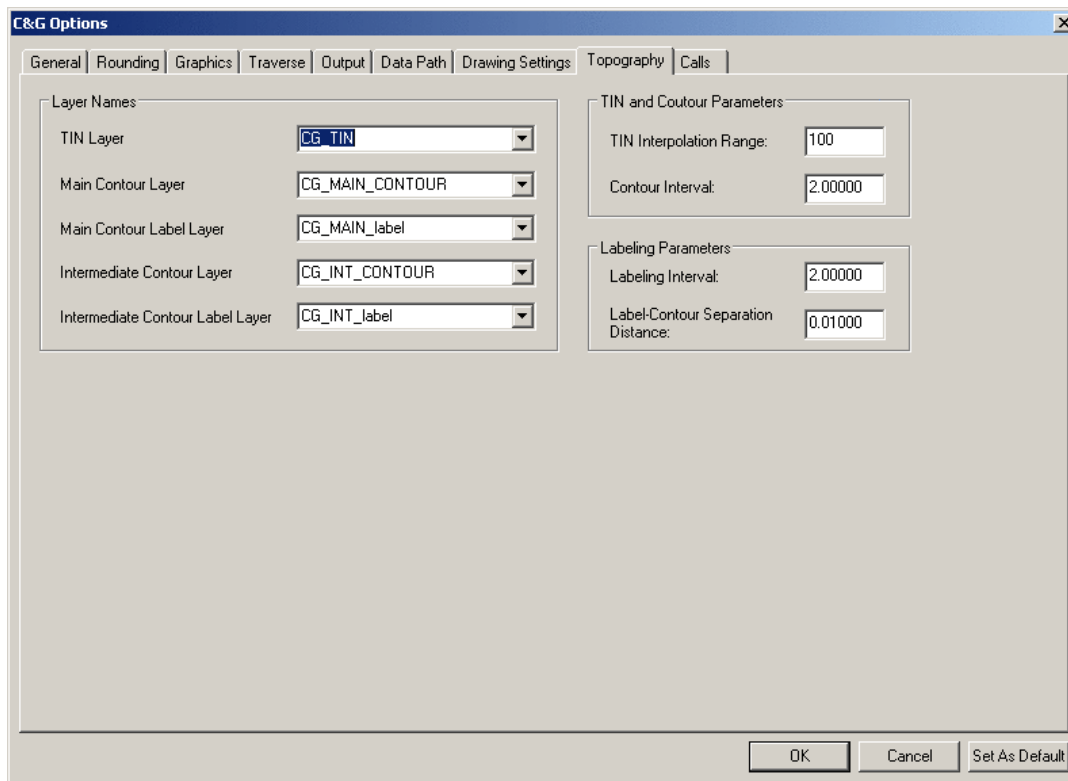
Number of description characters to show: Depending on the type of coordinate file being used, here may be as many as 255 characters in the description field. This option allows you to truncate the description at a given number of characters.

Elevation: This sets how many characters are displayed before and after the decimal point. On a flat piece of property 2 placed before the decimal may be enough information. On a steep mountain site 3 or 4 decimal places may be needed.

Topography tab

NOTE: The information on this tab is used for items on the CGTopo menu which has limited functionality and does not allow you to create a TIN. You must use the Carlson features to make, use and manipulate TINs (see Surface menu). These settings *may* be used when opening a CG-SURVEY for DOS drawing (*.PL1) when it has topo data in it.

The items on this tab allow you to specify contouring parameters and Tin, contour and other topographic entity layer specifications.



This dialog allows you to specify the drawing layers for the various topographic entities, as well as set various parameters for the creation of a new surface and placement of contour elevation labels.

Layer Names section

In this part of the dialog you can specify the layers for the various previously existing topographic entities found in the drawing. These allow you to label contours and, if necessary, remove contours and/or labels from the drawing.

TIN Layer: Specifies the layer on which triangulation network lines or TIN are found.

Main Contour Layer: Layer on which main contours are found.

Intermediate Contour Layer: Layer on which intermediate contours are found.

Main Contour Label Layer - Elevation labels for the main or index contour lines will be drawn on this layer.

Intermediate Contour Label Layer - Elevation labels for the intermediate contour lines will be drawn on this layer.

Note: The last two Contour Label Layer names will be used when labeling contours.

TIN and Contour Parameters section

TIN Interpolation Range: The interpolation range determines which points will be joined to form the triangles in the TIN. (MAY be used converting a CG-SURVEY for DOS PL1.)

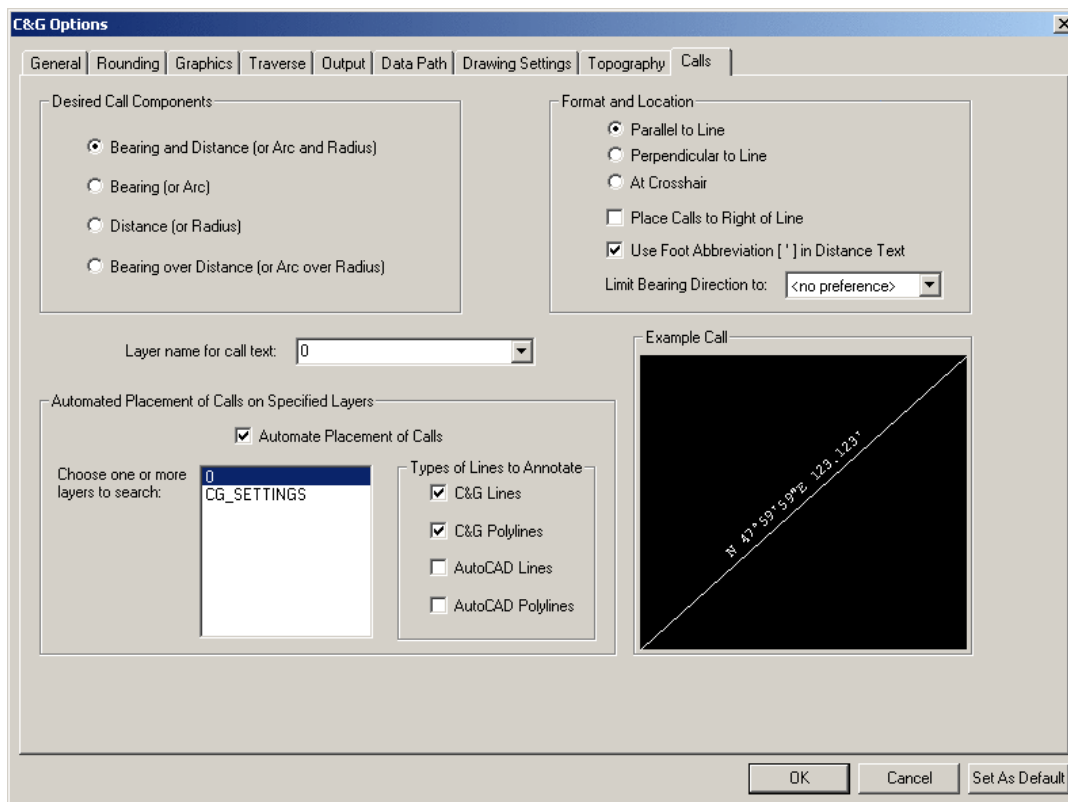
Contour Interval: (MAY be used converting a CG-SURVEY for DOS PL1.).

Labeling Parameters section

Label Interval: When labeling contours, only the contours falling on this interval will be labeled. For example, if you enter a 10' interval, only the contours at 900, 910, 920, etc will be labeled.

Label-Contour Separation Distance: This is the space between each end of the elevation label text and the contour line being labeled. A separation distance that is too small can make the elevation label hard to read, while a separation distance that is too large may not be visually pleasing.

This tab gives you several options for specifying the call or annotation format



Desired Components section

The of the dialog allows you to specify what you want displayed for a given call and whether the call text is stacked. The text in parentheses indicate the call items for a curve.

Format and Location section

The allows you to specify whether the call is placed **Parallel to Line**, **Perpendicular to Line** or requires the user to pick the location for horizontal call text (**At Crosshair**). If the **Place Calls to Right of Line** checkbox is checked the calls will be placed on the right side as determined by standing at the first point picked or the first point in a C&G line and looking toward the second point. You may also specify whether to use the foot symbol when units are feet. If bearings are being used, you may specify whether to limit bearing text to **NW,NE only** or **SW, SE only** or **<no preference>**.

Layer name for call text:

Specify the layer the call text is to be drawn on.

Automated Placement of Calls on Specified Layers section

This section of the dialog sets the parameters for a feature that allows you to place calls on C&G and/or CAD lines

and/or polylines found on specified layers. To use this option, check the **Automate Placement of Calls** checkbox. Choose one or more layer names from the list of layer names. You can specify multiple layers by holding the Ctrl key down while picking the layers to search.

In the **Types of Lines to Annotate** section, check the types of entities you wish to annotate.

Example Call section

The of the tab allows you to see a good approximation of how the call will look when drawn.

OK - click the **OK** button to save all the settings and close the dialog.

Cancel - click the **Cancel** button to close the dialog and discard any changes.

Set As Default

Click this button to save the settings to the CGSURVEY.OPT file. These settings will then be used whenever a new CGSurvey drawing is created.

Note: You can set the default settings and not affect any of the settings for the current drawing by clicking the **Cancel** button after clicking the **Set As Default** button.

Pulldown Menu Location: CG-Survey > Tools

Keyboard Command: cg_options

Prerequisite: None

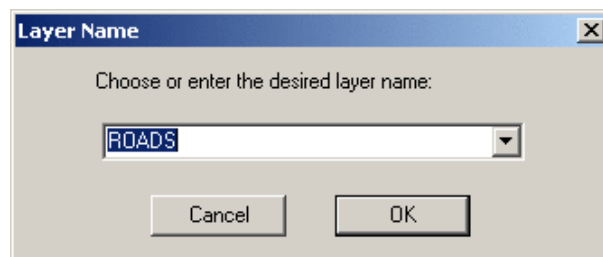
Copy Entity to Layer

This feature allows you to easily copy a single entity or group of entities from one layer to another.

Choose entities to copy:

Select entities: use the standard CAD selection methods to choose one or more entities to be copied. Once you have selected the entities press Enter.

Next use the **Layer Name** dialog to choose the layer to copy the entities to:



Prompts

Choose entities to copy:

Select entities: use the standard CAD selection methods to choose one or more entities to be copied.

Pulldown Menu Location: CG-Survey > Tools

Keyboard Command: cg_copyent

Prerequisite: None

Layer Control

These routines allow you to freeze, thaw, restore, turn on and off, and set the current layer without having to open the CAD layer manager.

Pick Layers to Freeze

This feature allows you to freeze layers by picking entities that are on layers you wish to freeze. You may pick as many entities as you wish.

Prompts

Select object on layer to FREEZE.

Select entities: choose objects on the layers you wish to freeze. Press Enter when done.

Pulldown Menu Location: CG-Survey > Tools > Layer Control

Keyboard Command: cg_lfreeze

Prerequisite: None

Pick Layers to Thaw

This feature allows you to thaw frozen layers by picking entities that are on a layer you wish to freeze. All frozen layers are turned on while you pick the entities. You may pick as many entities as you wish.

Prompts

Select object(s) on layers to keep THAWED.

Select entities: choose objects on layers you wish to thaw. Press Enter when done.

Pulldown Menu Location: CG-Survey > Tools > Layer Control

Keyboard Command: cg_lthaw

Prerequisite: None

Freeze ALL Layers

Choosing this menu item causes all layers, except the current layer, to be frozen.

Pulldown Menu Location: CG-Survey > Tools > Layer Control

Keyboard Command: cg_alfreeze

Prerequisite: None

Thaw ALL Layers

Choosing this menu item causes all layers to be thawed.

Pulldown Menu Location: CG-Survey > Tools > Layer Control

Keyboard Command: cg_althaw

Prerequisite: None

Pick Layers to turn Off

This feature allows you to turn off layers by picking entities that are on layers you wish to turn off. You may pick as many entities as you wish.

Prompts

Select object on layer to turn OFF. choose objects on layers you wish to turn off. Press Enter when done.

Pulldown Menu Location: CG-Survey > Tools > Layer Control

Keyboard Command: cg_loff

Prerequisite: None

Pick Layers to turn On

This feature allows you to turn on layers by picking entities that are on layers you wish to turn on. All layers will be turned on during the command to allow you to pick the desired entities. You may pick as many entities as you wish.

Prompts

Select object(s) on layers to keep ON.

Select entities: choose objects on layers you wish to turn on. Press Enter when done.

Pulldown Menu Location: CG-Survey > Tools > Layer Control

Keyboard Command: cg_lon

Prerequisite: None

Turn OFF ALL Layers

Choosing this menu item causes all layers, except the current layer, to be turned off.

Pulldown Menu Location: CG-Survey > Tools > Layer Control

Keyboard Command: cg_aloff

Prerequisite: None

Turn ON ALL Layers

Choosing this menu item causes all layers to be turned on.

Pulldown Menu Location: CG-Survey > Tools > Layer Control

Keyboard Command: cg_alon

Prerequisite: None

Pick Current Layer

This feature allows you to pick an entity that is on a layer you wish to make the current layer.

Prompts

Set Current Layer - Select entity on the desired layer: pick the entity which is on the layer you want to made the current layer.

Pulldown Menu Location: CG-Survey > Tools > Layer Control

Keyboard Command: cg_lset

Prerequisite: None

Elevations

If this menu item is checked, then point elevations are ON. If it is unchecked then point elevations are OFF.

When point elevations are ON and the **Enter Elev.** radio button is set on the **General** tab of the **C&G Options** dialog, you will be prompted to enter an elevation when new points are saved to the coordinate file. When point elevations are ON and the Calculate Elev. radio button is set you will not be prompted to enter an elevation. When point elevations are OFF, no elevation is stored when coordinate points are saved to the coordinate file.

Pulldown Menu Location: CG-Survey > Tools

Keyboard Command: cg_tog_elev

Prerequisite: None

Descriptions

If this menu item is checked, then point descriptions are ON. If it is unchecked then point descriptions are OFF. Generally, when point descriptions are on, you will be prompted to enter a description when new points are saved to the coordinate file. Also, when descriptions are ON, the description field will be enabled when editing coordinate point values.

Pulldown Menu Location: CG-Survey > Tools

Keyboard Command: cg_tog_desc

Prerequisite: None

Point Code

If this menu item is checked, then point codes are ON. If it is unchecked then point codes are OFF. Point codes are unique to C&G coordinate files that can be used to filter or group points in various C&G features.

When point codes are ON, you will be prompted to enter a point code when new points are saved to the coordinate file. When point codes are OFF, no code is stored when coordinate points are saved to the coordinate file.

Pulldown Menu Location: CG-Survey > Tools

Keyboard Command: cg_tog_code

Prerequisite: None

Auto Point Number

If the **Auto Point Number** menu item is checked then automatic point numbering is ON. This means that, as points are created they will automatically be assigned the next available point ID in the current coordinate file and will be saved without any user interaction.

When auto point numbering is OFF, as points are created you will be asked to enter the point ID. If you enter a point ID that already exists in the coordinate file, you will be asked if you want to overwrite the existing point or enter a new point ID.

See also: **Automatic Point Numbering ON** checkbox on the **General** tab of the **C&G Options** dialog.

Pulldown Menu Location: CG-Survey > Tools

Keyboard Command: cg_tog_auto_num

Prerequisite: None

Auto Point Plot

If the **Auto Point Plot** menu item is checked, when a point is calculated and stored in the coordinate file it will be plotted in the drawing.

See also: the **Auto Point Plot ON** checkbox on the **Graphics** tab of the **C&G Options** dialog.

Pulldown Menu Location: CG-Survey > Tools

Keyboard Command: cg_tog_auto_pt_plot

Prerequisite: None

Auto Lines

If the **Auto Lines** menu item is checked, automatic line plotting is ON. When automatic line plotting is ON, the following COGO features will automatically draw lines and curves using the newly calculated points as they are saved to the coordinate file:

Quick Traverse - but lines will not be drawn to side-shots

Curve Between Tangents and **Tangent Between Curves**

Bearing Area Cut-Off

Hinge/Radial Area Cut-Off

Roadway (**Right-of-way** and all **Cul-de-Sacs** and **Intersections** features)

Middle Ordinate Solution for curves

Best Fit

See also: **Auto Line Plot ON** checkbox on the **Graphics** tab of the **C&G Options** dialog.

Pulldown Menu Location: CG-Survey > Tools

Keyboard Command: cg_tog_auto_lines

Prerequisite: None

CG Snap

C&G Snaps are object snaps that are active only during a C&G command. These snaps allow you to pick point symbols and/or C&G lines by clicking near them. They work similar to the CAD snaps but only snap to C&G entities. The C&G snaps work in conjunction with the normal CAD snaps but, when a C&G command is run, the CAD snaps are automatically turned off at the start of the command and the C&G snaps become active. In almost all C&G features you have the option of turning the CAD snaps back on if desired. When both the CAD and C&G snaps are on, the CAD snaps are applied first to determine the x and y screen coordinates of the point on the appropriate CAD entity; these coordinates are then passed to C&G and they are used to apply the C&G snaps and find the nearest appropriate C&G entity.

Note: If CAD snaps are turned on during a C&G command and if C&G snaps are also on, a double snapping process occurs. Because of this double snapping, it is recommended that when C&G Snaps are on, CAD snaps should be left off during C&G commands.

Note: If snapping is desired and C&G snaps are off, then the CAD snaps must be turned on each time a C&G command is run.

Pull-down Menu Location: CG-Survey > Tools

Keyboard Command: None

Prerequisite: None

Off

This turns off all C&G snaps.

Note: This setting applies ONLY to C&G features and is not directly supported by whatever CAD software you are using.

Pull-down Menu Location: CG-Survey > Tools

Keyboard Command: cg_snap_off

Prerequisite: None

Points

This allows you to pick near and snap to a C&G point symbol whenever a point ID is required for a C&G feature.

Note: This setting applies ONLY to C&G features and is not honored by whatever CAD software you are using.

Pull-down Menu Location: CG-Survey > Tools

Keyboard Command: cg_snap_points

Prerequisite: None

Lines

This allows you to pick near and snap to a C&G line whenever a bearing, distance, or pair of points is required for a C&G feature.

Note: This setting applies ONLY to C&G features and is not honored by whatever CAD software you are using.

Pull-down Menu Location: CG-Survey > Tools

Keyboard Command: cg_snap_lines

Prerequisite: None

Points and Lines

This allows you to pick near and snap to a C&G point or line whenever a point ID, bearing, distance, or pair of points is required for a C&G feature.

Note: This setting applies ONLY to C&G features and is not directly supported by whatever CAD software you are using.

Pull-down Menu Location: CG-Survey > Tools

Keyboard Command: cg_snap_points_lines

Prerequisite: None

Zoom to Point ID

This feature pans the drawing in order to place the location of the point ID you specify at the center of the screen. It is not necessary to plot the point symbol prior to using this feature.

Prompts

Point ID of point to zoom to: specify the point ID of a point in the current coordinate file.

The drawing will be panned to center the point and a "rubber band" line will extend from the point to your cursor.

Stopping to view point [View another/Done] <D>: Press "D" and Enter or just Enter to clear the rubber band line and return to the CAD command line. Press "V" and enter to specify another point ID.

Pulldown Menu Location: CG-Survey > Tools

Keyboard Command: cg_zoom_pt

Prerequisite: coordinate file

Windows Calculator

Selecting this menu item will bring up the standard Microsoft Windows® calculator.

Pulldown Menu Location: CG-Survey > Tools

Keyboard Command: 'cg_cal

Prerequisite: None

CGEditor

The CGEditor is an integral part of preparing files for use for C&G applications. The CGEditor is a very powerful tool. You can open multiple data files of any supported file type and edit the files as needed. The CGEditor has a full complement of tools for searching and replacing and navigating within a file. It will also allow you to cut or copy records from one file and paste them into another file in order to merge files, move data between phases of a job, etc.

Types of data files supported

The CGEditor can create and/or edit four types of data files used by CGSurvey and Carlson.

Raw Data Files

Raw data files contain information pertaining to a field traverse. Raw data files are typically downloaded from the data collector and converted to the C&G raw data file format. These files have the extension .cgr.

Map Check Files

Map Check files contain bearing, distance and curve information and are typically used to calculate the closure of a deed description. These files have the extension .cgm.

Cross Section Files

Cross Section files contain one or more cross sections identified by their station along the alignment. Each cross section record has the percent grade defined for its left and right slopes. Following the "Station" record are several "Point" records containing the elevations and offsets of the points along the cross section. Cross section files consist

of a pair of files; the main data file has the extension .cew and the index file has the extension .cex.

Template Files

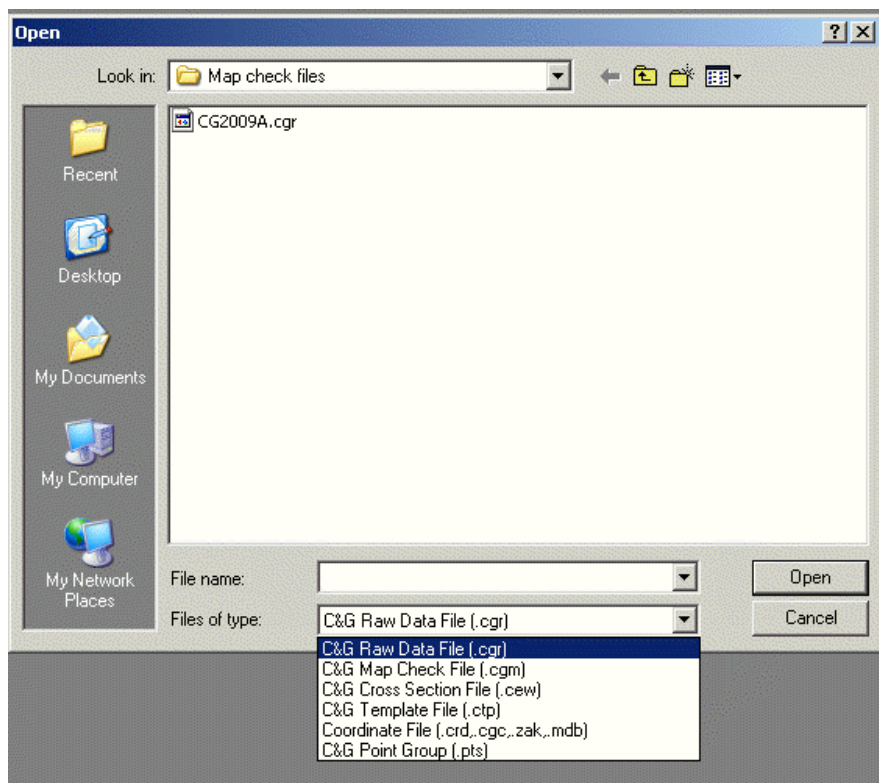
Template files are merely cross section files that represent a standard cross section and can be used to generate other cross section files. However, unlike cross section files, template files use an integer ID instead of a station to uniquely identify each template. Like cross section files, the percent grade is defined for the left and right slopes of each template and there are a set of "Point" records specifying the template elevation at a given offset. The centerline elevation at offset 0.00 is typically set to 0.00. Template files consist of a pair of files; the main data file has the extension .ctp and the index file has the extension .ctx.

NOTE: The CGEditor program sold as part of the standalone version of SurvNET can only be used to edit raw data files.

The CGEditor can be used to create new files or edit existing files. It uses a multi-document interface, so you can edit or view several files of several different types at the same time. The following sections will describe how to open and edit files.

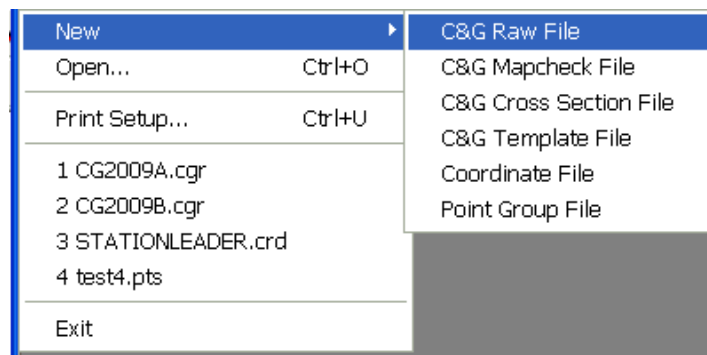
Opening Existing Files

To open an existing file, click on the **File** menu then choose **Open** in the submenu. You can then use the Open file dialog box to browse to the desired file. Check to make sure the **Files of Type:** is set correctly. Click on the desired file to highlight it, then click the **Open** button.



Creating Files

To create a new file, use the **File** menu and choose **New** and then click on the type of file you wish to create:



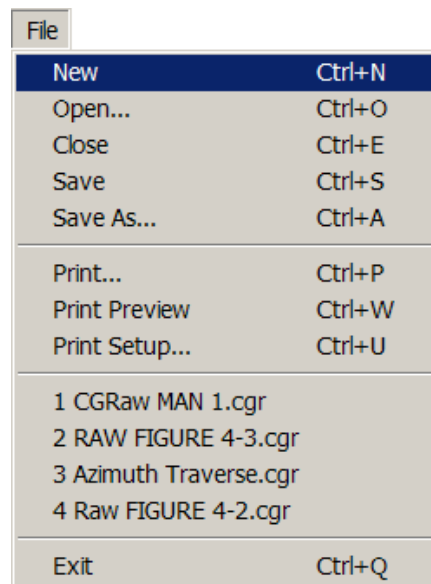
- C&G Raw File**
- C&G Mapcheck File**
- C&G Cross Section File**
- C&G Template File**
- Coordinate File**
- Point Group File**

After clicking the menu item for the type of new file you wish to create, a temporary file is created with no data in it and a spreadsheet-like window will open. At this point more menu items will be added to the main menu and, as you will see, the **Add** menu item will allow you to insert data rows (or records) where you can enter your data.

NOTE: The CGEditor program sold as part of the standalone version of SurvNET can only be used to edit raw data files.

The CGEditor Menus

File Menu



Many of the following File menu items will be familiar to experienced Windows users:

New: Allows you to create a new file.

Open (Ctrl + O): Brings up the Open File dialog box so you can select and edit an existing file.

Close (Ctrl + E): Closes the current data file. If more than one file is open, the file that is currently being worked on will be closed.

Save (Ctrl + S): Saves the current file.

Save As: Allows the user to save the current file to a file having a different name.

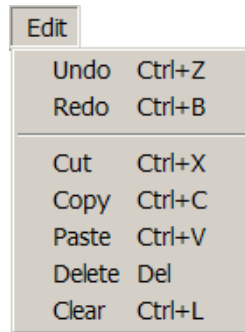
Print (Ctrl + P): Allows the user to print a copy of the currently active file.

Print Preview (Ctrl + W): Display a preview of the file about to be printed.

Print Setup (Ctrl + u): Printer selection as well as page size and layout.

Exit (Ctrl + Q): Exit the CGEditor application.

Edit Menu



As with the File menu, the Edit menu is typical of most Windows programs.

Most of the items in the Edit menu require that either a field within a record, or the entire record itself, be selected (highlighted) before clicking the menu item. To select a an individual data item (or field) in a data record simply click the field. To select a record (row) simply click in the first field (Type or Row#) for the desired record.

Undo (Ctrl + Z): Undoes the most recent editing action. (you need not have anything highlighted for this item)

Redo (Ctrl + Y): Reverses the most recent undo action. (you need not have anything highlighted for this item)

Cut (Ctrl + X): Cuts the currently highlighted cell or record. You may then use the paste command to put the cut cell or record in another location.

Copy (Ctrl + C): Copies the currently highlighted cell or record. You may then use the paste command to put the copied cell or record in another location.

Paste (Ctrl + V): Allows you to paste any previously cut or copied cell or record to the currently highlighted location.

If entire records are being pasted and only a field is currently highlighted, the pasted records will be inserted above the current record. However, if one or more entire records are currently highlighted, the pasted records will replace the highlighted records.

Delete (<Delete> key): Deletes the currently highlighted field or record.

Select All (Ctrl + A): Selects all the records in the current data file.

Clear (Ctrl + L): Removes the data from the selected field or record.

Add Menu

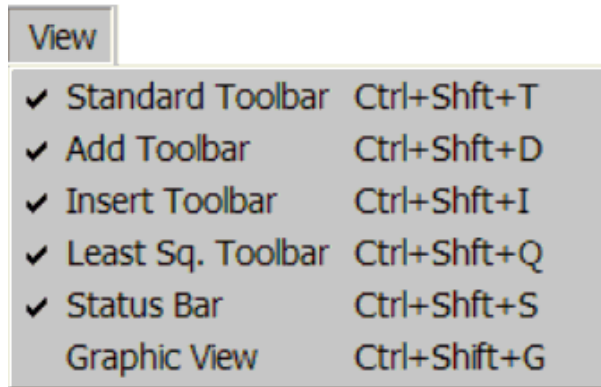
The Add menu allows you to add a record to the current file. The Add menu item appends the record to the end of the file. The types of records that can be added will depend on the type of file being edited, these record types will be described in more detail in later sections for each type of file you can editor.

Insert Menu

The Insert menu allows you to insert a record above the current record.

The types of records that can be inserted will depend on the type of file being edited, these record types will be described in more detail in later sections for each type of file you can edit.

View Menu



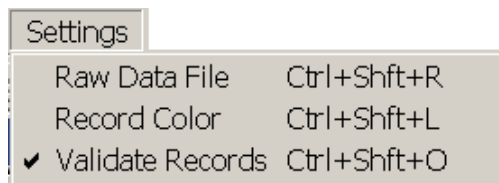
The View menu allows you to turn tool bars on or off. The items listed in the View menu will differ for different types of files. The individual tool bars will be discussed in the sections pertaining to the various types of files that can be edited.

Standard Tool Bar



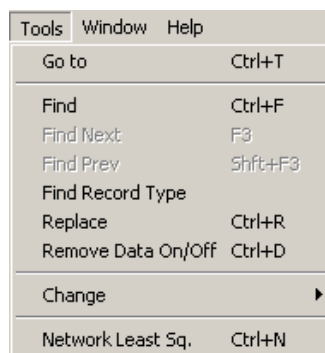
The above figure shows the standard tool bar. The Standard toolbar is the same for all types of files. It allows you to create all the various files that can be edited by the CGEditor. It allows you to open and save files. It allows you to cut, copy and paste and undo and redo as well as print the current file.

Settings



The Settings Menu will differ depending on the type of file being edited. But generally contains the settings for the file and the record colors.

Tools



The Tools Menu contains a variety of spread sheet tools, such as find, find next, find and replace etc. The menu will vary slightly for each type of data file and will be discussed in the sections pertaining to the various file types.

Windows

This menu contains many of the standard Window menu items found in other programs. It allows you to arrange the currently open windows in several configurations. It has the added functionality of the New Window command which allows you to have two or more views of a single file.

Traverse types

The raw data file can contain data pertaining to one or more traverses. If you will be using SurvNET to process the data, there is no need to delineate separate traverses in the raw data file. However, if you are using the old C&G traverse reduction program, and you want to combine more than one traverse in a raw data file, you will need to use the special traverse code records at the beginning and end of each traverse.

There are three basic types of traverses:

Closed Loop Traverse

Closed Traverse Beginning and Ending at Known Points

Open Traverse and Side Shots

Figures 1, 2, 3 and 4 show illustrations of each of these traverse types. Below each illustration you will also see the accompanying raw data as seen in the CGEditor.

Closed Loop

A closed loop begins and ends on the same two points as shown below in Figure 1

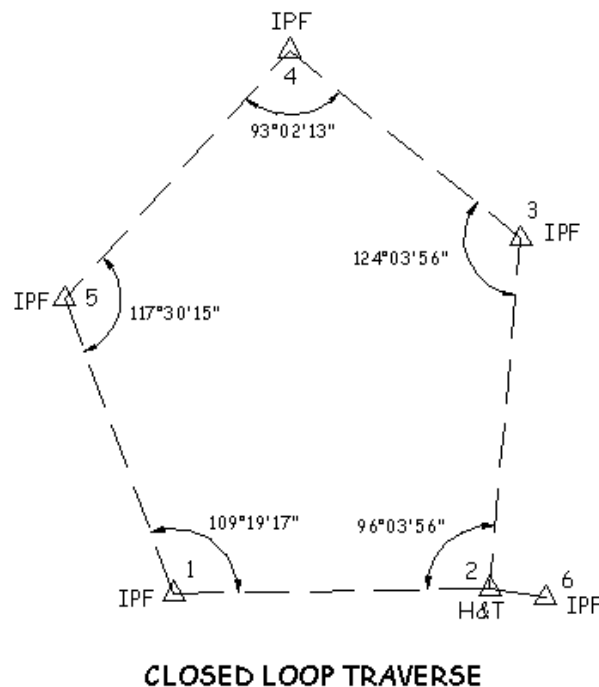
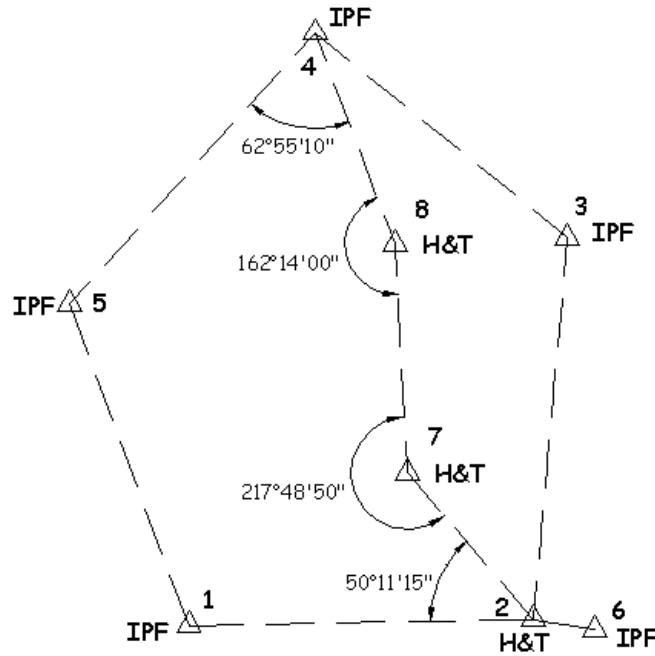


Figure 1

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Horz.Angle	Slope Dist.	Vert.Angle	Foresight	Code	Description
IP	1	5.32000	5		0.00000					
FS				5.00000	109.19170	292.31000	88.35000	2	TP	H&T
IP	2	5.20000	1		0.00000					
FS				5.00000	96.03560	324.26000	88.51320	3	TP	IPF
FS				5.00000	190.32100	52.39000	90.32550	6	SS	IPF
IP	3	4.90000	2		0.00000					
FS				5.00000	124.03560	275.84000	92.22400	4	TP	IPF
IP	4	5.12000	3		0.00000					
FS				5.00000	93.02130	309.68000	89.13000	5	TP	IPF
IP	5	5.15000	4		0.00000					
FS				5.00000	117.30500	290.55000	90.00000	1	TP	IPF

Closed Loop beginning and ending on known points

Figure 2 shows a closed traverse beginning on two known points (1 and 2) and ending on two known points (4 and 5). With this type of traverse, both a linear and angular closure can be calculated



CLOSED LOOP TRAVERSE BEGINNING AND ENDING AT KNOWN POINTS

Figure 2

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Horz.Angle	Slope Dist.	Vert.Angle	Foresight	Code	Description
IP	2	5.90000	1		0.00000					
FS				5.00000	50.11150	164.12000	88.20200	7	TP	H&T
IP	7	5.23000	2		0.00000					
FS				5.00000	217.48500	193.91000	90.58300	8	TP	H&T
IP	8	5.34000	7		0.00000					
FS				5.00000	162.14080	191.57000	92.03140	4	TP	IPF
IP	4	5.30000	8		0.00000					
FS				5.00000	62.55100			5		

Loop beginning on two known points and closing on an azimuth

Figure 3 illustrates a traverse that begins on two known points, or a single known point and a back sight azimuth, and ends on one known point. In this case it is only possible to calculate a linear closure.

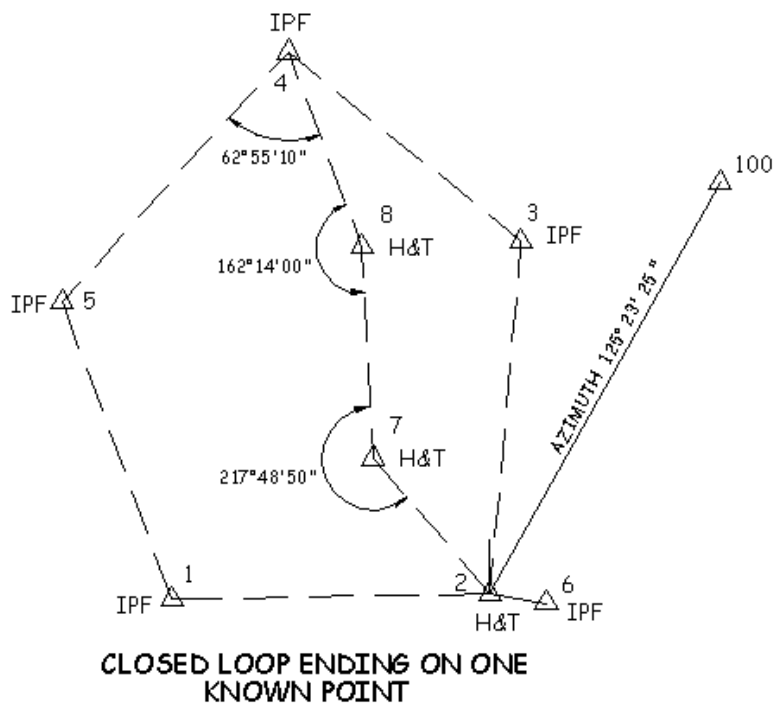


Figure 3

Set	From Point	To Point	Bearing							
DR	2	100	125.23240							
IP	2	4.90000	100		0.00000					
FS				5.00000	50.11150	164.12000	88.20200	7	TP	H&T
IP	7	5.23000	2		0.00000					
FS				5.00000	217.48500	193.91000	90.58300	8	TP	H&T
IP	8	5.34000	7		0.00000					
FS				5.00000	162.14030	191.57000	92.03140	4	TP	IPF
IP	4	5.30000	8		0.00000					
FS				5.00000	87.23150			101		

Open Traverse

Figure 4 shows an open traverse (side shots).

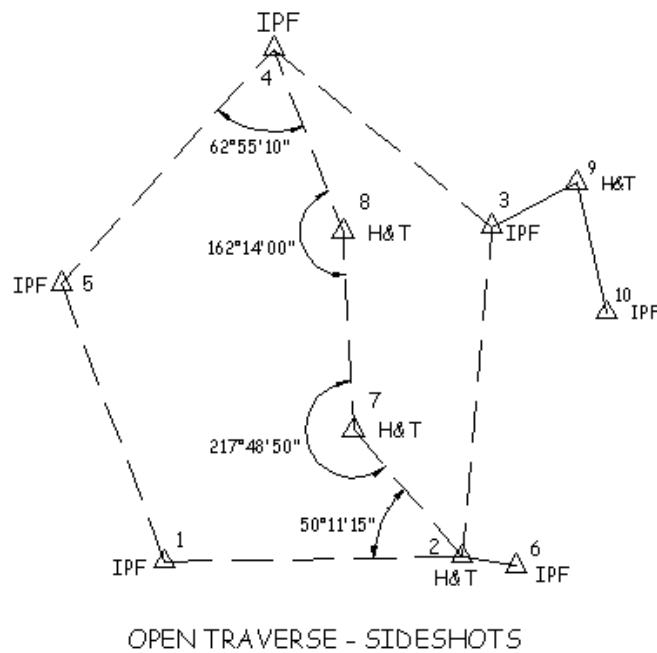


Figure 4

Set	Inst.Point	Inst.Height	Backsight	Rod Height	Horz.Angle	Slope Dist	Vert.Angle	Foresight	Code	Description
IP	3	5.12	4		0.00000					
FS				5.00	113.19430	92.91000	89.15300	9	TP	H&T
IP	9	5.26	3		0.00000					
FS				5.00	284.01250	130.44000	91.15210	10	TP	IPF

Note: The data shown in the CGEditor views accompanying the four illustrations include instrument height (HI) and rod height entries. However, if you have elevations turned off, these entries are optional. Also, the examples use single distance and angle entries but multiple measurements are allowed.

In these figures each traverse has been placed in a separate raw data file. However, with the use of special codes you can combine multiple traverses in a single raw data file.

Entering and Editing Traverse Data

In the CGEditor "Raw Data" refers to unadjusted field traverse data, typically downloaded to the PC from a data collector. C&G raw data files have the extension .CGR.

Creating or Opening a Raw Data File

To create a new file or open an existing file click on the **File** menu then either click on **New** or **Open**. If you click on **New**, another submenu will appear, pick **C&G Raw Data File**. In either case you will then see a file dialog. Browse to the directory where you wish to work and, if creating a new file, type in a file name, or, if opening an existing file, click on a raw data file (*.cgr). Next, click the **Save** button for a new file or the **Open** button for an existing file.

If you are creating a new file, an empty file will be shown in its own document window within the editor. If you are editing an existing file, the data from the file will appear in a similar document window. It is possible to have multiple documents open at the same time. So you could create a new file and open an existing file in the same editing session and each would appear in its own window in the editor. You can have as many new and/or existing

files open as your project demands.

Settings

Before entering any data you should check the current settings. Click the Settings menu item then click Raw Data File to review and/or change the current settings. (See Settings Menu section later in this section.)

Traverse Data Entry

A line or row in the raw data file is referred to as a record and each item of data in a record is referred to as a field. There are several types of records that you may use in a raw data file:

Instrument Point
Foresight
Foresight Tie
Reference Bearing
Coordinate Value
Standard Errors
Control
Measurement
Setup
Elevation
Scale
Loop Traverse
Closed Traverse
Open Traverse
End Traverse
Data on/off
Comment

The type of data required for each of these types of records varies. Some require no data entry and are only "flags" to signify the beginning or ending of a series of records, others require only one field to be filled out, while others require several fields of data.

Adding and Inserting new records

When creating a new file, to begin entering data you must select from the **Add** or **Insert** menus to create the first blank record and begin data entry. Depending on what type of record you are editing, when you press <Enter> for the last field in the record, the following record will be added automatically.

Note: If the **Add** and/or **Insert** toolbars are not showing, click on the View menu then click on the toolbar you want to turn on.

When you click on one of the **Add** menu items or toolbar icons, an empty record is added to the end of the file. If you click on one of the **Insert** menu items or toolbar icons, an empty record is inserted above the currently active record or field. To make a record the currently active record, just click on one of its fields.

Moving from field to field:

While entering data, to move to the next field, press the Enter or the Tab key. To move to the preceding field press the Esc key or both the Shift and Tab keys at the same time.

Insert and Add menus

Insert	
Instrument Point	Ctrl+Alt+P
Foresight	Ctrl+Alt+R
Foresight Tie	Ctrl+Alt+G
Reference Bearing	Ctrl+Alt+B
Coordinate Value	Ctrl+Alt+U
Coords From File	Ctrl+Alt+F
Standard Error	Ctrl+Alt+S
Elevation	Ctrl+Alt+I
Scale	Ctrl+Alt+C
Loop Traverse	Ctrl+Alt+L
Closed Traverse	Ctrl+Alt+D
Open Traverse	Ctrl+Alt+O
End of Traverse	Ctrl+Alt+N
Data On/Off	Ctrl+Alt+X
Comment	Ctrl+Alt+M

Instrument Point records

The first record of a raw data file is often an instrument point. Add or insert a blank record using the menus or toolbars. Fill in the following fields in the new instrument point record:

Inst. Point:

Enter the point ID of the instrument point.

Inst. Height (or HI):

Enter the instrument height. This may be either the distance from the IP on the ground ("Plus-up") or the actual elevation of the instrument, depending on how the data is to be reduced. This field will only be active if elevations are on. (See the Settings section in the Entering and Editing Traverse Data section of this chapter). If elevations are ON and you leave this field BLANK (zero is a valid height), all measurements taken at this setup will be considered 2D and no elevations will be calculated.

Backsight:

Enter the point ID for the backsight.

Rod Height:

Enter the rod height. This field will only be active if elevations are on. (See the Settings section in the Entering and Editing Traverse Data section of this chapter).

Horz. Angle:

Enter the instrument's initial horizontal angle reading at the backsight. When doing an azimuth traverse, no entry is required here.

Note: on doubled angles: Doubled angles require 2 Instrument Point records. Each new instrument setup requires a 0 to the back sight. The first angle to the foresight is the single angle. This angle is locked into the gun and the back sight is retaken. The second angle to the foresight is the doubled angle. You may also double angles to side shots.

Slope Distance and Vertical Angle or Horizontal Distance and Vertical Distance to the Back sight:

Enter the appropriate distance and/or angle. A blank is assumed to be a zero.

Note: When the **Slope Dist/Vert Angle** or **Horz. Dist/Vert. Dist.** column headings are preceded by a "^", it indicates that a record inserted before the current record (or added after the current record) will have the same type of distance entry mode. For example, if the heading shows Dist and Angle and you insert a record, the new record will be in the Slope Dist/Vert Angle distance entry mode. You can change this by clicking on one of the distance headings to remove or add the "^". If the "^" is not present it means that the inserted or added record will have the opposite distance entry mode than does the current record.

If, after entering the data in the last field of a given Instrument Point record, you press the Enter or Tab key, a **Foresight** record will automatically be created. If you want to change this newly created blank **Foresight** record into an **Instrument Point** record, press the Esc key. If you are at the end of the file, pressing Esc again to delete last blank record.

Foresight Point records

After entering the data for the last field in the **Instrument Point** record, press Enter. This will cause a **Foresight** record to be created below it. This record will contain the following columns (the explanations of several of these columns are as described for **Instrument Points**, only the differences will be noted here):

Rod Height:

This column is only active if elevations are on. If elevations are ON and this field is left BLANK, the point will be considered 2D and an elevation will not be calculated. **Horz. Angle:**

Enter the instrument's horizontal angle reading at the foresight point. Enter a positive value for a clockwise angle and a negative value for a counter-clockwise angle. This entry may be blank if you are entering only the distance readings to the foresight.

Slope Dist./Vert Angle or Horz. Dist/Vert. Dist.:

Enter the distance data for the foresight point.

Foresight:

Enter the Point ID for the foresight point.

Code:

Enter the code for the Foresight Point. This column is only active if Code is on. (See Settings in this section.)

Description:

Enter the description for the Foresight Point. The number of characters you are allowed to enter is set in the Settings under Description Length. If you enter an integer code here and the Translate Raw Descriptions Using Description Table is checked in the Settings and a matching description number is found in the description table, then the description from the table will replace the integer value you entered in the Description field. The integer value you entered will then be moved to the Code field.

Note: **Side shots** should be placed within the block of foresights immediately following the instrument point record for the instrument point from which they were shot. You may append side shots to the end of a traverse file, but they must be preceded by a begin open traverse record.

Foresight Tie records

In some cases, you will need to tie to an existing traverse. You use a **Foresight Tie** record to do this. This record is used in the reduction process to determine what known point you are tying into. It is necessary if there are side shots taken at the last setup along with the tie point.

In a closed traverse, you must end a traverse by occupying a known point and turning an angle to a second known point. The second known point is the tie point.

Reference Bearing

From Point

Enter the point ID of the from point.

To Point

Enter the point ID of the to point

Bearing (Azimuth)

Bearings must be entered in the form Qdd.mmsss where Q is the quadrant (1 = NE, 2 = SE, etc), d is whole degrees, m is minutes, and s is seconds (you can specify seconds to the nearest .1 seconds but when you do not wish to specify tenths of a second, a trailing zero is not required)

Azimuth is entered as ddd.mm.sss (when the leading d or the trailing s is zero, it is not required)

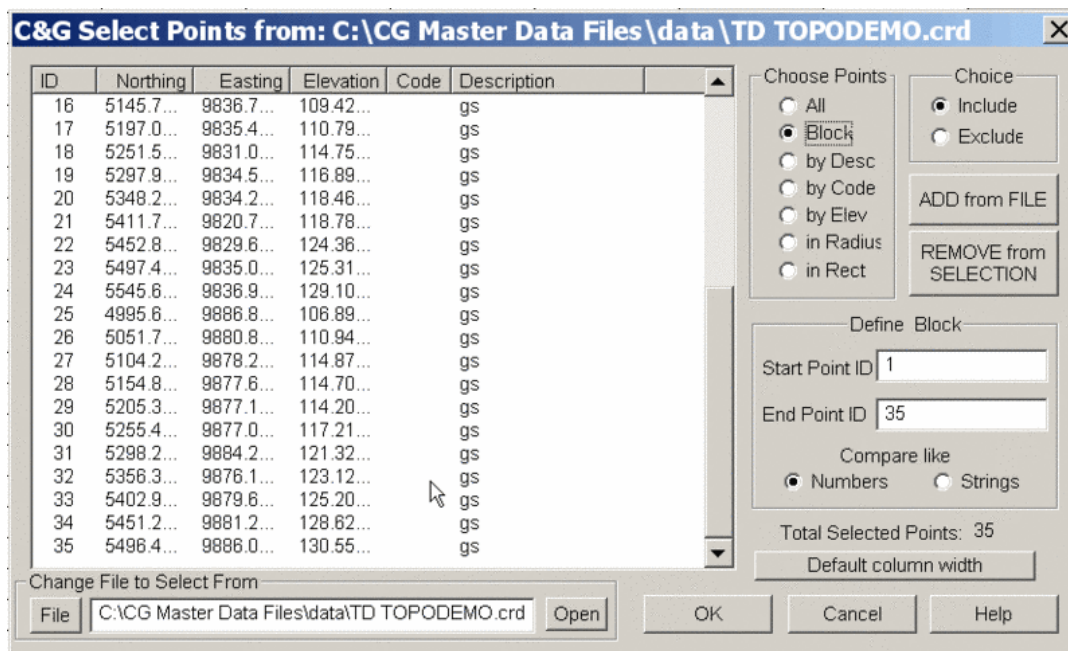
Coordinate Value record

You can use either the Add or Insert menus or toolbars to create a new coordinate record. You can then hand enter known coordinates for a point. Coordinates can be used as a reference point during the reduction process.

Entering Coordinate Value records from a Coordinate File

Instead of hand entering coordinate points, you can insert coordinate records from an existing coordinate file.

Click the Insert menu, then pick the Coords From File menu item.



Elevation

You can specify the elevation for a given point ID using an Elevation record.

Scale

You may specify a scale factor in a Scale record. A scale factor is a decimal number. You may enter as many scale factors as you wish. A scale factor will be used until another is encountered. Scale factors should be placed before an Instrument Point record.

Note: Multiple Traverses: If you are combining more than one traverse in a single raw data file, you must separate the traverses with special records. After inserting or adding a begin traverse record, you may type in a comment regarding the traverse in the Comment column. You may also specify the order in which the traverses are to be processed by using the first part of the Comment field. Please see Traverse reduction order below for more details.

Note: If you are processing the data with SurvNET, the Scale records are ignored. SurvNET calculates scale factors automatically when working on State Plane coordinates.

Beginning and/or ending a Traverse

Note: If you are processing the data with SurvNET, Traverse Records (LT, OT, CT, ET) are ignored. Since SurvNET adjusts all data simultaneously, it requires no traverse definitions.

Use **Loop Traverse**, **Open Traverse** and **Closed Traverse** records to delineate multiple traverses within a single file.

Traverse reduction order

The order in which the traverses appear in the raw data file is typically not important. Traverses are processed in the order in which they appear in the file. Traverses may be entered in a sequential order or you may embed one traverse within another. However, if the coordinates computed from one traverse are needed for the reduction of another traverse, then traverse order IS important. If this condition is true for a raw data file and the traverses have NOT been placed in the raw data file in the correct order, then you need to specify a **Traverse Order Number** for each traverse in the file.

Note: If you specify Traverse Order Numbers, the traverses in the file will be reduced in the order of their Traverse Order Numbers.

Traverse Order Numbers

Each **Loop Traverse**, **Open Traverse** or **Closed Traverse** comment field can contain a Traverse Order Number.

Note: The Traverse Order Number must be an integer and must appear as the first entry in the Comment field separated from the remainder of the comment by a space.

For example, the comment field of a **Loop Traverse** record having a **Traverse Order Number** of 3 should look like this:

3 this is a comment

If any one *Begin Traverse* record has a *Traverse Order Number*, then all *Begin Traverse* records MUST have a *Traverse Order Number*. Also, the *Traverse Order Numbers* in a given file must begin with 1 and continue sequentially. You may not duplicate a *Traverse Order Number* for any *Begin Traverse* record in a given file.

IMPORTANT NOTE: Reducing a raw data file having **Traverse Order Numbers** that violate any of the above specifications will have unpredictable results. Error messages during the reduction process may not reflect the fact that improper traverse order numbering is actually the root cause of the problem.

Loop Traverse

This record indicates the beginning of a loop traverse. A loop traverse begins and ends at the same point. If you wish to add a comment to identify the traverse in some way, just type it in the Comment column.

Closed Traverse

This record indicates the beginning of a closed traverse. A closed traverse ties into known points at both ends. If

you wish to add a comment to identify the traverse in some way, just type it in the Comment column.

Note: If you are running a Closed Traverse and tying into a single point, a reference azimuth must be placed at the last instrument point if you wish to adjust the angular error.

Open Traverse

This record indicates the beginning of an open traverse. An open traverse is a group of side shots. If you wish to add a comment to identify the traverse in some way, use the **Comment** column.

End Traverse

Signals the end of the data records for any of the traverse types.

Comment

Inserts a comment line above the current active line. Comment lines are ignored during processing.

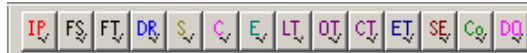
Data On/Off

Data On/Off records surround a series of records that are to be ignored during processing by C&G or SurvNET. The first **Data On/Off** record encountered causes processing to skip to the next **Data On/Off** record. Processing continues beginning at the record after the second **Data On/Off** record. This can be used when trying to isolate errors in a traverse.

The Add and Insert Tool bars



ADD Tool Bar: add the various types of traverse records to the end of the current file.



Insert Tool Bar: insert one of the various types of traverse records above the current record.

Notice that the only difference between the appearance of the Add Toolbar and the Insert Toolbar above is the check mark in the lower right hand corner of each icon of the Insert Toolbar.

Toolbar Icon Explanation

IP Add/Insert an Instrument Point record

FS Add/Insert a Foresight record

FT Add/Insert a Foresight Tie record

DR Add/Insert a Reference Bearing record

S Add/Insert a Scale record

C Add/Insert a known Coordinate point record

E Add/Insert an Elevation benchmark record

LT Add/Insert a Loop Traverse record

OT Add/Insert a nOpen Traverse record

CT Add/Insert a Closed Traverse record

ET Add/Insert an End Traverse record

SE Add/Insert a Standard Error record for Network Least Squares Adjustment (SurvNET) program

Co Add/Insert a Comment record

The Least Squares Toolbar



The "network" icon:

Selecting this icon will start the **SurvNET** Network Least Squares program. If **SurvNET** has already been started, clicking this icon will bring it to the front so you can work with it. (See the **Tools** menu section and the **SurvNET** section for additional info.)

The "eyeball" icon:

This icon brings up a separate window displaying a scaled map of the current raw data file. (See Graphic View under the View menu section)

The "C" icon:

Clicking this icon hides all **Comment** records. The **Comment** records still remain in the raw file, they are just not shown on the screen. You will find that there are some actions you cannot perform when Comments are off.

The "No DO" icon:

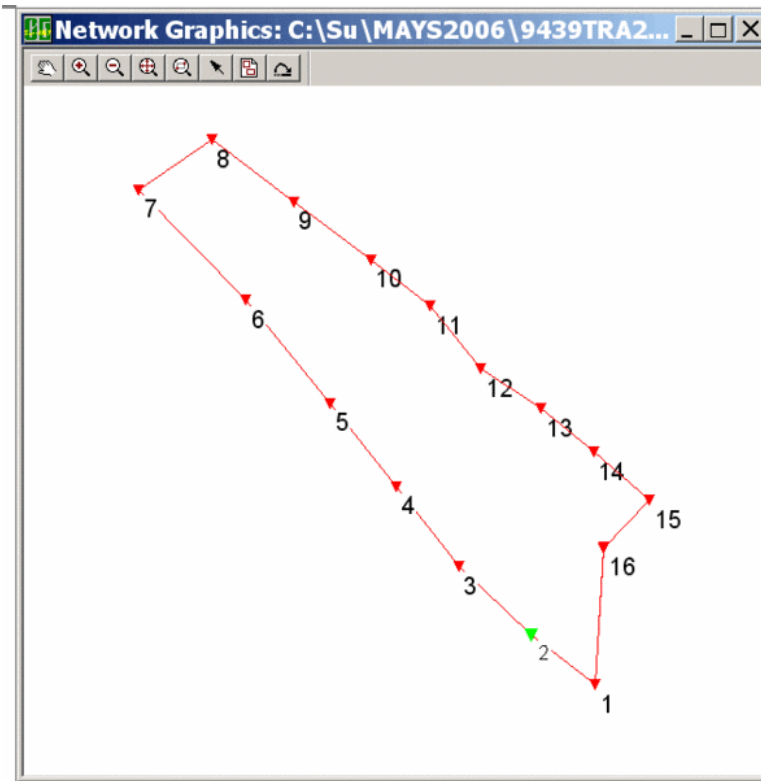
Clicking this icon removes all the **Data On/Off** records from the raw data file.

Status bar

When this menu item is checked, the status bar will display. The status bar is along the bottom border of the **CGEditor** window. On the left side of the status bar a brief help message is displayed when you hold the cursor over such things as menu items or toolbar icons. It also has indicators that tell you if Caps Lock or Num Lock are turned on and displays the Row/record number that is currently active.


Graphic View


Clicking on this menu item brings up a window containing a graphic representation of the traverse. The traverse lines and points are drawn to scale using the data from the current raw data file.





The Graphic View Window

The Graphic View window shows a scaled drawing of the current raw file traverse lines and points. The toolbar icons at the top of the window can be used to move around in the view and change its appearance. The icons will be discussed as they appear from left to right:


 **Pan:** This works very much like the CAD Pan command. When you click the hand icon the cursor changes to a hand. When you click on the graphic screen the first time you are "grabbing" the graphic. You can then move it to the proper view and click a second time to "put it down". You may repeat this as many times as you wish in order to move around the drawing. When done with the Pan command, click on the Pick Point icon.


 **Zoom In:** Clicking on this icon causes the graphic image to be enlarged a preset amount. The zoom factor cannot be configured. If you wish to see a certain area of the graphic image it is recommended that you click Zoom Extents then use Zoom Window to view the desired area.

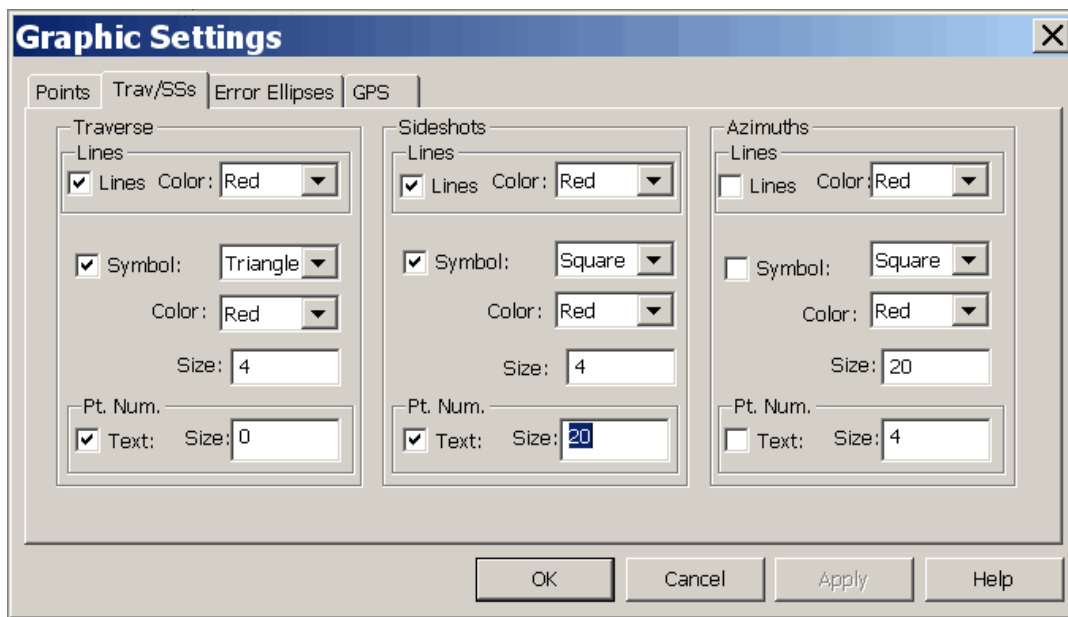
 **Zoom Out:** As with Zoom In, Zoom Out reduces the image size a pre set amount. The zoom factor is not configurable.

 **Zoom Extents:** Zooms the image so all points and lines can be seen on the screen.

 **Zoom Window:** Allows you to click on two diagonal corners of the rectangular area that you wish to see.

 **Pick Point:** Use this icon to allow you to pick a point on the graphics screen in order to "zoom" to the first instance of the associated point ID found in the raw data editor window. This allows you to rapidly and conveniently locate a given point ID in the data file. This is especially useful in trouble shooting for errors or other problems in the data that may be more easily detected in the graphic image than when viewing the raw data. When you pick near a plotted point on the graphics screen its point ID is noted. The raw data file is then searched for that point ID. The active field in the editor window is then set to the first instance of that point ID. You can pick the same location several times to move to the next instance of the point ID in the file. If you have a large Pick Radius set (See Graphic Settings) or are zoomed out, picking a point may result in more than one point being found. If this occurs, a dialog box listing the nearby points will pop up. Using the list box in the dialog choose the desired point ID and press <Enter> or click OK to find the point in the data file.

Clicking this icon also allows you to turn off the Pan feature when you are done panning.  Brings up the **Graphic Settings** dialog:



The graphic settings dialog allows you to configure the appearance of the various items that may be seen on the graphics screen.

Note: The **Graphic Settings** dialog is also used for the **SurvNET** program and thus the items on the **Error Ellipses** and **GPS** tabs have no effect on the **CGEditor Graphic View**.

Points and Trav/SSs tabs

Control Points, Fixed Control Points and Floating Points and Traverse, Sideshots and Azimuths:

Specify whether the symbols, labels or lines for any of these should be shown. Also, if they are to be shown, specify symbol and/or line color, symbol type and point ID label size.

Symbol:

Choose to represent the various types of points as a Square, Triangle or Circle using the drop down list.

Color:

For symbol or line color you can choose Red, Green, Blue, Cyan, Magenta or Yellow from the drop down list.

Size:

Specify the point symbol size.

Pt. Num.

Text: Check the check box if you want the points labeled.

Size: If the points are to be labeled, specify the label height.

Pick Radius

When you pick near a point plotted on the graphics screen, the current field in the editor window moves to the first instance of that point in the current raw data file. Setting the pick radius allows you to specify how large an area around the pick point is to be searched for raw data points drawn in the Graphics View window.

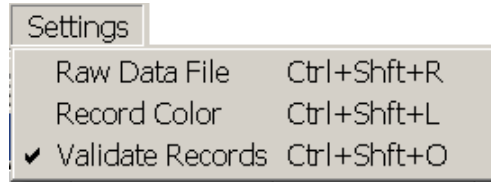
Error Ellipses tab (Has no effect in CGEditor)

GPS tab (Has no effect in CGEditor)

 **Refresh Graphics:** Allows you to refresh the graphics to view recent changes in the raw data due to editing.

Important Note: For the **Refresh Graphics** to reflect recent changes in the raw data file, you must save the file itself prior to refreshing the graphics.

Settings Menu

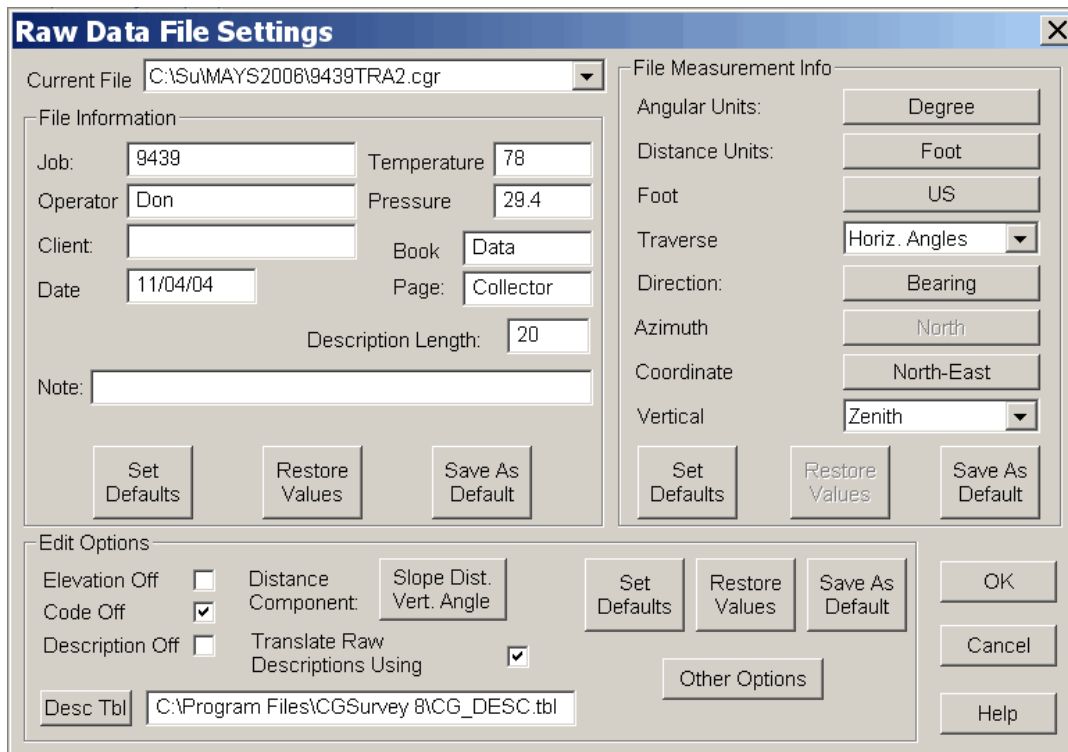


The items in the settings menu can be used to configure how the data in the raw data file will be interpreted and the appearance of that data as seen in the CGEditor.

Raw Data File Settings dialog

When you click on the **Raw Data File** menu item you will see a dialog box that allows you to specify many of the more important settings related to the currently open raw data files. You can also set up the defaults that will be used for newly created raw data files.

Note: See the More on Default Settings subsection at the end of the **Settings Menu** section.



The Raw Data File Settings dialog

Current File

To view and/or edit the settings for a given file, pick the file using the **Current File** list box. You can also view and/or edit the **DEFAULT** settings for newly created files.

File Information

This portion of the dialog allows the user to specify job or project specific information. Except for description length, these items are for your own information and do not affect processing of the raw data.

Job: Enter any name you wish to identify the job or project.

Operator: Enter the name of the person who led the field work.

Client: The name of the person or company for whom this work was done.

Date: Date in any format you wish to use.

Temperature: Temperature at the time the field work was done. For your reference only. May be Celsius or Fahrenheit.

Pressure: Atmospheric pressure at the time the field work was done. For your reference only. May be in any units.

Book: Field book number for the field work.

Page: Page number in the field book.

Description Length: Specify the length of the description field used in this file.

Set Defaults:

This button sets the items in the **File Information** portion of the dialog as the current default values. When a new raw data file is created, these default settings will be used. See the **More on Default Settings** heading at the end of the **Settings** section.

Restore Values:

This button allows you to set the values in the File Information portion of the dialog back to what they were when you opened the **Raw Data File Settings** dialog.

Save As Default:

Sets the default values for the **File Information** portion of the dialog. These values are used as the default settings when a new file is created. See the **More on Default Settings** heading at the end of the **Settings** section.

File Measurement Info

Angular Units: Clicking the button to the right changes the angular units from **Degrees** to **Grads** or vice versa.

Distance Units: Clicking the button to the right changes the distance units from **Foot** to **Meter** or vice versa.

Foot Definition: Clicking the button to the right changes the foot definition from **US** feet to **International** feet or vice versa. This button is only active when **Distance Units** are set to **Foot**.

Traverse Angles: Choose one of the items in the list to specify how the traverse angles were measured:

1. **Horiz. Angles**
2. **Azimuths**
3. **Deflection Angles**

Direction:

Specify what type of angle is used to define the direction of a line. Clicking the button to the right changes the direction from **Bearing** to **Azimuth** or vice versa.

Azimuth Direction:

Specify the reference direction for azimuths. Clicking the button to the right changes the azimuth direction from **North** to **South** or vice versa. This button is only active when **Direction** is set to **Azimuth**.

Coordinate Order: Clicking the button to the right changes the Coordinate Order from North-East to East-North or vice versa.

Vertical Reference: Pick one of the items from the list to the right to specify the reference orientation for measuring vertical angles:

1. **Zenith**
2. **Nadir**
3. **Horizontal**

Set Defaults:

This button sets the items in the **File Measurement Info** portion of the dialog to the current default values. See the **More on Default Settings** heading at the end of the **Settings** section.

Restore Values:

This button allows you to set the values in the **File Measurement Info** portion of the dialog back to what they were when you opened the **Raw Data File Settings** dialog.

Save As Default:

Sets the default values for the **File Measurement Info** portion of the dialog. These values are used as the default settings when a new file is created. See the **More on Default Settings** heading at the end of the Settings section.

Edit Options**Elevation Off:**

Check this check box to turn off the Elevation data entry column for this file. This makes data input more convenient since you do not have to enter any data in the Elevation column, nor do you have to tab through it. Turning off elevations does not cause any data to be deleted from the current file.

Code Off:

Check this check box to turn off the Code data entry column for this file. This makes data input more convenient since you will not have to enter any data in the Code column. Turning off codes does not cause any data to be deleted from the current file.

Description Off:

Check this check box to turn off the Description data entry column for this file. This makes data input more convenient since you will not have to enter any data in the Description column. Turning off descriptions does not cause any data to be deleted from the current file.

Note: You can turn the Elevation, Code and Description data entry columns on or off while editing a file by clicking on the column heading.

Distance Component:

Specify how distances are to be entered. Clicking the button to the right changes the Distance Component from Slope Dist-Vert Angle to Horiz. Dist-Vert. Dist. or vice versa.

Translate Raw Descriptions Using Description Table:

This check box is only active if descriptions are on. If you check this check box, integer codes entered in the Description field will be looked up in the specified description table (See the following item.). If a matching description number is found in the description table, the code will be moved to the Code field and the description found in the description table will be placed in the Description field. If no matching description number is found, the Description field remains as entered.

Desc Tbl:

Click on the Desc Tbl button use a file dialog to set or change the description table. The description table is used to set the Description field when an integer number is entered in the Description field. (See the previous item.) If you prefer, instead of clicking on the Desc Tbl button you can also type in the full file path in the edit box.

Set Defaults:

This button sets the items in the Edit Options portion of the dialog to the current default values. See the More on Default Settings heading at the end of the Settings section.

Restore Values:

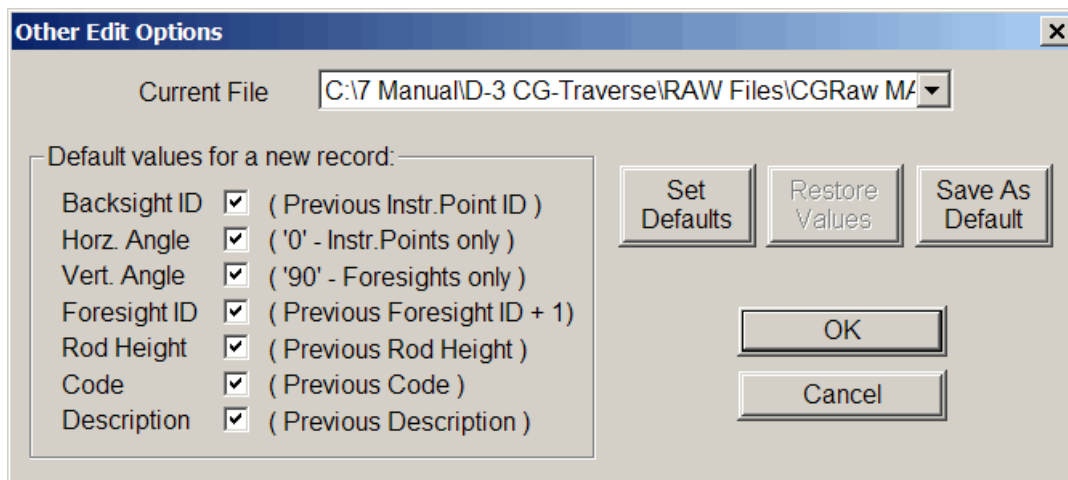
This button allows you to set the values in the Edit Options portion of the dialog back to what they were when you opened the Raw Data File Settings dialog.

Save As Default:

Sets the default values for the Edit Options portion of the dialog. These values are used as the default settings when a new file is created. See the More on Default Settings heading at the end of the Settings section.

Other Edit Options dialog

Click the **Other Options** button to bring up the Other Edit Options Dialog box.



Current File:

Click on the name of the file in the file list for which you wish to review and/or specify the settings. You can also choose to view or edit the DEFAULT settings.

Default values for new record:

Checking the check box for the following items causes CGEditor to "remember" the most recently entered value in the respective field. Thus when you insert or add a record containing one of the checked items, it will be filled in with a "default" value.

Backsight ID

Horz. Angle

Vert. Angle

Foresight ID

Rod Height

Code

Description

Note: The previously used field values are not "remembered" and thus will not be used to fill in new records the next time you open the **CGEditor**.

Set Defaults:

This button sets the items in the Other Edit Options dialog to the current default values. See More on Default Settings at the end of the Settings section.

Restore Values:

This button allows you to set the values in the Other Edit Options dialog back to what they were when you opened the Raw Data File Settings dialog.

Save As Default:

Sets the default values for the items found in the Other Edit Options dialog. These values are used as the default settings when a new file is created. See More on Default Settings at the end of the Settings section.

Click **OK** to close the **Other Edit Options** dialog.

Click **OK** to close the **Raw Data File Settings** dialog.

More on Default Settings:

When the CGEditor is started from CGSurvey, many of the initial default settings may not be those you had specified in a previous session. This is because many of the default settings you previously specified were overridden by the current CG Settings specified in CGSurvey. However, you may yourself override the default settings for the current session only by changing any of the settings and clicking the Save As Default button. If you wish to change the "default" settings for future editing sessions, you must change the CG Settings in CGSurvey.

Settings overridden by the settings on the various tabs in the CAD C&G Options dialog:

File Information: only **Description Length** is overridden by the settings in the CAD C&G Options dialog.

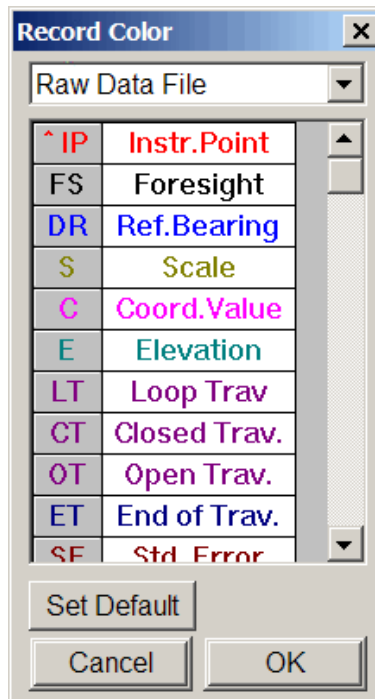
File Measurement Info: ALL items are overridden by the settings in the CAD C&G Options dialog.

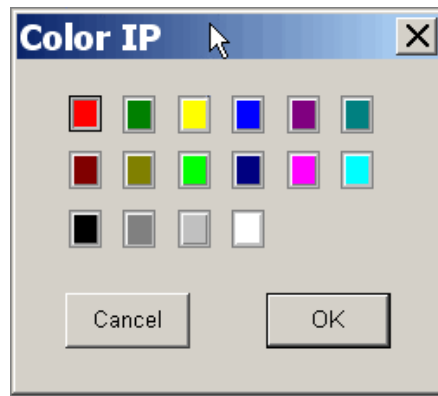
Edit Options: ALL items are overridden by the settings in the CAD C&G Options dialog.

Other Edit Options: NONE are overridden by the settings in the CAD C&G Options dialog.

Record Color

To set the color for a given record type click the **Record Color** menu item. Then, in the **Record Color** dialog, click on the record type and a color selection dialog will appear. Click on the color you want the that record type to have. If you click the **Set Defaults** button, the original program default colors are set. Click the **OK** button to save the color settings and close the dialog. Click the **Cancel** button to close the dialog without saving the changes.

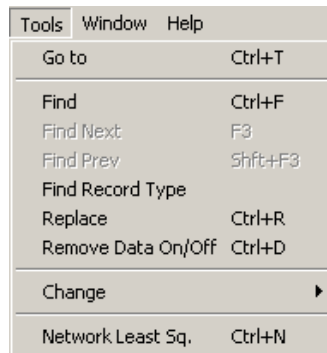




Validate Records

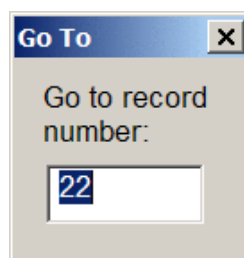
If this menu item is checked, all the records in the file will be validated prior to saving the file. To change the Validate Record setting, just click the menu item. If an invalid record is encountered when saving a file with the Validate Records menu item checked, you are asked if you want to edit the invalid field, ignore the error or ignore all errors. If you decide to edit the offending field, the field will be highlighted and you can edit it and attempt to save again.

The Tools menu has several items that can be used to find and replace specific text in specific types of fields. It even allows you to apply simple mathematical functions to allow you to edit the data in a group of fields in a single step.



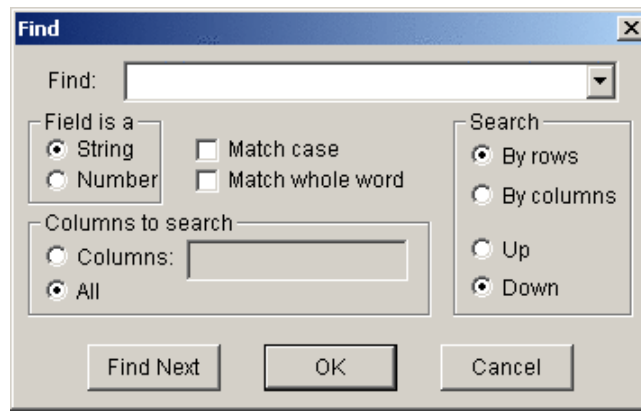
Goto (Ctrl + T):

Select this item to go to a certain row (or record) number. In the dialog box that comes up, type in the desired row number and click OK. The editor window will zoom to that record and set the current field to the first editable field in the record.



Find (Ctrl + f) menu item:

The **Find** dialog allows you to enter a value to find and set the detailed search criteria.



Find: Type in the string or number you are searching for in the edit box or pick a previous search string from the list.

Field is a: Choose what type of data is in the field you are looking for. Check appropriate checkbox for matching case and/or whole word.

Columns to search:

The default is to search **All** columns, but if you choose the **Columns** radio button, you can enter a comma separated list of column numbers. The column to the right of the TYPE column is column 1 and it is the first column in which you can search.

Search:

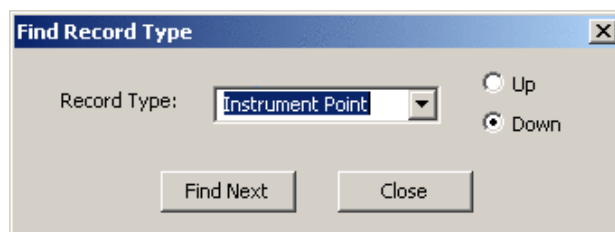
You can search **By Rows** or **By Columns** and you can choose to search **Up** or **Down** from the current field.

Once you have specified the parameters for the search, click the **Find Next** button to find the first instance of the search string. Continue to click the **Find Next** button to find the next instance of the string. To just find the next instance of a string and close the dialog box, you can click **OK**.

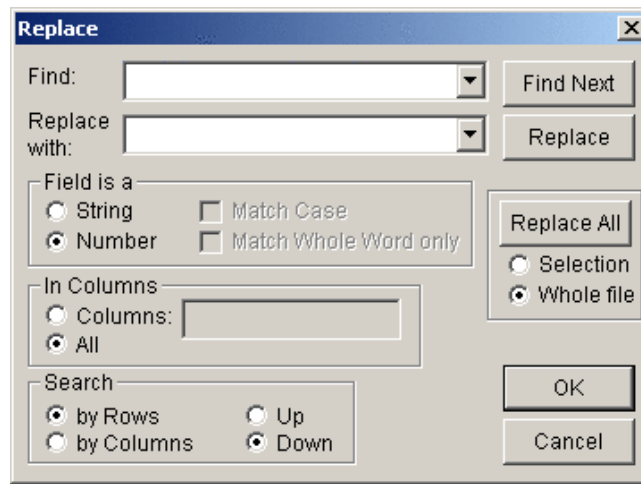
Find Next (F3) menu item: Finds to the next occurrence of the string previously specified in the **Find** dialog.

Find Prev (<Shift> + <F3>) menu item: Moves you to the previous occurrence of the string previously specified in the **Find** dialog.

Find Record Type menu item: Allows you to find the next record type of the type specified. The search starts at the current record. When you click this menu item, the **Find Record Type** dialog box is displayed. Choose the record type you wish to look for by picking from the list then specify the direction of search and click the **Find Next** button to find the record. Click **Cancel** when done.



Replace (<Ctrl> + r) menu item: When you click on this menu item, the **Replace** dialog appears.



The **Replace** dialog allows you to specify a **Find:** value and a **Replace with:** value. The other fields in the **Replace** dialog are the same as the **Find** dialog. You can view the **Find:** value one instance at a time by clicking the **Find Next** button, if you decide to replace a given value found just click the **Replace** button. Alternatively, you can allow the software to automatically replace all the instances of the **Find:** value encountered in the specified columns in the raw data file by clicking the **Replace All** button.

Note: Before clicking the **Replace All** button, be sure to specify whether you wish to replace matching fields in the highlighted **Selection** of fields/records or in all the fields in the **Whole File**.

Data On/Off (<Ctrl> + d) menu item:

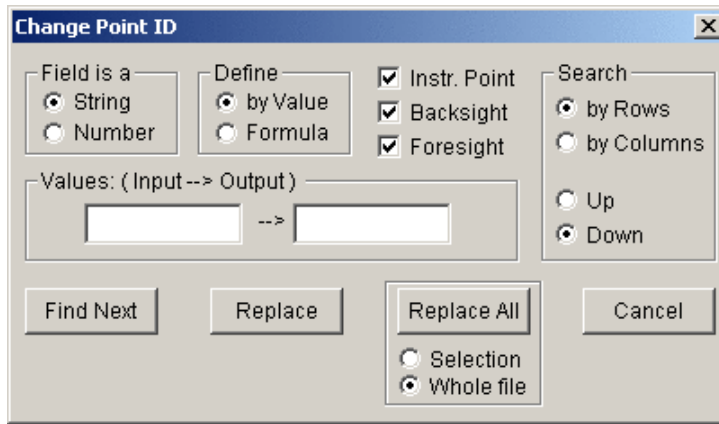
Selecting this menu item inserts a Data On/Off record above the current record. Records between pairs of Data On/Off records are ignored when the traverse is reduced. This can be useful when trying to find problems in a traverse.

Change

The items in this submenu allow you to change specific types of fields in the raw data file.



Point ID (<Ctrl> + I) menu item: This menu item allows you to change point IDs for instrument points, back sight points, or foresight points. You can change individual points one at a time or you can make a global change. You can specify a value to find and a value to replace it with. The **Change Point ID** dialog has several sections that are similar to the **Replace** dialog



Field is a: You must specify how you want to treat the point ID field. You can do this by clicking on the **String** or **Number** radio buttons.

Define: You must specify whether you wish to specify the replacement value **by Value** or **Formula**.

Note: The **Values: (Input -> Output)** section of the dialog changes its title to **Formula:** when you elect to **Define** by **Formula**. Also, the content of this portion of the dialog changes according to the field type (see **Values: or Formula:** section below).

Instr. Point, Backsight, and Foresight check boxes: Check the check boxes of the types of point IDs you wish to change.

Values: or Formula: section

When **Define** is set to **by Value** and **Field is a** is specified as either a **String** or a **Number** then the title of this section of the dialog becomes

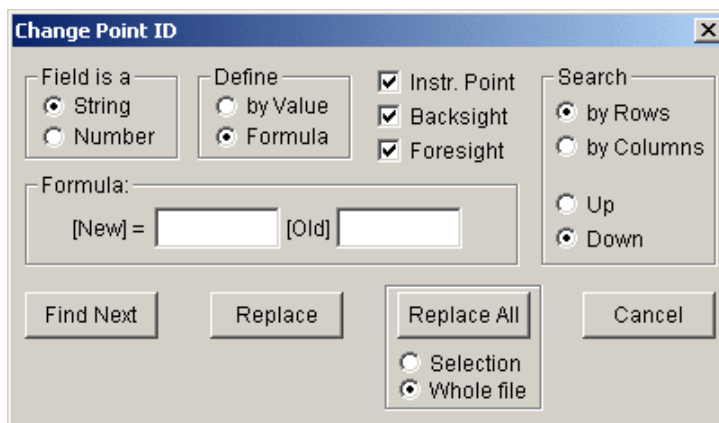
Values: (Input -> Output) (as shown in the dialog above). In this configuration the **Change Point ID** dialog functions like the **Replace** dialog except that it only searches the point ID fields specified.

Specify the value to search for in the edit box to the left of the "-->" and the value to replace it with in the edit box to the right of the "-->".

The **Find Next**, **Replace** and **Replace All** buttons act exactly the same as the **Find Next**, **Replace** and **Replace All** buttons in the **Replace** dialog.

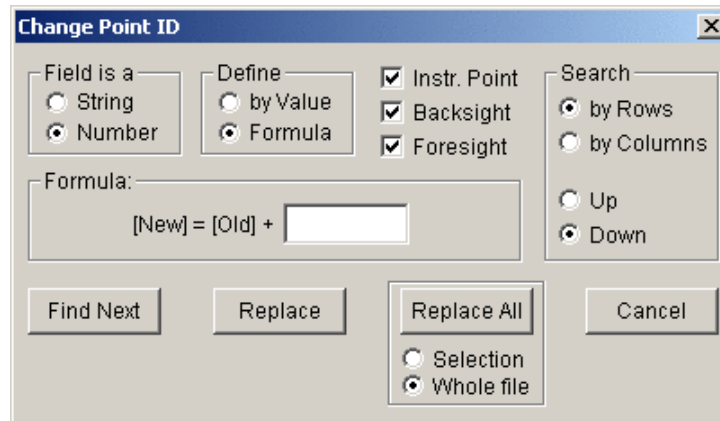
When **Define** is set to **Formula** the title of this section of the dialog becomes **Formula:**

If **Field is a** is specified as a **String**, the dialog is as shown below:



In this configuration the formula acts to add a prefix and/or a suffix to the existing point ID (represented by **[Old]**). Enter the prefix in the edit box to the left of **[Old]** and the suffix in the edit box to the right of **[Old]**. If you do not wish to add a prefix or you do not wish to add a suffix, you may leave either the left or right hand edit boxes empty.

If **Field is a** is specified as a **Number**, the dialog is as shown below:

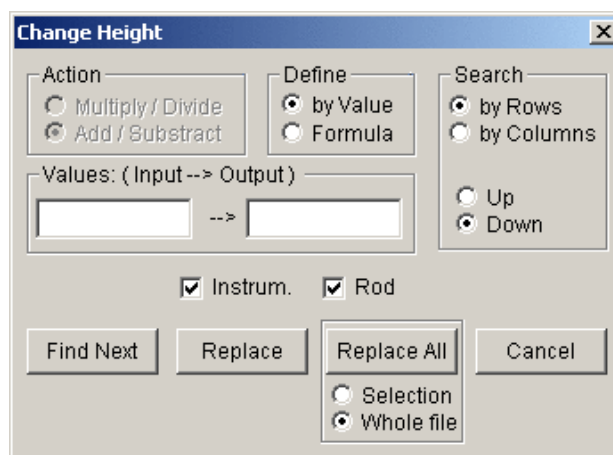


In this configuration the formula adds a specified number to a given point ID. Enter the positive or negative number in the edit box to the right of "[Old] +".

NOTE: When the **Field is a** is specified as a **Number** and a point ID containing non-numeric characters is encountered, it will be skipped and no change will be made to it.

Change Height (<Ctrl> + h)

Use this menu item to change the instrument height and/or rod height. Clicking this menu item brings up the **Change Height** dialog.



Action section of dialog

Use this section to determine how the height is to be changed when **Define** is set to **Formula**.

Multiply/Divide: Choose this if you wish to multiply or divide the height by a given number.

Add/Subtract: Choose this if you wish to add a specified number to the height or subtract a specified number from the height.

Define section of dialog by Value: If you choose by Value, this command becomes like the Replace command, except that it acts only on instrument heights and/or rod heights.

Formula: This allows you to specify a number to apply to the height by addition, subtraction, multiplication, or division. (See the **Action** and **Values:/Formula:** sections.)

Values:/Formula: section of dialog

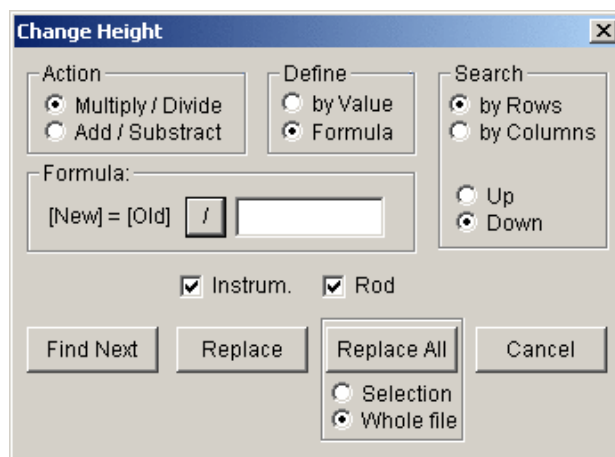
Depending on what you choose in the Action and Define sections there are several possibilities for this section of the dialog:

When **Define** is set to **by Value** the **Action** section of dialog is disabled and the title of this section becomes **Values: (Input->Output)**

In this configuration the feature functions like the **Replace** command, except that it acts only on instrument heights and/or rod heights.

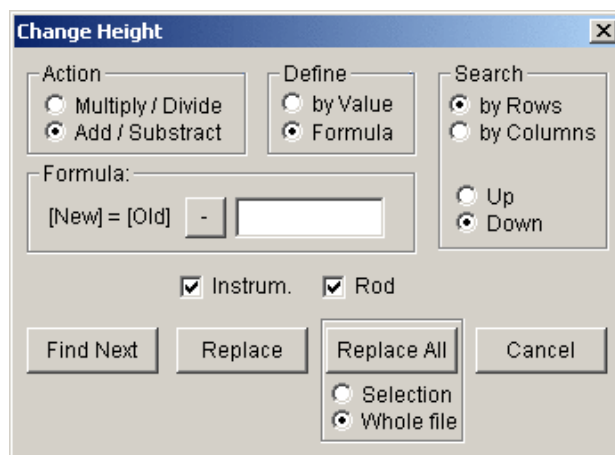
When **Define** is set to **Formula**, the **Action** section of dialog is enabled and the title of this section becomes **Formula:**

When the **Action** is set to **Multiply/Divide**, the **Formula:** section changes as seen below:



In this configuration you can multiply or divide the instrument height or rod height by the number specified in the edit box. To switch between multiply and divide, just click on the button with the multiply ("*") or divide ("/") symbol on it.

When **Action** is set to **Add/Subtract**, the **Formula:** section changes as seen below:



In this configuration you can add or subtract the number specified in the edit box to or from the instrument height or rod height. To switch between add and subtract, just click on the button with the add ("+") or subtract ("-") symbol

on it.

Search section of the dialog

Use this section of the dialog to specify how the records will be searched. The search begins at the currently active field.

Instrum. and **Rod** checkboxes: Check one or both of these check boxes to specify which types of heights are to be searched/changed.

Find Next button: Use this button to move to the next field that matches the specifications you entered.

Replace button: Use this button to replace the highlighted text that was found.

Change All button: Use this button to make the changes specified to all matching fields in the file. Be sure to specify whether to apply the changes to the highlighted **Selection** (records or fields) or to the **Whole file**.

Cancel button: Click the Cancel button to close the dialog.

Change Angle (<Ctrl> + g)

Choose this menu item to change vertical and/or horizontal angle fields. Clicking the **Change Angle** menu item brings up the **Change Angle** dialog: This dialog is almost identical to the **Change Height** dialog and will not be described in detail. The differences are: the **Multiply/Divide** action seen in the **Change Height** dialog is replaced by the **Make Opposite** action; you can check either the Vertical or Horizontal check boxes to specify the angles you wish to change; choosing **Formula** and **Make Opposite** disables the **Formula:** section of the dialog due to the fact that the action to be taken is merely to reverse the sign of the angle.

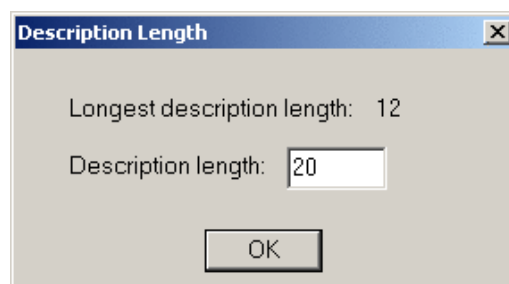
Change Distance (<Ctrl> + D)

The **Change Distance** dialog is almost identical to the **Change Height** dialog. The only difference is that you can choose to change the **Slope** distance and/or the **Horizontal** distance by checking the checkboxes.

Change DescLen (<Ctrl> + j)

This command allows you to set the description length for the current raw data file. It displays the **Longest description length:** that is found in the current records in the file. It allows you to specify a new **Description length:**.

Warning: If you specify a length less than the longest description found in the file, the descriptions that exceed that length will be truncated.



Network Least Sq. menu item

This menu item runs the **SurvNET Network Least Squares Adjustment** program. Please refer to the section on SurvNET for a detailed description of this very powerful traverse and level loop adjustment program.

Window menu

This menu contains many of the standard Window menu items found in other programs. It allows you to arrange the currently open windows in several configurations. It has the added functionality of the New Window command which allows you to have two or more views of a single file

Help

For information regarding the CGEditor program version click the About CGEditor... menu item.

Editing C&G Mapcheck Files

Mapcheck files are typically used to check the closure of a given parcel of land given the deed description of that parcel. A mapcheck file may contain straight line boundaries as well as boundaries described by both tangent and non-tangent curves.

Creating or Opening a Mapcheck File

To create a new file or open an existing file choose **File** on the main menu then either click **New** or **Open**.

If you choose **New** a submenu will appear, click the **C&G Mapcheck File** menu item.

In either case you will then see a file dialog. Browse to the directory where you wish to work and, if creating a new file, type in a file name, or, if opening an existing file, click on a mapcheck file (*.cgm). Next, click the **Save** button for a new file or the **Open** button for an existing file.

If you are creating a new file, an empty file will be shown in its own document window within the editor. If you are editing an existing file, the data from the file will appear in a similar document window. It is possible to have multiple documents open at the same time. So you could create a new file and open an existing file in the same editing session and each would appear in its own window in the editor. You can have as many new and/or existing files open as your project demands. You may also cut, copy and/or paste between files.

Settings: Before entering any data you should check the current settings. Click the **Settings** menu item then click **Map Check File** to review and/or change the current settings. (For more details, see the **Settings Menu** section of **Editing C&G Mapcheck Files**.)

Mapcheck Data Entry

Opening an existing template file or creating a new one is very similar to opening or creating a raw traverse data file. There are three types of records that you may use in a mapcheck file:

Straight line (identified as **Line** in the **Type** column)

Tangent Curve (identified as **TC** in the **Type** column)

Non-tangent Curve (identified as **NTC-C** or **NTC-R** in the **Type** column for Chord or Radius definition NTC records)

Adding and Inserting new records

To create a new record in the current file you must either use the **Add** or **Insert** menu item or the **Add** or **Insert** toolbar.

Note: If the **Add** and/or **Insert** toolbars are not showing, click the **View** menu then choose the menu item for the toolbar you want to turn show.

When you click on one of the **Add** menu items or toolbar icons, an empty record is added to the end of the file. If you click on one of the **Insert** menu items or toolbar icons, an empty record is inserted above the currently active record or field. To make a record the currently active record, just click on one of its fields.

Moving from field to field: While entering data, to move to the next field, press the Enter or Tab key. To move to the preceding field press the Esc key or the Shift and Tab keys at the same time.

Straight Lines

There are two fields to be filled out in a **Straight Line** (or **Line**) record:

Bearing or Azimuth: For a bearing, use the standard C&G bearing notation:

For Bearing: Qdd.mmsss

Where

q = quadrant (1 = NE, 2 = SE, 3 = SW, 4 = NW)

d = 2 digit bearing

m = minutes

s = seconds and tenths of seconds

For example: enter S 35° 22' 34.2" E as 235.22342

For Azimuth, use the notation: ddd.mmsss

Distance: Enter the length of the boundary in whatever units you have specified in the **Map Check File Settings**.

Code: Enter a code (optional).

Note: If **Code Off** is checked in the **Map Check File Settings** dialog, this field will not be active. However, clicking on the **Code** column title will turn it on.

Description: Enter a description (optional).

Note: If **Description Off** is checked in the **Map Check File Settings** dialog, this field will not be active. However, clicking on the **Description** column title will turn it on.

If **Translate Mapcheck Descriptions Using a Description Table** is checked in the **Map Check File Settings** dialog and you have entered an integer number description, then when you move to the next field, the description table will be searched for a description number matching the integer entered. If a matching description number is found, the description from the table will be placed in the **Description** field and the integer originally entered in the **Description** field will be placed in the **Code** field.

Tangent Curves

For a Tangent Curve record there are six possible fields to enter. Of the following six fields you must enter data for two of the first four:

Radius - decimal distance

Arc Length - decimal distance

Chord - decimal distance

Central Angle - angle specified as ddd.mmsss (degrees.minutes and seconds to nearest .1 sec.)

Code (optional - see Straight Lines above)

Description (optional - see Straight Lines above)

Non-Tangent Curves

The fields in a **Non-Tangent Curve** record vary according to whether it is defined using the chord bearing/azimuth or radius bearing/azimuth.

When using **Non-Tangent Curve** record it is necessary to specify whether the chord or radius definition will be used when specifying the curve . There are four ways to accomplish this:

1. Prior to Inserting or Adding the record, use the **Settings** menu then choose **Map Check File**. In the **Map Check File Settings** dialog set the **Curve Definition** in the **File Measurement Info** section of the dialog.
2. Prior to Inserting or Adding the record, use the **Settings** menu to check or uncheck the **Non-Tan Curves Use Chord** menu item. When the **Non-Tan Curves Use Chord** menu item is checked, newly created **Non-Tangent Curve** records will added or inserted that use the chord definition, otherwise they will use the radius definition.
3. Prior to Inserting or Adding the record, click the **C-R** toolbar icon. When the icon appears depressed, newly created **Non-Tangent Curve** records will use the chord definition, otherwise they will use the radius definition.
4. To change the type of curve definition for an existing **Non-Tangent Curve** record, use the Edit main menu and choose the Change Curve Def'n menu item. This changes the current record from what it is now to the opposite type of curve definition.

For both the **Chord** and **Radius** definitions the following fields are present in the record:

Chord or Radius Brg/Azimuth

used to orient the curve properly as it leaves the PC. As noted in the **Tangent Curves** section, bearings must be entered in the qdd.mmsss format and azimuths entered in the ddd.mmsss format. **Radius**

Arc Length

Chord

Central Ang

Code

Description

All but the first field has been discussed earlier in the **Tangent Curves** section and will not be described here.

Editing a Mapcheck File

Most of the menu items found in the mapcheck menus have been discussed in the **Editing Traverse Raw Data Files** section. Only the differences will be discussed here.

File Menu: The **File** menu when editing a mapcheck file is identical to the **File** menu discussed in the **Editing Traverse Raw Data Files** section.

Edit Menu: With the exception of the **Change Curve Def'n** menu item, the **Edit** menu is identical to the **Edit** menu discussed in the **Editing Traverse Raw Data Files** section. **Change Curve Def'n** was discussed above in the **Non-Tangent Curves** section

Add Menu: The Add menu allows you to add **Straight line**, **Tangent Curve** and **Non-Tangent Curve** records to the end of the file.

Insert Menu: The Insert menu allows you to insert **Straight line**, **Tangent Curve** and **Non-Tangent Curve** records above the current record.

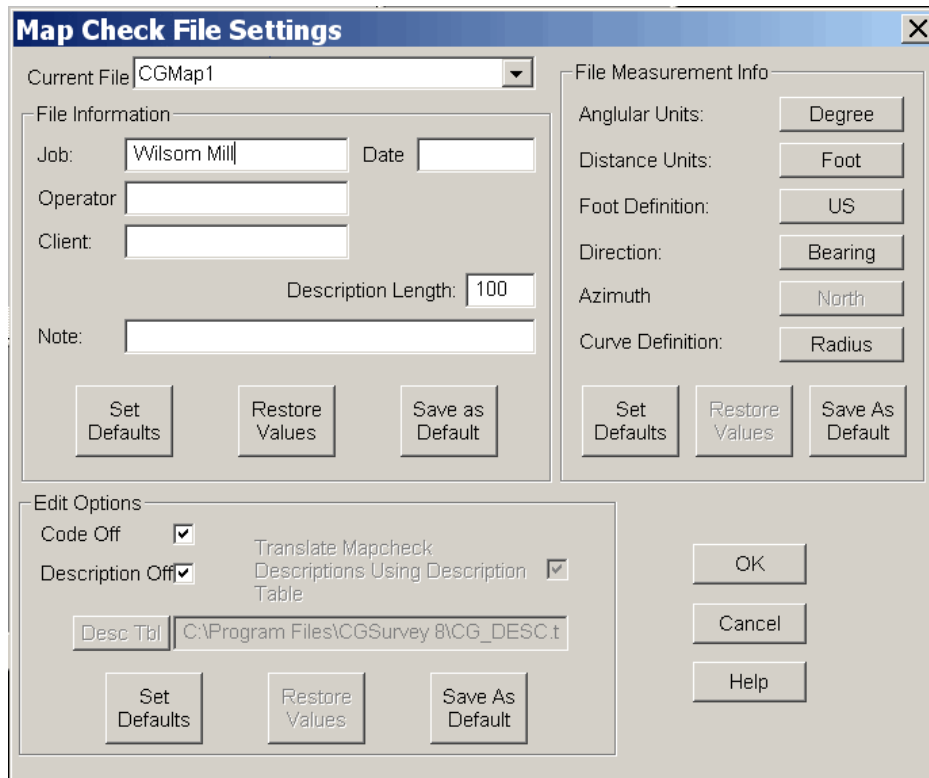
View Menu: Allows you to turn the toolbars on and off.

Settings Menu

The **Settings** menu contains items that allow you to specify the format of the data in a mapcheck file and how this data will appear in the **CGEditor**.

Map Check File settings menu item

The **Map Check File** menu item brings up the **Map Check File Settings** dialog (see below). This dialog allows you to specify settings for each of the mapcheck files currently open in the editor. It also allows you to specify the default settings for creating new map check files.



Current File: Use this list to choose a file you wish to set or view the settings for. You may also set or view the **DEFAULT** settings that are used for newly created files.

File Information and Edit Options:

The settings in the File Information and Edit Options sections have been discussed under the Settings Menu section of Editing a Raw Data File.

File Measurement Info:

Most of the settings in the File Measurement Info section have been discussed under the Settings Menu section of Editing a Raw Data File. However, a **Curve Definition** item has been added to this section for mapcheck files:

Curve Definition: click the **Curve Definition** button to change from Chord to Radius definitions and vice versa. **Curve Definition** only applies to the insertion or addition of **Non-Tangent Curve** records.

Record Color menu item

The **Record Color** menu item has been discussed under the **Settings Menu** section of **Editing Traverse Raw Data Files**. The only difference is that here you are setting the colors for the various types of mapcheck records instead of raw data records.

Validate Records

This menu item allows you to set whether records are validated prior to being saved. (See also, **Validate Records** in the **Settings Menu** section of **Editing Traverse Raw Data Files**.)

Non-Tan Curves Use Chord: Use this to switch which types of **Non-Tangent Curve** records are added or inserted.

Tools, Window and Help Menus the items in these menus have been discussed in the **Editing Traverse Raw Data Files** section

C&G Cross Section Files

Cross section files contain data which defines one or more topographic or design cross sections along an alignment. Any features using a cross section file assume that it is at right angles to the alignment. Each cross section is identified by its station along the alignment. Each cross section is defined by a **Station** record specifying a station on the alignment followed by a series of **Point** records specifying the offset and elevation of points on the cross section at that station. Cross sections can be used to visualize a site, specify design elevations and calculate volumes. Opening an existing cross section file or creating a new one is very similar to opening or creating a map check file.

Cross Section File Data Entry

Station Records: There are three fields to be filled out in a Station record:

Station: Specifies the station of this cross section along the alignment. For example: station 6+45.37 is indicated as 645.37.

Left Slope:

This field defines the slope at the left side of the cross section in feet per foot (or meters per meter if units are set to meters). This slope will be used to extend this cross section to meet any cross section it overlays.

Right Slope:

This field defines the slope at the right side of the cross section in feet/foot (meters/meter). This slope will be used to extend this cross section to meet any cross section it overlays.

Point Records

There are two fields in a Point record:

Offset: The **Offset** defines the perpendicular distance from the alignment to this point on the cross section.

Elevation: The Elevation specifies the elevation of this point on the cross section.

Cross Section File Data Editing

Adding and Inserting new records: To create a new record in the current file you must either use the **Add** or **Insert** menu or toolbars.

Note: If the toolbars are not showing, click on the **View** menu then click the item for the toolbar you want to turn on.

Settings Menu item

Record Color:

The **Record Color** menu item has been discussed under the **Settings Menu** section of **Editing Traverse Raw Data Files**. The only difference is that here you are setting the colors for the various types of cross section records instead of raw data records.

Validate Records: This menu item has been described in the **Settings Menu** section of **Editing Traverse Raw Data Files**

US Foot: If this menu item is checked units are US feet. If the **Meters** menu item is checked, this menu item is disabled.

International foot: If this menu item is checked units are International feet. If the **Meters** menu item is

checked, this menu item is disabled.

Feet: If this menu item is checked units are Feet.

Meters: If this menu item is checked units are meters.

Note: The settings for US Foot and International foot will be ignored if Meters is checked.

C&G Template Files

Template files contain data defining standard cross section templates that can be used to create a cross section file that represents the design cross sections for a proposed alignment. Cross section files created using templates can be overlaid on existing cross sections to allow the computation of cut and fill volumes and to visualize the design alignment. Opening an existing template file or creating a new one is very similar to opening or creating a map check file.

Entering and Editing Template Data

Entering and editing template data is analogous to that described in **Entering and Editing Cross Section Data** except that, instead of being identified by their station along the alignment, templates are identified by an integer template identifier. This identifier is used when building a cross section from templates in order to specify a template among the many that a template file may contain. Templates are placed along a proposed alignment at various stations and thus create a series of cross sections using the alignment elevation to set the elevation of the template points. When building cross sections along an alignment using templates, cross sections at stations between two template stations result in a series of cross sections being created to transition between the templates.

Template File Data Entry

Template Records: There are five fields to be filled out in a Template record:

Template: Template number for identifying the template

Left Slope:

Specifies the slope at the left side of the template in feet/foot (meters/meter). This slope will be used to extend this template generated cross section to meet any cross section it overlays. **Right Slope:**

Enter the slope at the right side of the template in feet per foot (or meters per meter units are set to meters). This slope will be used to extend this template generated cross section to meet any cross section it overlays. **Offset:**

The Offset defines the distance from the centerline of the template to this point on the cross section. The template centerline should be assigned a 0.0 offset. The 0.0 offset is placed on the alignment when cross sections are generated from templates.

Elevation:

The Elevation specifies the elevation of this point on the template. If the elevation of the centerline point is set to 0.0, then this elevation can be used to directly compute the elevation of the point based on the elevation of the alignment where the template is placed.

Editing a Template File: All template menu items and editing procedures are identical to those described for cross sections.

Editing Coordinate Files

Coordinate files contain data on the Point IDs, Northings, Eastings, Elevations, Descriptions and, for C&G files, Codes for various points located in the field and points created by calculations and/or by hand data entry. The coordinate file may have points from a single job, portions of a single job or many jobs. The Point ID must be a unique identifier for a given point. Typically Point IDs are integer numbers but may also be any combination of

letters and numbers depending on the format of the file.

The CGEditor can be used to edit six different types of coordinate files. All the supported coordinate file types have **Point ID**, **Northing**, **Easting**, **Elevation**, and **Description** fields. In all formats, any given point may have a blank **Description** field. The types of files supported and a brief description of their differences follows:

C&G Numeric (*.crd)

Point ID: any integer number between 1 and 65,536.

Description: The maximum description length for a given file can vary between 1 and 100 characters and is set when the file is created. A given point description entry may be blank.

Code: up to 4 characters long. Used to filter and sort points. The **Code** field may be blank.

C&G Alpha-numeric (*.cgc)

Point ID: up to 10 characters long and can contain any combination of alphabetic and numeric characters.

Description: The maximum description length for a given file can vary between 1 and 100 characters and is set when the file is created. A given point description entry may be blank.

Code: up to 4 characters long. Used to filter and sort points. The **Code** field may be blank.

Carlson Numeric (*.crd)

Point ID: any positive integer number containing 1 to 9 digits.

Description: entries can be from 0 to 31 characters long.

(the **Code** field is not supported.)

Carlson Alpha-numeric (*.crd)

Point ID: any a series of from 1 to 9 alphabetic or numeric characters.

Description: entries can be from 0 to 31 characters long.

(the **Code** field is not supported.)

Simplicity (*.zak)

Point ID: can be any positive integer number containing 1 to 8 digits.

Description: entries can be from 0 to 28 characters long.

(the **Code** field is not supported.)

Land Desktop (*.mdb)

Point ID: can be a series of from 1 to 255 alphabetic or numeric characters.

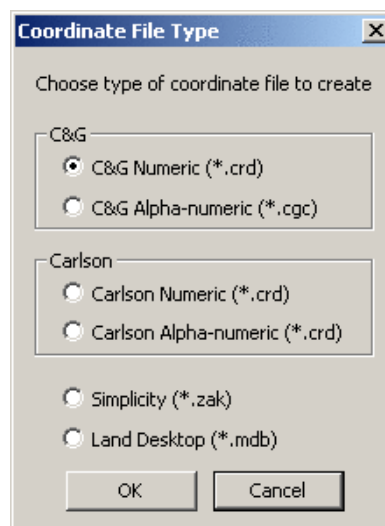
Description: entries can be from 0 to 255 characters long.

(the **Code** field is not supported.)

Creating or Opening a Coordinate File

To create a new file or open an existing file choose **File** on the main menu then either click **New** or **Open**.

If you choose **New** a submenu will appear, click the **Coordinate File** menu item or click on the "C" icon in the **Standard** toolbar. Next pick the type of coordinate file you wish to create using the **Coordinate File Type** dialog:



A new coordinate file with a temporary name will appear in its own document window in the CGEditor and will contain only a single blank coordinate **Point** record.

If you are opening an existing file using the **Open** menu item, you will be asked to choose the file using file dialog. Browse to the directory where you wish to work click on a coordinate file and click the **Open** button The coordinate records from the file will appear in a separate document window in the CGEditor.

It is possible to have multiple documents open at the same time. So you could create a new file and open an existing file in the same editing session and each would appear in its own window in the editor. You can have as many new and/or existing files open as your project demands. You may also cut, copy and/or paste between files.

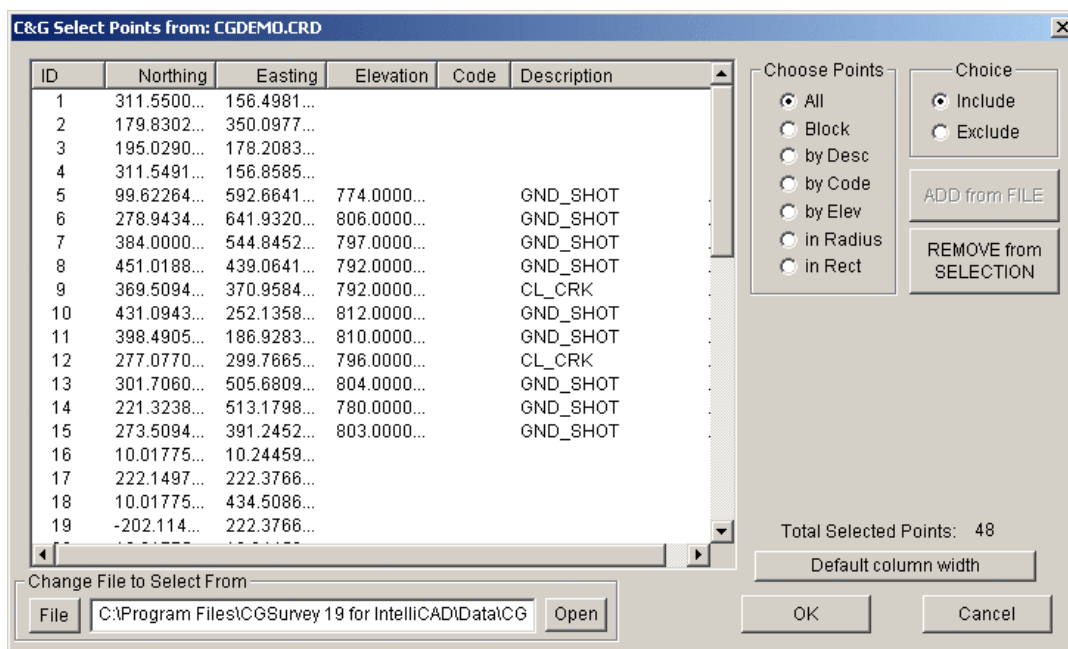
Settings: Before entering any data you should check the current settings. Click the **Settings** menu item then click **Coordinate File** to review and/or change the current settings. (For more details, see the **Settings Menu** section of **Editing Coordinate Files**.)

Entering and Editing Coordinate File Data

Once a coordinate file has been opened or created, you can edit any of the fields in any of the records. To create a new coordinate point you must use the Add/Insert main menu or toolbar. Both the Add/Insert menu and toolbar allow you to add or insert individual blank records or one or more records from an existing coordinate file. When you add a record or records, they are appended to the end of the file. When you Insert one or more records they are inserted just above the current record.

Insert and Adding Coordinate Records from an Existing Coordinate File

If you choose to either the Add or Insert Pts from File or the corresponding toolbar item, you will see the **C&G Select Points from:** dialog.



This dialog consists of a list of the points chosen so far from the file listed in the **Change File to Select From** edit box. Use the **File** button to set the file name and the **Open** button to open the file for use. When the dialog first comes the point list is empty.

Choose Points section

Choose one of the available methods you will use for choosing points. Different methods will cause data entry controls to appear below the **Choose Points section**.

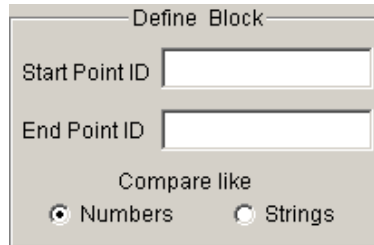
Note: Any time there are points in the list the REMOVE from SELECTION button will be enabled. Clicking this button will remove points from the list according to the current method being used to choose points.

All

If you choose all and not all of the points in the file are in the list, the **ADD from FILE** button will be enabled. Clicking the **ADD from FILE** button will add all the points from the file to the list.

Block

If you choose the **Block** method, the following **Define Block** section will appear below the **Choose Points section** of the dialog.

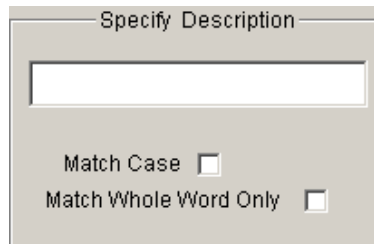


The 'Define Block' dialog box contains two text input fields: 'Start Point ID' and 'End Point ID'. Below these fields is a section labeled 'Compare like' with two radio buttons: 'Numbers' (which is selected) and 'Strings'.

Fill in the **Starting Point ID** and **End Point ID** then click the ADD from FILE button.

by Desc

If you choose the **by Desc** method, the following **Specify Description** section will appear below the **Choose Points section** of the dialog.



The 'Specify Description' dialog box features a large text input field at the top. Below it are two checkboxes: 'Match Case' and 'Match Whole Word Only', both of which are currently unchecked.

Fill in the description to look for and check the **Match Case** and **Match Whole Work Only** checkboxes as needed. Next click the **ADD from FILE** button

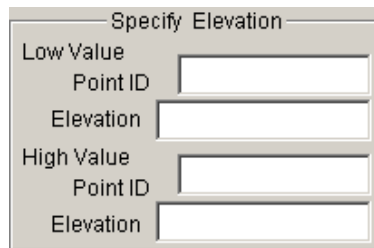
and any matching point records will be added to the list.

by Code

If you choose the **by Desc** method, the following **Specify Code** section will appear below the **Choose Points section** of the dialog. This section of the dialog looks and functions the same as the **Specify Description** section shown above except you must specify a code.

by Elev

If you choose the **by Desc** method, the following **Specify Elevation** section will appear below the **Choose Points section** of the dialog.



The image shows a dialog box titled "Specify Elevation". It contains two sections. The first section is labeled "Low Value" and has two input fields: "Point ID" and "Elevation". The second section is labeled "High Value" and also has two input fields: "Point ID" and "Elevation".

Specify the high and low elevation values. You can do this by directly entering the elevation values or you can type a point ID in the **Point ID** edit box then click on another edit box. When you do this the elevation of the point is written to the appropriate **Elevation** edit box. If you used a point ID to get the elevation, you can edit the value if necessary. Next click the **ADD from FILE** button to add the points to the list.

in Radius

in Rect

Coordinate File Data Entry

There is only one type of coordinate record called a **Point** record. This record has six fields:

Point ID: Point identifier must be unique. Its format varies according to the type of coordinate file.

Northing: Specifies the northing or Y coordinate of a point.

Easting: Specifies the easting or X coordinate of a point.

Elev: Specifies the elevation or Z coordinate of a point. (May be On or Off. To turn the column on click the column heading.)

Code (C&G coordinate files only):

a 4 character optional field used to group points. May be blank. (May be On or Off. To turn the column on click the column heading.)

Description:

Text describing the point. May be blank. Length limited by type of coordinate file. (May be On or Off. To turn the column on click the column heading.)

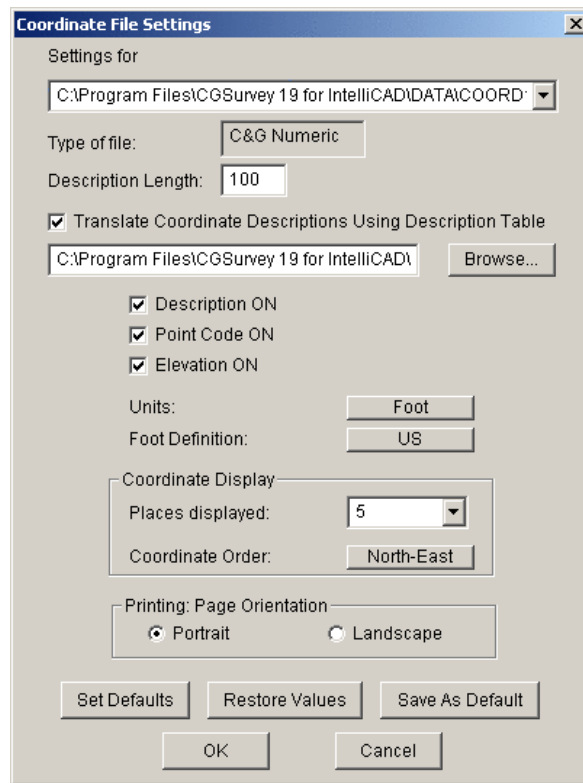
After adding or inserting a **Point** record, fill in the various fields as needed. Use the Tab or Enter keys to move from one field to the next. If you press Enter when in the last activated field in a record, a new blank record will be created just below the current record and the current field will be set to the **Point ID** field in the new record.

To replace the data in an existing record, just click once on the field you wish to replace and begin typing the new data. To edit the data in an existing record, click twice on the field you wish to edit and make any edits required in the existing data.

Settings Menu

Coordinate Files

Choosing this menu item brings up the Coordinate File Settings dialog



Settings for

drop down list box allows you to specify settings for any coordinate file currently open in the **CGEditor** as well as choose to set the **Default settings for new files you create.**

Type of File is only visible for the files currently open.

Description Length:

this edit box can be used to set a new description length for the file. If you choose to change the description length, any descriptions that already exist in the file will be truncated to the new length.

Translate Coordinate Descriptions Using Description Table checkbox

if this is checked then the description table file name edit box and the **Browse...** button will be enabled and you will be required to specify a description table to use.

Description ON checkbox - if this is checked then the **Description** column will be activated in the editor.

Point Code ON checkbox - if this is checked then the **Code** column will be activated in the editor. (Only applies to C&G coordinate files)

Elevation ON checkbox - if this is checked then the **Elevation** column will be activated in the editor.

Units: click the button to switch between **Foot** and **Meter.**

Foot Definition: click the button to switch between **US** and **International** feet. (Disabled if **Units** are set to **Meter**)

Coordinate Display section

Places displayed: drop down list - use to specify the number of decimal places displayed in the editor for northing, easting and elevation.

Note: the **Places displayed** setting does not affect the values actually stored in the coordinate file, only how they

are displayed in the editor window.

Coordinate Order: click button to switch between **North-East** and **East-North**.

Printing: Page Orientation section

choose **Portrait** or **Landscape**. You may wish to choose **Landscape** to avoid having the coordinate records with long descriptions causing each page to span 2 pages in width.

The **Set Defaults**, **Restore Values** and **Save As Default** have been covered elsewhere.

US Foot menu item

If this menu item is checked units are US feet. If the **Meters** menu item is checked, this menu item is disabled. The check will also be set or cleared by changes in the **Coordinate File Settings** dialog.

International foot menu item

If this menu item is checked units are International feet. If the **Meters** menu item is checked, this menu item is disabled. The check will also be set or cleared by changes in the **Coordinate File Settings** dialog.

Feet menu item: If this menu item is checked units are Feet. The check will also be set or cleared by changes in the **Coordinate File Settings** dialog.

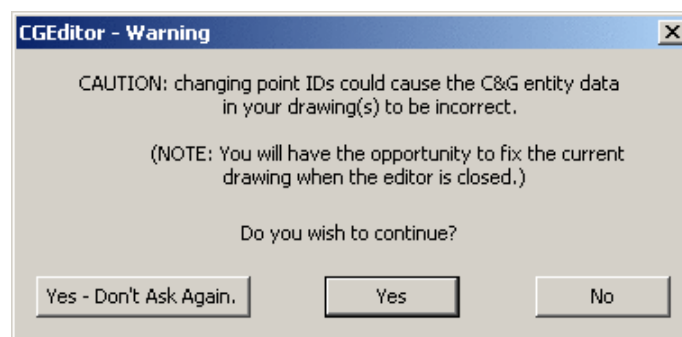
Meters menu item: If this menu item is checked units are meters. The check will also be set or cleared by changes in the **Coordinate File Settings** dialog. **Note:** The settings for US Foot and International foot will be ignored if Meters is checked.

Tools Menu

With the exception of **Renumber Points**, the items on the Tools menu are similar to those items already described for other file types.

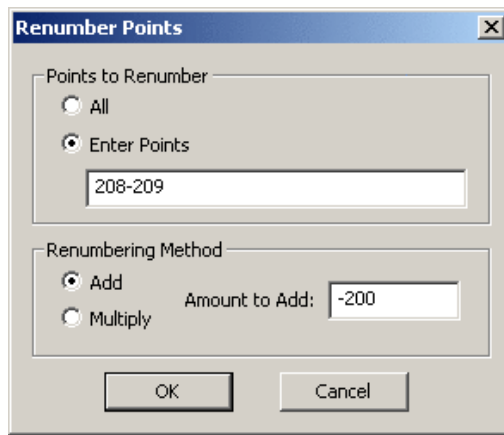
Renumber Points menu item

When you choose the **Renumber Points** menu item you will receive a warning regarding the problems that may be encountered in existing C&G drawings.



After considering the problems you may encounter due to point renumbering respond **Yes** or **Yes-Don't Ask Again** to continue with the renumbering operation, or **No** to cancel the operation.

If you choose to continue with the renumbering of points, you will see the **Renumber Points** dialog:



Renumber Points dialog

Points to Renumber section

First choose **All** or **Enter Points**.

If you choose **Enter Points** you must fill in the edit box specifying the points to renumber. The points to renumber can consist of single points or range(s) of points. Multiple entries of ranges and/or single points must be separated from the next entry by a comma (",") and ranges must be specified using a dash ("-")

Renumbering Method section

choose **Add** or **Multiply** then enter the **Amount to Add:** or **Multiply by:** in the edit box. You may specify a positive or negative whole number.

When done click **OK**. Click **Cancel** to end the command without changes to the coordinate file being edited.

Editing C&G Point Group Files

A C&G point group is essentially a list of points placed in a specially formatted text file (*.pts). It is possible to create and/or edit point group files using any plain text editor like Microsoft Notepad or Wordpad if you know the format of the file. Typically it is far easier to use the CGEditor to create and/or edit C&G point group files. Point groups have many uses in C&G commands: road alignments, property boundaries, define **Include** and **Exclude Boundaries** for **Topo** commands, etc. In the case of alignments, a point group can also include vertical curve information.

C&G point group files are organized into named subgroups. The subgroup name can be anything you wish to use to identify the points that follow. You may have several subgroups in a single point group file. For example, if you are defining subdivision lots, then you may choose the subgroup names to be lot 1, lot 2, etc. For an alignment you can make the subgroup name the starting station and C&G features will make this the default starting station when asking you for an alignment. Point groups can also be used to .

Creating and Opening Point Group Files

You may open and/or create as many files as are needed for your project.

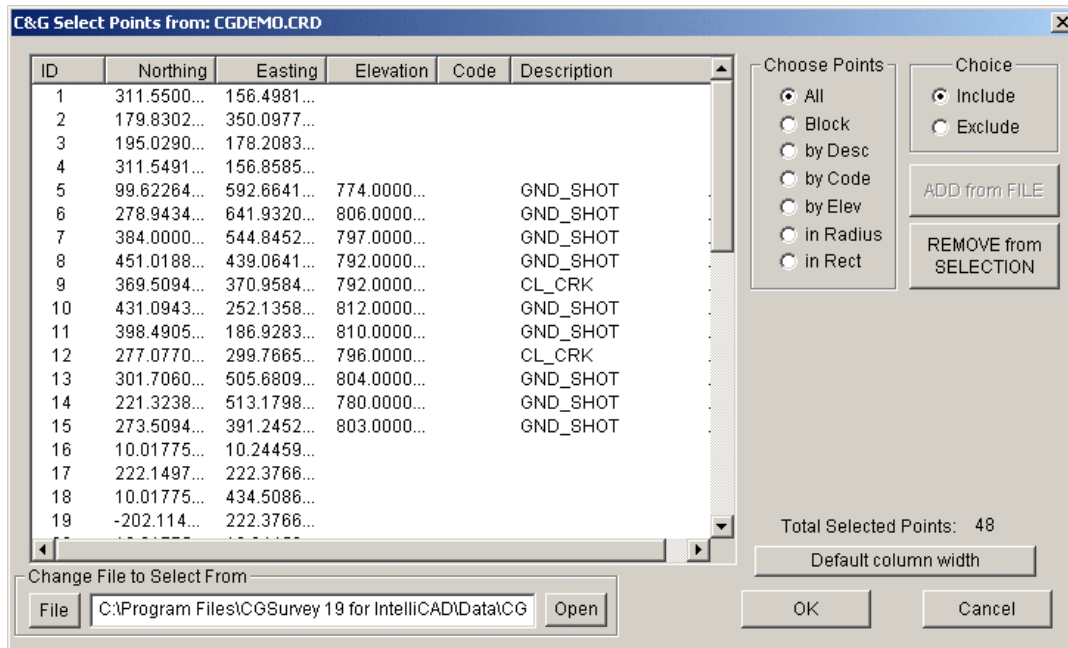
To create a new empty point group file choose the **File** menu then the **New** menu item then the **Point Group File** menu item or, more simply, just click the "P" toolbar icon on the **Standard** toolbar. In either case, a document window will appear within the main CGEditor window. This will have a single blank record or row for the **Subgroup** name - identified by **SGR** in the **Type** column.

To open an existing point group file, choose **File** then **Open...** Then, in the file dialog, browse to the directory where your point group file is located, highlight the desired file and click the **Open** button. The records will be read from the file and will be displayed in a separate document window

Entering and Editing Point Group File Data

If you do not have a subgroup name record, choose the **Add/Insert** main menu then, if you wish to place it at the end of the file, choose the **Add Subgroup** menu item or, if you wish to insert it above the current record, choose the **Insert Subgroup** menu item. After filling in the subgroup name, you can just press Enter to add a single Point record (identified by **PNT** in the **Type** column) below the subgroup name record. Alternatively, you can use the Add/Insert menu to add or insert a single point or you can choose to add or insert several points from a coordinate file. These same add and insert methods can be found on the **Add/Insert** Toolbar.

If you choose to **Add** or **Insert Pts from File** the following dialog comes up:



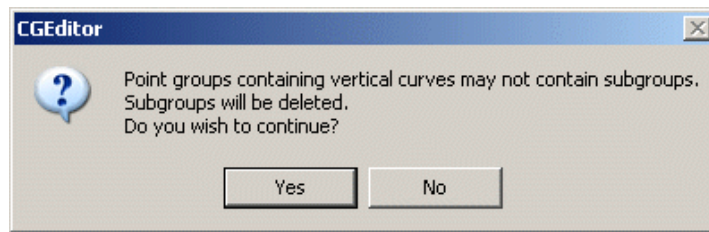
The use of this dialog to choose points has been described in detail under the section on **Editing Coordinate Files**. Please refer to that section for more details. After choosing the points you wish to add or insert into the point group using the **C&G Select Points from ...** dialog, click OK and the point records will be created in the point group file being edited.

Horizontal Curves

You may have noticed that the **Rad Pt Type** column is marked with **<None>** for the points you have inserted so far. However, if you wish to specify a curve in your alignment or lot boundary, you must designate the record as a radius point. If you click on the **Cw** toolbar icon (for clockwise) or the **Cc** toolbar icon (for counter clockwise) or choose similar items on the **Tools** menu, you will notice that the **Rad Pt Type** column for the point changes to **CW** or **CCW** to indicate that the point is a radius point. If a radius point is specified, the preceding point is assumed to be the PC and the following point is assumed to be the PT. If you wish to change the point back to not being a radius point, click the **Not Radius Pt** toolbar icon or use the **Tools** menu.

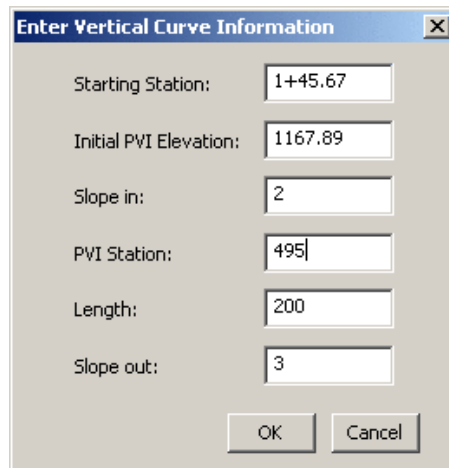
Vertical Curves

You may enter vertical curve information in a point group file. This allows you to not only specify the horizontal location of the alignment, but also its vertical alignment. A point group file that has vertical curves in it may not contain any subgroup records. If you attempt to place vertical curve data in a file having one or more subgroups, you will be given the following warning:



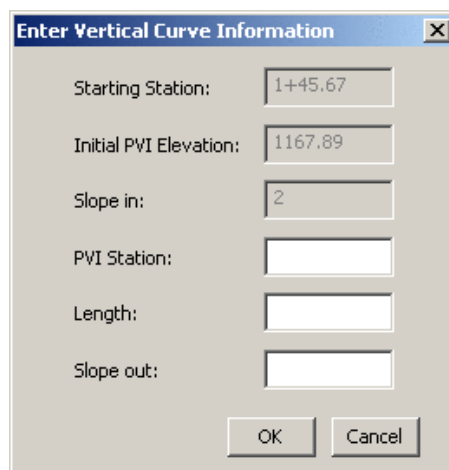
As indicated by the choices in the dialog, you may continue and place vertical curve info in a point group file containing subgroups, but it will not be usable in C&G commands.

If you have no subgroups (or if you do and answer "Yes" to the warning) and this is the first vertical curve in the file, the **Enter Vertical Curve Information** dialog will come up.



Enter the information in the dialog to specify the vertical curve. The **Starting Station** and the **PVI Station** should be entered as decimal numbers and when you click in another edit box the decimal station will be converted to standard station notation. The **Slope in** and the **Slope out** should be entered as a percent (For example, enter 2 or 2.0 for 2%). When you click **OK** the vertical curve records **VC1** and **VC** are added in the document window.

For the second and succeeding vertical curves, you can either **Add Vertical Curves** to the end of the vertical curve records or you can **Insert Vertical Curves** within the existing vertical curve records. For these vertical curve records, the dialog requires fewer entries:



This is a result of the fact that the initial vertical curve **Starting Station**, **Initial PVI Elevation** and **Slope in** control the overall vertical orientation of the succeeding vertical curves thus you need only enter the **PVI Station**, **Length** and **Slope out** for these vertical curves. When you click **OK**, another vertical curve record will be added to or

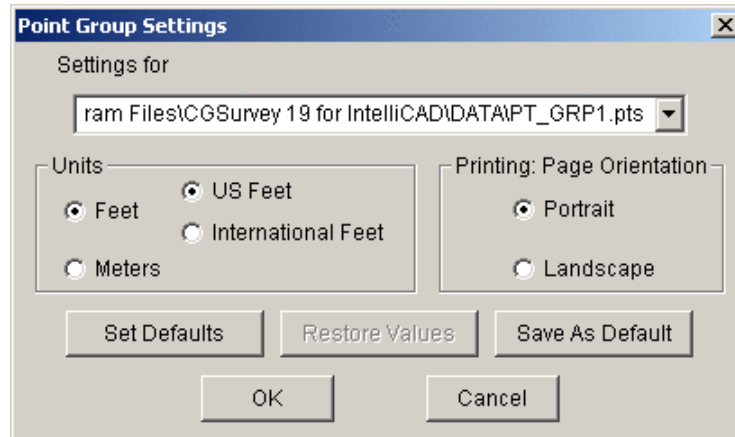
inserted into the document.

Once the vertical curve information has been specified you can go ahead and enter the points specifying the alignment.

Settings Menu

The Settings menu allows you to configure the point group file and the record appearance.

Choose the **Point Group Settings** menu item to bring up the **Point Group Settings** dialog:



In this dialog you can set the units and the page orientation for printing.

Note: the units setting only effects the display of stations in station notation.

Choose the **Record Color** menu item to bring up the **Record Color** dialog. Set the display color of the various records by clicking on the line for the record type. This brings up a color dialog that allows you to pick from the 16 available colors. Click **OK** when done.

Tools Menu

The items in the **Tools** menu are, for the most part, self-explanatory or have been covered in detail for other types of files.

Pulldown Menu Location: CG-Survey>Tools>CGEditor

Keyboard Command:eda, cg_edit_all

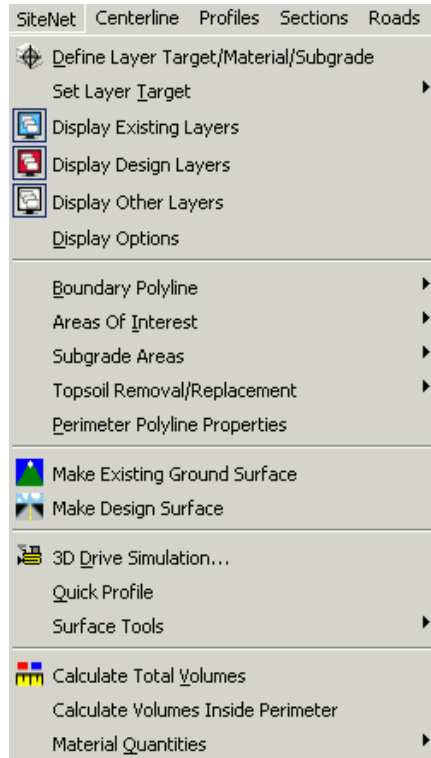
Prerequisite: May need existing C&G raw traverse data file (*.cgr), C&G Map Check file (*.cgm), C&G Cross Section file (*.cew), C&G Template file (*.ctp), coordinate files (*.crd, *.cgc, *.zak, *.mdb) and/or C&G Point Group files (*.pts)

Civil Module

5

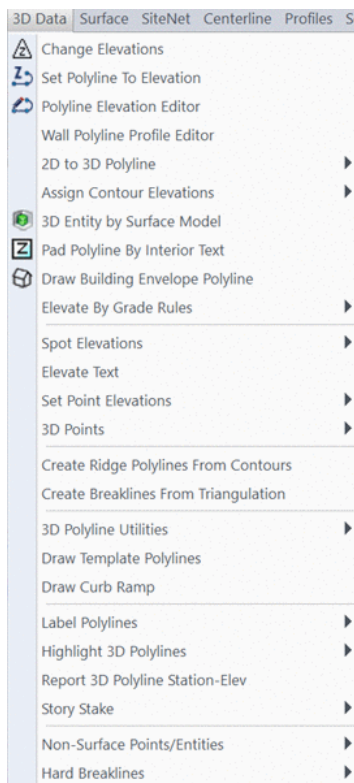
SiteNet Menu

The SiteNet programs build triangulation surfaces, apply surface adjustments, calculate volumes and report material quantities. Drawing layers are organized by target surfaces: design, existing or other. Polyline perimeters are used to define the site boundary, subgrade areas and topsoil areas. Once the layers and perimeters are setup, the surfaces are created by the Make Existing/Design Surface commands. Then there are commands to inspect the surfaces and report volumes and material quantities. See the Takeoff section of the Construction chapter of the manual for specifics on each command.



3D Data Menu

The 3D Data menu has commands for creating 3D entities to use for surface modeling. Several of these commands are shared in common with the Elevate menu of the Construction module.



Change Elevations

This command will change the elevation of selected entities. It can move the entity to a specified elevation from its current elevation (absolute) or do a differential change by adding or subtracting a value from its current elevation. If Carlson points are selected, their elevation attribute text and the elevation stored in the external coordinate file are changed. If the points are in the drawing at their real Z, this is also adjusted, however, if they are in the drawing at a fixed elevation, e.g. 0, the point blocks remain at that elevation.

There are options to move the changed objects to a new layer, and alternately to select a source object's elevation to supply the value of the elevation to change the selected object(s) to.

Prompts

Type of elevation change [<Absolute>/Differential**]:** *press A* to change to a specific (absolute) value, or *press D* to enter an amount of elevation change to apply to object's current elevation.

Change Layer for changed entities [Yes/No**]:** *press Enter* or *N* to keep on same layer, *press Y* to change layer of object after changing elevation.

Select/Enter Elevation **<100.0000>:** *125* By using the Absolute option all entities selected are changed to the elevation 125. You may simply *press Enter* to keep the value shown in brackets. *Press S* to select a source object's elevation for the new elevation to change the selected object(s) to.

Select objects: 1 found

Select objects: *press Enter* to conclude selection.

Tested 1 Entities

Carlson Software Points Changed > 1

Select/Input another Elevation (Enter to end): *press Enter*

Pulldown Menu Location: 3D Data

Keyboard Command: chgelev
Prerequisite: Something to change

Set Polyline To Elevation

This command allows you to assign elevations to one or more polylines. The command also affects lines, arcs, circles, inserts and points as well as polylines. The elevation can be assigned by entering the value or by picking an existing object or text entity that has the desired elevation. Note that this will change 3D objects to a single target elevation (making it a 2D object) as well as changing elevation values of 2D objects.

Note that this is the same command as **3D Entity to 2D** on the **Edit** menu.

Prompts

Enter/⟨Select text or linework of elevation⟩: *Press E to enter a value directly, or select text or linework of source value*

Linework Elevation: 440.000 (elevation of selected linework displays).

Select Lines, Arcs, Circles, Polylines, Inserts and Points for elevation change.

Select objects: 1 found

Select objects: 1 found, 2 total

Select objects: 1 found, 3 total

Select objects: *press Enter to conclude selection.*

LWPOLYLINE (indicates entity type)

LWPOLYLINE

3DPOLY to 2DPOLYLINE

Number of entities changed > 3

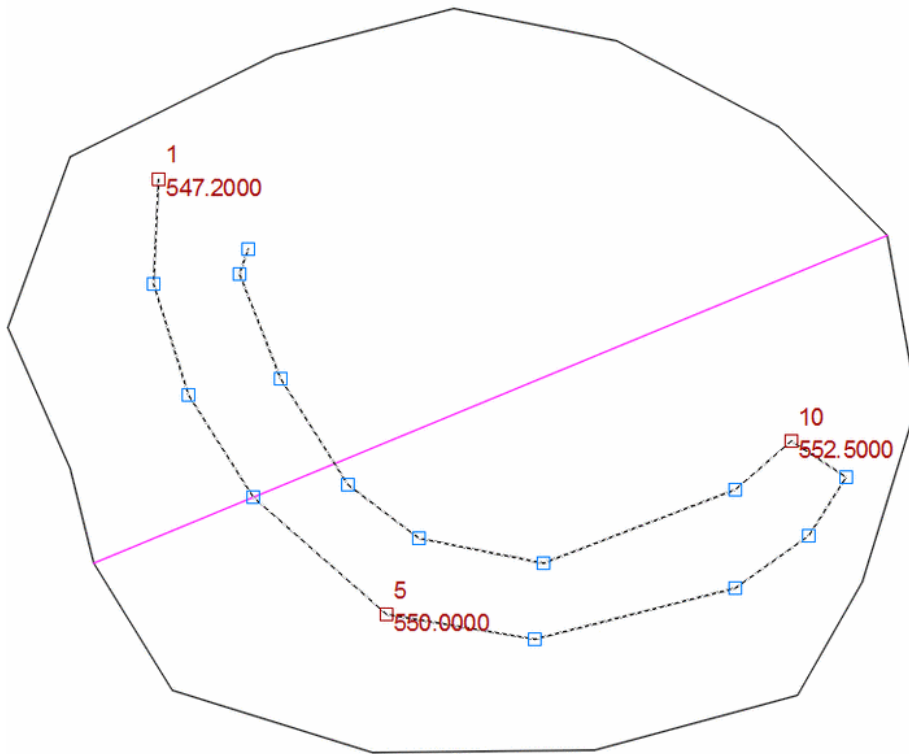
Pulldown Menu Location: 3D Data

Keyboard Command: 3dto2d

Prerequisite: A polyline and a target elevation to assign

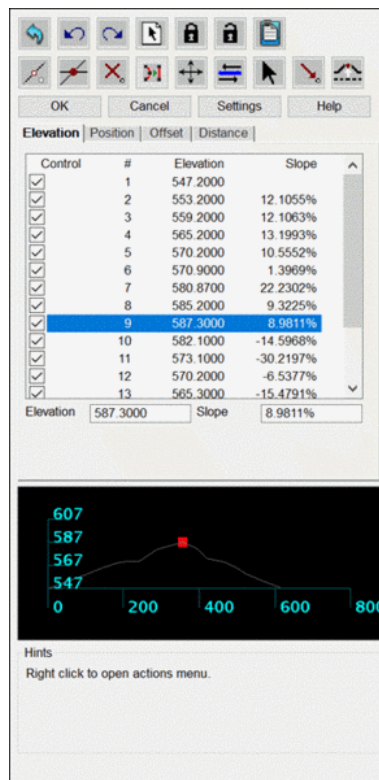
Polyline Elevation Editor

This command allows very precise control of 3D Polylines, specifically in the ability to edit vertex elevations, as well as add, delete, or move vertices. If these polylines are used in the creation of surfaces as breaklines, any editing you do may update the surface automatically. You can also control the location of Polyline vertices as defined by the station and offset of the vertices relative to a Centerline. Polyline vertices are designated as either control or free vertices. The elevation of control vertices are set and held, the elevations of free vertices are interpolated. In the drawing, control vertices are shown by default with red boxes, free vertices with blue boxes, but you may configure these using the Grip Color in the Settings.



Polyline Elevation Editor Dock Dialog:

When you run the Edit Assign Polyline Elevations routine, you are first prompted to select a Polyline to edit. After selecting a Polyline to edit, the following dock dialog appears on the left side of your screen. This dialog is "modeless" meaning that CAD commands and controls are still active while this dialog is active.



Tabs:

The four tabs in the panel provide access to control of Polyline vertex **Elevation, Position, Offset** and **Distance**.

- **Elevation:** This tab displays the vertices of the Polyline, each with a check box to set whether it is a control vertex or a free vertex (Control), its assigned number (#), its elevation (Elevation), and the slope from the previous vertex to that vertex (Slope). Selecting a vertex highlights its grip in the drawing. Once selected, you can enter an "elevation" or "slope" edits for that vertex. If multiple vertices are selected with varying elevation or slopes then "[multiple]" will be displayed in the "Elevation" and or "Slope" edits. When multiple vertices are selected, entering values in the "Elevation" or "Slope" edits will set all selected vertices based on the entered elevation or slope values.
- **Position:** The Position tab displays the Easting and Northing coordinates of each vertex. When one vertex is selected Northing and Easting can be entered in the "Northing" and "Easting" edits. When Multiple vertices are selected average Northing and average Easting are displayed in the "Average Northing" and "Average Easting" edits. When multiple vertices are selected entering values in the "Average Northing" and or the "Average Easting" edits will set all selected vertices Northing and or Easting values to based on the entered values.
- **Offset:** The Offset tab requires the selection of a Centerline to reference. Once a Centerline is designated, the Station and Offset of each vertex relative to the Centerline is displayed and can be edited.
- **Distance:** The Distance tab shows the horizontal distance for each Polyline segment. You can change a distance by highlighting the segment on the list and editing the value in the "Horz Dist" edit. If multiple vertices are selected "[multiple]" will be displayed in the "Horz Dist" edit. The user cannot set multiple segment's horizontal distances at once.

Toolbar Commands:

The toolbar provides access to several commands, some of which are tab specific.



Revert All: Reverts currently selected Polyline vertex positions and control information to vertex positions and control information when Polyline was initially selected. (Note: Selecting a new Polyline resets initial reversion location)



Undo: Undoes the most recent command line action. (Equivalent to ctrl+z)



Redo: Redoes the most recent command line action. (Equivalent to ctrl+y)



Select Polyline: Select Polyline to edit. (Note: if no Polyline is selected a Polyline can be selected using CAD selection)




Control Selected: Set selected vertices controlled, elevation will not be interpolated for control points. (This toolbar command is only available when elevation tab is active)





Free Selected: Set selected vertices free, elevation will be interpolated for free points. (This toolbar command is only available when elevation tab is active)





Output Report: Brings up Report Formatter dialog. Output reports can contain Easting, Northing, and or Elevation information for each vertex as well as whether or not each vertex is controlled or free.


 **Create Vertex:** Creates a vertex at the station nearest the mouse click position. Control/free state depends on the "Create Vertex Mode" setting (see Settings dialog).


 **Create Vertex At Crossing:** Prompts user to select crossing Polyline(s). If Polyline(s) selection is(are) applicable, crossing vertex point(s) are added at the elevation of the selected Polyline(s) at crossing. All created vertices are added as control points.


 **Delete Vertices:** Deletes currently selected vertices.


 **Reduce Vertices:** Prompts user to select an offset cutoff. Removes those vertices from the active Polyline that will not change the position of the Polyline at that vertex by more than offset cutoff. Will not removed controlled vertices.


 **Move Selected:** Move currently selected vertices based on drag offset. (Equivalent of click and drag)


 **Reverse Polyline:** Reverses indices of vertices. This will be reflected by reversing the vertices in the currently active list view.

 **Pick Elevation/Position From Screen:** Elevation- prompts user to pick an entity from screen. Will set elevation based on entity's elevation information. Vertex set controlled. Position-prompts user to pick position from screen sets Northing and Easting based on selected position. (This toolbar command is only available when elevation tab or position tab is active).

 **Pick Elevation From Surface:** Prompts user to select a surface model (tin;ft). Then prompts user to pick a direction point to run the slope from highlighted vertex. Elevation of highlighted point is determined from surface and slope is then applied along the line to the next control point. (This toolbar command is only available when elevation tab is active and one or more vertices is selected).

 **Pick Elevation With Slope:** Prompts user to pick a point or elevation text. Then prompts user to pick slope. An elevation is then calculated based on the run to the point selected at the slope the user input. Elevated vertices will be set controlled. (This toolbarcommand is only available when elevation tab is active).

 **Offset Selected Elevation:** Prompts user to input an offset amount. Adds(subtracts) offset amount from currently selected vertices. Offset vertices will be set controlled. (This toolbar command is only available when elevation tab is active)

 **Select Centerline:** Prompts user to select a Centerline or Polyline to apply Centerline information from screen. (This toolbar command is only available when Offset tab is active)

OK:

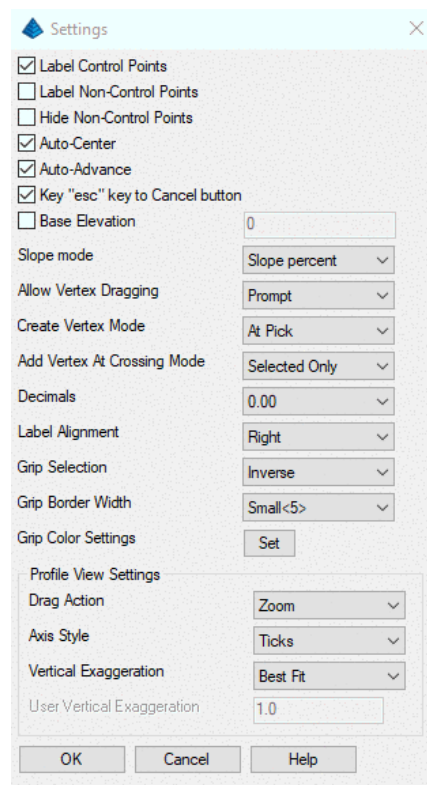
The "OK" button closes the Polyline Elevation Editor dialog, and saves the vertex position and control information of the currently selected Polyline.

Cancel:

The "Cancel" button closes the Polyline Elevation Editor dialog, and reverts the currently selected Polyline's vertex position and control information to the vertex position and control information when the Polyline was first selected.

Settings:

The "Settings" button provides control over various options pertaining to the use of the Polyline Elevation Editor routine.



- **Label Control Points:** Toggle on to display control vertices' numbers and elevations.
- **Label Non-Control Points:** Toggle on to display free vertices' numbers and elevations.
- **Hide Non-Control Points:** Toggle on to remove all free vertices' grips.
- **Auto-Center:** Toggle on to automatically center view such that all currently selected grips when they are selected.
- **Auto-Advance:** Toggle on to automatically advance the currently selected grip to the next available vertex (by vertex number).
- **Key "esc" key to Cancel button:** Toggle on to key "esc" to the Polyline Elevation Editors' dialog "Cancel" button. That is pressing the "esc" key will cancel and exit the Polyline Elevation Editor routine.
- **Base Elevation:** When toggled on the leading significant digits of the Base Elevation value will be added as a prefix to any elevation value entered. Base Elevation values must be whole numbers greater than or equal to zero. Typical use: (Base: 6000)+(User: 100)=(Elevation: 6100 displayed as 100).
 - *Conflicts:* Some user entered elevation values are ambiguous in terms of how to combine the Base Elevation, and the user entered elevation. These conflicts are dealt with as follows:
 - * *Significant Digit Conflicts:* In the case where a user entered value, and the Base Elevation value both contain significant digits in the same column(s), then the user entered value supersedes the Base Elevation value in that(those) column. Ex: (Base: 6500)+(User: 100)=(Elevation: 6100 displayed as 100).
 - * *Negative Value Conflicts:* Negative values are processed as values below the Base Elevation. However, adding a negative value can result in elevations that result in "Significant Digit Conflicts" with the Base Elevation. In this case the logic of the "Significant Digit Conflicts" applies. Ex: (Base: 6000)+(User: -100)=(Elevation: 5900 displayed as 5900).
- **Slope Mode:** Determines how slope at a vertex is displayed and interpreted.

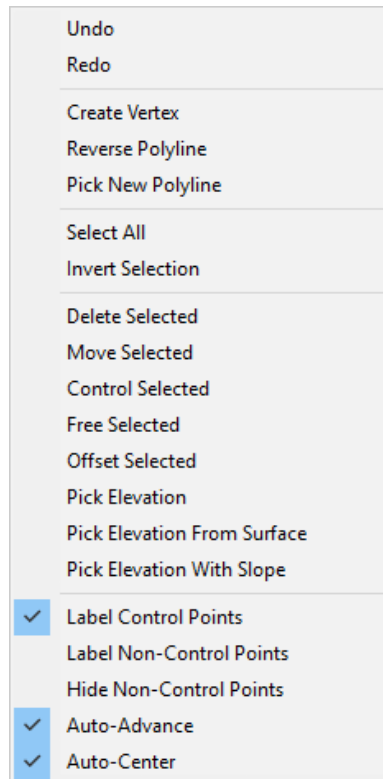
- *Slope Ratio*: Slope as a ratio of line length to line elevation [*run/rise*]
- *Slope Percent*: Slope as a percent [*rise/run*100*]
- *Slope Degree*: Slope as degree offset from 0° from -90° to 90° [*atan(rise/run)*180/PI*]
- **Allow Vertex Dragging**: Determines how dragging selected vertices functions.
 - *Never*: Never allow dragging.
 - *Prompt*: On completion of vertex dragging user is prompted on whether or not to accept the results of the dragging operation.
 - *Always*: Always allow dragging.
- **Create Vertex Mode**: Determines how elevation is calculated for vertices inserted using the Create Vertex toolbar command, double click on line, or right-click Create Vertex command.
 - *On Line*: Creates a vertex at the station nearest the pick point. Elevation is interpolated based on the equation [$elv=(InsertDistance2D/TotalDisatance2D)*(TotalElevationChange)$]. If no control points exist, then the elevation is set to the elevation of all other free vertices on the Polyline. Inserted vertex is a free point.
 - *At Pick*: Creates a vertex at pick point. Vertex is inserted in the Polyline after the vertex preceding the nearest station to the pick point. If the pick point is the result of a snap, the elevation of the new vertex is taken from the elevation of the snapped entity, and the created vertex is added as a control point. Otherwise, the created vertex is added as a free point.
- **Add Vertex Crossing Mode**: When running the function to add vertices at crossings with other linework, this option chooses whether to add intersection vertices on the crossing linework in addition to the current polyline.
- **Decimals**: Determines how many decimal places are displayed in Polyline Elevation Editor dialog. User can choose between 0 and 6 decimal places for display.
- **Label Alignment**: Determines where vertex index, and vertex elevation for all labeled vertices is drawn.
 - Right: Draw label to the "right" of vertex grip. Takes into account screen rotation.
 - Top: Draw label "above" vertex grip. Takes into account screen rotation.
- **Grip Selection**:
 - *Standard*: left-click to drag vertex, shift+left-click to select single grip vertex.
 - *Inverse*: shift+left-click to drag vertex, left-click to select single grip vertex. (legacy control)
- **Grip Border Width**: Allows user to set the relative size of the gripbox for all vertices based on the internal ACAD/ICAD GRIPSIZE variable. (Note: the values associated with each Grip Border Width choice represent the value that the internal ACAD/ICAD variable GRIPSIZE is set to for each Grip Border Width selection. If the current GRIPSIZE ACAD/ICAD variable is not one associated with any of the Grip Border Width choices ["Small," "Medium," "Large," "XLarge"] the Grip Border Width will be set to current ACAD/ICAD GRIPSIZE.)
- **Grip Color Settings**: Set the grip colors for easier editing of the vertices:
 - Hover Vertex
 - Control Vertex
 - Free Vertex
- **Profile View Settings**:
 - Drag Action: Choose between doing Zoom or Pan for click-n-drag in the profile view.
 - Axis Style: Set the profile grid display.
 - Vertical Exaggeration: Control the display for the profile vertical scale.

Help:

Display Carlson documentation for Polyline Elevation Editor.

Right-click Dialog:

There is a right-click menu available when the Polyline Elevation Editor routine is running which gives access to a variety of functions and settings. "Select All" and "Invert Selection" are the only commands unique to the right-click dialog. "Select All" selects all currently visible vertices. "Invert Selection" inverts all currently visible vertices. "Undo," "Redo," "Create Vertex," "Reverse Polyline," "Pick New Polyline," "Delete Selected," "Move Selected," "Control Selected," "Free Selected," "Offset Selected," "Pick Elevation," "Pick Elevation From Surface," "Pick Elevation With Slope," and "Pick Position" are equivalent to the Toolbar Command with the same name. "Label Control Points," "Label Non-Control Points," "Hide Non-Control Points," "Auto-Advance," and "Auto-Center" are equivalent to their settings dialog toggles with the same name.



Pulldown Menu Location: 3D Data

Keyboard Command: EDIT_PLINE.Z

Prerequisite: 3D Polylines with vertices

Wall Polyline Profile Editor

This command creates a retaining wall object using a polyline to define the alignment and attaching top and bottom profiles. This polyline is recognized by surface modeling commands like Triangulate and Contour as a retaining wall and the top and bottom profiles get built into the model. The wall polyline is also used in SiteNet and Takeoff for building the design surface including the subgrade surface for the base of the wall.

The polyline should be drawn so that the left side is the low side and the right side is high. Use the Reverse Polyline command in the Edit>Polyline Utilities menu if you need to switch the polyline direction.

The command prompts to select the polyline for the wall. If the polyline isn't already set as a wall polyline, the program prompts for a reference surface file which is used to define the initial top and bottom wall profiles. When the polyline is already set as a wall, the program loads the wall definition and goes directly to the profile editor.

The profile editor has a toggle to switch between editing the top and bottom profiles. Please see the Road Profile Editor command for a description on how to use the editor.

Surface View: Sets the color or texture of the wall for the 3D Viewer commands.

Layback Slope: Sets the slope from the top front to the bottom front of the wall.

Top Width: Sets the width of the wall at the top.

Bottom Width: Sets the width of the wall at the bottom.

Freeboard: Sets the height of the wall above the backfill surface on the back side of the wall.

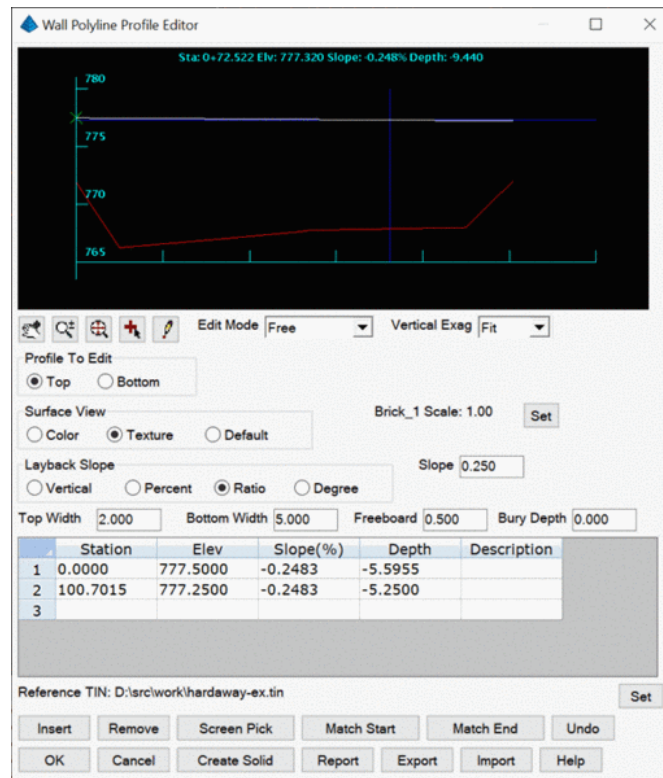
Bury Depth: Sets the depth of the wall below the surface at the front of the wall.

Match Start: Sets the top and bottom profiles to the same starting elevation.

Match End: Sets the top and bottom profiles to the same ending elevation.

Create Solid: Makes a solid model (.mdl) for the wall.

Report: Creates a report of the wall dimensions, profiles, volume, face area and backfill.



Prompts

Pick retaining wall polyline: *select polyline for wall*

Profile Editor dialog

Pulldown Menu Location: 3D Data

Keyboard Command: rwallpro

Prerequisite: polyline and surface file

2D to 3D Polyline by Surface Model

This command converts a 2D polyline into a 3D polyline by calculating 3D polyline vertices at all the intersects of the 2D polyline with the surface definition contained in the selected grid (.grd) or triangulation (.flt, .tin) surface file.

The command then interpolates elevations from these intersections at the original vertex locations. An application for this command is to "drape" linear features along a surface such as a 2D centerline. For example, a 2D centerline may be created in the drawing and then elevated to the target surface, thus providing a quick "profile" of the centerline which can then be offset for left and right edges, constructing a 3 dimensional model of the corridor alignment.

When you first run the command, a dialog pops up for selecting the desired surface file to use. Once selected, the following prompts appear:

Prompts

Loading edges...

Loaded 8788 points and 26097 edges

Created 17310 triangles

Select polylines to convert. *select one or more 2D polylines*

Select objects: 1 found

Select objects: *press Enter* to conclude selection.

Use current polyline elevations as vertical offset from surface [Yes/<No>]? *press Enter* to accept default.

Keep existing polylines [Yes/<No>]?

Set layer name for converted polylines [Yes/<No>]? *Press Y* to assign a new layer name or *press Enter*

Converting polylines ...

Elevated 1 polylines.

Pulldown Menu Location: 3D Data > 2D to 3D Polyline

Keyboard Command: 2dto3dpf

Prerequisite: polylines and a grid file or triangulation file.

2D to 3D Polyline by Screen Entities

This command converts a 2D polyline into a 3D polyline by calculating 3D polyline vertices at all the intersects of the 2D polyline with surface entities (contour polylines, triangulation lines) and by interpolating elevations from these intersections at the original vertices locations. An application for this command is to create breaklines. For example, a ridge or valley breakline could be generated from contour lines by drawing a 2D polyline along the ridge top or valley bottom, crossing the contours. Then the command would "grab" the contour line elevations along the polyline to make a ridge or valley breakline.

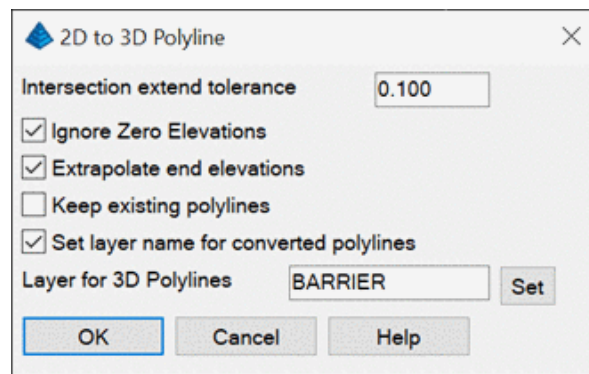
Intersection extend tolerance: Sets how far past the ends of the polyline to check for intersections.

Ignore Zero Elevations: Filters out any intersections at zero.

Extrapolate end elevations: Sets the start and end polyline elevations by extrapolating the last slope.

Keep existing polylines: Creates separate 3D polylines and leaves the original 2D polylines unchanged. Otherwise the original polylines are changed to 3D polylines.

Set layer name for converted polylines: Changes the layer name of the elevated polylines.



Prompts

2D to 3D Polyline dialog

Select polylines to convert. *select polylines*

Select objects: 1 found

Select objects: *press Enter* to conclude selection.

Select surface 3DFaces, lines and polylines. *select source 3D objects*

Select objects: Specify opposite corner: 7 found

Select objects: *press Enter*

Reading points ... 5353

Converted 1 polylines from 24 intersections.

Pulldown Menu Location: 3D Data > 2D to 3D Polyline

Keyboard Command: 2dto3dps

Prerequisite: Polylines and 3D surface entities

2D to 3D Polyline by Points

This command converts a 2D polyline into a 3D polyline by using the elevations of points. At each vertex of the polylines, the program looks for a point with elevation at the same x,y location. The points can be Carlson point blocks or POINT entities. This routine can be useful if the linework is created in 2D at zero elevation with commands like Draw 2D Polyline, and points with elevation are located along the linework. Then the linework can be converted into 3D polylines with this command. For example, a centerline polyline with arcs may need to be created in 2D for stationing because AutoCAD does not allow arcs on 3D polylines. To use this polyline as a breakline in surface modeling, this command can convert the polyline into a 3D polyline.

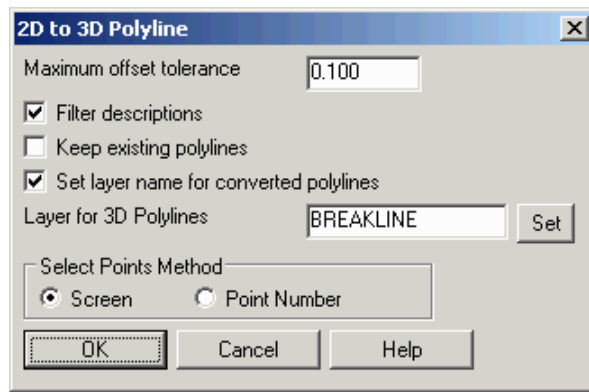
Maximum offset tolerance: points must be within this distance of the polyline vertices to get applied.

Filter descriptions: filters point by their description with ability for different vertical offsets by description.

Keep existing polylines: controls whether to erase for original 2D polylines.

Set layer name for converted polylines: assigns the layer name to the new 3D polylines.

Select Points Method: choose between screen picking the points or entering point number to process.



Prompts

2D to 3D Polyline dialog

Select polylines to convert.

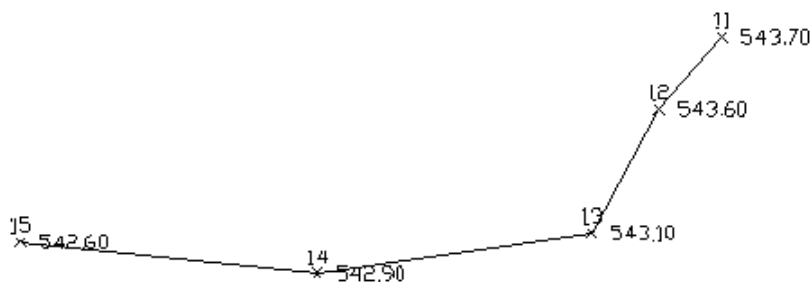
Select objects: *select polyline(s)*

Select points.

Select objects: *pick point entities*

Changed 1 polylines.

The routine will convert the 2D polyline into a 3D polyline and place it on the indicated layer.



Pulldown Menu Location: 3D Data > 2D to 3D Polyline

Keyboard Command: 2dto3dpt

Prerequisite: A polyline and points

2D to 3D Polyline by Slope From Reference

This command converts a 2D polyline into a 3D polyline using a slope from a reference 3D polyline or point. For example, this command can be used to elevate a bottom-of-bank polyline given an elevated top-of-bank polyline and bank slope. After selecting the polyline to elevate and the 3D reference, there is a prompt for the slope percent. Use negative for downhill. For each vertex on the 2D polyline, the program finds the nearest position on the 3D reference and assigns the vertex elevation using this reference position elevation plus the slope multiplied by the distance from the vertex to the reference.

Prompts Select polyline to assign elevations: *pick polyline to elevate*

Select reference 3D polyline: *pick reference polyline*

Enter slope percent: -25

Select polyline to assign elevations (Enter to end): *press Enter*

Pulldown Menu Location: 3D Data > 2D to 3D Polyline

Keyboard Command: elev_slope

Prerequisite: A 3D polyline or 3D points to reference

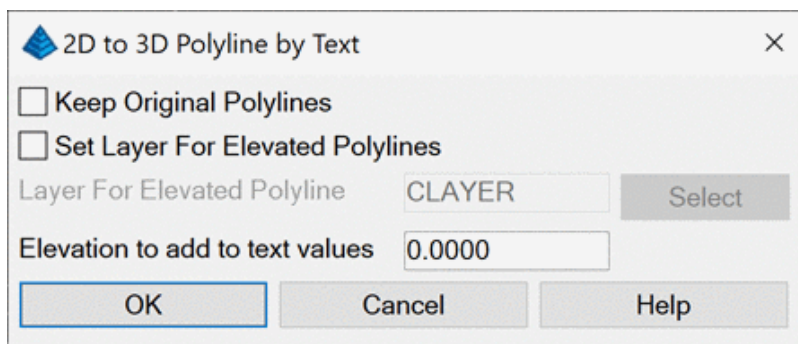
2D to 3D Polyline by Text

This command allows you to change 2D polylines to 3D polylines by elevation labels.

This command will prompt you for samples of the elevation labels and the polylines to convert. The program uses these samples to know the layer names for the labels and linework to process. Then select all the polylines with their labels you want to convert.

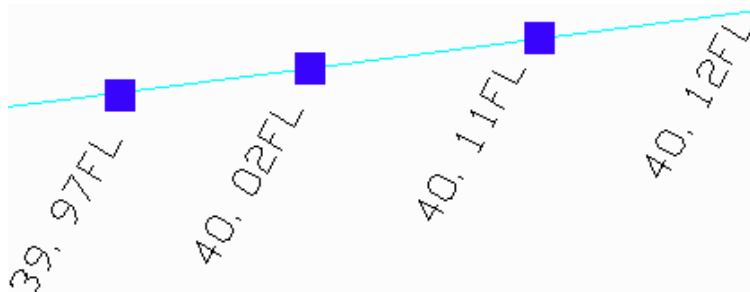
There is a dialog to enter in an elevation to add to label values. Often elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 539.97, 540.02, 540.11 sometimes, like in the example below, they are listed as 39.97, 40.02, 40.11. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing.

The dialog also has options to Keep Original Polylines which leaves the original 2D polylines. Otherwise the 2D polylines are replaced by the elevated polylines. The Set Layer For Elevated Polylines allows you to change the polyline layer for the elevated polylines.



This command will assign elevations from the labels to nearby vertices. If vertices do not have a nearby elevation label, then they will be interpolated from vertices that do have nearby elevation labels.

To decide how to apply the elevation text to the polyline, the routine computes the Center Point of the elevation text. Then the Nearest Vertex on the polyline to the elevation text Center Point is computed. Next the Center Point is projected onto the polyline to calculate the Projected Point. If the Center Pointer to Nearest Vertex distance is less than 2/3rd the Projected Point to Nearest Vertex distance then the Nearest Vertex is elevated base on the elevation text. Otherwise, a new vertex is inserted into the polyline at the Projected Point, and this new vertex is elevated based on the elevation text.



Prompts

Select sample of elevation text: *pick a text label*

Select sample of a polyline to convert: *pick a polyline*

Select polylines to convert and elevation labels.

Select objects: *select all the entities to process*

19 found, 19 total

Enter elevation to add to label values <0.00>: 500

Pre-processing entity #19 of 19

Processing elevation text #18

Remaking polyline #1

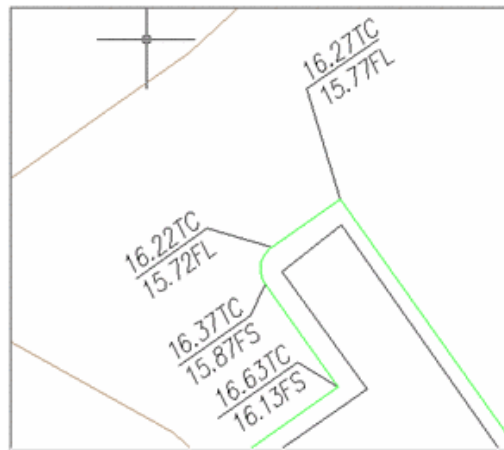
Pulldown Menu Location: 3D Data > 2D to 3D Polyline

Keyboard Command: elevfb

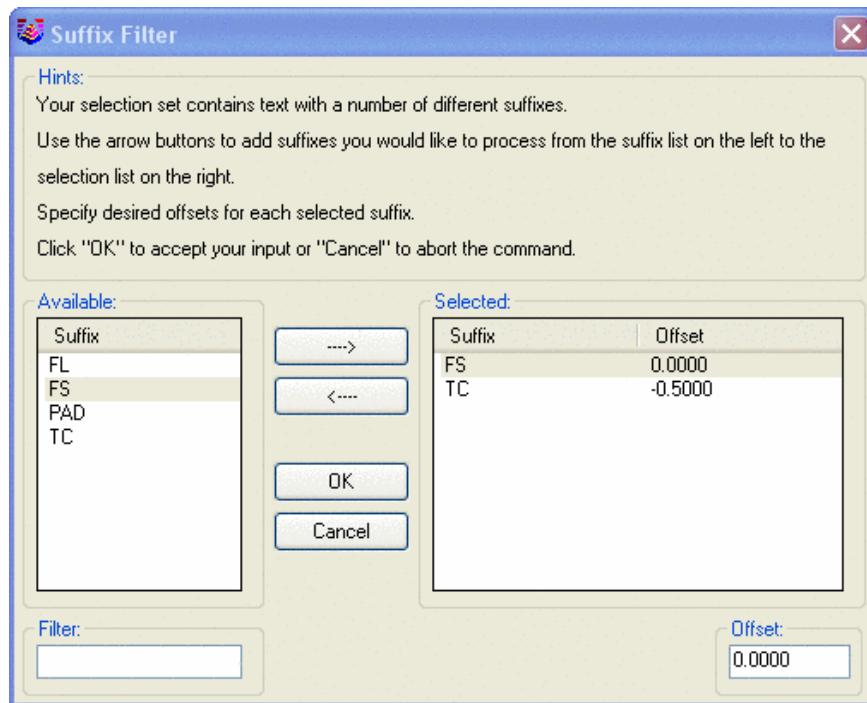
Prerequisite: 2D polyline and elevation labels

2D to 3D Polyline by Text With Leader

This command will assign elevations from the labels to the polylines by following the label leaders to their corresponding vertices on the polyline.



The command will prompt you for samples of the elevation labels, the leaders, and the polylines to convert. The program uses these samples to know the layer names for the labels and linework to process. Then select all the labels and leaders for the polylines you want to convert. Since you pre-specified the layers involved, you may safely window everything or type the keyword *ALL* and only those layers specified will be examined. You will then be prompted to enter an elevation to add to label values. Often, elevations are abbreviated to save time and space. For example, if every elevation in the drawing is in the 800s instead of labeling every elevation 817.85, 817.40, 817.30 sometimes, like in the above example, they are listed as 17.85, 17.40, 17.30 This option allows you to add a given amount, such as 800, to every label elevation to produce the correct elevation in the drawing.



The program searches for all leaders and gathers their associated text. If the program finds different labels in the elevation text, then this dialog box allows you to select the text you want to use to create the 3D polylines. In this example you might want to use elevations followed by TC. This dialog box allows you to select that text and exclude the other text which is not to be used in the elevations of the polyline, such as FS.

If you are creating 3D polylines from multiple elevation labels, then this dialog box will allow to offset certain labels by a given amount. In the above example you can offset an elevation labeled TC by -0.50 so that it matches vertices set by FS labeled elevations. **Note:** you must press *Enter* after setting the offset amount to assign it to the selected prefix in the list above.

Prompts

Options/Select sample of elevation text: *pick a text label* (Press *O* to set tolerance options).

Select sample of an annotation leader: *pick an annotation leader*

Select sample of a polyline to convert: *pick a polyline*

Select polylines to convert, leaders and elevation labels to process.

Select objects: *select the desired entities* This will filter by only those layers identified in above steps.

Joining adjacent polylines...

Reading the selection set ...

Enter elevation to add to label values <0.00>: *800*

Pre-processing entity #19 of 19

Filtering text entities

Conflict detected: pick polyline corresponding to current leader

Press N for next selection, A for all objects, or Enter to accept current object only: *press Enter*

Processing leader #6

Remaking polyline #1

Pulldown Menu Location: 3D Data >> 2D to 3D Polyline

Keyboard Command: elevfl

Prerequisite: 2D polyline, elevation labels, and leaders

2D to 3D Polyline by Start/End Elevations

This command allows you to convert a 2D polyline to a 3D polyline by specifying the starting and ending elevations of the polyline. All intermediate polyline vertices elevations are linearly interpolated from these end point elevations.

Prompts

Select polyline to assign elevations:

Enter starting elevation: *109.85*

Percent/Ratio/<Enter ending elevation>: *112.16*

Select polyline to assign elevations (Enter to End): *press Enter*

Pulldown Menu Location: 3D Data > 2d to 3D Polyline

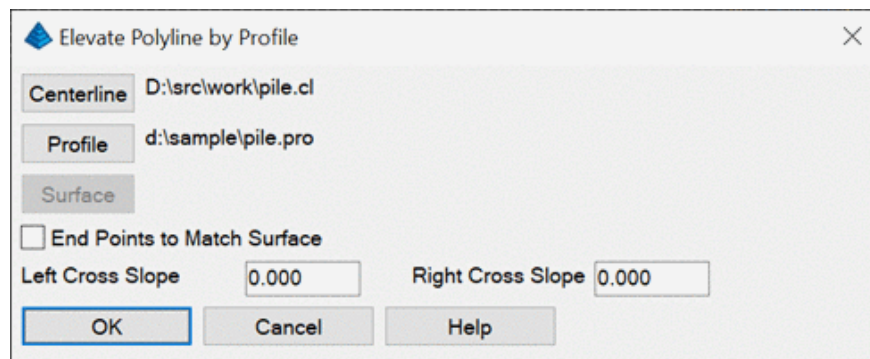
Keyboard Command: 2dto3dpl

Prerequisite: A polyline

2D to 3D Polyline by Profile

This command converts a 2D polyline into a 3D polyline by using a reference polyline, centerline and cross slope. For each vertex on the polyline, the program locates the station and offset from the centerline and then elevation on the profile for that station. Then the vertex elevation is set from that profile elevation plus the cross slope multiplied by the offset. For example, this command can be used to elevate a polyline for edge of pavement at a -2% cross slope from a centerline and profile for the road centerline. To elevate a polyline that is along the profile alignment without an offset, the Profile To 3D Polyline command is more direct method.

There is an option to use a surface to set the elevations at the start and end vertices of the polyline.



Prompts

2D to 3D Polyline dialog

Pick polyline to elevate: *pick polyline*

Pick polyline to elevate (Enter to end): *press Enter*

Pulldown Menu Location: 3D Data > 2D to 3D Polyline

Keyboard Command: elev_profile

Prerequisite: A polyline and profile

LWPolyline to 3D Polyline

This command converts the entity type of LWPolylines (light-weight polylines) to Polylines. This command does not change the geometry of the polylines or any other property.

Prompts

Select polylines to convert to 3D.

Select entities: *pick lwpolylines*

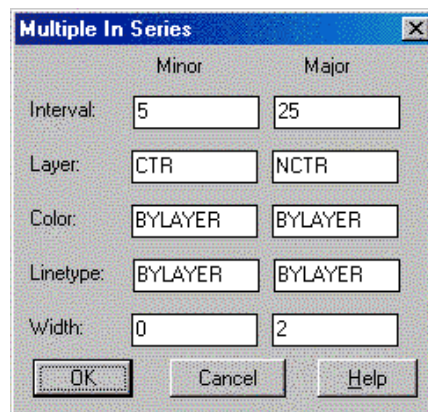
Pulldown Menu Location: 3D Data > 2D to 3D Polyline

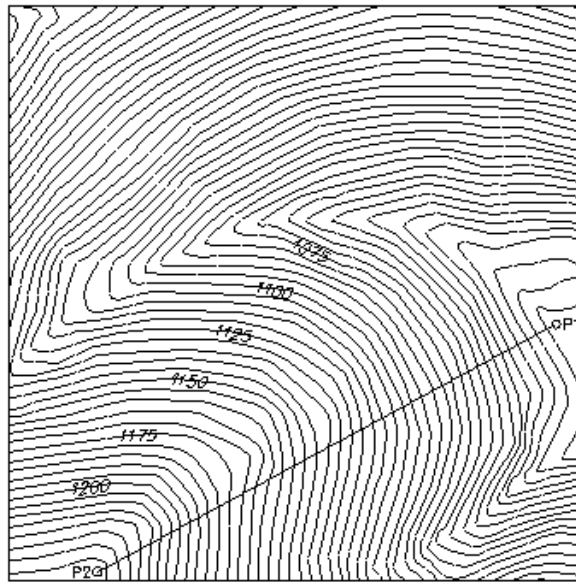
Keyboard Command: lwto3dp

Prerequisite: lwpolyline

Assign Contour Elevations: Multiple in Series

This command can be used to quickly and accurately assign the elevation of series of polylines that have been converted from raster or digitized without correct elevations. The routine will automatically assign elevations to the polylines crossing the fence line selected by two points. At the same time the elevations are changed, the program can assign it a new layer, color, linetype, and polyline width. This process usually works best if contours are in a temporary (white) layer to start. When they are processed, they will take on the color of the new layers making it easy to distinguish which polylines have been processed.





Prompts

Settings/First Point: *press S to change settings or pick first point*

Second Point: *pick second point*

Beginning Elevation <0.00>: 1020

Increment Direction U/D <U>: *press Enter*

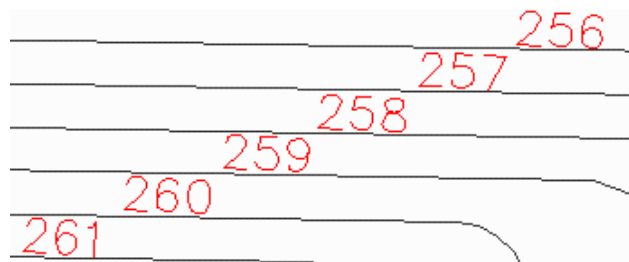
Pulldown Menu Location: 3D Data > Assign Contour Elevations

Keyboard Command: grpcelev

Prerequisite: Digitized polylines

Assign Contour Elevations: From Contour Labels

This command allows you to set elevations to contours from elevation labels.



Select a sample of the elevation text to be used on the contouring. Next, select a sample of the contouring that you want to add the elevations to. Now select all the contours and their corresponding elevation labels and press Enter. Carlson Civil will then add elevations to all the contours. You may be prompted to distinguish what contour goes with what elevation label. You can either press Enter to accept the contour that Carlson Civil has selected or you can Press N to choose another contour.

Prompts

Select sample of elevation text: *pick an example text elevation*

Select sample of a contour line: *pick an example contour polyline*

Select contour lines and elevation text to process.

Select objects: *select the entities to process*

Joining adjacent polylines...

Reading the selection set ...

Joining ...

Pre-processing entity #1008 of 1008

Filtering text entities

Processing elevation text #518

Conflict detected: *pick contour corresponding to current elevation text*

Press N for next selection or Enter to accept current:

Remaking polyline #311

Pulldown Menu Location: 3D Data > Assign Contour Elevations

Keyboard Command: txtcelev

Prerequisite: Contours and contours labels

Assign Contour Elevations: Single Elevation Group

This command changes the elevations of polylines and can be used to set the elevations of contour polylines. The routine begins at a specified elevation and prompts for a selection set of polylines to set to the elevation. Then the routine repeats using the last elevation plus the elevation increment. Enter an empty selection set to exit the routine.

Prompts

Starting elevation <0.0>: 500.0

Contour interval (negative for down) <1.0>: 5.0

Select polylines to set to elevation 500.0.

Select objects: *pick the polylines*

Select polylines to set to elevation 505.0.

Select objects: *pick the polylines*

Select polylines to set to elevation 510.0.

Select objects: *press Enter*

Pulldown Menu Location: 3D Data > Assign Contour Elevations

Keyboard Command: setcelev

Prerequisite: polylines

3D Entity By Surface Model

This command assigns the elevations of the selected points, inserts, lines and polylines to the elevations defined by the specified grid (.GRD) or triangulation (.FLT, .TIN) file. For lines and polylines, additional vertices are added to model the surface. 3D Polyines cannot have arcs, so any arcs in the original 2D polylines are converted to a series of chords. This command is useful to "drape" objects onto a surface and to easily update them if they are moved to a different location on the surface later by executing the command again.

Prompts

Choose Grid or Tmesh File to Process dialog

Select points, inserts, lines and polylines to convert.

Select objects: *pick the entities to convert*

Converting entities ...

Done.

Pulldown Menu Location: 3D Data

Keyboard Command: convert3d

Prerequisite: entities to be converted, grid or triangulation file

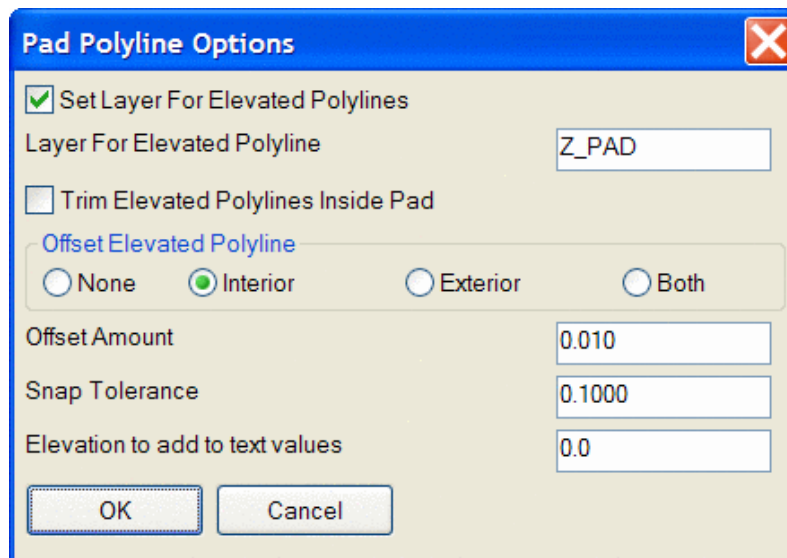
Pad Polyline By Interior Text

This command allows you to set one or more pad elevations using interior text labels.

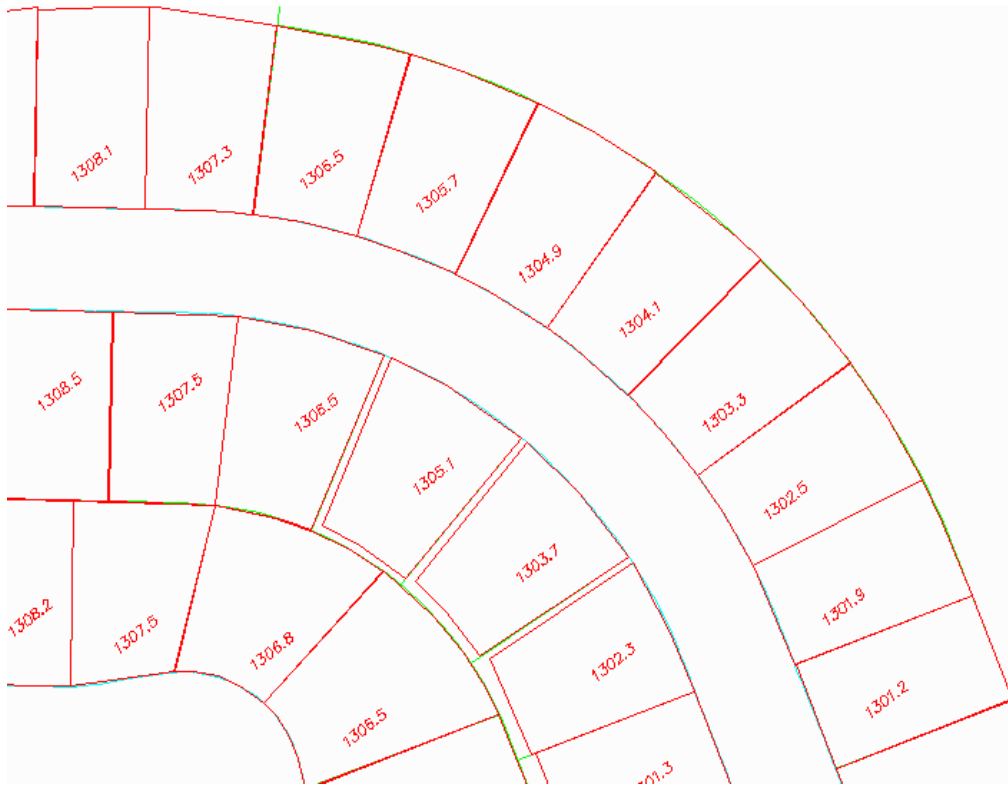
In the Pad Polyline Options dialog box, you can choose whether to have the new geometry created on a new layer, and if so, specify the name for the layer. You also specify whether to trim any elevated polylines crossing through the new elevated pad. You also specify whether to horizontally offset the new polyline from the original geometry, and if so, whether to offset to the inside, outside, or both, and by how much.

The Snap Tolerance field joins linework which falls within the range you set to create a pad.

Elevation to add to text values adds to the values from the elevation labels. Often, elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 523.5, 543.3, 537.2 sometimes they are listed as simply 23.5, 43.3, 37.2. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing.



After completing the dialog box, click OK. You are prompted to select the layers you want to use for the text representing pad elevations and for the boundaries of the pads. You do this to tell the routine which layers to process once you are prompted to select objects to convert. Also, sometimes pads are drawn with linework from two different layers and Carlson Civil allows you to pick all the linework layers. If you chose to Trim Elevated Polylines Inside Pad, you are also prompted to identify by selection the layer(s) with the elevated polylines that you want trimmed. Finally, select all the pads, labels, and existing elevated polylines, press Enter, and the new polylines with elevations will be created.



Prompts

Select layer sample of elevation text: *pick a label text*

Selected text layer —TX07

Select layer sample of boundary linework: *pick a pad polyline*

Selected linework layer PAD

Select another layer sample of boundary linework (Enter to continue):

Select text and linework to process.

Select objects: *select pad and text entities*

Analyzing entire selection...

Set elevation for 1 polylines.

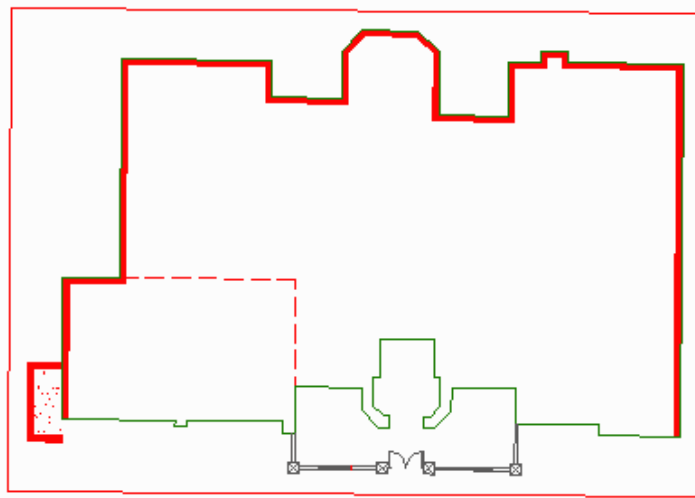
Pulldown Menu Location: 3D Data

Keyboard Command: pad_by_text

Prerequisite: Pad polylines and elevations

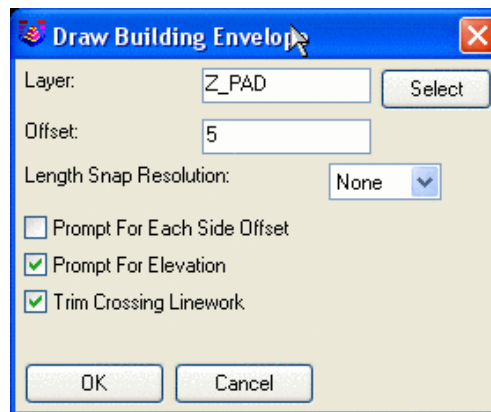
Draw Building Envelope Polyline

This command creates a rectangular polyline around selected linework. This can be used to give a building all one elevation.



In the Draw Building Envelope dialog box, set the layer for the new polyline and the desired offset value, (the above example is offset by 5 feet). Select whether or not to be prompted for specified offsets for each side, an elevation for the new polyline, and whether or not to trim existing linework that crosses the new polyline. Click OK. Select the entities that make up the building.

Prompts



Select building lines:

Select objects: *pick the building linework*

Draw another building envelope [**<Yes>/No**]?

Pulldown Menu Location: 3D Data

Keyboard Command: bldg_perim

Prerequisite: Building linework

Define Lot Edge Grade Rules

This command establishes grade rules for use in elevating lot edges with respect to a reference 3D polyline such as a road edge or other feature that must remain at its established elevations. Elevating lot edges will create 3D linework along selected lot edges in the drawing and set them at their designated elevations based on the rules established by this command (see *Elevate Lot Edges By Grade Rules* for a more detailed explanation).

This routine defines the actions to take when elevating lot edges by grade rules, establishing three categories of slopes to create (normal, minimum, and maximum) in the two possible conditions that will occur (cut or fill), for

the two edges of the lots being elevated (front edges and back edges). You may also establish intermediate grade breaks along the side lot lines by adding or editing additional grading rules in the middle windows. You define both cut and fill conditions in this routine, and the *Elevate Lot Edges By Grade Rules* command will use the appropriate condition based on whether the lot edge is in cut or fill at any given location with respect to a selected reference grade line such as a road edge or other feature.

Prompts

On execution, the routine prompts for a grade rules filename (.grr) to create if new, or select an existing .grr file. Once chosen, the following settings dialog will appear:

Options:

Slope Type: Choose which type of slope to define, either *percent*, *ratio*, or *vertical difference*.

Normal slope: Set the desired slope to use in initial lot line elevation. This is the slope that is applied whenever you first execute *Elevate Lot Lines By Grade Rules*. Note that positive slope values are from the reference grade line **up** to the front lot line, negative slope values are from the reference grade line **down**, and may be set either positive or negative for either cut or fill conditions. This is useful to force positive drainage toward a roadway, even where the reference grade line is in a fill condition.

Min/Max Slope: These slope values are the limits for automatic adjustment in balancing the final grading of the lots in the *RoadNET* routines. The lot lines' slopes are adjusted to values between these two limits to achieve balanced cut and fill, and if balance is not achieved within these limits, then the entire site is raised or lowered to reach a balanced condition.

Side Line Grade Breaks: The middle windows in the dialog allow you to add, edit, or remove additional grade breaks along the side lot lines and are established at a distance from the previous grade break (either the front line or the previously defined break point), at a given slope (again, defined using percent, ration, or vertical difference).

Back Slopes: These values are for establishing the back lot lines' slopes from the last encountered grade break (either the front line, or intermediate grade breaks).

Back Lot Edge At Existing Ground: Selecting this option will make the back line slopes portion unavailable, and

establish their elevations at the existing grade in their location. This will also prompt for selection of the existing surface model when executing the *Elevate Lot Lines By Grade Rules* routine to establish the required elevations along the back lot lines. Besides locating the back lot corners at the existing ground, this option will also drape the back lot edge on existing ground and add vertices into the back lot edge as needed to follow any undulations in the existing ground surface.

Pulldown Menu Location: 3D Data >> Elevate By Grade Rules

Keyboard Command: roadnet_grr

Prerequisite: None.

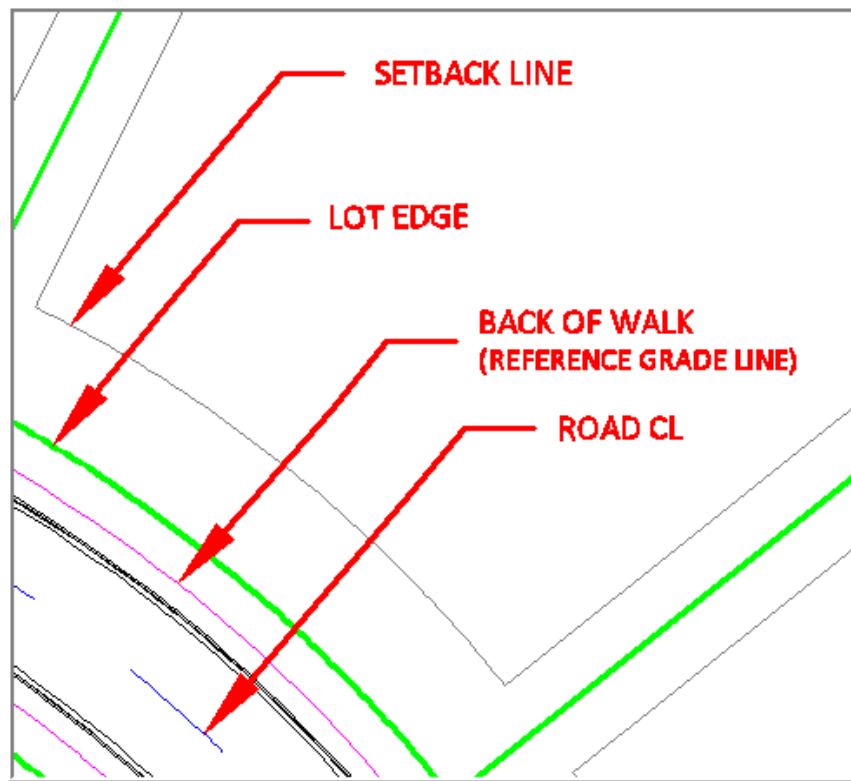
Elevate Lot Edges by Grade Rules

Elevating lot edges is useful to establish final grading for subdivisions or selected lot boundaries, and is also used in the *Road Network* routines by linking to the grade rule file. Once this final grade is established (by executing *Triangulate and Contour* selecting the elevated 3D lot lines as source objects), there are options to balance cut and fill by adjusting the lot lines' elevations based on the settings established in the grade rules (.grr) file. The initial elevation of lot lines will apply the **normal** slope, and the balancing routines will only adjust within the limits established by the **minimum** and **maximum** slope settings in the .grr file.

Consider the following example: you have a small subdivision with lot lines established using *LotNET* or other routine, and wish to elevate them with respect to the edge of the proposed roadway. Initially, the lot lines are at zero elevation. In this example, you will use the back edge of the sidewalk that is established as part of the roadway cross section template that creates all linework, at proposed grade based on the design profile of the roadway, and you wish to maintain this edge of walk at its design elevations.

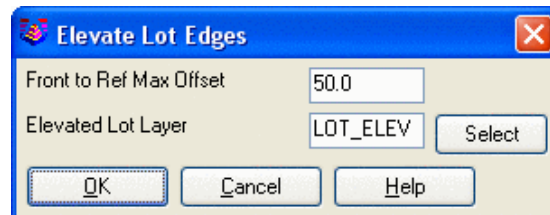


Small Subdivision Design with Roads and Lots



Detail of roadway and lot edges

Execute the routine, and you will be prompted for some initial information:



Specify the distance away from the reference grade line to seek the front lot lines to elevate and establish the layer to place the elevated lot lines on. This retains the original 2D zero-elevation lot lines to use in annotation, etc. on their original layers.

Prompts

Select reference elevation polylines.

Select objects: *select reference 3D polyline with elevations*

Select lot linework to elevate.

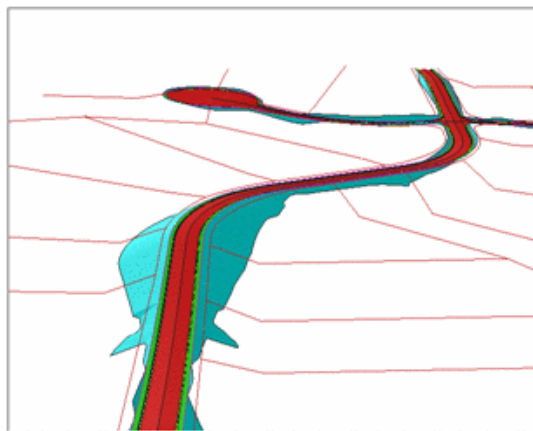
Select objects: *select lot lines to elevate, including side lines and rear lines*

Loading edges...

Loaded 826 points and 2250 edges

Created 1425 triangles

Elevated 18 lot edges.



Front, side and back lot lines elevated

Pulldown Menu Location: 3D Data >> Elevate By Grade Rules

Keyboard Command: elevate_lots

Prerequisite: 3D reference grade line, lot lines to elevate

Elevate Pads by Grade Rules

This command sets the elevation of closed polylines at a specified slope from reference 3D polylines. For each closed polyline, the program finds the nearest offset to a reference polyline. Then the pad polyline is elevated using the reference polyline elevation at this nearest offset position and applying the vertical offset as the slope times the offset distance.

When the original pad polyline is 3D, the program maintains the original slopes of the 3D polyline and elevates by raising or lowering the 3D polyline instead of setting all the vertices to the same elevation. This method applies for a pad with varying elevations such as for an exposed basement.

Elevate Pad Polylines ✕

Reference Elevation: Highest Elevation ▾

Slope Type: Percent ▾

Cut Normal Slope: 2.000

Cut Min Slope: 1.000

Cut Max Slope: 10.000

Fill Normal Slope: 2.000

Fill Min Slope: 1.000

Fill Max Slope: 10.000

Vertical Offset: 0.000

Assign New Layer

Elevated Pad Layer: PADS Select

Retain Original Polyline

Add Grade Breaks to Reference Polylines

Swale Options

Draw Swale Polyline

Swale Layer: SWALE Select

Offset From Pad: 10.000

Center Taper: 2.000 Edge Extend: 5.000

Slope From Pad %: -1.000 Cross Slope %: -0.500

OK Cancel Help

The **Reference Elevation** has four methods. Elevation At Middle uses the middle position of the pad polyline segment that is on the frontage side of the pad and locates the station along the reference polyline that is at the perpendicular offset from this position. Highest Elevation takes the frontage segment of the pad polyline and checks the station range for this segment along the reference polyline to find the highest elevation within this range which is used as the elevation reference. Likewise, Lowest Elevation uses the lowest elevation on the reference polyline in front of the pad. The Reference Point method uses a point placed along the pad by the Tag Pad Reference Point command.

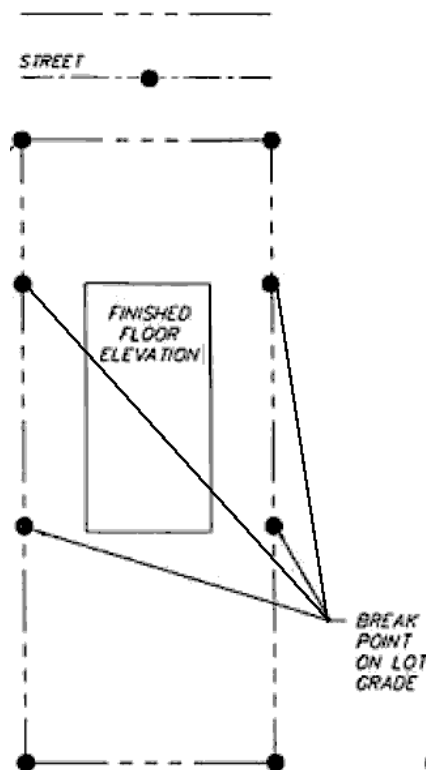
The program prompts for the existing surface triangulation file which is used to compare to the reference elevation and choose between using the specified cut or fill slope from the reference elevation to the pad. This feature allows you to have steeper slopes in cut than fill conditions to help with site balancing.

The **Min** and **Max Slopes** are stored with the pad polylines for use by the site balancing option in Calculate Total Volumes in SiteNet. These slopes are used as grading rules to make sure the pad stays within this slope range from the reference polyline when the pads are raised or lowered during the site balancing. The Min and Max Slopes are not used during the Elevate Pads By Grade Rules which only uses the **Cut/Fill Normal** slopes. Use the Grade Rules Editor command to edit these slopes and elevation reference info that is assigned to the pad polyline.

The **Vertical Offset** is added to the elevations of the pad polylines.

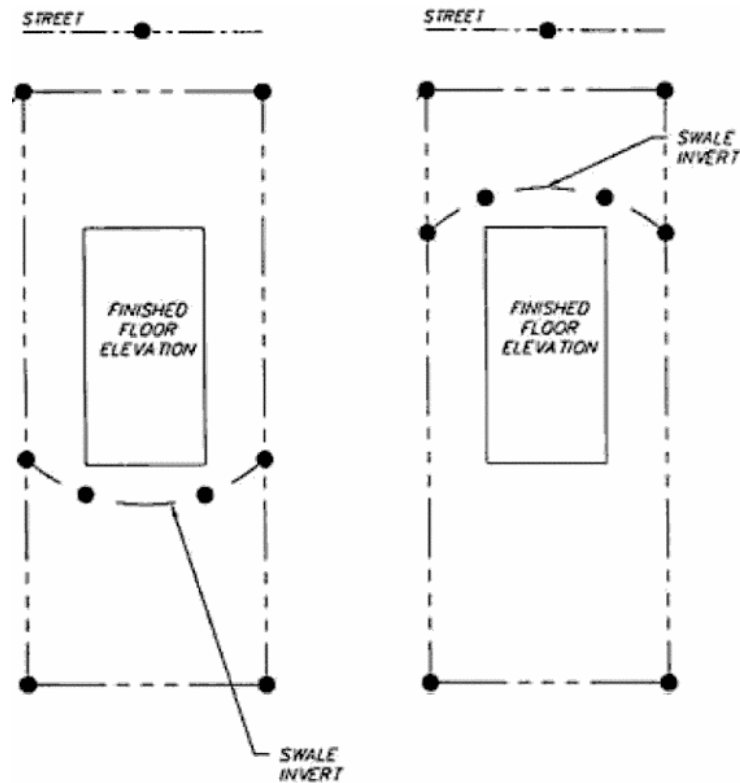
The **Assign New Layer** option changes the original layer to the specified layer for the elevated pad polylines. The **Retain Original Polyline** option creates new elevated polylines and leaves the original polylines unmodified.

The **Add Grade Breaks to Reference Polylines** adds break points to the side lot lines at the ends of the building pad as shown in this graphic.



The **Draw Swale Polyline** creates 3D breakline polylines to control storm water flow around the building pad. When the lot is sloping towards the street, the swale polyline is at the back of the pad as shown in the left graphic. When

the lot is sloping away from the street, the swale polyline is at the front of the pad as shown in the right graphic. The swale polyline elevation is set relative to the building pad elevation by the **Slope From Pad**, and the **Cross Slope** controls the swale elevations for connecting with the lot sides. The **Offset From Pad** sets the distance from the building pad to the swale polyline. The **Center Taper** controls how much the swale polyline bulges out in the middle and the **Edge Extend** sets the position of the swale polyline break points from the side lot lines.



Prompts

Select Existing Ground Surface Pick the existing triangulation surface

Select reference elevation polylines.

Select objects: *select reference 3D polyline with elevations*

Select pad polylines to elevate.

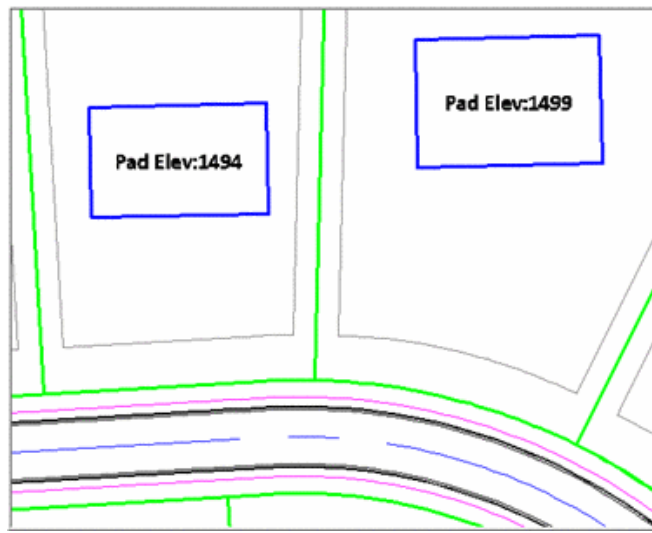
Select objects: *select pad polylines to elevate*

Loading edges...

Loaded 826 points and 2250 edges

Created 1425 triangles

Elevated 4 pad polylines.



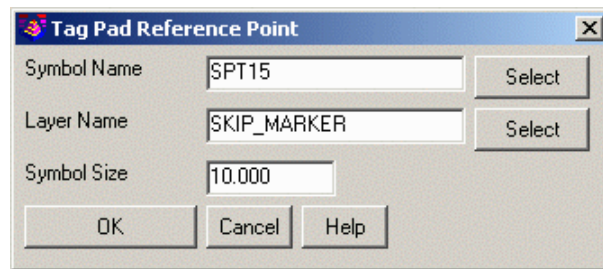
Pulldown Menu Location: 3D Data >> Elevate By Grade Rules

Keyboard Command: elevate_pads

Prerequisite: 3D reference grade line, closed polyline pads to elevate

Tag Pad Reference Point

This command creates points on building pads that are used by the Elevate Pads By Grade Rules command when using the Reference Elevation by Reference Point method. Each pad should have a single reference point. You set the symbol, layer and size for the reference points in the options dialog.



Prompts

Tag Pad Reference Point dialog

Pick pad reference point: *point a point*

Pick pad reference point (Enter to end): *press Enter*

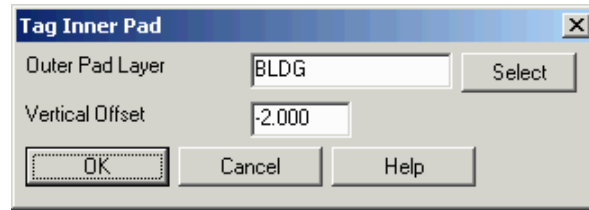
Pulldown Menu Location: 3D Data > Elevate By Grade Rules

Keyboard Command: tag_pad_ref_pt

Prerequisite: None

Tag Inner Pads

This command assigns polylines to use as building pads within larger building pads such as a garage slab within a building. These inner pads are assigned a vertical offset from the main pad. The options dialog has a setting for the layer name of the main pad polyline. The Elevate Pads By Grade Rules command will process the inner pads to set the elevation by the vertical offset from the main pad.



Prompts

Tag Inner Pads dialog

Select inner pads for layer BLDG.

Select objects: *select pad polylines to tag*

Pulldown Menu Location: 3D Data > Elevate By Grade Rules

Keyboard Command: tag_inner_pad

Prerequisite: closed pad polyline

Grade Rules Editor

This command allows you to elevate other types of linework (besides lot edges and pads) with respect to a single reference elevation and offset point (or two single points if elevating a previously-elevated object using Lot Edge rules). This differs from the typical *Elevate Lot Edges By Grade Rules* in that instead of referencing the object to a reference line of constantly changing elevation, you will be setting a single point (or two single points) as a starting reference value.

You choose the linework you wish to elevate and establish for it a reference elevation and offset, if any. You may enter these values directly, or pick an object from the screen to establish the reference. If you leave the fields at their default of 0.000, you will elevate the selected object with respect to that zero reference elevation, so make sure that you have some valid reference data set in the fields. If you are elevating an object that has not previously been elevated using *Elevate Lot Edges By Grade Rules*, you will establish a single reference point/offset. If elevating a previous "Lot Edge Rules" object, you will establish two reference points/offsets, one for the front, and one for the back. You will also establish the grade rule for the back portion in addition to a single set of rules for a 2D object. (see dialogs below for differences).

The settings work much the same as the other grade rules (lot edges and pads) except that instead of elevating things based on a linear, changing reference elevation line, you are setting a single point of reference (or two single points) to elevate the selected object.

Prompts

Select grade polyline: *select the linework you wish to elevate*

If the selected object has not been previously elevated using *Elevate Lot Edges By Grade Rules*, you are then presented with the following dialog to establish the relationship of the selected linework to a reference elevation and offset. You may enter the values directly, or pick a point in the drawing and the values are calculated from the picked point.

Grade Z:	1487.341
Reference Z	1485.965
Reference Offset	27.514
Pick Reference Point	
Slope Type	Percent
Normal Slope	5.000
Min Slope	1.000
Max Slope	20.000

If selected object has been previously elevated using *Elevate Lot Edges By Grade Rules*, you will be presented with the next dialog, which illustrates the difference in the two cases:

Grade Z:	1511.230
Front Reference Z	1496.564
Front Reference Offset	27.500
Pick Front Reference Point	
Back Reference Z	1496.564
Back Reference Offset	55.030
Pick Back Reference Point	
Slope Type	Ratio
Front to Ref Max Offset	50.000
Front Normal Slope	1.875
Front Min Slope	25.000
Front Max Slope	3.000
Distance Vertical	50.000 25.517:1
Add Edit Remove	
<input type="checkbox"/> Back Lot Edge At Existing Ground	
Back Normal Slope	22.642
Back Min Slope	100.000
Back Max Slope	10.000

The fields and their values function the same as those in *Define Lot Edge Grade Rules*, so please refer to that command for more information.

Pick reference point: *<Osnap on> select a point in the drawing as a reference, and the values are calculated and populate the dialog fields for you.*

Select grade polyline (Enter to end): *select another, or press Enter to conclude selection*

Pulldown Menu Location: 3D Data >> Elevate By Grade Rules

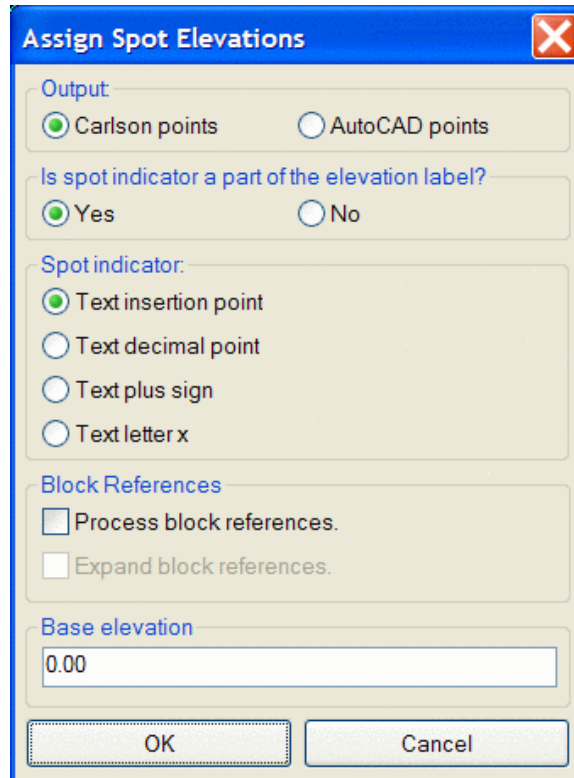
Keyboard Command: edit_grades

Prerequisite: linework to elevate

Convert Spot Elev To Points

This command takes spot elevation entities with zero elevations and assigns them elevations according to corresponding elevation labels.

This dialog box allows you to choose the format of the spot elevations entities that you want to convert.



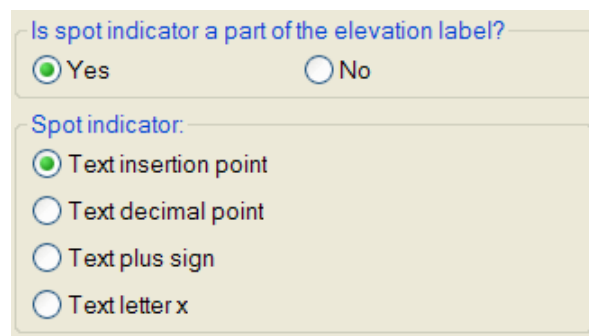
Output:

Carlson points: creates Carlson points at elevation of spot and stores them in coordinate file

AutoCAD points: creates AutoCAD point objects at elevation of spot

Is spot indicator a part of the elevation label?

If set to "Yes", four choices for Spot indicator are available to select from:



Text insertion point: uses the insertion point of the text for the location of the new point

Text decimal point: uses the decimal point in the text for the location of the new point

Text plus sign: uses the plus sign in the text for the location of the new point

Text letter x: uses the letter x in the text for the location of the new point

If set to "No", seven choices for Spot indicator are available to select from:

Is spot indicator a part of the elevation label?
 Yes No

Spot indicator:
 Linework leader
 Linework cross
 Text plus sign
 Text letter x
 AutoCAD point
 Circle or arc
 Closed linework



Linework leader: creates a data point at the end of a leader

Linework cross: creates a data point at the intersection of a linework cross

Text plus sign: creates a data point at the insertion point of a text plus sign

Text letter x: creates a data point at the middle of a text letter x

AutoCAD point: creates a data point at the node of an AutoCAD point

Circle or arc: creates a data point at the center of a circle or arc. If a circle is composed of two 180-degree arcs, however, it only creates 1 data point

Closed linework: creates a data point at the geometric center (centroid) of closed linework, such as a square or triangle shape

Block References:

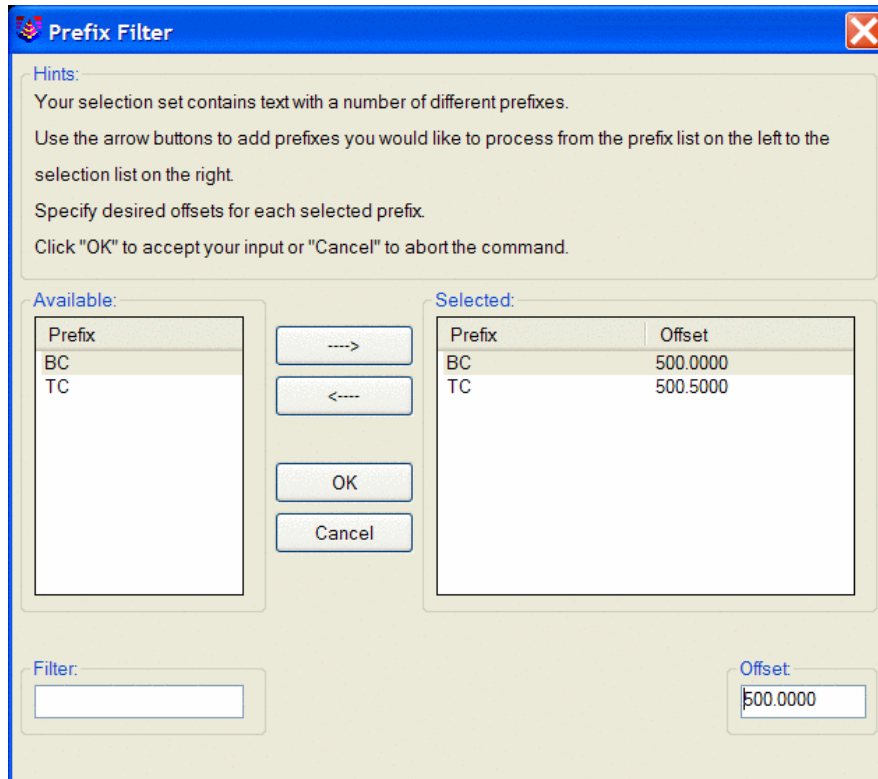
Process Block References: If check box is cleared, Carlson Civil searches only text entities for elevations, but if checked, Carlson Civil will search block references for elevations that are stored as attributes of a block. Use this option if the elevation is an attribute and the symbol designating the location of the spot elevation are both part of the block definition.

Expand Block References: Use this option to search block references when the elevation is stored as an attribute of a block, but the symbol designating the location of the spot elevation is a different block or even other geometry that is not defined within a block.

Base elevation: The value entered here is added to the existing spot elevations for all newly created points. Often, elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 523.5, 543.3, 537.2 sometimes they are listed as simply 23.5, 43.3, 37.2. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing. Note: The base elevation will not be added to any elevations that are closer to the base elevation value than they are to 0; e.g. if a base elevation of 500 is specified, 500 will be added to elevations like 23.4, 45.5, etc, but will not be added to elevations like 456.4 or 468.9.

Prefix Filter: Carlson Civil examines all selected spot elevations for prefixes or suffixes. If they are all the same, the command proceeds, but if there are different prefixes and/or suffixes found, the Prefix Filter dialog box is

invoked. This dialog box allows you to select which prefixes and/or suffixes to use to create spot elevations, and also allows you to use different offset values for each.



Prompts

Starting point number <1>: *press Enter*

Select TEXT, MTEXT spot elevations to process and any associated leader lines:

Select objects: *pick entities to process*

Pre-processing entity #40 of 40...

Filtering text entities

Processing elevation #40...

Converted 40 spot elevations.

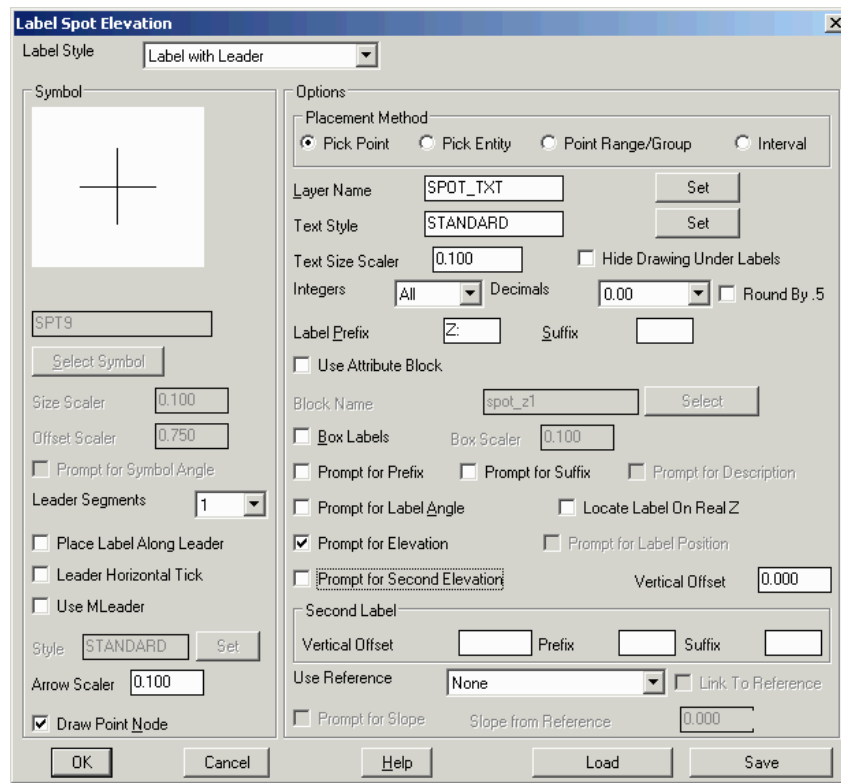
Pulldown Menu Location: 3D Data

Keyboard Command: spotelv2

Prerequisite: Spot elevations

Draw Spot Elevations

This command creates spot elevation labels based on user settings.



Label Style: This option at the top of the dialog determines which options are available in the rest of the dialog.

Label with Leader: This option draws a leader between the spot location and the label. The style of the leader is controlled by the current DIMSTYLE settings.

Label with Symbol: Draws the specified symbol at the spot with the label off to the side.

Carlson Point: This option creates a Carlson point entity with the point#, elevation, description attribute block and stores the point to the current coordinate file. When creating Carlson points, there is an option for **Prompt For Description** for entering descriptions for the points for the point entities and coordinate file.

Label Decimal on Point: Draws only the elevation label and positions the label so that the label decimal is on the spot.

Label Insertion on Point: Draws only the elevation label and uses the spot location for the insertion point of the label.

Label Only: Draws the label without any indicator of the exact reference point for the label.

Symbol: Specify the symbol to be used for the location of the spot elevation. If Draw with Leader is selected, the use of a symbol is automatically disabled. Along with the symbol name, you can also set the **Size Scaler** to size the symbol and the **Offset Scaler** to control the offset between the symbol and the label. The **Prompt For Symbol Angle** option allows you to rotate the symbol. Otherwise, the symbol is drawn horizontal to the current twist screen.

Leader Segments controls how many leader segments the program will prompt for. **Place Label Along Leader** draws the label along the leader instead of horizontal off the end of the leader. **Leader Horizontal > Tick** draws a short horizontal segment at the label end of the leader. **Leader Horizontal > Underline** draws a leader line under the label. **Draw Symbol with Leader** combines the symbol with the leader. **Arrow Scaler** controls the size of the arrowhead. The **Use MLeader** creates an mleader entity. Otherwise a regular leader and separate text entities are drawn.

Draw Point Node: Specify whether to create a node (point entity) at the spot.

Options: Under Options, there are settings to control the layer, text style, text size and prefix/suffix for the labels.

Box Labels controls whether to draw a box around the label and **Box Scaler** controls the size of the box and is applied as a factor of the text size.

Locate Label On Real Z sets whether the label and symbol are placed at the spot elevation or at zero elevation. There are several settings for fields to prompt for when placing each spot label including prefix, suffix, position and angle.

Use Attribute Block: This option uses a block with attributes to draw the spot elevations. This allows you more control of the label layout. There is one default spot elevation attribute block called `spot_z1.dwg` in the Carlson Support folder. To customize the layout, open the attribute block drawing, make your edits and then save the drawing. The attribute block uses three attribute definitions: Integers, Decimals, Suffix. Integers is for the elevation digits to the left of the decimal point. Decimals is for the digits to the right of the decimal point. Suffix is a text description.

Use MText: This option draws the labels as MText instead of regular Text.

Prefix/Suffix on New Line: This option labels the elevation value on a separate line from the prefix or suffix.

Placement Method:

Individual: Prompts the user for a screen pick or a point number to specify the location of the spot elevation.

Pick Entity: This method labels the elevation of the selected 3D entity. For 3D polylines, the elevation comes from the polyline at the location the polyline is selected.

Point Range/Group: Draw spot elevations at the specified point numbers or point group name from the current coordinate file.

Interval: Prompts the user to select an existing polyline and specify an interval distance to set spot labels along the polyline. It also prompts for number and interval for additional labels to be set at offset distances right and/or left of the polyline.

Use Reference:

Off: User is prompted to supply elevation for each new spot elevation.

Single Surface File: Prompts user to specify the triangulation or grid surface model file to read the elevations from. Spot elevations created with this method are automatically updated if the surface model elevation changes.

Two Surface Difference: Prompts user to specify an existing surface file and a design surface file, and labels the elevation difference between the two.

Slope Percent/Ratio From Point/Polyline: These methods allow you to set the spot elevation by a slope percent or ratio from a reference point/polyline. These methods have another option for whether to **Prompt for Slope** which will prompt for the slope from the reference for each spot. Otherwise, the **Slope From Reference** setting in the dialog is used. When using the Polyline reference method, the program finds the station-offset of the spot point along the polyline and uses the elevation of the polyline at that station for the reference.

Link To Reference applies to using a surface file or 3D polyline for the spot elevations. This link option will automatically update the labels when the reference surface file or 3D polyline change.

The **Hide Drawing Under Labels** option creates a wipeout under the spot label to hide any other drawing entities behind the label.

The **Integers** setting controls how many decimal places to the left of the decimal point to label and the **Decimals** settings controls how many decimal places to the right of the decimal point to label. The **Round By .5** option rounds the last digit to either 0 or 5.

Prompt for Prefix/Suffix: These options will prompt for the labels for each point which is useful when labeling different kinds of spots.

Prompt for Label Position: When using Label With Symbol style, this option will prompt for where to place the label for each spot.

Prompt for Elevation: Prompts to enter the elevation for each spot. Otherwise, the elevation of the picked point is used.

Prompt for Second Elevation: Prompts for a second elevation for each spot. This option applies when labeling two elevations at the same point such as top and bottom of curb when they have varying elevation differences between them. If the two elevation have a fixed elevation difference, then you can use the **Vertical Offset** for the **Second Label**.

Prompt for Label Angle: This option allows you to rotate the label for each spot. Otherwise, the label is drawn horizontal to the current twist screen.

Vertical Offset adjusts the elevation label by the specified amount.

Additional Label has separate settings for **Vertical Offset**, **Prefix** and **Suffix** which allows for labeling two elevations for the same spot such as labeling both bottom and top of curb.

$$+123.45 + \boxed{123.45} \quad \swarrow \quad 123.45 + 123.45$$

The **Save** and **Load** buttons write and recall the settings to a .LSE file so that you can store different label styles and share them.

Pulldown Menu Location: 3D Data

Keyboard Command: labspot

Prerequisite: None

Move Spot Elevations

This command allows you to reposition spot elevation label on the reference surface. The value of the label will update for the new position. In order for this command to run, the spot elevation labels must be created using the Draw Spot Elevations command with the Use Reference set to Single Surface File and the Link To Reference option turned on.

Prompts

Select spot elevation to move: *pick entity*

Pick new position: *pick a point*

Select spot elevation to move (Enter to end): *press Enter to end*

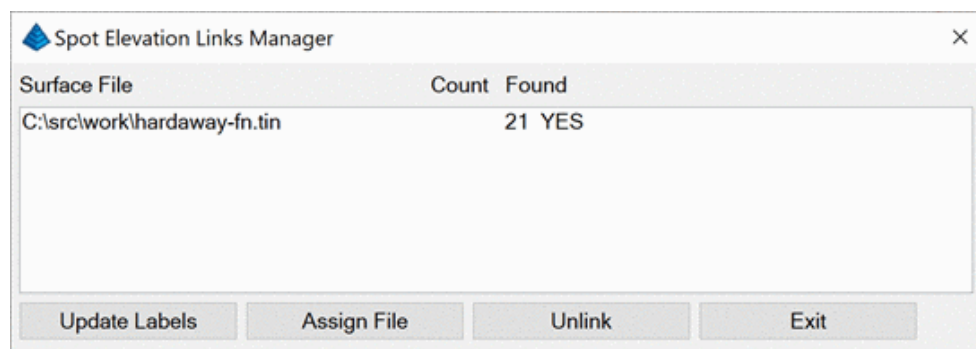
Pulldown Menu Location: 3D Data > Spot Elevations

Keyboard Command: movespot

Prerequisite: Spot Elevation labels drawn using Draw Spot Elevations command

Spot Elevation Links Manager

This command reports the number of spot elevations in the drawing created by the Draw Spot Elevations command and created with the Link To Reference option turned on. The report includes the name of the linked reference surface file and whether the file is found on the computer. The **Update Labels** function updates the spot elevations to match the current surface elevations in case the labels are out-of-sync. The **Assign File** function changes the reference surface file name for the spot elevations. The **Unlink** function removes the reference surface file link from the spot elevations.



Pulldown Menu Location: 3D Data

Keyboard Command: labspot_mgr

Prerequisite: Spot elevations

Adjust Elevation Labels

This command has several functions that allow you to modify spot elevation labels.

The **Remove Base Elevation** function removes the base elevation amount from the labels. For instance, often elevations are abbreviated to save space. If every elevation in a drawing is in the 500's instead of labeling every elevation 523.5, 543.3, 537.2 sometimes you may wish to have them displayed as simply 23.5, 43.3, 37.2. This command allows you to adjust the labels by a given amount, such as 500, to every label elevation. This does not affect the actual elevation of entities in the drawing or in the associated surface model file.

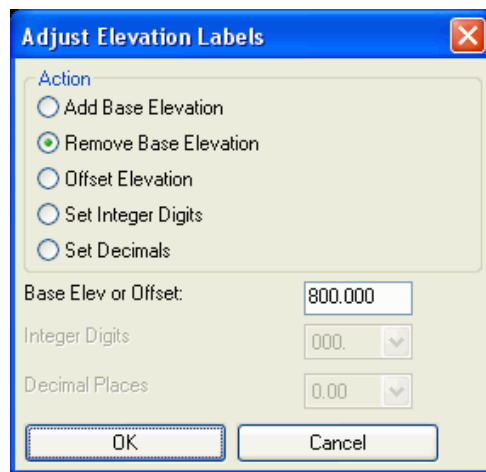
The **Add Base Elevation** function is the reverse of Remove and applies when the labels are missing the base elevation and you want to add this elevation into the labels.

The **Offset Elevation** function adds the specified offset amount to the elevation labels and applies when elevation labels need to be adjusted by a fixed vertical offset.

The **Set Integer Digits** function sets the number of digits to the left of the decimal point for the elevation labels.

The **Set Decimals** function sets the number of digits to the right of the decimal point for the elevation labels.

The **Elevation Block Attribute Tag** edit allows the user to specify the name of the tag in a block reference to apply elevation adjustment functions to. This settings does not apply to Carlson Coordinate Point block references for which the "ELEV2" tag attribute is always adjusted.



Prompts

Select a sample elevation label: *select single label to identify the source layer to process*

Select spot elevation labels to process.

Select objects: *select the text to process*

Pulldown Menu Location: 3D Data

Keyboard Command: adjust_elevation_labels

Prerequisite: Spot Elevation label text

Elevate Text

This command sets the elevation of text entities by reading the elevation value from the text label. For example, this command can elevate spot elevation labels. Having the text entities at real elevation instead of zero can be useful for viewing in 3D or using for surface modeling.

Prompts

Select text to elevate.

Select objects: *pick text entities*

Elevated 12 text entities.

Pull-down Menu Location: 3D Data

Keyboard Command: ztext

Prerequisite: None

Interpolate Points

This command divides the distance between two points and inserts one of the point symbols at the specified distances. It can also interpolate elevations. (To interpolate elevations the points picked must be at their real Z axis elevation.)

Prompts

Interpolate Elevations [<Yes>/No]? *press Enter*

Point w/elevation to calculate from?

Pick point or point number: *3*

2nd Point w/elevation?

Pick point or point number: *pick a point*

Number of Segments/Divisions: *5*

The command then locates 4 points.

Pull-down Menu Location: 3D Data > 3D Points

Keyboard Command: divlin

Prerequisite: Execute *Drawing Setup* to set defaults. Locate two points to divide between and if you want to interpolate elevation they should have a real Z axis elevation.

Elevation Between Points

This command interpolates new points between two reference points at a single elevation or elevation interval. The routine uses the elevations of the two reference points together with the target elevation to figure the interpolation distance. For example, with one reference point at 100, the other reference at 104 and the target at 101, then the new point will be created 1/4th the distance from the first reference point towards the second reference. The target elevations are used as elevations for the new points.

Prompts

Point to interpolate from.

Pick point or point number: *pick a point*

Point to interpolate to.

Pick point or point number: *pick a point*

Add single elevation or elevation interval [Single/<Interval>]? *press Enter*

Enter elevation interval: *1*

Pulldown Menu Location: 3D Data > 3D Points

Keyboard Command: addptz

Prerequisite: None

3 D Points - Interpolate Entity

This command divides the distance of a *LINE*, *ARC* or *PolyLINE* and locates points at the computed distances. It also interpolates elevations (To interpolate elevations the points picked must have an AutoCAD or real *Z*/elevation). The figure below shows a graphic example.

Prompts

Interpolate Elevations <Y>: *press Enter*

Point w/elevation to calculate from?

Pick point or point number: *1*

2nd Point w/elevation?

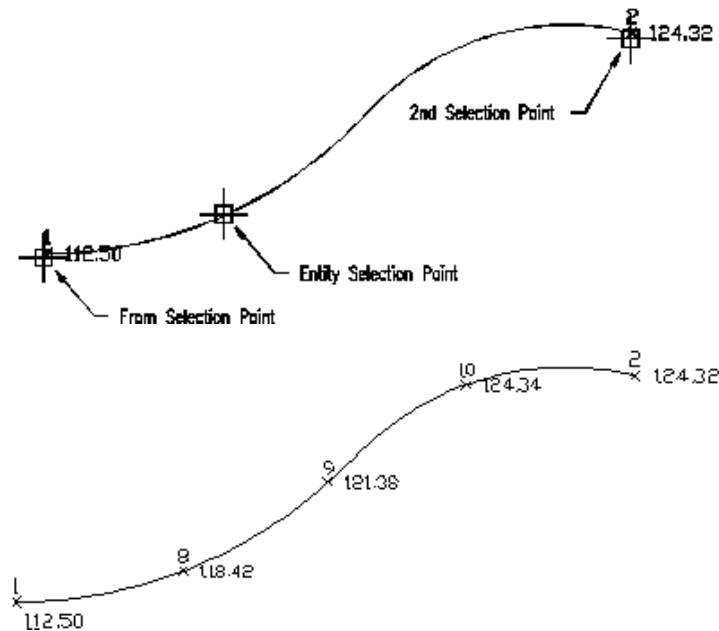
Pick point or point number: *2*

These points don't have to be on the entity selected to divide.

Select Entity to Divide: *pick point on entity*

Number of Segments/Divisions: *4*

The command then locates 3 points along the selected entity.



After selecting the points above (top), new points are located along the selected entity (above, bottom).

Pulldown Menu Location: 3D Data >> 3D Points

Keyboard Command: divent

Prerequisite: Two elevation points and the entity to divide between.

Points by Slope Ratio

This command allows you to locate points by defining a slope from a reference elevation. There is an options dialog where you can define the slope format as percent or ratio. Enter the slope as positive for up, negative for down. You can also choose to enter elevation difference (vertical) instead of a slope. Or match the slope between two other points.

There are several difference methods for defining the X,Y location for the points.

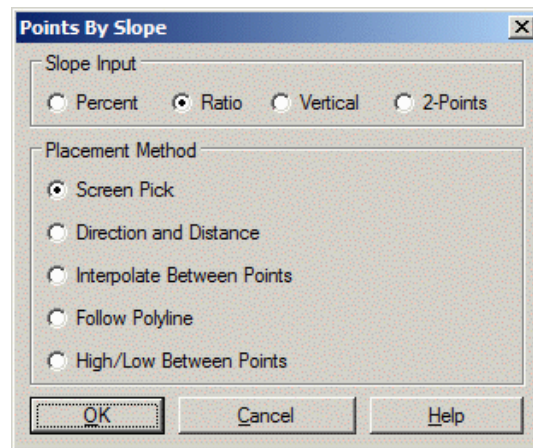
Screen Pick: Prompts for a reference starting point and then you screen pick the new points to create and enter the slope from the reference.

Direction and Distance: First pick the starting point. If the picked starting point is at zero elevation, the program will prompt for the reference elevation. Next pick a point for the direction. Then enter the slope and the horizontal distance.

Interpolate Between Points: First pick the starting point. If the picked starting point is at zero elevation, the program will prompt for the reference elevation. Next pick the second reference point. Then enter the slope and the number of points to create between the two reference points. This number of points will be evenly interpolated between the reference points.

Follow Polyline: This method is similar to Screen Pick except that instead of using the straight line distance between the reference and target points, you pick a polyline to get the distance. The program gets the distance by projecting the reference and target points onto the polyline and then following the polyline between these projected positions.

High/Low Between Points: With this method, you specify two reference points and two slopes. On the line between these two reference points, the program locates the point where the two slopes intersect.



Prompts

Slope ratio + for uphill - for downhill, Start point should be 3D.

Slope Ratio (? : 1) <2.0>: 1

Starting point ?

Pick point or point number: *screen pick a point*

Direction Point: *screen pick a point*

Enter Hz Distance or [I]nterpolate to Direction Point <I>: *press Enter*

Horizontal Dist: 214.94

Number of Points to Interpolate <2>: 5

Start Point Elevation: 0.0

End Point Elevation: 214.942

Difference in elevation: 214.942 Elevation add: 42.9884

Select/<Enter Point Elevation <42.99>>: press Enter

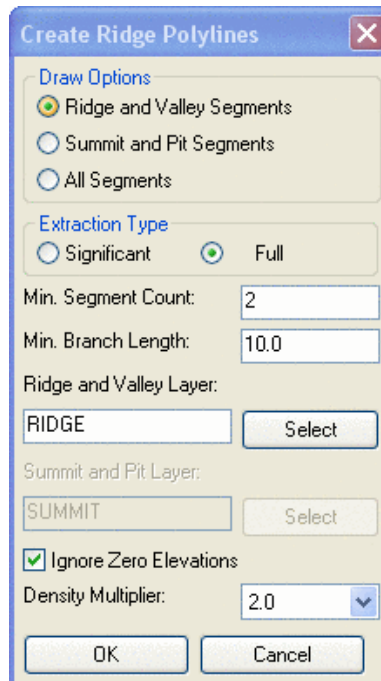
Pulldown Menu Location: 3D Data > 3D Points

Keyboard Command: SLP

Prerequisite: None

Create Ridge Polylines From Contours

This command creates 3D polylines along the tops of ridges and in the bottom of drainages for more accurate modeling of the surface.



Draw Options: Specify whether to draw Ridge/Valley segments, Summit/Pit segments, or all. Ridge/valley segments travel along the tops of ridges or bottom of valleys and are extracted from the contours lines by finding acute vertices and looking to the next adjacent contour for the corresponding acute vertex. Summit or pit segments model closed polyline contours and extend the trend either upward or downward to the peak or bottom point in the geometry.

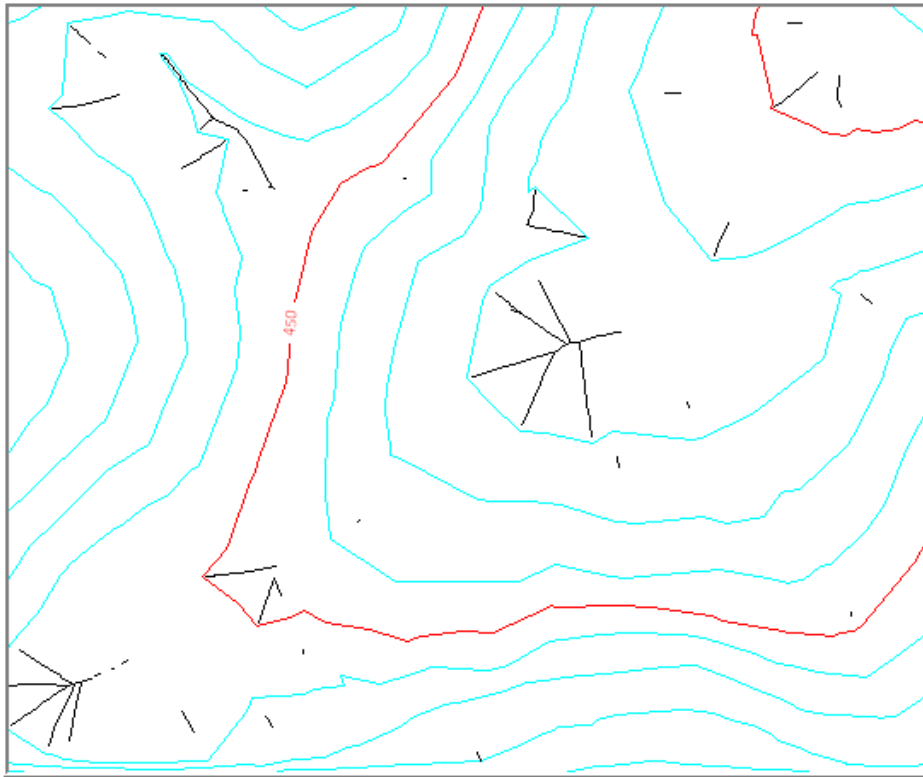
Extraction Type (Significant and Full): Significant segments are those segments that fall into relatively flat areas of the triangulation (large open spaces between the contours). Use this setting to reduce the number of segments generated by this command. Full extraction will model all segments.

Minimum Segment Count and Branch Length: Use this setting to prevent small segment branches from being generated. This can reduce the occurrence of errors in the output.

Layer Selection: Specify the layer that the ridge/valley and summit/pit polylines will be drawn on. You may choose the select button to specify from the list of existing layers, or specify a new layer by simply typing the new layer name.

Density Multiplier: The quality of output produced by this command is directly proportional to vertex density of contour polylines. This multiplier can be used to temporarily increase contour vertex density for the duration of calculations at the cost of additional processing time. Setting this multiplier to higher values generally reduces occurrence of errors in the output.

Ignore Zero Elevations: select this to ignore linework set at zero elevation.



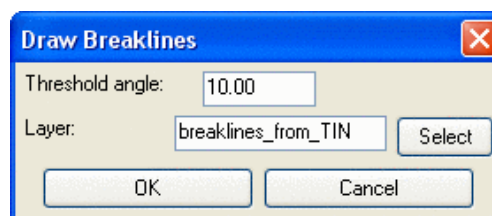
Pulldown Menu Location: 3D Data

Keyboard Command: cs_extract

Prerequisite: Contours with ridges and valleys

Create Breaklines From Triangulation

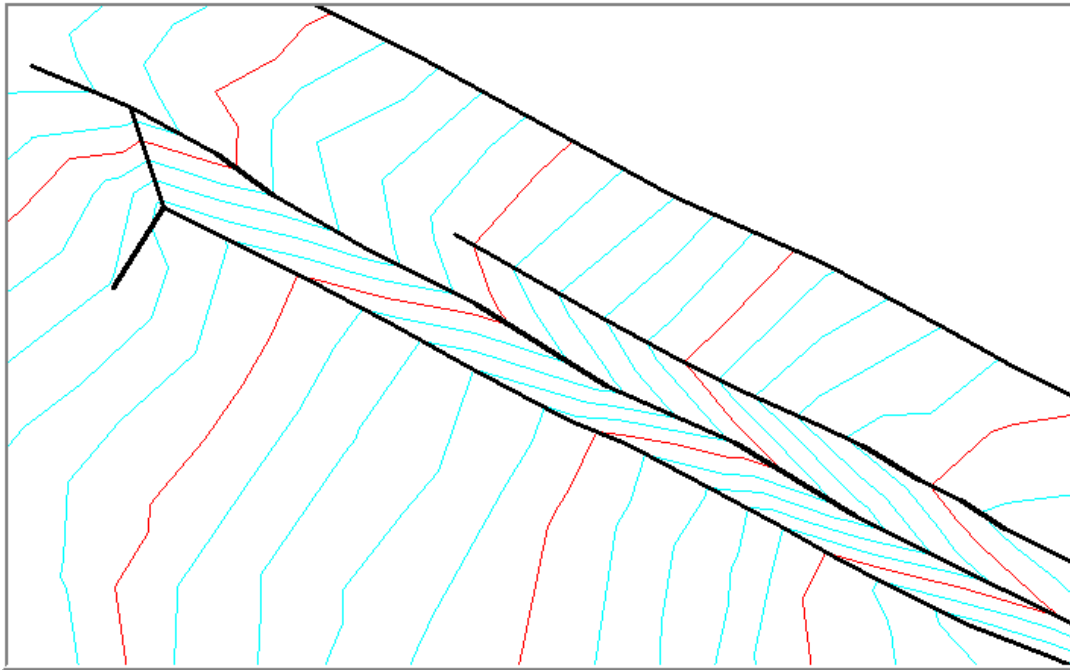
This command creates 3D polylines along the tops of ridges and in the bottom of drainages for more accurate modeling of the surface.



Threshold Angle: Specify the minimum angle between adjacent triangles in the surface model to generate a breakline in the drawing.

Layer: Specify the layer to draw the breaklines on. You may type a new or existing layer name, or press the Select button to choose from a list of existing layers.

Once you press *OK*, you will be asked to select the triangulation file to process. Once selected, the routine draws the breaklines on the indicated layer.



Pulldown Menu Location: 3D Data

Keyboard Command: draw_breaklines

Prerequisite: Triangulation file surface model

Offset 3D Polyline

This command allows you to offset a 3D polyline entity in both the horizontal and vertical directions. There are four offset methods. The *Interval* method applies one horizontal and one vertical offset to all the vertices of the polyline. The *Constant* method has a horizontal offset and sets the elevation of the polyline to one constant elevation. The *Variable* method allows you to specify each horizontal and vertical offset individually either by polyline segment or for each point. The vertical offset can be specified by actual vertical distance, percent slope or slope ratio.

Finally, the *Surface* method allows you to offset the 3D Polyline to intersect a target surface defined by a triangulation or grid file. This functions much like the **Design Pad Template** command on the **Surface** menu, but without creating side slope faces; only the intercept (or "daylight") line is created.

Prompts

Enter the offset method [<Interval>/Constant/Variable/Surface]: *press Enter*

Vertical/<Horizontal offset amount>: *15*

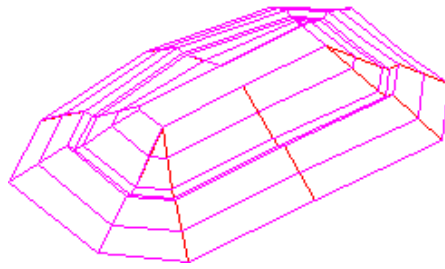
Percent/Ratio/Vertical offset amount <0>: *10*

Select a polyline to offset (Enter for none): *select a 3D poly*

Select side to offset: *pick a point*

Select a point on the graphics screen that is in the direction of the side of line to offset.

Select a polyline to offset (Enter for none): *press Enter*



Pulldown Menu Location: 3D Data

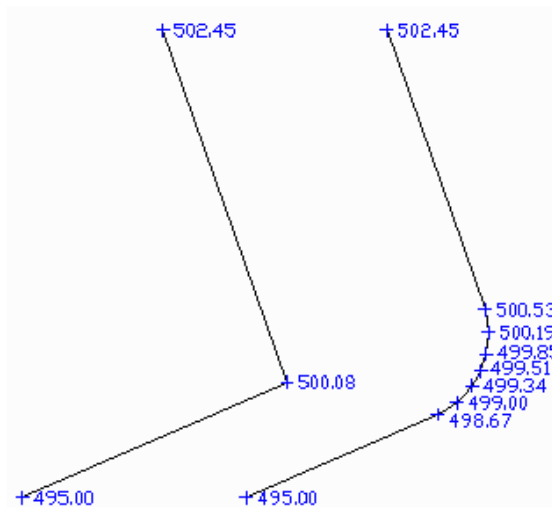
Keyboard Command: offset3d

Prerequisite: 3D polylines to use for selection; surface file for *Surface* method.

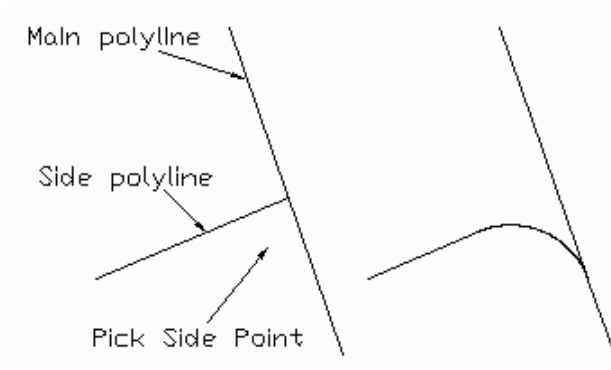
Fillet 3D Polyline

This command fillets two segments of a 3D polyline (or two un-joined 3D Polylines) with the given radius. AutoCAD's *FILLET* command does not support 3D Polyline entities. Since 3D polylines cannot have arcs, this command draws the fillet arc as a series of short chords. The elevations along the curve are interpolated from the 3D polyline.

There are two processing modes: corner and intersection. The corner mode works like the standard Fillet command except that it's in 3D. The intersection mode works at the intersection point between two 3D polylines. One polyline is set as the main polyline and the other as the side polyline. The main polyline is used for reference only and is not modified. The side polyline is modified to fit in the fillet radius. The intersection mode works for crossing and T intersections. An application of the intersection mode is for curb 3D polylines at road intersections.



Before and after fillet3d on the corner



Before and after fillet3d at intersection

Prompts

Fillet corner of a polyline or intersection of two polylines [*<Corner>/Intersection*]? *press Enter*

Enter fillet radius <10.00>: *press Enter*

Select a corner point on polyline: *pick 3D polyline near meeting point of two segments*

Select a corner point on polyline: *press Enter* (to end command)

Pulldown Menu Location: 3D Data > 3D Polyline Utilities

Keyboard Command: fillet3d

Prerequisite: 3D polyline

Join 3D Polyline

This command joins *3DPOLY* entities into a single 3D polyline entity. The routine requires that two endpoints be coincident, at the same elevation. A similar function is obtained with the **Join Nearest** command on the **Edit** menu, but will allow options to join across gaps and various options for treating the resulting common endpoint.

Prompts

Select the 3D polyline to join: *pick a 3D polyline*

Select the other 3D polyline to join: *pick a 3D polyline that has a common endpoint with the first 3 segments added to the polyline.*

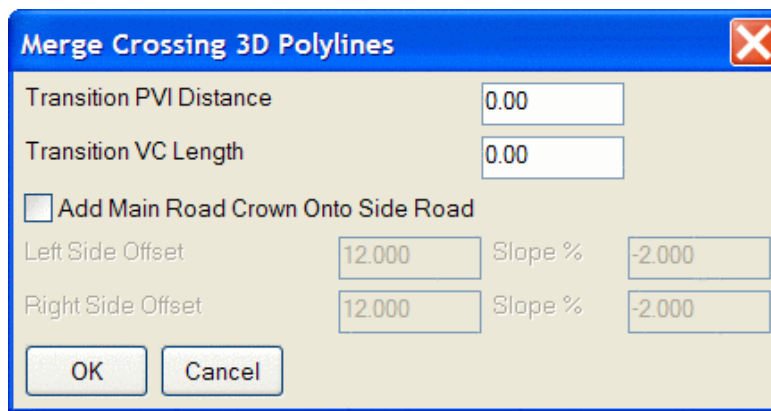
Pulldown Menu Location: 3D Data and Edit >> 3D Polyline Utilities

Keyboard Command: join3d

Prerequisite: 3D Polylines to use for selection

Merge Crossing 3D Polylines

This command works with 2 crossing 3D polylines, adding one or more vertices to one of them at the virtual point of intersection to match the elevation of the other. The 3D polyline that is vertically unchanged is referred to as the "Main 3D polyline", the 3D polyline that is edited is referred to as the "Side 3D polyline." The command uses the 2 vertices on the Main 3D polyline on either side of the virtual intersection to determine an interpolated elevation on the Main 3D polyline at the point of virtual intersection, and adds a vertex on the Main 3D polyline at that location with the calculated elevation, but the vertical characteristics of the Main 3D polyline are otherwise unchanged. The Side 3D polyline gets a new vertex at the virtual intersection with the same interpolated elevation, thereby changing it's vertical definition as much as necessary to match. The characteristics of the transition are controlled by the settings in the Merge Crossing 3D Polylines dialog box.



Transition PVI Distance: This option creates 2 additional vertices on the Side 3D polyline, each at the specified distance from the virtual intersection, and both with the same elevation as the vertex at the virtual intersection, essentially creating a flat section.

Transition VC length: This option creates a vertical curve for the transition, passing through the interpolated elevation at the virtual intersection. The start of the vertical curve is the specified value from the virtual intersection, as is the end, so the overall length of the entire vertical curve is actually twice the value specified in the dialog box.

Add Main Road Crown Onto Side Road: This option creates the transition by assuming the Main 3D polyline is a crowned roadway, and creates corresponding additional vertices on the Side 3D polyline.

Prompts

Select the Main 3D polyline: *pick the 3D polyline that will determine the crossing elevation, but will remain essentially unchanged*

Select the Side 3D polyline: *pick the 3D polyline that is to be changed to match the Main 3D polyline elevation at the virtual intersection*

Merge Crossing 3D Polylines dialog Adjust variables as desired in Merge Crossing 3D Polylines dialog box, pick OK.

Pulldown Menu Location: 3D Data

Keyboard Command: merge3d

Prerequisite: 2 crossing 3D polylines

3D Polyline by Slope on Surface

This command creates a 3D polyline at a user-specified slope. The user picks the starting point and then the polyline continues along the surface at the slope until it reaches a point where the maximum slope at the point is less than the design slope. The surface is defined by a grid or TIN file which must be created before running this routine. Applications for this command include designing haul roads or ditches.

Prompts

Enter the polyline layer <SLOPE_ROAD>: *press Enter*

Select the Grid File dialog

Reading row > 51

Extrapolate grid to full grid size (Yes/<No>)? *Y*

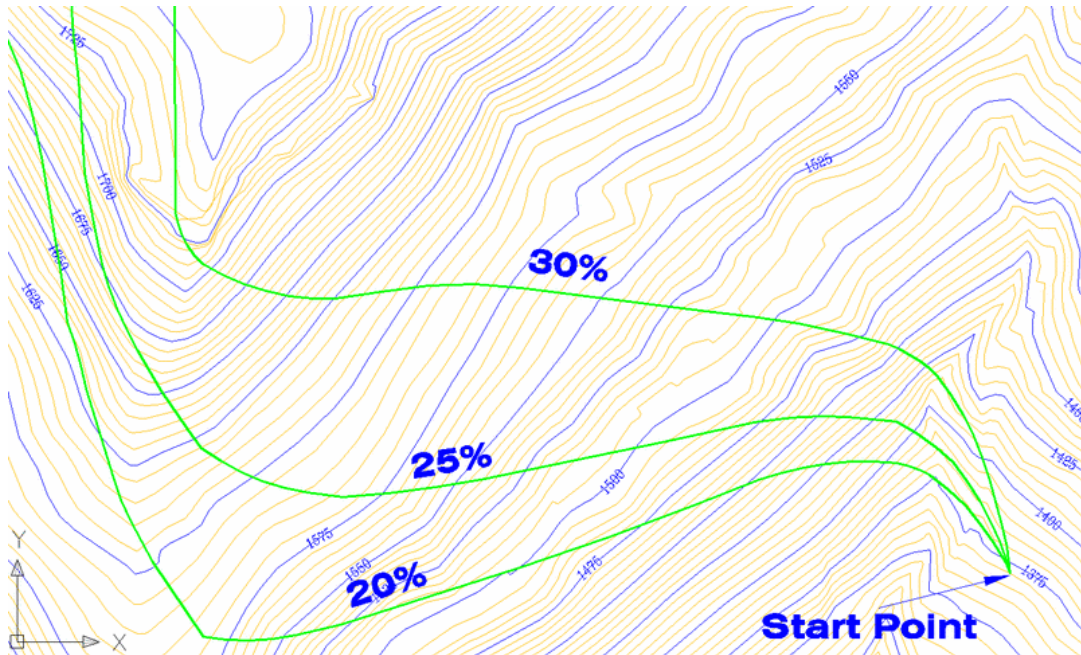
Limiting length for polyline (Enter for none):

Pick origin point of 3D polyline: *pick a starting point*

Direction of 3D Polyline (<Up>/<Down>)? *press Enter* The slope must go either uphill or downhill.

Direction of 3D Polyline facing up slope (<Left>/<Right>)? *R* Imagine facing uphill. Do you want the polyline to go to the left or right?

Enter the design slope: *10* This value is in percent slope.



Pulldown Menu Location: 3D Data

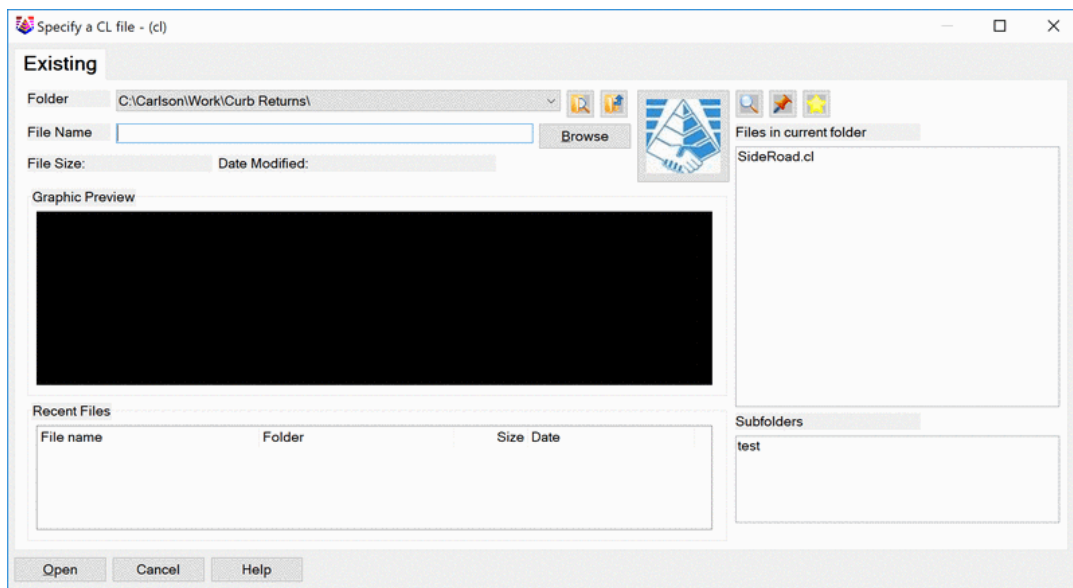
Keyboard Command: surfpl

Prerequisite: Existing surface file

Elevate Intersection Curb Return

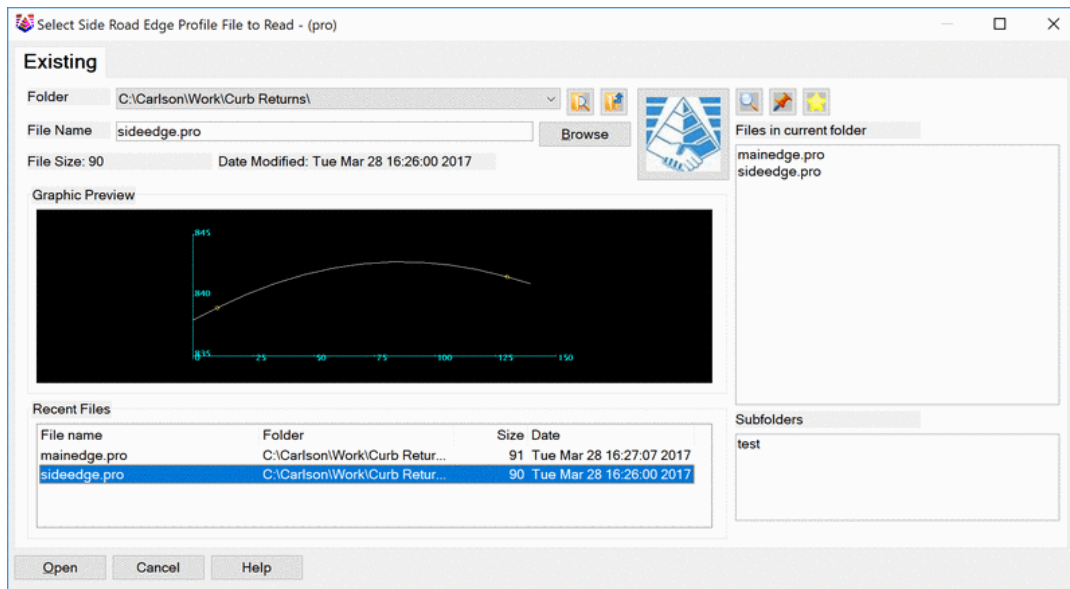
This command elevates an intersection curb return based on the distance along the side road in relationship to the curb return. Instead of an interpolation of the elevation this command will apply the interpolation from the side road to the curb return.

Side Road CL File/<Select polyline>: *select the side road's polyline or choose the centerline .CL file*



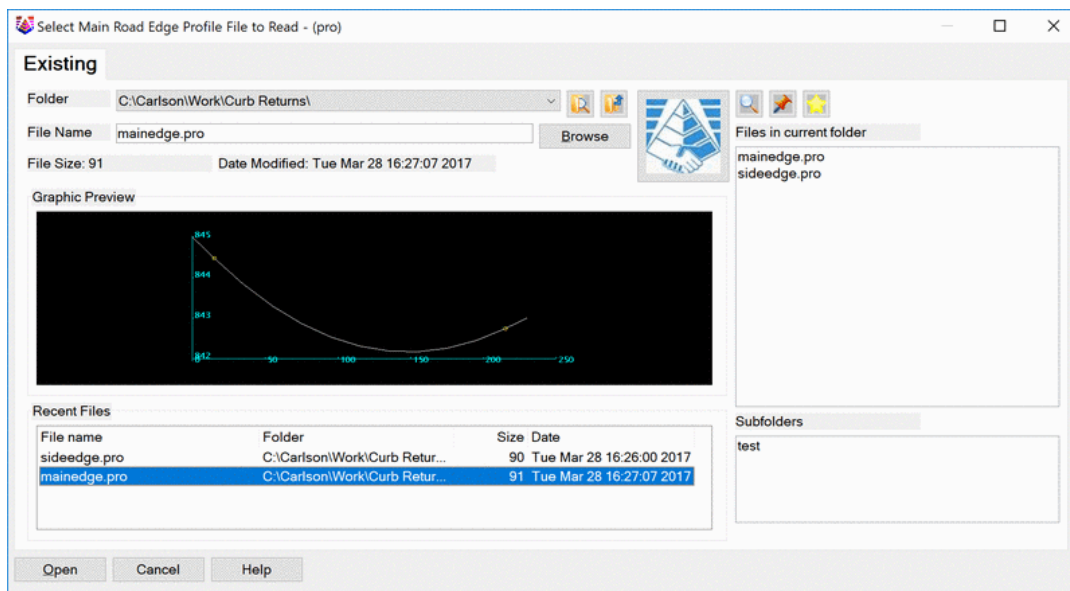
Select Side Road Edge polyline: *select the side road's edge polyline*

If the selection is not a 3D polyline, next you will be prompted to choose the Side Road Edge profile .PRO file

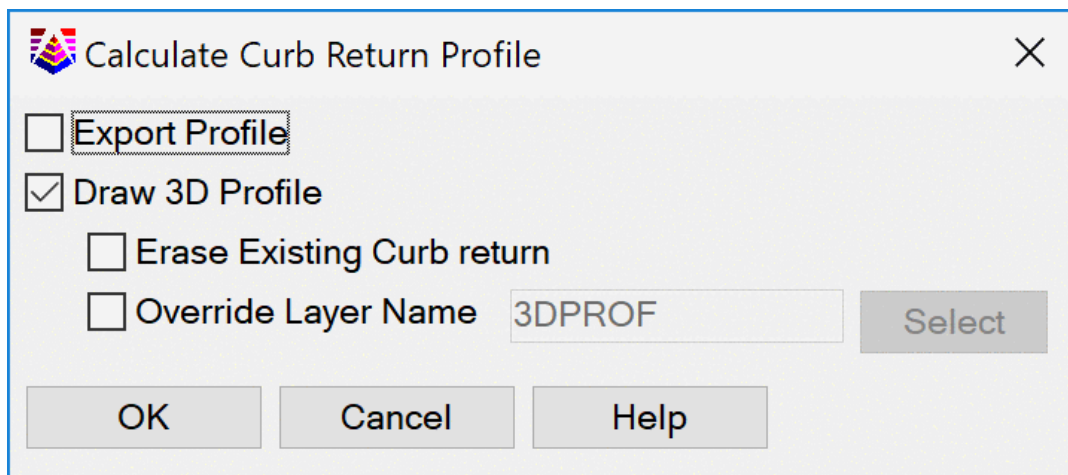


Select Main Road Edge polyline: *select the main road's edge polyline*

If the selection is not a 3D polyline, next you will be prompted to choose the Main Road Edge profile .PRO file

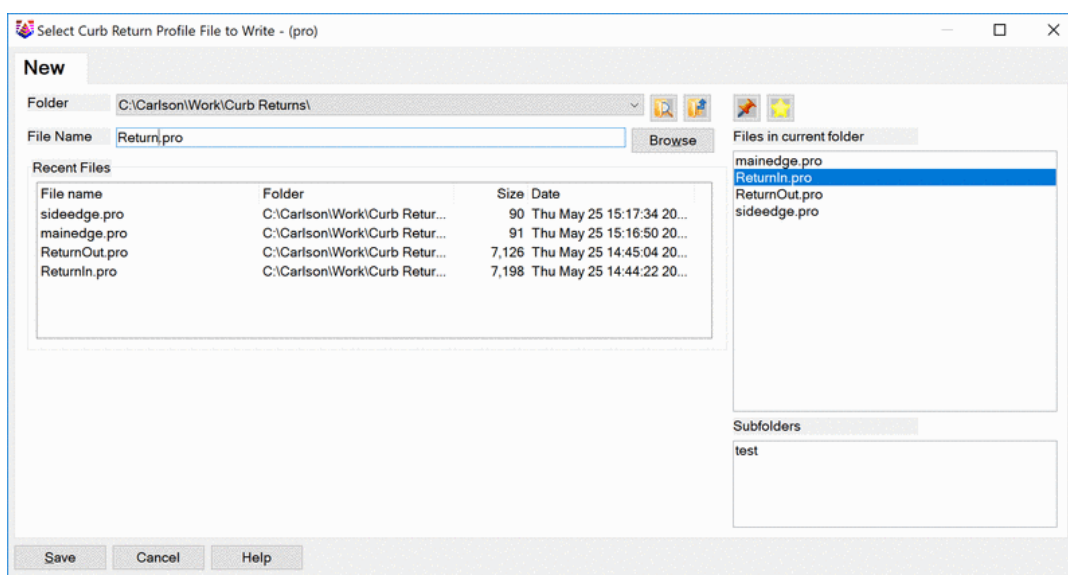


Select Curb Return polyline: *select the corresponding curb return edge polyline to create profile for*



Export Profile

Select the profile .PRO file to create for the curb return.



Draw 3D Profile

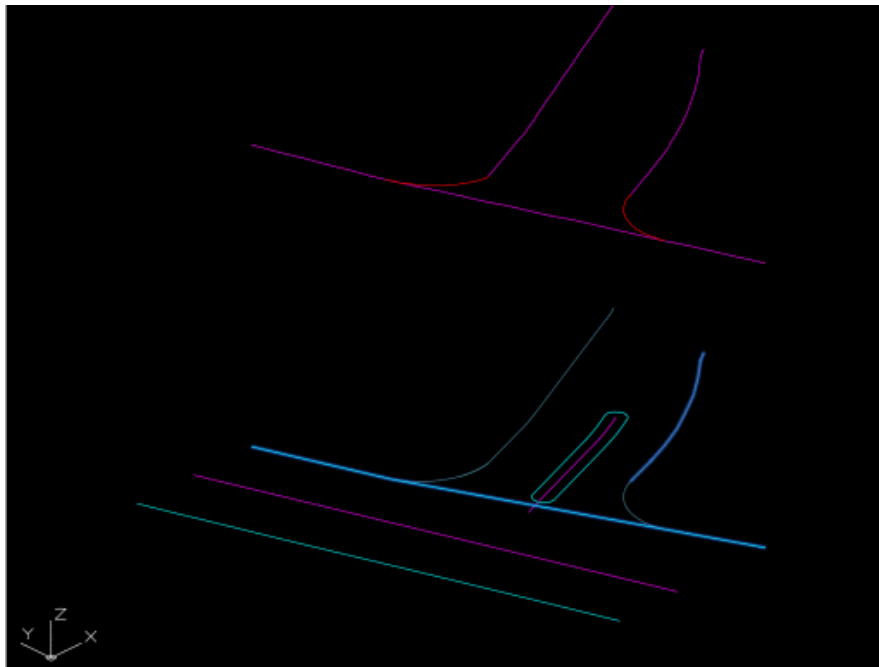
This will draw the profile as a 3D-Polyline on the original curb return's layer.

Erase Existing Curb Return

This will erase any already created 3D-Polyline drawn for the selected curb return.

Override Layer Name

This will override the original selected curb return's layer with the layer chosen.



Pull-down Menu Location: 3D Data - 3D Polyline Utilities

Keyboard Command: calcapron

Prerequisite: Existing profiles for Side and Main Road Edges

Min/Max Slopes 3D Polyline

This command checks 3D polylines to make sure slopes are within the specified range. For any segment that is outside the range, the program will set the elevation of the polyline vertex to put the segment within range. This routine could be used on a 3D polyline for a ditch to make sure the slope has a minimum grade. Also this routine applies to a 3D polyline for a road that has a maximum grade.

Prompts

All slopes up, down or either [Up/Down/<Either>]? U for up

Min slope percent: 1

Max slope percent: 9

Select polylines to process.

Select entities: *pick 3D polylines*

Changed 3 polylines.

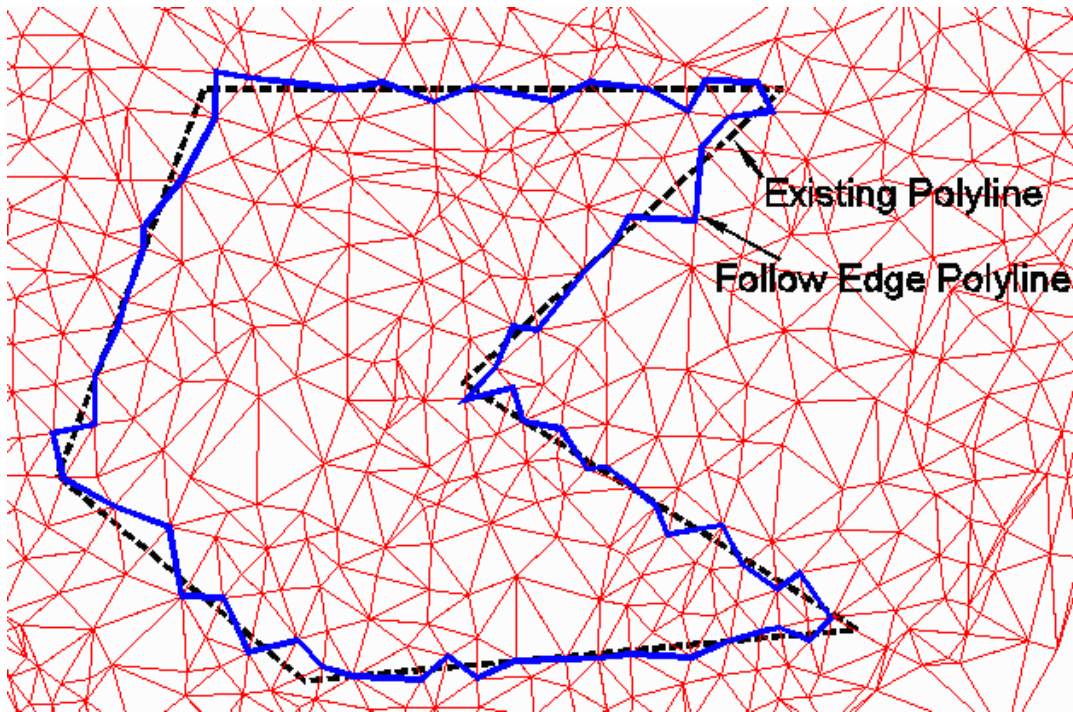
Pull-down Menu Location: 3D Data > 3D Polyline Utilities

Keyboard Command: minmax3dp

Prerequisite: 3D polyline

Follow TIN Edges

This command creates a polyline by connecting edges in a triangulation to approximately follow the path of an existing polyline. One application is to create an inclusion or exclusion perimeter for carving out a portion of a triangulation surface. By having the polyline follow the triangulation edges, the polyline can be applied to the triangulation without trimming edges or adding points. Otherwise, when the polyline doesn't follow the triangulation edges, the program will embed the polyline into the triangulation by draping the polyline onto the triangulation.



Prompts

Enter the polyline layer <INCLUSION>: *PERIMETER*
 Select TIN File *Select a triangulation file*
 Select boundary polyline to follow: *pick a polyline*
 Loading edges...
 Done.

Pulldown Menu Location: Edit > 3D Polyline Utilities
Keyboard Command: tinline
Prerequisite: a polyline

Extend To Elevation

This command extends the end segment of a 3D polyline or line until the segment reaches the specified elevation. The slope for the extension is user-specified and defaults to the existing segment slope.

Prompts

Select line or polyline to extend: *pick a 3D line or polyline*
 End point elevation: **496.130**
 New elevation: *495*
 Slope to extend <-1.482>: *press Enter*
 Select line or polyline to extend (Enter to end): *press Enter*

Pulldown Menu Location: 3D Data
Keyboard Command: extend2z
Prerequisite: 3D linework

Break 3D Polyline by Surface

This command breaks 3D polylines against a surface. The surface is defined by a 3D grid or triangulated surface model which can be selected from a .grd, .flt or .tin file. Alternately, the user can select 3D entities on the screen from which the program internally calculates a 3D grid. This routine is one step in 3D polyline design. In this example, a valley fill is designed using 3D polylines as follows:

1. Draw 3D Polyline - draw a 3D polyline at elevation 1450 across valley
2. Offset 3D Polyline - offset the 3D polyline at 2:1 slope and 10:1 for benches and top
3. Break 3D Polyline by Surface - trims the 3D polylines by the valley sides
4. Draw 3D Polyline - with OSNAP endpoint, draw 3D polyline perimeter around 3D polylines ends
5. Make 3D Grid File - create 3D grid file of fill surface
6. Plot 3D Grid - draw the 3D grid using 3D polyline perimeter as inclusion polyline
7. Viewpoint 3D - creates the last figure view

Prompts

Source of surface model (File/<Screen>)? *press Enter* The *File* option allows you to choose the .grd, .flt, or .tin file that models the site. Otherwise, a grid will be calculated by picking the grid location and selecting surface entities on screen (e.g., contour polylines). Using the *File* option can be quicker because the surface is already calculated. Also the .grd file can be drawn to preview the existing surface.

Pick Lower Left limit of surface area: *pick a point*

Pick Upper Right limit of surface area: *pick a point*

Make GRiD Setting Dialog *OK*

Select polylines to clip.

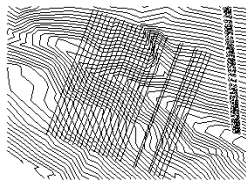
Select objects: *pick the 3D polylines*

Select surface entities.

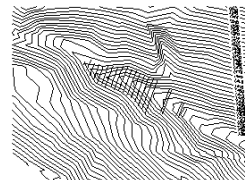
Select objects: *select objects that define the surface.*

Erase polyline below surface (<Yes>/No)? *press Enter* If you answer yes, the segments of the polylines below the surface will be erased from the intersection, if any, of the polyline with the surface. Otherwise the polylines will only be broken into separate polylines at the intersection.

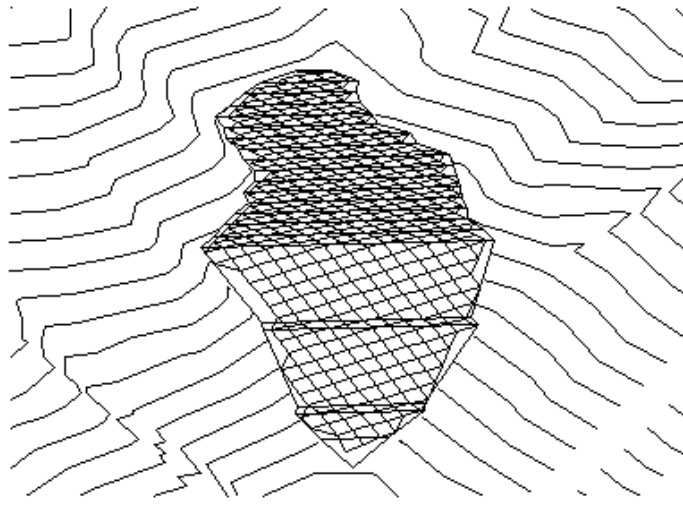
Specify layer names (Yes/<No>)? *press Enter* If you answer yes, you will have the option to place the broken polylines into different layers.



Before Break 3D Polyline by Surface



After Break 3D Polyline by Surface



3D view of fill with grid mesh

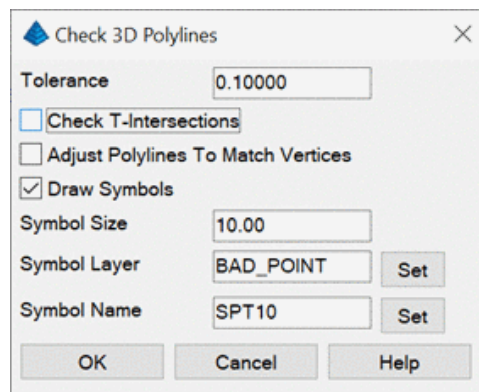
Pulldown Menu Location: 3D Data

Keyboard Command: surfbreak

Prerequisite: Plot the 3D Polylines to use for selection and entities that define a surface.

Check 3D Polylines Match Vertices

This command checks 3D polylines for vertices that match within the specified tolerance and for T-intersections. This command is a way to clean up 3D polylines to be used as breaklines in surface modeling. If the Triangulate & Contour routine reports a Data Problem Log with Vertical Edges or T-intersections, then this command can help clean up the polylines. This command can be used to identify the 3D polyline vertices within the tolerance. Besides creating a report of vertices, there is also an option to draw symbols at these vertices. Use the Adjust Polyline To Match Vertices to have the program modify the polyline vertices which will combine vertices within the tolerance into a single vertice.



Prompts

Check 3D Polylines dialog

Select 3D polylines to process.

Select objects: *select the polylines*

Check Report

Pulldown Menu Location: 3D Data > 3D Polyline Utilities

Keyboard Command: tin_pt_check

Prerequisite: 3D polylines

Add Points At Elevation

This command inserts vertices into a 3D Polyline at a specific elevation, or elevation interval, by interpolating between existing elevations in the polyline.

Prompts

Add single elevation or elevation interval [Single/<Interval>]? press Enter

Enter Elevation Interval: 50

Select 3D polylines to process. pick 3D polyline(s)

Select objects: 1 found

Select objects:

Processing polylines ...

Added 10 points to polylines.

Pulldown Menu Location: Edit > 3D Polyline Utilities

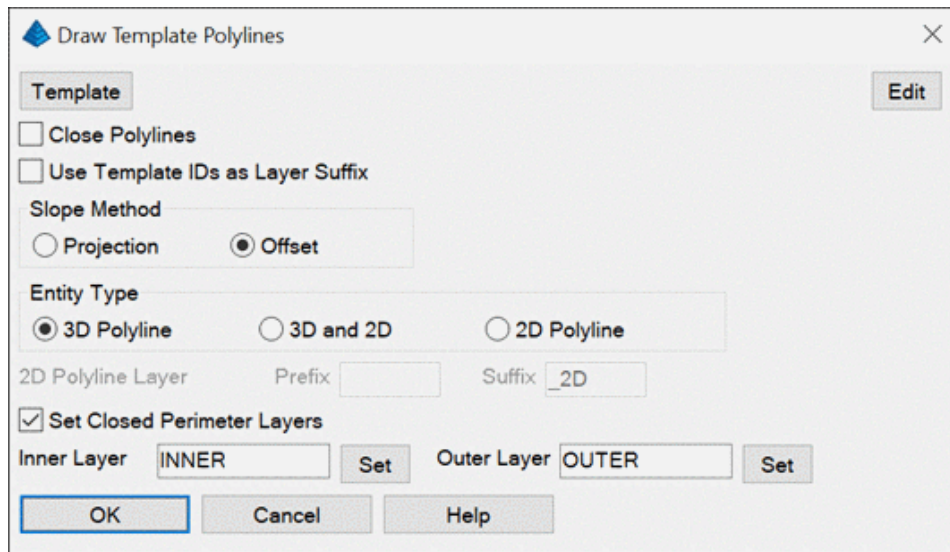
Keyboard Command: addplz

Prerequisite: 3D Polylines

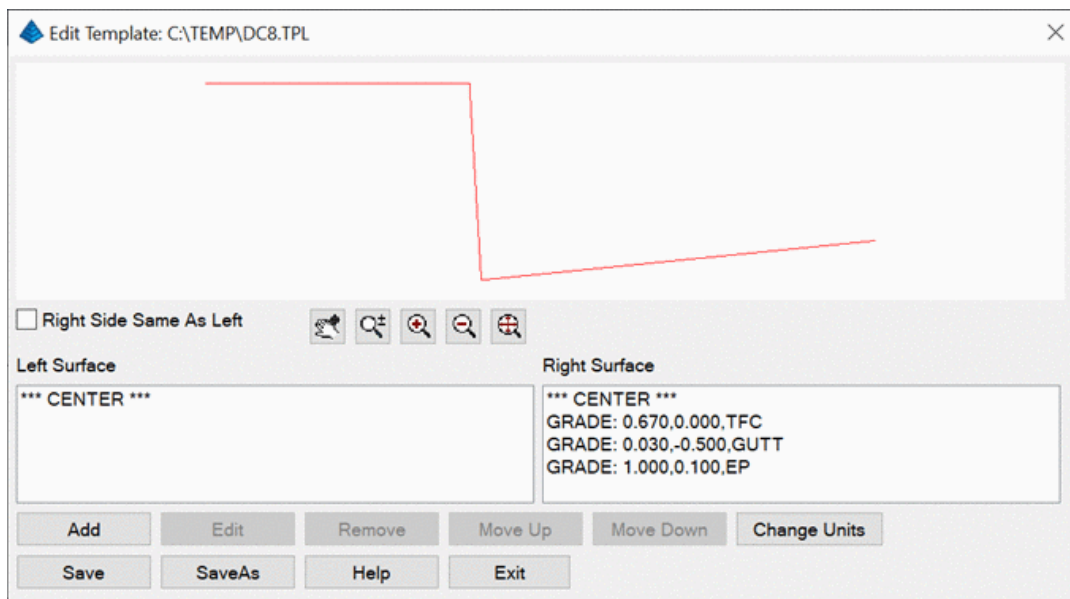
Draw Template Polylines

This command draws polylines that are offset parallel to an existing alignment polyline. A template file (TPL) from road design is used to define the offsets for the new polylines. This command can be used to turn 3D polylines into template polylines such as for curbs. The template 3D polylines can then be used for breaklines in surface modeling such as with the Triangulate & Contour routine.

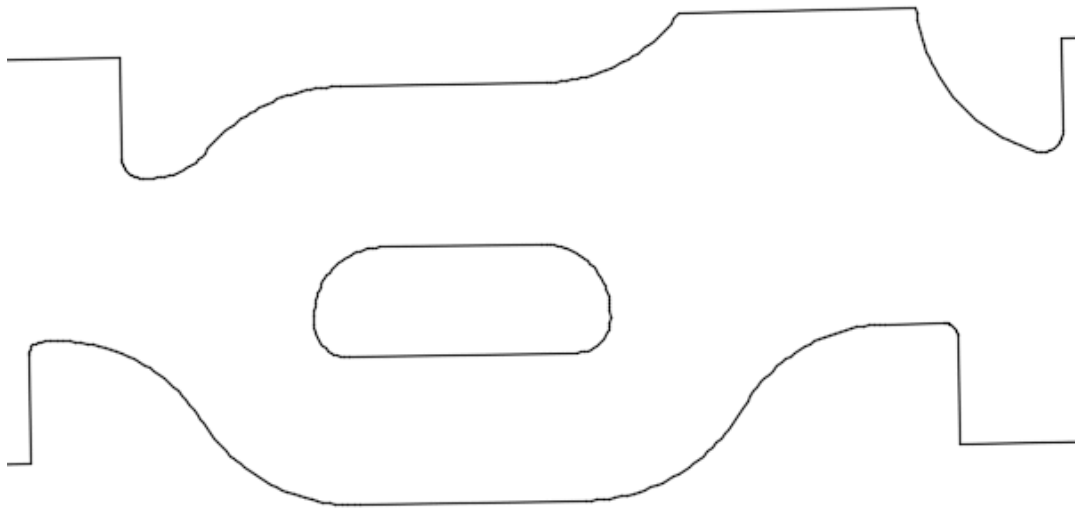
In the dialog, pick the Template button to select a TPL file. Use the Edit button to edit the grades in the template. Each grade in the template is used to create a new template polyline. The grade distance is used for the horizontal offset and the grade slope is used for the vertical offset. The layer of the existing alignment polyline is used as the base layer for the template polylines. The Close Polyline option applies when the input polyline is unclosed. This option will when create a closed polyline using the farthest left and right template polylines. The Use Template IDs as Layer Suffix option adds the template ID to the layer name for each template polyline. The template polylines can be drawn as 3D polylines or as 2D polylines as zero elevation. For 2D polylines, there are layer prefix and suffix settings for creating separate layers for the 2D polylines. The Densify 3D Polyline option creates additional vertices at the specified interval. The Set Closed Perimeter Layers option uses the specified layers for the inner and outer template polylines.



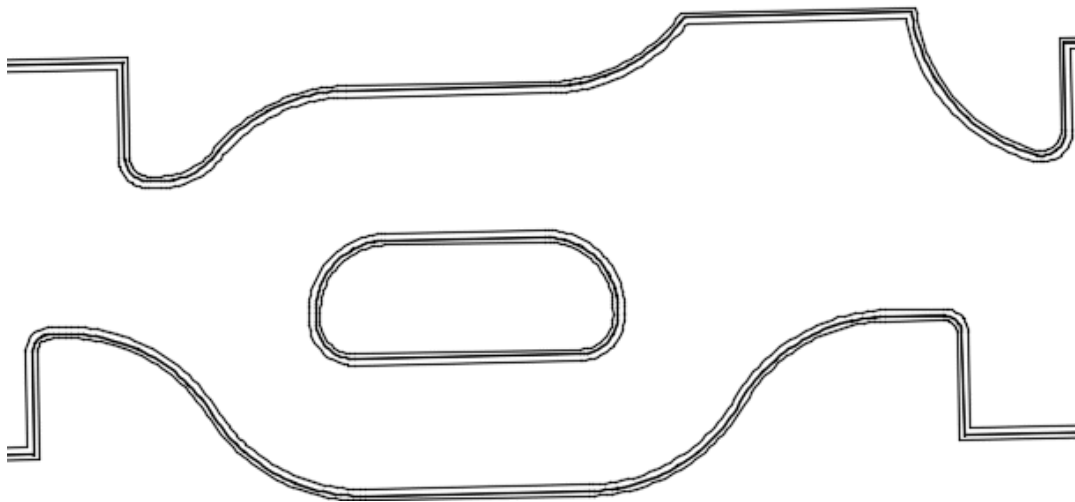
Main dialog



Edit Template dialog for a curb



Original alignment polylines



Template polylines created from curb template

Prompts

Draw Template Polylines dialog

Pick 3D polyline: *pick a polyline*

Pick side for template: *pick a polyline*

Pick 3D polyline (Enter to end): *press Enter*

Pulldown Menu Location: 3D Data > 3D Polyline Utilities

Keyboard Command: `tpl_3dp`

Prerequisite: 3D polyline

Draw Curb Ramp

This command modifies 3D curb polylines to fit in a ramp of the specified dimensions. For input, this command requires three 3D polylines that define the face of curb (bottom and top) and the back of curb. There is a dialog to enter the curb dimensions. For the **Pick Method**, the **Center** method places the ramp centered at the pick point. The **Sides** method prompts to pick two points on the curb polylines to define the left and right sides of the ramp. The **Driveway** method allows you to create multiple ramps at once by selecting closed polylines that represent driveways. These polylines must be closed polylines with two vertices on the curb polyline. The ramp is fit between these two vertices. The **Separate Pick Position Prompt** option prompts for the curb position point after selecting the curb polyline. Otherwise, the program uses the position that you select the curb polyline as the curb position. For the **Curb Method**, the **Curb Cut Only** modifies the three curb polylines and doesn't draw the ramp polylines. The **Elevate Driveway** option sets the elevation of the driveway polyline to match the curb polyline. For **Style**, the **Taper To End** option applies the **Front Taper** width to the back of the ramp as well as the front, the **Hold Fixed** option uses the full **Front Taper** width for the first ramp and then tapers, and the **Taper First Ramp** option applies the **Front Taper** only to the first ramp. The **Separate Curb Taper** is the distance the curb drop before the ramp. Leave this field blank to not use this curb taper. The **Direction Method** chooses between making the ramp perpendicular to the curb or having the program prompt for picking the ramp direction. The **Lip Method** controls the height between the bottom-of-curb and top-of-curb polylines at the bottom of the ramp. The **Ramp Slope** method sets the slope between these polylines to match the ramp slope and the **Height** method sets a fixed height. The **Slope Method** controls the ramp slope. The **Match Curb Height** method sets the slope so that the back of the ramp reaches the original height of the top-of-curb. The **Slope** method sets the ramp slope to the specified value. The **Second Ramp** and **Third Ramp** allow for additional ramps such as for a landing pad.

After the dialog, the program prompts to pick the curb 3D polylines to draw the ramp. The existing 3D curb polylines are modified to lower the curb for the ramp and new 3D polylines are drawn for the rest of the ramp. Also, the program updates other 3D linework that crosses the ramp such as sidewalks. These 3D polylines can then be used for breaklines in surface modeling such as with the Triangulate & Contour routine.

Curb Ramp

Pick Method
 Center Sides Driveway

Separate Pick Position Prompt

Curb Method
 Curb Cut Only Curb Cut and Ramp

Width Elevate Driveway

Depth

Front Taper Style

Separate Curb Taper

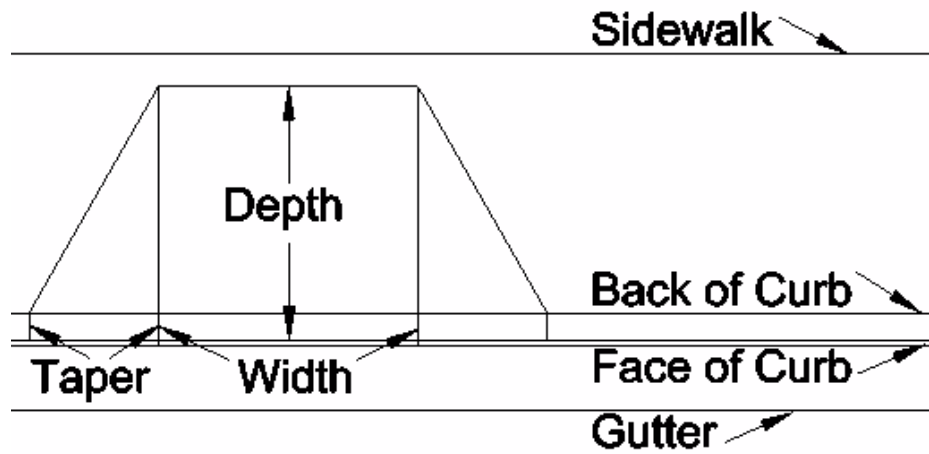
Direction Method
 Perpendicular Pick

Lip Method Ramp Slope Height Height

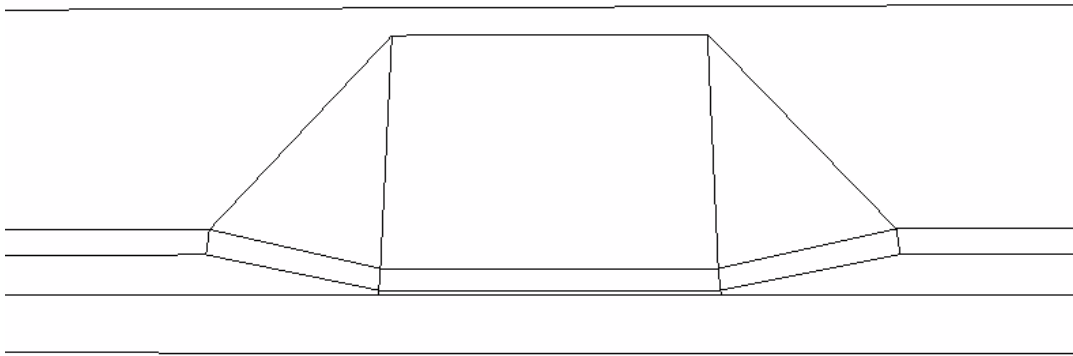
Slope Method Match Curb Height Slope Slope

Second Ramp
Depth Slope

Third Ramp
Depth Slope



Plan view of 3D curb polylines with ramp dimensions



3D view of curb polylines and ramp

Prompts

Curb Ramp dialog

Select curb polyline: *pick a polyline*

Select curb polyline (Enter to end): *press Enter*

Pulldown Menu Location: 3D Data > 3D Polyline Utilities

Keyboard Command: curb_ramp

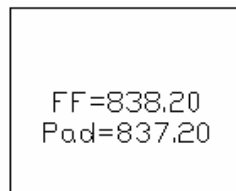
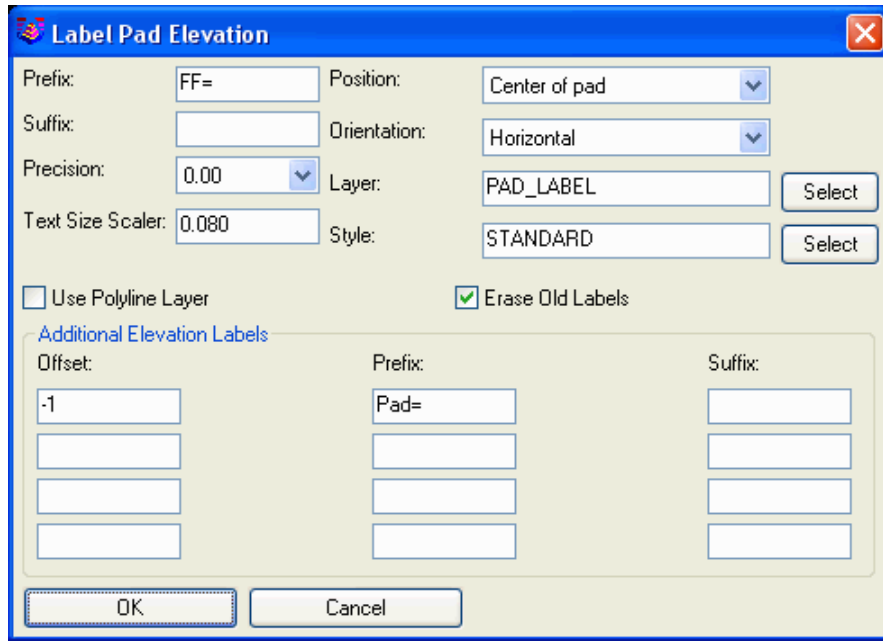
Prerequisite: 3D curb polylines

Label Pad Elevation

This command creates elevation labels inside closed polylines such as building pad polylines. The elevation value for the label comes from the elevation of the polyline. In the dialog, there are settings to control the label prefix, suffix, decimal place precision, text size position, orientation, layer and style for the labels. For **Position**, the Center of pad option creates the labels in the center of the polyline and the Edge of pad creates the label along the top segment of the polyline and draws a leader from the label to the polyline. For **Orientation**, the Horizontal option creates the label horizontal to the current twist screen and the Align With Pad option rotates the label to align with

the longest segment of the polyline. The **Use Polyline Layer** option will use the layer of the polyline for the label layer. The **Erase Old Labels** option erases previous labels when labeling the same polyline another time. The **Draw on Real Z Axis** creates the labels at the pad elevation instead of zero elevation. The **Additional Elevation Labels** are optional labels that are a fixed vertical offset from the polyline elevation. For example, you can use this option when the pad polyline elevation represents the finished floor elevation and you also want to label the basement elevation that is at a fixed offset.

If the Link Labels With Linework option is on in Carlson Configure > General Settings when the labels are created, then the labels will automatically update when the elevations of the polylines are changed.



Pulldown Menu Location: 3D Data > Label Polylines

Keyboard Command: labelpad

Prerequisite: Closed polyline with assigned elevation

Update Pad Elevation Labels

This command reads all previously created Pad Elevations, and updates their values if their associated Pad Polyline elevations have been changed.

Pulldown Menu Location: 3D Data >> Label Polylines

Keyboard Command: update_labelpad

Prerequisite: Previously created pad elevation(s)

Label Elevations Along Polyline

This command labels point elevations and aligns with a polyline based on settings shown in the dialog. These settings can be divided into five groups.

Label Settings: The **Source of Elevations** are read from Carlson points drawn on the screen, polyline vertices, elevations of grade break vertices and can also be picked on the screen. The **Side for Labels** is relative to the direction the polyline is drawn. Labels can be **aligned** horizontally, parallel or perpendicular to the polyline or according to the picked alignment. The **Offset distance scalar** offsets the label from the actual point.

Text Settings: The labels will be drawn on **Layer** with selected **Style**. The **Text size scalar** is relative to the current horizontal scale, which is set in *Drawing Setup*. These scalars are multiplied by the horizontal scale to obtain the actual drawing units. The number of **Integers** and **Decimals** can also be specified along with **Prefix** and **Suffix** for the main elevation label.

Leader Settings: The Leader Settings are used to **Draw Leader** with **Arrowhead** on the leader **Layer** with length of leader equal to **Leader Scaler**. The option **Draw text above leader** extends the leader tick to the length of the label.

Additional Settings: **Draw box around label** draws box around the elevation label. **Flip text for twist screen** changes the text direction if the text is drawn upside down. If the option **Ignore zero elevation** is on zero elevation labels will be ignored. The Carlson points or picked points are beyond **Maximum offset to use** will be ignored.

Additional Offset Settings: If the **Additional offset** is other than 0, it will be labeled with **Prefix** and **Suffix** using the other text settings on the next line of main elevation label.

The overlapping labels can be moved using Move Elevation Labels command to remove the overlap.

If the Link Labels With Linework option is on in Carlson Configure > General Settings when the labels are created, then the labels will automatically update when the elevations of the polylines are changed.

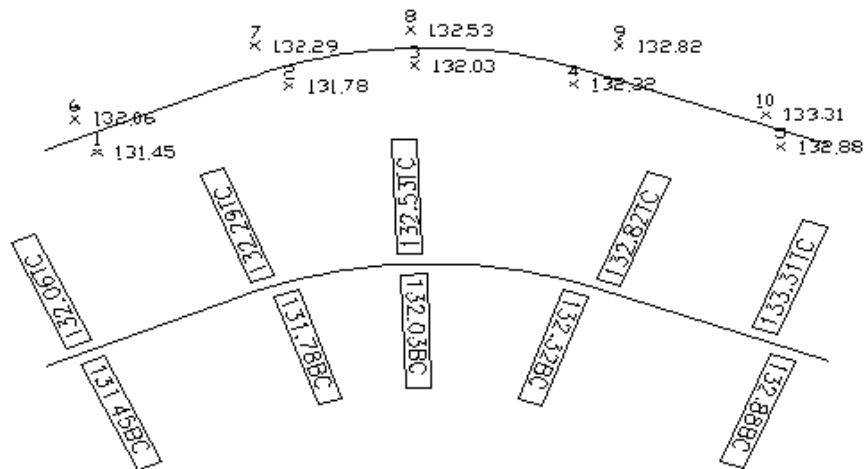
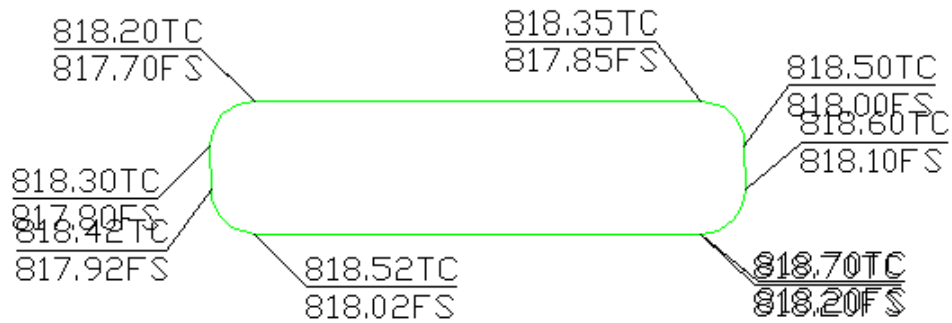
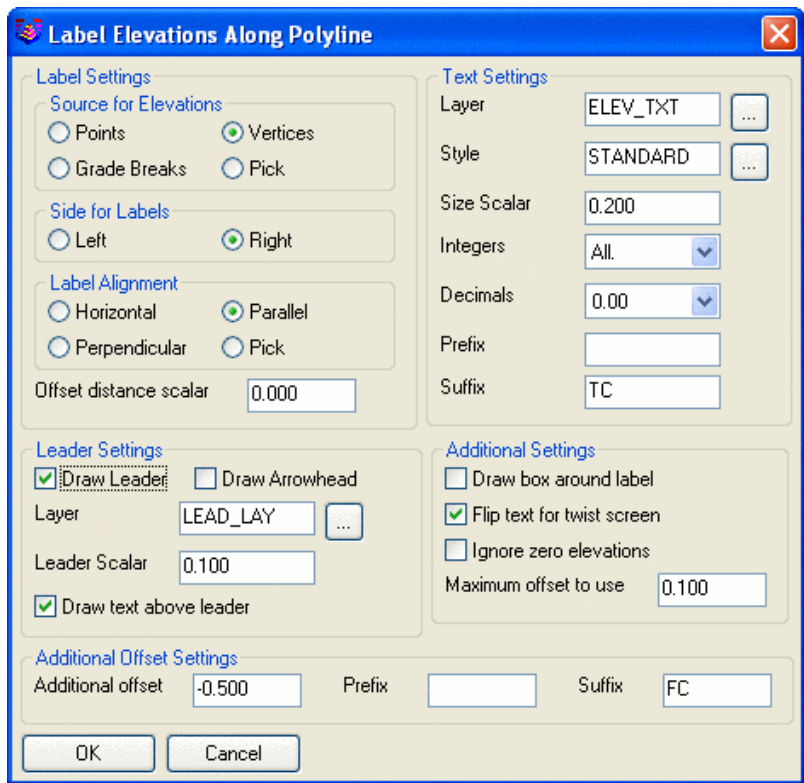
Prompts

Label Elevations Along Polyline dialog

Select alignment polyline: *pick a polyline*

Select points to label.

Select objects: *pick the points*



The alignment polyline with points to label is shown

Pulldown Menu Location: 3D Data->Label Polylines

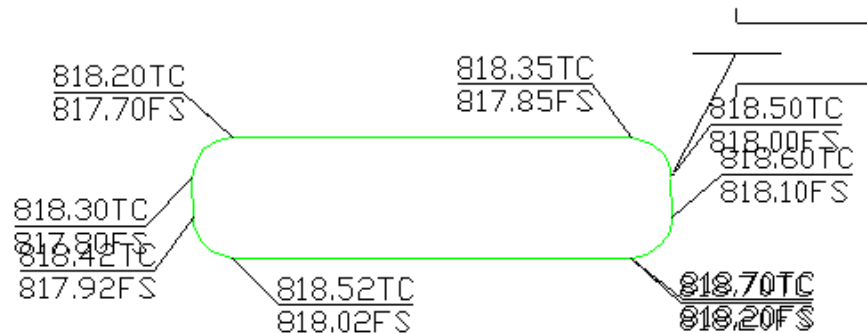
Keyboard Command: elevlab
Prerequisite: Polyline and points

Move Elevation Labels

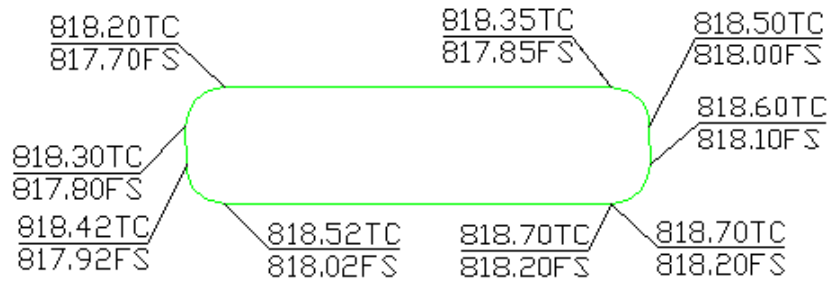
This command moves the selected elevation labels drawn using Label Elevations Along Polyline command with a leader. The purpose is to clean up label overlaps. To move a label, pick any one of the elevation labels text and the program will pick up all the other associated labels. Then pick the new location and while the pointer is moved, the program shows an outline of the label area. The program remembers the moved locations for each label so that when the elevation labels are redrawn, the moved locations are retained. The Restore function puts the labels back to their default position. The following graphics show the elevation labels before and after Move Elevation Labels was used to clean up the label overlaps.



Elevation Labels with Overlap



Move a label away



Elevation Labels without Overlap

Prompts

Select elevation label to move (R for Restore): *pick a elevation label text with leader*

Pick label position: *pick a point*

Select elevation label to move (R for Restore): *press Enter to end*

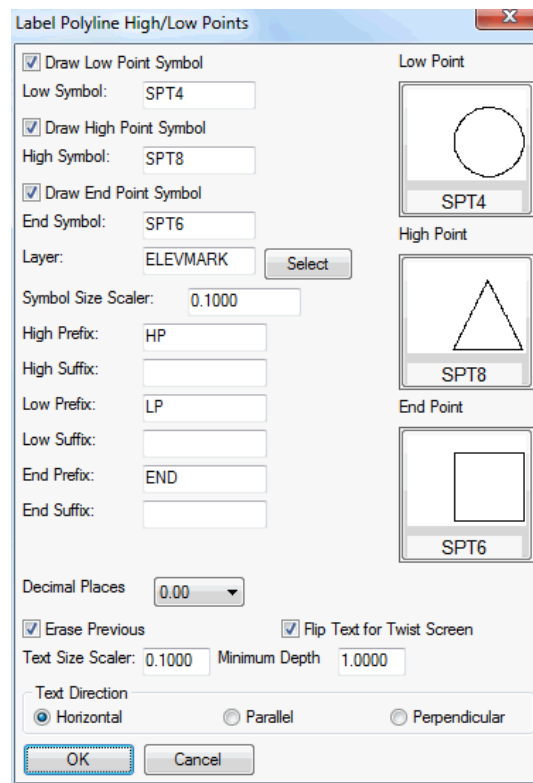
Pulldown Menu Location: 3D Data > Label Polylines

Keyboard Command: move_elevlab

Prerequisite: Elevation labels drawn using Label Elevations Along Polyline command

Label Polyline High/Low Points

This command finds and labels the high, low and end points of selected 3D Polylines. This is useful for drainage studies, finding low spots for placement of culverts or inlets, checking overhead clearances, etc. The **Erase Previous** option will erase labels created by this routine when running again on the same polyline. The **Flip Text for Twist Screen** option applies to text labels that would be upside-down in the current view depending on the polyline orientation. When a polyline has elevations that go up and down, multiple high and low points can be labeled for the different local highs and lows. The **Minimum Depth** setting controls how much elevation difference is needed to label a local high or low point.



Prompts

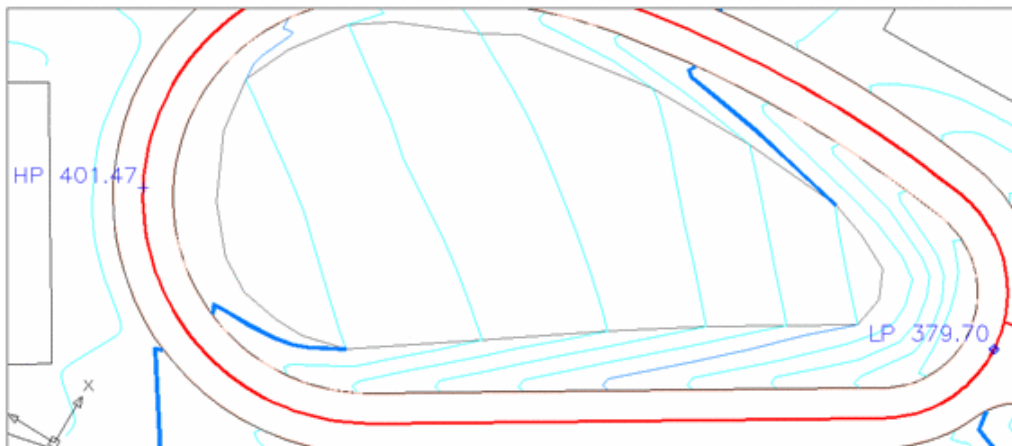
Label Polyline High/Low Points dialog

Select 3D Polylines.

Select objects: *select one or more 3D Polylines*

Processing polylines ...

Done.



Pulldown Menu Location: 3D Data >> Label Polylines

Keyboard Command: elevpl

Prerequisite: 3D polyline(s)

Label Polyline Segment

This command labels the distance and slope of 3D polyline segments, or between two picked points, in plan view.

In the dialog, choose the selection method; either the Entire polyline, Polyline segment, or Any two points. Then choose the location and/or visibility for the Horizontal distance, Slope distance, Slope %, and Slope ratio, enter any desired Prefixes and/or Suffixes, and the desired decimal Precision for each. Choose whether to Draw Slope Leaders, which are arrows pointing in the direction of the slope. The Slope Direction controls whether to draw the slope leaders in the direction of the polyline, always pointing uphill or always pointing downhill. Leader Position controls whether to draw the slope leaders alongside the slope label or above the label. Choose whether to Erase old annotations, and set the Text size scaler, Text offset scaler, and other related variables. Scalers are multiplied by the drawing scale to determine the actual sizes for the specified objects. Choose the Annotation layer from the drop list. Pick OK, and you are prompted to select the polyline or segment or two points to annotate.

	Location	Prefix	Suffix	Precision
Hz Distance	No annotation			0.00
Slope Distance	No annotation			0.00
Slope %	No annotation			0.00
Slope Ratio	No annotation			0.00

Text Size Scaler: 0.100 Label Layer: POLYLINE_LABEL Select

Text Offset Scaler: 0.025 Label Style: STANDARD Select

Leader Length Scaler: 0.100 Selection Method: Entire Polyline

Leader Offset Scaler: 0.025 Slope Direction: Polyline Direction

Arrow Size Scaler: 0.050

Draw Slope Leaders

Leader Position: Next to Label

Erase Old Annotations

OK Cancel



Prompts

Set up variables as desired in Annotate polyline dialog box, pick OK. Depending on your Selection method, the prompt will either read:

Select a polyline segment to annotate: *pick the segment*

or

Select a polyline to annotate: *pick the polyline*

or

Pick first point: *_nea to* For snap on.

Pick second point: *_nea to*

Press enter to return to the Annotate polyline dialog box, in the dialog box, pick Cancel to end.

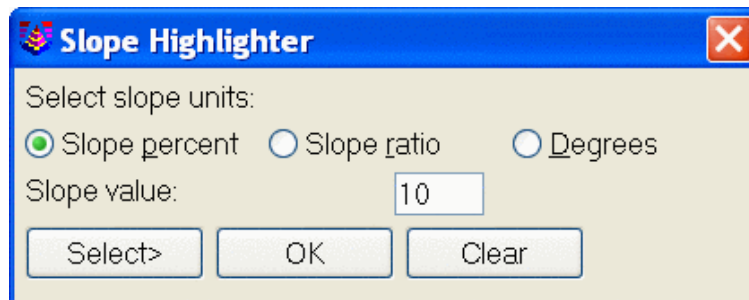
Pulldown Menu Location: Annotate

Keyboard Command: label3dp

Prerequisite: A 3D polyline

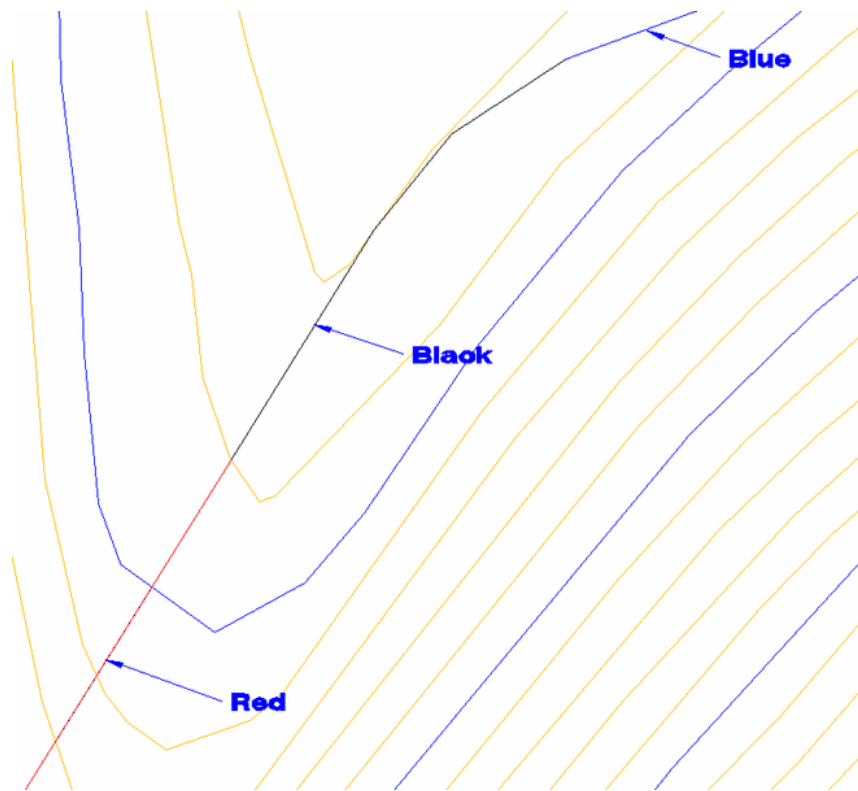
Highlight Segments by Slope

This routine highlights segments of a 3D polyline with slopes greater than the entered slope value. The RED segments are uphill, BLUE is downhill and BLACK/WHITE is for slopes less than the target slope. The slope can be entered as percent, ratio or degree. Choose Select to select the polyline and it is colored by slope.



Prompt

Select a polyline to highlight: *Select the 3D pline*



Pulldown Menu Location: 3D Data >> Highlight 3D Polylines

Keyboard Command: hlslope3dp

Prerequisite: 3D polylines

Highlight Crossing Breaklines

Breaklines are lines or polylines that are used in surface modeling to represent a feature such as a ridge, stream or curb. Breaklines force triangulation and interpolation to occur between the vertices of the breakline, which prevents any other triangulation line from crossing the breakline. It is important to avoid crossing breaklines in the surface model because the program cannot force triangulation along both since holding one will cross the other. Also the program cannot hold the interpolation along both because the elevation at the intersection point can be different for the two breaklines. For example, consider a breakline going from 101 to 105 and another going from 107 to 109. If these lines had an intersection at the midpoint, the elevation for one would be 103 and the other 108. This is an error that needs to be fixed by the user because those two breaklines are holding different elevations at the same point.

This command checks for intersections between the selected breaklines and then identifies any crossing breaklines by highlighting them. You can then edit these crossing breaklines before doing surface modeling such as Make 3D Grid or Triangulate & Contour.

Prompts

Ignore zero elevations [Yes/No]?

Reading points ...

Select surface entities to check.

Select objects: *select polylines and lines*

2915 found

Select objects: *press Enter to conclude selection*

Reading points ... 4442

122 crossing breaklines are highlighted.

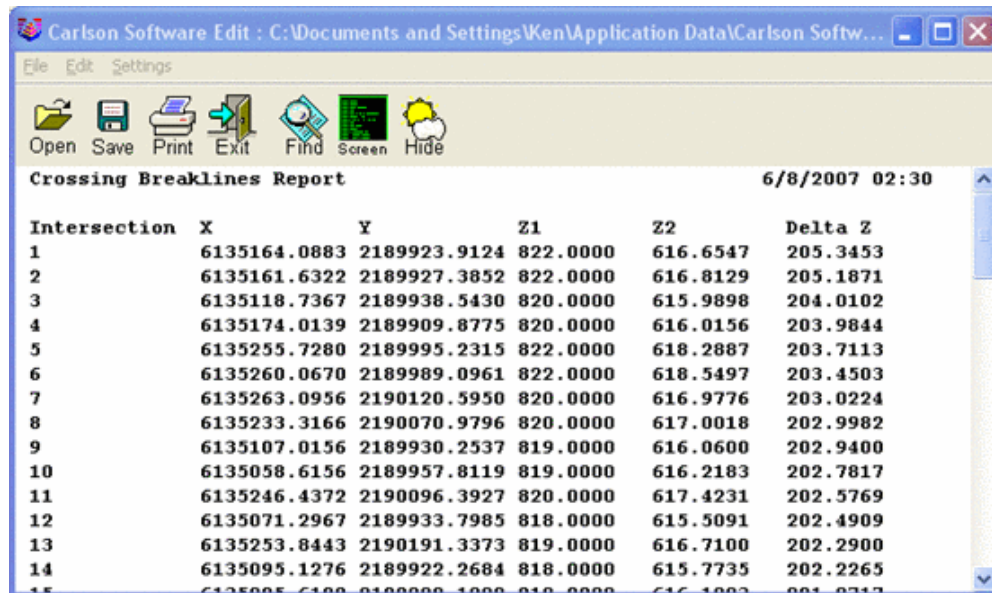
Use Report Formatter [Yes/No]? *press Enter*

Minimum delta Z to report <0.0>: *press Enter*

Add polyline vertices at intersections [Yes/No]? *press Enter*

or

Found no crossing breaklines if there are none.



Intersection	X	Y	Z1	Z2	Delta Z
1	6135164.0883	2189923.9124	822.0000	616.6547	205.3453
2	6135161.6322	2189927.3852	822.0000	616.8129	205.1871
3	6135118.7367	2189938.5430	820.0000	615.9898	204.0102
4	6135174.0139	2189909.8775	820.0000	616.0156	203.9844
5	6135255.7280	2189995.2315	822.0000	618.2887	203.7113
6	6135260.0670	2189989.0961	822.0000	618.5497	203.4503
7	6135263.0956	2190120.5950	820.0000	616.9776	203.0224
8	6135233.3166	2190070.9796	820.0000	617.0018	202.9982
9	6135107.0156	2189930.2537	819.0000	616.0600	202.9400
10	6135058.6156	2189957.8119	819.0000	616.2183	202.7817
11	6135246.4372	2190096.3927	820.0000	617.4231	202.5769
12	6135071.2967	2189933.7985	818.0000	615.5091	202.4909
13	6135253.8443	2190191.3373	819.0000	616.7100	202.2900
14	6135095.1276	2189922.2684	818.0000	615.7735	202.2265

Pulldown Menu Location: 3D Data >> Highlight 3D Polyline

Keyboard Command: xbar

Prerequisite: Polyline

Report 3D Polyline Station/Elevation

This command reports station or elevation information on a 3D polyline. If the station information is entered the program will return the elevation at the entered station. If the elevations are entered the program will determine and report the station at which the entered elevation occurred. If the elevation entered occurs at more than one location along the 3D polyline, all occurrences of the elevation are reported.

A prompt is provided allowing you to designate a starting station, or accept the default value of <0.0>. All entry and values are recorded and are displayed in the Carlson Standard Report Viewer upon completion.

Prompts

Select 3D polyline to report: pick a polyline

Starting Station <0.0>: press Enter

Enter elevation or station (<Elevation>/Station)? S

Enter Station to calculate elevation: 100

Station: 1+00.000 **Elevation:** 1052.262

Enter Station to calculate elevation (Enter to end): press Enter

Select 3D polyline to report: pick a polyline **Starting Station <0.0>:** press Enter

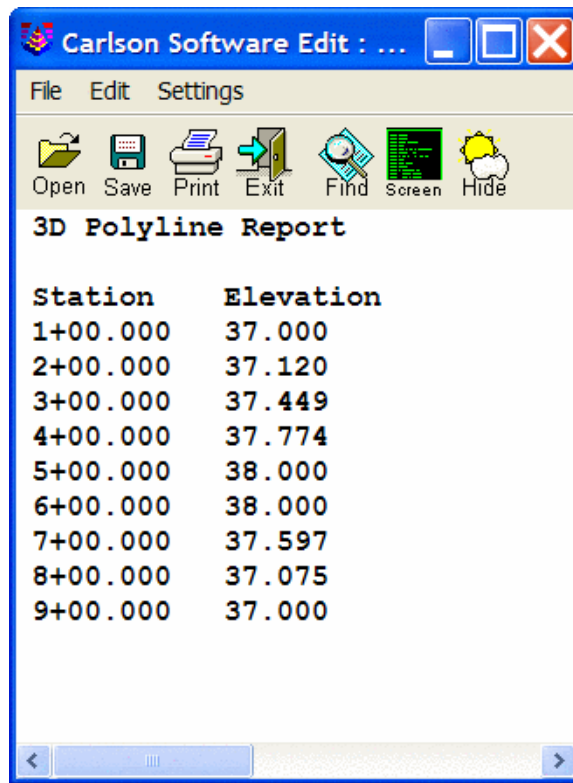
Enter elevation or station (<Elevation>/Station)? E

Enter Elevation to find stations: 2140

Enter Elevation to find stations: 2144

Enter Elevation to find stations: 2150

Enter Elevation to find stations (Enter to end): press Enter



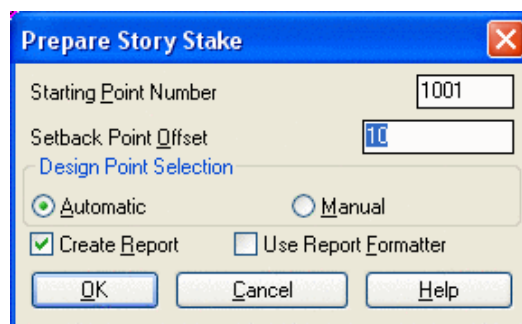
Pulldown Menu Location: 3D Data

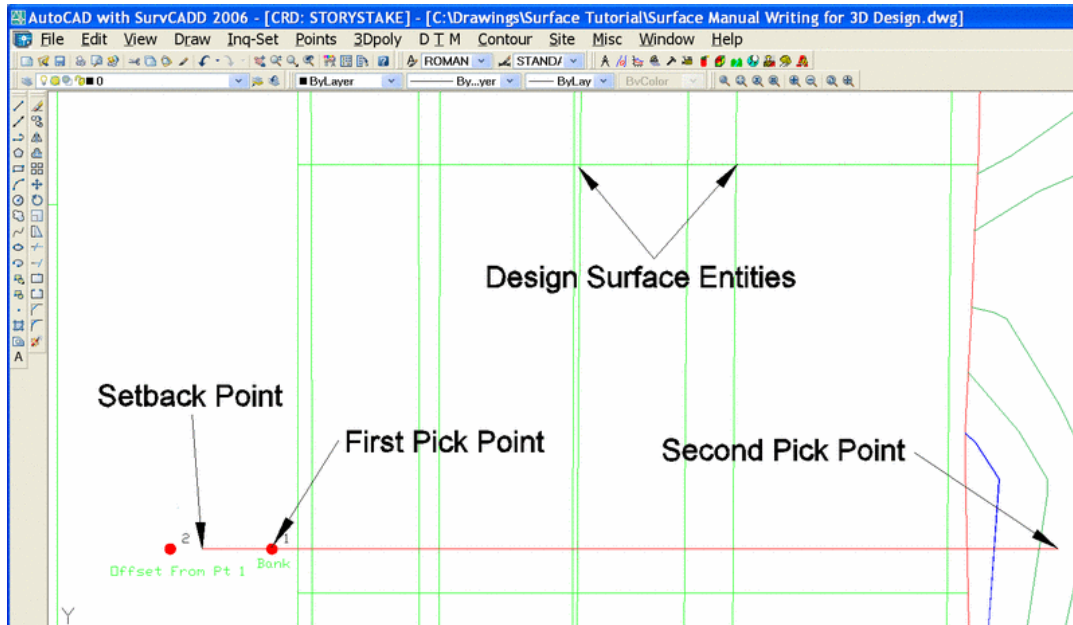
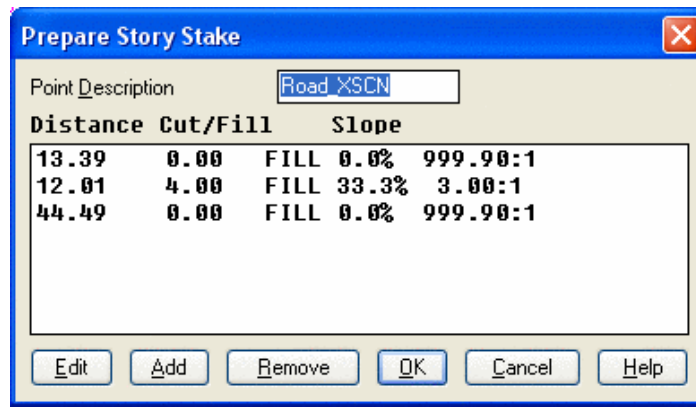
Keyboard Command: plreport3

Prerequisite: A 3D Polyline

Story Stake from Surface Entities

This command creates points with cut/fill information stored in the note fields for the points. Beginning at a point and facing a specified direction, the cut/fill information describes a design surface that is defined by contours and 3D polylines in the drawing. The program prompts you to pick the starting point followed by a direction point. Then the intersections for all the contours and 3D polylines between these two points are calculated and the resulting horizontal distances and slopes are shown in a dialog. In this dialog, you can edit, add or remove these slopes descriptions. The Point Description can also be specified. When OK is clicked, a point in the coordinate file is created at the starting point with this information stored in the note file. An offset point is also created at the specified offset distance back from the starting point. At the end of Story Stake from Surface Entities, a report of all the created points and the corresponding cut/fill data is shown if the Create Report option was set. Story Stake from Surface Entities does not draw the points in the drawing. These points can be drawn using the Draw-Locate Points command.





Prompts

Pick starting point: *pick the first point*

Pick direction point: *pick the second point to determine the direction*

Pick next starting point (Enter to end): *Enter if done*

Pulldown Menu Location: 3D Data >> Story Stake

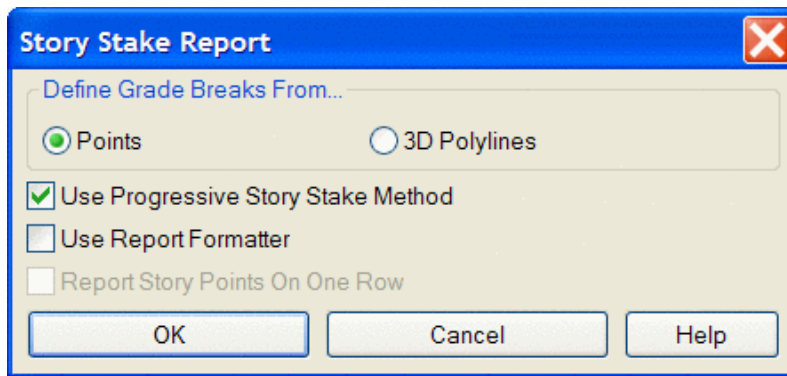
Keyboard Command: prepare_story

Prerequisite: Screen entities such as 3D polylines and contours

Story Stake By Points/Polyline

By Points

This option creates a report of cut/fill slopes and distances of a design surface from point to point. First you select the starting point. This starting position is shown as point 100 in the drawing below. Next you enter the subsequent point numbers to get the direction. The resulting horizontal distances and slopes are shown in a report dialog.



If the use Progressive Story Stake Method is turned off, the Cut/Fill, Distance and Slope is calculated from the first point to each point. The result is shown below. If it is turned ON, then it is point to point.

SurvCADD Edit : C:\scad2006_2006\USER\scadrprt.tmp

File Edit Settings

Open Save Print Exit Find Screen Hide

5/4/2005 13:44

Story Stake Report

Point#	Elev	Description	Cut/Fill	Distance	Slope
100	4612.000		Fill 18.00	13.236	136.0% 0.74:1
101	4630.000		Fill 0.00	100.000	0.0% 999.99:1
102	4630.000		Fill 19.00	18.365	103.5% 0.97:1
103	4649.000		Fill 0.00	125.000	0.0% 999.99:1
104	4649.000		Fill 4.00	3.502	114.2% 0.88:1
105	4653.000		Fill 0.00	100.000	0.0% 999.99:1
106	4653.000		Fill 45.00	45.000	100.0% 1.00:1
107	4698.000		Fill 0.00	200.000	0.0% 999.99:1
108	4698.000		Fill 25.00	25.008	100.0% 1.00:1
109	4723.000				

SurvCADD Edit : C:\scad2006_2006\USER\scadrprt.tmp

File Edit Settings

Open Save Print Exit Find Screen Hide

5/4/2005 13:52

Story Stake Report

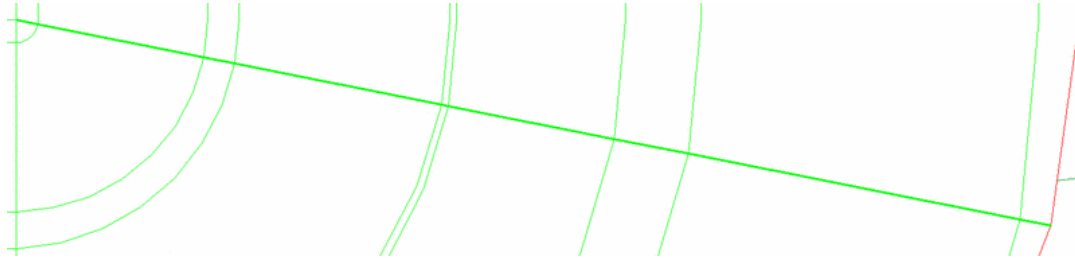
Start Point: 100 Elev: 4612.000 Desc:

Point#	Elev	Description	Cut/Fill	Distance	Slope
101	4630.000		Fill 18.00	13.236	136.0% 0.74:1
102	4630.000		Fill 18.00	113.236	15.9% 6.29:1
103	4649.000		Fill 37.00	131.601	28.1% 3.56:1
104	4649.000		Fill 37.00	256.601	14.4% 6.94:1
105	4653.000		Fill 41.00	260.103	15.8% 6.34:1
106	4653.000		Fill 41.00	360.103	11.4% 8.78:1
107	4698.000		Fill 86.00	405.103	21.2% 4.71:1
108	4698.000		Fill 86.00	605.103	14.2% 7.04:1
109	4723.000		Fill 111.00	630.111	17.6% 5.68:1

By Polyline

This command creates a report of cut/fill slopes and distances of a 3D polyline across a design surface. First you select the 3D polyline. The resulting horizontal distances and slopes are shown in a report dialog. The same applies

for the Progressive Story Stake Method when using the 3D Polyline.



Carlson Software Edit : C:\carlson2007b4\USER\scadrprt.tmp

File Edit Settings

Open Save Print Exit Find Screen Hide

Story Stake Report 5/3/2006 11:26

Point#	Elev	Description	Cut/Fill	Distance	Slope
	91.498		Cut 3.072	51.671	-5.9% -16.82:1
	88.426		Fill 0.02	0.270	5.6% 17.71:1
	88.441		Fill 0.16	25.909	0.6% 159.60:1
	88.604		Fill 0.41	18.644	2.2% 45.15:1
	89.017		Fill 0.01	19.976	0.0% 2372.93:1
	89.025		Fill 3.04	15.756	19.3% 5.18:1
	92.070		Fill 1.33	6.522	20.3% 4.91:1
	93.397		Fill 12.55	43.871	28.6% 3.50:1
	105.943		Fill 0.09	0.616	15.0% 6.68:1
	106.035		Fill 0.35	0.826	42.5% 2.35:1
	106.386		Fill 12.93	50.306	25.7% 3.89:1
	119.318		Fill 2.50	12.564	19.9% 5.03:1
	121.815		Fill 6.20	19.437	31.9% 3.14:1
	128.011		Fill 0.65	1.416	46.2% 2.16:1
	128.665		Cut 0.031	1.607	-1.9% -51.66:1

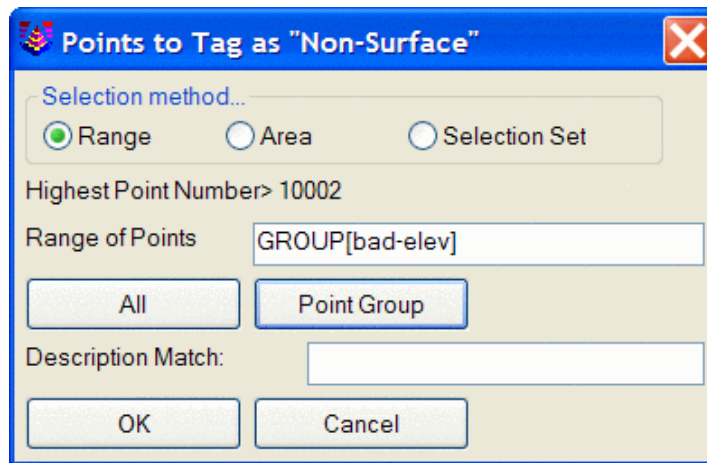
Pulldown Menu Location: 3D Data >> Story Stake

Prerequisite: Screen entities such as 3D polylines and contours

Keyboard Command: story_report

Tag Non-Surface Points

This command allows you to tag Carlson points in the drawing so that they will not be used when creating a surface. These could be points that are far from the site, such as off-site horizontal control, or points with elevations that are not on the ground, such as a TBM taken on the top of a fire hydrant. There are several methods available to select the points for tagging as non-surface points. One key to remember is that they must be present in the drawing to be tagged.



Range: This option allows you to specify a range of point numbers, or select ALL of the points currently in the drawing, or specify a Point Group, remembering, however, that only points that are currently in the drawing can be tagged. So if you select a Point Group, but only some of the points listed in the Point Group are currently present in the drawing, the whole Point Group will not be tagged

Area: This option allows you to utilize inclusion and/or exclusion polyline(s) to specify an area in the drawing within which any points currently in the drawing are tagged as non-surface points.

Selection Set: This option allows the manual selection of points within the drawing.

Description Match: This option allows the filtering of selected points by descriptions. For example, you could use a Range of ALL, but set the Description Match to TBM, and only the points with that description would be tagged.

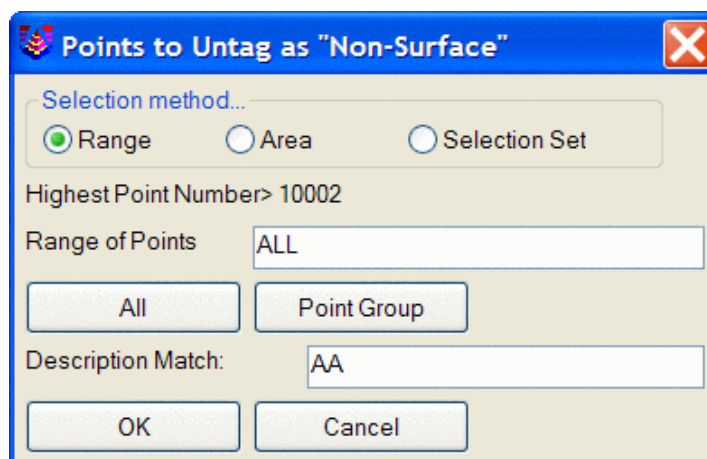
Pulldown Menu Location: 3D Data >> Non-Surface Points

Keyboard Command: tagns

Prerequisite: Carlson points in a drawing

Untag Non-Surface Points

This command allows you to Untag Carlson points in the drawing that have been tagged as non-surface points, so that they will again be used when creating a surface. As with tagging non-surface points, there are several methods available to select the points for untagging, and the points must be present in the drawing to be untagged.



Range: This option allows you to specify a range of point numbers, or select ALL of the points currently in the drawing, or specify a Point Group, remembering, however, that only points that are currently in the drawing can be tagged. So if you select a Point Group, but only some of the points listed in the Point Group are currently present in the drawing, the whole Point Group will not be tagged

Area: This option allows you to utilize inclusion and/or exclusion polyline(s) to specify an area in the drawing within which any points currently in the drawing are tagged as non-surface points.

Selection Set: This option allows the manual selection of points within the drawing.

Description Match: This option allows the filtering of selected points by descriptions. For example, you could use a Range of ALL, but set the Description Match to TBM, and only the points with that description would be tagged.

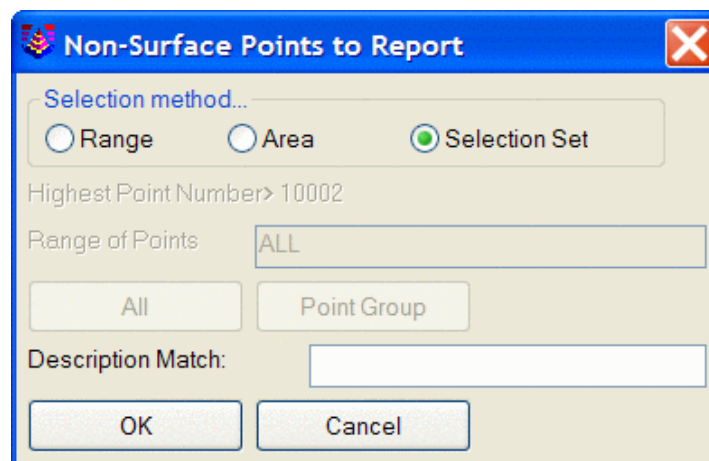
Pulldown Menu Location: 3D Data >> Non-Surface Points

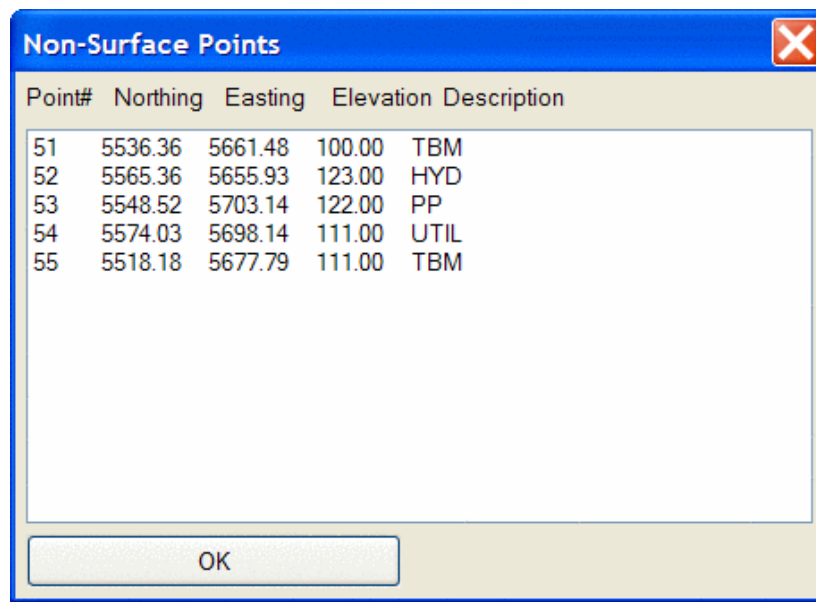
Keyboard Command: untagns

Prerequisite: Carlson points in a drawing

Report Non-Surface Points

This command allows you to generate a report of Carlson points in the drawing that have been tagged as non-surface points. As with tagging and untagging non-surface points, there are several methods available to select the points for the report, and again, the points must be present in the drawing to be included in the report.





Range: This option allows you to specify a range of point numbers, or select ALL of the points currently in the drawing, or specify a Point Group, remembering, however, that only points that are currently in the drawing can be tagged. So if you select a Point Group, but only some of the points listed in the Point Group are currently present in the drawing, the whole Point Group will not be tagged

Area: This option allows you to utilize inclusion and/or exclusion polyline(s) to specify an area in the drawing within which any points currently in the drawing are tagged as non-surface points.

Selection Set: This option allows the manual selection of points within the drawing.

Description Match: This option allows the filtering of selected points by descriptions. For example, you could use a Range of ALL, but set the Description Match to TBM, and only the points with that description would be tagged.

Pulldown Menu Location: 3D Data >> Non-Surface Points

Keyboard Command: tagns

Prerequisite: Carlson points in a drawing

Non-Surface Entities

This command tags selected entities in the drawing so that they will not be used when creating a surface in commands like Triangulate & Contour. For example, you could tag a 3D polyline for a building roof so that it's not used for modeling the ground surface. There are three commands to manage these tags:

Tag Non-Surface Entities: adds the non-surface tag to the selected entities

Untag Non-Surface Entities: removes any non-surface tag from the selected entities

Report Non-Surface Entities: reports the entity type and location for any selected entities that have active non-surface tags

Prompts

Select entities to tag as non-surface.

Select objects: *pick entities to tag*

Pulldown Menu Location: 3D Data >> Non-Surfaces Points/Entities

Keyboard Command: tagns_ent, untagns_ent, reportns_ent

Prerequisite: Entities in the drawing

Tag Hard Breakline Polylines

This command tags polylines with a description so that Triangulate & Contour can identify these polylines as hard breaklines. The tag is invisible and doesn't change the polyline. Triangulate & Contour will not smooth the contours as they cross these hard breaklines, even with contour smoothing turned on. For example you could tag 3D polylines that represent a wall or a curb so that the contours go straight across without smoothing curves. If contour smoothing is turned off, this tag had no effect.

Prompts

Select hard breakline polylines. (For no smoothing in Triangulate & Contour)

Select objects: *Select breaklines to tag*

Select objects: *press Enter* to conclude selection

Set 14 polylines as hard breaklines.

Pulldown Menu Location: 3D Data >> Hard Breaklines

Keyboard Command: hardbrk

Prerequisite: Polylines

Highlight Hard Breakline Polylines

This command visually highlights all polylines in the drawing that have been tagged as hard breaklines.

Pulldown Menu Location: 3D Data >> Hard Breaklines

Keyboard Command: highlight_hardbrk

Prerequisite: Polylines tagged as hard breaklines

Identify Hard Breakline Polylines

This command prompts to select polylines and reports to the command line whether they are tagged as hard breaklines or not.

Prompts

Select polyline: *select polyline*

Polyline is a hard breakline

Select polyline ([Enter] to End): *select polyline*

Not a hard breakline

Select polyline ([Enter] to End): *press Enter* to conclude.

Pulldown Menu Location: 3D Data >> Hard Breaklines

Keyboard Command: id_hardbrk

Prerequisite: Polylines, tagged or not as hard breaklines

Untag Hard Breakline Polylines

This command removes hard breakline description tags from polylines. These tags are used by Triangulate & Contour to identify polylines as hard breaklines. Contours are not smoothed as they cross these hard breaklines, even with contour smoothing turned on. This routine untags polylines so that contours are smoothed across them. If contour smoothing is turned off, hard breaklines have no effect.

Prompts

Select polylines to remove hard breakline tag from.

Select objects: *select polylines*

Pulldown Menu Location: 3D Data >> Hard Breaklines

Keyboard Command: softbrk

Prerequisite: Polylines with hard breakline tag

Extend 3D Polyline to Surface

This command extends 3D polylines to a surface. The 3D polyline is extended from the selected end of the polyline using the slope of the polyline end segment until this vector intersects the surface. The surface is defined by a 3D grid or triangulated surface model which can be selected from a .grd, .flt or .tin file.

Prompts

Select Surface File *Pick a triangulation or grid surface*

Pick polyline to extend: *pick a 3D polyline*

Pick polyline to extend (Enter to end): *press Enter*

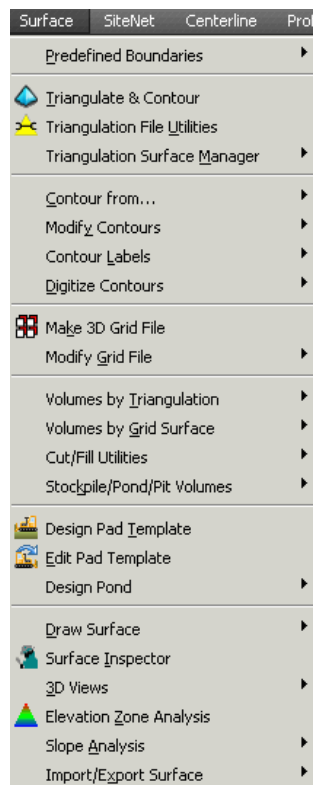
Pulldown Menu Location: 3D Data

Keyboard Command: surf_extend

Prerequisite: 3D Polylines and a surface model file

Surface Menu

The Surface menu has command for building and processing triangulation and grid surface models. The Survey and Hydrology modules have a subset of the surface commands. The Civil module has the full set of surface commands.



Tag Predefined Boundaries

This command allows you to identify closed polylines to be used as inclusion or exclusion boundaries. These boundaries are applied in Surface Menu commands such as *Two Surface Volumes* and *Triangulate & Contour*. Inclusion polylines limit processing to inside the polyline(s). For example, an inclusion polyline for volumes would be the limit of disturbed area. Exclusion polylines prevent processing inside the polyline(s). For example, a building perimeter or pond surface could be an exclusion polyline for contouring. *Tag Predefined Boundaries* assigns a site name to polylines and flags the polyline as either inclusion or exclusion.

Many Surface commands will prompt for inclusion and exclusion polylines. The advantage to Predefined Boundaries is that you don't have to select the boundary polylines each time that you run the Surface command. Instead, the program will recognize that the boundary is already specified and will prompt "Use predefined boundary Area 1 (<Yes>/No)?" This lets you simply press enter to use your predefined boundaries. If you want to pick different boundaries, you can type *N* for No and select them prior to processing. When you have more than one set of predefined boundaries, the routine lets you choose from a list of the boundary names as shown in the dialog.

Prompts

Boundary name <Site 1>: *Area 1*

Select Inclusion perimeter polylines.

Select objects: *pick the closed polylines or press Enter for none*

Select Exclusion perimeter polylines.

Select objects: *pick the closed polylines or press Enter for none*

Pulldown Menu Location: Surface >> Predefined Boundaries

Keyboard Command: plzone

Prerequisite: Closed polyline

Identify Predefined Boundaries

Identify Predefined Boundaries identifies any polylines in the drawing that have been previously tagged as *Predefined Boundaries*. The *Pick* option prompts to pick a polyline, and if it has been tagged as a Predefined Boundary, the Boundary Type and Boundary Name are presented at the command line. The *Search* option scans the entire drawing for Tagged Predefined Boundaries, highlights them in the drawing, and lists their location, layer, type and name at the command line.

Predefined Boundaries are applied in Surface commands such as *Two Surface Volumes* and *Triangulate & Contour*. Inclusion polylines limit processing to inside the polyline(s); e.g., an inclusion polyline for volumes would be the limit of disturbed area. Exclusion polylines prevent processing inside the polyline; e.g., a building perimeter or pond shoreline could be an exclusion polyline for contouring. Identifying polylines that have been previously tagged as Predefined Boundaries is often helpful during surface modeling.

Prompts

Pick polylines to check or search drawing [**<Pick>/Search**]: *press Enter for default pick method*

Select boundary polyline: *pick a polyline*

Inclusion boundary polyline for Site 1

Select boundary polyline (Enter to end): *press Enter*

Pulldown Menu Location: Surface >> Predefined Boundaries

Keyboard Command: plzoneid

Prerequisite: Assigned, predefined inclusion/exclusion closed polylines

Untag Predefined Boundaries

This command removes the previously tagged predefined boundary names from selected polylines. These polylines will no longer be automatically recognized as boundary polylines.

Prompts

Select polylines to remove boundary tag from.

Select objects: *pick the boundary polylines*

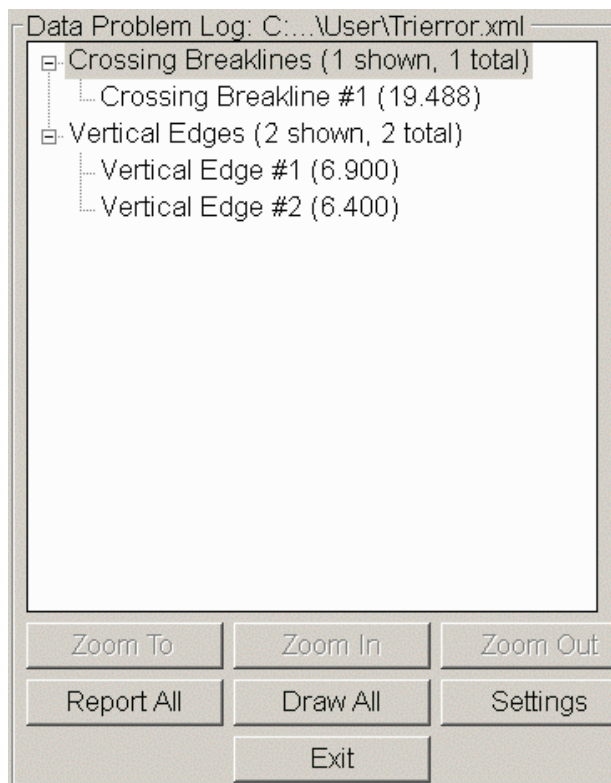
Pulldown Menu Location: Surface >> Predefined Boundaries

Keyboard Command: nozone

Prerequisite: predefined boundary polylines

Triangulation Data Check

This command checks 3D drawing entities for errors with being used to build a triangulation surface. For example, this routine checks for 3D polylines that cross and have different elevations at the crossing which is not allowed because a triangulation surface can't have different elevations at the same point. This command prompts to select the entities to check and then has a dialog to review any errors. See the Error Log section of the Triangulate and Contour topic of the manual for more information.



Pulldown Menu Location: Surface
Keyboard Command: tri_check_data
Prerequisite: Entities to check

Triangulate & Contour

At the heart of nearly every land design project is at least one terrain model. These models go by several names and one of the most common is that of a "TIN" or Triangulated Irregular Network; another common name is that of a "DTM" or Digital Terrain Model. Since accurate representations of a surface model are significantly important to most land development projects, having a thorough understanding of the Triangulate & Contour controls is very important.

Surface models are generally comprised of combinations of the following general data types:

- Points - Most surface models are comprised of points whose coordinates (x,y,z) contribute to the formation of triangular planes that connect three points that are in close proximity to one another. Within Carlson, most points come from the Draw Field to Finish command and/or the Draw-Locate Points command. Points can be selectively filtered from the triangulation engine through the use of the Tag Non-Surface Points command.
- Breaklines - Breaklines (or "fault lines") are used to control the connection sequence between four points which results in two triangles. Common uses of breaklines include ravines, ditches, berms and other areas where distinct grade discontinuity occurs. The "leg" of a triangle can travel along a breakline but cannot cross the breakline. Breaklines must be in the form of 3D polylines or simple lines whose vertices or endpoints define a valid "Z" elevation. A common problem related to breaklines is when two breaklines cross one another in 3D space. In these situations, an impasse results and will result in a "crossing breakline" report. Within Carlson, most breaklines come from the Draw Field to Finish command and/or the 3D Polyline command. Breaklines fall into one of two general categories:
 - "Soft" breaklines - Unless otherwise specified, all breaklines are considered "soft" breakline. The nature of soft breaklines allows a degree of contour smoothing across the breakline itself resulting in a "weathered-" or natural-looking contour.

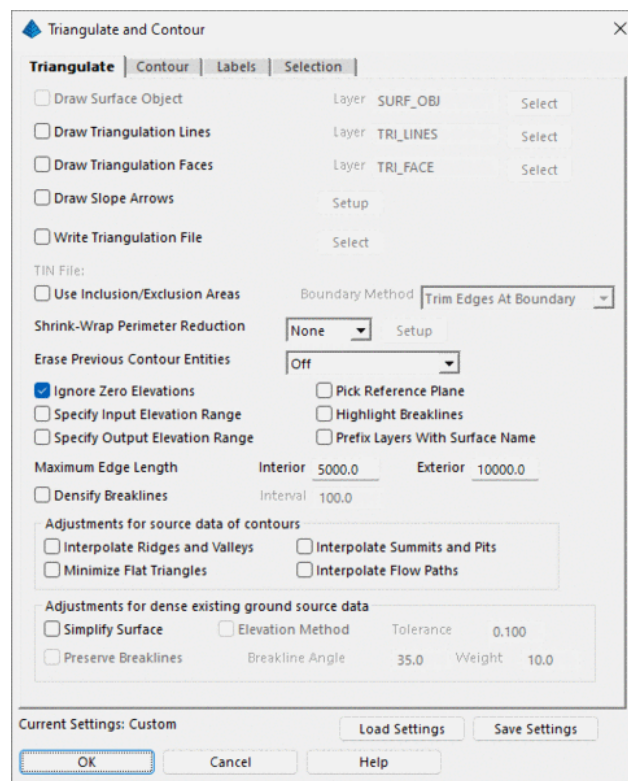
- "Hard" breaklines - Breaklines tagged as "hard" breaklines prevent contour smoothing through the breakline. Hard breaklines are generally used to represent man-made terrain breaks that commonly occur during excavation and construction. Breaklines can be changed to hard breaklines through the use of the Tag Hard Breaklines command.

Breaklines and other triangulate-able entities can be selectively filtered from the triangulation engine through the use of the Tag Non-Surface Entities command.

- Inclusions - Inclusions (or "boundaries") are used to identify the entities that can be used for triangulation and multiple inclusion regions can be selected for a given surface model. Entities that fall outside of an inclusion boundary and are not otherwise bound by a different inclusion boundary are ignored by the triangulation engine. Inclusion regions must be in the form of a closed 2D or 3D polylines. Within Carlson, most inclusion polylines come from the Shrinkwrap Entities command.
- Exclusions - Exclusions (or "void regions") are the antithesis of Inclusions and are used to prevent triangulation from occurring between points that are bound by an Inclusion region. Common uses of exclusion regions include building footprints and free-standing water limits (*e.g.* ponds). Entities that fall inside an exclusion region are ignored by the triangulation engine. Exclusion regions must be in the form of a closed 2D or 3D polylines. Within Carlson, most exclusion polylines come from the Boundary Polyline command and/or the 3D Polyline command.

Carlson provides a programming interface for these file types and also offers a third file type (*.GRD) for the representation of terrain data. See the Notes section for additional details.

The **Triangulate** tab provides options and settings that control the creation and analysis of the TIN itself.

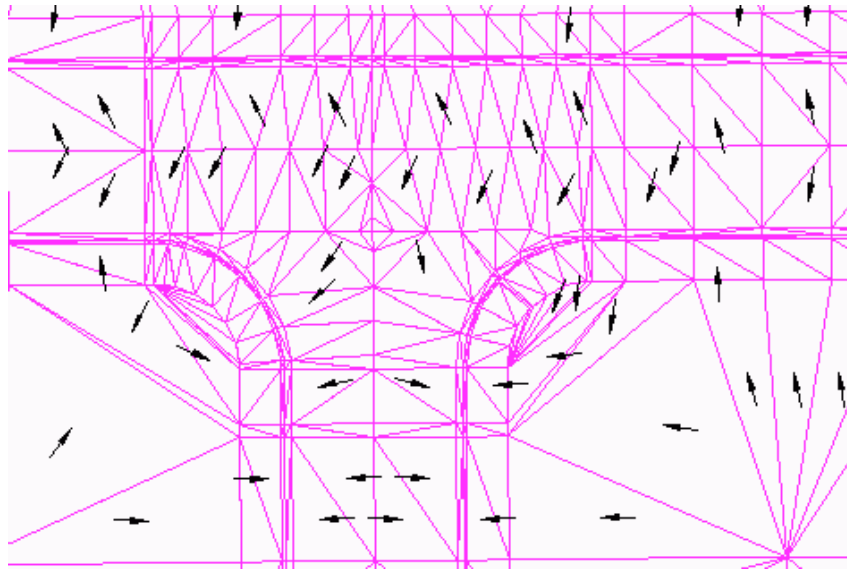


Draw Surface Object: This option draws the triangulation faces as a Carlson Surface Object.

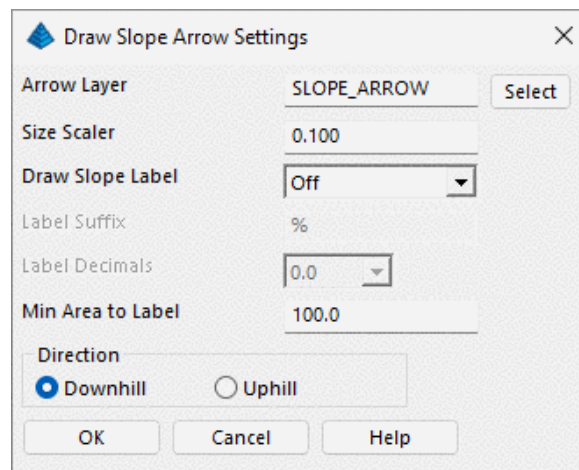
Draw Triangulation Lines: When enabled, the program will draw the triangulation using simple line entities at the appropriate elevation(s). Use the **Select** button or specify the layer for these lines.

Draw Triangulation Faces: When enabled, the program will draw the triangulation using a collection of 3D Face entities. These 3D Faces can then be used rendering routines such as *HIDE* and *SHADE* or in Carlson routines such as 3D Viewer Window, 3D Surface Fly-Over and Slope Zone Analysis. Use the **Select** button or specify the layer for these 3D Faces.

Draw Slope Arrows: When enabled, slope arrows are created within the triangles indicating the downhill dip direction as illustrated below.



Clicking the **Setup** button yields the Draw Slope Arrow Settings dialog box.



Arrow Layer: Indicate the layer to which the slope arrows are to be placed.

Size Scaler: Indicate a positive, non-zero value for the scale factor that should be applied to the slope arrows.

Draw Slope Label: When enabled, the slope value (expressed as a percentage or a ratio) of the triangle is labeled onto the slope arrow. Specify the desired unit suffix (*e.g.* "%") to apply to the end of the numerical value that is calculated from the TIN triangle(s).

Label Decimals: Indicate the amount of precision that is to be displayed on the slope label.

Min Area to Label: Indicate the smallest allowable triangle size that can be used for the slope percentage labels.

Write Triangulation File: When enabled (strongly suggested), an external surface model file is created which can subsequently be used for volume calculations, the creation of profiles, cross-sections and graded pads. Carlson

currently provides two file types to store the DTM data created by the Triangulate & Contour routine:

1. *.TIN - The TIN file format is the default and preferred file format due to its compact file size and organizational efficiency. The Carlson TIN format is governed by Carlson and is in a binary (non-human readable) format.
2. *.FLT - The FLT file format is a legacy ASCII-based (human-readable) file format and is used in some older machine control applications.

Use Inclusion/Exclusion Areas: When enabled, the program will prompt you for inclusion and exclusion polylines and prevents the use of the Shrink-Wrap Perimeter Reduction option. These are used to further control the area of activity for triangulation and contouring. The inclusion and exclusion polylines must be closed polylines and when used, must be drawn before using *Triangulate & Contour*. It is suggested that the height of the **Command:** line display must be set to show at least two lines so that the additional prompts can be easily viewed. Refer to the Notes section for additional information on Inclusion/Exclusion polyline selections.

Boundary Method: This option controls whether edges that cross the inclusion or exclusion perimeter are trimmed or removed. Use the Remove option if you don't want the triangulation to interpolate across the perimeters. Use the Trim option if you want the triangulation to fill the area up to the perimeter.

Shrink-Wrap Perimeter Reduction: This option produces an inferred Inclusion region around the data to be selected and mimics the results of the Shrinkwrap Entities command. Under the Setup, there is an option to draw the shrink-wrap perimeter polyline on a specified layer.

Erase Previous Contour Entities: In the event that a TIN needs to be recreated and Carlson-produced contours are in the drawing, three options exist that allow you to control whether or not the contour data should be removed from the drawing:

- Off - All existing Carlson-generated contours are left intact in the drawing. If these contours satisfy all of the triangulation requirements, they can be utilized by the Triangulation algorithm.
- Current Surface - Only the Carlson-generated contours that are associated with the active Triangulation file are removed from the drawing.
- All Contour Entities - All Carlson-generated contours are removed from the drawing, regardless of the surface model that created them.

Ignore Zero Elevations: When enabled, this option will filter out all data points and entities at an elevation of zero from the triangulation data set.

Pick Reference Plane: When enabled, this option allows you to contour an overhang or cliff by changing the reference plane to a side view. The reference plane can be specified by using the *View>Viewpoint 3D>View* command (see the AutoCAD/IntelliCAD Help menu for additional details) or by specifying three data points on the cliff (two along the bottom and one at the top).

Specify Input/Output Elevation Range: If you would like to manually set the range over which to contour, select either or both of the aforementioned toggles. One controls the triangulation of the source data and the other for the contour output. The program will automatically contour from the lowest elevation in the data set up to the highest at the increment specified in Contour Interval.

Highlight Breaklines: When enabled, this routine highlights breaklines in the triangulation network by drawing the triangulation lines along breaklines in yellow.

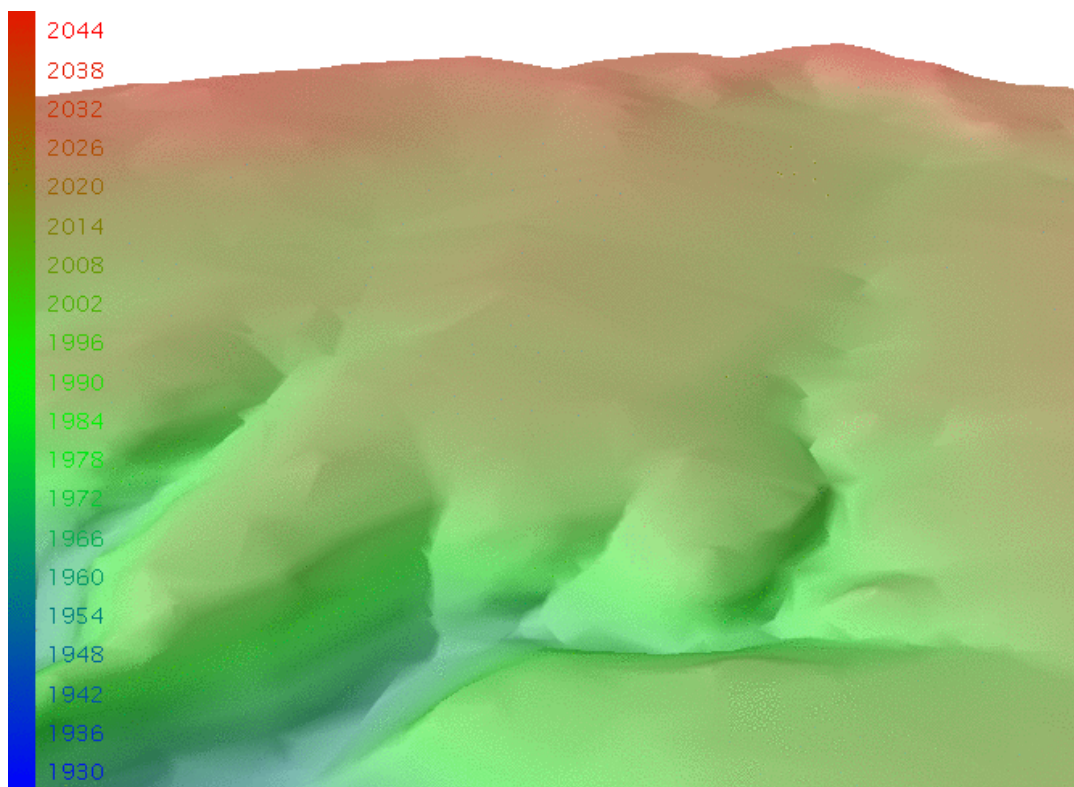
Prefix Layers With Surface Name: This option applies when using Write Triangulation File. The file name is added as a prefix to all the layers created during Triangulate and Contour as a way for layer management to organize all the contouring layers for a surface file.

Maximum Edge Length: Two bounds are provided to limit the length of the "legs" within a triangulation network. Based on the available data, if the edge length of a triangle exceeds the respective bound, the triangle will not be formed:

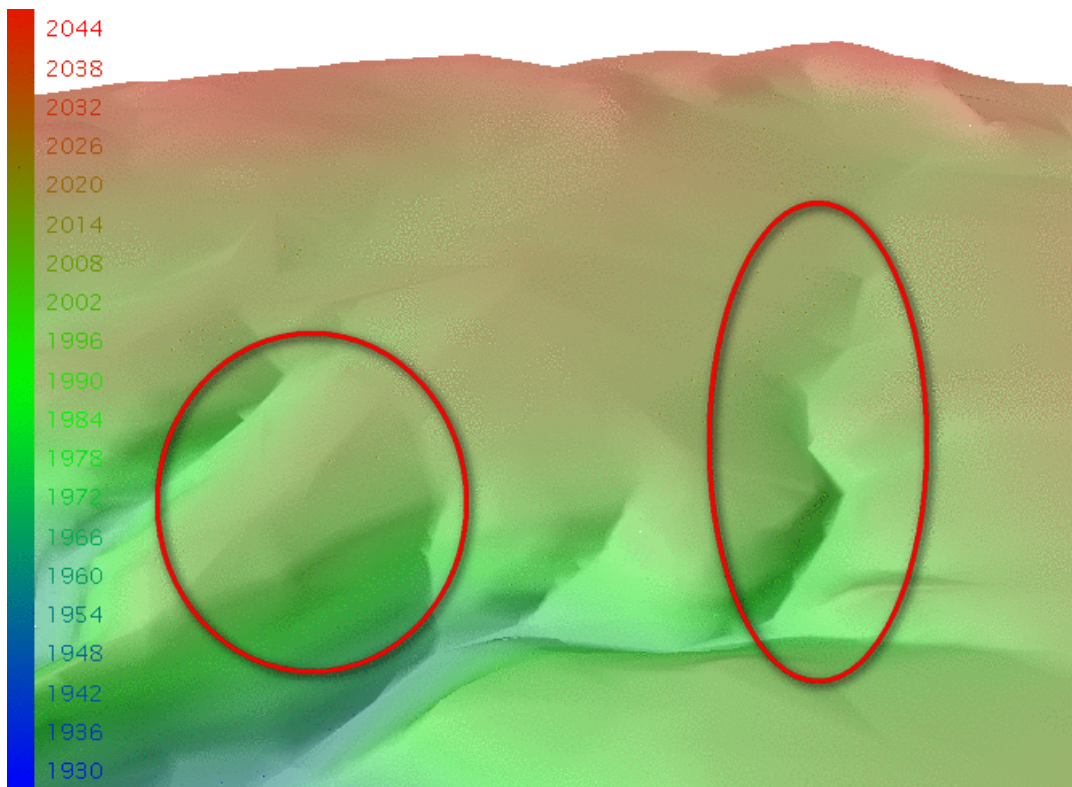
- **Exterior:** This value applies to triangulation lines around the perimeter of the triangulation area.
- **Interior:** This value applies all the other triangulation lines. Generally you would have the Exterior value larger than the Interior value.

Densify Breaklines: This option subdivides linework segments for the input data so that the maximum length of the segments is the specified Interval. This option is similar to the Densify Polyline Vertices command except that the entities are not modified in the drawing. Having shorter breakline segments is often helpful for holding the breaklines in the TIN and making more regularly sized triangles.

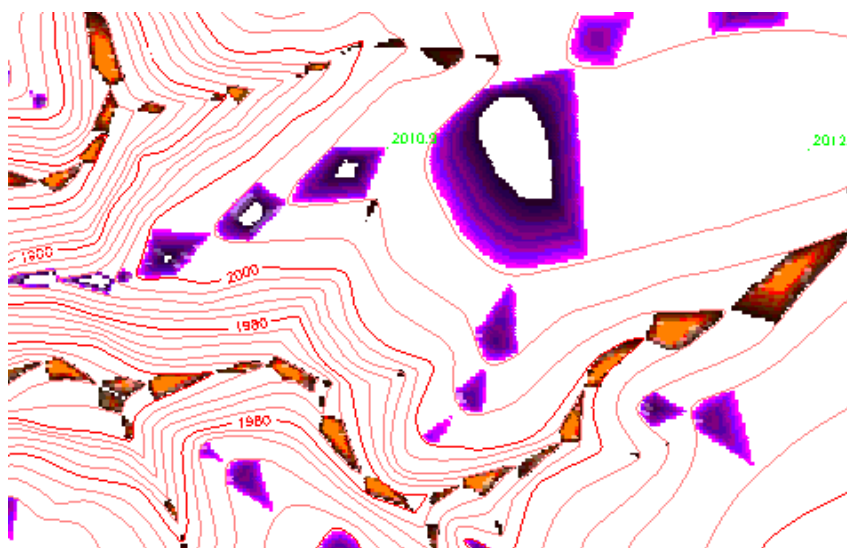
Minimize Flat Triangles: When enabled, this toggle instructs the triangulation "engine" to iterate through the triangulation permutations to minimize the occurrence of "flat" (or more precisely, horizontal) triangles. Flat triangles often occur when creating surface models from contour data. In this scenario, the often used Delaunay triangulation algorithm may produce unrealistic results. The Minimize Flat Triangle option will perform additional permutations of the triangulation network through the use of the Surface Manager > Swap Edge routine in an attempt to maximize the number of "sloped" triangles. Another option that produces similar results is the Interpolate Ridges and Valleys option.



Before: Surface made from an existing contour map with Minimize Flat Triangles disabled.



After: The same surface with Minimize Flat Triangles enabled. Note the better defined ravine and ridge definitions.



Difference: A Cut/Fill Color Map showing the regions of significant triangulation difference between the "Before" scenario and the "After" scenario of "Minimize Flat Triangles."

Interpolate Ridges and Valleys: The purpose of this option is to avoid flat areas by better defining surface ridges and valleys. When enabled, this option inserts breaklines into the drawing which are subsequently used in the triangulation process in an attempt to minimize flat, horizontal triangles.

Interpolate Summits and Pits: When enabled, this option creates additional triangulation in a summit or pit situation to more accurately represent existing ground conditions from a surface model created from contour entities. Since the tops of hills and the bottom of pits are often not shown on existing ground contour maps, this option often helps improve the accuracy of existing terrain conditions.

Interpolate Flow Paths: this function generates 3D data polylines from a contour map that approximate the downhill flow of water through valleys. These polylines are then used when creating a triangulation and can

improve the it in these valley regions by giving more definition to these areas. The polylines are created by looking for "sharp" or angled regions of the contours, and then connecting these regions with a smooth curve.

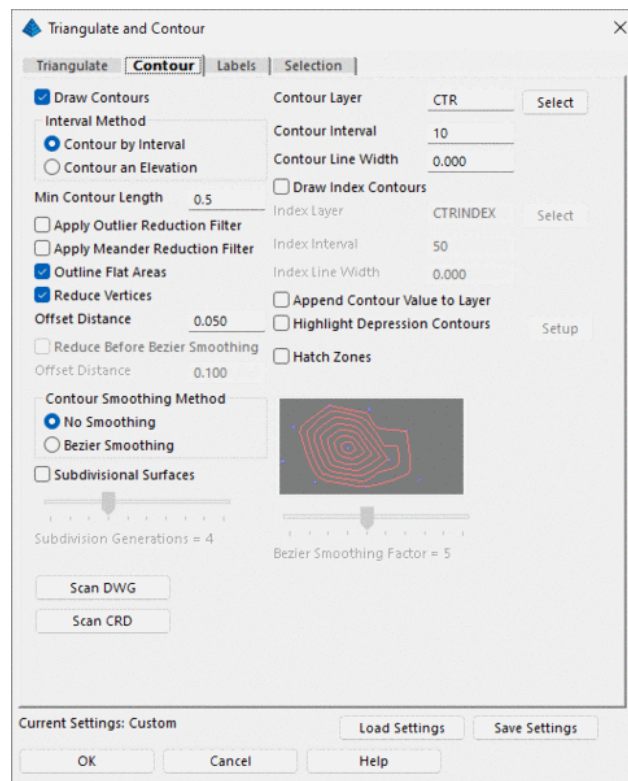
Simplify Surface: When enabled, this option reduces the digital size of a surface without significantly compromising the integrity or accuracy of the surface itself. The most common application to enable this option is when using very large datasets, such as smoothed contours. Its use is less applicable to design surfaces or surfaces based on surveyed points, but it can still be utilized.

Elevation Method: When enabled, this option reduces the size of the surface file by analyzing the difference in elevation between each vertex of the TIN and the vertices directly surrounding it, assigning a numerical weight or value to each vertex. If it is determined that the calculated weight for a particular vertex is less than the *Tolerance* factor, the vertex is a candidate for removal. The number of vertices removed is directly proportional to the *Tolerance* factor, so the higher the *Tolerance* factor, the more vertices are removed and vice versa.

Preserve Breaklines: When enabled, this option analyzes the TIN by focusing on the edges; calculating the angular difference between adjacent triangular faces. If the angular difference between edges is greater than the specified *Breakline Angle*, it is considered to be a breakline, and it is preserved. If its angular difference is determined to be below the *Breakline Angle*, it becomes a candidate for removal. In that case, the *Weight* factor is applied to the corresponding vertex, adjusting its original value. If the resulting value is still below the *Tolerance*, it is then removed. The number of vertices removed is inversely proportional to the *Weight* factor, so the greater the *Weight* factor. The fewer vertices that are removed, the lower the *Weight* factor, the more vertices that are removed.

A good rule-of-thumb that can be used when deciding whether or not to use these options is:

- If the surface contains no man-made features, use *Simplify Surface* option (with or without the *Elevation Method* option).
- If the surface contains man-made features, such as roads, use both *Simplify Surface* and *Preserve Breaklines*.



Draw Contours: When enabled, the program will draw contour lines using the designated settings after triangulation process is complete. Otherwise, only the designated Triangulation operations are performed. If this option is

disabled and contours are subsequently desired, use the Contours from TIN File command.

Interval Method: Indicate the desired elevation(s) for contours to be drawn:

- **Contour by Interval:** Specify the desired interval (*e.g.* every 2 feet) into the *Contour Interval* field.
- **Contour an Elevation:** Specify a desired elevation (*e.g.* a floodplain elevation or other unique elevation of interest) and set the desired value into the *Contour Interval* field.

Contour Layer/Index Layer: Specify the layer to which the contours/index contours are to be drawn.

Contour Interval/Index Interval: Specify the interval to which the contours/index contours are to be drawn. When contouring an elevation, you can enter multiple space-separated values to draw multiple contours. For example, entering "5 8 14" into this field would draw contours at elevations, 5, 8, and 14.

Contour Line Width/Index Line Width: Specify the line width to be applied to the contours/index contours.

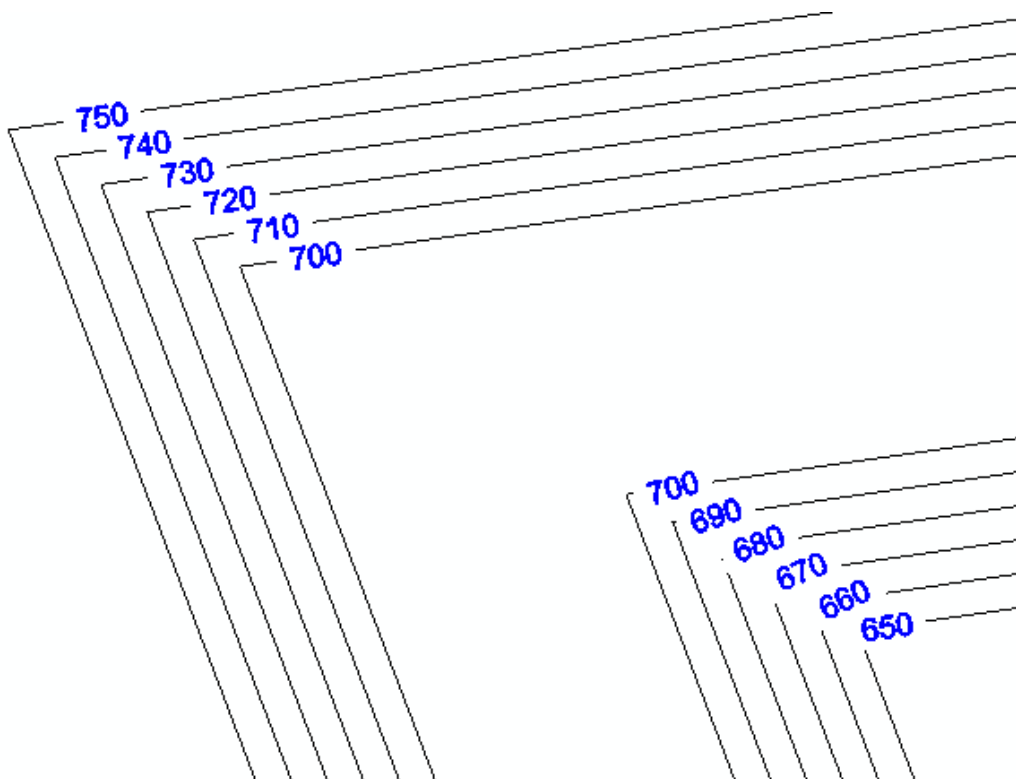
Draw Index Contours: When enabled, index (or "major") contours will be created with independent characteristics from the regular contours.

Min Contour Length: Specify the minimum linear threshold that should be used to draw contours.

Apply Outlier Reduction Filter: When enabled, this option attempts to remove "the jaggies" which tend to occur along long, thin triangles.

Apply Meander Reduction Filter: This option smooths contours by removing back-tracking.

Outline Flat Areas: This option will outline flat areas of the surface. This is particularly useful for showing catch-benches that have been designed into a steeply angled portion of the surface. In the below image, a the 700' contour line is drawn on both sides of the catch bench. If the Outline Flat Areas option were disabled, only one side of the catch bench would be drawn.



Reduce Vertices: When enabled, this option removes extra vertices from the contours using the *Offset Distance* value. The Offset Distance is the maximum amount that the polyline can move horizontally when removing a point.

The result of this action is often a significant reduction in vertex locations along the contour resulting in a more efficiently-sized and compact drawing file.

Offset Distance: Specify the maximum allowable distance for shifting the original contour line in order to reduce vertices. The reduced contour will shift no more than this value, at any point, away from the original contour line. A lower value will decrease the number of vertices removed and keep the contour line closer to the original. A higher value will remove more vertices and allows the contour to shift further from the original location.

Reduce Before Bezier Smoothing: When enabled, this option removes extra vertices from the contours before they undergo Bezier Smoothing using the *Offset Distance* value. The Offset Distance is the maximum amount that the polyline can move horizontally when removing a point. Removing points before smoothing gives the Bezier smoothing more freedom to make the contour curvy.

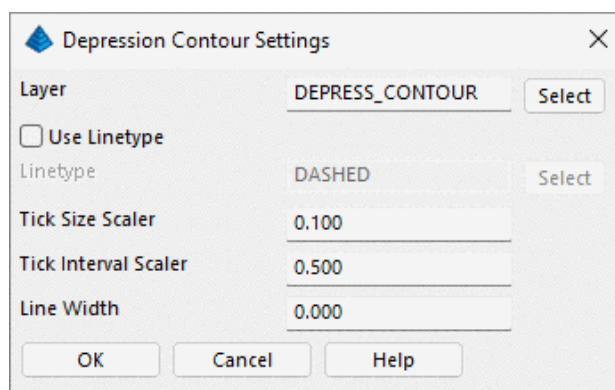
Contour Smoothing Method: Indicate the desired amount of smoothing (often used for existing, natural ground conditions to simulate a "weathered terrain" effect) that should be applied to the contours:

- **No Smoothing:** This option is often used for proposed, man-made surface considerations where the terrain has been shaped with earth-moving equipment. For applications where a "nature-emulated" man-made terrain is desired, refer to the Carlson Natural Regrade documentation.
- **Bezier Smoothing Factor:** This option holds all the contour points calculated from the triangulation and only smooths between the calculated points.
- **Polynomial Smoothing:** This option applies a fifth degree polynomial equation through the contour data points for a smooth transition between the triangulation faces.

Subdivisional Surfaces: When enabled, adjust the horizontal slider to indicate the degree of triangular subdivisions. This causes each triangle in the triangulation surface model to be subdivided into $(x + 1)^2$ triangles, where $x = \text{Subdivision Generations}$. The mathematically generated sub-triangle vertices are raised or lowered to provide smoother contours. More generations increase the smoothness of the contours but incur increased processing time. Although this algorithm does not produce "crossing contours," it can result in undesired contours in terrain scenarios such as where graded slopes abruptly transition to nearly horizontal slopes (*e.g.* the sides and bottom of a detention pond).

Bezier Smoothing Factor: Adjust the horizontal slider to obtain a preview of how much smoothing can be expected at each setting. Sliding the bar to the left results in a lower setting which have less looping or less freedom to curve between contour line points. Likewise, moving the slider to the right results in a setting that increases the looping effect. Note that too much smoothing applied in some situations can result in crossing contours.

Highlight Depression Contours: Depression contours are the lowest elevation closed contours that are surrounded by higher closed contours. This option highlights depression contours by either using a separate linetype or by drawing tick marks along the contour. Use the *Setup* button to get this dialog of settings for depression contours.



Linetype: Set the CAD linetype to use for depression contours.

Layer: Specify the layer to which the depression contours are to be drawn.

Tick Size Scaler: Indicate the relative scale factor that should be applied to the depression ticks.

Tick Interval Scaler: Indicate the desired interval scaler which controls the spacing of the depression ticks.

Line Width: Specify the line width to be applied to the depression contours.

Hatch Zones: When enabled, this option will create hatching between the contours based on elevation zones. The following dialog will open allowing the user to specify the hatch type and color for each elevation zone. The entire elevation range of selected data is displayed under Current Values.

Elevation	Range	Color	Pattern	Scale	Layer
500.000	<= 500.000	[Red]	SOLID	Set	zone1
510.000	500.00 to 510.00	[Orange]	SOLID	Set	zone2
520.000	510.00 to 520.00	[Yellow-Orange]	SOLID	Set	zone3
530.000	520.00 to 530.00	[Yellow]	SOLID	Set	zone4
540.000	530.00 to 540.00	[Light Yellow]	SOLID	Set	zone5
550.000	540.00 to 550.00	[Light Green]	SOLID	Set	zone6
560.000	550.00 to 560.00	[Green]	SOLID	Set	zone7
570.000	560.00 to 570.00	[Light Green]	SOLID	Set	zone8
580.000	570.00 to 580.00	[Green]	SOLID	Set	zone9
590.000	580.00 to 590.00	[Light Green]	SOLID	Set	zone10
600.000	590.00 to 600.00	[Green]	SOLID	Set	zone11
610.000	600.00 to 610.00	[Light Green]	SOLID	Set	zone12
620.000	610.00 to 620.00	[Cyan]	SOLID	Set	zone13
630.000	620.00 to 630.00	[Blue]	SOLID	Set	zone14

Auto: Opens the following dialog, allowing for automatic configuration of the range of elevations in each zone, assigning of colors and hatch patterns, and the scale.

Value Range
 Starting: 1
 Set Values
 Starting Value: 500.000
 Value Interval: 10.000

Pattern
 Set Pattern
 Pattern: 2X2BRIKC
 Select

Set Scale
 Scale: 1.000

Set Layer
 Layer: zone1
 Select

Color Setting
 Set Colors
 Starting Color#: 11
 Color: 10

Set Gradient Colors
 Low: 1
 Medium: 1
 High: 1

Starting Zone: Sets the zone with which to begin the application of the setting defined in this dialog. For Instance, if the Starting Zone was set to 10, the settings definitions applied here wouldn't affect Zones 1-9, but would start at

Zone 10.

Set Values: Enables the Starting Value and Value Interval fields, which allow the user to specify the starting elevation for the given zone and set the zone increment.

Starting Value: Sets the elevation of the beginning zone to define.

Value Interval: Sets the elevation increment for subsequent zones.

Set Pattern: Sets the hatch pattern for the defined zones.

Set Scale: Enables the Scale option.

Scale: Sets the scale for the selected hatch pattern.

Set Colors: Modifies the colors in the table using either Starting Color and Color Increment, or the Gradient option.

Starting Color: Sets the starting color number based on the standard CAD color chart.

Color Increment: Sets the color number to increase for subsequent zones. So if the increment was set to 5, and the starting color was 60, the next color would be 65, 70, and so on.

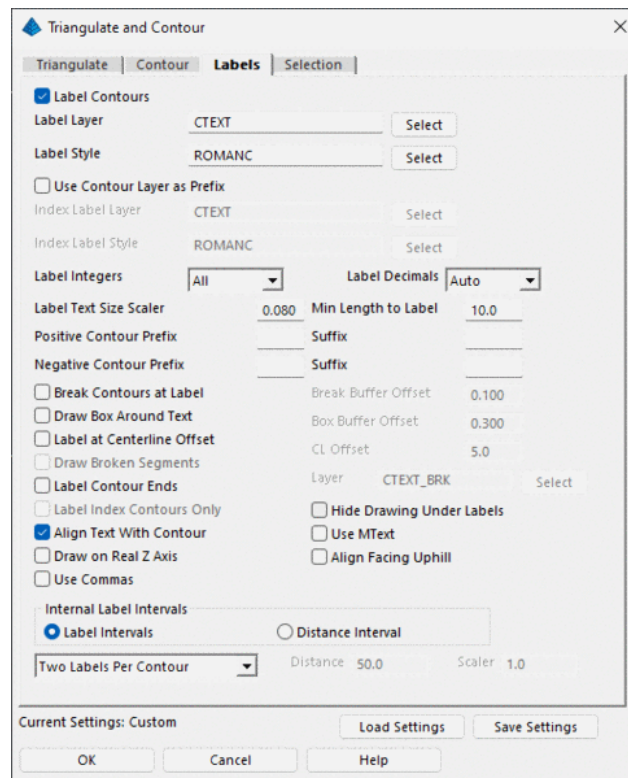
Set Gradient Colors: Sets the colors according to **Low**, **Medium**, and **High** colors. Note that the program will use the Index Colors (total of 256 color options) when setting gradients, which can result in duplicated colors if there are many entries in the table. This option is best used when there are only a small number of hatch zones.

Clear: Clears the all of the Elevation fields in the dialog.

Load: Loads previous settings from a saved .pat file.

Save: Saves the current setting configuration to a .pat file.

Scan DWG/Scan CRD : These buttons report the elevation range which can help with choosing the contour interval.



Label Contours: When enabled, contours will be labeled based on the settings below. If this option is disabled and further contour annotation is desired, utilize the Contour Elevation Label command.

Label Layer: Specify the layer name for intermediate contour labels. To only label index contours, enable the Label Index Contours Only option.

Label Style: Specify the text style that will be used for the contour label text.

Use Contour Layer as Prefix: This option adds the Contour Layer name to the Label Layer name.

Index Label Layer: Specify the layer name for index contour labels.

Index Label Style: Specify the text style that will be used for the index contour label text.

Label Integers controls how many digits to label to the left of the decimal. For example, if all contours are in the 5000's, then setting for three digits would label the 5280 contour as 280.

Label Decimals: Specify the amount of precision to display on the contour labels.

Label Text Size Scaler: Specify a relative text size scale factor to be applied to the label(s).

Min Length to Label: Specify the desired minimum length of contours that should be annotated. In other words, Contours whose length is less than the value will not be labeled.

Positive/Negative Contour Prefix: Indicate a desired string of prefix text (*e.g.* Elev=) that is to precede the positive and/or negative contour elevations, respectively.

Positive/Negative Contour Suffix: Indicate a desired string of suffix text that is to follow the positive and/or negative contour elevations, respectively.

Break Contours at Label: When enabled, the contour lines will be broken and trimmed at the label location for label visibility. As an alternative to physically placing a gap into the contour, consider using the Hide Drawing Under Labels option.

Break Buffer Offset: Specify the offset distance which determines the gap between the end of the trimmed contour line and the beginning or ending of the text.

Draw Box Around Text: When enabled, a rectangle is drawn around the contour elevation labels.

Box Buffer Offset: Specify the offset distance which determines the gap between the box and the beginning or ending of the text.

Label At Centerline Offset: When creating contours and subsequent plan sheets for roads, enable this option to position the labels at a fixed offset from a centerline. The program automatically uses any polylines in the drawing that are tagged as centerlines. To check whether a polyline is a centerline, use the Centerline ID command. To create a centerline polyline from a centerline file, use the Draw Centerline File command.

Draw Broken Segments: When enabled, the segments of contours that have been broken out for label visibility will be redrawn as independent segments. To join these segments back into the contour, use the Join Nearest command.

Layer: Specify the layer that is to receive the newly drawn broken segments.

Label Contour Ends: When enabled, the ends of "open" contours will be labeled.

Label Index Contours Only: When enabled, only the index contours are labeled. This option is active only when Draw Index Contours has been selected in the Contour tab.

Hide Drawing Under Labels: When enabled, a "Wipeout" entity is placed with the annotation label that will create the appearance of trimmed segments at the contour label, even though the contour line is still fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels yet the contour lines

themselves remain contiguous. This feature will also hide other entities that are in the immediate vicinity of the contour label.

Align Text with Contour: When enabled, the contour elevation labels will be rotated to align with their respective contour lines.

Use MText: When enabled, contour labels are created using the MText entity type. Otherwise, the standard DText entity type is used.

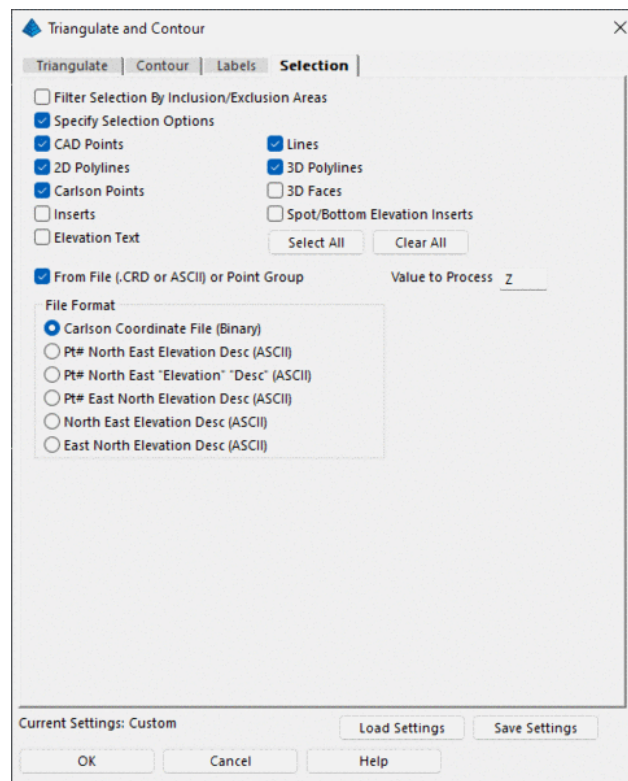
Draw On Real Z Axis: When enabled, the contour labels are placed at the same "Z" (elevation) value of the contour itself. When disabled, the contour labels are placed at a "Z" (elevation) value of 0 (zero).

Align Facing Uphill: When enabled, the contour elevation labels will still be rotated to align with their respective contour lines, but the labels will be placed in such a manner that the top of the text label will always be toward the uphill side of the contour.

Use Commas adds a comma into the labels for the thousands place such as "5,000" instead of "5000".

Internal Label Intervals: Indicate the desired method for contour labels within the contour itself:

- **Label Intervals:** This option will label each contour with a set number of labels.
- **Distance Interval:** This option allows you to specify an interval distance between labels. The distance can be set in drawing units or as a scaler that is multiplied by the drawing horizontal scale.



Filter Selection By Inclusion/Exclusion Areas: This option filters out selected entities from the triangulation that are outside the surface area defined by the inclusion/exclusion perimeter polylines. Otherwise, all the selected entities are used for triangulation and then the triangulation is trimmed at the inclusion/exclusion perimeters. Whether to prompt for inclusion/exclusion perimeters is specified on the Triangulate Tab.

Specify Selection Options: When enabled, indicate the type(s) of entities that are to be used during the triangulation process. This is an excellent method of "filtering out" unwanted entity types or enabling the use of desired entity

types.

CAD Points, Lines, 2D Polylines, 3D Polylines, 3DFaces, Elevation Text and Inserts (blocks) are standard CAD entities types.

Carlson Point Inserts refer to Carlson points (such as those placed with the Draw Field to Finish command or which utilize the Carlson "SRVPNO*" family of blocks with point number, elevation, and description attributes).

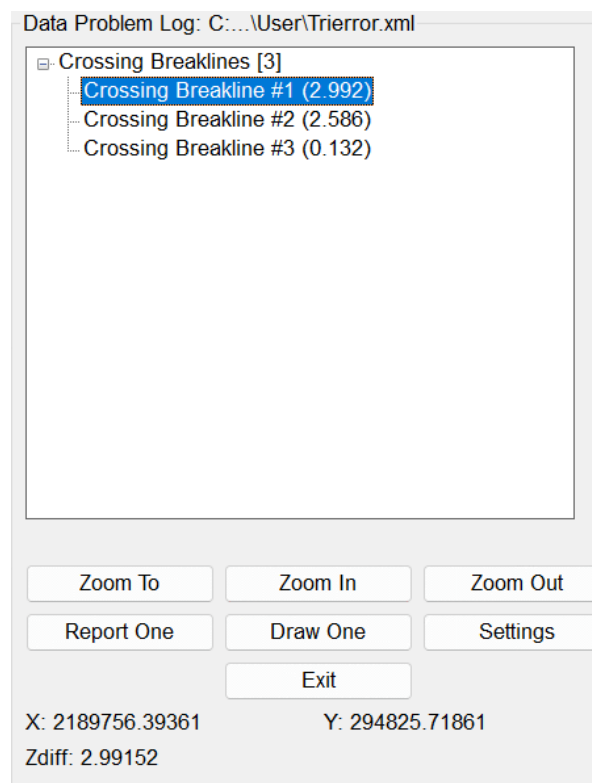
Spot/Bottom Elevation Inserts include text entities that start with 'X'.

From File: When enabled, allows you to triangulate from the points in an external coordinate (.CRD) or ASCII file. This option also provides access to the use of Point Groups as a data source.

An *Error Log* is generated if the triangulation routine finds vertical conflicts between breaklines or other surface entities and displays the conflicts in a "docked dialog box." Three types of conflicts are reported (each conflict type is listed into its own category):

1. Crossing Breaklines - Indicates the common X,Y location of two breaklines that do not share a common "Z" elevation.
2. Vertical Edges - Indicates that two entities or vertexes of differing elevations have the same x-y location, thus forming a vertical plane to another point.
3. Breakline T-Intersections - Indicates that a third entity is abutting another entity, but the second entity doesn't have a vertex at the point of intersection.

Click the "+" sign beside a category to display the individual conflicts within that category and click the "-" sign to collapse the list. When a line item error is selected, a highlighted arrow is temporarily placed in the drawing to indicate the exact location of the specific conflict. Zoom functionality allows the user to more closely inspect the specific problem area, and if needed a marker can be drawn or a report generated for an individual conflict or conflicts.

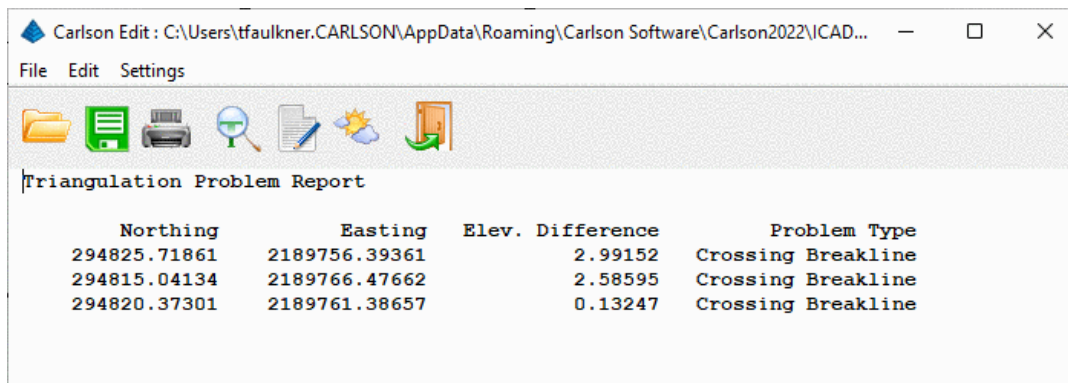


Zoom To: Centers the display on the location of the error without affecting the zoom resolution.

Zoom In: Increases the ability to see detail.

Zoom Out: Decreases the ability to see detail.

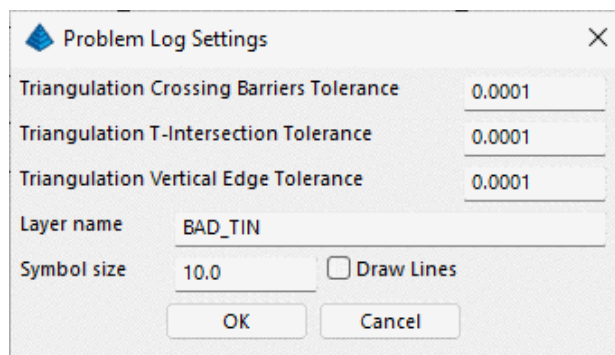
Report One/All: This option toggles between "One" and "All" depending whether a single line item conflict or an entire category is selected from the error log. An error report is generated listing the x-y position and the elevation difference of the entities in conflict.



Northing	Easting	Elev. Difference	Problem Type
294825.71861	2189756.39361	2.99152	Crossing Breakline
294815.04134	2189766.47662	2.58595	Crossing Breakline
294820.37301	2189761.38657	0.13247	Crossing Breakline

Draw One/All: This option toggles between One and All depending whether a single conflict or a category is selected from the list. This option draws an "X" symbol at each selected conflict.

Settings: Indicate the desired configuration settings for the error log:



Problem Log Settings

Triangulation Crossing Barriers Tolerance: 0.0001

Triangulation T-Intersection Tolerance: 0.0001

Triangulation Vertical Edge Tolerance: 0.0001

Layer name: BAD_TIN

Symbol size: 10.0 Draw Lines

OK Cancel

Tolerances: Indicate the lowest elevation difference threshold that should be reported for Crossing Breaklines, Vertical Edges and Breakline T-Intersections, respectively.

Layer Name: Specify the layer name for the "X" entities drawn with *Draw One/All* option. This also sets the layer name for the *Draw Lines* option.

In the case of crossing polylines, *Draw Lines* will trace over the polylines responsible for the conflict.

Symbol Size: Specify the size of the "X" symbol that is drawn to delineate the selected errors. This will determine the actual size of the symbol in the drawing. This value is not multiplied by the horizontal drawing scale.

Note:

- When selecting Inclusion/Exclusion polylines, you may select any number of Inclusion polylines and any number of Exclusion polylines. Selecting multiple Inclusion polylines results in "islands" of terrain data within a given TIN file.
- If *Triangulate & Contour* reports zero points found and fails to do anything when you're using Carlson points, then those points are probably located at zero elevation. To fix this problem, make sure that Carlson Point

Inserts is toggled on in the Selection tab. This will enable *Triangulate & Contour* to read the elevation from the elevation attribute of the point.

- For those experienced in programming, Carlson offers a DTM API (Application Programming Interface) which provides functions that can be used to access and manipulate information stored within a DTM file.
- In surface situations where a series of rectangular grid cells are desired, explore the Make 3D Grid File command.

Prompts

The following are the most often encountered prompts:

Select the Inclusion perimeter polylines or ENTER for none.

Select entities: Select the desired closed polylines that form the bounding inclusion area(s) of the surface model and press Enter when complete.

Select the Exclusion perimeter polylines or ENTER for none.

Select entities: Select the desired closed polylines that form the regions(s) of the surface model where triangulation should not occur and press Enter when complete.

Select the points and breaklines to Triangulate.

Select entities: Select the desired entities from CAD using standard CAD selection methods and press Enter when complete.

Pulldown Menu Location(s): Surface (Survey, Civil, Hydro, Construction, Field, Natural Regrade), Takeoff > Surface Tools

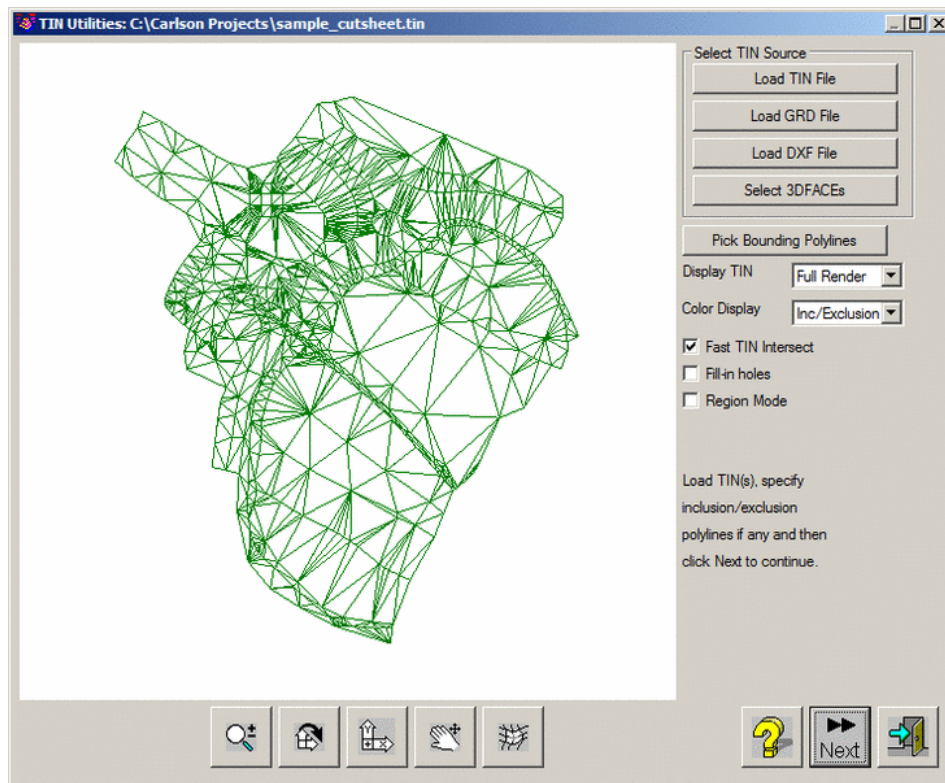
Keyboard Command: tri

Prerequisite: 3D entities in the drawing (defined by the Selection Tab) and/or an external point file

Triangulation File Utilities

This command allows you to modify TIN surfaces in a variety of different ways, then allows for 3D viewing and shading of the modified surface and finally for saving the file with a choice of output formats. The focus of the routine is to elevate or lower the TIN or selected areas within the TIN, merge TINs with other surfaces, or use data from other TIN files to apply to the current TIN.

Operations can be performed on the entire TIN or just on user selected Inclusion and/or Exclusion areas. The routine will automatically rework the TIN network for conformation to a selected boundary, say a building outline. In the case of said building, a value of 10 could be subtracted from the building outline. This will drop all of the triangulation within the outline by 10', thus creating a model of the excavated area for the building. The modified TIN can then be saved to a new file, which could be used to compute an excavation volume with Volumes by Triangulation. This routine does not allow for manual editing of individual portions of the TIN; refer to the editing options found under Surface Manager.



Load TIN File: Allows you to specify a triangulation (.flt or .tin) file to load.

Load GRD File: Allows you to specify a grid (.grd) file to load.

Load DXF File: Allows you to specify a DXF file to load. Only loads 3DFACE entities from the selected DXF file.

Select 3D Faces: Allows you to select 3DFACE entities from the current drawing. This also includes rectangular 3D faces from a plotted grid.

Pick Bounding Polylines: Allows you to select any inclusion/exclusion perimeter(s). When this button is selected, the user is taken back to the drawing and prompted to select the perimeters. Press *Enter* when the selections are finished to return back to the dialog.

Display TIN: Controls the detail level of the TIN in the graphic display. When the TIN is very large, the graphic display can cause delays. The Low Detail mode shows a simplified TIN graphic. The Outline Only mode shows only the TIN perimeter. The Full Render mode shows all the triangles.

Color Display: Displays the triangles based on whether or not they are part of the Inclusion/Exclusion set via the Pick Bounding Polylines option or colored based on the individual triangle color property.

Fast TIN Intersect: When checked, this command will perform a simple and fast check for overlapping triangles and is the preferred choice in most cases. However, if problems with the TIN are suspected, this option should be unchecked so a complete and thorough check and repair of the TIN is performed.

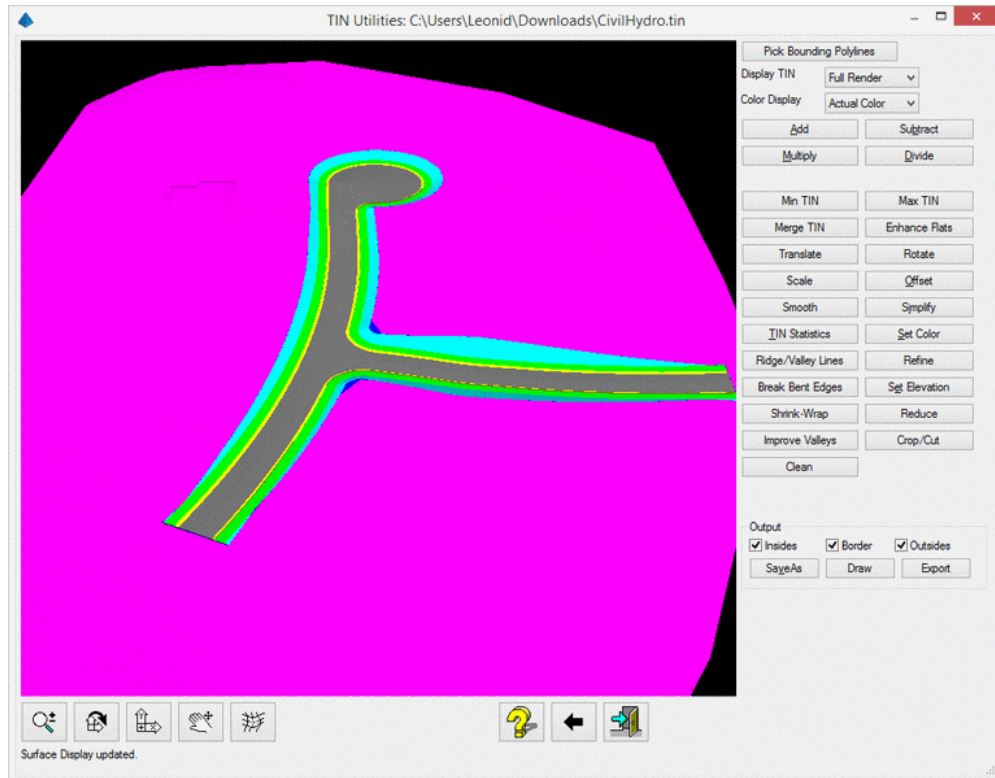
Fill-in-holes: When checked, any missing triangulation (or "gaps") in the surface will be automatically filled in with additional triangles. This option has to be set *before* loading the TIN file to take effect.

Region Mode: This option deals with nested or overlapping boundaries. When checked, "hatch pattern logic" is applied in which all nested boundaries are used in an alternating fashion, so that an Inclusion Boundary within an Exclusion Boundary is still recognized. If this option is not checked, everything within an Exclusion Boundary is ignored.

Next: Press this button to proceed to the next dialog after all selections have been made.

The next dialog allows you to perform mathematical operation(s) on the loaded TIN. Each operation is described below. Keep in mind that generally these operations are to be performed on an area inside your inclusion perimeter (but excluding anything inside your exclusion perimeters). If you do not specify any perimeters, the desired

operation(s) will be performed on the entire TIN.



Add: Adds to the TIN elevations using either a Value or another TIN. For Value, it prompts for a value to Add to the elevations of the subject area of the TIN. The TIN method raises the subject area of the current TIN by the elevation value from a second user-selected TIN file. This function is most applicable to applying a strata thickness (e.g. landfill cover) on top of a TIN to get a new target surface elevation.

Subtract: Subtracts from the TIN elevations using either a Value or another TIN. For Value, it prompts for a value to Subtract from the elevations of the subject area of the TIN. The TIN method lowers the subject area of the current TIN by the elevation value from a second user-selected TIN file.

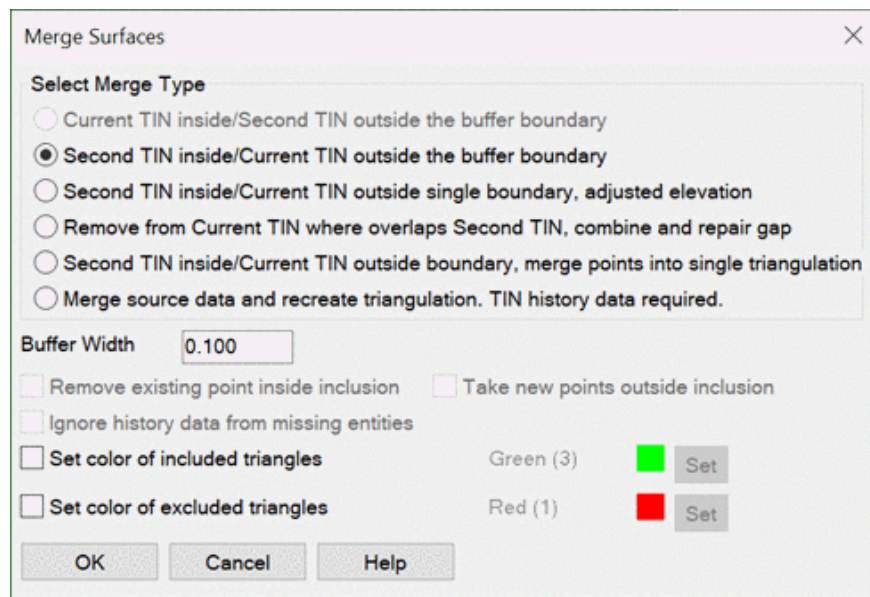
Multiply: Multiplies the TIN elevations by either a Value or another TIN. For Value, it prompts for a value to Multiply to the elevations of the subject area of the TIN.

Divide: Divides the TIN elevation by either a Value or another TIN. For Value, it prompts for a value to Divide into the elevations of the subject area of the TIN.

Min TIN: This does a comparison between the current TIN and a second user-selected TIN file and applies the lower value of the two TINs to the subject area.

Max TIN: This does a comparison between the current TIN and a second user-selected TIN file and applies the higher value of the two TINs to the subject area.

Merge TIN: Merges the current subject TIN into a second user-specified TIN file. There are six methods to merge tins. The methods for merging effect the way the gaps and elevation differences between the TINs are handled. If the two TINs being merged meet within a relatively close tolerance, the different methods will have little effect. In situations where the two surfaces being merged vary significantly, ie. an asbuilt of an area where the existing ground 'tie in points' are also disturbed, the method of merging will be more obvious.



1. **Current TIN inside/Second TIN outside the buffer boundary:** This method is only available when Bounding Polylines are selected in the first Triangulation File Utilities dialog. The current TIN will be used inside the boundary polylines and the second TIN is used everywhere else. Any gap or elevation difference between the two TINs are directly joined and interpolated within the confines of that gap. The current TIN file should be the smaller of the two surfaces since the subject file will be joined or merged into the second file. For example, to merge a pad design into existing ground with this method, choose the pad design as the current TIN, pick the pad perimeter as the bounding polyline and use existing ground as the second TIN. This method is effective if there is little difference between the two TINs.
2. **Second TIN inside/Current TIN outside the buffer boundary:** This method uses the second TIN inside the boundary and the current TIN everywhere else. The outline of the second TIN is used as the boundary if no bounding polylines were selected in the initial dialog. Any gap or elevation difference between the two TINs are directly joined and interpolated within the confines of that gap. For example, to merge a pad design into existing ground with this method, choose the existing ground as the current TIN and choose the pad design as the second TIN. This method is effective if there is little difference between the two TINs.
3. **Second TIN inside/Current TIN outside single boundary, adjusted elevation:** This method removes triangles from the current TIN for areas that overlap the second TIN and then adjusts the elevations to meet the second TIN edges. This method creates a smoother transition between the two TINs but will densify the triangles along the edges. For example when blending an As Built TIN into an previously surveyed area, choose the As Built TIN as second.
4. **Remove from Current TIN where overlaps Second TIN, combine and repair gap:** This method removes triangles from the current TIN for areas that overlap the second TIN. Then the second TIN is added into the current TIN surface and the gap between the current and second TINs is triangulated to stitch them together. This method is useful when the two TINs don't have matching elevations on their common boundary. This method will create a transition zone between the TINs but creates more distortion along the first TIN but will have less dense triangles than the *adjusted elevation* method.
5. **Second TIN inside/Current TIN outside boundary, merge points into single triangulation:** This method merges the two TINs by triangulating a new surface from the point data from each TIN, eliminating any overlapping areas. This method will create all new triangles for the entire surface so any previous edits (ie. swapped edges) will not be maintained.
6. **Merge source data and recreate triangulation. TIN history data required.** This method merges two TINs and recreates a new surface utilizing any edits made and recorded in the TIN history file. This is method

is effective for users who have performed particular edits to either TIN that may not be contained in the underlying data.

Set color of Included Triangles: When enabled, the color of triangles included in the merge are set as specified.

Set color of Excluded Triangles: When enabled, the color of triangles excluded from the merge are set as specified.

Enhance Flats: There are two methods:

1. **Interpolate Ridges, Valleys, Summits:** This routine eliminates flat triangles by adding a data point inside the triangle at a different elevation to subdivide the triangle. The elevation of this point is calculated based on the slopes of the neighboring triangles.

2. **Swap Edges to Minimize Flat Triangles:** This routine swaps edges of flat triangles when it can find a neighboring point with a different elevation and still maintain the triangulation.

Translate: This function has two methods:

1. **Delta:** This method moves the TIN points by the specified delta x, y and z.

2. **Geoid:** This method adjusts the elevations by the selected geoid. To use the Geoid method, the grid projection for the drawing must be defined in the Drawing Setup command.

Rotate: Rotates the TIN points by the specified rotation angle around a base point.

Scale: Scales the TIN points by the specified scale factor from a base point. The scale can be applied to just the x,y of the TIN points or the z, too. As an example, the Scale feature can be used to convert a TIN between Imperial and Metric units.

Offset: Performs a perpendicular offset (from the face/s) to the TIN surface by the specified amount. The routine offsets each point in the TIN vertically by looking at the slopes that connect to the point. For points at slope transition points such as at the bottom of a ditch, these corner points are effected by both slopes which means the program can't hold either exactly. To hold both slopes exactly would require changing the x/y position of the TIN points which this routine doesn't do to avoid more complications. So if the offset surface needs to exactly hold the slopes, then use another method like extracting Cross-sections from the Surface, offsetting these sections, creating 3D polylines from Sections and modeling the 3D polylines.

Smooth: There are three methods to smooth the triangulation surface.

1. **Moving Least Squares:** Smooths the surface by adjusting the elevations using a moving least squares method.

2. **Subdivide:** Performs a subdivision on the triangles within a TIN for the intent of developing a smoother rendition of the surface. This option is commonly used in scenarios where surface models are derived from man-made data (e.g. surfaces from contour maps, on-line sources, etc). The routine starts to subdivide each triangle "n" times with each iteration finding the centroid of the triangle. New triangles are formed using this centroid location and the process continues.

3. **Polynomial:** This option uses a polynomial interpolation in an effort to add additional points into the surface model in an effort to produce a more "rounded" (less man-made) surface.

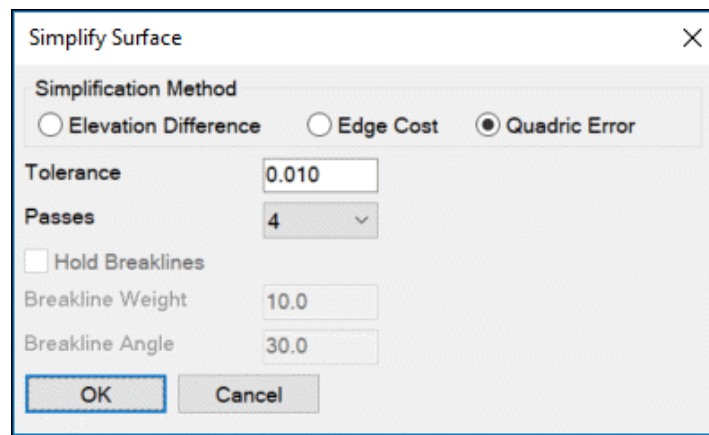
4. **Laplace:** This method uses the Laplacian method that adjusts each vertex using the positions of the neighboring vertices.

Simplify: Causes edges within the TIN mesh to be collapsed to reduce the number of triangles, edges, and points within the TIN while having a minimal impact on the overall shape of the TIN. There are three methods:

1. **Elevation Difference:** Looks at the effect of removing a point from the TIN. The point is removed if the elevation difference between the original point and the updated TIN is less than the tolerance.

2. **Edge Cost:** Looks at the effect of removing an edge from the TIN.

3. **Quadric Error:** Looks at vertex pairs and minimizes surface changes using quadric matrices.



Tolerance: This setting is used by both of the Simplification options described above. Specify the maximum average distance that any point can be moved outside of the plane of any triangle that connects to that point. Values might range from .01 to .1 for most purposes.

Passes: For the Elevation Difference option, this is the number of times the program will check through all the points.

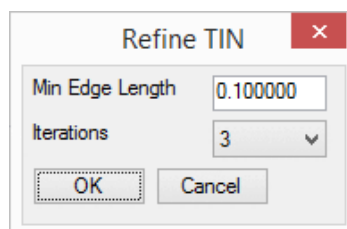
Hold Breaklines: Further analyzes the TIN by focusing on the edges, calculating the angular difference between adjacent triangular faces. If the angular difference between edges is greater than the specified **Breakline Angle**, it is considered to be a breakline, and it is preserved. If its angular difference is determined to be below the **Breakline Angle**, it becomes a candidate for removal. In that case, the **Breakline Weight** factor is applied to the corresponding vertex, adjusting its original value. If the resulting value is still below the **Tolerance**, it is then removed. The number of vertices removed is inversely proportional to the **Breakline Weight** factor, so the greater the **Breakline Weight** factor, the fewer vertices that are removed, the lower the **Breakline Weight** factor, the more vertices that are removed.

TIN Statistics: Generates a report of the TIN statistics, including number of points, edges, and triangles, and minimum and maximum Z values.

Set Color: Assigns a selected color to the TIN triangles. The color is used when viewing the TIN file in the 3D viewer commands. To set the color within an inclusion perimeter, use the Pick Boundary Polyline option.

Ridge/Valley Lines: This option permits you to select previously drawn polylines that represent surface model breaklines to enhance the approximate appearance or effect of ridges and/or valleys within the surface model. A more precise procedure would be to first utilize the 3D Polyline routine and re-create the surface model by specifying the 3D polylines within the Triangulate & Contour routine.

Refine: This function adds points to the triangulation to make more uniform triangles and fewer skinny triangles. The Min Edge Length prevents adding points that result in an edge length below this minimum. The Iterations is how many passes the routine does through the triangulation.



Break Bent Edges: Used commonly in scenarios for a TIN created from contour entities, there is a likelihood of areas where horizontal triangles (also known as "flat triangles") adjoin sloped triangles. The routine will detect such edges, add an additional point on the horizontal triangles and assign its elevation representing the trend of the surface, by extrapolation, looking to restore presumed surface data that was lost in recreating the TIN from

contours. This methodology provides additional corrective options to that of the Minimize Flat Triangles option of the Triangulate & Contour routine.

Set Elevation: This button has four functions.

Set New Elev: Sets all TIN faces in the subject area to the elevation specified.

Set NULLs to Elev: Sets all NULL (empty) values in the subject area to the elevation specified.

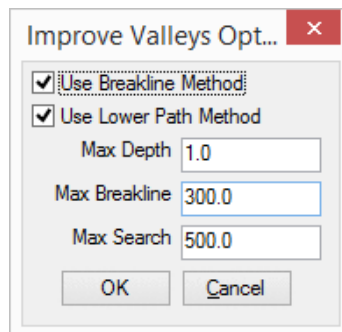
Set Elev to NULL: Sets all of the elevation values in the subject area to NULL (empty).

Set Elev by Surface: Sets all TIN faces within the subject area to the elevations from a second surface file within the same area. You will be prompted to select a second TIN file or grid file. Only areas common to both surfaces will be applied to the subject TIN.

Shrink-Wrap: This function removes skinny triangles around the perimeter without causing any triangulation points to become disconnected. There are three reduction levels: low, medium and high. The low removes the fewest triangles and high the most.

Reduce: This function removes triangulation points that are within the specified distance and then re-triangulates the surface.

Improve Valleys: The purpose of this function to clean up the bottom of the valley to permit unrestricted water flow for watershed analysis routines. The typical TIN may include small ridges, crossing the very bottom of the valley, obstructing the flow. There are two separate clean up methods in this function: the breakline method and the low path one.



Max search distance controls how far the program looks for another low point to try to connect the flow to.

- The **Breakline Method** improves TIN surface by inserting breakline segments between each eligible sink and the closest lower point to that sink such that the highest crest point between these two points is not higher than a given height tolerance (max depth) above the sink. Maximum breakline is a limit to the length of used breakline.
- The **Lower Path Method** works by finding optimal paths between neighboring sinks, and lowering the necessary points between them to allow water to run from one to the other. The maximum depth controls the maximum amount a point along the path can be lowered by.

Crop/Cut: This function removes the triangulation by a closed perimeter polyline. The routine embeds the polyline into the triangulation surface using the elevations of the triangulation. You can either remove the triangulation

inside the perimeter (Cut), or remove the triangulation outside the perimeter (Cut).

Clean: This function removes skinny and tiny triangles that have an edge less than the specified Distance Tolerance. The Spike Removal option removes steep V-shaped ditches or ridges in the surface. A Level of High removes more than Low.

Low Points: This function draws symbols at low points in the TIN which can be helpful for finding spots where rainfall will collect. There is a dialog with settings for the symbol to draw, size and layer.

Output Options

The following three options determine what part or parts of the TIN modifications that will be saved to the new TIN file. If the entire TIN is to be saved, all three options should be enabled.

- **Inside:** If this is the only option checked, only changes made to the TIN within the inclusion perimeter will be saved. TIN entities outside of the perimeter will not be saved to the named file.
- **Border:** When the routine re-works the TIN to fit around a perimeter, a small horizontal offset is automatically applied to prevent the formation of vertical faces. The Border function will save changes made to TIN in this offset area.
- **Outsides:** If this is the only option checked, TIN entities inside of the inclusion perimeter will not be saved to the named file. Everything outside of the perimeter will be saved.

Save: Saves the current TIN as an .flt or .tin file.

Draw: Draws the current TIN as 3D Faces in the current viewport. The Layer window is used to specify the layer that the faces will be created in.

Export: There are three export formats:

1. Export To Text (points only): Writes the TIN point locations to a user-specified ASCII-based coordinate file.
2. Export To Text (points and triangles): Outputs a text file of the points and triangle indexes.
3. Export To DXF: Saves the current TIN as a .dxf file. This format can be used by many other CAD programs.

Control



Action

Converts the left mouse button to a zoom function. Hold the button down and move the mouse up or down to zoom in and out.

Converts the left mouse button to a rotate function. Hold the button down to rotate the view in any X, Y or Z direction. When the XY appears in the window, the rotation will occur relative to the XY axis. When the mouse is moved toward the outer perimeter of the window, the XY will change to a Z. Holding the button down while the Z is visible will rotate the drawing on the Z axis.

Restores the graphics to plan view.

Converts the left mouse button to a pan function. Hold down on the button while moving the mouse to pan. Holding down the mouse wheel will also serve as a pan function in any of the above modes.

Toggles shading on and off.

Reverses the effects of all operations performed on the TIN and reverts it back to its original status.



This icon exits the routine. If the TIN has been modified, you will be prompted to save.

Usability Controls

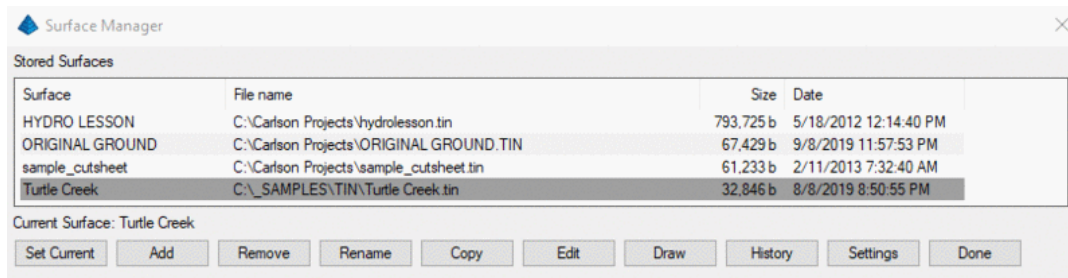
Pulldown Menu Location(s): Surface

Keyboard Command: tinutil

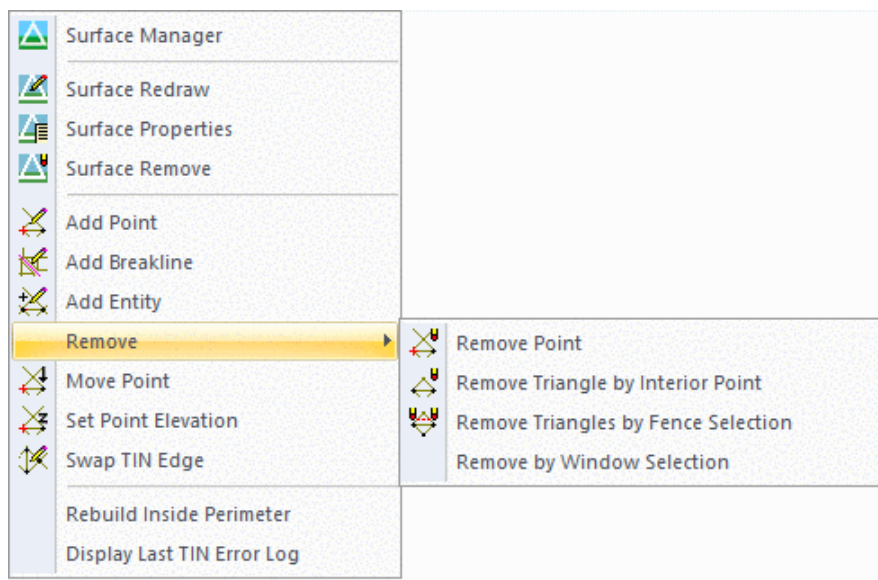
Prerequisite: 3D Faces, a TIN file or a DXF file containing 3DFACE entities.

Surface Manager

The Surface Manager toolkit allows you to modify pre-defined triangulated surfaces, making real-time modifications and updates to contours and associated TIN (Triangulated Irregular Network) definitions. Functionality includes swapping TIN lines, adding or removing (untagging) breaklines, adding or removing points by range or group, adjusting point elevations, removing TIN lines, drawing or removing contour lines and labels, re-contouring at a different interval or with different label settings, etc. Contour lines are automatically updated to reflect any changes made to the TIN. A surface must be named and saved by one of the surface modeling routines (in the Triangulate tab) as a prerequisite to using the Surface Manager.



All of the tools available in the Surface Manager are also available in the fly-out menu, as shown in this figure. Their functions are identical but require a surface to be set current. Changes made apply only to the current surface.



Set Current Designates a surface as current for editing with various surface tool functions, such as modifying TIN lines, setting a new contour interval, labeling contours, etc.

Add Allows you to add a surface by selecting a surface model file (.TIN or .FLT).

Remove Allows you to remove a surface from the list of stored surfaces.

Rename Allows you to rename a surface.

Copy Creates a copy of the TIN file and adds the copy as a new entry.

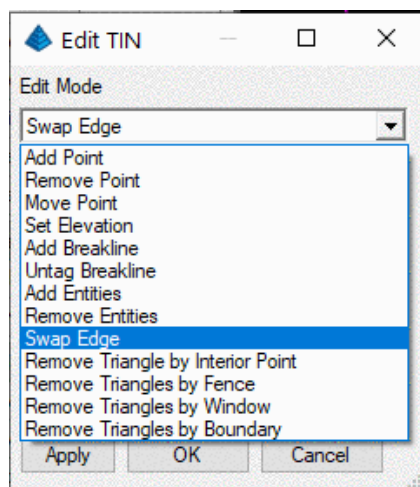
Edit Displays various edit tools. Clicking on any of these commands will activate the command line, where you will be prompted based on the Edit command selected.

Draw Allows you to alter the drawing display properties for TIN lines, contours and labels for the selected surface. Applicable dialogs from Triangulate and Contour are used to provide a full set of options. When accessed, settings for the current surface display configuration are set. To make a modification, simply specify the desired change and press ok. For instance, if Draw Triangulation Lines was checked on, unchecking the box and pressing ok will redraw the surface without the TIN lines. If the contours were drawn at 1 foot intervals, setting the interval value to 2 and pressing OK will redraw the contours at 2 foot intervals. Refer to the Triangulate and Contour section of the manual for a more detailed explanation of the options.

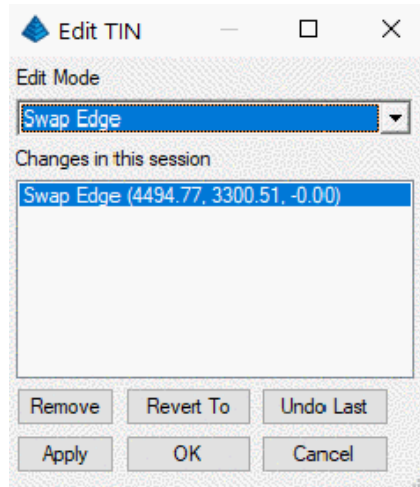
History The Triangulation History Editor displays all functions performed during the creation and editing of a TIN file. Each edit can then be eliminated from the process without recreating the TIN from scratch. In cases where the user does recreate the TIN, any edits previously performed can be recalled and re-implemented.

Settings Allows you to set the triangle edge and breakline Color and Transparency for easier editing. Many times the triangle edges or breaklines can get lost or match colors for say the contours and this allows you to better differentiate the items when editing.

The dialog for this function has the pulldown with the choice of the editing operation. You can pick it from the pull-down or type the corresponding keyword at the command line to switch.



As the operations are performed, they are recorded in the log.



Remove allows to remove any operation performed in this session. The TIN will be re-processed from the beginning of the session, skipping the edit being removed

Revert To removes all the edits following the edit selected. Allows the user to go back to last good edit and after re-processing continue editing

Undo Last removes the last edit and reprocesses TIN

Apply saves the current TIN as edited and continues editing

Operations:

Add Point (AP) prompts to pick point or point number to add [Group/Range]. This allows you to add triangulation point(s) to the network by either graphically picking points from the screen; typing a point number; or by adding an entire (R)ange or point (G)roup to the TIN. You can add points to the inside or the outside of the TIN. The elevation for graphically selected points is interpolated from the surrounding TIN network. This is a good method for adding additional triangulation to the surface in a sparse area. Also, a new elevation can be specified for the picked point. There is an option for whether to store the graphically selected point to the coordinate file. When 3D polylines are created by Field to Finish and you add point by range or point group that covers the point numbers used to make the 3D polylines, then the segments from these 3D polylines are used as breaklines in the TIN.

Remove Point (RP) removes an existing triangulation intersection from the TIN network. The affected triangulation re-adjusts to compensate for the missing intersection. Contours update accordingly.

Move Point (MP) is a combination of removing a point and adding it at a new location.

Set Elevation (SP) sets a new elevation for a specified TIN intersection. The affected TIN is adjusted and the contours are updated.

Add Breakline (AB) adds a breakline to the surface by picking beginning and ending points on the screen. The endpoint snap automatically turns on. Only one breakline can be created at a time. The TIN network will reconfigure to follow the new breakline and update the contours. This does not create 3d polylines in the drawing.

Untag Breakline (UB) removes breaklines from the TIN that were either added through Add Breakline Edit

command or through the initial selection of lines in the drawing.

Add Entities (AE) adds selected points and breaklines into the TIN by graphical selection of existing entities on screen.

Remove Entities (RE) removes selected points and breaklines from the TIN by graphical selection of entities used in its creation.

Swap Edge (SW) swaps common TIN edges to create two different triangles from the original triangle configuration. Contours automatically update to reflect changes made to the TIN. Some common edges may not be swapped because of the orientation of the two triangles.

Remove Triangle by Interior Point (RT) removes a TIN line from the surface by picking a TIN line or selecting an interior point. Contours are removed from the affected area.

Remove Triangles by Fence (RF) removes existing triangles that touch the fence as it is being drawn across the triangles.

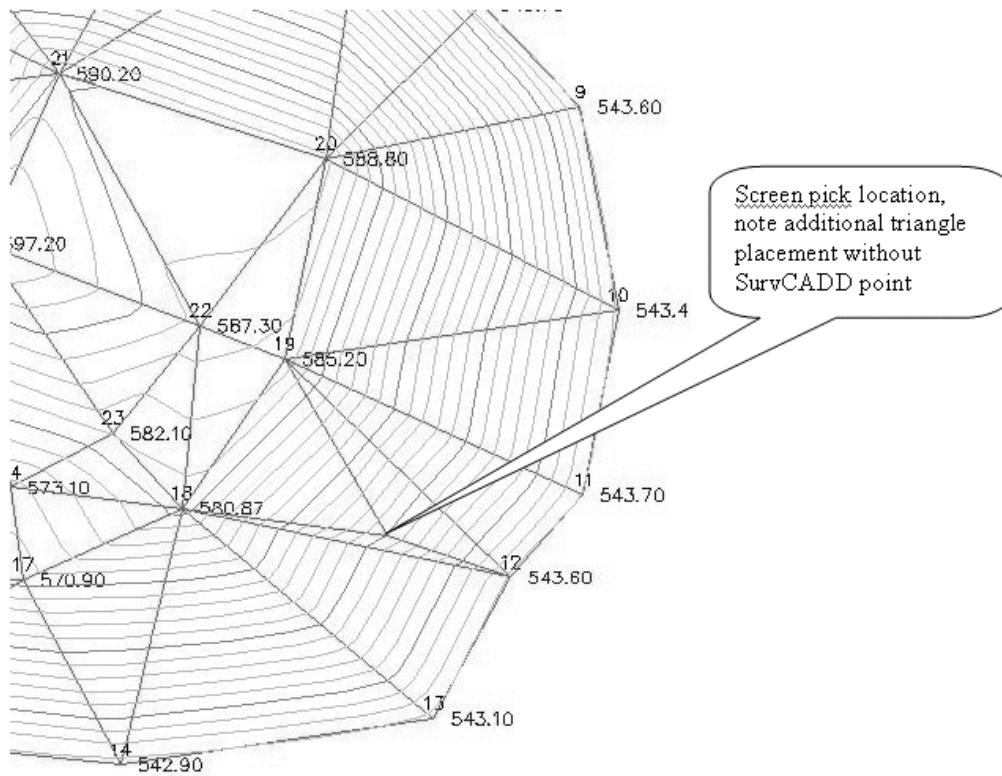
Remove Triangles by Window (RW) removes existing triangles using a crossing-type window whereby any triangles that touch or are contained within the window when drawn will be removed.

Remove Triangles by Boundary (RB) removes existing triangles that touch or are contained within a selected closed boundary or polyline.

Add Pnt(AP), Move Pnt(MP), Set elev(SP), Add Breakline(AB), Untag Breakline (UB), Add Entities(AE), Swap edge(SW), ID Pnt(ID), Show/Hide Tris(ST), Remove Pnt(RP), Remove Tri(RT), Remove Entities(RE), Remove by Window(RW), Remove by Fence(RF), Remove by Boundary(RB), Update Tris(UT) Press Enter to end. Press Esc to cancel.

Add Pnt(AP)

Pick point or point number to add [Group/Range]: *Graphically pick points from the screen; type a point number; or add an entire (R)ange or point (G)roup to the TIN. When graphically selecting a point, you will be prompted for the elevation of the new point, and if you would like to add the new point to the current coordinate file.*



Move Pnt(MP)

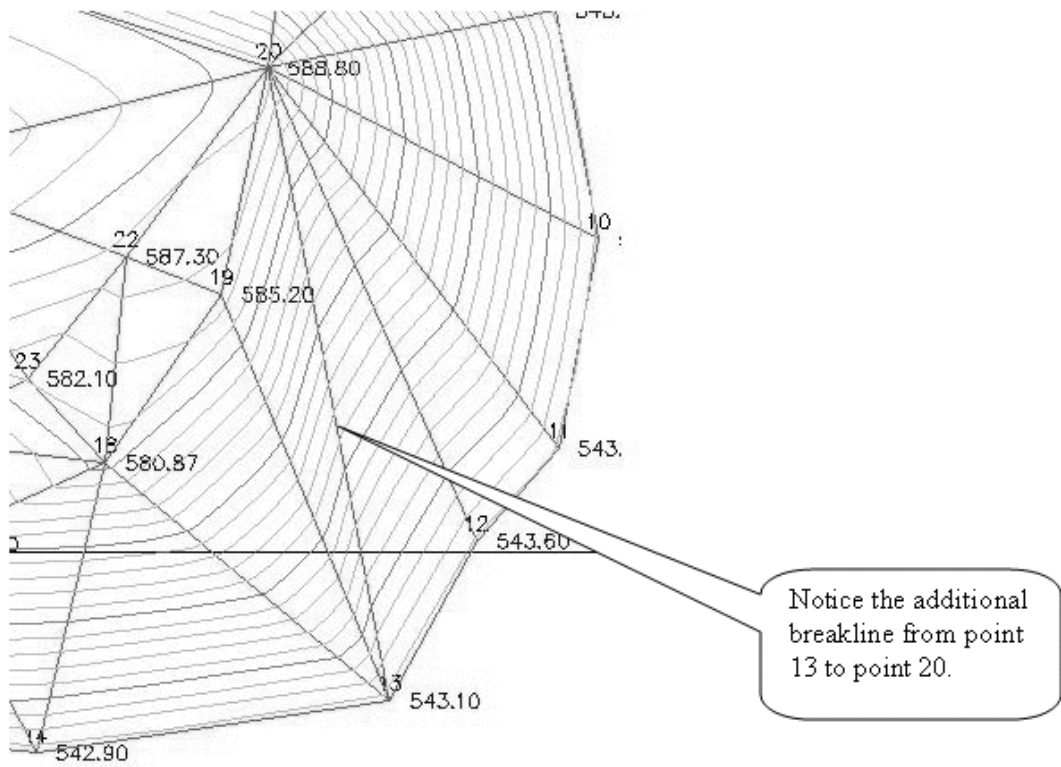
Pick near point to move: *Graphically pick near a point to move. Carlson will take that point and place it in a new location when prompted.*

Set elev(SP)

Pick near point to set elevation: *Graphically pick near a point to change the elevation and you will be prompted for the new elevation.*

Add Breakline(AB)

Pick point or point number for 1st breakline point: *Graphically snap 2 points on the screen for the new breakline.*



Untag Breakline(UB)

Select an breakline edge to untag: *Graphically select a triangle edge that represents a breakline to remove.*

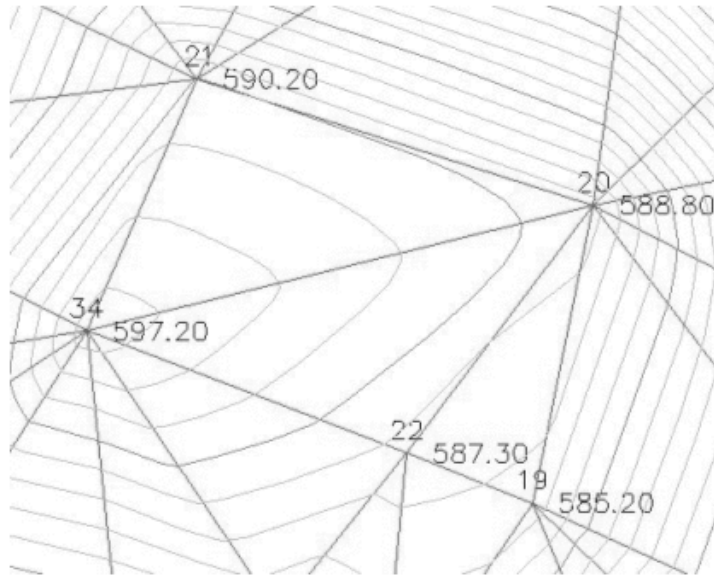
Add Entities(AE)

Select points and breaklines to add to triangulation. FILTER/Select entities: *Graphically select entities to add to the TIN by pick or window selection.*

SWap edge(SW)

Select an internal edge to swap: *Graphically pick near a Triangle Edge to swap. This will switch the direction of two triangles sharing the same edge.*





ID Pnt(ID)

(Prompt is from current edit command) *The coordinates of the nearest triangle vertex are displayed on the commandline (Point: x=, y=, z=).*

Show/Hide Tris(ST)

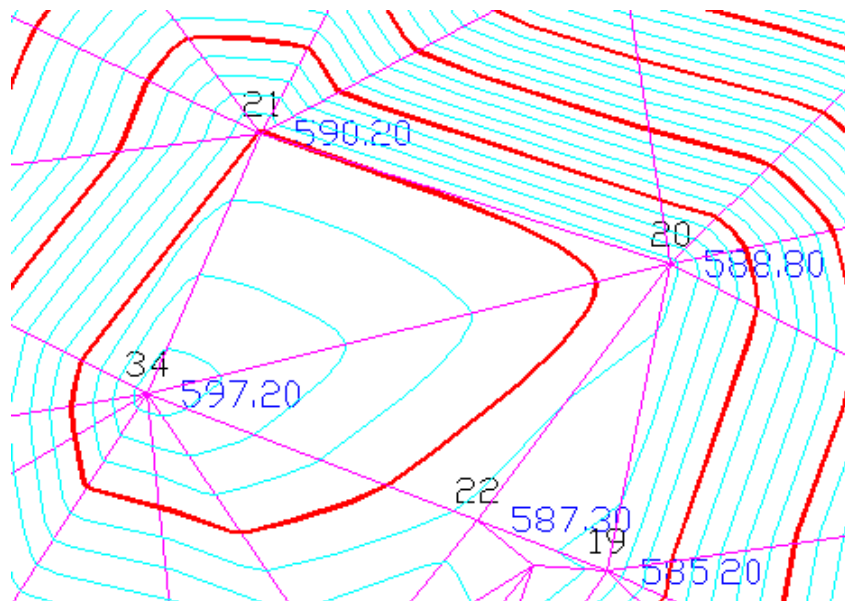
(Prompt is from current edit command) *Temporarily toggles the display of the triangles ON or OFF the screen for easier editing.*

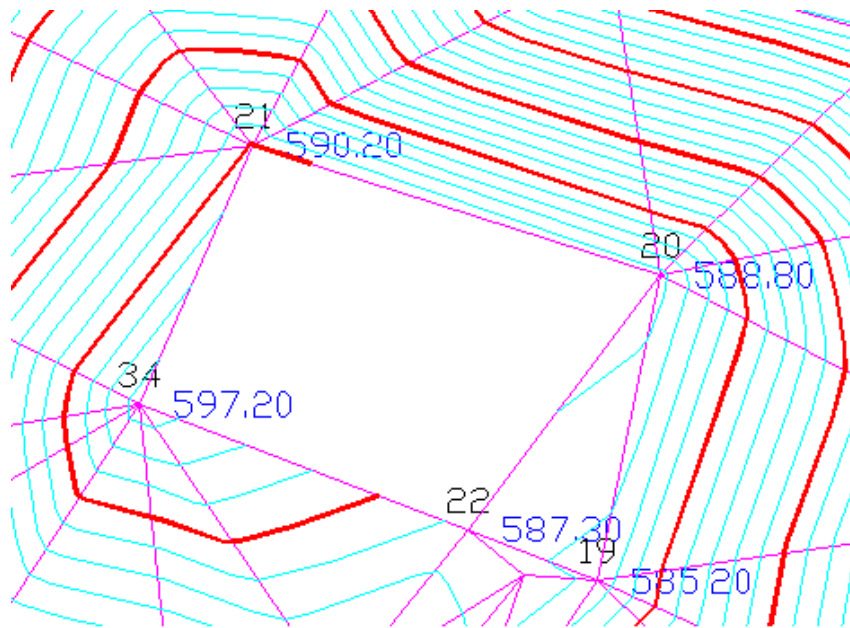
Remove Point(RP)

Pick point to remove: *Graphically pick near a point you wish to remove from the TIN.*

Remove Tri(RT)

Removing Triangles, Pick point or enter keyword: *Graphically pick inside a triangle you wish to remove.*





Remove Entities(RE)

Select entities to remove from triangulation. FILTER/Select entities: Graphically remove breaklines, points, etc. that were used in the creation of the TIN that you want to remove from the TIN. This will reform the TIN without the selected entities.

Remove by Window(RW)

Pick 1st corner of area to remove: Graphically select 2 points of a window. Any TIN triangles inside or touching the window selection will be removed from the TIN.

Remove by Fence(RF)

Pick beginning of fence: Graphically select points on a screen to create a fence. Any TIN triangles touching the fence selection will be removed from the TIN.

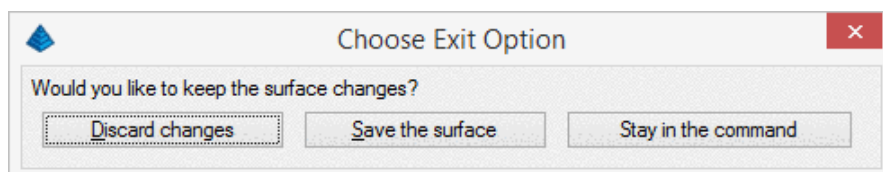
Remove by Boundary(RB)

Select boundary polyline: Graphically select a closed polyline. Any TIN triangles inside or touching the polyline will be removed from the TIN.

Update Tris(UT)

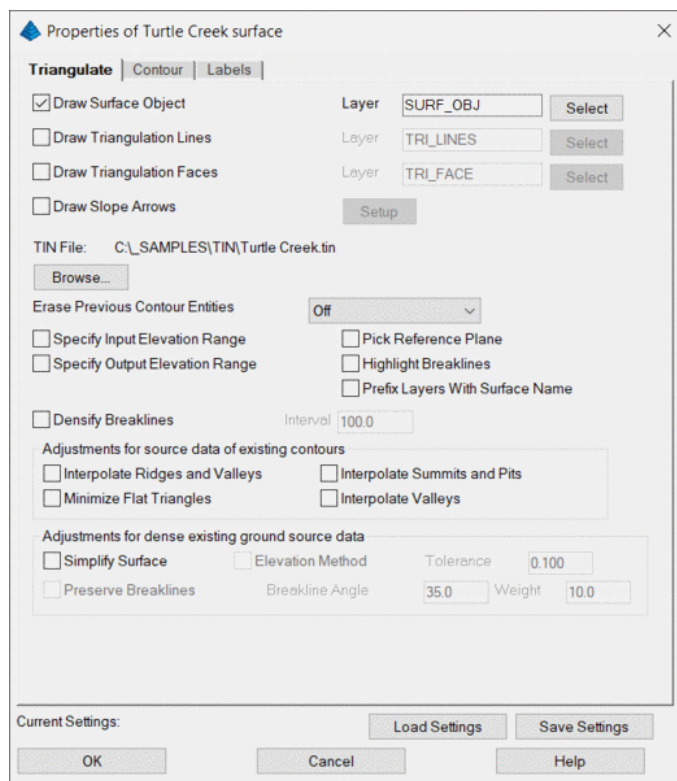
(Prompt is from current edit command) Refreshes the TIN drawn on the screen from the TIN file.

To conclude the Surface Edit mode, press Enter at the end of the internal command sequence. This will return to the Surface Manager dialog. If you press Escape key instead, the following dialog is displayed:

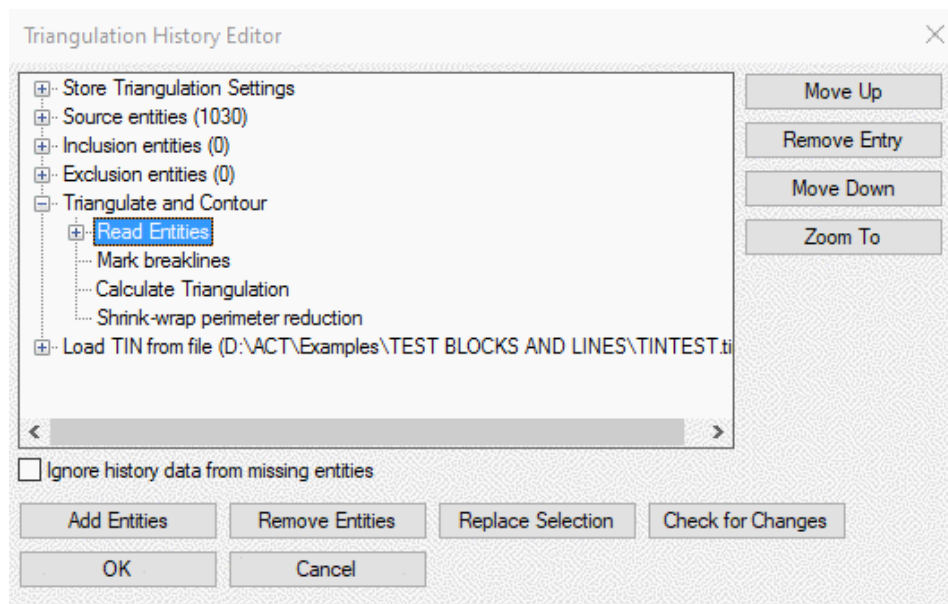


This prevents you from accidentally undoing all of your changes by unintentional pressing the ESC key.

(Draw) Properties allows you to draw the surface and control the properties for the TIN lines, contours and labels for the selected surface. Applicable dialogs from Triangulate and Contour are used to provide a full set of options. When accessed, settings for the current surface display configuration are set. To make a modification, simply specify the desired change and press ok. Refer to the Triangulate and Contour section of the manual for a more detailed explanation of the options.



The Triangulation History Editor displays all functions performed during the creation and editing of a TIN file. Each edit can then be eliminated from the process without recreating the TIN from scratch. In cases where the user does recreate the TIN, any edits previously performed can be recalled and re-implemented. For example, if you have added several breaklines and later decide that they were unnecessary, highlight the added breaklines and click Remove Entry. You can also change the order in which an edit appears, such as the application of an exclusion boundary toward the end of the list of edits.



Move Up / Move Down moves the selected edit up or down in the chronological list of edits.

Remove Entry removes an edit from the TIN.

Zoom To centers the CAD screen on the location of the selected edit.

Ignore history data from missing entities If toggled OFF, all history data in list will be used to process the TIN. If toggle ON, any entities not available will not be used to recreate the TIN, and therefore will be removed until the missing TIN entities are restored.

Add/Remove Entities allows you to select new items to add to the TIN or select existing items to remove from the TIN.

Replace Selection allows you to replace a TIN edit in the history with a new edit.

Check for Changes will read the TIN file and check to see if any changes have been made that are not currently reflected in the current TIN file.

Pulldown Menu Location: Surface

Keyboard Command: surface_mgr

Prerequisite: Need to have at least one TIN (.TIN/.FLT) file

Rebuild Inside Perimeter

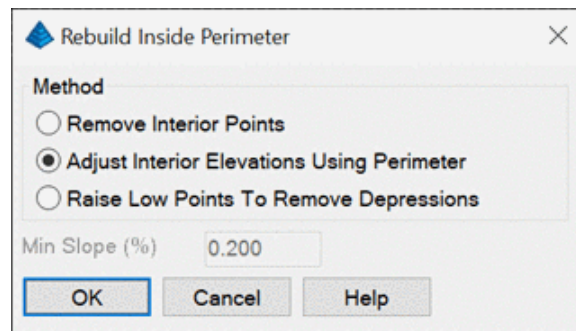
This command rebuilds a triangulation surface within a perimeter. One application of this command is to discard a problem area of the triangulation and rebuild this area using the good elevations around the perimeter of this area.

The perimeter is selected using a closed polyline. The command embeds the perimeter polyline into the triangulation using the elevations of the triangulation. There are three methods for adjusting the surface inside the perimeter:

Remove Interior Points: The surface points within the perimeter are removed. Then the area within this area is retriangulated using the elevations around the perimeter.

Adjust Interior Elevations: The elevations of the surface points within the perimeter are adjusted using a weighted average of the perimeter elevations.

Raise Low Points: The elevations of the surface points within the perimeter are raised to remove any points that are lower than the lowest perimeter elevation or make a slope lower than the specified Min Slope. This method can be used to make sure storm water drainage flows across this area.



Prompts

Select Triangulation File: Select file to load

Options dialog

Select perimeter polyline: *pick a closed polyline*

Output Triangulation File: Select file to write

Pulldown Menu Location: Surface > Triangulation Surface Manager

Keyboard Command: tin_rebuild

Prerequisite: A triangulation file and closed polyline

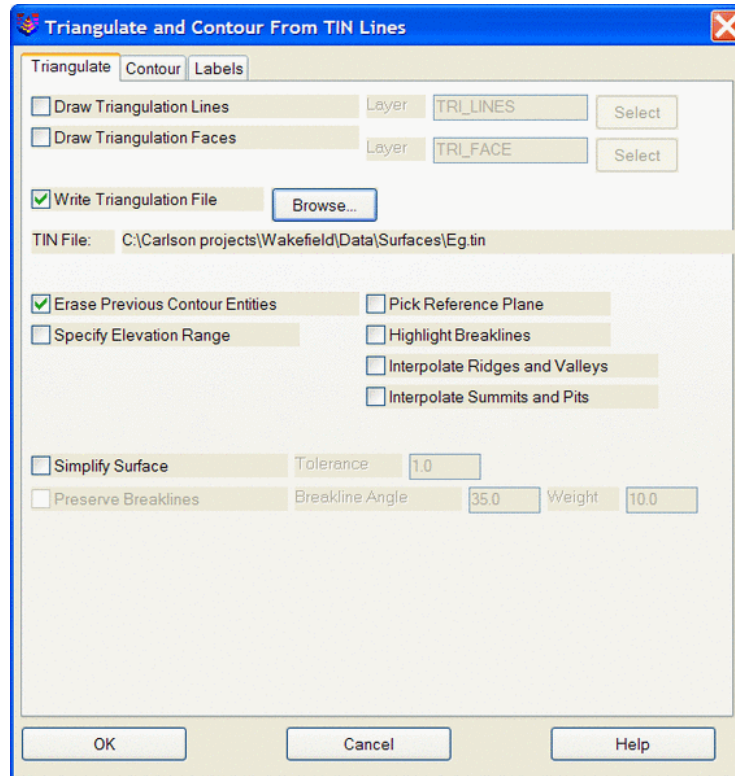
Contour from Triangular Mesh

This command creates contours directly from displayed Triangulated Irregular Network (TIN) surface features (triangles), optionally creating an external Triangulation File (.TIN) in the process. The user is prompted to pick a sample of the TIN to process. This will automatically filter out all other entities that don't reside on the layer of the displayed TIN faces. The triangles must be drawn on the screen as 3D Lines or 3D Faces. All of the settings for Triangulation, Contouring and Labeling, found in *Triangulate and Contour* are available. See the *Triangulate and Contour* section in the manual for a detailed description of each of these settings.

It is not recommended to use the "Subdivisional Surfaces" option in the Contour tab when using this routine. Use of this option with this routine may allow the internal reconfiguration of triangles that have been formed along breaklines.

Prompts

Triangulate and Contour From TIN Lines dialog box



Determining layer name for triangulation lines.

Select sample of triangulation line: *pick a tmesh line and press Enter*

Select all the triangulation lines to contour.

Select objects: *select the triangulation entities*

Reading points... 82

Contouring elevation 404

Inserted 1195 contour vertices.

The user may be prompted for additional information depending on settings used in the Triangulate and Contour dialog box.

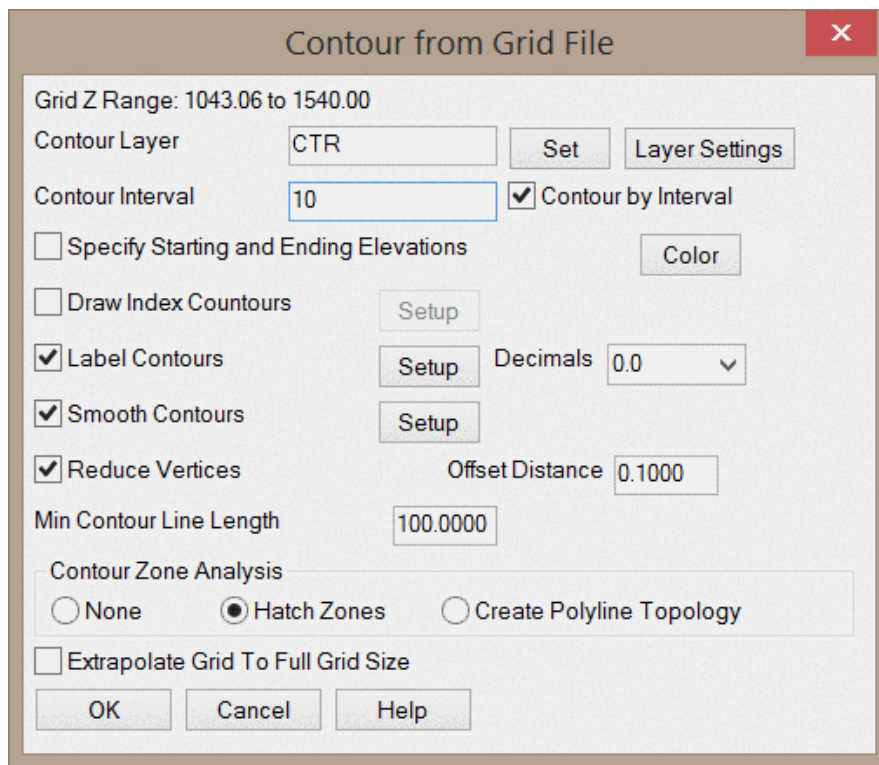
Pulldown Menu Location: Surface >> Contour from...

Keyboard Command: contour

Prerequisite: A triangulated irregular network drawn as lines or triangular 3D Faces.

Contours from Grid File

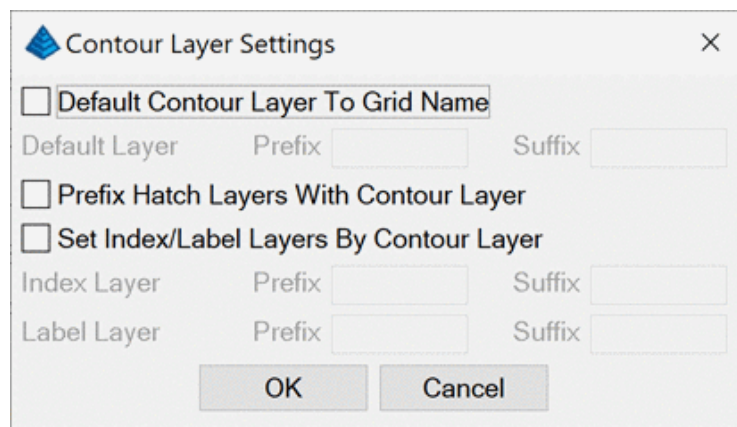
This command creates contours from a grid file. Contouring from a grid employs a different method than from a triangulation network and generally produces contours that loop more. The grid has data points at a regular interval while the triangulation has edges for every point and breakline in the surface. The smoothness of the contours depends a great deal upon the grid resolution. When the command is executed, you will be prompted to select inclusion/exclusion perimeter polylines to limit the contours to certain areas. If no perimeter polylines are selected, the entire grid will be contoured. After selecting a grid file to contour, the below dialog will appear.



Grid Z Range: This text displays the current range of Z values in the grid file to help you determine the optimal Contour Interval/Values.

Contour Layer: This sets the layer for the contours. You may select an existing layer by clicking the **Set** button.

Layer Settings: This button will open the below dialog, which controls default naming for layers of the various linework.



Default Contour Layer To Grid Name: When enabled, this toggle will automatically set the **Contour Layer** to match the grid name. For example, a grid named "Topo.grd" would default to draw contours on the "Topo" layer. The Prefix and Suffix are combined with the grid name. For example, a grid named "Topo.grd" with a prefix of "Contour" would create a layer of "Contour.Topo".

Prefix Hatch Layers With Contour Layer: When enabled, this toggle will use the **Contour Layer** as a prefix for all of the hatch layers. For example, if the first hatch zone is set to be drawn on the "Zone 1" layer and the Contour Layer is set to "CTR", the actual layer name of the hatch will be "CTR Zone 1".

Set Index/Label Layers by Contour Layer: When enabled, this option will automatically set the **Index Layer** and the **Text Layer** to be the same as the **Contour Layer**. However, you may also specify a prefix or suffix for each layer. For example, if the Contour Layer is set to "CTR" and the Index Layer Suffix is set to " Index", then the index contours will be drawn on the "CTR Index" layer. It is important to note that this option only sets these layer names when the command is first executed or when the above dialog is opened, meaning that you may still override these layer names if you wish.

Contour Interval/Values: This value is dependent on the **Contour by Interval** toggle.

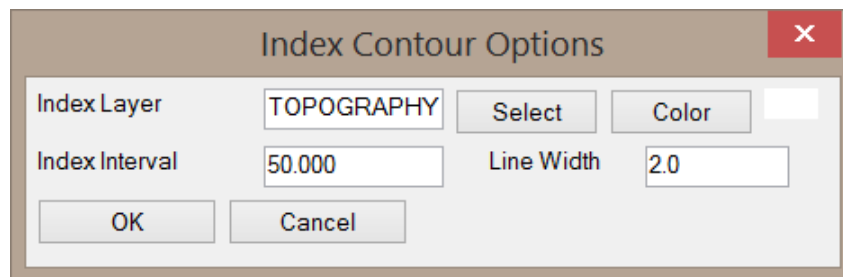
When the toggle is enabled, this value will set the interval for the contours. For example a value of 2 will draw contours along the grid values of 2, 4, 6, 8, etc. The first value contoured will be the first value divisible by that contour value. For example, if an interval of 3 is selected for a grid that ranges from 5 to 20, the first contour will be drawn at a value of 6.

When the toggle is disabled, this value(s) will set the specific value(s) to contour. For example, if a value of 10 is selected, only one contour will be drawn where the grid has a value of 10. Multiple contours may be drawn by entering values separated by spaces. For example, entering "10 12 16" into this text box will draw contours at values of 10, 12, and 16.

Specify Starting and Ending Elevations: When enabled, this toggle will prompt you to enter the starting and ending values to contour. For example, if the grid ranges in values from 0 to 100, you can use this option to only draw contours for values ranging from 30 to 60.

Color: This button will open the color palette, where you can select the color for the contours. Please note that this does not set the color of index contours.

Draw Index Contours: When enabled, this option will draw some contours differently than the main contours. You can control the setup of the index contours by clicking the **Setup** button to the right of this toggle. When clicked, the below dialog will appear.



Index Layer: This sets the layer for the index contours. You may select an existing layer by clicking the **Select** button.

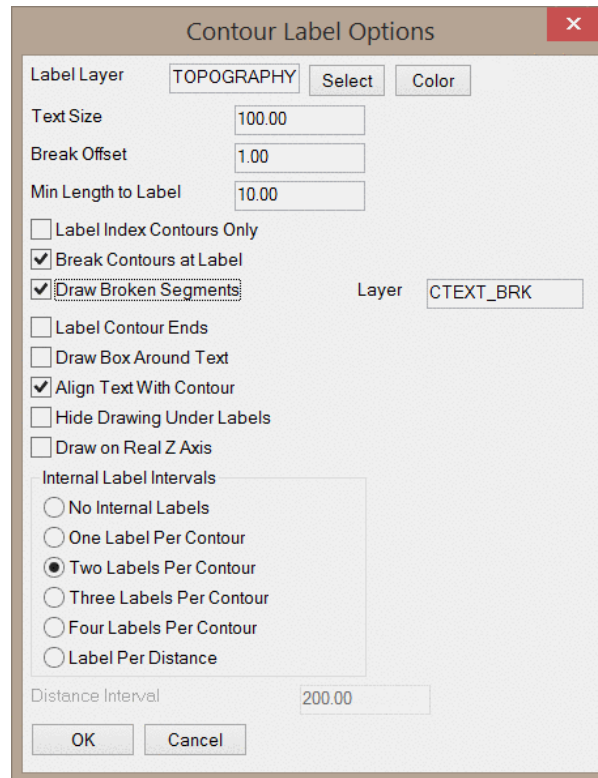
Color: This button will open the color palette, where you can select the color for the index contours. Please note that this does not set the color of non-index contours.

Index Interval: This value sets the interval of the index contours. This value will default to be the **Contour Interval** multiplied by 5.

Line Width: This value sets the width of the index contours. The non-index contours will have a line

width of 0 by default.

Label Contour Lines: When enabled, this toggle will add labels to the contours. The number of decimals in the labels is controlled by the **Decimals** dropdown list. You can control the setup of the labels by clicking the Setup button to the right of this toggle. When clicked, the below dialog will appear.



Label Layer: This sets the layer for the labels. You may select an existing layer by clicking the **Select** button.

Color: This button will open the color palette, where you can select the color for the index contours. Please note that this does not set the color of non-index contours.

Text Size: This value sets the height of the text in drawing units.

Break Offset: This value sets the distance between the end of the trimmed contour and the start/end of the text. This value is only used when the **Break Contours at Label** toggle is enabled.

Min Length to Label: This value sets the minimum length of contours to be labeled. For example, if this value is set to 10, a contour with a length of 5 will not be labeled.

Label Index Contours Only: When enabled, this toggle will only add labels to index contours.

Use MText: Sets whether to create the labels as Text or MText entities.

Break Contours at Label: When enabled, this toggle will break the contour line near the label, making it easier to read the labels.

Draw Broken Segments: When enabled, this toggle will draw the broken segments of the contour (the por-

tion of the contour under the labels). These segments will be drawn on the layer specified to the right of this toggle.

Label Contour Ends: When enabled, this toggle will draw additional labels at the ends of contours. This option will not draw additional labels on contours draw as closed polylines.

Draw Box Around Text: When enabled, this toggle will draw a box around the text labels.

Align Text with Contour: When enabled, this toggle will align the label with the contour itself. When disabled, labels will be aligned horizontally.

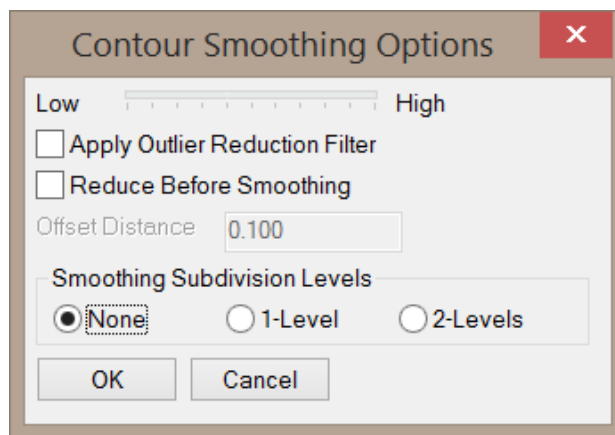
Hide Drawing Under Labels: When enabled, a WIPEOUT entity will be draw around the labels, hiding the contour and any other linework underneath it. This option has the same visual effect as breaking the contour at the label, but allows you to retain a continuous contour.

Draw on Real Z Axis: When enabled, labels will be drawn at the same elevation as the contour. When disabled, labels will be drawn at zero elevation.

--- **Labels Per Contour:** These radio buttons set the number of labels to draw on each contour.

Label Per Distance: This radio button will draw a contour label on a regular interval along the contour, set by the **Distance Interval** value.

Smooth Contours: When enabled, the contours will be smoothed to reduce sharp edges. You can control the smoothing options by clicking the **Setup** button to the right of this toggle. When clicked, the below dialog will appear.



Low-High Slider Bar: This slider bar controls the degree of smoothing. This is based on Bezier smoothing, which holds all the original points calculated from the grid and only smooths between the calculated points.

Apply Outlier Reduction Filter: When enabled, this toggle will remove spikes in the contours that don't follow the general trend direction.

Reduce Before Smoothing: When enabled, this toggle will apply the Reduce Vertices function to the contours before smoothing. The **Offset Distance** is the maximum distance the contour is allowed to move when removing vertices.

Smoothing Subdivision Levels: This option will internally subdivide the grid cells with a quadratic smoothing algorithm to help create smoother contours.

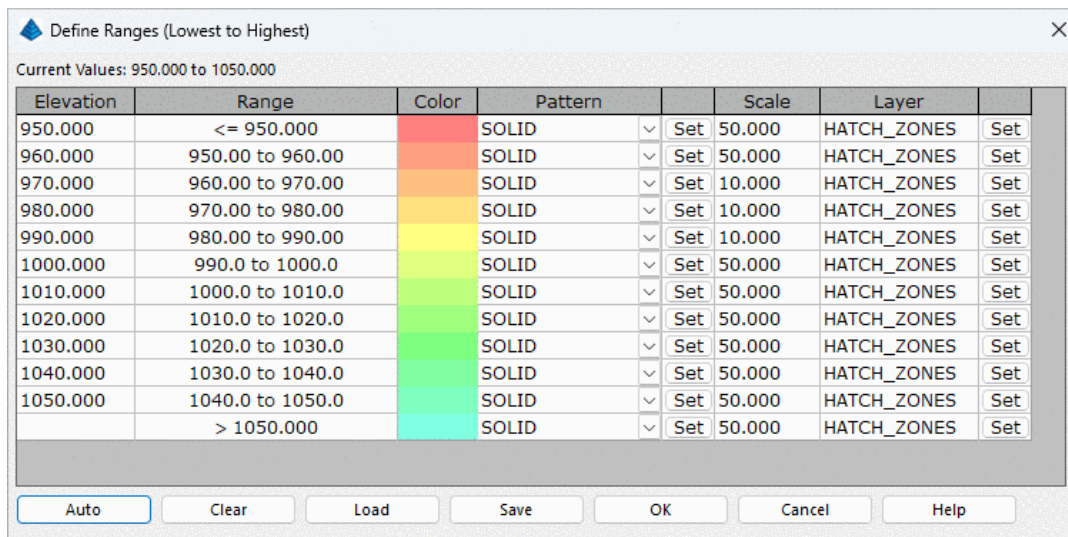
Reduce Vertices: When enabled, this toggle will apply the Reduce Vertices function to the contours. The **Offset Distance** is the maximum distance the contour is allowed to move when removing vertices.

Min Contour Line Length: This values sets the minimum length required for contours to be drawn. For example, if this value is set to 100, a contour with a length of 50 will not be drawn. This filter helps to prevent drawing relatively insignificant contours.

Contour Zone Analysis: This option controls how the contour zones will be drawn. A zone is simply the space between two contour lines.

None: The contour zone boundaries will not be drawn. Only the contours themselves will be drawn.

Hatch Zones: This option will draw the contour zone boundaries and hatch them according to the below dialog. When this option is selected, you will be prompted to pick a location to draw a legend. It is recommended that you select an inclusion boundary when using this option even if you are contouring the entire grid.



The above dialog controls the appearance of the hatch zones. A description of each column is given below. Note that you may need to scroll down to other pages to view all of the contour zones.

Elevation: Sets the maximum contour value for the zone.

Range: Displays the range of values included in the zone.

Color: Sets the hatch color for the zone.

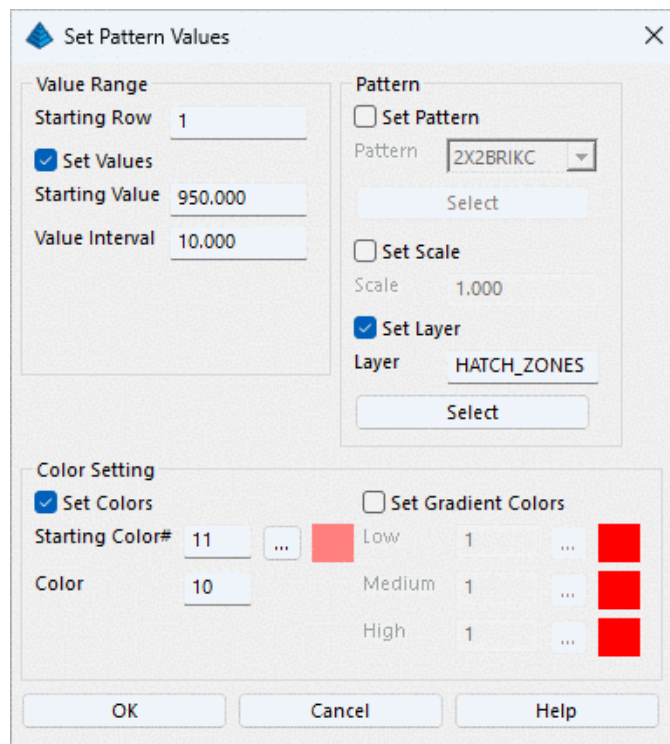
Pattern: Sets the hatch pattern for the zone.

Scale: Sets the hatch pattern scale for the zone.

Layer: Sets the layer for the hatch pattern.

Clear: This button will clear all settings in the dialog.

Auto: This button will display the below dialog, which will let you automatically populate each hatch zone with various properties.



Starting Zone: This value sets the first zone to be modified. If this value is set to 1, all zones in the dialog will be modified. You can toggle the various options on this dialog to only modify certain properties of each zone, however.

Set Values: When enabled, the values of each zone will be modified. The **Starting Value** sets the value of the first zone selected. That value will increment by the **Value Interval** value.

Set Colors: When enabled, the colors of each zone will be modified. There are two ways to set the colors of the zones, controlled by the **Set Gradient Colors** option.

When **Set Gradient Colors** is disabled, the colors for each zone will be defined by a **Starting Color #** and a **Color Increment**. The Starting Color # relates to a color in the CAD color palette. The Color Increment sets the color for the next zone. For example, if the Starting Color # is 10 and the Color Increment is 2, the second contour zone will use color 12. The default values of 10 and 10 will create a rainbow color gradient. Similarly, a Starting Color # of 11 and a Color Increment of 10 will produce a lighter shade of the rainbow gradient. It can be difficult to create other color gradients with this system, but the other method of setting color allows for easier selection of other gradients.

When **Set Gradient Colors** is enabled, you will be able to set color gradients by selecting a **Low**, **Medium**, and **High** color value. All intermediate colors will be selected automatically. It should be noted that this command can only use the colors from the CAD color palette, which includes 255 colors. As a result, attempting to use this option with a large number of hatch zones may result in some hatch zones using the same color.

Set Pattern: When enabled, the pattern for each zone will be set to the selected hatch pattern.

Set Scale: When enabled, the hatch pattern scale for each zone will be set to the **Scale** value.

Set Layer: When enabled, the layer for each zone will be set to the **Layer** value. You may select an existing layer by clicking the **Select** button.

Create Polyline Topology: This option will draw the contour zone boundaries, but will not hatch them. This option will add an extra label in the zone boundary to indicate the range of grid values in that zone. This can

be useful for GIS applications in other modules.

Extrapolate Grid to Full Grid Size: This option will extend the contour lines to the rectangular limits of the grid when values are left as null. Note that this will not modify the grid file itself.

Prompts

Select the Inclusion perimeter polylines or ENTER for none.

Select objects: *pick a closed polyline for the contour boundary if any*

Select the Exclusion perimeter polylines or ENTER for none.

Select objects: *pick a closed polyline for the area to exclude*

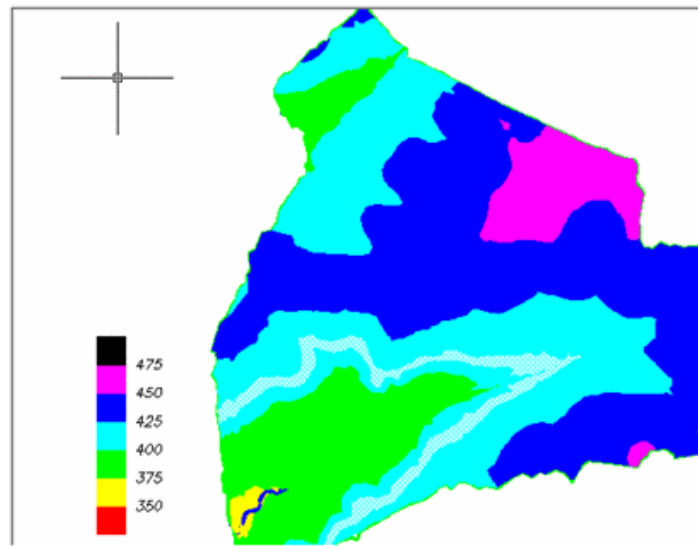
Grid File to Process dialog *select a .grd file*

Contour from Grid File options dialog

Extrapolate grid to full grid size (Yes/<No>)? *press Enter* This prompt appears if your grid extends beyond the limits of your data points in some areas.

Contours interpolated from GRID

Setting color ranges using "Hatch Zones" option



Pulldown Menu Location: Surface >> Contour from...

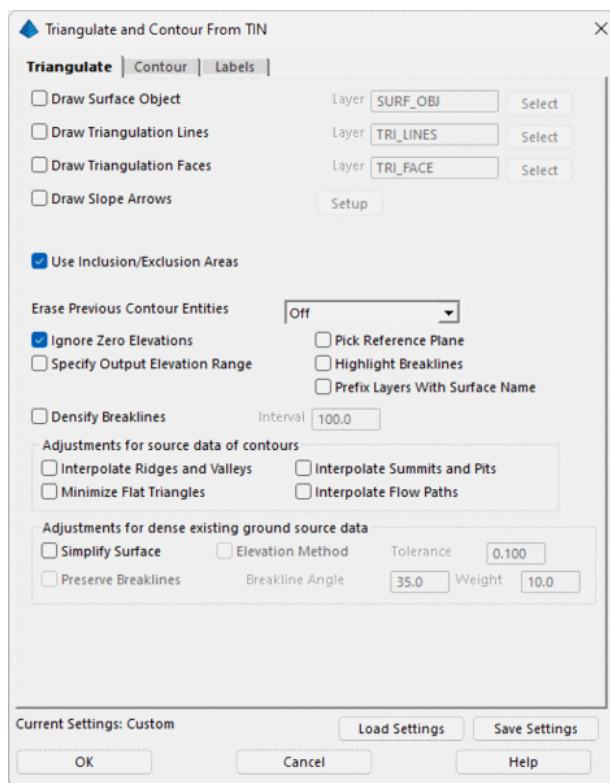
Keyboard Command: cntgrd

Prerequisite: A grid file

Contour from TIN File

This command creates contours directly from a TIN file (.flt or .tin) without the need to have the TIN drawn on the screen. The routine starts by opening the dialog for *Triangulate and Contour*, allowing the user to specify triangulation, contour and label settings. After pressing *OK* on the initial dialog, a second dialog opens, allowing for the selection of the TIN file from which to create the contours.

See the *Triangulate and Contour* section in the manual for a detailed description of each of the settings.



Prompts

Fill out the Triangulate and Contour Dialog information with the desired options. Select the desired TIN file and choose Open.

Loading edges...

Loaded 1994 points and 5944 edges

Created 3936 triangles

Removed 9 disconnected edges.

Reading points... 0

Contouring elevation 497

Inserted 1926 contour vertices.

The user may be prompted for additional information depending on settings used in the Triangulate and Contour dialog box.

Pulldown Menu Location: Surface >> Contour from...

Keyboard Command: cntrTIN

Prerequisite: A TIN file (.flt or .tin)

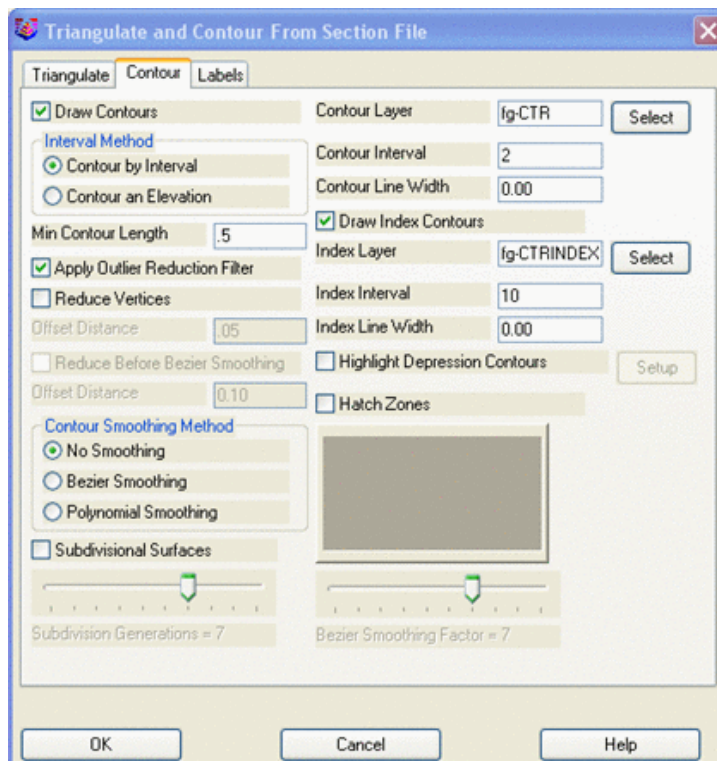
Contour From Section File

This command creates contours from an existing cross section file. Both a section file (.sct) and a centerline file (.cl) are required to generate contours. All of the settings for Triangulation, Contouring and Labeling, found in *Triangulate and Contour* are available.

See the *Triangulate and Contour* section in the manual for a detailed description of each of the settings.

Prompts

Triangulate and Contour dialog box



After pressing *OK*, the user will be prompted to select an existing section file (.sct) and a centerline file (.cl).

Reading points... 314

Inserted 314 points

Inserted 518 breakline segments

Contouring elevation 2016

Inserted 3091 contour vertices.

The user may be prompted for additional information depending on settings used in the Triangulate and Contour dialog box.

Pulldown Menu Location: Surface >> Contour from...

Keyboard Command: cntr_sct

Prerequisite: A section file (.sct) and a centerline file (.cl).

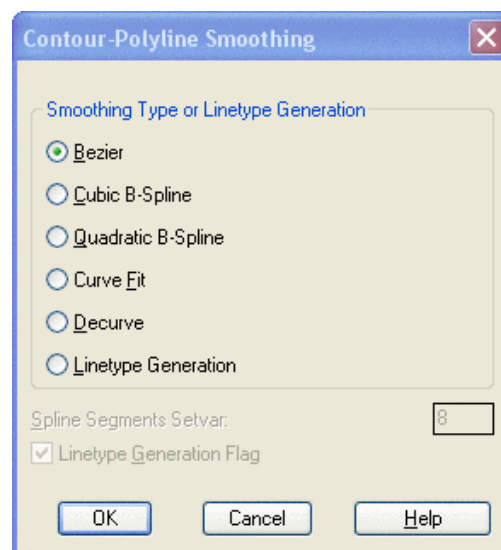
Smooth Contours

This command has options for applying smoothing to polylines. Select the radio button for the smoothing option you want to apply. If you use Quadratic B-Spline type smoothing or Cubic B-Spline type smoothing, the *Spline Segments* AutoCAD system variable is relevant. The Curve Fit option provides the least smoothing, and the Cubic B-Spline option applies the most. Another effective way of smoothing is by creating the contours from rectangular meshes using various grid resolutions. Increase the smoothing by lowering the grid resolution and decrease by raising the grid resolution. The Bezier option provides an incremental type of smoothing. The Linetype Generation option turns on the *Ltype Gen* flag for the selected polylines. For more information on this option and the spline smoothing options, look up the *PEDIT* command in the AutoCAD Reference Manual. After selecting the *OK*

button the routine will prompt for needed values.

Bezier smoothing is also embedded in many of the routines that create contours. Bezier smoothing applies the Bezier smoothing algorithm to polylines. This smoothing technique has two advantages over Spline or Curve Fit smoothing. One is that a Bezier smoothed polyline will pass through all of the vertices in the original polyline, while a Spline smoothed polyline only curves towards the original vertices and can pull away from vertices at sharp corners. Hitting all the original vertices can be an important feature in contour maps for maintaining the exact location of the contours. Another benefit of Bezier smoothing is the ability to control the looping and vertex factors. A higher looping factor increases the curving effect. Use this setting with some care, as too high a looping factor may cause nearby contour lines to cross after the smoothing has been applied.

Vertex reduction can also be applied along with the smoothing. This avoids having to create smoothed polylines with numerous vertices and then having to reduce these vertices in a second step. Be sure not to make the cutoff offset for reduction too high or you can negate or even reverse the smoothing effect. One disadvantage to Bezier smoothing is that it cannot be decurved like the other smoothing techniques.



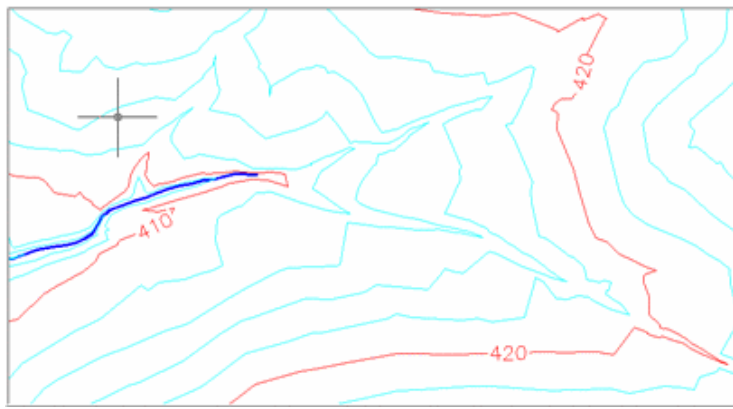
Prompts

Enter the looping factor (1-10) <5>: *press Enter* This determines the extent of curving. 1- least curvy, 10 - most curvy.

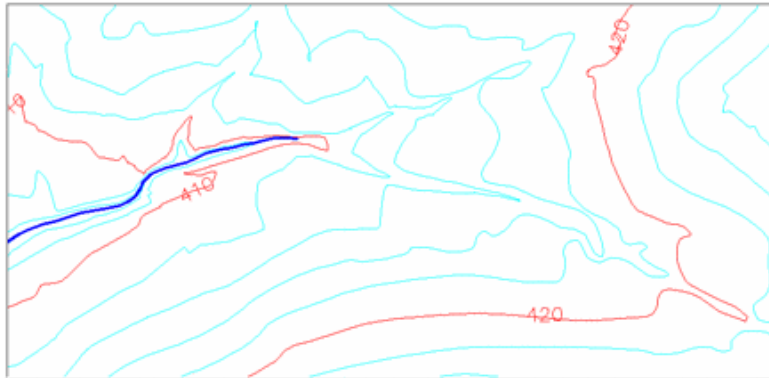
Enter the offset cutoff <0.05>: *press Enter* This value is the maximum shift distance for vertices reduction. A higher value removes more vertices.

Select polylines to smooth.

Select objects: *pick polylines*



Before Smoothing



After Smoothing

Pulldown Menu Location: Surface >> Modify Contours

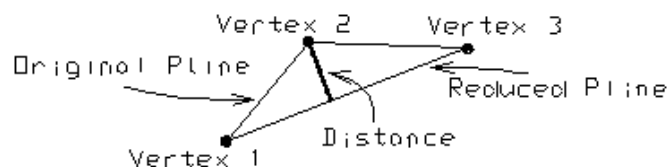
Keyboard Command: Smooth

Prerequisite: Create Contour lines to smooth

Reduce Contour Vertices

Contouring and smoothing often creates an explosion in file size due to the many vertices it adds to the individual contour polylines. Fortunately, many of these vertices are very close together, some of which can be removed with no visible effect on the contour polylines themselves. *Reduce Contours Vertices* can reduce the total number of vertices up to 90%. This has the benefits of a smaller drawing file, faster drawing loading, and faster regens.

This command removes vertices in a polyline that are within a user specified offset cutoff. The algorithm looks at three vertices at a time, and calculates the distance between the second point and the line from the first to the third point. If this distance is less than the user specified cutoff, the second point is removed. In theory, reducing the polyline vertices should not shift the polyline more than the user's cutoff distance. The default for this cutoff is one tenth of a foot. Increasing the cutoff will remove more vertices while decreasing it will more closely preserve the original contour line. When combining vertex reduction with smoothing, it is suggested to smooth before reducing, although it can be done the other way around.



Prompts

Enter the offset cutoff <0.1>: .3

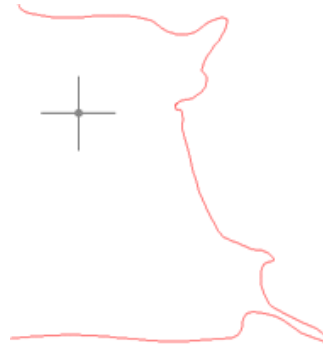
Select polylines to reduce. *select polylines*

Select objects: *press Enter* to conclude selection

Processed polylines: 1

Total number of vertices: 1125

Number of vertices removed: 939



Before Vertex Reduction



After Vertex Reduction

Pulldown Menu Location: Surface >> Modify Contours

Keyboard Command: reduce

Prerequisite: Polylines (contours) with vertices to reduce

Edit Contours

This command revises a segment of a contour polyline. Begin by picking a point on the contour where you want to start editing. Then pick new points for the polyline. When finished picking new points, press Enter and then pick a point on the contour to connect with the new points. The polyline segment between the start and end points is then replaced with the new points.

If there is a triangulation file associated with the contours, then the command prompts for whether to update the triangulation surface file to match the contour edits. When this option is used, data points are added to the triangulation surface along the edited contour segment to make the triangulation surface match the contour line. Existing triangulation source data is retained. So the updated triangulation is the combination of the original source data and the additional points from Edit Contours. One way to get a triangulation surface associated with the contours is to use the Triangulate & Contour command with both Write Triangulation File and Draw Contours

options active.

Note: If the triangulation association is not used, then this routine has no effect on the actual triangulation or grid surface model file that the contours may have been drawn from. It only revises the drawn contour or polyline on the screen. If the contours are later regenerated from this file, the edits will be discarded.

Prompts

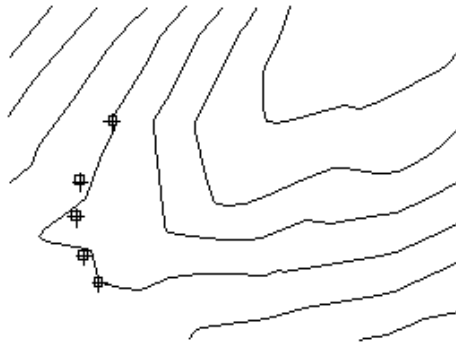
Select contour to edit: *pick the contour polyline at the place to start editing*

Pick intermediate point (Enter to End): *pick a point*

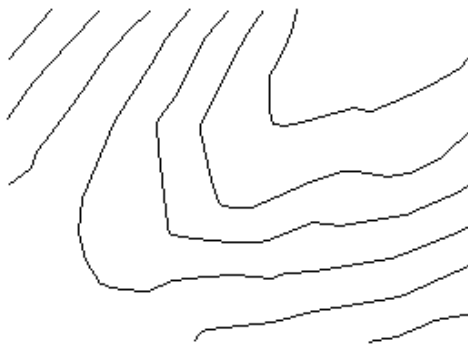
Pick intermediate point ('U' to Undo, Enter to End): *pick a point*

Pick intermediate point ('U' to Undo, Enter to End): *press Enter*

Pick reconnection point on contour: *pick the contour polyline at the place to join*



Edit this contour by picking new points



Contour with segment replaced with new points

Pulldown Menu Location: Surface >> Modify Contours

Keyboard Command: editctr

Prerequisite: Polylines with elevation (contour polylines)

Contour ID

Contour ID reports the routine and source data used to generate the selected contour polyline.

Prompts

Select contour polyline to identify: *pick a polyline*

Surface Name: **Triangulate & Contour by screen entities**

Select contour polyline to identify (Enter to end): *press Enter*

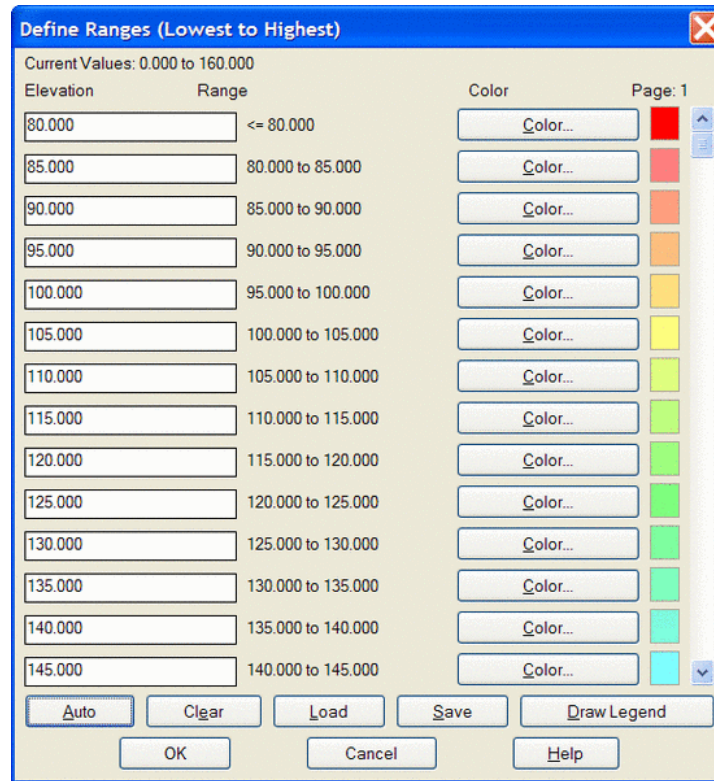
Pulldown Menu Location: Surface->Modify Contours

Keyboard Command: CTR_ID

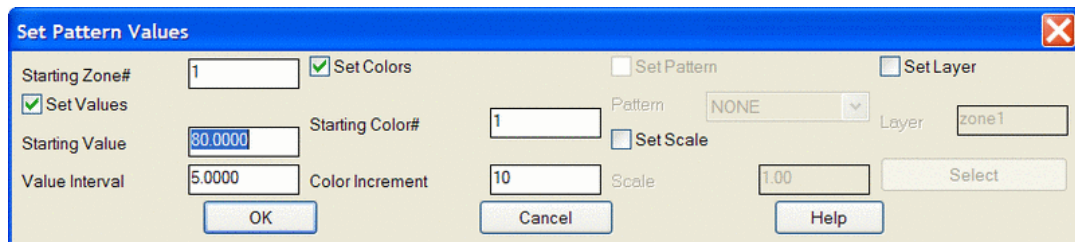
Prerequisite: a contour polyline

Color Contours by Elevation

This command sets the color of the selected contour polylines and text based on elevation. The color to use is defined in elevation range table.



- **Auto** - This button opens the following dialog, allowing for automatic configuration of the range of elevations and colors.



- **Starting Zone #** - Sets the zone with which to begin the application of the settings defined in this dialog. For Instance, if the Starting Zone was set to 10, the settings definitions applied here wouldn't affect Zones 1-9, but would start at Zone 10.
- **Set Values** - Enables the Starting Value and Value Interval fields, which allow the user to specify the starting elevation for the given zone and set the zone increment.
- **Starting Value** - Sets the starting elevation value for the first zone.
- **Value Interval** - Sets the elevation increment for subsequent zones.
- **Set Colors** - Enables the Starting Color and Color Increment fields.
- **Starting Color #** - Sets the starting color number, based on the AutoCAD standard color chart.

- **Color Increment** - Sets the color number to increase for subsequent zones. So if the increment was set to 5, and the starting color was 60, the next color would be 65, 70, and so on.
 - **Note:** The Pattern, Scale, and Layer options do not apply to this command.
- **Clear** - Clears the all of the Elevation fields in the dialog
 - **Load** - Loads previous settings from a saved .pat file
 - **Save** - Saves the current setting configuration to a .pat file.

Prompts

Select polylines and text to color: *pick the entities*

Define Ranges Dialog

Pick point for color legend: *pick a point to a clear area of the drawing to place a legend or press Enter for no legend*

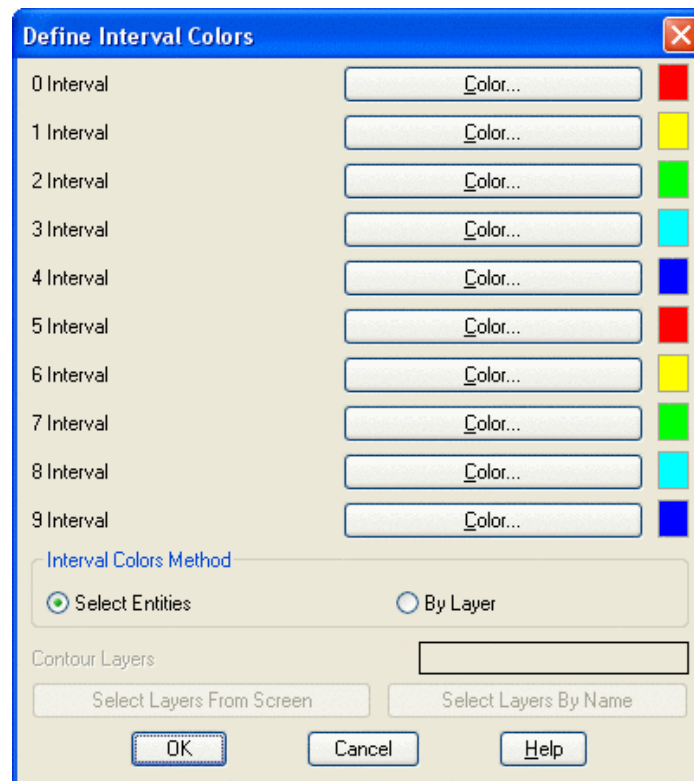
Pull-Down Menu Location: Surface >> Modify Contours >> Color Contours

Keyboard Command: ctrcolor

Prerequisite: Contours polylines

Color Contours by Interval

This command sets the color of the selected contour polylines based on the elevation interval values, which are essentially the number that the elevation ends with, so specific colors are assigned for elevations ending in 0, 1, 2, etc. The color assignments are defined in the Define Interval Colors dialog box.



Select Entities: User is prompted to select the contour polylines to change.

By Layer: Contour polylines are selected automatically by their layer.

Prompts

Define Interval Colors Dialog If Select Entities is set as Interval Colors Method, *pick OK*, and you are prompted to:

Select polylines and text to color: *pick the entities* If By Layer is set as Interval Colors Method, set the *layers by Screen* selection or *from a list by Name*, then *pick OK*.

Pull-Down Menu Location: Surface >> Modify Contours >> Color Contours

Keyboard Command: ctrcolor2

Prerequisite: Contours polylines

Highlight Index Contours

This command will move contours of a specified interval to another layer. This allows the user to change the color or width of a certain interval. This is useful if all the contours had been generated on a single layer, and you wish to display the index contours differently based on a new layer setting.

Prompts

Layer name of existing contours <CTR>: *press Enter*

Layer name for highlight contours <NCTR>: *press Enter*

Select Contours to Highlight.

Select objects: Select contours using any standard selection methods.

Select objects: *press Enter* to conclude selection. The program then sorts and displays the High and Low interval of the selected contours.

Contour increment to highlight: *10*

Starting Highlight at elevation <98.0>: *100*

Ending Highlight at elevation <152.0>: *150*

Assuming we had drawn 1 foot intervals, the above example would move the contours on elevations 100, 110, 120, 130, 140 and 150 to the layer NCTR.

Pulldown Menu Location: Surface >> Modify Contours

Keyboard Command: indexctr

Prerequisite: Contours should be plotted and visible on the screen.

Highlight Depression Contours

This command highlights depression contours by changing their layer, color, and adding tick marks. A depression contour is a closed contour line that leads to a local minimum such that there are no contour lines with a higher elevation within the contour. This routine finds the depression contours out of the selected polylines. The depression contours are highlighted, and the user selects which ones to label.

Prompts

Layer name of existing contours <CTR>: *Enter*

Layer name for depression contours <DCTR>: *Enter*

Width for depression contours <1.0>: *Enter*

Tick Interval for depression contours <50.00>: *Enter*

Tick Size for depression contours <6.0>: *Enter*

Select the existing contours.

Select objects: *Select all the contour polylines*, even the contours that aren't depression contours.

Select objects: *press Enter* to conclude selection. The program then sorts and displays the high and low elevations

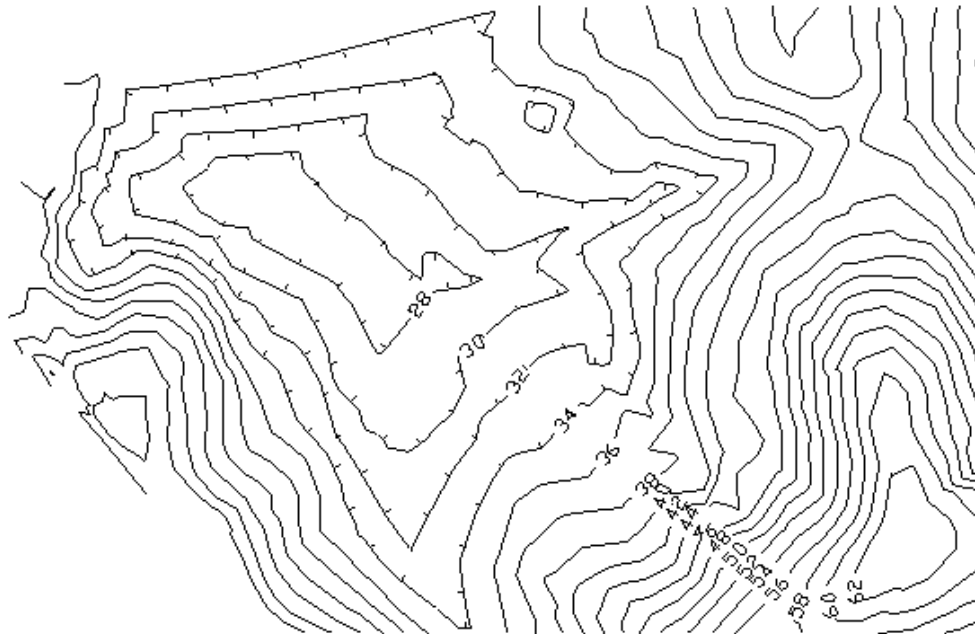
of the selected contours.

Reading the selection set ...

Locating the depression contours ...

Highlight all or selected depression contours [All/<Selected>]? A The "All" option changes all contours identified as depression contours to the specified layer and adds tick marks. The "Selected" option highlights all contours identified as depression contours and then user is prompted to select which ones to change to specified depression contour layer and add tick marks to.

Drawing the depression contours ...



Highlighted depression contours

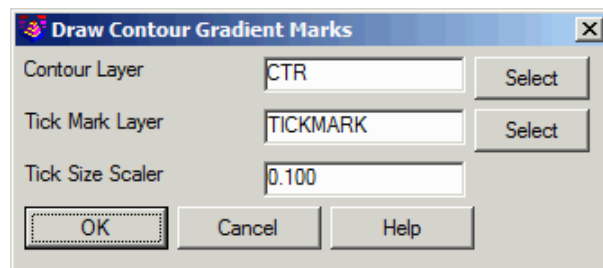
Pulldown Menu Location: Surface >> Modify Contours

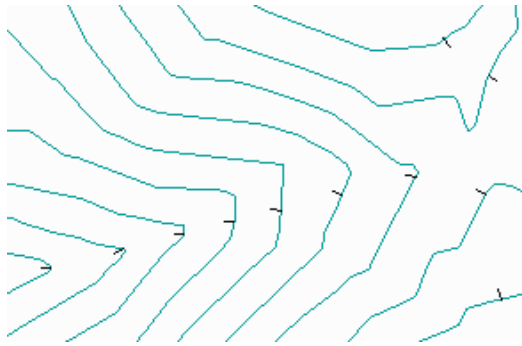
Keyboard Command: depress

Prerequisite: Contours should be plotted and visible on the screen.

Draw Contour Gradient Marks

This command draws lines perpendicular to contours to show the downhill slope direction. In the options dialog, set the layer for the existing contour polylines, the layer for the new gradient lines, and the size scaler for the new lines which is relative to the current scale from Drawing Setup. Next, the program prompts for the surface triangulation file which should be the same surface that the contours represent. The program uses this surface file to determine the direction for the gradient lines. Then the program prompts for where to draw the gradient marks. The Polyline option draws marks at each intersection of the selected polyline with the contours. The Point method prompts for two points and draws marks at the contour intersections with the line defined by these two points.





Prompts

Options Dialog

Select Surface File

Define a line which slices the contours at the place to label them.

Pick 1st point (P-Polyline, Enter to end): *pick point*

Pick 2nd point: *pick point*

Define a line which slices the contours at the place to label them.

Pick 1st point (P-Polyline, Enter to end): *press Enter*

Pulldown Menu Location: Surface > Modify Contours

Keyboard Command: ctr.grad

Prerequisite: contour polylines and surface file

Change Contour-Plines Width

This command allows the user to select a group of contours/plines and change their width for emphasis when plotting. Prior to running this command, the desired contours can be isolated to their own layer using the *Highlight Index Contours* command, or if already on a separate layer you may use *Isolate Layer* from the *View* menu.

An alternate to using this routine is to assign an AutoCAD *lineweight* to the layer that the contours or polylines are on and set the *Display Lineweight* toggle at the bottom of the screen. If using this routine to assign a polyline width, then this new width will display regardless of the lineweight toggle.

Pulldown Menu Location: Surface >> Modify Contours

Keyboard Command: cwidth

Prerequisite: Contour polylines should be drawn and visible on the screen.

Trim Contour-Plines by Pline

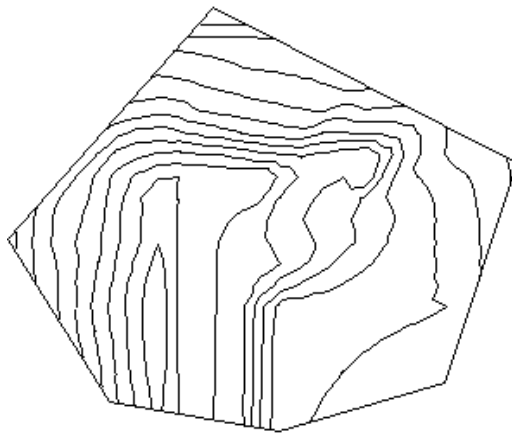
This command can be used to trim a group of contour lines or polyline entities that cross a perimeter defined by a 2D polyline. The trim can be executed on the inside or the outside of the perimeter.

Prompts

Warning: All of the trim perimeter should be visible on the screen!

Select polyline which represents perimeter: *select trim perimeter*

Pick point on the side of perimeter to trim from: *pick a point* To trim contours on the inside of the perimeter, pick a point on the inside of the perimeter (this is useful for deleting contour lines that fall inside a building or some area that you want void of contours). To trim contours on the outside of the perimeter, pick a point outside of the perimeter.



Pulldown Menu Location: Surface >> Modify Contours

Keyboard Command: polytrim

Prerequisite: Draw a 2D closed polyline perimeter.

Contour Elevation Label

This command can be used to simultaneously create elevation labels on a group of contour polylines at elevation. First the command starts with a dialog with the label options. Then to place the labels, pick two points crossing the contour polylines at the desired label location. The program will find all the contour polylines that intersect the picked line (defined by the two picked points) and will place labels at the intersection point of each contour. A second crossing line can be initiated immediately, so multiple areas can be quickly labeled while remaining in the command. Alternatively, you can type P for Polyline at the Command prompt and select a polyline. Then the program finds all the intersections between the selected polyline and the contours and places labels at these intersections. The actual "z" elevation of the contour line determines the label value.

Label Layer specifies layer name for the contour labels that will be created.

Label Style specifies the text style to be used for labels.

Horizontal Scale is used in conjunction with the Text Size Scaler to determine unit height of the contour labels. **Text Size Scaler** is a scaler that will be multiplied by the horizontal scale to set the actual text height of the labels in AutoCAD units.

Integers controls how many digits to label to the left of the decimal. For example, if all contours are in the 5000's, then setting for three digits would label the 5280 contour as 280.

Decimals sets the decimal precision for the labels to be created.

Label Position determines the label position in relation to the contour polyline.

- **On Contour** centers the label on the contour line.
- **Above Contour** places the label above the contour line. If this option is used, the options for Break Contours at Label and Draw Broken Segments become inactive.

Ignore Zero Elevation Polylines enables the routine to filter out all entities with an elevation of zero.

Hide Drawing Under Labels activates a text wipeout feature that will create the appearance of trimmed segments at the contour label, even though the contour line is still fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels, yet the contour lines themselves remain contiguous. This feature will also hide other entities that are in the immediate vicinity of the contour label.

Align Facing Uphill makes the label parallel to the contour and flips the label so that it reads facing uphill. Otherwise, the labels are made to face up relative to the current screen view. When this option is on, the program prompts for a triangulation surface file that should match the surface the contours represent.

Use Commas adds a comma into the labels for the thousands place such as "5,000" instead of "5000".

When **Align Text with Contour** is checked, contour elevation labels will be rotated to align with their respective contour lines.

When **Break Contours at Label** is checked, the contour lines will be broken and trimmed at the label location for label visibility.

When **Draw Broken Segments** is checked, segments of contours that are broken out for label visibility will be redrawn as independent segments. Specify the layer for these broken segments in the box to the right of this toggle.

Label Contour Ends creates labels off the ends of the contours.

Label By Distance places the labels by distance along the contour. The user is not prompted for screen picks of contour crossing when this option is used.

- **Interval** sets the distance interval to be used between labels on each contour.

When **Draw Box Around Text** is checked, a rectangle will be drawn around the elevation labels. The Offset Scaler controls the size of the rectangle.

The **Draw On Real Z Axis** chooses between creating the text entities at the elevations of the contours or at zero elevation.

The **Use MText** chooses between creating MText and DText label entities.

Index Contours: Label All will label both index and intermediate contours with the same settings. Label Index Only labels only the index contours. Separate Index Layer will label both index and intermediate contours with the index labels on a different layer.

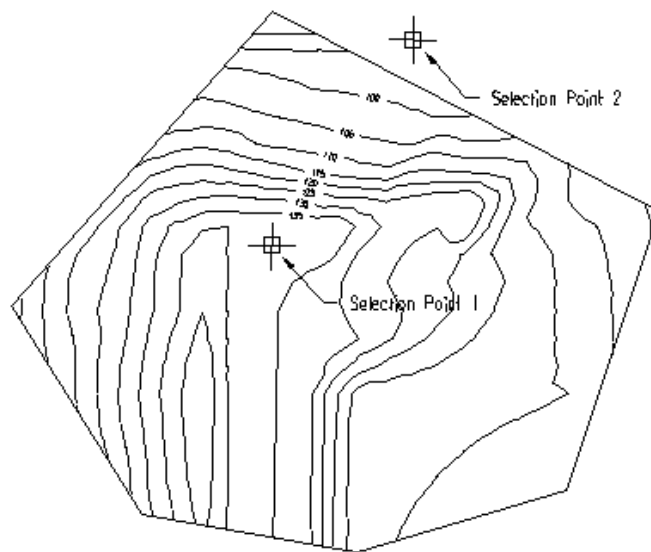
Prompts

Contour Label Options Dialog Opens Select the desired options and press OK.

Define a line which slices the contours at the desired label locations.

Pick 1st point (P-Polyline, Enter to end): *pick a point*

Pick 2nd point: *pick a point*



By selecting two points the contour lines that cross the line defined by the two points are labeled.

Pulldown Menu Location: Surface >> Contour Labels

Keyboard Command: gclabel

Prerequisite: polylines with elevation (contour polylines)

Local Elevation Label

This command allows the user to place elevation labels on contour lines or other entities with elevation. The command prompts for two points to align the rotation angle of the label, centers the label between the picks, and then prompts for two break points which are used to erase/break the contour that runs through the elevation label. If a break is not required just press [Enter] at the break point prompts.

Pulldown Menu Location: Surface >> Contour Labels

Keyboard Command: clabel

Prerequisite: polyline with elevation

Draw Contour Label Marker Line

This command draws lines that Triangulate and Contour looks for to control where to draw contour elevation labels. The labels are drawn where the contour polylines cross the marker lines. These marker lines can be drawn on a separate layer that can be turned off for plotting the drawing.

Prompts

Layer selection dialog

Pick start point: *pick a point*

Pick end point: *pick a point*

Pick start point (Enter to end): *press Enter*

Pulldown Menu Location: Surface > Contour Labels

Keyboard Command: mark_ctr_label

Prerequisite: None

Move Label Along Contour

This command slides an existing contour label along a contour, maintaining its alignment with the contour. After moving the label, you can type F for Flip at the Command prompt to rotate the label orientation by 180. The label must have originally been created with the *Break Contours at Label* option *Off*. If the option to *Hide Drawing Under Labels* was used when the label was created, the wipeout will move with the label when using this command.

In addition to moving a label, an existing label can be copied and placed at a new position along the contour by using the Copy option at the first prompt.

Prompts

Copy/<Select contour label to move>: *Pick label*

Pick new contour label position: *Move mouse to relocate label*

Flip last/<Select contour label to move (Enter to end)>: *press Enter*

Pulldown Menu Location: Surface >> Contour Labels

Keyboard Command: move_ctr_label

Prerequisite: generated contour labels

Flip Contour Labels-Text

This command individually rotates each of the selected text entities by 180 degrees.

Pulldown Menu Location: Surface >> Contour Labels

Keyboard Command: fliptext

Prerequisite: Text labels on contours

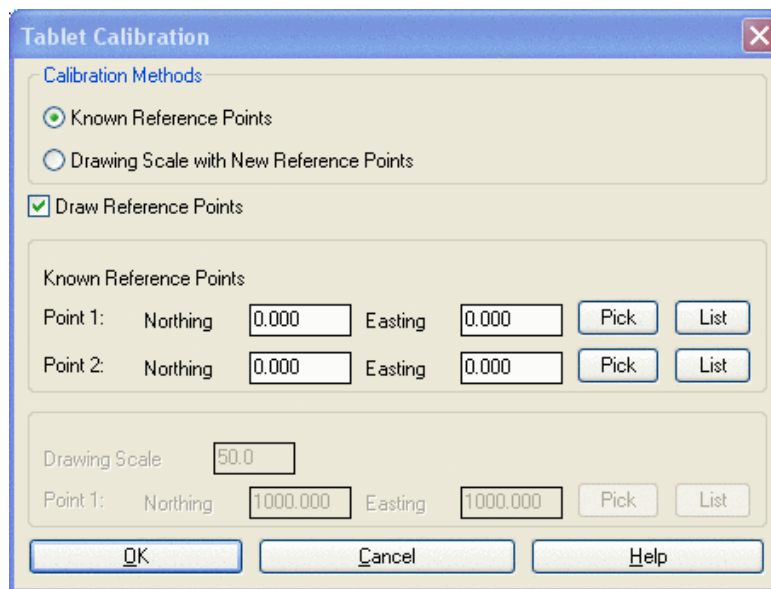
Tablet Calibrate

This command executes the routine to calibrate the digitizer tablet to a hardcopy drawing. There are two methods of calibration: *Known Reference Points*, and *Drawing Scale with New Reference Points*, which are explained in detail below. The Calibrate routine must be used prior to using the *Digitize Contours* command.

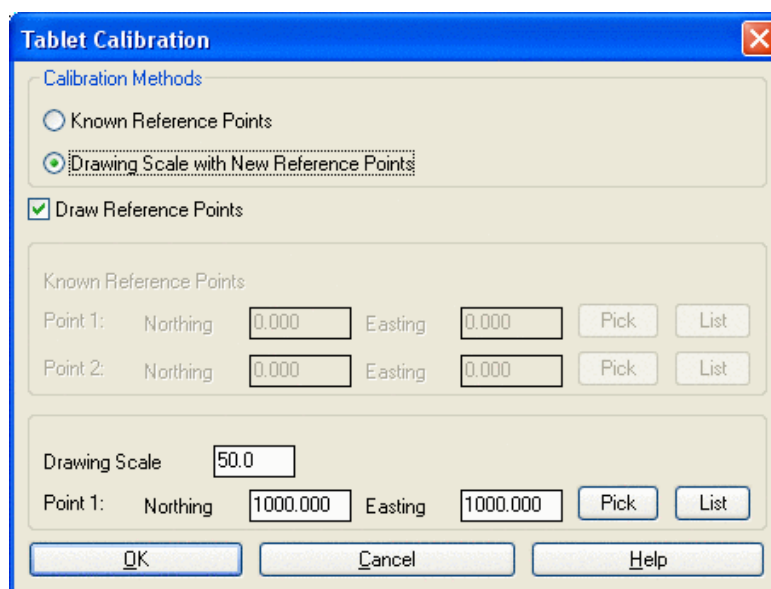
Please refer to Configure, General Settings and Digitizer Puck Layout for selection of the correct puck layout before proceeding.

Tablet Calibration

Known Reference Points uses two known coordinates for reference points on the hardcopy drawing. When this option is selected, the fields for coordinate information activate. Enter the known northing and easting values for the reference points from the information on the hardcopy drawing in the appropriate fields and select the *Pick* button. Pick the points from the hardcopy drawing using the tablet. Carlson Civil saves the coordinates of the two reference points for future calibrations and displays them on the *Tablet Calibration Dialog* the next time it is accessed, so if you are working on the same drawing, you can use the **Known Reference Points** method with the saved coordinates to calibrate to your previous coordinates. For greater calibration accuracy, choose two points that are farther apart rather than closer together.



Drawing Scale with New Reference Points is very convenient when you don't know the precise coordinates of the entities on your hardcopy drawing. You must specify the drawing scale from the plan. This method establishes a coordinate system relative to the position of the plan on the digitizer board. In addition to the drawing scale, you are required to enter a random coordinate for the first reference point, the default coordinate is (1000,1000). You then select the *Pick* button and pick the point on the hardcopy drawing to assign the specified coordinate to. The routine will compute the coordinate of the second reference point that you pick based on the first point. The coordinates of these two reference points would be saved and will be display in the *Tablet Calibration Dialog* as **Known Reference Points** the next time you calibrate the tablet, so you can digitize the previous coordinates if you are working on the same drawing, even though you may have moved or rotated your drawing on the digitizer tablet.



Prompts

Tablet Calibration Dialog

Specify the Calibration Methods. If you select *Drawing Scale* method, enter the drawing scale and the coordinate of the first reference point. Otherwise enter the exact coordinates of the first and second reference points.

Pick first reference point: *pick a point*

Pick second reference point: *pick another point*

Pulldown Menu Locations: Surface >> Digitize Contours

Keyboard Command: digsetup

Prerequisite: Affix a drawing to your digitizer tablet. Have a digitizer board and a puck connected to your computer, correctly configured, and have Wintab driver installed. Select the puck layout in Configure.

Digitize Contours (Polyline)

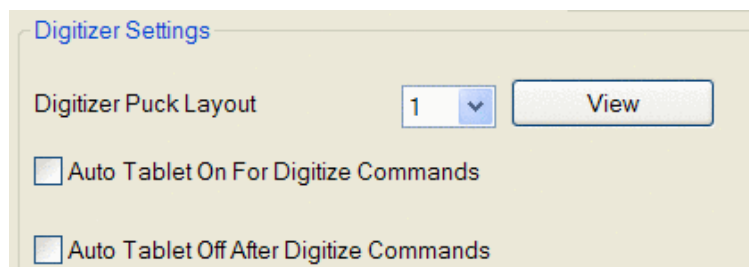
A contour is drawn as a polyline which consists of a series of connected points with a constant elevation. There are two ways to digitize contour lines: sketch mode or point mode. You can start digitizing a contour with one mode and switch to the other during digitizing the contour. Sketch mode uses more points than pick mode. In general, we recommend using pick mode to digitize the straight parts of lines because it reduces the number of points and speeds up calculations, but using sketch mode to digitize the curved parts because it is fast and accurate.

This command lets you digitize contours as polylines one at a time. The first time it prompts you with the *Digitize Contours Dialog*. Enter the layer name or select it from a list of existing layers. Look at your hardcopy plans and determine an elevation interval that is between most of the contours and enter it in the Elevation Interval field. You are able to modify both the value and the direction of the elevation interval between digitizing contour lines, using the buttons on the puck.

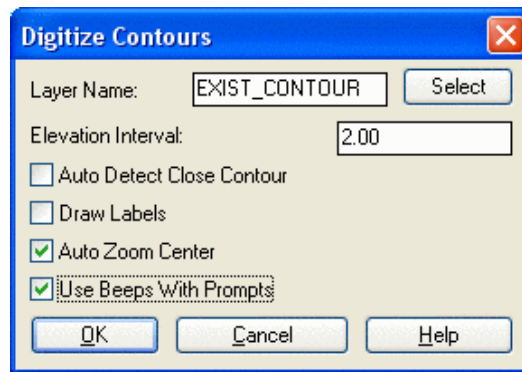
To have Carlson Civil automatically close contours whose beginning and ending points are within a specified range, check the *Auto Detect Close Contour*. *Draw Labels* will draw the elevation at the starting point of the contour. In Pick mode, if you want the program to automatically zoom the display to center around the last point when you get near the edge of the screen while picking points, check the *Auto Zoom Center*. Click *OK* to start digitizing.

If this is your first time digitizing a contour, you are defaulted to the Pick Mode digitizing, otherwise you will be defaulted to the previous digitizing mode. If you want to use the other digitizing mode, press *0* (the number "zero") on the puck or enter *0* from the keyboard. Place your cursor at one end of the contour line and begin digitizing the line. While digitizing a line, you can force a contour to close on itself by pressing *A* on the puck to end the contour and connect the last point to the first point, remove a mistake by pressing *B* on the puck, or switch to the other digitizing mode by pressing *0*. During Sketch Mode digitizing, you can stop digitizing by pressing the *Pick* or *Enter* button on the puck, take some rest or make edits, and start sketching again. At the end of the contour line, press *Enter* on your puck or keyboard. The contour is completed, and the elevation for the next contour is automatically incremented. You will be asked to digitize the next contour. If you press *A* on the puck or enter *Yes* on the keyboard, you can digitize another contour, or press *B* on the puck or enter *No* on the keyboard to end digitizing contours.

To digitize with a mouse instead of a digitizing tablet, go to the *Settings* menu, *Configure*, and *General Settings*. In the General Settings dialog box, under Digitizer Settings, clear the check box for *Auto Tablet On For Digitize Commands*.



This is the Digitize Contours dialog box.



Prompts

Digitize Contours Dialog

Enter Layer Name, Elevation Interval, and toggle on/off Auto Detect Close Contour etc.

Increment(1.00)[A]/Direction(+)[B]/Elevation <573.00>: 450 Enter elevation or press Enter to accept current value.

Start Digitizing...

Sketch[0]/Pick the first point: *pick a point to start Pick Mode digitizing* Press 0 to switch to Sketch Mode.

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): 0 Press 0 on the puck or enter 0 on the keyboard to use Sketch Mode.

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): *pick and drag*

Drag to digitize (Pick or press Enter to stop sketching)... *pick or press Enter to stop sketching*

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): B Undo the last point.

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): B Undo the last point.

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): *pick and drag again*

Drag to digitize (Pick or press Enter to stop sketching)... *pick or press Enter to stop sketching*

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): 0 Press 0 on the puck or enter 0 on the keyboard to use Pick Mode.

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): *press Enter to finish digitizing*

Digitize Another Contour [<Yes(A)>/No(B)]? B Press B to finish digitizing.

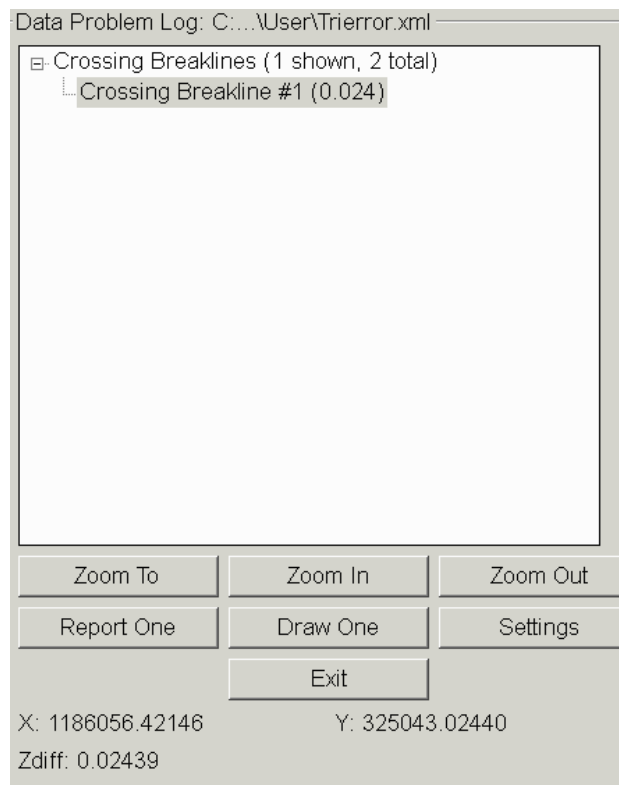
Pulldown Menu Location: Surface >> Digitize Contours

Keyboard Command: digcont

Prerequisite: Calibrate the tablet with the *Tablet Calibrate* menu option if contours are to be taken off an existing hardcopy drawing.

Display Last TIN Error Log

This command displays the data error log from the last run of Triangulate and Contour. The Triangulate and Contour command reports any errors in the source data such as crossing breaklines. If you exit this report during Triangulate and Contour, then you can bring up this report again using this command. See the Triangulate and Contour section of the manual for more information on this error report.



Pulldown Menu Location: Surface > Triangulation Surface Manager

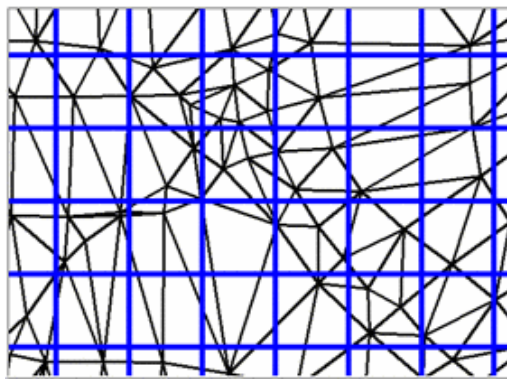
Keyboard Command: tin_error_log

Prerequisite: data errors from Triangulate and Contour

Make 3D Grid File

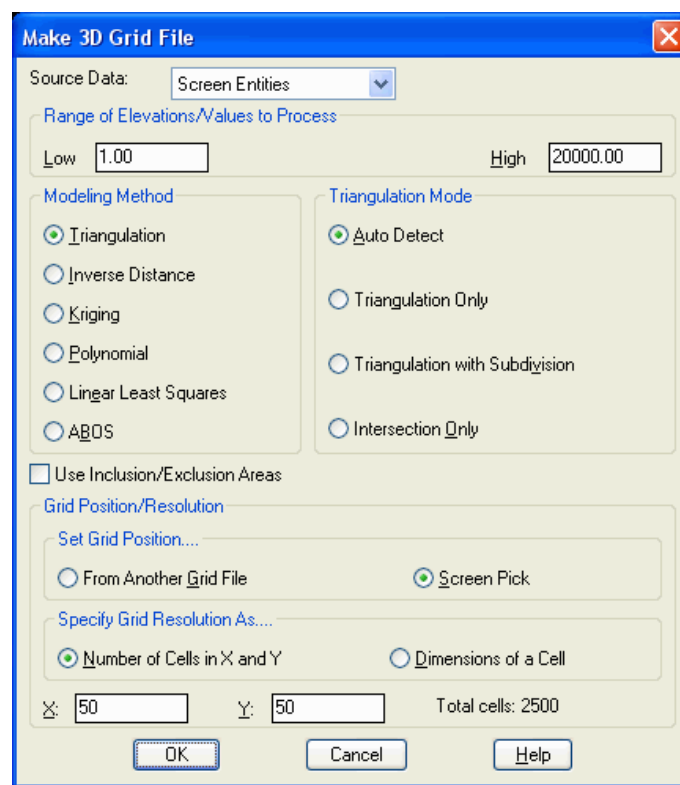
This command creates a grid (.GRD) file which serves as a surface model for use in many of the other Surface routines. The program internally makes a triangular network of the data points (if *Triangulation* is selected as the modeling method) and then interpolates the elevation values of a rectangular grid at the specified grid resolution. Data points can be either points, inserts, lines, or polylines. Lines and polylines are treated as breaklines in the triangulation.

Gridding as a means of modeling surface features is generally less favorable than triangulating as the surface is defined only at the intersection of the grid lines. This can lead to inaccuracies around local features such as ditches or curb lines, since the grid resolution must be small enough to adequately capture the changes in these local regions. Contrast this with Triangulated Networks which carry all this information at every point along the features. Gridding can, however, be useful for modeling large sites in general trends such as watershed analyses and large-scale volume computations.



Grid superimposed over triangulated features

The grid location is specified by first picking a lower left corner and then an upper right corner. The screen cannot be twisted when this is done because grids always run north-south and east-west.



The dialog box sets the range of elevations to process, modeling method and grid resolution. Each of these items is described below.

- **Source Data:** This option selects the type of data to use for gridding. The Screen Entities option processes selected 3D entities from the drawing including points, lines, polylines, 3D faces and inserts. The Coordinate File and Text File options read point data from the selected file. These methods are useful for large datasets that would take extra memory and time to draw as points in the drawing. For the Text File, the program will prompt for the order of the fields and the delimiter. The Triangulation File option will interpolate the grid elevations from the selected triangulation surface.
- **Range of Elevations/Values to Process:** Entities with elevations or values outside the range to process are ignored and will not be used for the gridding.
- **Modeling Method:** The modeling method almost always should be triangulation for surface topographic grid files. Polynomial, inverse distance, kriging and linear least squares apply to random data points for surfaces

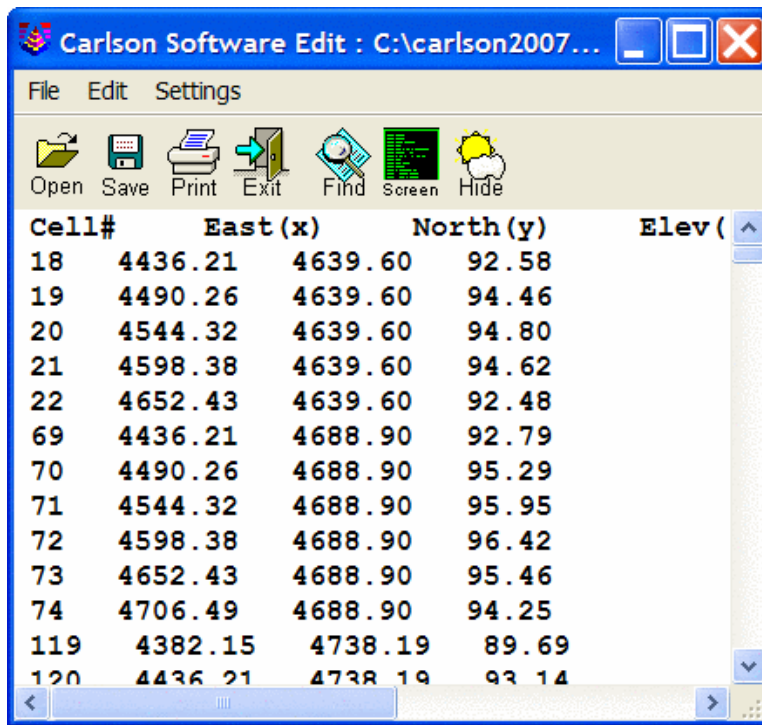
- like underground features, usually sourced by such methods as drillholes, data tables, etc.
- **Triangulation Mode:** When using Triangulation and Polynomial methods, There are four triangulation modes: AutoDetect, Triangulation Only, Intersection with Triangulation and Intersection Only.
 - **Auto Detect** method automatically chooses between the Triangulation Only and Intersection with Triangulation methods. If the selected surface entities are primarily made of polylines, then the Intersection with Triangulation method is used. Otherwise the Triangulation Only method is used.
 - **Triangulation Only** method builds a triangulation surface out of all the selected points, lines and polylines. All lines and polylines are treated as breaklines. Grid node elevations are calculated based on the triangulation.
 - **Triangulation with Subdivision** method uses the subdivisional surfaces modeling method. This option causes each triangle in the triangulation surface model to be subdivided into an average of three smaller triangles per subdivision generation. This gives a much smoother surface model, where instead of one triangle, there are now three or more.
 - **Intersection Only** method goes directly to the Steepest Intersection method using the selected lines and polylines. The Steepest Intersection method is used to assign the grid node elevations from the linework of the triangulation lines and the selected lines and polylines. The triangulation step is skipped and any selected point data is not used. This method can be used for making grids out of polylines such as a contour map as long as the surface is defined just by contour polylines without needing spot elevation points. Skipping the triangulation step makes this method a lot faster especially for large files.
 - **Use Inclusion/Exclusion Areas:** This option will prompt for inclusion and/or exclusion perimeter polylines and will only assign grid cell elevations within these areas and leave the rest of the grid cells as Null.
 - **Grid Resolution:** The grid resolution is specified by either the number of grid cells or by the size for each grid cell. It is usually best to set the Dimensions of a Cell to a known size, and the program will calculate the "number of cells in X and Y." While the program can handle really large grids with no limit, a general rule of thumb is to keep the total number of grids cells under 500,000 (about 700 by 700 cells) to limit the processing time. The grid location and resolution can also be specified by using the position/resolution from an existing grid file. In this case, the location and resolution of the new grid will match those of the selected grid file which is useful for routines that require two grid files with identical locations and resolutions.

No elevations are calculated on grid cells that extend beyond the extent of the data. The figure shows an example of how the grid is calculated to the limits of the data points. Extrapolation can be used to calculate elevations for the grid cells that are beyond the data limits. When there are grid cells with no elevation in a grid (.GRD) file, many routines will prompt *Extrapolate grid to full grid size?* Extrapolation fills in all the grid cells. The method to extrapolate uses a safe calculation that tends to average out or level the extrapolated values. So extrapolated grid areas are not as accurate as grid areas within the limits of the data. *Grid File Utilities* can be used to apply and save extrapolation to a grid file. The *Plot 3D Grid* command can then draw the grid file so that you can see the extrapolation.

A Carlson grid (.GRD) file has the following format:

Line 1 is the lower left Y coordinate
 Line 2 is the lower left X coordinate
 Line 3 is the upper right Y coordinate
 Line 4 is the upper right X coordinate
 Line 5 is the X direction grid resolution
 Line 6 is the Y direction grid resolution

The rest of the lines are the Z values of the grid intersects starting from the lower left moving in the left to right direction and ending at the upper right. If the intersect has no value, the letter 'N' is saved instead of the Z value for Null values. An example is shown in the Display-Edit Report dialog.



Cell#	East (x)	North (y)	Elev (z)
18	4436.21	4639.60	92.58
19	4490.26	4639.60	94.46
20	4544.32	4639.60	94.80
21	4598.38	4639.60	94.62
22	4652.43	4639.60	92.48
69	4436.21	4688.90	92.79
70	4490.26	4688.90	95.29
71	4544.32	4688.90	95.95
72	4598.38	4688.90	96.42
73	4652.43	4688.90	95.46
74	4706.49	4688.90	94.25
119	4382.15	4738.19	89.69
120	4436.21	4738.19	93.14

Gridding from Contour Maps

A grid file can be created from contours represented as polylines with elevation. The program calculates the elevation of each grid corner by looking for contour intersections in eight directions (N, S, E, W, NE, SE, SW, NW) and then interpolating the elevation between the two steepest intersections.

To accurately model the surface, it might be necessary to add entities in addition to the contour polylines. For one, spot elevation points can be added for the high and low points. Otherwise the grid model might plateau at the last contour. Also 3D breaklines need to be added on long narrow ridge and valley contours because in these areas the program will find the same contour when it looks for intersections in the eight directions. When all eight intersections are the same contour, the interpolated grid elevation equals the contour elevation instead of rising up the ridge or dipping in the valley. The 3D breaklines force interpolation along the ridge or valley. To draw these polylines, set the OSNAP to *Nearest* and run the *3D Polyline* command. Then draw the polyline by picking the contour polylines where the breakline crosses them. Another way to quickly create breaklines is to first draw 2D polylines. Then convert these polylines into 3D polylines with the *Screen* option in the *2D to 3D Polyline by Surface Model* command found on the 3Dpoly menu. There is also an automatic way to draw these breaklines. Under 3D Data, use the command: *Create Ridge polylines from Contours*.

Prompts

Grid File to Create File Selection Dialog

Enter a name for the grid file.

Use position from another file or pick grid position [<Pick>/File]?

Pick Lower Left grid corner <8111.88,3985.08>: pick a point for the lower left limit of the grid

Pick Upper Right grid corner <8366.88,4195.08>: pick a point

Make Grid File dialog box

In this dialog, you specify the grid resolution and whether or not to include data points with zero elevations. You can specify the resolution by entering the number of grid cells in the X and Y directions. By the Dimensions option, you to set the X and Y size for each grid cell.

Reading points ...

Select points, lines, polylines and faces to grid from.

Select objects: Specify opposite corner: 1075 found

Select objects:

Reading points ... 980

Finding points on breaklines ...

Ignored 2729 duplicate points.

Inserting breaklines 3480 ...

Triangulating points ... 980

Assigning grid values > 1800

Writing grid file: C:\Carlson 2008\WORK\example1.grd

Pick the Lower Left grid corner: *pick a point for the lower left limit of the grid*

Pick the Upper Right grid corner: *pick a point*

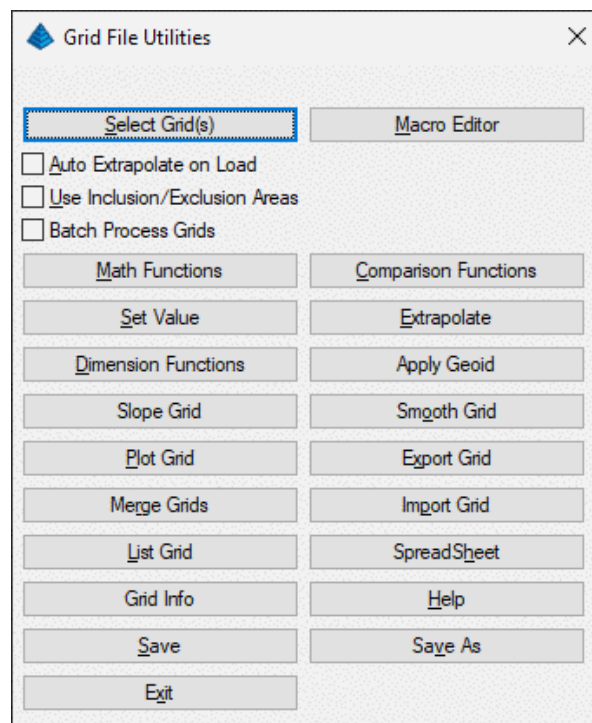
Pulldown Menu Location: Surface

Keyboard Command: mkgrid

Prerequisite: Entities that define the surface

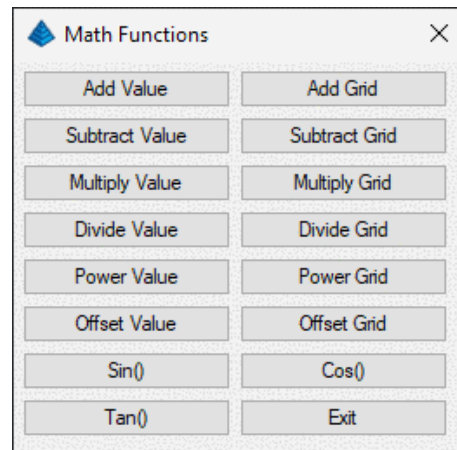
Grid File Utilities

This command is used to modify and create grid files. The modifications can be done manually on a single grid, on multiple grids in a batch mode, or saved and rerun using the grid macros created with the macro editor. To modify manually, start by picking the Select Grid(s) button. There is an option to use inclusion and exclusion polylines to only modify the grid within/outside these perimeters. With this option active, the program will prompt for inclusion and exclusion polylines when a function is selected. Only grid cells inside the inclusion polylines will be modified. Grid cells inside the exclusion polylines will not be modified. If no inclusion and exclusion polylines are selected, then the entire grid will be modified. Each function is described below.



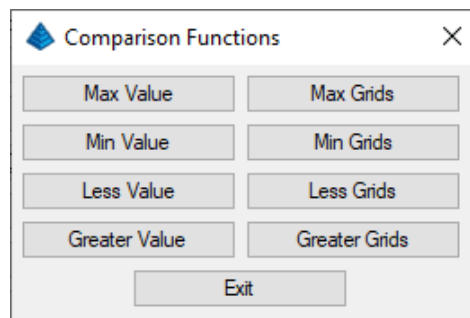
- **Select Grid(s):** This is the first step to load a grid. Usually a grid needs to be loaded before running a function. If Batch Process Grids is turned on, then multiple grids may be selected while holding down the Shift or CTRL buttons.

- **Auto Extrapolate On Load:** This will extrapolate values for any null or empty values in the grid as the grid is loaded.
- **Use Inclusion/Exclusion Areas:** If this is turned on, then the GFU function will only be applied within the selected inclusion polyline and outside the selected exclusion polyline.
- **Batch Process Grids:** When this option is turned on, GFU functions can be executed on many grids at once. It is recommended to move the grids to a backup directory, or create a copy of them, as the grids are over-written with the same name. The functions that cannot be batched are: Plot Grid, Merge Grid, List Grid, Import Grid, and Spreadsheet.
- **Math Functions:**

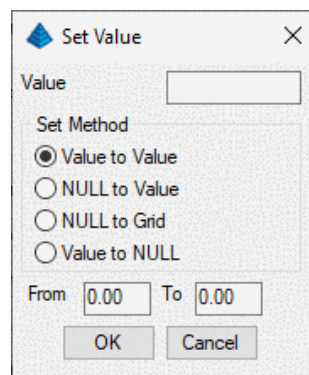


- **Add Value:** Adds an entered value to the grid values. ($\text{GridA} + X$)
- **Add Grid:** Adds one grid to another grid. ($\text{GridA} + \text{GridB}$)
- **Subtract Value:** Subtracts an entered value from the grid values. ($\text{GridA} - X$)
- **Subtract Grid:** Subtracts one grid from another grid. ($\text{GridA} - \text{GridB}$)
- **Multiply Value:** Multiplies the grid values by an entered value. ($\text{GridA} * X$)
- **Multiply Grid:** Multiplies the grid values by another grid. ($\text{GridA} * \text{GridB}$)
- **Divide Value:** Divides the grid values by an entered value. (GridA / X)
- **Divide Grid:** Divides the grid values by another grid. ($\text{GridA} / \text{GridB}$)
- **Power Value:** Raises the grid values to the specified power. ($\text{GridA} ^X$)
- **Power Grid:** Raises the grid values to another grid for the "power". ($\text{GridA} ^\text{GridB}$)
- **Offset Value:** Offsets the grid elevations perpendicular to the grid surface. Use a negative value to offset down.
- **Tan(), Sin() and Cos():** Applies these functions to the grid values which need to be in decimal degrees. ($\tan(\text{GridA})$, $\sin(\text{GridA})$, $\cos(\text{GridA})$)

- **Comparison Functions:**

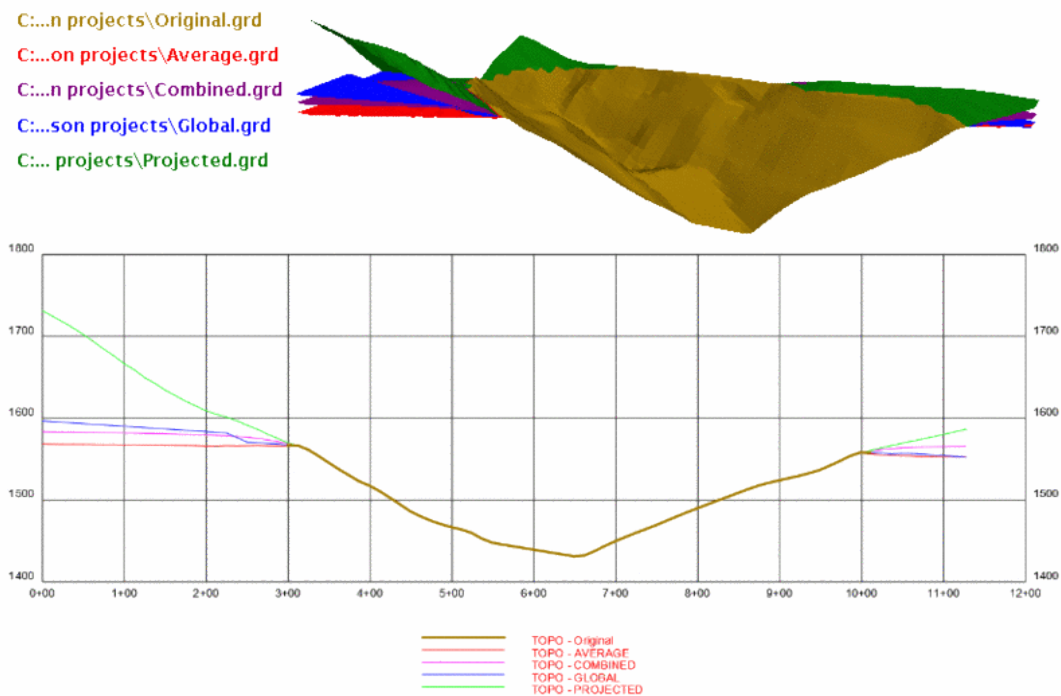


- **Max Value:** Compares a grid and a value and takes the Maximum value of either. This is a way to stop a grid from going negative, below zero.
 - **Max Grids:** Compares a grid with another grid and takes the Maximum (higher) value of either.
 - **Min Value:** Compares a grid and a value and takes the Minimum value of either. This is a good way to cap a grid off at a certain value so it never goes higher than the specified value.
 - **Min Grids:** Compares a grid with another grid and takes the Minimum (lesser) value of either.
 - **Less Value:** Asks for a value to compare and a value to assign and uses the following logic:
If $\text{GridA} < \text{compare_value}$ then $\text{GridA} = \text{assign_value}$, otherwise no change
 - **Less Grids:** Asks for a grid to compare and a grid to assign and uses the following logic:
If $\text{GridA} < \text{compare_GridB}$ then $\text{GridA} = \text{GridC}$, otherwise no change
 - **Greater Value:** Asks for a value to compare and a value to assign and uses the following logic:
If $\text{GridA} > \text{compare_value}$ then $\text{GridA} = \text{assign_value}$, otherwise no change
 - **Greater Grids:** Asks for a grid to compare and a grid to assign and uses the following logic:
If $\text{GridA} > \text{compare_GridB}$ then $\text{GridA} = \text{GridC}$, otherwise no change
- **Set Value:** assigns the grid elevations to the user-specified value. Using inclusion and exclusion perimeters are usually required for this command. For example by using Set Value with the inclusion perimeter option, you could set the grid values to 0.0 within the inclusion polyline for a strata thickness grid. The four options are:
 - Value to Value will set all values to one value
 - Null to Value will set all Nulls to one value
 - Null to Grid will set all Nulls to another specified grid
 - Value to Null will set all values to Null.

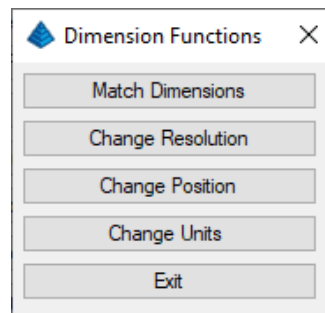


- **Extrapolate:** This function will assign values to null grid nodes by one of four methods.
 - Global Trend finds the average slope and slope direction from the existing grid elevations and applies this slope to calculate the missing elevations.
 - Average method calculates a grid elevation as the average of its nearest neighbors. This will create a very flat extrapolation.
 - Projected method extends the trend at the edge (as opposed to finding an average slope of the entire surface like the Global option).
 - Combined method uses a combination of both the Average and Projected methods.

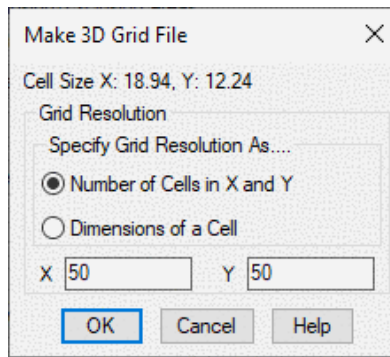
An example extrapolation of a surface using each method is shown below in 3D and in a profile view. The Original surface (brown color) is an example valley that only contains data in the center of the grid.



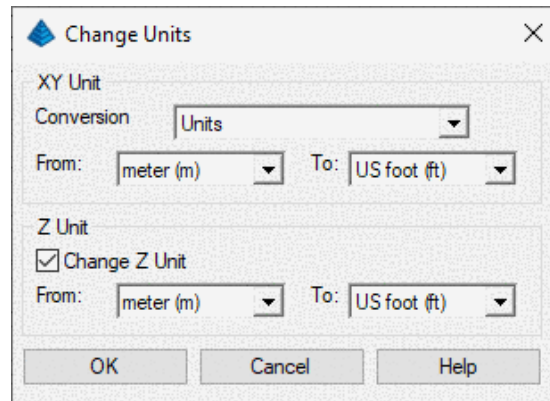
- **Dimension Functions:**



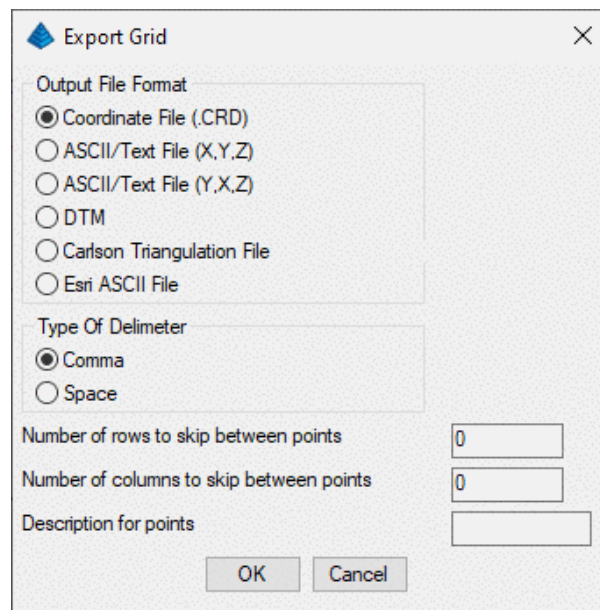
- **Match Dimensions:** Sets the grid position and resolution to match another grid file. The program will prompt for a grid file to get the position from. Certain commands require grids match position and resolution. Running this command will ensure grids will match.
- **Change Position:** This lets you change the lower left and upper right corners of the grid file. For example, you can use this routine to localize a grid file if you have a large grid for the entire site but are currently working on a smaller area. If the new position covers area outside the original position, any grid cells in this area will be assigned a null value. Otherwise the program uses the original grid values for the new grid position.
- **Change Resolution:** This changes the grid resolution (number or dimensions of grid cells). The program uses the original grid values for calculating the grid values at the new resolution. Enter a new value for X and Y number of cells or dimensions of cells.



- **Change Units:** Scales the grid X/Y and/or Z values to switch units such as meters to feet.



- **Apply Geoid:** Adjusts the grid elevations by the selected geoid. To use this function, the grid projection must be defined in the Settings > Drawing Setup command.
- **Slope Grid:** Converts a grid of elevations to a grid of slope values expressed as a percentage.
- **Smooth Grid:** This function has three smoothing method. The **Least-Squares** method applies a moving least-squares algorithm to adjust the grid elevations. The **Eliminate Spikes** method adjusts grid nodes with elevations too different (spikes) from neighbors within the **Search Radius**. The **Search Scaler** is used to identify spikes to adjust by using this scaler multiplied by the standard deviation of the grid values within the search radius. The **Interpolate Radius** is used to select the grid values for recalculating the grid value at the spike. **Quadratic Smoothing** uses neighboring nodes to adjust each grid node. This routine can be used to refine a grid so that the contours from the Contour from Grid routine appear smoother. Typically this adjustment is relatively small. To get more smoothing, run the routine more times.
- **Plot Grid:** This runs the Draw 3D Grid command.
- **Export Grid:** There are several choices for export options: Carlson Coordinate File (CRD), ASCII text as XYZ, ASCII text as YXZ, DTM, Digital Elevation Model (DEM), TIF, Carlson Triangulation (TIN) and Esri (ASC). There are two options for the ASCII delimiter, either a comma or a space. There is an option to skip a number of rows and columns between the exported points. When exporting into the Carlson CRD file, the description for the points is set at the bottom.



Export to DTM writes the current grid file to a DTM format text file. The format of this file is the following:

DTM 1.0 Header Line

test.dtm Name of file

51 Number of cells in X direction

51 Number of cells in Y direction

79442.4697 Lower left grid corner Y coordinate

14899.0326 Lower left grid corner X coordinate

0.0 Lower left grid corner Z coordinate

11.5618 Dimension of cell in X direction

7.0639 Dimension of cell in Y direction

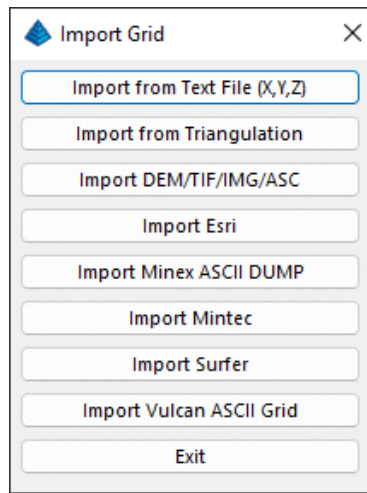
1581.2612 Grid cell values starting from lower left, moving from left to right

1580.8879

1580.3257

etc...

- **Merge Grids:** creates a grid file by merging together two existing grid files, grid1 and grid2. The current grid is grid1 and the program will prompt for a second grid. These two grids must overlap with the same location and resolution. The inclusion and exclusion perimeters apply to grid2 such that the merged grid will consist of grid2 cells within the inclusion perimeters and outside the exclusion perimeters and grid1 cells everywhere else. The result is stored in the current grid.
- **Import Grid:** There are several formats that may be imported.
 - Text File (ASCII): This function allows for various formats. The data can be comma or space separated or in fixed width columns.
 - Carlson Triangulation (TIN, FLT)
 - Digital Elevation Model (.DEM, .TIF .IMG or .ASC) such as from the USGS (US Geological Survey)
 - Esri (ADF)
 - Minex ASCII DUMP (ASC, CSV)
 - Mintec
 - Surfer (GRD both ASCII and Binary)
 - Vulcan ASCII Grid (ASC, CSV)



Import from Text File (X, Y, Z) creates a grid file from X Y Z data in any text file. There does not need to be a current grid file loaded since this routine will create a grid file. The text file should consist of one X Y Z coordinate per row with the first coordinate being the lower left grid corner and the last coordinate as the upper right grid corner. There are options for space or comma separated coordinates and for the order of the coordinates as either row (left to right) or column (bottom to top). The prompting will be as follows:

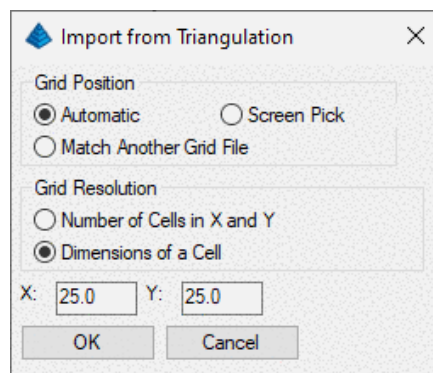
Separation type [<Space>/Comma/FixedWidth]?

Column number for X coordinate <1>:

Column number for Y coordinate <2>:

Column number for Z value <3>:

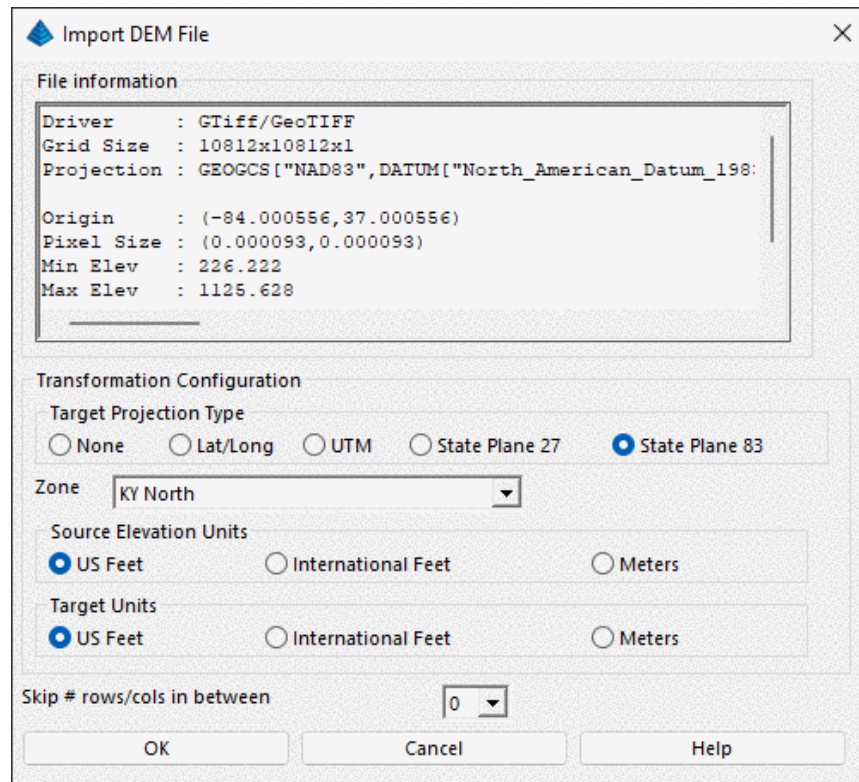
Import from Triangulation prompts user to select a tin or flt file and allows user to adjust grid position and resolution. The grid file is created with the same name in the same directory as the selected tin/flt file.



Import DEM/ESRI prompts the user to select a DEM, ADF, or ASCII ESRI grid file to be imported. The ESRI grid files can be created from ArcMap using the Raster To ASCII tool. If the file format is recognized, the program reads, and displays information about the source projection of the DEM/ADF/ESRI file, and allows the user to define a target projection for transforming the grid to a local coordinate system. the Source Elevation Units define the units of the Z values if they cannot be automatically detected. The Target Units will set the units for the X, Y, and Z values of the new grid. Note that if the Source Elevation Units and the Target Units use different units, the Z values will be scaled accordingly.

The "Skip every # rows/cols" option allows the user to reduce the size of the imported grid file. In the case of a DEM/ESRI files each pixel of elevation information of the DEM/ADF image data is interpreted as representing an elevated point located in the center of that pixel (not multiple points located on the edges of that pixel). In contrast, the "Import DEM/ADF File Dialog" will output the "Origin" of the DEM/ADF file in terms of the raster pixel extents. The output grid file is created with the same name in the same directory as selected DEM/ADF/ESRI

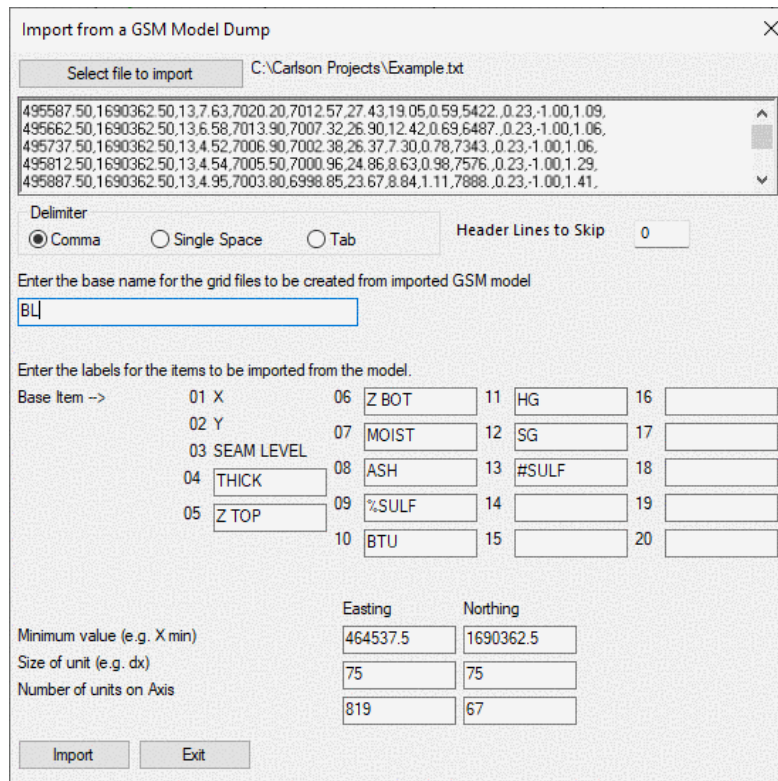
file.



Import Minex ASCII DUMP allows user to import multiple Minex ASCII grids from a single .asc or .csv file. The imported grids will be automatically named according to the header information in the .asc/.csv file. An example File Format is provided below.

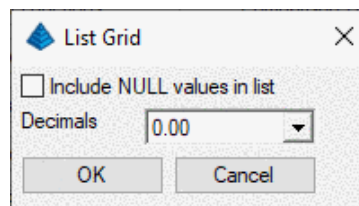
```
; origin      3051045.00      952377.25
; extent      5375.50      4042.63
; mesh        15.00      15.00
; rotation    0 00 00
; x    1    12
; y    13    12
; SeamRoof    25 10
; SeamFloor   35 10
; SeamThick   45 10
3051052.50    953764.75      920.76
3051052.50    953779.75      922.01
3051052.50    953794.75      923.29
3051052.50    953809.75      924.60
3051052.50    953824.75      928.38      925.93
3051052.50    953839.75      940.24      927.30
3051052.50    953854.75      941.63      928.69
3051052.50    953869.75      943.06      930.11
3051052.50    953884.75      944.52      931.56
3051052.50    953899.75      946.01      933.04
3051052.50    953914.75      947.54      934.55
3051052.50    953929.75      949.10      936.10
```

Import Mintec allows user to import Mintec GSM Model Dump as grd files. User is prompted to select GSM Model Dump (txt) file, which is then processed to determine minimum northing, easting (lower left corner), resolution and size of the grid. First three columns of the GSM model dump must represent the X, Y and SEAM LEVEL respectively, a base name for the grid files is specified along with name of the quality that each column represents. User can define up to 17 qualities. When the import button is pressed all the imported grids are created with the name "BASE NAME-SEAM LEVEL-BASE ITEM.grd" in the same directory as source dump file.



Import Surfer prompts to select a grd file from Surfer program and creates a grd file.

- **List Grid:** displays a list of the northing, easting and elevation of each grid corner. There is an option to Include NULL values in the list. A grid node will have no value, or a Null value (listed as None) if the grid node was outside the limits of the data during Make 3D Grid File.



Carlson Software Edit : C:\Users\tfaulkner.CARLSON\...

File Edit Settings

Cell#	East (x)	North (y)	Elev (z)
55	540100.00	157050.00	1380.25
56	540200.00	157050.00	1404.41
57	540300.00	157050.00	1431.12
58	540400.00	157050.00	1440.00
59	540500.00	157050.00	1440.00
60	540600.00	157050.00	1440.00
61	540700.00	157050.00	1440.00
62	540800.00	157050.00	1440.00
63	540900.00	157050.00	1440.00
64	541000.00	157050.00	1436.77
65	541100.00	157050.00	1432.11
66	541200.00	157050.00	1428.90
67	541300.00	157050.00	1426.21
68	541400.00	157050.00	1424.52
69	541500.00	157050.00	1423.21
70	541600.00	157050.00	1421.83
71	541700.00	157050.00	1418.36
72	541800.00	157050.00	1411.76
73	541900.00	157050.00	1406.39
74	542000.00	157050.00	1398.28
75	542100.00	157050.00	1372.76
76	542200.00	157050.00	1355.95
77	542300.00	157050.00	1352.00
78	542400.00	157050.00	1351.04
79	542500.00	157050.00	1354.00

- **Spreadsheet:** displays the grid elevations in a row and column spreadsheet that is in the same layout as the grid file. Grid elevations can be edited in this spreadsheet and saved upon exiting the spreadsheet.

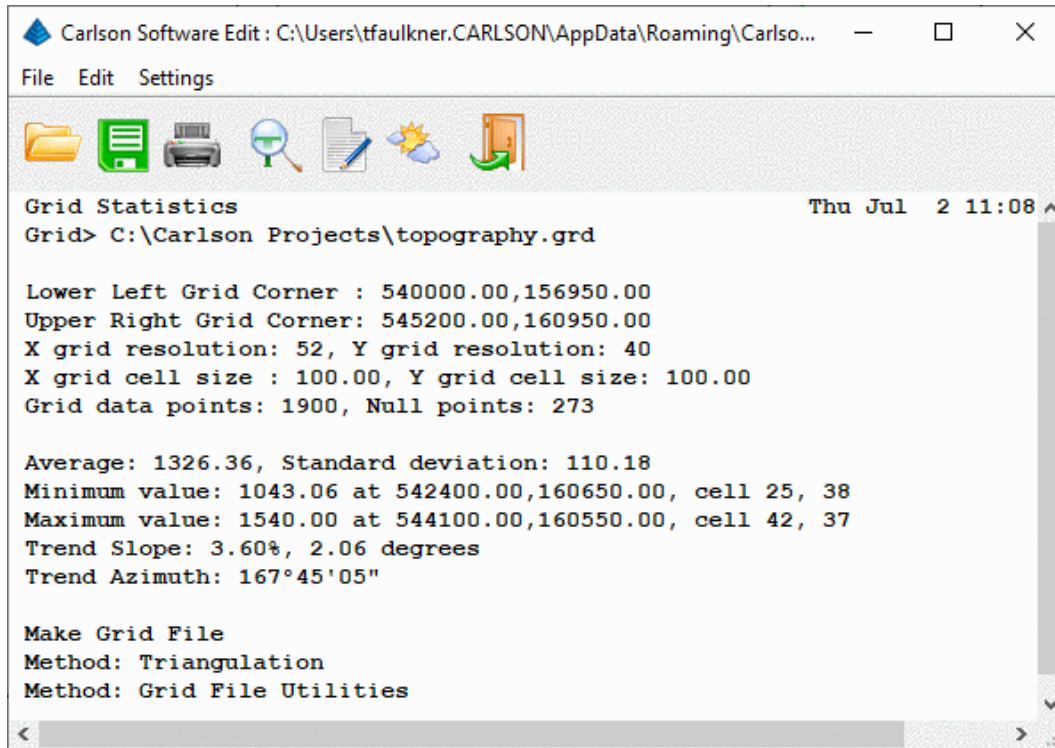
Grid Data Sheet

File

	19	20	21	22	23	24	25	
37	1172.9800	1187.6800	1196.2600	1181.6900	1153.4900	1114.9000	1065.3600	108
36	1222.6500	1240.8700	1252.8000	1216.0700	1173.0200	1130.7200	1078.8100	108
35	1282.9700	1298.5100	1281.1100	1238.4400	1186.5300	1141.9800	1093.7400	106
34	1340.6600	1340.0000	1306.0100	1253.2400	1196.3400	1154.9000	1107.0500	105
33	1343.7100	1322.3400	1285.5400	1249.6900	1191.9600	1163.1700	1115.7900	106
32	1295.3000	1255.6000	1222.1300	1194.7100	1170.8000	1142.8600	1106.1500	106
31	1237.7000	1210.0100	1184.3700	1159.1000	1133.6700	1100.0600	1070.4200	107
30	1190.9800	1172.8700	1148.5300	1119.7800	1088.9600	1101.7200	1115.7900	112
29	1154.5600	1137.0200	1110.5800	1102.7500	1119.2100	1135.5100	1151.2100	116
28	1119.9500	1110.2200	1128.2200	1141.1400	1153.6600	1169.9300	1194.0500	123
27	1140.7700	1150.7100	1158.5900	1172.5700	1188.4700	1212.5000	1243.2100	127
26	1170.0100	1180.8900	1199.6400	1225.0700	1236.0300	1264.4300	1301.2900	133
25	1229.3600	1246.8800	1258.4100	1281.1900	1292.6900	1311.1400	1351.5100	136

- **Grid Info:** This function displays information about the grid file. It is a form of Grid Statistics. The items it

displays are shown in the report below:



Macro Editor: The Macro Command Recorder allows you to create a script for grid modification and save these steps to a .gfu file. These macros can be rerun to repeat grid modifications when the source grids are modified, thus saving time on tedious manipulation. Macros can also be reviewed for accuracy, whereas manual manipulation of grid files cannot be checked with as much detail. When writing a macro, you essentially perform the normal grid manipulations using the functions on the left of the dialog. Although the prompts will be slightly different than manual manipulation, the steps will be automatically added to the text window. You can then copy/paste the text as needed to speed up the macro-writing process.

Quick notes on the macro editor:

- You may enter the script manually (typing each variable, function, etc), or you may use the function buttons on the left. Some functions are more easily written manually compared to using the function dialogs on the left.
- Function templates may be inserted using the "Template to Insert" droplist on the left. Using these templates, you can see an example of the syntax for each function.
- You may add comments to the script to allow for easier review. Comments are simply preceded by a semi-colon (;). With the exception of the PERIM, INCLU, and EXCLU functions, any text following a semi-colon will be seen as a comment (the program will not attempt to read it upon execution of the script)
- Most all function follow a syntax of RESULT=FUNCTION. For example, to evaluate the result of adding Grids A and B together to create Grid C, the function would be written as C=A+B.
- When a variable is first defined, the variable will take on the grid dimensions (cell size and lateral extent) of the right side of the equal sign. For Example, consider an exmaple in which variable A is defined as a grid and B is not yet defined. If you write the equation B = A + 5, then variable B will cover the same lateral extent and will use the same grid cell size as variable A.

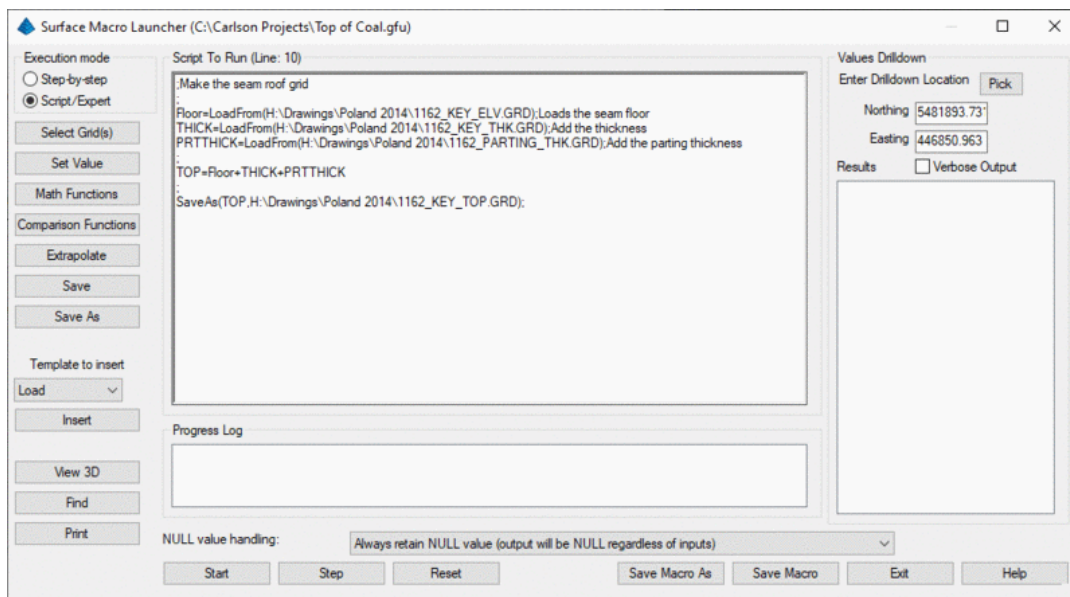
If two grid variables do not use the same grid dimensions, they can still be compared (note that this is not possible with manual manipulation of the grids). For example, if Grid A and Grid B cover only partially overlap, the equation B = A + 5 will allow Grid B to retain it's original dimensions, but it will only be updated where it overlaps A. In this way, you can avoid the intermediate step of settings grids to the same location/cell

size in order to compare them.

- When using the functions on the left side of the dialog (Math Functions, Comparison Functions, etc), you will sometimes be able to select existing grid variables from a dropdown list. In order for this list to be populated with the existing grid variables, you must first Start the macro, otherwise the program will not display the existing grid variables.
- When adding a step that involves another grid file, there are three options for storing this grid into the script. Use Grid Variable will use the grid assigned to the Grid Variable Name. Prompt for Grid File Name will bring up a grid file selection dialog each time the macro is run. Store the Grid File Name will save the specific grid file name into the macro.

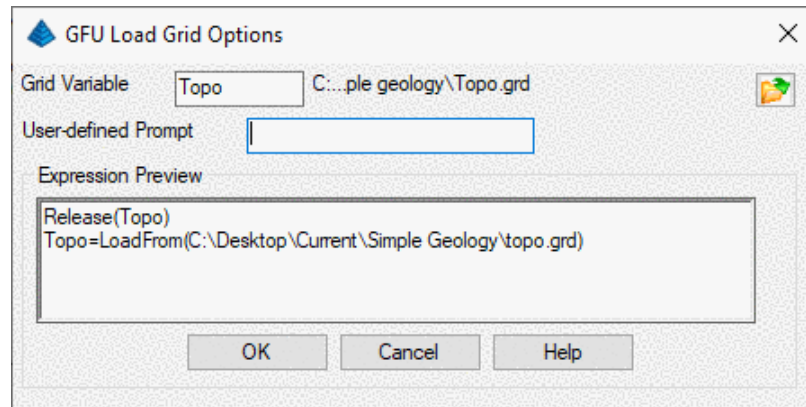
Surface Macro Launcher Details

The Surface Macro Launcher displays the GFU file name in the upper menu bar. This is an editor, and you can use basic functions like CTRL-X, CTRL-C & CTRL-V for cut, copy, & paste. GFU files also can be edited easily in any text editor, such as Notepad, WordPad or K-Edit. If there are any errors in the GFU during its execution, they will be displayed in the error log section when the macro runs. The Values Drilldown on the right side of the dialog allows you to inspect the calculations/grid variables at an actual Northing-Easting location, which is especially useful for checking that the results are correct. You may screen-pick a location by clicking the Pick button, or you may manually enter the Northing and Easting coordinates. As the macro runs, each variable will be display in the text readout below these options. If the Verbose Output option is disabled, only a summary of the grid variables will be displayed, with each variable updating as the macro runs. If the Verbose Output option is enabled, the results from each line of the macro will be displayed in the text readout.



- **Step-by-Step:** This execution mode is a simpler method and is intended to be executed immediately. It is similar to the previous windows simple "button pushing" to execute a function. These two modes have completely different dialogs when the functions are selected.
- **Script/Expert:** This mode stores the history and is a replayable history of the operations. It contains all of the higher level features and options, and is for the users who want to program it manually.

- **Select Grids:** In the Step-by-Step mode, the File Open dialog appears to select the file. In Script/Expert mode, the following dialog initially appears to add the expression to the macro for replaying. This function releases a variable if it is previously defined, and loads a grid file and assigns a variable to it. Another option is to enter an User-defined Prompt instructing the user on which grid to select at the time the macro is run. The dialog appears as shown. The preview shows how it will look when inserted into the macro.



- **Set Value:** This function sets the grid to new values based on the selected options. The Step-by-Step method has the following Set Functions shown in the first image below. The Expert method functions are shown in the second image. The expression #1 can either be set to a Value/Expression or to a New Grid file. The operation type contains four options of Set Value.

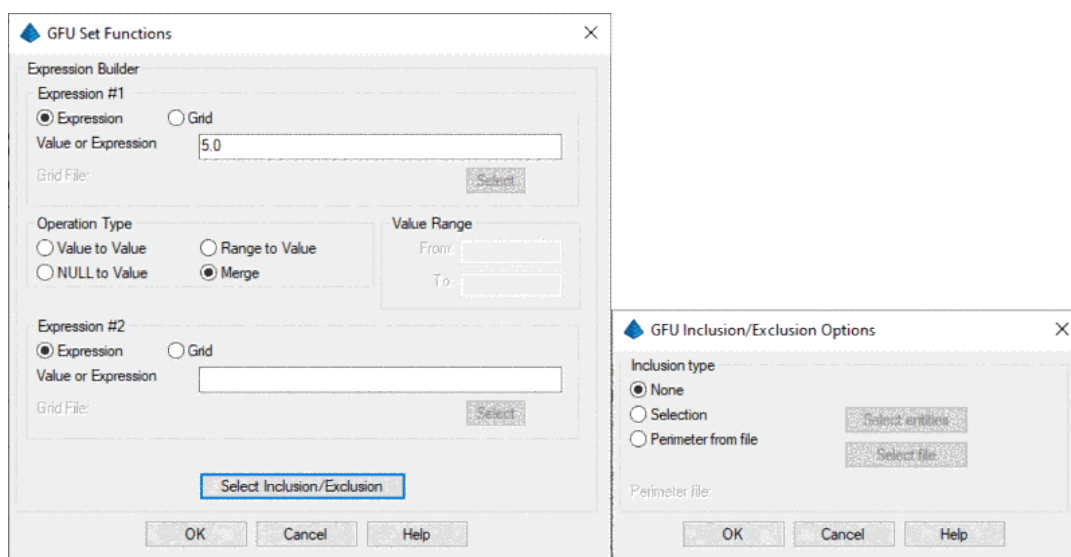
Value to Value (A=B for each point of A, sets the value to the value of B, Expr#1))

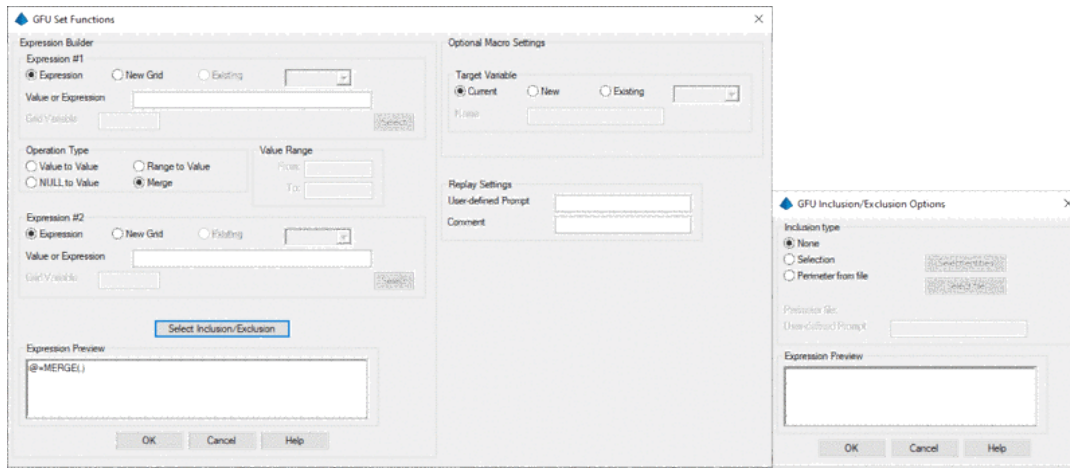
Null to Value (C=SET_NULL (A,B), For each point of C, sets the value to the value of A (Expr#1) if defined and the value of B (Expr#2) otherwise)

Range to Value (A=CHANGE_RANGE(B,C,D,E), for each point of A, if value of B (Expr#1) is between C and D, sets the value to value of E (Expr#2) or Null if not specified)

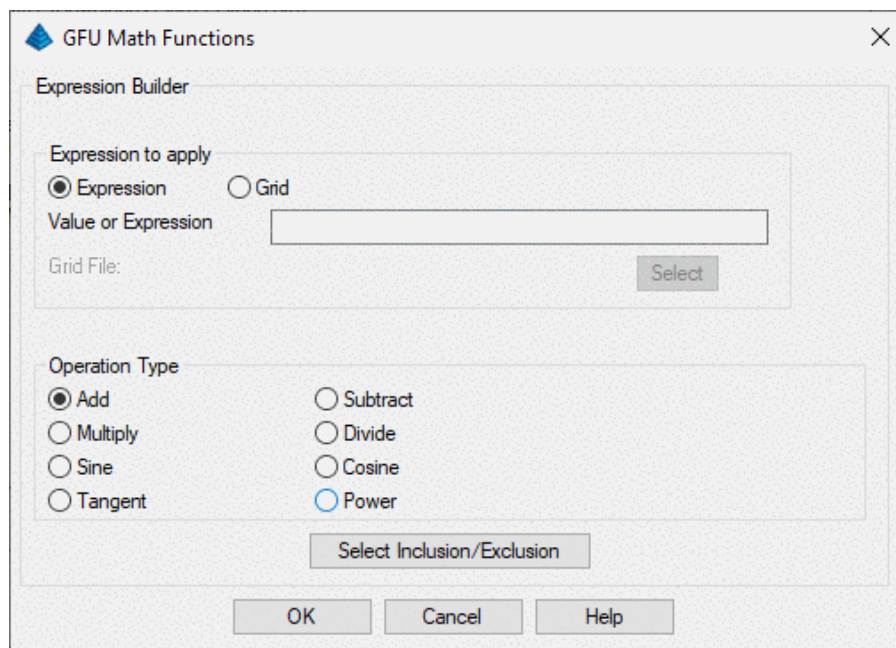
Merge (C=MERGE(A,B), for each point of C, sets the value to the value of B (Expr#2) if defined and the value of A (Expr#1) otherwise). Expression#2 is what the value will be set to, or to a new grid.

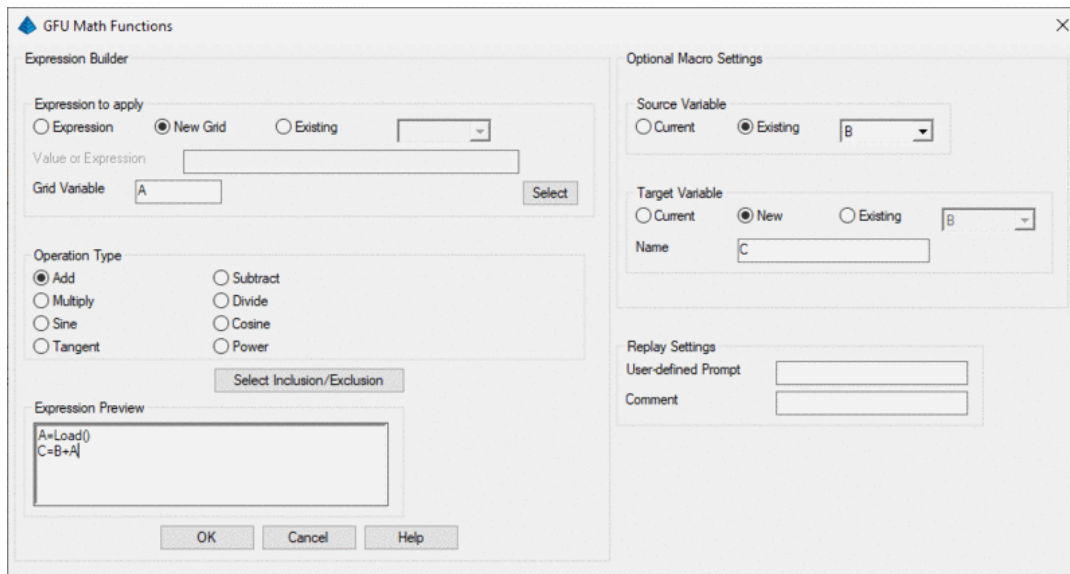
The Select Inclusion/Exclusion options can be set to None, Selection (manually select polyline), or a PLN polyline file. Note that PLN files may be created with the Export Polyline File command. The preview of the expression appears in the expert mode. These two modes appear the same in all the various functions. To set a grid to NULL, where the values fall within a range, use the following format: A=CHANGE_RANGE_VALUE(A,lowrange,highrange,NULL)



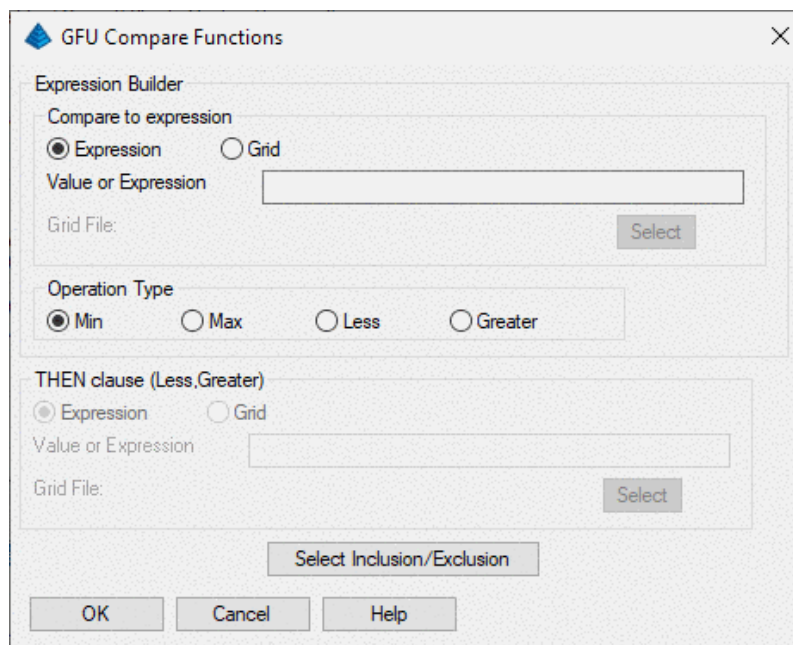


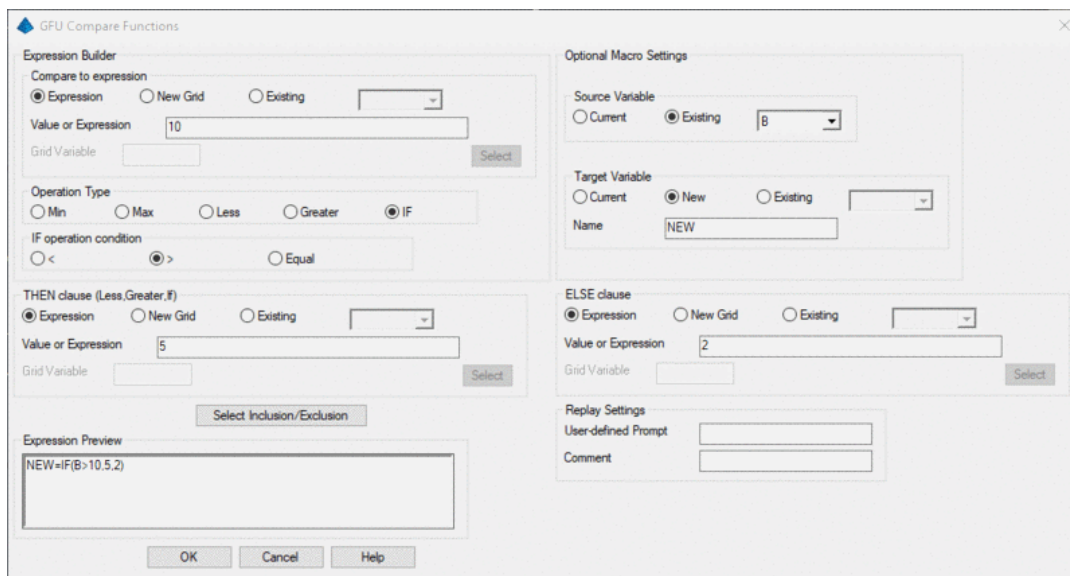
- Math Functions:** These functions apply the mathematical operations selected to the loaded grid. The Step-by-Step mode dialog is shown first, followed by the Script/Expert mode. The operations are Add, Subtract, Multiply, Divide, Sine, Cosine, and Power function.



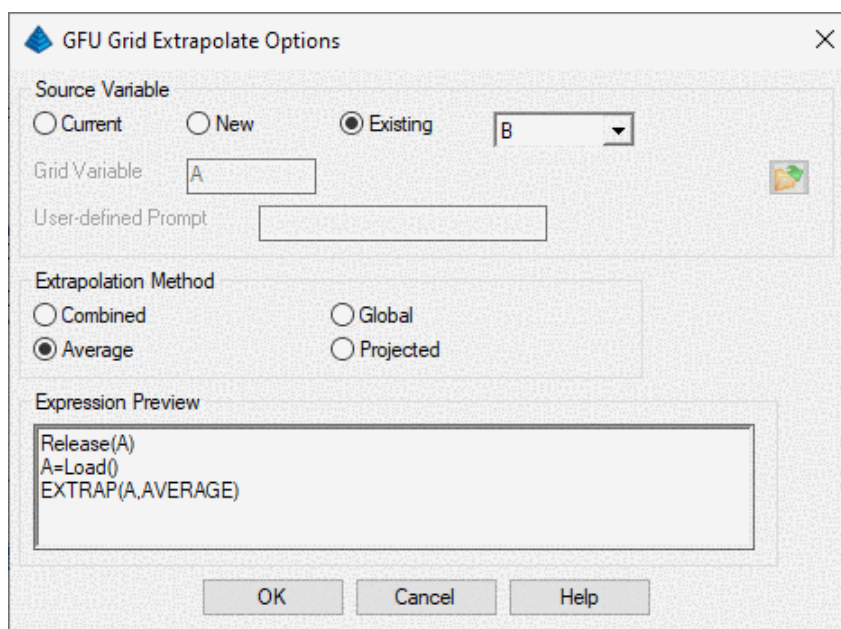
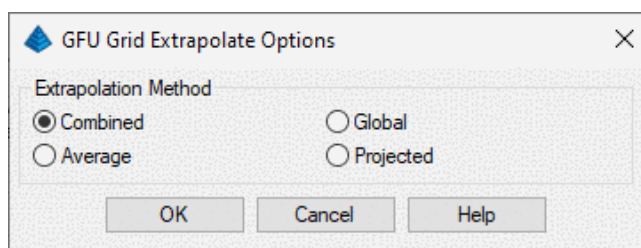


- Comparison Functions:** The Step-by-Step mode uses the Min, Max, Less, and Greater commands for grid editing. This is shown below in the first dialog. The second one shows the Script/Expert dialog that also contains the If Statement functions. The IF statement is the standard format of IF This, Then This, Else This. The Preview shows how it will appear in the GFU syntax.

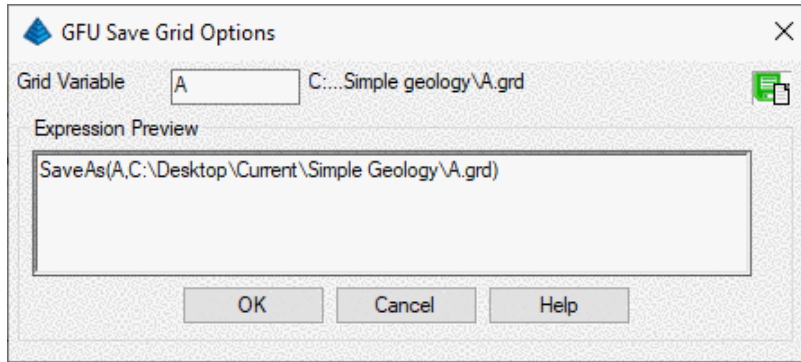




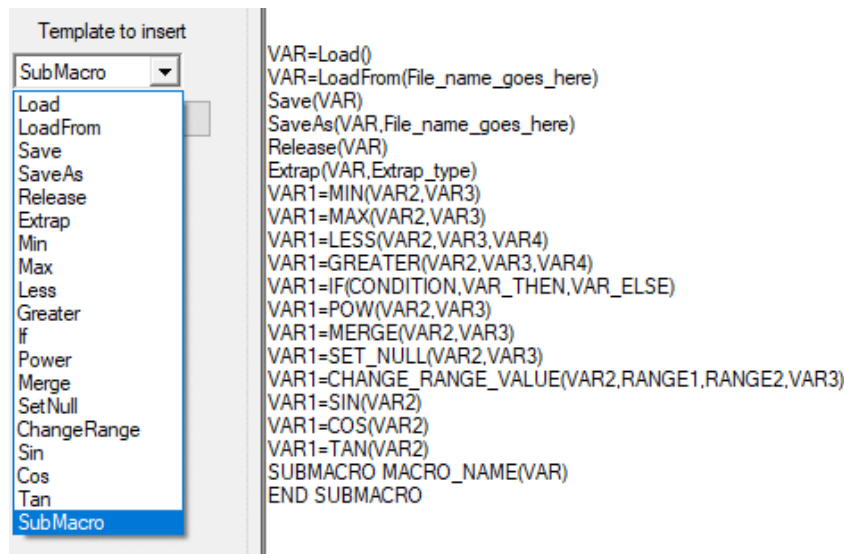
- **Extrapolate:** This function uses the four methods described above to extrapolate the grid to fill in any Null values. Details on each extrapolation method are provided above.



- **Save/Save As:** These commands add the syntax of saving to the macro. The variable can be the existing, or set to a new one here.



- **Template to insert:** This option allows for easy insertion of the syntax for the following functions.



- **Nested Submacros:** It is possible to define a macro for a certain function, and then call that macro and apply it to other variables in either the same macro, or in a different macro. Here are 3 examples of Submacros.

Submacro in the same macro that calls it. There are two submacros defined at the top, and then they are both called at the bottom, THICK2 and THICK.

```

SUBMACRO THICK()
A1=LOADFROM($PROJECTPATH\one.grd)
B1=LOADFROM($PROJECTPATH\two.grd)
C1=A1+B1;
SAVEAS(C1,$PROJECTPATH\three.grd)
END SUBMACRO
;
SUBMACRO THICK2(A,B,D)
C=A+B;
SAVEAS(C,D);
END SUBMACRO
;
A1=LOADFROM($PROJECTPATH\one.grd);
B1=LOADFROM($PROJECTPATH\two.grd);

```

```
THICK2(a1,B1,$PROJECTPATH\three_b.grd);
THICK()
```

Submacro in a different macro that calls it. The first step is to define the submacros in their own gfu, this one contains THICK and THICK2 and is saved as mymacros.gfu.

```
SUBMACRO THICK()
A1=LOADFROM(C:\downloads\one.grd);
B1=LOADFROM(C:\downloads\two.grd);
C1=A1+B1;
SAVEAS(C1,C:\downloads\three.grd);
END SUBMACRO;
;_____
SUBMACRO THICK2(A,B,D)
C=A+B;
SAVEAS(C,D);
END SUBMACRO;
```

INCLUDE: To include a previously defined macro in another macro, the INCLUDE function is used. Once mymacros.gfu is created, then it can be loaded (with INCLUDE) and referenced in another, separate GFU macro.

```
INCLUDE($PROJECTPATH\mymacros.gfu);

A1=LOADFROM($PROJECTPATH\one.grd);
B1=LOADFROM($PROJECTPATH\two.grd);
THICK2(a1,B1,$PROJECTPATH\three_b.grd);
THICK();
```

Here is an example, with comments, where the bottom elevation and the thickness are added together to get the roof, for 6 seams. The Submacro is defined by the equation just once. Then it is called and used 6 times at the bottom of the macro.

```
;Add the Floor and Thickness to get the seam Roof.
;Coal seams to apply this to are: C1,C2,C3,C4,C5,C6
;_____
;LOAD THE GRIDS INTO MEMORY
;
C1_KEY_ELV=LOADFROM(C:\Carlson Projects\C1_KEY_ELV.GRD)
C1_KEY_THK=LOADFROM(C:\Carlson Projects\C1_KEY_THK.GRD)
C2_KEY_ELV=LOADFROM(C:\Carlson Projects\C2_KEY_ELV.GRD)
C2_KEY_THK=LOADFROM(C:\Carlson Projects\C2_KEY_THK.GRD)
C3_KEY_ELV=LOADFROM(C:\Carlson Projects\C3_KEY_ELV.GRD)
C3_KEY_THK=LOADFROM(C:\Carlson Projects\C3_KEY_THK.GRD)
C4_KEY_ELV=LOADFROM(C:\Carlson Projects\C4_KEY_ELV.GRD)
C4_KEY_THK=LOADFROM(C:\Carlson Projects\C4_KEY_THK.GRD)
C5_KEY_ELV=LOADFROM(C:\Carlson Projects\C5_KEY_ELV.GRD)
C5_KEY_THK=LOADFROM(C:\Carlson Projects\C5_KEY_THK.GRD)
C6_KEY_ELV=LOADFROM(C:\Carlson Projects\C6_KEY_ELV.GRD)
C6_KEY_THK=LOADFROM(C:\Carlson Projects\C6_KEY_THK.GRD)
;_____
;DEFINE THE SUBMACRO EQUATION
;
```

SUBMACRO ROOF(X,Y,Z)

TOP=X+Y

SAVEAS(TOP,Z)

END SUBMACRO

;
;

ROOF(C1_KEY_ELV,C1_KEY_THK,C:\Carlson Projects\C1_KEY_TOP.grd)

ROOF(C2_KEY_ELV,C2_KEY_THK,C:\Carlson Projects\C2_KEY_TOP.grd)

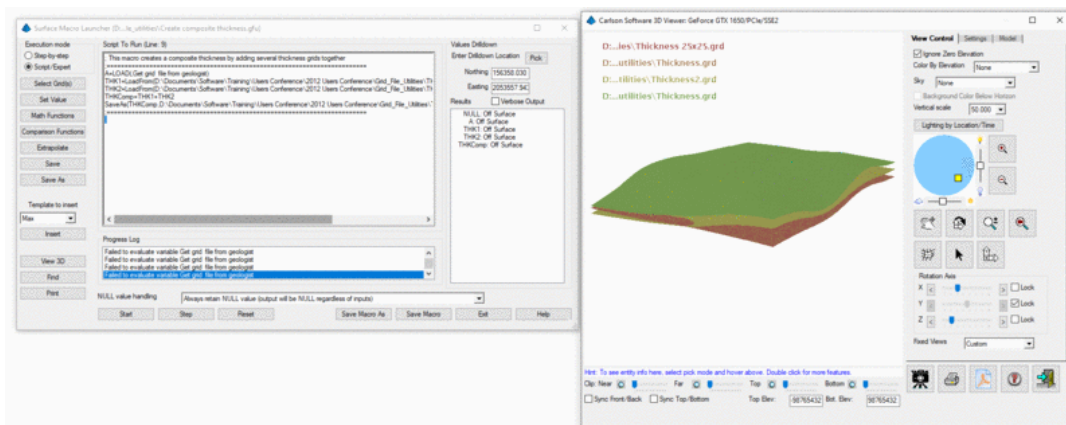
ROOF(C3_KEY_ELV,C3_KEY_THK,C:\Carlson Projects\C3_KEY_TOP.grd)

ROOF(C4_KEY_ELV,C4_KEY_THK,C:\Carlson Projects\C4_KEY_TOP.grd)

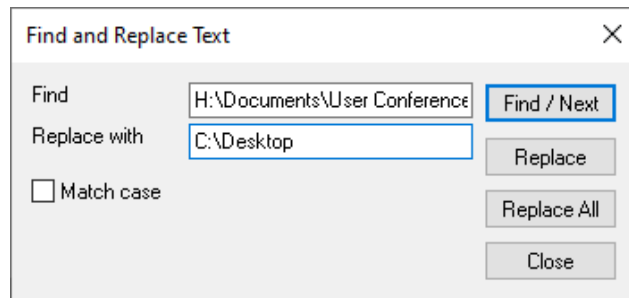
ROOF(C5_KEY_ELV,C5_KEY_THK,C:\Carlson Projects\C5_KEY_TOP.grd)

ROOF(C6_KEY_ELV,C6_KEY_THK,C:\Carlson Projects\C6_KEY_TOP.grd)

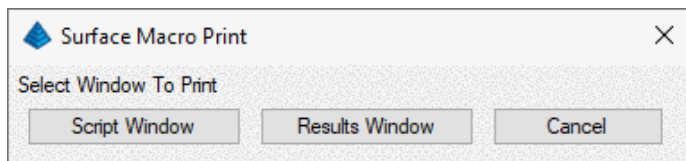
- **View 3D:** Using this option brings up the 3D Surface Viewer that steps through live all of the grids and displays them in 3D. This can be left open and moved to the side, or to another monitor to see all the surfaces and how they are modified, in a 3D view.



- **Find:** This is a Find, or Find and Replace to search the GFU text and make edits. It is useful if a file path has changed and all need to be reset to somewhere else.



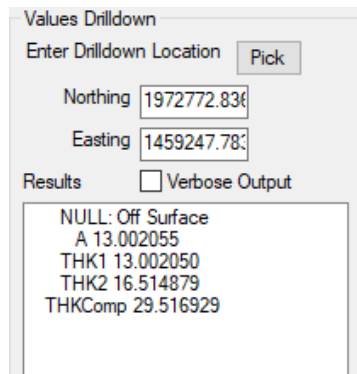
- **Print:** There are two options for printing. The Script Window prints out the entire GFU syntax. The Results Window prints out



- **NULL Value Handling:** This option determines how null values will be handled in the grid calculations. The "Always retain NULL Value (output will be NULL Regardless of inputs)" option will force any calculations with NULL inputs to create a NULL output. The "Treat NULL as 0.0 if one or more inputs are non-NULL";

Retain NULL if all inputs are NULL)" option will replace null values with zeros for calculations, but will retain null values in the output if all inputs are null.

- **Start/Step/Reset:** These buttons allow for individual stepping through the macro. It will advance only one line at a time with the Step. Start will run the entire macro. Reset will begin back at the beginning.
- **Save Macro As/Save Macro:** This saves the GFU macro either as its original name, or as a new name with Save As.
- **Values Drilldown:** The Values Drilldown is a good method for error checking on the macro. Use the Pick button to select a spot in plan view to fill in the Northing and Easting boxes. Then when the GFU is executed, the results of each line will be displayed. The Verbose Output will show the value of each line L1, L2, etc., instead of overwriting the variable each time it is encountered.



Variables

Variables in the GFU can be any keywords providing meaningful identification of the data loaded. A variable can be either just a value or constant; or most commonly a surface (Grid or TIN). Individual macro lines typically have one of the following forms:

Variable1=Variable2

Variable1=Expression

Variable1=Function(Expression1,Expression2)

Whenever new variable name is encountered on the left side of the equation, the new variable will be created. The program will use its knowledge of the right side of equation to define a type of the new variable. For example:

A=1.0 Variable A will be just a value

A=LoadFrom(abc.grd) Variable A is a grid loaded from file

B=A Variable B is same variable type as A

B=(1+C+A)/D Variable B will be 1+C+A, all divided by D. C and D will need to be defined somewhere before this line in the GFU.

Important!: Once a variable is defined, its type (like grid location and resolution) does not change. Therefore, for the existing variable A, the following expression:

A=Min(B,C)

is interpreted in the following way: for every point of the existing surface A calculate values of surface B and C and use the smaller of the two values to set new value of point elevation on surface A.

The following operators may be used in the expressions:

+, -, *, / - regular arithmetic operators

<, >, =, ! (not) - logic operators

— (or), & (and) - binary operators

Changing the scope of the equation

The scope of any line of the script can be modified by adding one of the following inclusion/exclusion operators:

A=Min(B,C);INCLU(handle_here) - will use AutoCAD entity with specified handle for the inclusion

A=Min(B,C);EXCLU(handle_here) - will use AutoCAD entity with specified handle for the exclusion

A=Min(B,C);PERIM() - will prompt for inclusion/exclusion at run-time

A=Min(B,C);PERIM(file_name) - will use specified file with inclusions/exclusions

Multiple inclusions or exclusions can be appended in this manner. Only points of target surface (A) passing inclusion/exclusion filter will be evaluated.

For custom, user define prompting, the following text should be used:

PERIM(,Prompt goes here) for user defined interactive inclusion and exclusion selection in CAD

The following script functions are currently defined:

Macro functions (performing operations on the entire surface at once)

- LOAD() Prompt user for the file to load. Returns a variable.
- LOAD(,Prompt goes here) for user defined prompting
- LOADFROM(string) Load surface from file. Grids (GRD) and TINs (FLT, TIN) are supported. Returns a variable.
- SAVE(Variable) Saves surface back to original file.
- SAVEAS(Variable,FileName) Saves surface into a file with given name.
- RELEASE(Variable) Releases memory used by a surface and undefines it for further use.
- EXTRAP(Variable[,Type])

Micro functions (taking effect on point by point basis as controlled by left side of the equation) Expressions can be complex ones with variables, value and functions

- MAX(Expression1,Expression2) Sets value to larger of two expressions evaluated.
- MIN(Expression1,Expression2) Sets value to smaller of two expressions evaluated.
- LESS(Expression1,Expression2,Expression3) If result of Expression1 is less than Expression2 then result is Expression3. Otherwise the source point is not changed. If Expression3 is not specified value is set to NULL.
- GREATER(Expression1,Expression2,Expression3) If result of Expression1 is greater than Expression2 then result is Expression3. Otherwise the source point is not changed. If Expression3 is not specified value is set to NULL.
- IF(Expression1,Expression2,Expression3) If Expression1 (can be logic expression like $(A+B)>C$ or $A=B$ or $A!B$ 'not equal') not 0 then result is Expression2, otherwise it is Expression3.
- POW(Expression1,Expression2) Result is value of Expression1 in power of Expression2
- MERGE(Expression1,Expression2) If Expression2 is valid at a point, then result is that value, otherwise it is value of Expression1
- SET_NULL(Expression1,Expression2) If Expression1 is valid at a point, then result is that value, otherwise it is value of Expression2
- CHANGE_RANGE_VALUE(Expression1, Range1, Range2, Expression2) If Expression1 is a valid point and its value is greater than equal to Range1 and less than equal to Range2, then result is Expression2. If Expression2 is not specified value is set to NULL.
- TIN_INTERSECT(Expression1,Expression2) Expects both Expression 1 and Expression 2 to be TIN's. This will add vertices to Expression 1 where Expression 2 has vertices. In this way, one TIN can be made to have vertices at the same XY locations as another TIN. Note that this function does not need to be preceded by "Variable=".

Here is an example of a complex IF statement used for coal recovery based on thickness of the seam:

```
COALTHK=LoadFrom(C:\Carlson Projects\Grids\C40_THK.GRD);
ROM_COAL=COALTHK
ROM_COAL=if((COALTHK<2)—(COALTHK=2),COALTHK - (COALTHK * 0.50),ROM_COAL)
ROM_COAL=if(((COALTHK<4)—(COALTHK=4))&(COALTHK>2),COALTHK      -      (COALTHK      *
0.10),ROM_COAL)
ROM_COAL=if((COALTHK<7)&(COALTHK>4),COALTHK - (COALTHK * 0.075),COALTHK - (COALTHK
* 0.05))
SaveAs(ROM_COAL, C:\Carlson Projects\Grids\C40_ROM_THK.GRD)
```

Pulldown Menu Location: Surface and Grids

Keyboard Command: GFU

Prerequisite: Make a grid (.GRD) file with the *Make 3D Grid File* command.

Edit 3D Grid

This command edits the elevation of a grid node by graphically picking the grid corner and entering a new elevation. The grid is a surface model that is represented by a rectangular mesh of grid cells. Each grid cell has four corners with elevation. This command modifies the elevation of one of these grid corners. After picking the grid node to edit, the program draws a temporary X marker on the selected point and shows the current elevation for the point. Before running this routine, a grid (.GRD) file must be created with the *Make 3D Grid File* command. Also the grid must be drawn on the screen using the *Draw Surface >> Draw 3D Grid File* command. Besides updating the elevation of the grid in the drawing, the grid file may also be updated. Whether to update the grid file is specified at the first prompt in the program.

Prompts

Update drawing only (Yes/<No>)? *press Enter* Choose between modifying the grid drawing or both the grid drawing and file.

Select Grid node to edit: *pick a grid cell corner*

Enter new Grid node elevation <305.519>: *press Enter*

Select Grid node to edit: *press Enter to end*

Pulldown Menu Location: Surface > Modify Grid File

Keyboard Command: editgrid

Prerequisite: A .GRD File and drawn grid 3D Faces

Merge Grid Files

This command creates a grid file by merging together two existing grid files, grid1 and grid2. The current grid is grid1 and the program will prompt for a second grid. These two grids must overlap with the same location and resolution. The inclusion and exclusion perimeters apply to grid2 such that the merged grid will consist of grid2 cells within the inclusion perimeters and outside the exclusion perimeters and grid1 cells everywhere else. The result is stored in the current grid.

No elevations are calculated on grid cells that extend beyond the extent of the data. Extrapolation can be used to calculate elevations for the grid cells that are beyond the data limits. The prompt *Extrapolate grid to full grid size?* shows when there are grid cells with no elevation in a grid (.GRD) file. Extrapolation fills in all the grid cells. The method to extrapolate uses a safe calculation that tends to average out or level the extrapolated values. So

extrapolated grid areas are not as accurate as grid areas within the limits of the data.

Prompts

Select Source Grid 1 File Dialog (*file select dialog*)

Reading cell> 93058

Extrapolate grid to full grid size [Yes/<No>]? *press Enter*

Select Source Grid 2 File Dialog (*file select dialog*)

Reading cell> 62137

Extrapolate grid to full grid size [Yes/<No>]? *press Enter*

Overlap method: Hold grid 1, replace with grid 2 or average [Hold/Replace/<Average>]?*press Enter*

Specify inclusion and exclusion areas for grid 2. Grid 1 used everywhere else.

Select the Inclusion perimeter polylines or ENTER for none. *select perimeter(s) or press Enter*

Select objects: *press Enter* to conclude selection

Select the Exclusion perimeter polylines or ENTER for none. *select perimeter(s) or press Enter*

Select objects: *press Enter* to conclude selection

Merged Grid File to Write Dialog(*new file select dialog*)

Pulldown Menu Location: Surface >> Modify Grid File

Keyboard Command: mergegrd

Prerequisite: Two grid (.GRD) files

One Triangulation Surface Volumes

This command calculates and reports the volume between a triangulation surface (.tin, .flt) file and a reference. The reference can be either an elevation or the TIN perimeter. The elevation method applies to cases like having a triangulation surface for the bottom of a pond and calculating the water volume to a water elevation. The TIN perimeter method creates a second surface by triangulating the perimeter points of the TIN. This method applies to cases like stockpile volumes where the base of the stockpile is not level and is covered so that the perimeter is the only data available to model the base.

In the Volume Report Options, the Cut and Fill Swell Factors are multiplied by the Cut and Fill volumes for the report. The Report Tons option multiplies the volumes by the specified density to report tons. The Write TIN Difference option creates a new TIN surface of the elevation difference between the triangulation and reference surfaces.

Note: The volume comparison of this routine uses the triangulation surface file as the *base* surface, and the reference as the *fnal* surface. So be aware that if your reference is set primarily below the surface defined by the triangulation file, it will report as cut, when in reality you may be filling above the reference to reach the defined surface.

Prompts

Select Triangulation File dialog

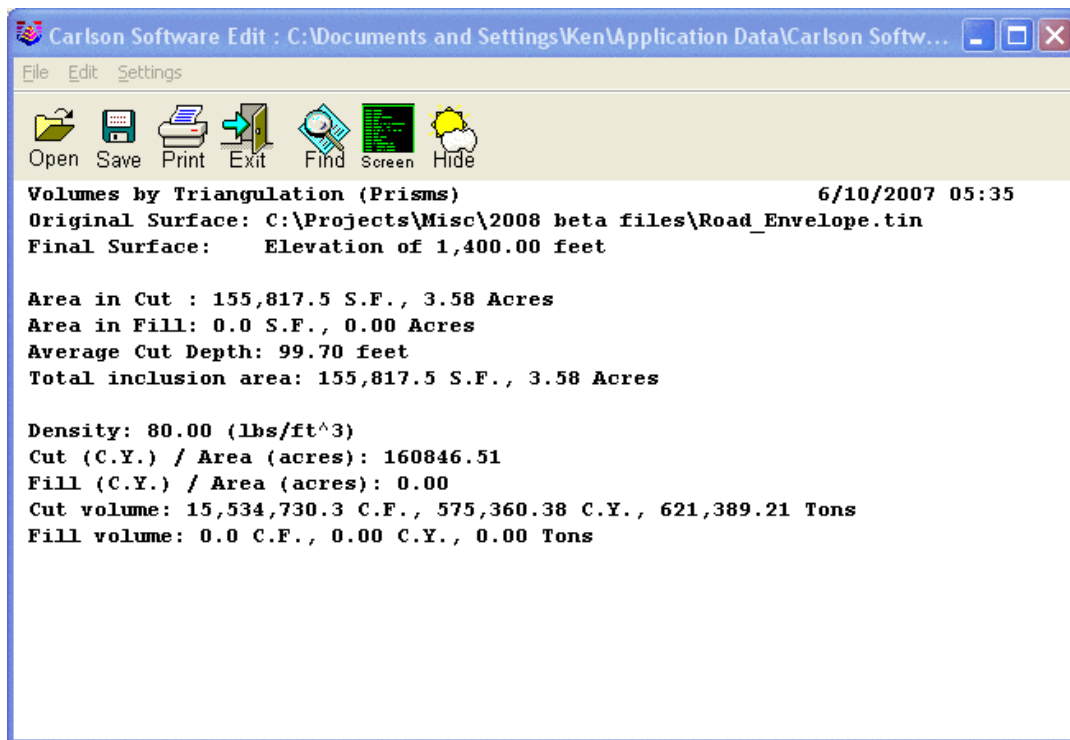
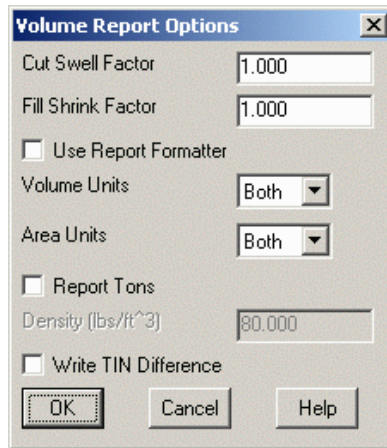
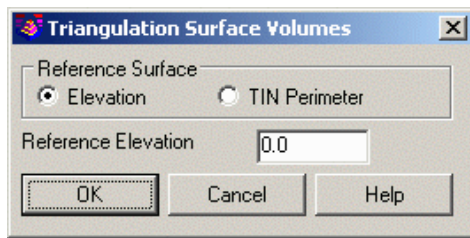
Triangulation Volumes options dialog

Select Inclusion polylines.

Select objects: *select inclusion boundary(ies) or Enter* for none.

Select Exclusion polylines.

Select objects: *select exclusion boundary(ies) or Enter* for none.



Pulldown Menu Location: Surface > Volumes By Triangulation

Keyboard Command: trivoll

Prerequisite: Triangulation (.flt or .tin) file

Two Triangulation Surface Volumes

Volumes By Triangulation is a volume method that compares two triangulation networks. This method is different from the grid based volume routines (*Volumes By Layer*, *One Surface Volumes*, *Two Surface Volumes*, *Stockpile Volumes*, etc.) and the cross section volume routine (*Calculate Section Volume*). Volumes by Triangulation calculates faster in most cases than the other methods, and it is the most accurate because it uses true TIN to TIN prismatic volumes. This added accuracy in general is very small. The grid resolution is usually sufficient to model the surface for the grid based volumes. The Volume By Triangulation accuracy applies well when there is a feature like a 5 foot wide ditch. Then the grid resolution would need to be less than 5 feet to model the ditch which might be difficult on a large site.

When multiple inclusion polylines are selected, there is an option to Report Multiple Perimeters Separately which reports the volumes for each inclusion separately. Otherwise the total volumes from all the inclusions is reported. For the separate inclusions report, the report will include the names assigned to the polylines by either the Tag Predefined Boundaries, Tag Area Descriptions, Tag Area Of Interest or Name Pit Polylines commands.

The disadvantage to this routine is that it lacks the output options that help the analysis of the volume such as Difference Contours. Also Volumes by Triangulation does no extrapolation and stops calculating volume at the perimeter of the smaller of the two triangulation networks. Volumes By Triangulation is better when used with point data instead of contour data because contour data requires triangulating all the contour polylines as breaklines which creates a large triangulation network and is slower.

The triangulation networks to compare are defined in .tin or .flt files that are created by *Triangulate & Contour* with the Write Triangulation File option. Note that while both file formats are supported, the newer binary triangulation file format (.tin) is twice as fast to load and save, and half the size, of the .flt triangulation file format. For this reason, the .tin file format is recommended. Before using this command, run *Triangulate & Contour* twice to create an triangulation (.TIN or .FLT) file for each surface. The volume calculation is limited by either the extent of the triangulation networks or by an inclusion/exclusion perimeter(s). These perimeters must be closed polylines.

Output data includes area, tons by density, average thickness, shrink and swell, ratio, and total volume.

Shrink/Swell Factors

An optional aspect of the Volumes by Triangulation routine is the ability to supply either a *Cut "Swell" Factor* and/or a *Fill "Shrink" Factor* to the results of the volume calculation. Having a solid understanding on the ramifications of each factor is important for determining how (and when) the values should be used for earthwork considerations.

The factors are commonly expressed as decimal differences from the "factor neutral" value of 1.00. In most cases, surface models are representations of what currently exists in the field or what is desired to exist after construction. Consider the following examples:

Excavating a Pit

Suppose you are given the task of designing a below ground storage pit. Based on your design surface model, the amount of Cut has been determined to be 1,000

C.Y.Cut Swell Factor > 1 (example 1.15)

Supplying a Cut Swell Factor greater than 1 would usually be taken to mean "How much volume will my 1,000 C.Y. of material occupy when it comes out of the ground?" With a 15% swell factor (1.15) applied, the 1000 C.Y. of excavated material would now occupy 1,150 C.Y. of space.

Cut Swell Factor < 1 (example 0.85)

Supplying a Cut Swell Factor less than 1 would usually be taken to mean "How much volume will 1,000 C.Y. of material occupy in this hole when it has been compacted?" With a 15% compaction factor (0.85) applied, the 1000 C.Y. of material getting compacted would now occupy 850 C.Y. of the hole space.

Working with a Stockpile

Suppose you have a stockpile of material that is suitable for building purposes. Based on your design surface model, the amount of material has been determined to be 1,000 C.Y.

Fill Shrink Factor > 1 (example 1.10)

Supplying a Fill Shrink Factor greater than 1 (see NOTE below) would usually be taken to mean "How much volume would this 1,000 C.Y. of material occupy if it were picked up and deposited elsewhere?" With a 10% swell factor (1.10) applied, the 1000 C.Y. of stockpile material would occupy 1100 C.Y. of space.

Fill Shrink Factor < 1 (example 0.90)

Supplying a Fill Shrink Factor less than 1 would usually be taken to mean "How much volume will 1,000 C.Y. of stockpile material occupy when it has been compacted?" With a 10% compaction factor (0.90) applied, the 1000 C.Y. of material getting compacted would now occupy 900 C.Y. of the hole space.

Note:

- In a design Fill scenario (such as a berm), often it is desired to know how much material would need to be brought in at a given compaction factor to occupy the design fill. To determine this value, use the following equation:

$$\text{Fill Factor} = 100.0 / (100.0 - \text{shrink_percentage}), \text{ using } 15\% \text{ shrink as an example,}$$
$$\text{Fill Factor} = 100.0 / (100.0 - 15.0) = 1.17647$$

Prompts

Select EXISTING Surface Triangulation File *Choose an .flt or .tin file*

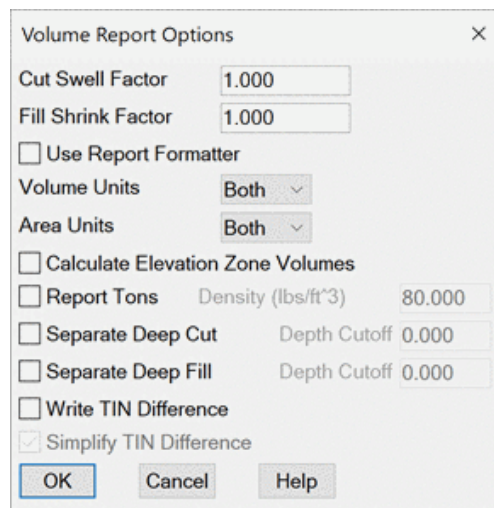
Select FINAL Surface Triangulation File *Choose an .flt or .tin file*

Select Inclusion polylines.

Select objects: *select objects* that form a perimeter around the area of study

Select Exclusion polylines.

Select objects: *select objects* that form an exclusion area within the area of study



Cut Swell Factor: Supply an appropriate factor by which the calculated Cut volume should be multiplied.

Fill Shrink Factor: Supply an appropriate factor by which the calculated Fill volume should be multiplied.

Use Report Formatter: Choose between customizing the report and using the standard report.

Volume Units and Area Units: Choose the units to include in the report.

Calculate Elevation Zone Volumes: This option calculates cut/fill volumes within elevation ranges. The ranges use a specified elevation interval and can start from the top or bottom.

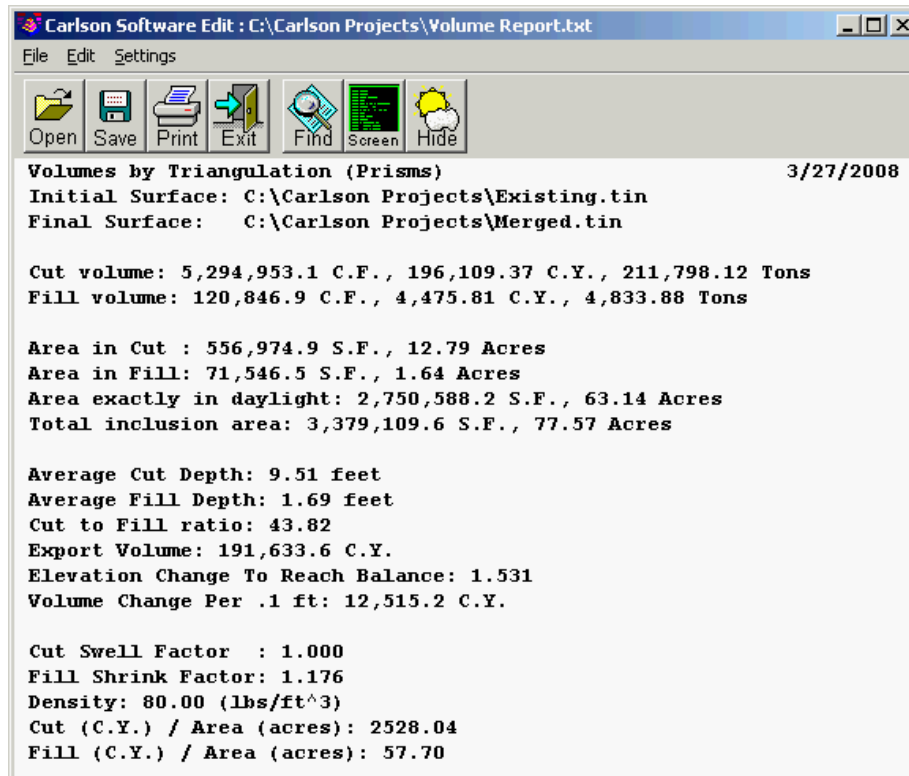
Report Tons: Enable this option to report the tonnage of Cut material and Fill material based on the material density.

Density: Specify the average material density for calculating the tons.

Separate Deep Cut/Fill: These options report the amount of cut/fill that is deeper than the specified Depth Cutoff.

Write TIN Difference: Enable this option to create a TIN based on the elevation difference between the EXISTING surface and the FINAL surface.

Simplify TIN Difference: Combining TINs to make the difference TIN can result in many small triangles. This option reduces the small triangles in this difference TIN.



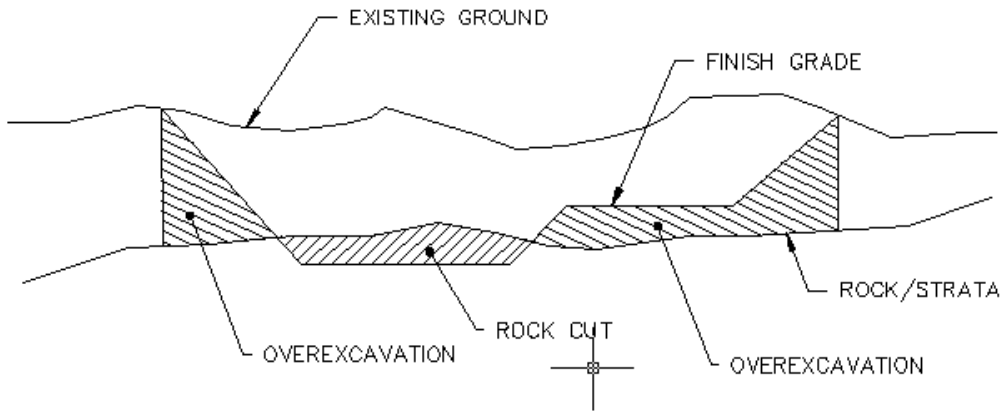
Pulldown Menu Location(s): Civil > Surface > Volumes by Triangulation, Survey > Surface > Volumes by Triangulation

Keyboard Command: trivol

Prerequisite: Two .flt or .tin files

Three Triangulation Surface Volumes

This command uses a set of three surface files, (.TIN, .FLT) to generate volumes between them. The user is prompted for the names of the three files as: Existing, for existing conditions, Final, for final design, and Strata, for a subsurface, such as rock. An Inclusion or Exclusion Perimeter can be selected, and then the Volume Report Options are presented. Picking OK generates the Volume Report.



Volume Report Options

Cut Swell Factor: 1.000

Fill Shrink Factor: 1.000

Use Report Formatter

Volume Units: Both

Area Units: Both

Report Tons

Density (lbs/ft³): 80.000

Write TIN Difference

OK Cancel Help

Carlson Software Edit : C:\Documents and Settings\grosen\Application Data\Carlson Software\Carl...

File Edit Settings

Open Save Print Exit Find Screen Hide

Volumes by Triangulation (Prisms) Wed Sep 02 09:36:22 2009

Initial Surface: C:\Carlson Projects\2010_1\DATA\SURFACES\EG1.tin

Final Surface: C:\Carlson Projects\2010_1\DATA\SURFACES\design.tin

Strata Surface: C:\Carlson Projects\2010_1\DATA\SURFACES\EG1-9.tin

Strata Cut volume: 10,319,708.2 C.F., 382,211.41 C.Y.

Other Cut volume: 17,847,651.2 C.F., 661,024.12 C.Y.

Total Cut volume: 28,167,359.4 C.F., 1,043,235.54 C.Y.

Fill volume: 9,716,336.4 C.F., 359,864.31 C.Y.

Strata cut area: 1,521,109.5 S.F., 34.92 Acres

Area in Cut : 2,868,880.7 S.F., 65.86 Acres

Area in Fill: 2,029,881.3 S.F., 46.60 Acres

Total inclusion area: 4,898,881.5 S.F., 112.46 Acres

Average Cut Depth: 9.82 feet

Average Fill Depth: 4.79 feet

Cut to Fill ratio: 2.90

Export Volume: 683,371.2 C.Y.

Elevation Change To Reach Balance: 3.766

Volume Change Per .1 ft: 18,144.0 C.Y.

Cut (C.Y.) / Area (acres): 9276.27

Fill (C.Y.) / Area (acres): 3199.85

Overexcavate volume: 25,820,433.4 C.F., 956,312.35 C.Y.

Overexcavate area: 3,363,077.2 S.F., 77.21 Acres

The amount of Cut into the Strata surface is reported, as well as Other Cut, Total Cut and Total Fill. The Areas for the same are also reported. The volume and area of Overexcavation is also reported.

Pulldown Menu Location: Surface >> Volumes By Triangulation

Keyboard Command: trivol3

Prerequisite: 3 Surface Files (.FLT or .TIN)

One Grid Surface Volumes

This command calculates the cut and fill volumes between the surface modeled by one grid (.GRD) file and a constant elevation or value. This is the same as *Two Grid Surface Volumes* except that the second surface is a flat plane at a constant elevation instead of a 3D grid surface. Please refer to that section for additional details. If the grid contains grid cells that have no elevations, you have the option to extrapolate elevations from the grid cells with elevations. When you choose not to extrapolate, no volume is calculated for these grid cells. There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the volume within this inclusion area is calculated. Volumes within an exclusion area are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to using this command.

Note: the volume comparison of this routine uses the selected grid file as the *base* surface, and the target elevation plane as the *fnal* surface, so be aware that if your target elevation is set primarily below the surface defined by the grid file, it will report as cut, when in reality you may be filling above the target elevation plane to reach the defined surface.

Prompts

Select the Inclusion perimeter polylines or ENTER for none:

Select objects: *press Enter*

Select the Exclusion perimeter polylines or ENTER for none:

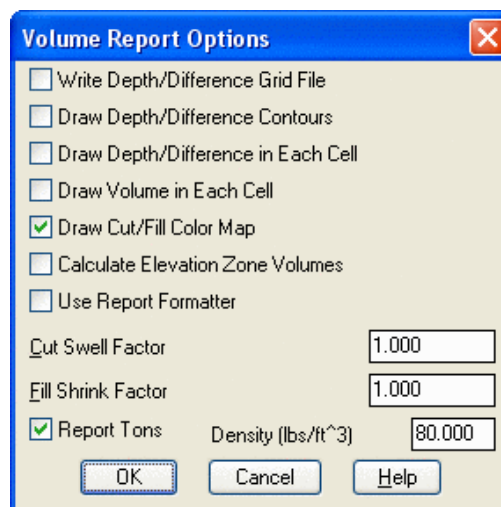
Select objects: *press Enter*

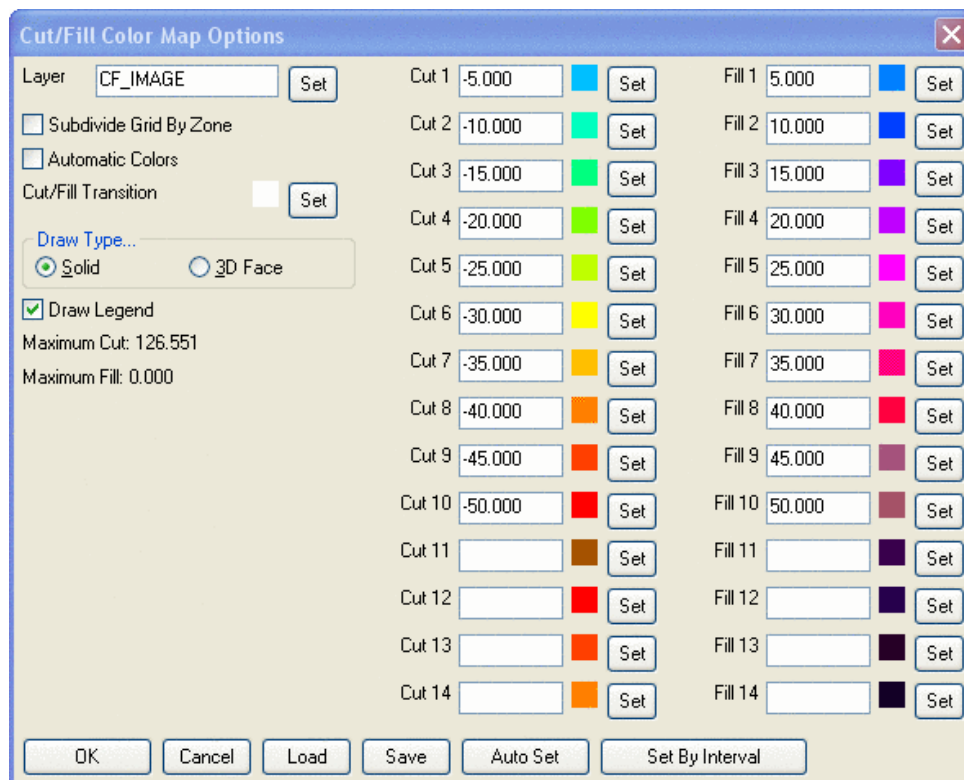
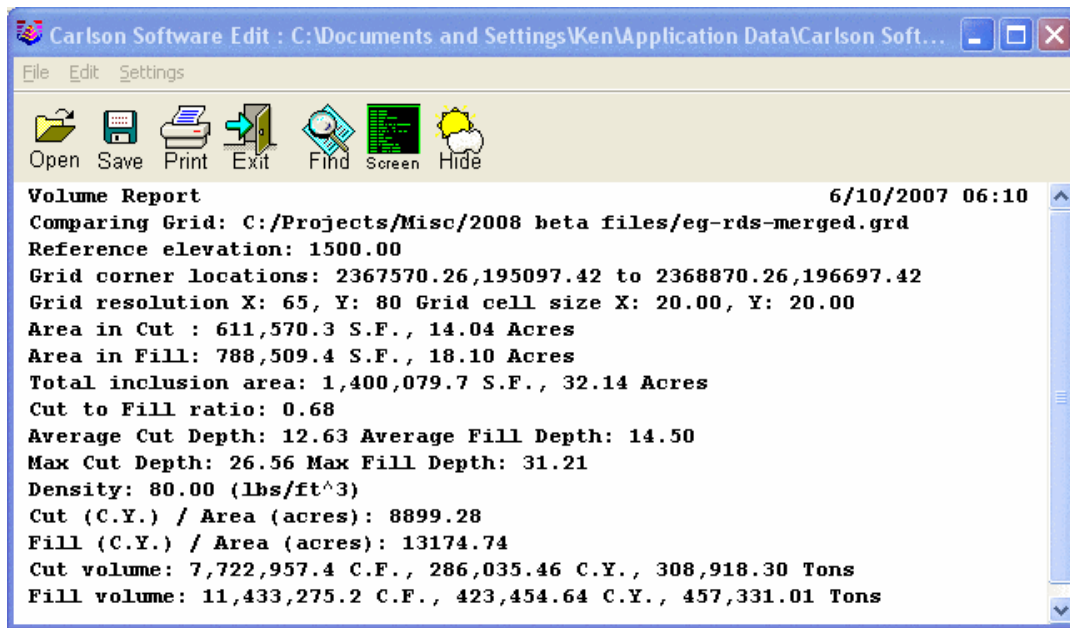
Specify Grid File Selection Dialog Choose a grid (.GRD) file to process.

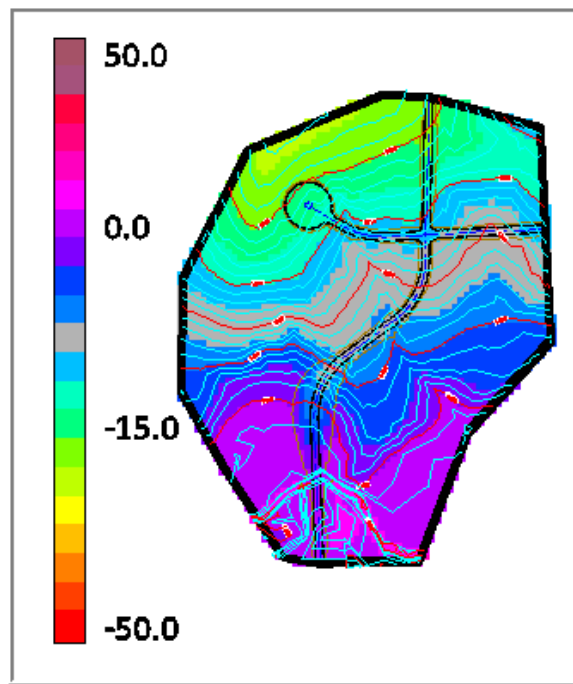
Extrapolate grid to full grid size (Yes/<No>)? *press Enter* If you enter Yes to this prompt, surface elevations will be computed for any grid cells that have null elevations.

Enter the base elevation: *1500* This defines the second surface.

Volume Report Options Dialog







Pulldown Menu Location: Surface >> Volumes By Grid Surfaces

Keyboard Command: volcalc1

Prerequisite: A grid (.GRD) file

Two Grid Surface Volumes

Two Grid Surface Volumes calculates the cut and fill volumes between two surfaces modeled by grid (.GRD) files. These two grid files must have the same location and resolution. To create the grid files, use the *Make 3D Grid File* routine. When creating the second grid file, choose *Use position of another file* and select the first grid file. Using the position of the first grid file sets the location and resolution of second grid to match the first.

There are several other routines that calculate volumes based on grid files. Grid based volumes can be calculated by *One Grid Surface Volumes*, *Volumes by Layer*, *Stockpile Volumes*, and *Pond/Pit Volumes*. These routines have special prompting and calculate the grid surfaces and volume in one step.

Volumes by Two Surface Volumes has three steps:

1. Creating the first grid file with *Make 3D Grid File*
2. Creating the second grid file with *Make 3D Grid File*
3. Running *Two Grid Surface Volumes*

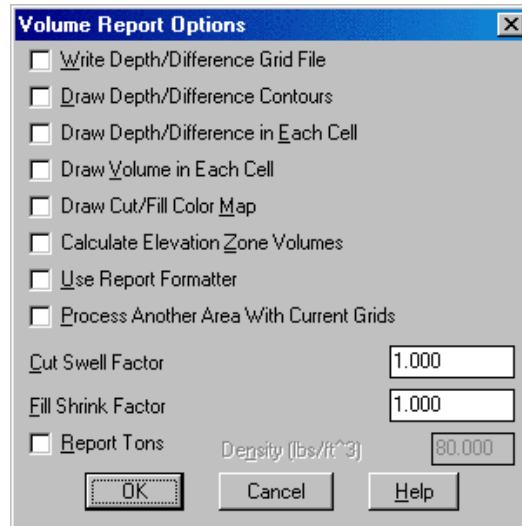
One advantage to this command is that you have more output options to help analyze volumes.

Besides grid based volumes, volumes can also be calculated between triangulation surfaces using the *Volumes by Triangulation* commands. Cross section end area is another volume method that is used by the *Calculate Sections Volume* command in the Civil Design module.

There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the volume within this inclusion area is calculated. **Important:** Whenever possible you should use a polyline that represents the limits of disturbed area as the inclusion perimeter. Volumes within an exclusion area are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine.

If the grid contains grid cells that have no elevations, you have the option to extrapolate elevations from the grid cells with elevations. When you choose not to extrapolate, no volume is calculated for the grid cells left without elevations. In general, extrapolation is not very accurate and should be avoided whenever possible. Sometimes you may get small amounts of cut in stockpiles that should only be fill, or small amounts of fill in pits that should only

be cut. These extraneous quantities are due to extrapolation at the border and should be small enough to be ignored. When inclusion or exclusion polylines are used, the program will automatically extrapolate the grids. In addition to writing a volume report to the file, printer or screen, there are several volume report options.



Write Difference Grid File creates a grid (.GRD) file of the elevation difference of the two grid files.

Draw Difference Contours creates a contour map of the difference or depth between the two grid files.

Draw Elevation Difference in Each Cell plots the elevation difference at the grid corners which is the same as the Elevation Difference routine.

Draw Volume in Each Cell plots the calculated volume for each grid cell and is an excellent way to verify the volume calculation. If a cell contains both cut and fill, both values will be plotted.

Calculate Elevation Zone Volumes calculates the cut and fill between different elevation ranges.

Draw Cut/Fill Color Map fills each grid cell with different shades based on the average cut or fill in the cell. Red shades are used for cut and blue for fill. There is an option to draw a color legend. You can subdivide the grid cells at zone transitions. Also, there is an option to control the zone intervals and range.

Use Report Formatter allows you to customize the report by choosing the fields to report and their order. Also the report formatter can be used to output the report data to Microsoft® Excel or Microsoft® Access.

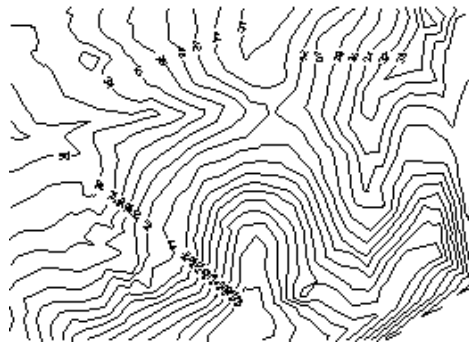
Process Another Area with Current Grids runs Two Surface Volumes again using the same grid files but different inclusion/exclusion polylines. This option saves the step of reloading the grid files to calculate volumes from the same grids for multiple areas.

The **Cut Swell Factor** value is multiplied by the cut volume in the report.

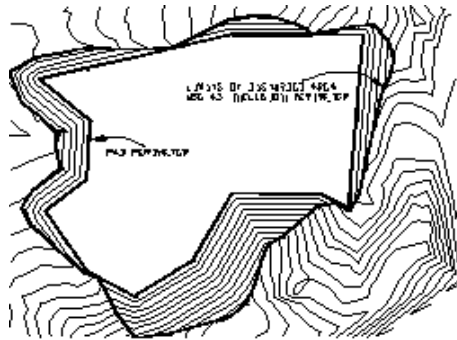
The **Fill Swell Factor** value is multiplied by the fill volume in the report.

Report Tons allows you to enter the material density and the program will report the cut and fill tons in addition to volume.

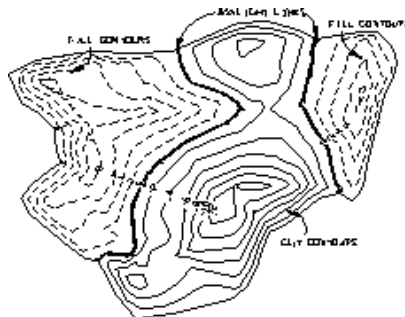
Given two accurate grid (.GRD) files, this routine will calculate accurate volumes. To verify the volume calculation, it is a good idea to check the grid (.GRD) files either by drawing them with *Draw Surface >> Draw 3D Grid File* and viewing them with *the 3D Viewer* or by contouring the grids with the *Contour Grid File* command.



Existing surface



Final surface contours with a closed polyline



Contours from the Draw Depth/Difference Contours option. Cut contours are red, fill contours are blue, daylight contours are green. This is a good way to check that both surfaces are modeled correctly and to verify the volumes.

Sample Two Surface Volumes report:

Volume Report

```

Comparing Grid: C:\scad2006\data\simo.grd
and Grid: C:\scad2006\data\final.grd
Lower left grid corner : 186551.67,57624.98
Upper right grid corner: 186828.81,57897.09
X grid resolution: 75, Y grid resolution: 75
X grid cell size: 3.70, Y grid cell size: 3.63
Total inclusion area: 37016.71 sq ft, 0.850 acres
Cut to Fill ratio: 1.14
Cut (C.Y) / Area (acres): 3642.35
Fill (C.Y) / Area (acres): 3182.70
Cut vol: 83570.89 cubic ft, 3095.22 cubic yards
Fill vol: 73024.56 cubic ft, 2704.61 cubic yards

```

Prompts

- Select the Inclusion perimeter polylines or ENTER for none:**
- Select objects:** *pick a closed polyline for the limits of disturbed area*
- Select objects:** *press Enter*

Select the Exclusion perimeter polylines or ENTER for none:

Select objects: *press Enter*

Specify Base Grid File Selection Dialog

Choose a grid (.GRD) file to process.

Extrapolate grid to full grid size (Yes/<No>)? *press Enter* If you enter *Yes* to this prompt, surface elevations will be computed for any grid cells that have null elevations.

Sample report from the Calculate Elevation Zone Volumes option:

(Calculates the cut and fill in different elevation ranges at a user-specified interval and beginning at a user-specified starting elevation.)

Volumes by elevation zone

Zone 20.00 to 30.00

Cut volume : 0.30 cubic ft, 0.01 cubic yards

Fill volume: 107.90 cubic ft, 4.00 cubic yards

Zone 30.00 to 40.00

Cut volume : 4.88 cubic ft, 0.18 cubic yards

Fill volume: 73021.14 cubic ft, 2704.49 cubic yards

Running total:

Cut volume : 5.18 cubic ft, 0.19 cubic yards

Fill volume: 73129.05 cubic ft, 2708.48 cubic yards

Zone 40.00 to 50.00

Cut volume : 65044.26 cubic ft, 2409.05 cubic yards

Fill volume: 0.25 cubic ft, 0.01 cubic yards

Running total:

Cut volume : 65049.44 cubic ft, 2409.24 cubic yards

Fill volume: 73129.29 cubic ft, 2708.49 cubic yards

Zone 50.00 to 60.00

Cut volume : 17786.85 cubic ft, 658.77 cubic yards

Fill volume: 0.00 cubic ft, 0.00 cubic yards

Running total:

Cut volume : 82836.29 cubic ft, 3068.01 cubic yards

Specify Final Grid File Selection Dialog

Choose a grid (.GRD) file to process.

Extrapolate grid to full grid size (Yes/<No>)? *press Enter*

Volume Report Options dialog

-83.8	-82.1	-63.2	-31.8	-7.5 +0.4	+15.1	+37.2	+59.4
-81.2	-78.8	-61.2	-33.8	-11.6	-0.1 +10.6	+32.7	+54.9
-77.2	-73.9	-60.4	-38.3	-16.1	-0.6 +6.7	+28.2	+50.4
-71.7	-65.6	-53.3	-38.9	-18.7	-2.1 +3.6	+23.7	+45.8
-64.1	-53.8	-40.3	-28.7	-12.7	-1.1 +4.7	+21.8	+41.6
-53.2	-40.3	-26.8	-13.3	-2.1 +2.5	+14.3	+28.5	+44.4

This shows a grid drawn by *Plot 3D Grid File* and volume values drawn by the Draw Volume in Each Cell option of the Two Surface Volumes routine. Cut appears as negative and fill as positive. Notice that cells bordering cut and fill regions contain a little of both.

Pulldown Menu Location: Surface >> Volumes By Grid Surfaces

Keyboard Command: volcalc2

Prerequisite: Two grid files

Volumes By Layer

This is the easiest yet still equally accurate method for calculating volumes. For this command, volumes are calculated in one step by a simple window of the area, selecting the items, and *calculate*.

First, you must specify the grid location and resolution. The grid location should enclose the area for volume calculations. Next the program asks for the layer names of the entities for the base and final surfaces. You designate the layers to use for each surface either by typing the layer names or by picking from the screen, then during the routine you select the entities to use. You may safely use the keyword *ALL* to select the entities, since you have pre-defined the layers to use, and all those entities not on the specified layers will be filtered out. These entities, for use in modeling the surfaces, can be points, lines (such as triangulation lines), 2D polylines (such as contours), and 3D polylines (such as breaklines).

Inclusion and exclusion perimeters may optionally be specified to limit the volume calculation area on the grid. An inclusion perimeter should be used if there is a closed polyline for the limit of the disturbed area. Then the program internally generates grids of the surfaces from the entities on the corresponding layers and then calculates and reports the volume. The main disadvantage to this routine is that it doesn't have the special output options of *Two Grid Surface Volumes* such as Depth Contours.

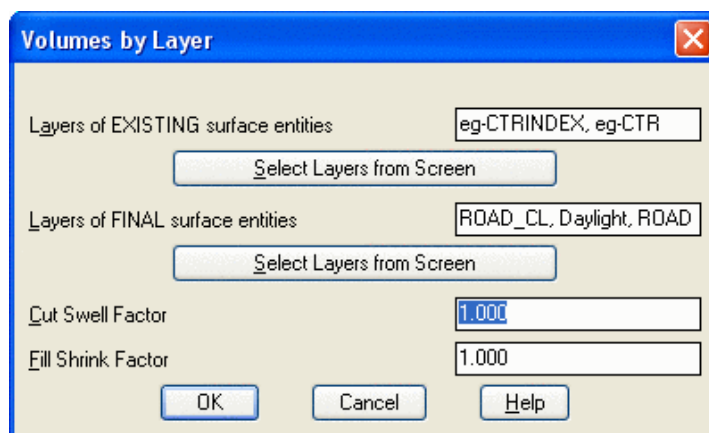
Prompts

Command: *layervol*

Pick Lower Left limit of surface area: *pick lower left corner of grid*

Pick Upper Right limit of surface area: *pick upper right corner of grid*

You are then prompted to designate layers:



Press *Select Layers from Screen* to show the routine which layers to use by selecting sample objects from those layers.

Select entities on layers of Existing surface. *select sample object(s)*

Select objects: Specify opposite corner: 3 found

Select objects: *press Enter* to conclude selection.

Select entities on layers of Final surface. *select sample object(s)*

Select objects: Specify opposite corner: 10 found

Select objects: *press Enter* to conclude selection.

Reading points ...

Select surface entities on corresponding layers.

Select objects: *all filters out those objects not on designated layers*

85 found

Select objects: *press Enter* to conclude selection.

Reading points ... 9396

Assigning grid values> 5300

Pass> 28 Null Z values left> 0

Writing grid file: C:\Documents and Settings\... \USER\grid1.grd

Assigning grid values> 5300

Pass> 43 Null Z values left> 0

Writing grid file: C:\Documents and Settings\... \USER\grid2.grd

Select the Inclusion perimeter polylines or ENTER for none: *select inclusion perimeter*

Select objects: 1 found

Select objects: *press Enter* to conclude selection.

Select the Exclusion perimeter polylines or ENTER for none.

Select objects: *press Enter* for none.

Reading cell> 5346

Pass> 28 Null Z values left> 0

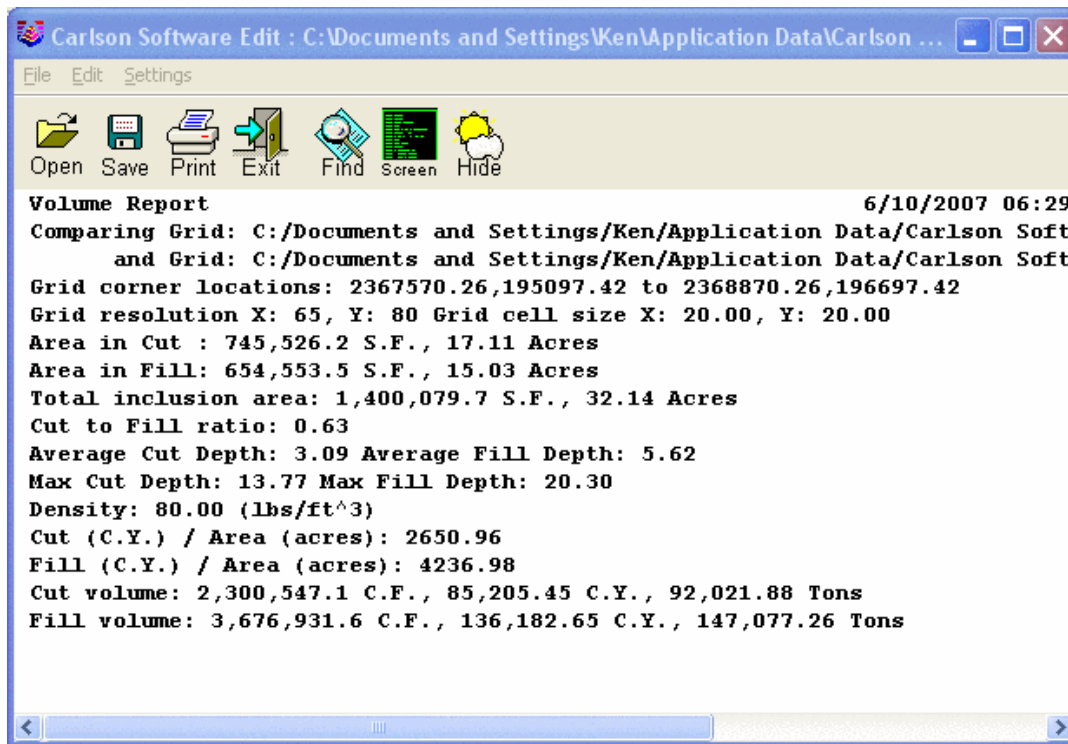
Reading cell> 5346

Pass> 43 Null Z values left> 0

Pre-processing grid cells

Processing cells ...

Select point for color legend (Enter for None): *press Enter*



Pulldown Menu Location: Surface >> Volumes By Grid Surfaces

Keyboard Command: layervol

Prerequisite: Entities that define both the base and final surfaces.

Cut/Fill Report

This command reports the cut/fill between two surfaces at an interval. The existing and design surfaces can be either triangulation (TIN) or grid (GRD). After selecting the surfaces, the program prompts for optional inclusion and exclusion polylines to control the report area. Then there is an options dialog. The report works with the Report Formatter for choosing which fields to show including cut/fill, existing elevation, design elevation, northing and easting.

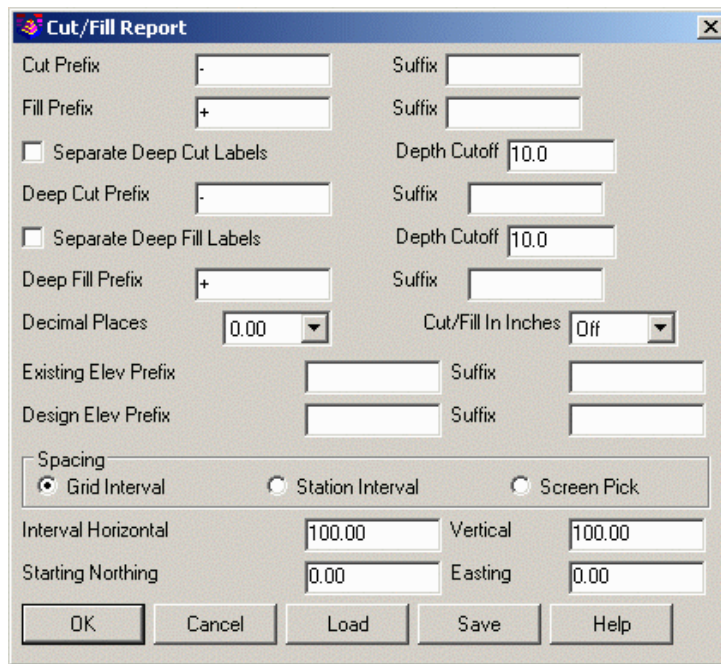
In the options dialog, you can customize the Cut/Fill labels. Text can be added either before or after the Cut/Fill amount, the Existing elevation, and the Design elevation with the Prefix and Suffix fields. Decimal Places sets to how many decimal places the labels will report. The Cut/Fill In Inches labels in feet and inches to the specified precision. The Separate Deep Cut/Fill Labels settings allow for highlighting deep areas by changing prefix/suffix labels.

The Spacing methods include:

Grid Interval: Places the labels at the specified Horizontal and Vertical Intervals starting with the specified Northing and Easting coordinate.

Station Interval: Uses a centerline polyline and places the labels at a station interval along this alignment.

Screen Pick: Prompts for each label position.



Pulldown Menu Location: Surface > Cut/Fill Utilities

Keyboard Command: cfreport

Prerequisite: Existing and design surfaces

Cut/Fill Labels

This command displays the design elevation, the existing elevation, and the amount to either cut or fill directly on the screen. The design and existing elevations can be defined by triangulation files, grid files or points.

In the Elevation Difference Label Options dialog, you can customize the Cut/Fill labels. Text can be added either before or after the Cut/Fill amount, the Existing elevation, and the Design elevation with the Prefix and Suffix fields. You can also choose whether or not to display the Existing Surface elevations and the Design Surface elevations. The Draw Marker Symbol option draws the specified symbol block at each label point. The Hide Drawing Under Labels option creates Wipeout entities around the labels so that you can read the labels clearly. Text Size chooses the text size for each line of the label. Text Style allows you change the Font Style displayed in the labels. Decimal Places sets to how many decimal places the labels will report. The Cut/Fill In Inches labels in feet and inches to the specified precision. The Separate Deep Cut/Fill Labels settings allow for highlighting deep areas by making them a darker color or adding prefix/suffix labels. The Label Cut and Label Fill toggles allow you to turn off cut or fill and label only the other.

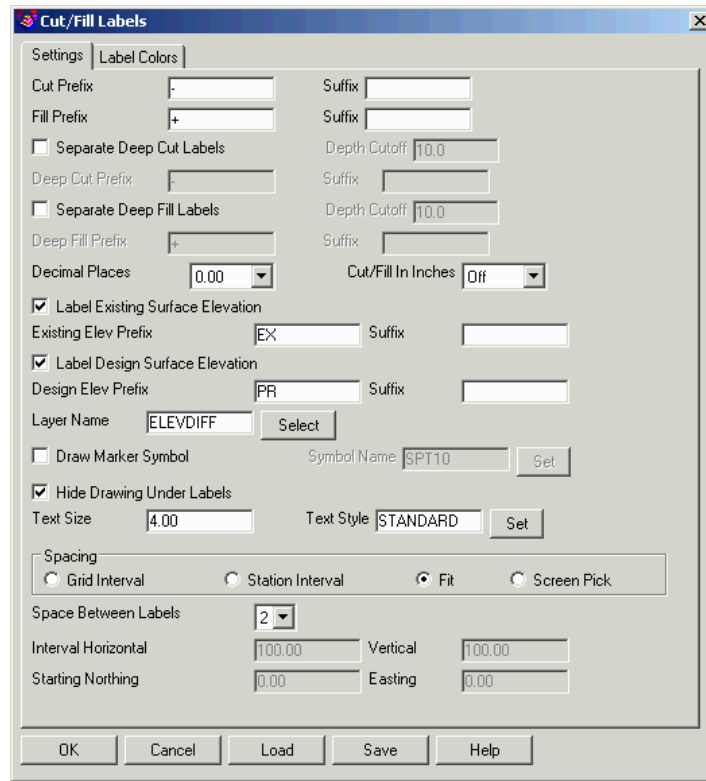
The Spacing Methods include:

Fit: Uses an inclusion perimeter and the size of the labels to make a series of rows and columns of labels that fit within the perimeter. The Space Between Labels sets the buffer around labels. The size of each space is determined by the Text Size.

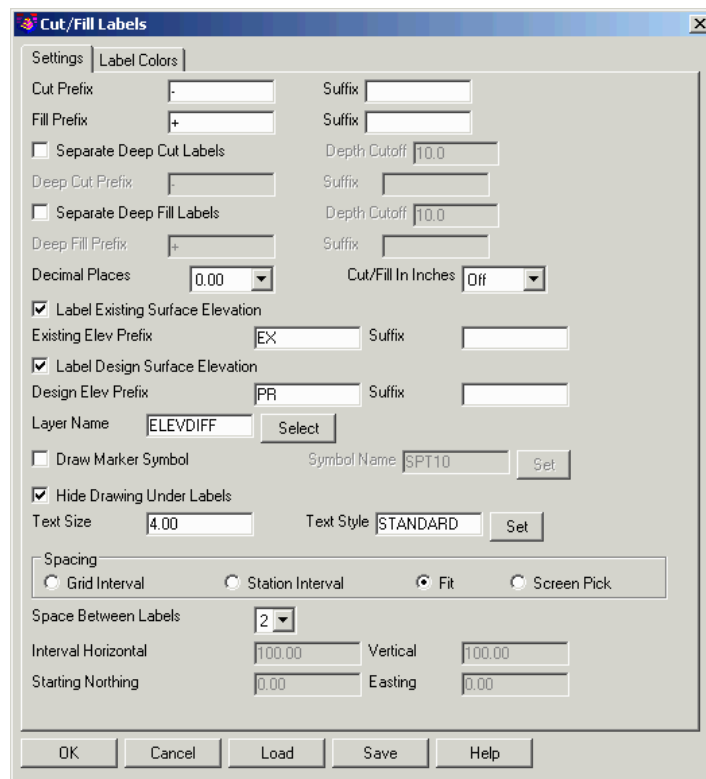
Grid Interval: Places the labels at the specified Horizontal and Vertical Intervals starting with the specified Northing and Easting coordinate.

Station Interval: Uses a centerline polyline and places the labels at a station interval along this alignment.

Screen Pick: Prompts for each label position.



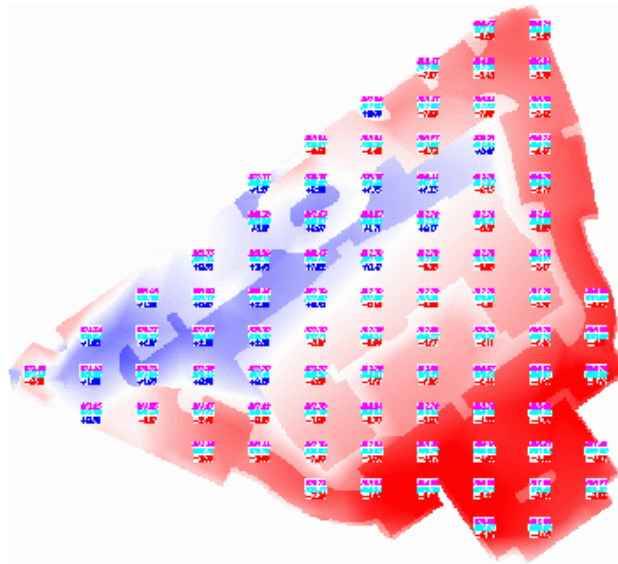
On the Label Colors tab, the Automatic Coloring options choose a color scheme for the Cut-Daylight-Fill colors. The program will use a gradient color for the labels depending on the cut/fill value for the label. The deeper cut/fill values will have a deeper color. The Max Depth sets the value for the deepest color. You can also choose None for the Automatic Coloring and then set fixed colors for the cut and fill. The Custom method lets you specify all the colors and depth ranges.



The following image shows the main dialog box for setting the labeling options. The labeling created with these options looks like this:

PR117.00 EX129.59 -12.59	PR117.00 EX129.00 -12.00
PR117.00 EX127.40 -10.40	PR117.00 EX127.31 -10.31

The distribution of the labels on the site looks like this:



Pulldown Menu Location: Surface > Cut/Fill Utilities

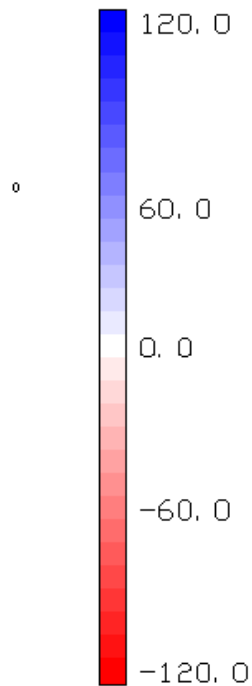
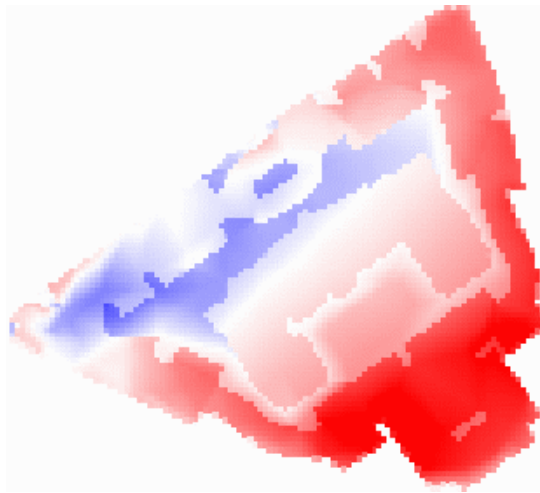
Keyboard Command: elevdiff

Prerequisite: Existing and design surfaces

Cut/Fill Color Map

This command creates a cut/fill color map typically in red and blue in order to show the difference between grid or triangulation surfaces. In the options dialog, the ranges of the cut and fill zones are set along with the colors. The Automatic Colors has several colors styles to choose from for the cut to daylight to fill colors. When not using the Automatic Colors, the Cut/Fill Transition color is used for the zone between the smallest cut and smallest fill. For triangulation surfaces, the Map Subdivisions is how many times the surfaces will be divided in the X and Y directions to make cells for coloring. For example, Map Subdivisions of 50 makes 2500 colors cells from 50x50. The Limit Cut/Fill Map Range option is a way to draw the color map only for a specified range such as only showing the deep cut. The Auto Set function fills out the cut/fill ranges based on the current two surfaces. The Set By Interval function fills out the cut/fill ranges by a user-specified depth interval.

For analyzing using the grid option, you need to already have two existing grid files. If the grids are not visible in plan view, you may want to have them display on-screen using the *Draw 3D Grid File* command. The grids should overlap with the same location and resolution. The resulting red/blue map with legend is shown below.



No coloring is done on tin or grid cells that extend beyond the extent of the data. Extrapolation can be used to calculate elevations for the grid cells that are beyond the data limits. The prompt *Extrapolate grid to full grid size?* shows when there are grid cells with no elevation in a grid (.GRD) file. Extrapolation fills in all the grid cells. The method to extrapolate uses a safe calculation that tends to average out or level the extrapolated values. So extrapolated grid areas are not as accurate as grid areas within the limits of the data.

Prompts

For a color map showing differences between two grids:

Type of surface model source [Tin/<Grid>]? *press T* for a Triangulation (.TIN) file, or *press Enter* to accept default choice in brackets.

Select Base Grid File Dialog Select an existing .grd file.

Select Final Grid File Dialog Select a second existing .grd file.

Select Inclusion polyline: *pick a closed inclusion perimeter*

Select Exclusion polylines (Enter for none).

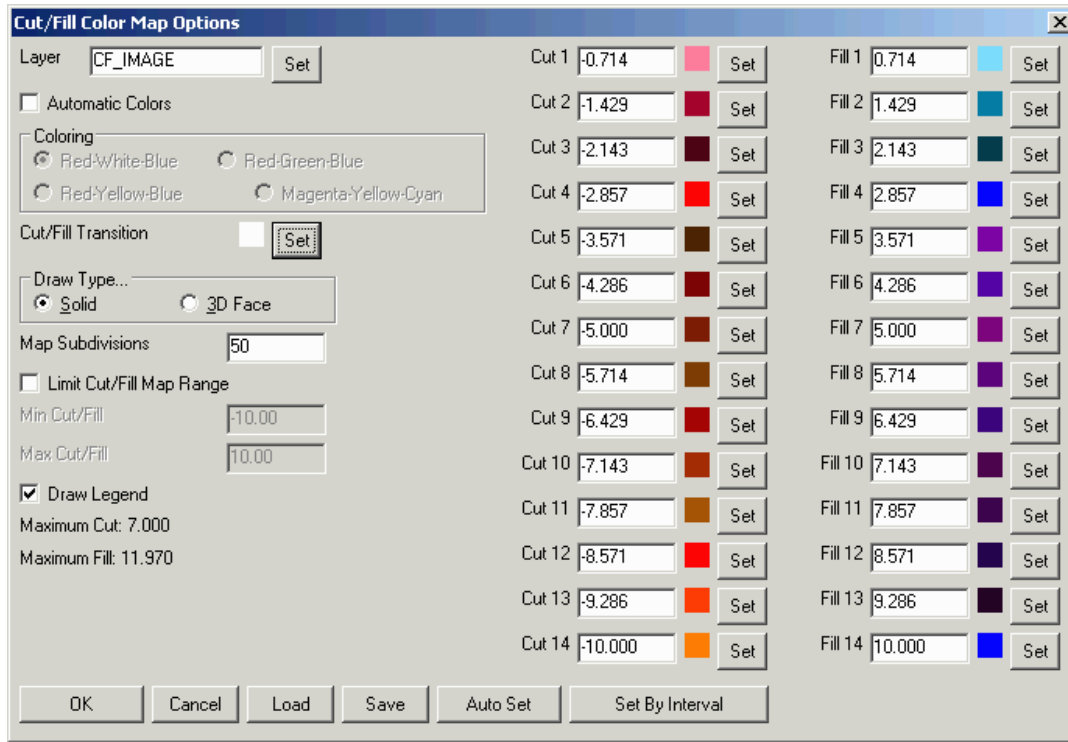
Select objects: *pick exclusion polylines or press Enter*

Cut/Fill Color Map Options Dialog

Select point for color legend: *pick a point*

Legend size <10.0>: *press Enter*

Label all zones or summary [All/<Summary>]?: *press Enter*



Pulldown Menu Location: Surface >> Cut/Fill Utilities

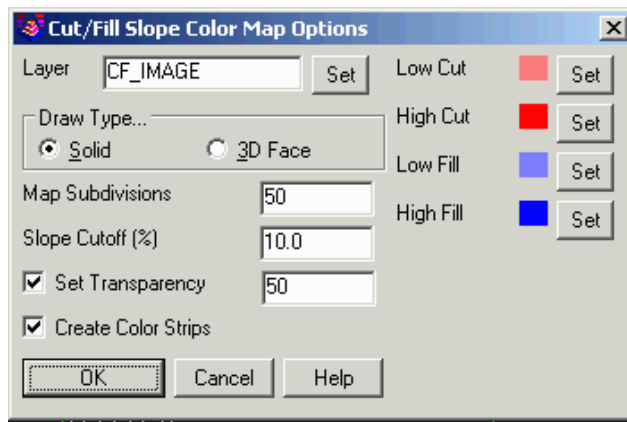
Keyboard Command: cf_map

Prerequisite: Two grid (.GRD), triangulation mesh (.FLT) or tin (.TIN) files

Cut/Fill Slope Color Map

This command creates color map like the Cut/Fill Color Map routine except instead of coloring by the amount of depth, this map uses different colors based on the amount of slope of the design surface. In the options, the Slope Cutoff sets the slope amount for using the specified low and high colors. Areas of cut that have a design surface slope below the cutoff are drawn with the Low Cut color and areas of cut over use the High Cut color. Likewise, areas of fill with a design surface slope below the cutoff use the Low Fill color and areas of fill over use the High Fill color.

The options dialog has settings for the color map properties including layer, entity type and transparency. The Map Subdivisions controls the resolution of the color map cells. A higher number makes more cells for higher resolution. The Create Color Strips option combines neighboring cells with the same color which speeds up the routine and makes for a smaller dwg size.



Prompts

For a color map showing differences between two grids:

Type of surface model source [Tin/<Grid>]? *press T* for a Triangulation (.TIN) file, or *press Enter* to accept default choice in brackets.

Select Base Triangulation File Dialog

Select Design Triangulation File Dialog

Select Inclusion polyline: *pick a closed inclusion perimeter*

Select Exclusion polylines (Enter for none).

Select objects: *pick exclusion polylines or press Enter*

Cut/Fill Color Slope Map Options Dialog

Pulldown Menu Location: Surface > Cut/Fill Utilities

Keyboard Command: cf_slope_map

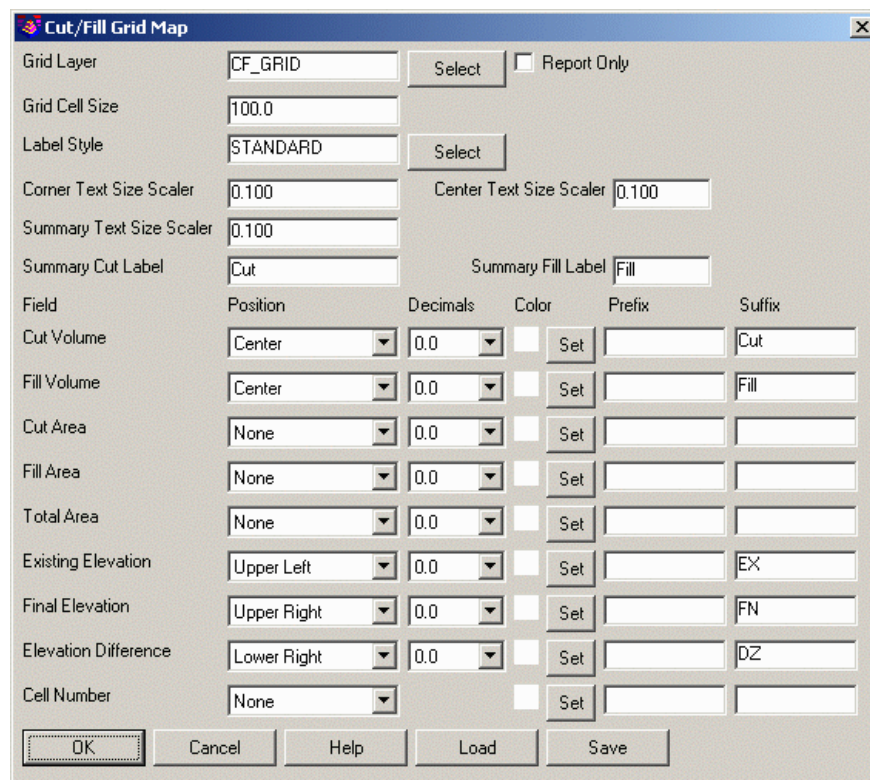
Prerequisite: Two triangulation surfaces

Cut/Fill Grid Map

This command labels cut/fill quantities and creates a report at a grid interval over the site. The grid cells are square at a specified size. The cut/fill quantities are calculated separately within each grid cell. The options dialog controls which cut/fill fields to label and the label position within the grid cell. There is a summary row at the bottom of the grid with the overall totals and sub-totals for each column.

After the options dialog, the program prompts for the corners for the area to grid. These corners should create a window around the site.

Then after drawing the grids and labels, the Report Formatter shows the cut/fill quantities for the grid cells. You can choose which fields to include in the report.



Prompts

Select Existing Triangulation File

Select Design Triangulation File

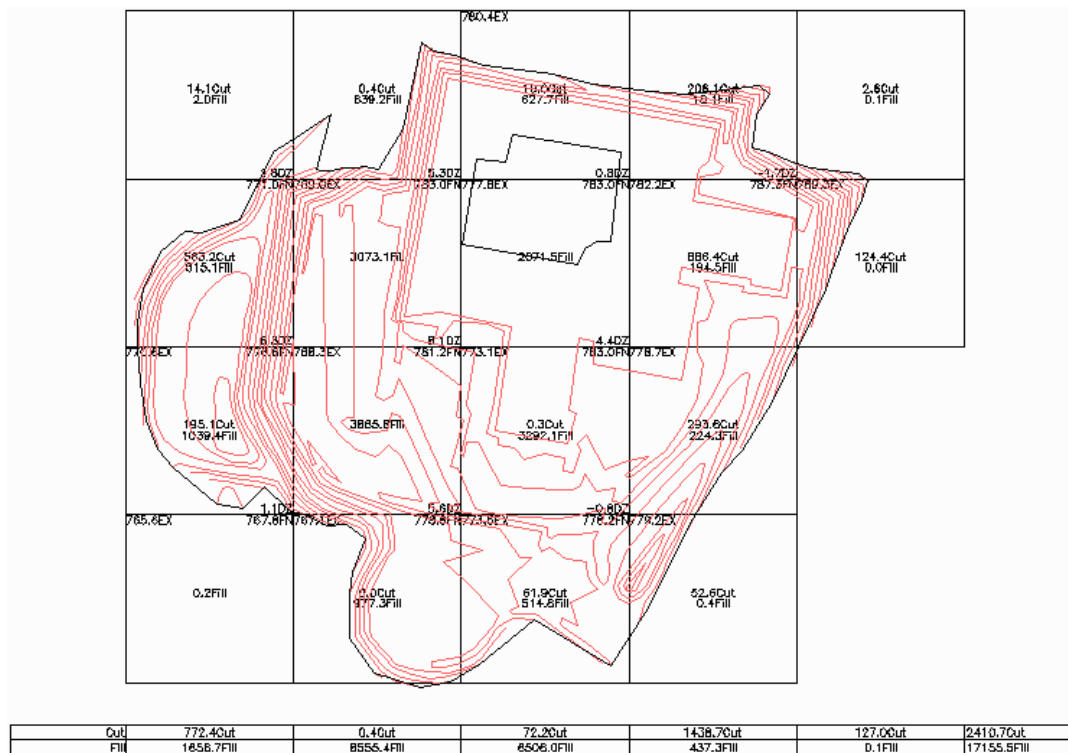
Select Inclusion polyline (Enter for none): *select polyline*

Select Exclusion polylines (Enter for none): *press Enter*

Cut/Fill Grid Map dialog

Pick first grid corner: *pick a point*

Pick second grid corner: *pick a point*



Sample Report:

Cell	Cut (C.Y.)	Fill (C.Y.)	Cut Area	Fill Area	Total Area
1	14.1	2.0	505.3	113.7	674.0
2	0.4	639.2	45.2	4693.4	4739.4
3	10.0	627.7	565.6	6691.1	7256.7
...					
----- Grand Total -----					
	2410.7	17155.5	30959.6	93877.1	125151.5

Pulldown Menu Location: Surface >> Cut/Fill Utilities

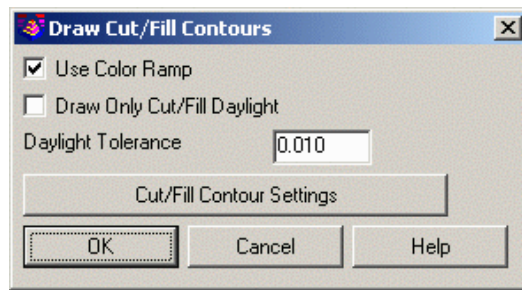
Keyboard Command: cf_grid

Prerequisite: Existing and design surfaces

Cut/Fill Contours

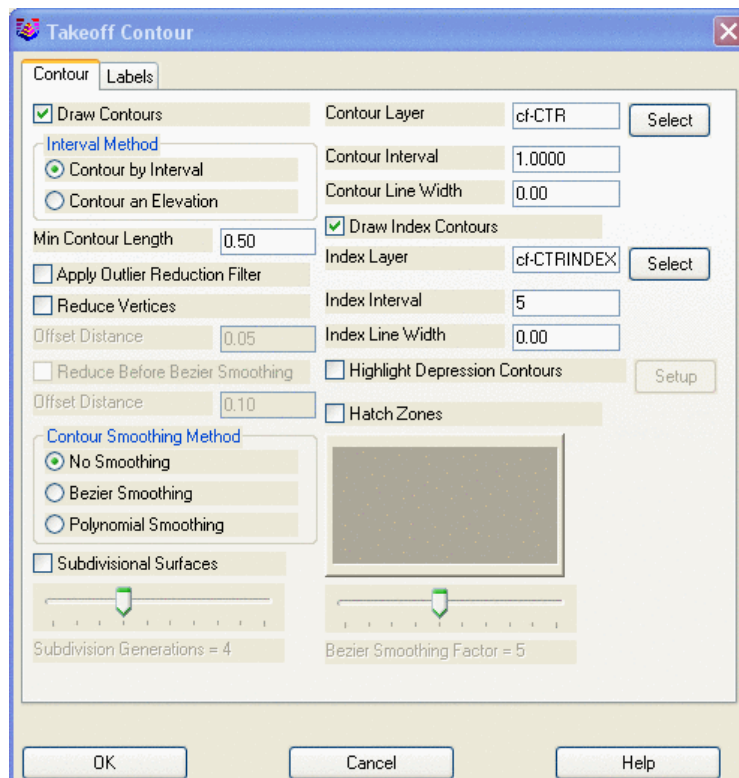
This command displays the amounts of cut and fill between two surfaces by computing and displaying cut/fill contour lines representing the amount of cut or fill along that line. Cut contours are displayed in red (with negative values), fill in blue (positive values), while the lines of zero cut (the "daylight" lines) are displayed in green ("0" labels).

The **Use Color Ramp** option changes the color of the contours to be darker as the cut and fill get deeper. Use the **Draw Only Cut/Fill Daylight** option to draw only the daylight lines, indicating the areas where the two surfaces intersect. The **Daylight Tolerance** setting controls which contours are drawn in green.



Cut/Fill Contour Settings:

You set the contour settings in the opening dialog by selecting the *Cut/Fill Contour Settings* button, which is comprised of the *Contour* and *Labels* tabs identical to those in *Triangulate and Contour*. Please refer to that command for details on the dialog options. You may wish to designate alternate layer names for these sets of contours to avoid overwriting previous contours on surface layers, and generally you will set the contouring interval to 1 foot.



Prompts

Draw Cut/Fill Contours dialog

Select Base Surface File

Select Design Surface File

Select Inclusion polyline: *select inclusion boundary(ies) or Enter for none.*

Select Exclusion polylines (Enter for None).

Select objects: *press Enter*

Loading edges...

Loaded 9507 points and 27345 edges

Created 17839 triangles

Loading edges...

Loaded 826 points and 2250 edges

Created 1425 triangles

Loading edges...

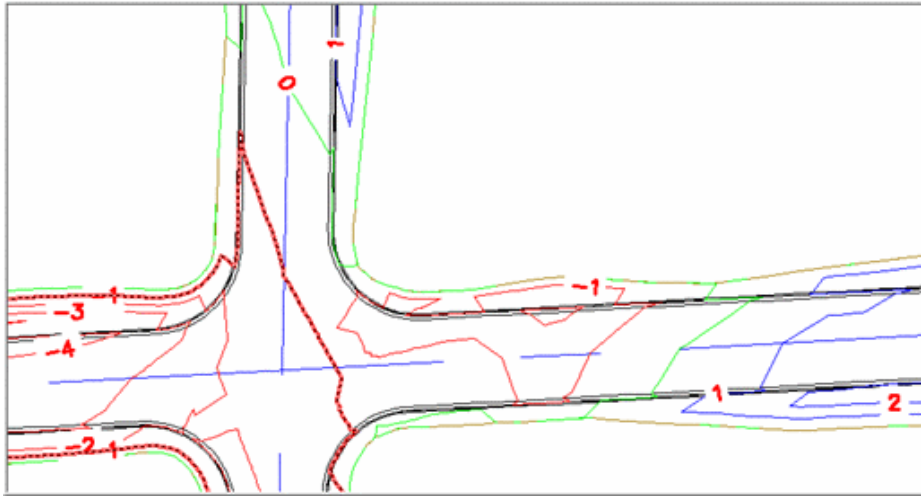
Loaded 18927 points and 54691 edges

Created 35765 triangles

Ignored 2942 points with zero elevation.

Contouring elevation 14 - Routine displays and updates the value in process

Inserted 10273 contour vertices.



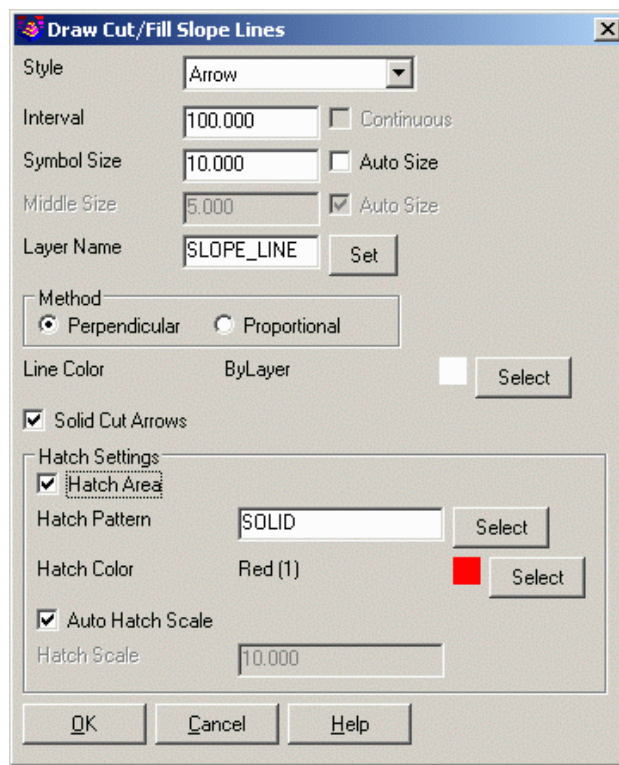
Pulldown Menu Location: Surface >> Cut/Fill Utilities

Keyboard Command: cf_ctr

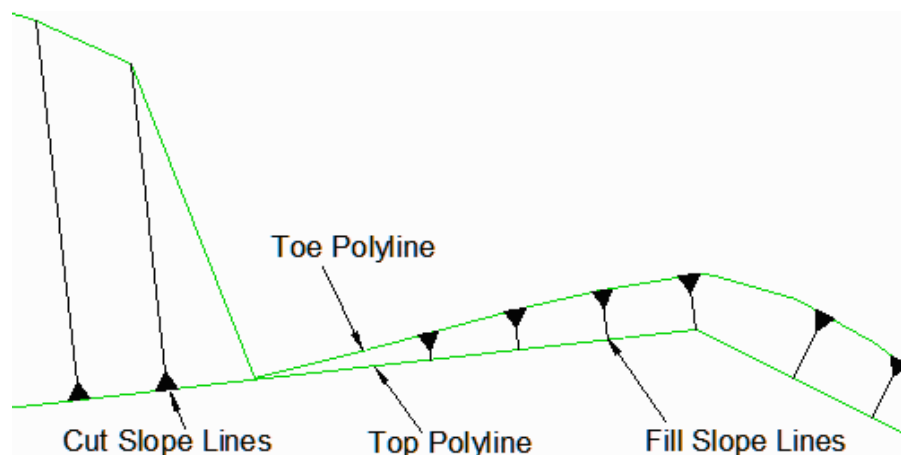
Prerequisite: Existing and design surfaces

Cut/Fill Slope Lines

This command draws cut/fill slope lines with slope direction arrows. The arrowhead points in the downward direction of the slope. The cut/fill slopes are defined by selecting a 3D polyline for the top of slope and another 3D polyline for the toe of slope.



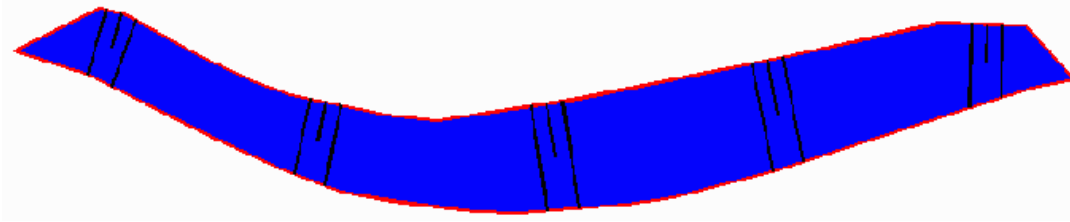
In the options dialog, the Style chooses the type of symbol to draw: arrow, Y, V, 3-line, Middle Dot, End Dot, 4-line or line. The Interval sets the spacing of the slope lines along the top of slope polyline. The Continuous method makes pattern repeat without interval spacing. There are settings for the arrowhead size and the color and layer for the slope lines. The Auto Size option adjusts the symbol size relative to the length of the cut/fill line. The Solid Cut Arrows option allows for different style arrowheads for cut and fill slopes. The Hatch Settings control whether to hatch the area between the top and toe 3D polylines and the hatch properties to use. The Perpendicular method draws the lines perpendicular from the top polyline to intersect with the toe polyline. The Proportional method draws the lines from the top polyline to a relative proportion position on the toe polyline. For example, when the top polyline position is 25% along the top polyline, then the toe position will be 25% along the toe polyline.



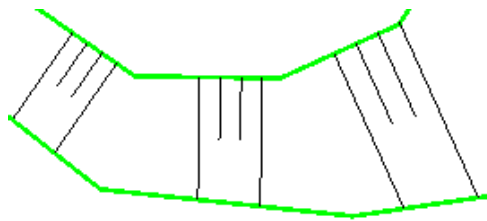
Arrow style slope lines



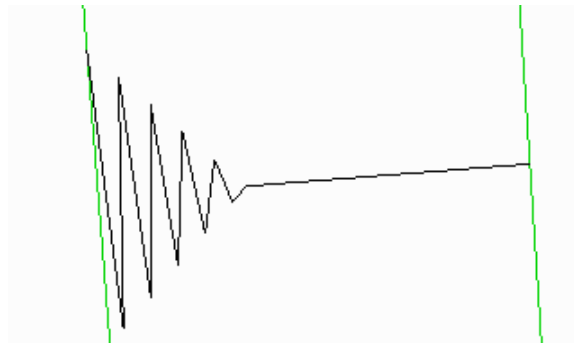
Y style with solid hatch in yellow



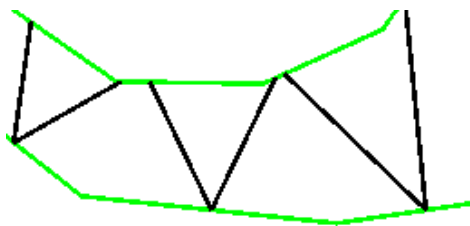
3 line style with hatch in blue



4 line style



Zigzag style



V style

Prompts

Draw Cut/Fill Slope Lines dialog

Pick top of slope polyline: *pick a 3D polyline*

Pick toe of slope polyline: *pick a 3D polyline*

Pulldown Menu Location: Surface > Cut/Fill Utilities

Keyboard Command: slope_lines

Prerequisite: 3D polylines for top and toe of slope

Cut/Fill Movement

This command optimizes earth movement from cut to fill between existing and design surfaces. The earth movement is reported separately by haul ranges which is useful when using different types of earth movers such as dozers and truck/shovel.

Select inclusion and exclusion boundaries:

If an exclusion or inclusion boundary needs to be set for the existing/design tins used in Cut/Fill Movement, first draw either the existing or design tin to screen using the Draw Triangular Mesh command, then select boundaries from the screen when prompted. If no exclusion or inclusion boundaries are selected, Cut/Fill Movement will process the entire existing/design tin (though other options such as **auto size grid limits** still apply).

The screenshot shows the 'Cut/Fill Movement' dialog box with the following settings:

- Grid Cell Size X: 100.0
- Grid Cell Size Y: 100.0
- Min Truck And Shovel Distance: 0.0
- Cut Swell factor: 1.000
- Fill Shrink factor: 1.000
- Auto Size Grid Limits
- Report Details
- Draw Grid
- Grid Layer: Draw_Grid (Set button)
- Draw Transport Paths
- Scale Path Width Relative To Volume
- Path Layer: CF_Movement (Set button)
- Label Volumes (Text Size: 8.000)
- Label Layer: FENCE (Set button)

Buttons: OK, Cancel, Help

Cut/Fill Dialog options:

Grid Cell Size X: Determines the x value (width) of generated cut/fill grid cells.

Grid Cell Size Y: Determines the y value (height) of generated cut/fill grid cells.

Min Truck and Shovel Distance: Defines the truck and shovel cutoff, see *Report Details*.

Cut Swell factor: The cut volume is multiplied by this factor.

Fill Shrink factor: The fill volume is multiplied by this factor.

Auto Size Grid Limits:

Toggle on: The X and Y limits of the existing/design surfaces are calculated then subdivided into grid cells according to Grid Cell X and Y sizes.

Toggle off: User is prompted to window a section of the existing/design surfaces. This windowed area is then subdivided into grid cells according to Grid Cell X and Y sizes.

Report Details:

Toggle on: Total Work, Average distance moved, total fill, and total in-cell movement volume is reported. In addition – for each cell – net fill, a breakdown of cut and fill, easting and northing cell center locations is reported. Finally a breakdown of transported cut is reported by From and To region, Total Volume transported, and Distance cut is moved (excluding internal transport).

Toggle off: Cut fill movement report reports total work average distance moved, total fill and total in-cell movement volume.

*Note: If a *minimum truck and shovel distance* greater than 0.0 is set the report will also include the Truck and Shovel cut off as well as volume moved less that cutoff, volume moved more than cut off and the average distance moved for both of these values.

Draw Grid: Draws a grid on "Grid Layer" (can be set using Draw Grid Layer Name edit box) representing grid cells as defined by Grid Cell Size X, Grid Cell Size Y and the Auto Size Grid Limits toggle.

Draw Transport Paths: Arrows representing the transportation of cut from its initial cut cell to its destination fill cell are drawn to screen according to the "Path Layer" (can be set using Path Layer Name edit box). The Arrows can be scaled to show the relative amount of transported cut by using the "Scale Path Width Relative To Volume" toggle.

Label Volumes: Draws text of the volume for each cut/fill area using the specified layer and text size.

Example: Detailed report

Cuts					
Index	Net Cut	Cut/Fill		Easting	Northing
1	46537.0	48667.4/	2130.4	2233.91	1130.84
2	32356.9	39800.7/	7443.7	2215.62	1334.99
4	16319.4	19722.6/	3403.2	2257.34	1218.23
5	14025.1	17886.4/	3861.3	2204.37	1537.45
6	13718.3	17114.7/	3396.3	2293.28	1333.72
9	6138.4	10280.8/	4142.5	2287.09	1218.40

*Example of Cut information reported by **Detailed** report.

Fills					
Index	Net Fill	Cut/Fill		Easting	Northing
3	18645.4	1699.7/	20345.1	2209.27	1427.56
7	13638.8	1838.0/	15476.8	2332.95	1432.98
8	8238.0	778.8/	9016.8	2299.97	1525.33
10	4598.9	2341.2/	6940.1	2329.34	1130.55
11	182.2	3.6/	185.7	2374.83	1244.80
12	7.2	17.1/	24.3	2374.83	1303.17
13	2.4	0.5/	3.0	2374.83	1387.10

*Example of Fill information reported by **Detailed** report.

Earth Movement Report			
From Region	To Region	Volume (C.Y.)	Distance
1	10	4598.9	95.4
1	External	41938.2	1702.65
2	3	18645.4	92.8
2	External	13711.5	1702.65
4	External	16319.4	1702.65
5	8	8238.0	96.4
5	External	5787.1	1702.65
6	7	13638.8	106.9
6	12	7.2	87.1
6	13	2.4	97.5
6	External	69.9	1702.65
9	11	182.2	91.6
9	External	5956.2	1702.65

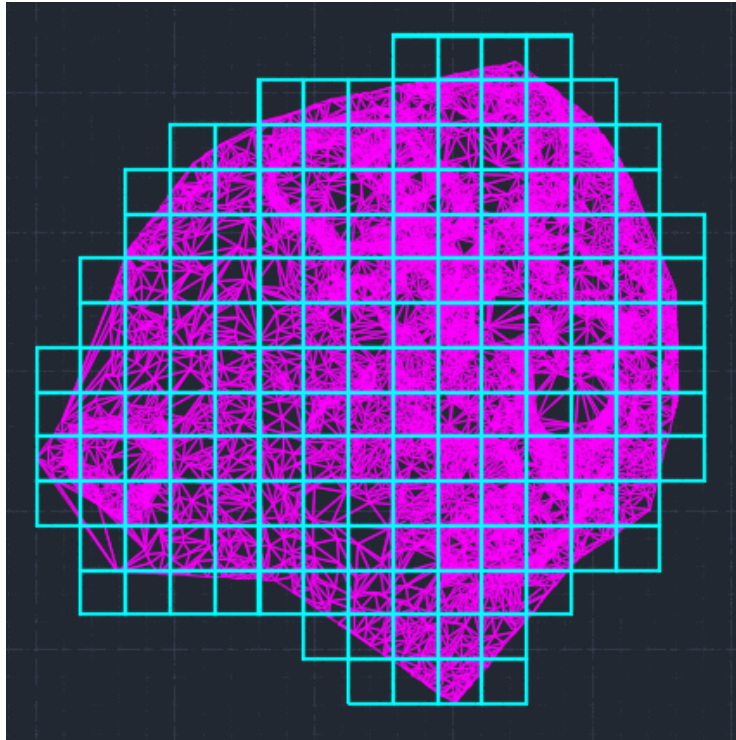
*Example of earth movement reported by **Detailed** report.

Total Work (Distance * Volume)	6,634,319.6	
Average Distance Moved	146.4	
Total Cut	160,151.5	C.Y.
Total Fill	76,369.3	C.Y.
Total In-Cell Movement Volume	31,056.5	C.Y.
Truck And Shovel Cutoff	400.0	
Volume Moved Less Than Cutoff	45,312.8	C.Y.
Average Distance Moved	97.9	
Volume Moved More Than Cutoff	0.0	C.Y.
Average Distance Moved	0.0	

*Example of totals and averages for cut movement (notice that when a **Minimum truck and shovel distance** above 0.0 is defined the report breaks down total and average movement by greater than and less than **Minimum truck**

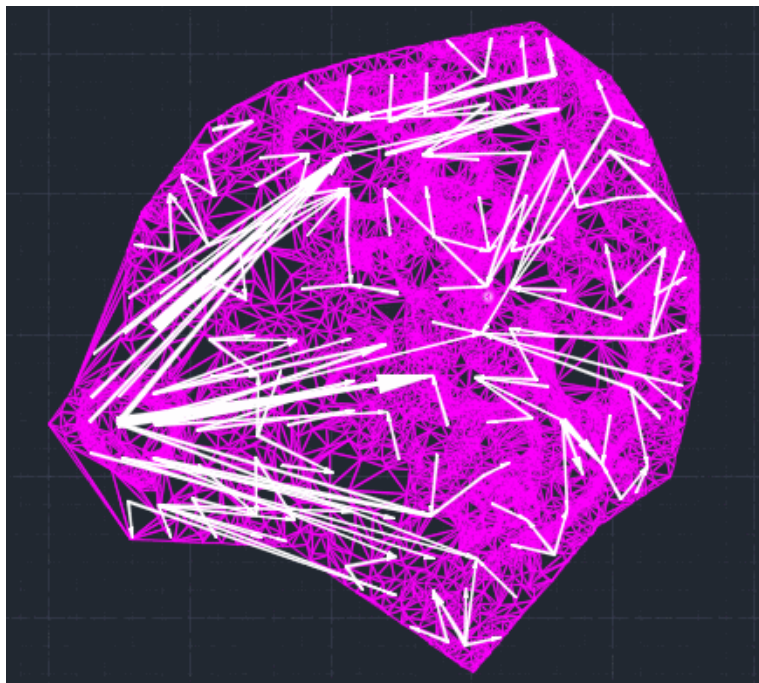
and shovel distance).

Example: Grid



*grid **Autosized to grid limits**. Individual grid cell dimensions are determined by **Grid Cell Size X** and **Grid Cell Size Y**,

Example: Cut movement represented by arrows



*Arrows Scaled to path width relative to transported volume

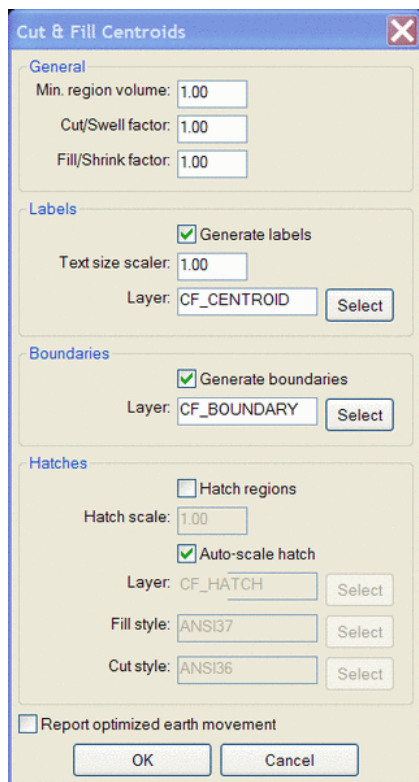
Pulldown Menu Location: Surface > Cut/Fill Utilities

Keyboard Command: cf_move

Prerequisite: Two surface models

Cut/Fill Centroids

This command finds the centroids for each cut and fill area between two triangulation surfaces (.flt or .tin files), with options to draw centroid boundaries, label centroid amounts (in cubic yards for English units), and hatch the areas. Included is a routine to find the optimum movement of the cut to fill volumes which minimizes the total haul distance moved. This routine finds all the areas of cut and fill, and locates the centroid for each area.

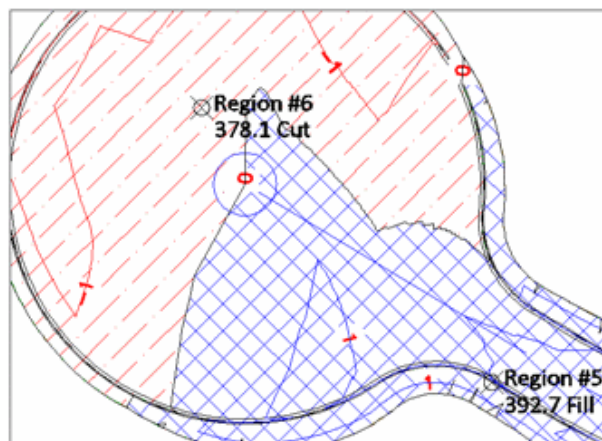
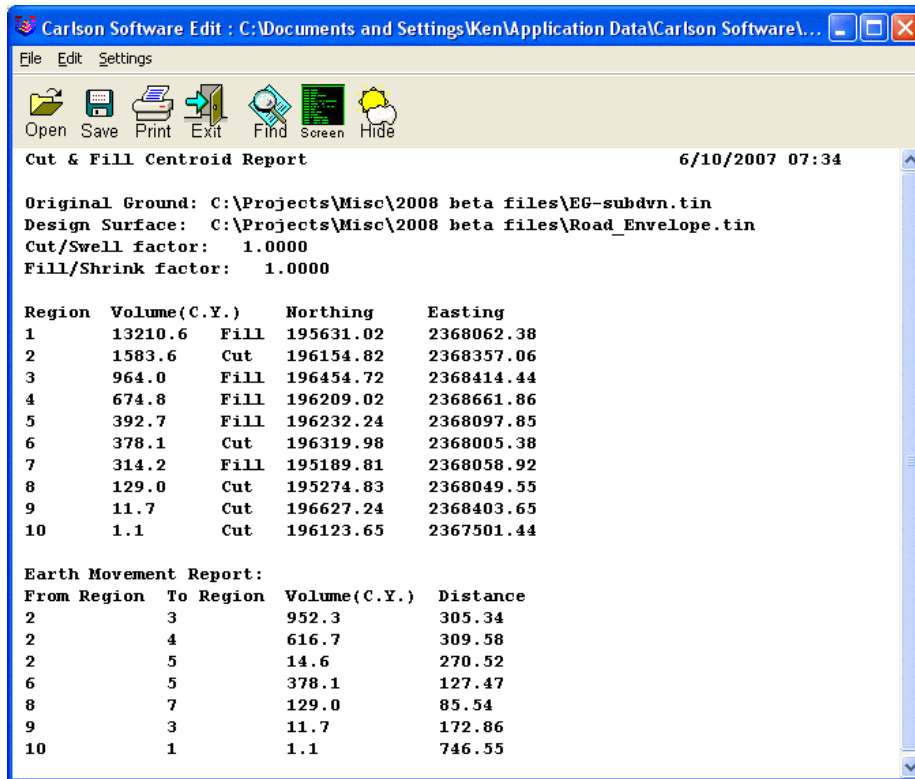


Prompts

Cut & Fill Centroid Locator Dialog

Select Original Ground Surface Dialog *Select an existing surface (.tin, .flt) file*

Select Design Surface Dialog *Select an existing surface (.tin, .flt) file*



Pulldown Menu Location: Surface >> Cut/Fill Utilities

Keyboard Command: cutfillc

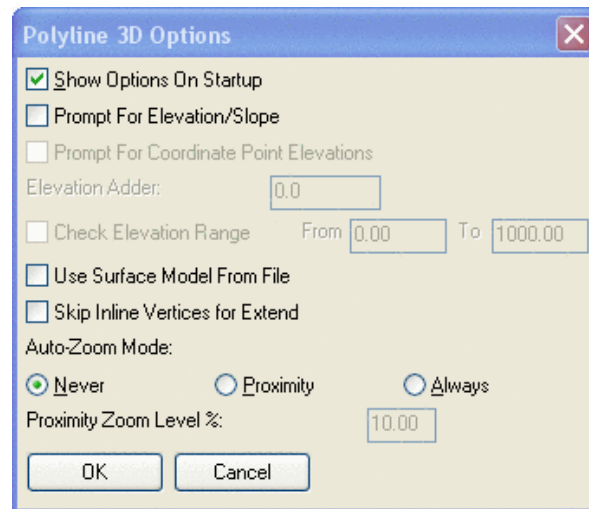
Prerequisite: Two triangulation files, (.flt or .tin)

Draw 3DPoly Perimeter

This command draws a 3D polyline on the PERIMETER layer. This is one way to generate the polyline that is required by the *Calculate Stockpile Volume* and *Calculate Pond/Pit Volume* routines. In these routines, this polyline is used as the inclusion perimeter for volumes. If you are using Carlson points to define the polyline, make sure they are present in the drawing at their real Z elevation, and then set your Object Snap to Insert prior to running this routine. Alternately, you can use point numbers from the current coordinate (.CRD) file. A third option is to obtain the elevations of picked points from a specified surface model. You have a further option to be prompted for each elevation, thus overriding the values found from the included points.

Prompts

If set to display, the 3D Poly options dialog will appear, and then you will be prompted for points to use to draw the 3DPoly Perimeter. Standard Carlson point number input or screen picks using selection methods are valid.



Dialog Options

Prompt for Elevation/Slope: Using the .XY filter allows the user to pick the X and Y coordinate from the screen and type in the elevation. If you use the No response then the Z coordinate of the point picked will be applied.

Use surface model from file: With this option, a surface file is specified, and then with each screen pick, the surface elevation is determined. If Prompt for elevations is set to No, the surface elevation is applied to the polyline vertex. If set to Yes, the surface elevation is displayed as the default, and can be accepted by pressing Enter, or a different elevation can be typed in instead.

Pick point or point number: *pick a point or type a point number*

Arc/Close/Undo/Pick point or point number: *15* This is a point number from the current coordinate (.CRD) file.

Note that if the response to Use Surface model from file is *Yes*, the elevation used is not the point elevation from the coordinate file (.CRD), but the elevation interpolated from the surface.

Arc/Close/Undo/Pick point or point number: *press Enter*

Draw another 3D polyline [Yes/<No>]? *press Enter* Pressing Enter ends the command.

Pulldown Menu Location: Surface >> Stockpile/Pond/Pit Volumes

Keyboard Command: 3dperim

Prerequisite: None

Draw 3DPoly Base Breakline

This command draws a 3D polyline in the BASE_BREAKLINE layer. This polyline is used by the *Calculate Stockpile Volume* and *Calculate Pond/Pit Volume* routines to model the base surface. You may want to set your Object Snap prior to running this routine so that you obtain the elevations of existing points while creating the 3D polyline. Besides picking and entering the points, you can also use point numbers from the current coordinate (.CRD) file.

This routine functions identically to the *Draw 3DPoly Perimeter* command, only placing the resulting 3D polyline on a different layer.

Pulldown Menu Location: Surface > Stockpile/Pond/Pit Volumes

Keyboard Command: 3dbase

Prerequisite: None

Calculate Stockpile Volume

This command is a customized and simplified method for calculating volumes in a situation in which the entire volume to be calculated is above the perimeter elevation, such as in the case of a stockpile of material. The complimentary command, *Calculate Pond/Pit Volume*, is for the opposite situation, in which the entire volume to be calculated is below the elevation of the perimeter.

The program internally computes BASE and FINAL grid surfaces from drawing geometry. The base surface is calculated from a 3D polyline representing the perimeter of the area being analyzed. If that 3D polyline is drawn on the PERIMETER layer, the command will automatically detect and use it. If no 3D polyline is found on that layer, you have an opportunity to manually select another 3D polyline to use. The 3D polyline perimeter can be drawn with the *Draw 3D Polyline Perimeter* command before using this routine.

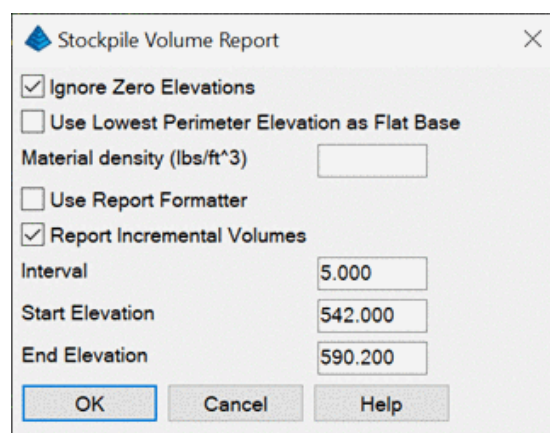
The 3D polyline perimeter is also used as the inclusion perimeter for the volume calculation.

Additional 3D polylines can also be specified to more precisely define the BASE surface. These must be on the BASE_BREAKLINE layer to be used for this purpose. These can be generated by the *Draw 3DPoly Base Breakline* routine.

The FINAL surface is calculated from all of the other selected drawing entities such as points, line, inserts, and polylines, along with the perimeter polyline, **but not including the BASE_BREAKLINE polylines**. These features are used only in computing the BASE surface.

You have the option of setting the resolution of the grids. There is also an option to report the fill volume in stages at an elevation interval.

The *Make 3D Grid File* and *Two Grid Surface Volumes* commands, used in combination, are an alternative to this command, and in any situation in which there are cut and fill volumes between the surfaces, that combination must be used to generate accurate results.



Option	Value
Ignore Zero Elevations	<input checked="" type="checkbox"/>
Use Lowest Perimeter Elevation as Flat Base	<input type="checkbox"/>
Material density (lbs/ft ³)	<input type="text"/>
Use Report Formatter	<input type="checkbox"/>
Report Incremental Volumes	<input checked="" type="checkbox"/>
Interval	5.000
Start Elevation	542.000
End Elevation	590.200

In the options dialog, the **Ignore Zero Elevations** filters out entities at zero elevation. The **Use Lowest Perimeter Elevation as Flat Base** models the base surface at a fixed level using the lowest elevation in the perimeter polyline. The **Material Density** is used to report tons and can be blank to skip the tons report. The **Use Report Formatter** option is for creating custom reports. The **Report Increment Volumes** option reports the volumes in stages.

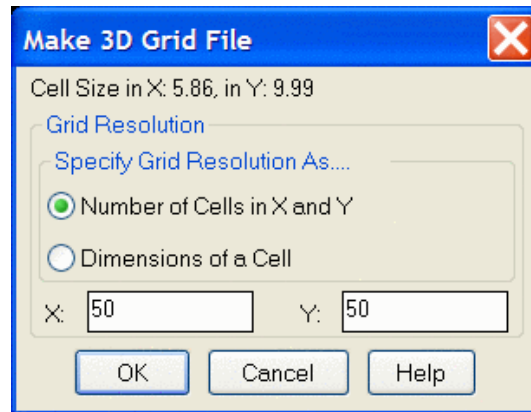
Prompts

Select stockpile entities and perimeter.

Select objects: *pick the objects that define the stockpile and the 3D polyline perimeter*

Stockpile Volume Report dialog

Select stockpile perimeter polyline: *pick the closed 3D perimeter polyline.* If the program didn't find a closed 3D perimeter polyline in the initial selection set, it prompts for this polyline.



Make Grid File dialog Set the resolution and then click OK.

Sample volume report

Volume Report

Lower left grid corner : 5173.56,3970.45

Upper right grid corner: 5477.51,4219.69

X grid resolution: 50, Y grid resolution: 50

X grid cell size : 6.08, Y grid cell size: 4.98

Stockpile volume : 1,251,818.0 C.F., 46,363.63 C.Y.

Area: 58,790.9 S.F., 1.350 Acres

Elevation Range: 543.03 to 596.32

Increment Volume(C.Y.) (C.F.)

549.00-555.00 9863.2 266305.4

555.00-561.00 7868.5 212450.6

561.00-567.00 6161.5 166359.4

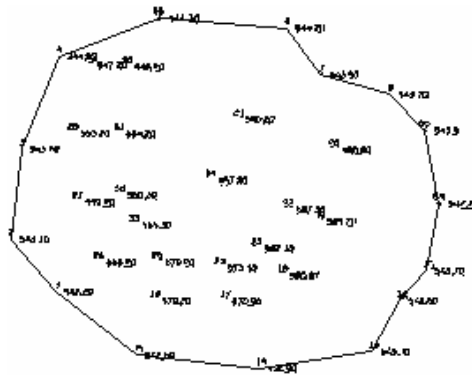
567.00-573.00 4618.4 124697.4

573.00-579.00 3226.7 87121.7

579.00-585.00 2175.9 58750.5

585.00-591.00 1004.7 27127.8

591.00-596.32 111.7 3015.6



Stockpile defined by points and a 3D polyline perimeter
Window these objects to obtain the volume report

Pulldown Menu Location: Surface >> Stockpile/Pond/Pit Volumes

Keyboard Command: stockvol

Prerequisite: Data representing the stockpile surface and a 3D polyline representing the perimeter of the stockpile.

Calculate Pond/Pit Volume

This command is a customized and simplified method for calculating volumes in a situation in which the entire volume to be calculated is below the perimeter elevation, such as in the case of a pond or pit. The complimentary command, *Calculate Stockpile Volume*, is for the opposite situation, in which the entire volume to be calculated is above the elevation of the perimeter.

The program internally computes BASE and FINAL grid surfaces from drawing geometry. The base surface is calculated from a 3D polyline representing the perimeter of the area being analyzed. If that 3D polyline is drawn on the PERIMETER layer, the command will automatically detect and use it. If no 3D polyline is found on that layer, you have an opportunity to manually select the 3D polyline to use. The 3D polyline perimeter can be drawn with the *Draw 3D Polyline Perimeter* command before using this routine.

The 3D polyline perimeter is also used as the inclusion perimeter for the volume calculation.

Additional 3D polylines can also be specified to more precisely define the BASE surface. These must be on the BASE_BREAKLINE layer to be used for this purpose. These can be generated by the *Draw 3DPoly Base Breakline* routine.

The FINAL surface is calculated from all of the other selected drawing entities such as points, line, inserts, and polylines, along with the perimeter polyline, **but not including the BASE_BREAKLINE polylines**. These features are used only in computing the BASE surface.

You have the option of setting the resolution of the grids.

Besides reporting the entire volume between the two surfaces, the report also includes the volumes at an elevation interval from the bottom to the top. These stage-storage volumes can also be stored to a capacity file (.cap) that can be used with the Hydrology module. In addition to the stage-storage volumes, the *Report Incremental Volumes* option reports the storage within an elevation range instead of relative to the surface.

The *Make 3D Grid File* and *Two Grid Surface Volumes* commands, used in combination, are an alternative to this command, and in any situation in which there are both cut and fill volumes between the surfaces, that combination must be used to generate accurate results.

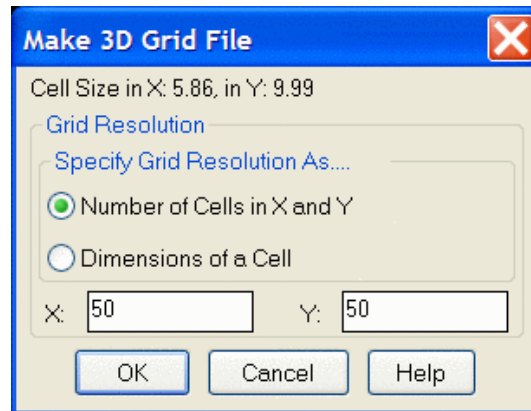
Prompts

Ignore Zero Elevations [<Yes>/No]? *press Enter*

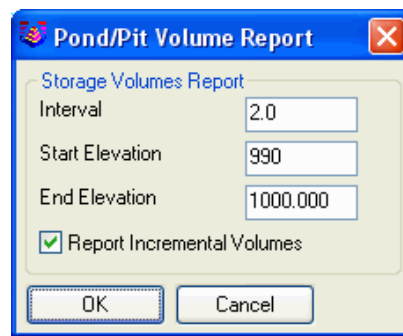
Select Pond/Pit entities and perimeter.

Select objects: *pick the objects that define the surface and the 3D polyline perimeter*

Select Pond/Pit perimeter polyline: *pick the polyline*



Make Grid File dialog Set the resolution and then click OK.



Sample Volume Report:

Lower left grid corner : 8361.29,10856.76

Upper right grid corner: 8940.96,11302.31

X grid resolution: 50, Y grid resolution: 50

X grid cell size : 11.59, Y grid cell size: 8.91

Pond/Pit volume : 602,182.5 C.F., 22,303.06 C.Y., 13.82 Acre-Ft

Area: 114,312.7 S.F., 2.624 Acres

Elevation Range: 987.08 to 1000.00

Storage Volumes

Elevation Storage(AcreFt) (C.Y.) (C.F.) Area(Acre)

990.00 0.26121 421.4 11378.5 0.281

992.00 1.18631 1913.9 51675.9 0.646

994.00 2.85639 4608.3 124424.4 1.033

996.00 5.46346 8814.4 237988.4 1.591

998.00 9.13982 14745.6 398130.8 2.083

1000.00 13.82421 22303.1 602182.5 2.624

Increment Storage(AcreFt) (C.Y.) (C.F.) Area(Acre)

990.00-992.00 0.92510 1492.5 40297.4 0.365
 992.00-994.00 1.67008 2694.4 72748.5 0.387
 994.00-996.00 2.60707 4206.1 113564.0 0.558
 996.00-998.00 3.67636 5931.2 160142.4 0.492
 998.00-1000.00 4.68438 7557.5 204051.7 0.541

Pulldown Menu Location: Surface >> Stockpile/Pond/Pit Volumes

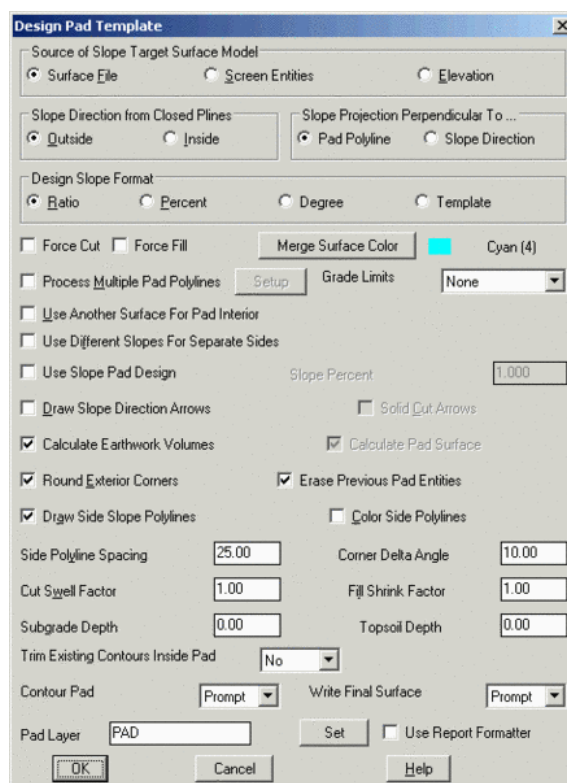
Keyboard Command: pitvol

Prerequisite: Data representing the pond/pit surface and a 3D polyline representing the perimeter of the pond/pit.

Design Pad Template

This command creates design slopes from a perimeter polyline at specified cut/fill slopes to reach existing ground. This routine can be used to design building pads, pits, roads, ditches, stockpiles, etc. The design is drawn as 3D polylines for the cut/fill slopes and for the daylight perimeter where the design meets existing ground.

Before beginning this routine, you must have drawn the polyline representing the outside edge of the feature to model. The edge is drawn as a polyline which can be either a 2D or 3D closed or open polyline. For a 2D polyline, the program will prompt for an elevation for the pad perimeter. With a 3D polyline, the pad perimeter is set to the elevations of the 3D polyline. For an open polyline, the program will prompt for the side for the design. With a closed polyline, the program designs the slopes either outward or inward depending on the settings in the dialog.



Under **Source of Slope Target Surface Model**, choose between a Surface File (.GRD, .FLT, .TIN), Screen Entities, or a specific Elevation. If using Screen Entities, the routine internally calculates a gridded model, the limits of which are specified by screen picks. Make sure that the grid area covers the entire area for the pad including room for the

cut/fill slopes.

For closed pad perimeters, there is a **Slope Direction from Closed Plines** option to draw the slopes inward or outward from the perimeter. The outward method starts the slopes at the design elevation of the perimeter and projects out to intersect the existing surface. The inward method projects the slopes inside to reach the grid surface or a set elevation. Outward sloping would be used for such things as building pads, parking lots, etc. where the interior remains as a defined surface. Inward sloping would be used for such things as the top edge of an excavated pit or pond where the interior side slopes project downward at the specified slopes until reaching the original ground surface.

The **Slope Projection Perpendicular To** option applies to sloping pad perimeters. The Pad Polyline method creates the user-specified slope perpendicular to the pad perimeter. The Slope Direction method accounts for the slope of the pad perimeter and makes the final surface to match the user-specified slope. For example, if the pad perimeter is at a 10% slope and the fill slope is at 2:1, then the Pad Polyline method would create fill slopes that are 2:1 perpendicular to the pad while slightly steeper (1.96:1) for the actual slope that goes in the slope direction with the effect of the sloping pad perimeter. For the same case except with the Slope Direction method, the resulting slope perpendicular to the pad is less steep (2.04:1) while the actual slope in the slope direction is exactly 2:1.

Under **Design Slope Format**, choose between *Ratio*, *Percent*, *Degree* or *Template*. The use of a Template allows for complex slopes to be applied, and is also an alternative approach to road design. The template (.TPL) file is created in the *Design Template* routine in the Roads menu. When using a template, the pad perimeter represents the centerline. One way to create the pad perimeter for the template is to use the *Profile to 3D Polyline* command which converts a 2D centerline to a 3D polyline using a design profile. With a template, the program uses not only the cut and fill slopes from the template file but also draws all the template grade points such as edge of road, curb and ditch. The subgrade, superelevation and template transition options of the template file are not used in this command. These options are only applied in the *Process Road Design* command. The grade points are drawn as 3D polylines parallel with the centerline. Cross section 3D polylines that include the grade points are also drawn at the specified interval.

The **Force Cut** option will try the cut slope to find a catch point even when the pad perimeter starts out in fill. This is possible when the existing ground is rising faster than the cut slope. Likewise the **Force Fill** option will try the fill slope to find a catch point when the pad starts out in cut.

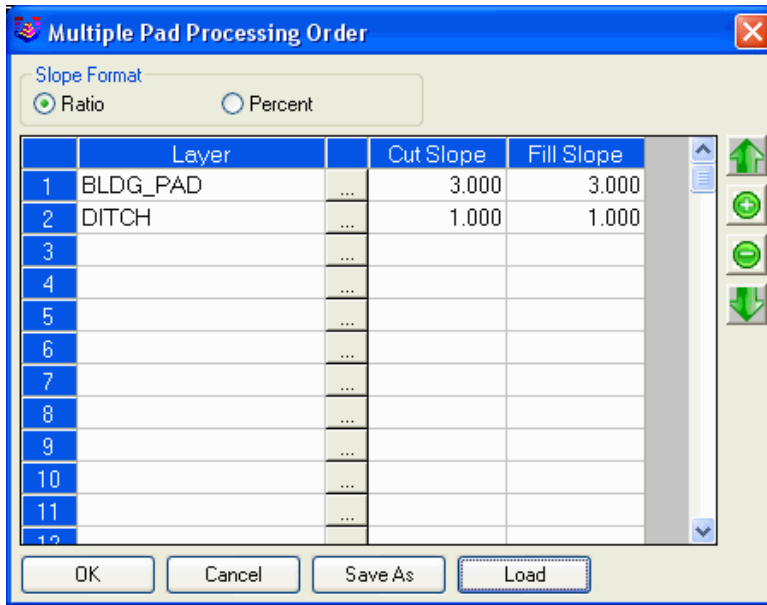
The **Grade Limits** option makes the cut/fill slopes fit within the grade limits either by making the slopes steeper or by putting in a vertical retaining wall. The retaining wall can be placed at the grade limit or at the pad perimeter. When this option is on, the program prompts to select a closed polyline to set the grade limits.

The **Min Cut/Fill Height** setting creates no tie slope when the cut/fill is less than a minimum. To always draw the tie slope, set this value to zero.

The **Merge Surface Color** is used to set the color of the triangles within the pad when using a triangulation surface as the target surface and updating this surface with the pad design. The triangle colors are shown in the 3D Viewer graphics.

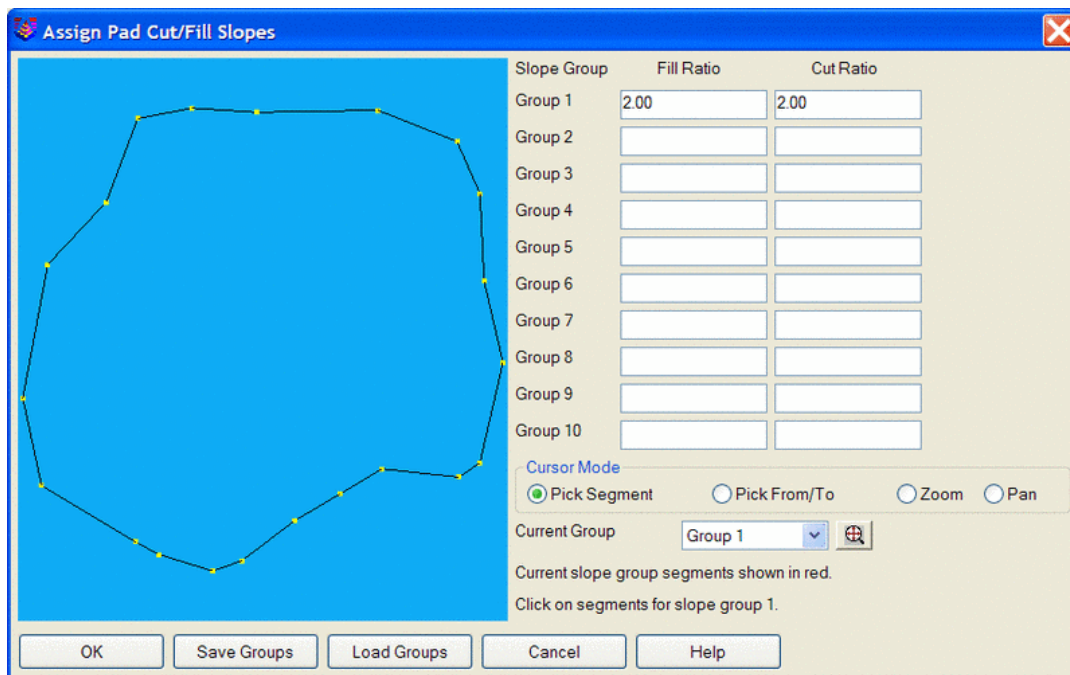
The **Process Multiple Pad Polylines** option allows you to process multiple pad perimeter polylines at a time instead of a single pad perimeter. The program will prompt for a selection set of pad perimeter polylines and then cycle through and run the design on each one. There will be one final report for the earthworks for all the pads. The Setup function allows you to specify different cut/fill slopes by layer and also to set the processing order by layer. For example, in the case of processing both building pads with a shallow slope and ditch polylines at a steeper slope, you could set up the processing order to do the building pad first and the ditch last so that the ditch cut slopes will carve out any overlap with the building fill slopes. These pad layer slope and order assignments can be saved

and loaded from a .PAD file.



Use Another Surface for Pad Interior will bring up a prompt for another Surface file (.GRD, .FLT, .TIN) to use for the design surface within the starting pad perimeter. Otherwise the program will model the pad interior by straight interpolation from the starting pad perimeter elevations. For example, if a building pad has a starting pad perimeter at a set elevation and the pad is supposed to be flat, then this option is not needed. This option is needed in a case where you are designing a pit and the starting pad perimeter is a 3D polyline that follows an undulating pit bottom surface. The pad design will model the pit side slopes. In order to model the undulating bottom of the pit, you need the Use Another Surface for Pad Interior option to select a surface that models the pit bottom.

Use Different Slopes for Separate Sides allows you to specify different slopes for different sides of your pad polyline. If this is toggled ON, the Assign Pad Cut/Fill Slopes dialog is invoked, where you can create multiple Slope Groups along the Pad Template polyline and set the Cut and Fill design ratios for each.



Use Slope Pad Design allows you to set a cross slope amount for the top of the pad. You will be prompted to

screen pick two points that designate the slope direction. For automatic balancing of cut/fill quantities, you will be prompted to find the optimal slope and slope direction.

Draw Slope Direction Arrows draws an arrow on the outslopes that points in the downhill direction. Arrows on fill slopes are drawn as solid filled.

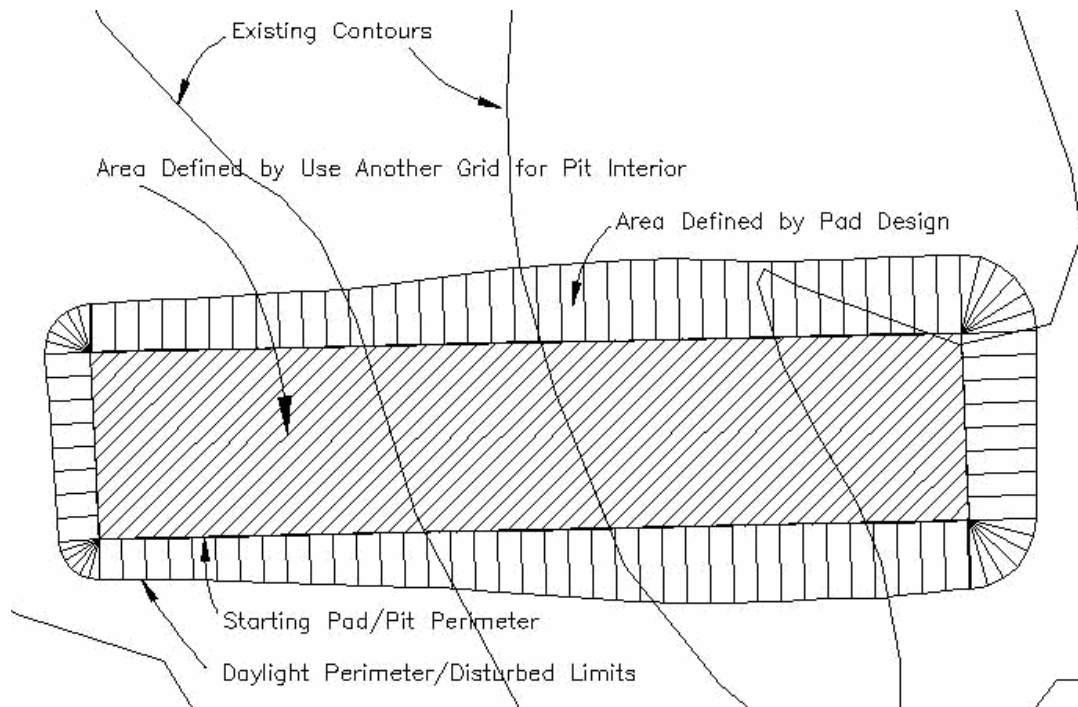
Solid Cut Arrows allows you to choose between drawing the cut arrows as solid filled or as wire frame.

Round Exterior Corners holds the outslopes around the corners. Otherwise the side outslopes stay straight until they meet at the corners as shown in the figure.

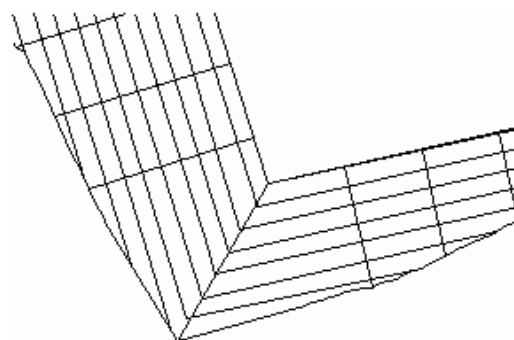
Erase Previous Pad Entities erases drawing geometry created with this command previously.

When **Draw Side Slope Polylines** is ON, Design Pad Template will draw 3D polylines perpendicular to the pad perimeter from the pad to the catch point.

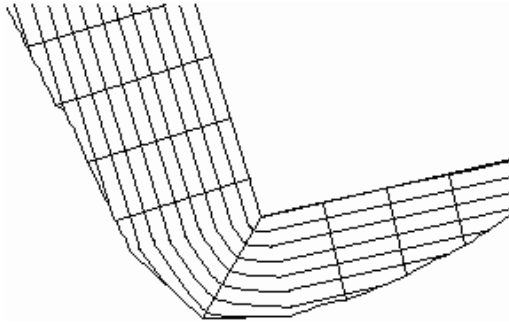
Color Side Polylines assigns different colors to Cut and Fill Side Polylines to make them easier to distinguish.



Example of pit design for option of Use Another Grid for Pad Interior



Pad corner without round corners option



Pad corner with round corners option

Side Polyline Spacing specifies the interval at which to draw the Side Slope Polylines. Besides at the interval, side slope polylines are also drawn at grid corners.

Corner Delta Angle is the delta angle in degrees between side slope polylines to span the delta angle around exterior corners.

The **Subgrade Depth** is applied to lower the design surface within the pad for the volumes calculations. The volumes are from the existing surface to the base of the subgrade. The effect of the subgrade is to increase cut and decrease fill.

The **Topsoil Depth** is used to report the topsoil removal volume by removing this depth within the pad disturbed area.

Cut volume is multiplied by the **Cut Swell Factor** in the final volume report.

Fill volume is multiplied by the **Fill Shrink Factor** in the final volume report.

The **Contour Pad** option draws contours on the pad. At the end routine, a dialog lets you set the contouring options. Usually you should specify a new contour layer and turn off smoothing.

The **Write Final Surface** option creates a surface model of the pad using the elevations of the pad within the disturbed area polyline and using the original ground surface everywhere else. At the end of the routine, the program will prompt for the surface file name to create.

The **Trim Existing Contours Inside Pad** option trims existing contours inside the disturbed limits of the pad.

You must specify the **Pad Layer Name** that the pad 3D polylines will be drawn on.

The **Use Report Formatter** option allows for making a customized report and outputs to various formats.

There is an option to calculate volumes for the pad design. The volumes are calculated by comparing the existing surface with the pad design. The inclusion perimeter for the volume calculation is the daylight perimeter polyline which represents the limits of disturbed area. The existing surface model is defined by the existing surface file (.GRD, .FLT, .TIN) or screen entities selected at the beginning of the command. The pad design surface is calculated by making a surface from the pad 3D polylines including the starting pad perimeter, the side polylines and the daylight perimeter.

Besides calculating the volumes in the *Design Pad Template* routine, you can also calculate the volumes with the *Two Surface Volumes* command, or the *Volumes by Triangulation* command. Two Surface Volumes works with two grid files, Volumes by Triangulation works with two TIN files. The design surface for Two Surface Volumes can be the final output surface from Design Pad or you can create a design surface with *Make 3D Grid File* using the 3D

polylines created in *Design Pad*. You could also create a TIN surface of the design surface using *Triangulate and Contour*. Some of the reasons to use either the Two Surface Volumes command or the Volumes by Triangulation command are that these volume routines have more output options (cut/fill color maps, etc.) and you can check the volumes by plotting or contouring the surface files. Also, you can combine several pads and other final surfaces by running *Make 3D Grid File* or *Triangulate and Contour* and then use these volume commands to calculate the overall site volumes.

The design is drawn as 3D polylines and the earthwork volumes are calculated. Before ending, the program allows you to adjust the design by changing the pad elevation, slopes and offset. The program can find the cut/fill balance by automatically adjusting the pad elevation. If adjustments are specified, the pad polylines are redrawn and the volumes recalculated.

A few key notes:

1. If the Source of Slope Target Surface Model is set to a Surface File, and the surface file used is a grid file, then the surface produced from the designed pad will be a grid surface and a grid file (.GRD).
2. If the Source of Slope Target Surface Model is set to a Surface File, and the surface file used is a TIN file, then the surface produced from the designed pad will be a triangulated surface and a TIN file (.TIN).
3. If the Surface used as a Target Surface is listed in the Surface Manager, the prompt seen in the Design Pad Template command is whether or not to Update the Surface, which is the Target Surface, so if you say "Yes," your Existing Ground Surface will now essentially contain the designed pad. So if you want to maintain an unedited version of Existing Ground, you may want to start with a copy of the Existing Ground Surface.
4. If the Surface used as a Target Surface is not listed in the Surface Manager, the prompt seen in the *Design Pad Template* command is whether or not to create a new surface of the combined surfaces.
5. If you respond "Yes" to the prompt about whether to contour the designed pad, the contouring dialog box has an option of whether to write the designed pad as a new surface, which will be only the area within the limits of the new design, not the entire Target Surface and design pad surface combined.

Prompts

First you are presented with the Design Pad Template dialog box.

If the Source of Slope Target Surface Model is set to a Surface File, you will first be asked to:

Pick the top of pad polyline: *select perimeter polyline*

Then the Select Slope Target Surface dialog box is presented. Choose the Slope Target Surface file, pick Open. You then proceed to enter the slope parameters of the pad...

If the Source of Slope Target Surface Model is set to a Screen Entities, you will first be asked to:

Pick Lower Left limit of pad disturbed area: *pick lower left* These prompts appear for the Screen Entities surface model method.

Pick Upper Right limit of pad disturbed area: *pick upper right* Be sure to pick these limits well beyond the area of the top of pad polyline in order to make room for the outslopes.

Make Grid File Dialog After selecting the limits of the disturbed area the program will generate a 3D grid that represents the surface. Specify the grid resolution desired and select OK.

Then,

Pick the top of pad polyline: *select perimeter polyline*

Then proceed to enter the slope parameters of the pad...

Enter the fill outslope ratio <2.0>: 2.5

Enter the cut outslope ratio <2.0>: 2.5 After entering outslopes slope ratios, a range of elevations along the pad

top will be noted.

Enter the pad elevation <29.54>: 39

Calculate earthwork volumes (<Yes>/No)? *press Enter*

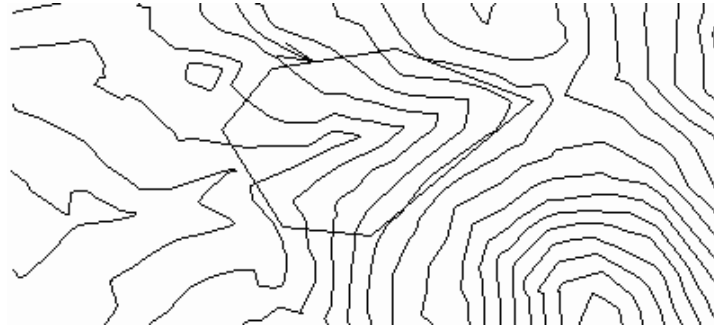
Report Viewer Reports cut/fill volume.

Adjust parameters and redesign pond (Yes/<No>)? *press Enter*

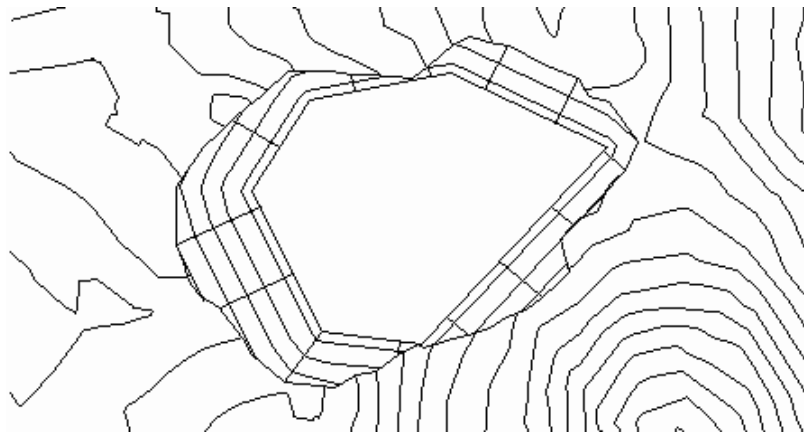
Write final surface to grid file (Yes/<No>)? *press Enter*

Trim existing contours inside pad perimeter (Yes/<No>)? *press Enter*

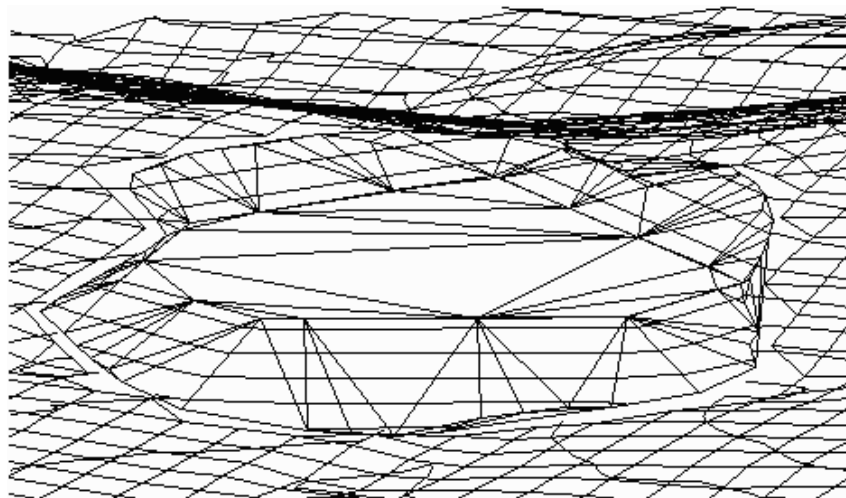
Contour the pad (<Yes>/No)? *press Enter*



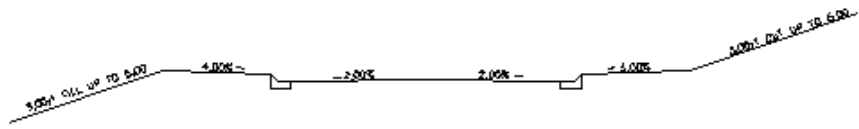
Existing contours with top of pad perimeter polyline



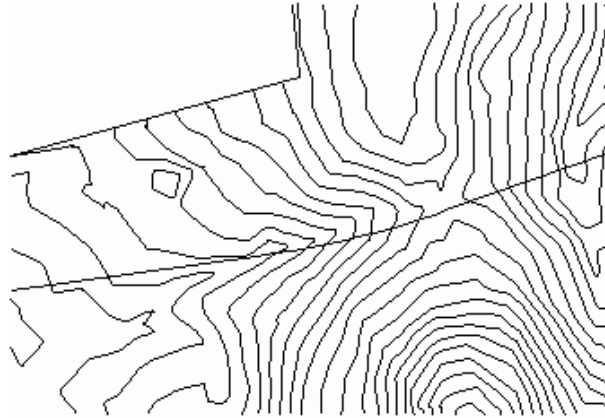
Pad template with contours



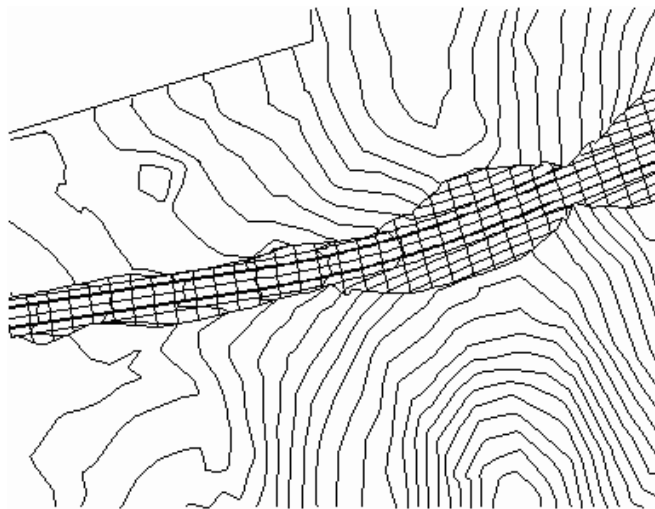
3D view of pad with DTM of surface and triangulation faces of pad



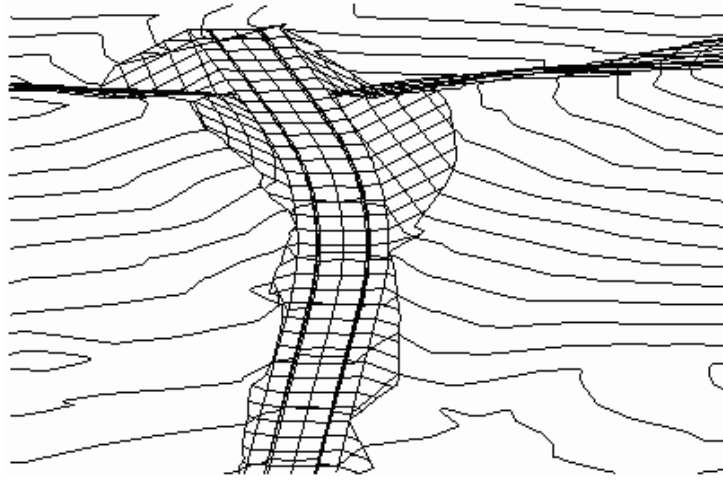
Template to apply in Design Pad Template



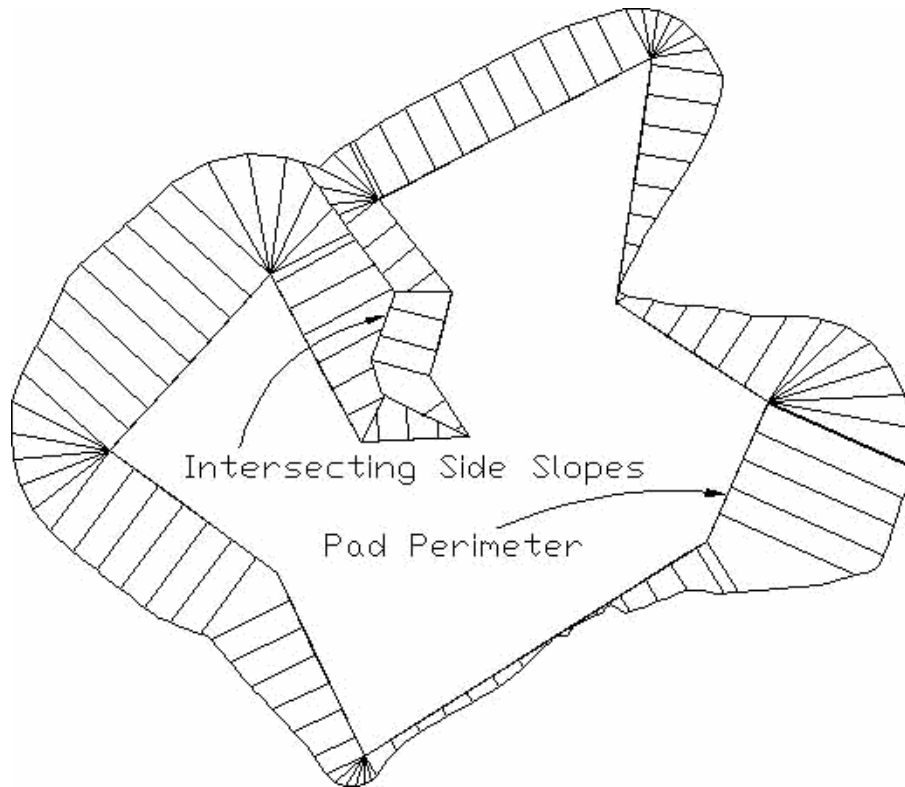
Existing surface with 3D polyline centerline



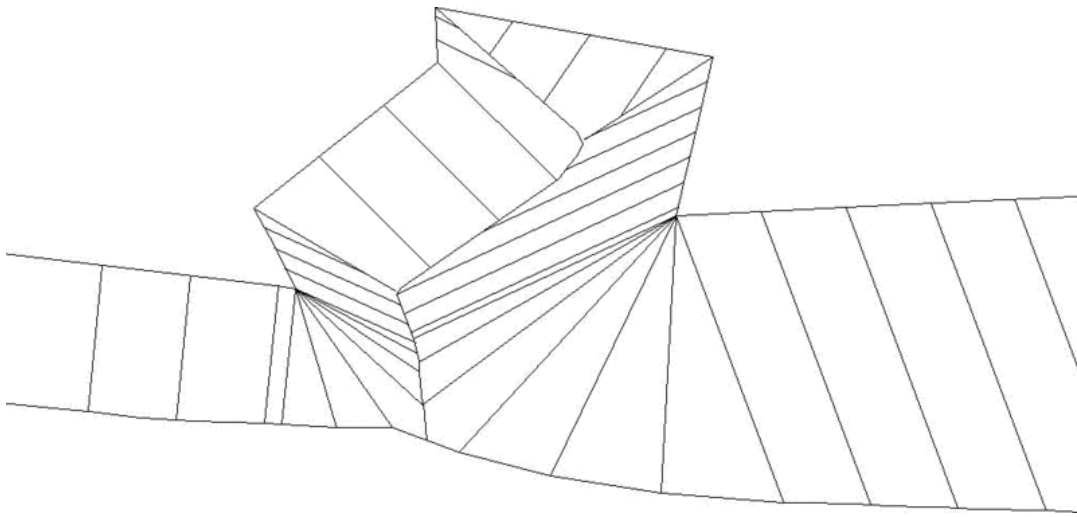
Result of Design Pad Template showing template grade polylines, cross section polylines, cut/fill slopes, and final contours



Viewpoint 3D view of Design Pad Template



Design Pad Template can also handle self-intersecting side slopes



Viewpoint 3D view of intersecting side slopes

Pulldown Menu Location: Surface

Keyboard Command: pad

Prerequisite: A pad perimeter polyline and surface entities or a surface file for an intercept target.

Edit Pad Template

This command is used in conjunction with *Design Pad Template*. It allows the user to modify an existing pad design that has been generated with *Design Pad Template*. The original design criteria of the pad can be modified and the routine will automatically re-calculate volumes, re-draw contours, disturbed limits, tin lines, etc. Possible modifications to the pad include cut/fill slopes, pad elevation, x-y position (the pad can be moved to a new location), horizontal offset, a single pad vertex can be moved to a new location, a new target surface can be specified, and finally, the pad and all of its elements can be deleted. Once the new design parameters are set, the routine will regenerate the pad based on the new parameters. The standard report viewer opens to display the new volume calculation and other design info.

Edit Pad Template requires a target surface to project to, which is typically the surface designated as the Source of Slope Target Surface Model in the Design Pad Template command. In Design Pad Template, when prompted to "Update Surface File" select *Yes*.

First the user is prompted to pick the pad polyline, which must be the original polyline used with *Design Pad Template* command. Subsequent edits to the Pad Template must also be initiated with the selection of that polyline. The Editing functions are displayed in an interactive docked sidebar dialog.

<input type="checkbox"/> Use Slope Groups	Set
Fill Slope	1.00
Cut Slope	1.00
Pad Elevation	1010.0000
Pad Volume	
Fill 9496.0 C.Y.	
Cut 1773.9 C.Y.	
Surface SIMO2	Set
Move Pad	Move Vertex
Add Vertex	Remove Vertex
Offset	Edit Vertex Z
Balance	Rotate Pad
Report	Delete
Write TIN	Exit Help
Process	

Use Slope Groups allows the assignment of varied slopes along the length of the Pad Template. Select the checkbox, and pick the Set button to access the Assign Pad Cut/Fill Slopes dialog box.

Fill Slope: displays the original fill slope criteria, *select* to edit to a new value.

Cut Slope: displays the original cut slope criteria, *select* to edit to a new value.

Elevation Delta: *enter a value* to adjust the elevation of the entire pad template.

Pad Volume: displays the cut/fill volume generated with the last use of either *Design Pad Template* or *Edit Pad Template*.

Surface: displays the target surface, *pick Set button* to change.

Move Pad: *Pick* to move the Pad Template to a new location on the site. The user is prompted at the command line to "Pick start point for translation" and then "Pick end point for translation." The pad is moved to the new location, with contours, tin lines, disturbed area and volumes regenerated on the fly.

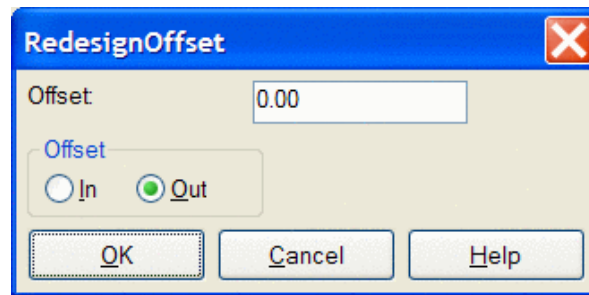
Move Vertex: *Pick* to move a vertex on the Pad Template. The user is prompted to screen select any vertex within the pad and place it in a new location. The pad vertex is moved to the new location, with contours, tin lines, disturbed area and volumes regenerated on the fly.

Add Vertex: *Pick* the position for a new point to add into the pad perimeter.

Remove Vertex: Select a point on the pad perimeter to remove.

Offset: *Pick* to apply a horizontal offset of the pad perimeter polyline. Enter the amount of offset desired in the field and then specify whether it is to go in or out.

Note: For subsequent operations of *Edit Pad Template*, the original pad polyline must be selected when prompted. The offset pad polyline is not eligible for selection.



Edit Vertex Z: Select a point on the pad perimeter to change the elevation. This function applies for 3D pad perimeters.

Rotate Pad: *Pick* to rotate the Pad Template.

Balance: *Pick* to automatically adjust the Pad Template elevation to balance the cut and fill volumes.

Delete: removes all of the pad design entities that were generated with *Design Pad Template* or *Edit Pad Template*.

Report: *Pick* to generate a cut/fill volume report.

Write TIN: Saves a TIN surface file of the pad surface.

Process: proceeds with the re-design of the pad using the current criteria.

Pulldown Menu Location: Surface

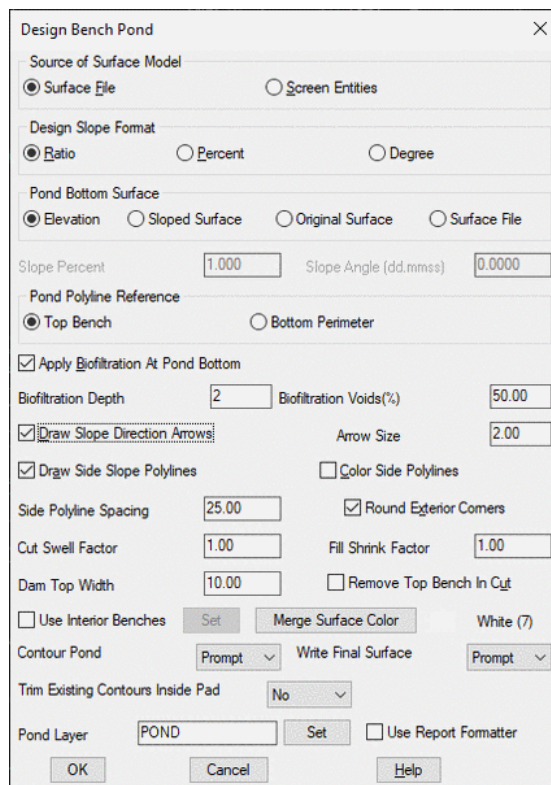
Keyboard Command: repad

Prerequisite: A Pad Template generated with the *Design Pad Template* command, and the surface model file used as a target surface.

Design Bench Pond

This feature will design a pond from a closed 2D polyline that defines the top dam perimeter of the pond. Before beginning this routine, you must have surface entities or surface files and a closed polyline that represents the top of the dam. The command creates a top of dam of the specified width, and then projects inward to model the pond, and outward to model the slopes to match to the target surface. Besides drawing the bench pond in 3D, the command also reports the earthwork to build the pond and the stage-storage data. There is also an option to output the stage-storage data to a .CAP file to use with Draw Stage-Storage or for hydrograph routing. After creating the pond and reporting the earthwork and stage-storage, the program prompts whether to adjust the design. There are three types of adjustment. One adjustment is to balance the earthwork cut/fill by adjusting the design elevation. Another adjustment is to resize to meet a specified stage-storage. The target storage adjustment can be done by either changing the pond bottom elevation or by offsetting the pond perimeter in or out. The third type of adjustment is to manually change one of the design parameters such as slopes or elevations.

This command starts with the dialog shown here.



Source of Surface Model: The existing surface may be defined by a 3D rectangular grid mesh (.grd file), a triangulation file (.flt or .tin file), or by screen entities. If the "Screen Entities" option is chosen, a grid file representing the existing surface will be internally created by using the user selected screen entities that depict the surface (Contours, Tri-Mesh, 3D Polys).

Design Slope Formula: Choose how you want to specify the slopes, either by ratio, percent or degree.

Pond Bottom Surface: Choose how to specify the pond bottom surface by either the original surface, sloped surface, surface file, or use a fixed set elevation. The Slope Surface has settings for the slope amount and direction that the program will use to build the bottom of pond surface. The Surface File method will prompt for a separate TIN surface file to use for the bottom of the pond. This method is useful for building a pond with bottom features like a V-ditch or specific low point. For this method, the bottom surface must be built using Triangulate & Contour before running Design Bench Pond.

Pond Polyline Reference: The program will prompt for you to select a pond perimeter polyline from the drawing. With this option, you can choose between using a perimeter polyline that represents the pond top bench or the pond bottom perimeter.

Apply Biofiltration At Pond Bottom: this option provides a storage at the bottom of the pond that isn't 100% water capacity. The depth is from the bottom of the pond, and the void is smaller than 100%, in the example below it is 50%. The Biofiltration is calculated in the storage and shown in the stage-storage report.

Draw Slope Direction Arrows: This option draws an arrow on the outslopes that points in the downhill direction. Arrows on fill slopes are drawn as solid filled.

Arrow Size: Specify the size for the slope direction arrows.

Draw Side Slope Polylines: This option draws 3D polylines from the pond top bench to the outside catch perimeter and inside to the pond bottom.

Color Side Polylines: This option will color the side slope polylines as red/blue for cut/fill.

Side Polyline Spacing: Defines the interval along the top of pond perimeter to draw 3D lines from the top of pond to the tie at the outslope.

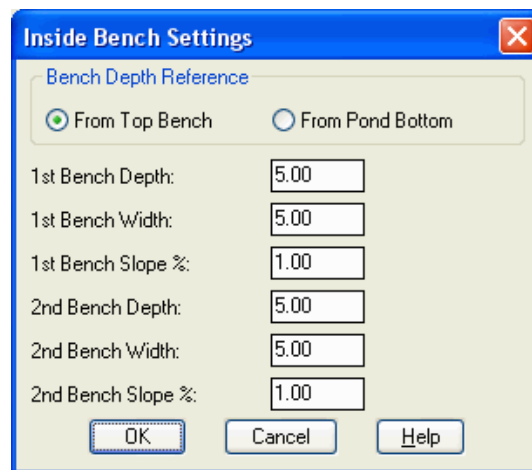
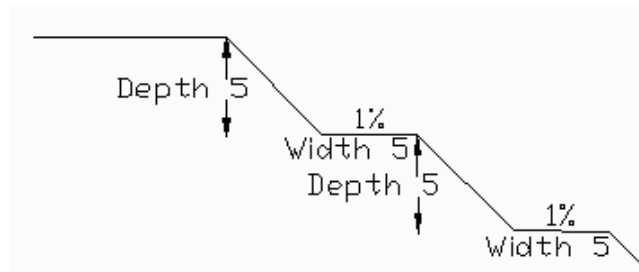
Cut Swell Factor: This value is multiplied by the earthwork cut volume for the report.

Fill Shrink Factor: This value is multiplied by the earthwork fill volume for the report.

Dam Top Width: Specify the width for the top of the dam.

Remove Top Bench in Cut: If this option is checked ON and the top of dam is in cut, then the bench will be removed.

Use Interior Benches: This option puts in up to two safety benches on the pond interior slope. The depth for the benches can be specified from either the top or from the bottom of the pond. If you only have one bench, then the fields for the second bench should be set blank.



Pond Layer Name: Specify the layer for the pond entities.

The design is drawn as 3D polylines with an option to draw contours on the pond, and the earthwork volumes and stage-storage volumes are calculated.

Prompts

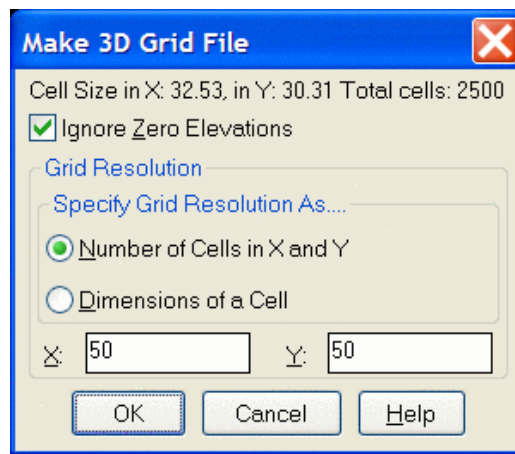
First the Design Bench Pond dialog box is presented.

If the Source of Surface Model is set to Screen Entities, when you pick OK you are prompted to:

Pick Lower Left limit of pond disturbed area: *Pick lower left*

Pick Upper Right limit of pond disturbed area: *Pick upper right* Be sure to pick these limits well beyond the area of the top of dam polyline in order to make room for the outslopes.

Make Grid File Dialog: Specify the grid resolution desired and select OK. Carlson Civil generates a 3D grid that represents the existing surface, using the drawing entities that fall within the specified area.



You are then prompted to:

Pick the top of dam polyline: *Select closed polyline*

Then proceed to design the pond...

If the Source of Surface Model is set to Surface File, when you pick OK, you are prompted to:

Pick the top of dam polyline: *Select closed polyline*

Then select the Existing Ground Surface Model file to use. Then proceed to design the pond...

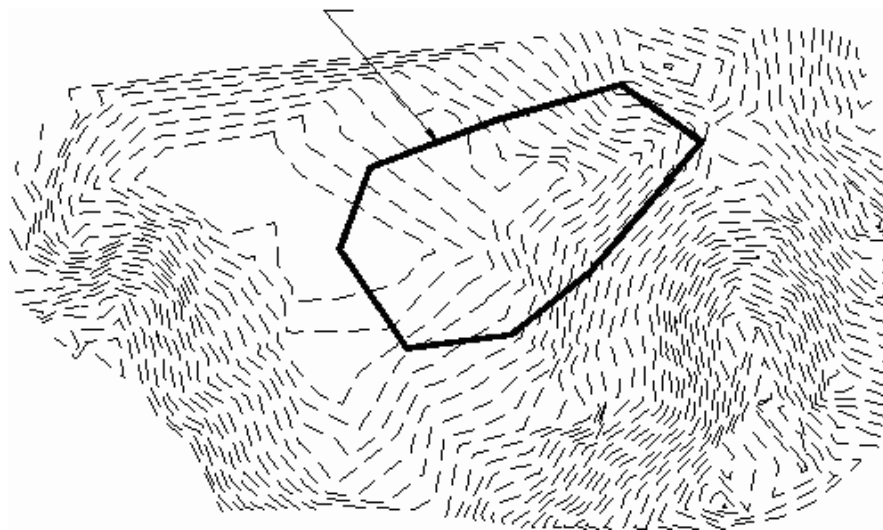
Enter the fill outslope ratio <2.0>: *Enter*

Enter the cut outslope ratio <2.0>: *Enter*

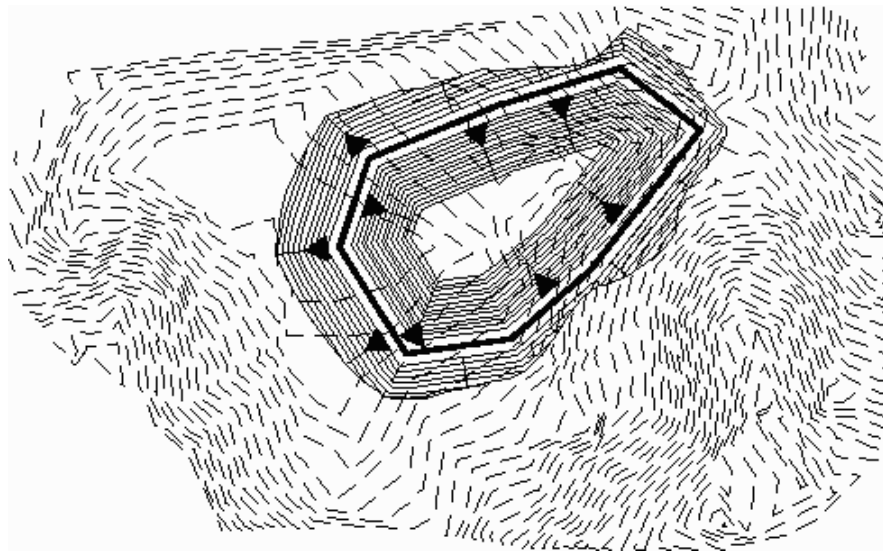
Enter the fill interior slope ratio <2.0>: *Enter*

Enter the cut interior slope ratio <2.0>: *Enter*

TOP OF DAM POLYLINE



Existing contours with top of dam polyline



Bench Pond showing Slope Direction Arrows and complete with contours

Range of existing elevations along dam top: 2033.75 to 2041.81

Enter the top of bank elevation <2033.75>: *press Enter to accept the default which is the lowest surface elevation along the perimeter*

Enter the pond bottom elevation: 2012.55

Method to specify storage elevations [<Automatic>/Interval/Manual]? *press Enter* If manual is selected the user can specify the elevation(s) to calculate.

Pond Report viewer that shows the earthwork volumes and stage-storage data

Adjust parameters and redesign pond [Yes/<No>]? *Y*

If yes is chosen for adjust, the following 7 prompts appear along with the report again:

Balance cut/fill [Yes/<No>]? *press Enter for No. Yes will adjust the pond elevation to balance the cut/fill earthwork.*

Enter the fill outslope ratio <2.00>: *2.5*

Enter the cut outslope ratio <2.00>: *2.5*

Enter the interior slope ratio <2.00>: *2.5*

Enter the top of dam width <10.00>: *press Enter*

Enter the top of bank elevation <2033.75>: *press Enter*

Enter the pond bottom elevation <2012.55>: *press Enter*

Offset top of dam polyline [Yes/<No>]? *Y*

If yes is chosen for offset, the following 2 prompts appear:

Offset inwards or outwards [<In>/Out]? *press Enter*

Enter the amount to offset: *5*

Method to specify storage elevations [<Automatic>/Interval/Manual]? *press Enter*

Pond Report viewer that shows the earthwork volumes and stage-storage data

Adjust parameters and redesign pond [Yes/<No>]? *N*

Write stage-storage file (Yes/<No>)? *press Enter.* This option creates a .cap file to use with Draw Stage-Storage Curve and to use for hydrograph routing.

Update target surface file [Yes/<No>]? **press Enter.** This option is available when the target surface is a triangulation file. The bench pond design will be merged into the target surface to update the triangulation file.

Write final surface to grid file [Yes/<No>]? *press Enter*

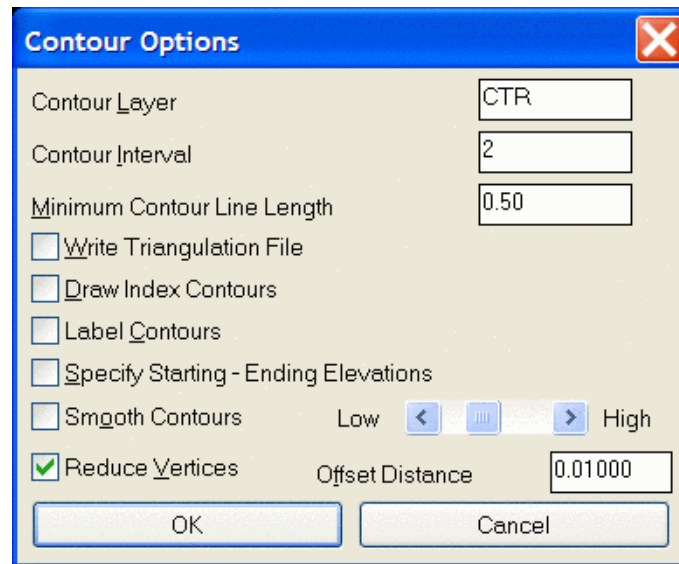
Trim existing contours inside pond perimeter [Yes/<No>]? *Y*

If yes is chosen, the following 2 prompts appear:

Retain trimmed polyline segments [Yes/<No>]? *Y*

Specify layer name for trimmed segments [Yes/<No>]? *press Enter*

Contour the pond [**<Yes>/No**]? *press Enter*



Refer to *Triangulate & Contour* section of the manual for a full explanation of Contour Options settings.

Bench Pond Design Report

Top of dam elevation: 40.0000
Bottom of pond elevation: 28.0000
Top of dam width: 10.0000
Cut slope percent grade: 40.00, slope ratio: 2.50
Fill slope percent grade: 40.00, slope ratio: 2.50
Interior slope percent grade: 50.00, slope ratio: 2.00

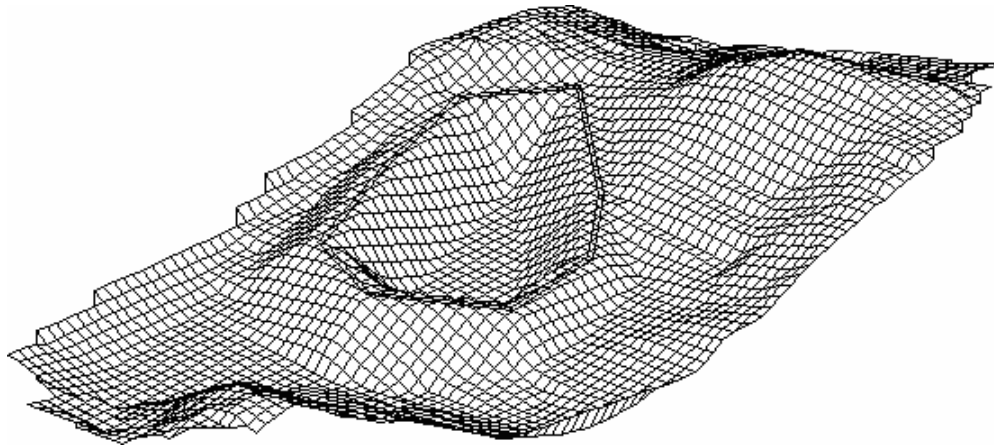
Lower left grid corner: 186395.20,57620.23
Upper right grid corner: 186803.82,57872.93
X grid resolution: 50, Y grid resolution: 50
X grid cell size: 8.17, Y grid cell size: 5.05

Pond EarthWork Volumes

Total fill: 2087.624 C.Y., 56365.84440 C.F.
Total cut: 772.791 C.Y., 20865.34961 C.F.

Pond Storage Volumes

Water Elev: 30.00, Pond Storage: 179.887 C.Y., 4856.94921 C.F.
Water Elev: 32.00, Pond Storage: 453.708 C.Y., 12250.12126 C.F.
Water Elev: 34.00, Pond Storage: 817.981 C.Y., 22085.48186 C.F.
Water Elev: 36.00, Pond Storage: 1321.035 C.Y., 35667.93750 C.F.
Water Elev: 38.00, Pond Storage: 2001.216 C.Y., 54032.83340 C.F.



3D view of Bench Pond

Pulldown Menu Location: Surface > Design Pond in Civil, Structure in Hydro

Keyboard Command: bpond

Prerequisite: Closed polyline that defines top of dam and surface entities

Edit Bench Pond

This command is used in conjunction with *Design Bench Pond*. It allows you to modify an existing pond design that has been generated with Design Bench Pond. The original design criteria of the pond can be modified and the routine will automatically re-calculate volumes, re-draw contours, disturbed limits, tin lines, etc. Possible modifications to the pond include cut/fill slopes, pond elevations, x-y position (the pond can be moved to a new location), horizontal offset, a single pond vertex can be moved to a new location, a new target surface can be specified, and finally, the pond and all of its elements can be deleted. Once the new design parameters are set, the routine will regenerate the pond based on the new parameters when the Process button is selected. The standard report viewer opens to display the new volume calculation and other design info.

Edit Bench Pond requires a target surface to project to, which is typically the surface designated as the Source of Slope Target Surface Model in the Design Bench Pond command. In Design Bench Pond, when prompted to "Update Surface File" select Yes.

The Edit Bench Pond command starts with a prompt to pick a pond polyline which can be any of the side slope or perimeter polylines created by the Design Bench Pond command. Another way to start Edit Bench Pond is to double-click on one of the pond polylines.

When the command starts, a dialog is docked on the left side of the drawing window. This dialog allows you to edit the pond while still being able to run other commands.

Fill Out Slope:	<input type="text" value="2.00"/>
Cut Out Slope:	<input type="text" value="2.00"/>
Fill Interior Slope:	<input type="text" value="2.00"/>
Cut Interior Slope:	<input type="text" value="2.00"/>
Top Elevation:	<input type="text" value="1040.0000"/>
Bottom Elevation:	<input type="text" value="1020.0000"/>
Pond Earthworks:	
Fill:	188145.4 C.Y.
Cut:	291438.8 C.Y.
Storage (AcreFt/C.Y./Gallons):	
<input type="text" value="83.50 134707.6 27207436.3 at 104"/>	
Surface:	Quads3b <input type="button" value="Set"/>
<input type="button" value="Move Pond"/>	<input type="button" value="Move Vertex"/>
<input type="button" value="Add Vertex"/>	<input type="button" value="Remove Vertex"/>
<input type="button" value="Offset"/>	<input type="button" value="Rotate Pond"/>
<input type="button" value="Balance"/>	<input type="button" value="Delete"/>
<input type="button" value="Set Storage"/>	<input type="button" value="Stage-Storage"/>
<input type="button" value="Report"/>	<input type="button" value="Exit"/> <input type="button" value="Help"/>
<input type="button" value="Process"/>	

Fill Out Slope: displays the original fill slope criteria, select to edit to a new value.

Cut Out Slope: displays the original cut slope criteria, select to edit to a new value.

Fill Interior Slope: displays the pond interior slope for fill condition, select to edit to a new value.

Cut Interior Slope: displays the pond interior slope for cut condition, select to edit to a new value.

Top Elevation: enter the top elevation of the pond.

Bottom Elevation: enter the bottom elevation of the pond.

Pond Earthworks: displays the cut/fill earthworks and storage volume calculated with the last use of either Design Bench Pond or Edit Bench Pond.

Surface: Displays the target surface, pick Set button to change.

Move Pond: Pick to move the pond a new location on the site. The user is prompted at the command line to "Pick start point for translation" and then "Pick end point for translation". The pond is moved to the new location, with contours, tin lines, disturbed area and volumes regenerated on the fly.

Move Vertex: Pick to move a vertex on the top of pond perimeter. The user is prompted to screen select the vertex to edit and place it in a new location.

Add Vertex: Pick the position for a new point to add into the pond perimeter.

Remove Vertex: Select a point on the pond perimeter to remove. **Offset:** Pick to apply a horizontal offset of the top of pond perimeter polyline. Enter the amount of offset desired in the field and then specify whether it is to go in

or out.

Rotate Pond: Pick to rotate the pond.

Balance: Pick to automatically adjust the pond elevation to balance the cut and fill volumes.

Delete: removes all of the pond design entities.

Set Storage: prompts for a target elevation and storage, and then resizes the pond to fit.

Stage-Storage: write a stage-storage .cap file for the current design.

Report: Pick to generate a cut/fill volume and stage-storage report.

Process: proceeds with the re-design of the pond using the current criteria.

Pulldown Menu Location: Surface->Design Pond

Keyboard Command: edit_bpond

Prerequisite: A Bench Pond generated with the *Design Bench Pond* command, and the surface model file used as a target surface.

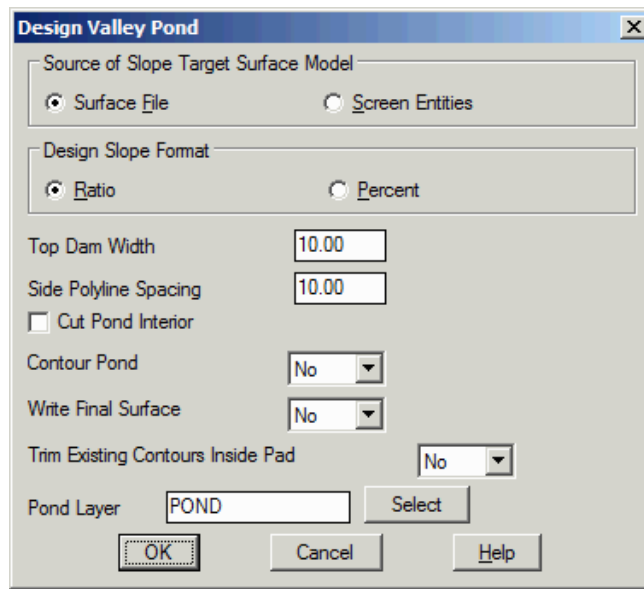
Design Valley Pond

This feature will design a valley pond, essentially by creating a dam across a low area, beginning with a surface model file (.GRD, .TIN, .FLT), or screen entities representing the surface, and a 2D polyline that defines the top of dam. These components must be present before starting. If Screen Entities are used, the program internally creates a grid mesh from the surface entities (Contours, Tri-Mesh, 3D polys) found by doing a crossing selection of the grid limit.

The design is drawn as 3D polylines with an option to drawn contours on the pond, and the earthwork volumes and stage-storage volumes are calculated.

The Cut Pond Interior option has two methods for cutting volume from the pond interior. This cut will create more water storage. The trace method prompts you to define a 3D polyline by picking points starting at the dam and going around the pond counterclockwise. At each point you enter an elevation. The default is the current ground elevation and typically you would enter a lower elevation. Then you enter a cut slope and the program will cut from this perimeter polyline at the entered slope. The polyline method requires a pre-drawn closed polyline inside the pond. The program will ask for a depth to cut and a cut slope. Polyline is set to the current ground elevation minus the cut depth. Then the program cuts out from the polyline to the original ground at the cut slope.

Prompts



For Screen Entities method:

Pick Lower Left limit of pond disturbed area: *pick lower left*

Pick Upper Right limit of pond disturbed area: *pick upper right* Be sure to pick these limits well beyond the area of the top of dam polyline in order to make room for the outslopes.

Make 3D Grid File Dialog

After selecting the limits of the disturbed area the program will generate a 3D grid that represents the surface. Specify the grid resolution desired and select OK.

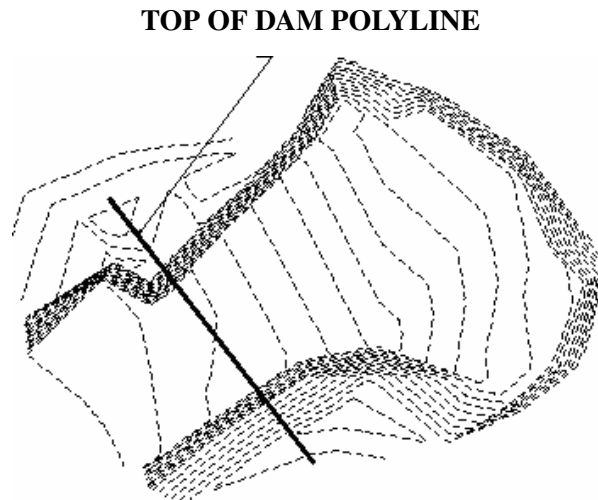
Pick the top of dam polyline: *select a 2D polyline*

Pick a point within the pond: *pick a point*

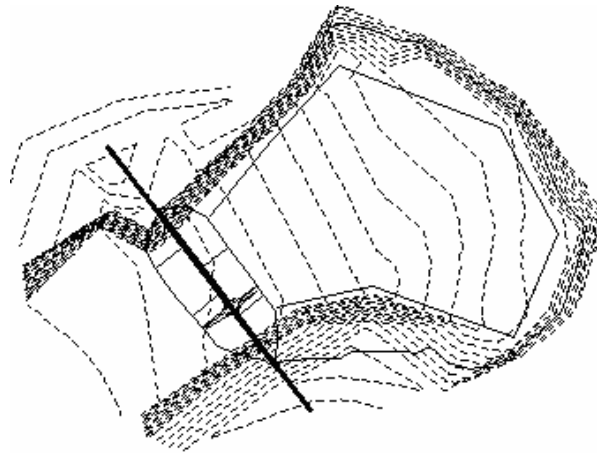
Enter the fill outslope ratio <2.0>: *2.5 Enter*

Enter the cut outslope ratio <2.0>: *2.5 Enter*

Enter the top of dam elevation: *90 Enter*



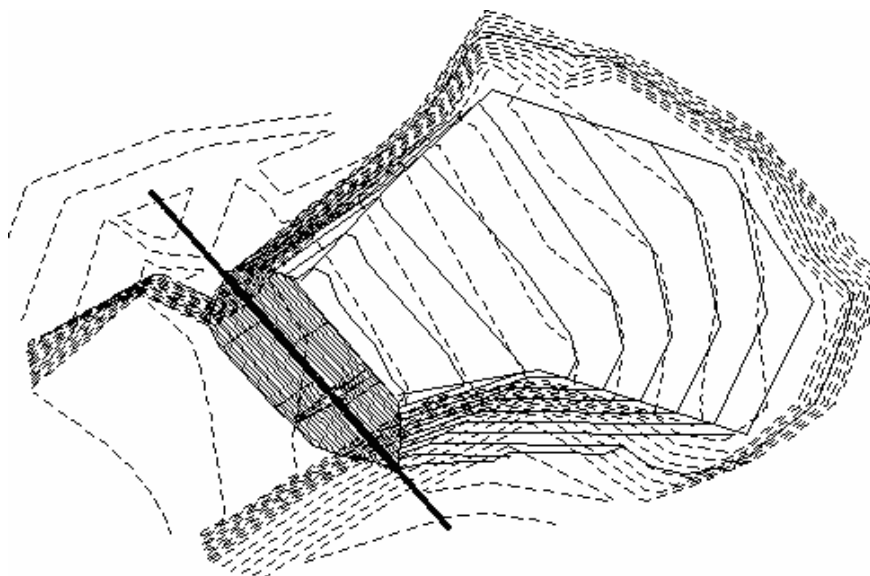
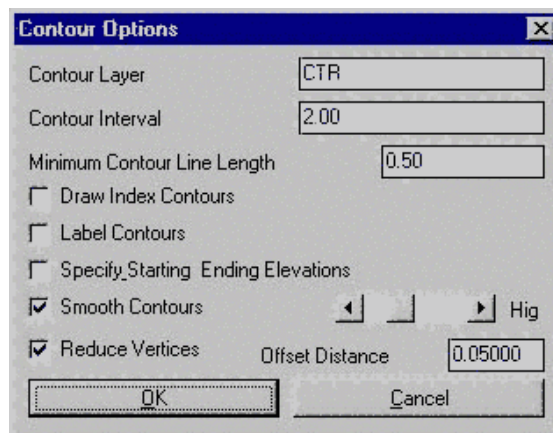
Existing contours with top of dam polyline



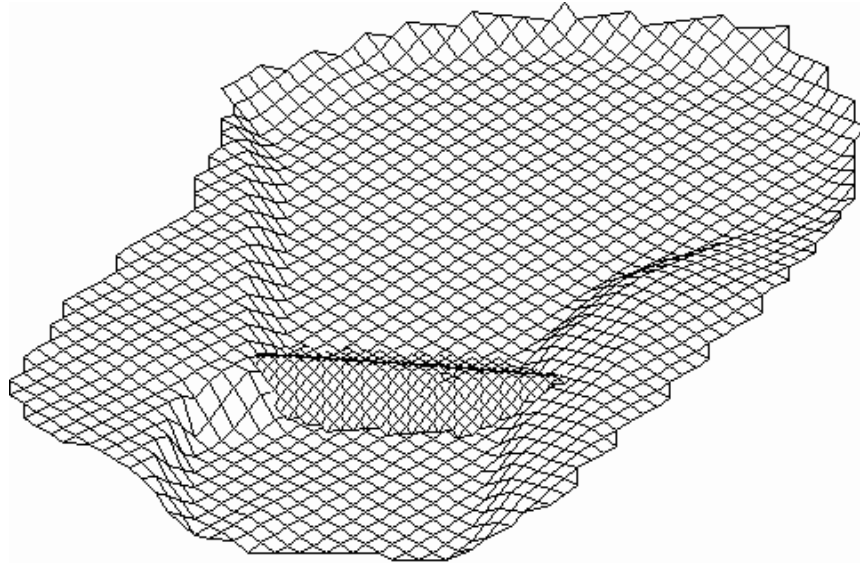
Valley Pond showing Emergency Spillway

Choose method to specify storage elevations (<Automatic>/Manual)? *Enter* If manual is selected the user can specify the elevation(s) to calculate.

Adjust parameters and redesign pond (Yes/<No>)? *Enter* If yes, the user will be able to enter new slope ratios and dam widths etc...



Valley Pond Design complete with contours



3D view of Valley Pond

Pulldown Menu Location: Surface > Design Pond in Civil, Structure in Hydro

Keyboard Command: vpond

Prerequisite: Polyline that defines top of dam

Draw Triangular Mesh

This command draws a triangulation (.flt or .tin) file as either Surface Object, 3D LINES or 3DFACEs. Since 3DFACE entities can be shaded within the *3D Viewer Window* or *3D Surface FlyOver*, or with the AutoCAD *3D Orbit* command, this is an excellent tool for visual surface inspection. 3D Lines cannot be shaded.

The Surface Object is a custom object for displaying the TIN in the drawing. This custom object also supports the EndPoint snap. The main advantage of the Surface Object is with displaying large TIN files because the Surface Object is more efficient than drawing individual 3D Faces or Lines. The Surface Object requires Carlson to be loaded to display.

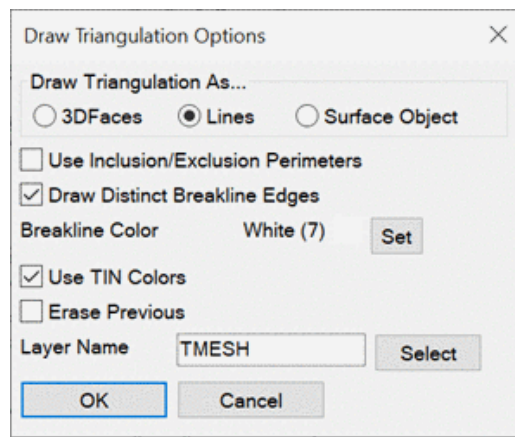
In the options dialog, choose the method for how to draw. The Use Inclusion/Exclusion Perimeters will prompt for selecting closed polylines to limit the area where to draw. The Use TIN Colors will color the entities using the colors for triangles stored in the TIN file. Colors can be stored in the TIN files using commands like Triangulation File Utilities, Road Network and Color Surface By Layer. The Erase Previous will erase the existing entities in case the TIN file has already been drawn. For Lines, there is an option to use a different colors for edges in the TIN that are from breaklines.

Triangulation (.flt or .tin) files can be created by routines such as *Triangulate & Contour*.

Prompts

Select Triangulation File to Draw

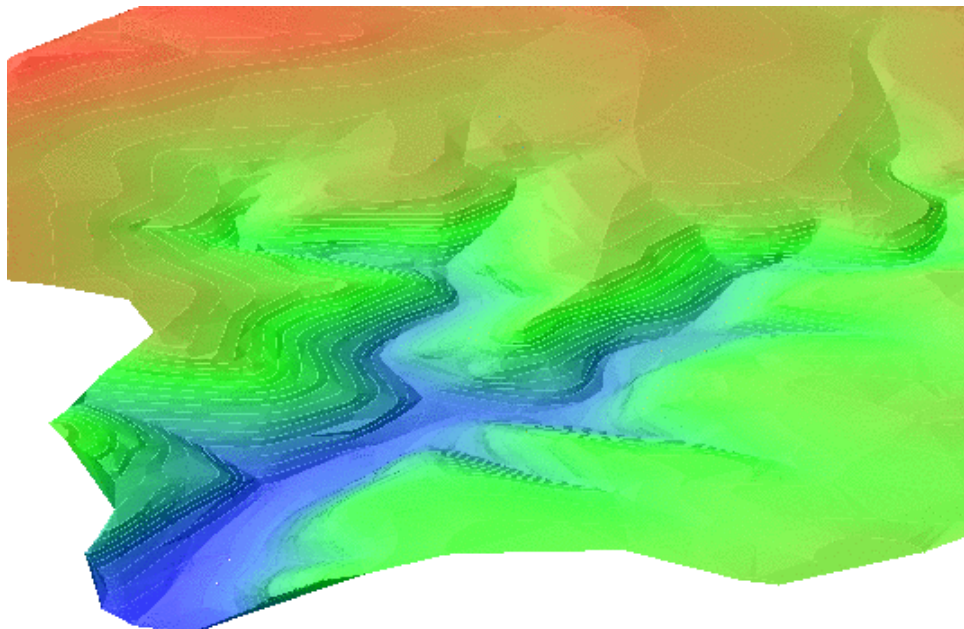
Choose a triangulation (.flt or .tin) file from the file selection dialog. You are then prompted for options:



If using Inclusion/Exclusion Perimeters, you will be prompted to select them as the routine executes.

Loading edges...

Loaded 198 points and 234 edges



This Triangulation mesh was drawn as 3DFaces with the Draw Triangular Mesh command, and then colored by elevation within 3D Viewer Window

Pulldown Menu Location: Surface >> Draw Surface

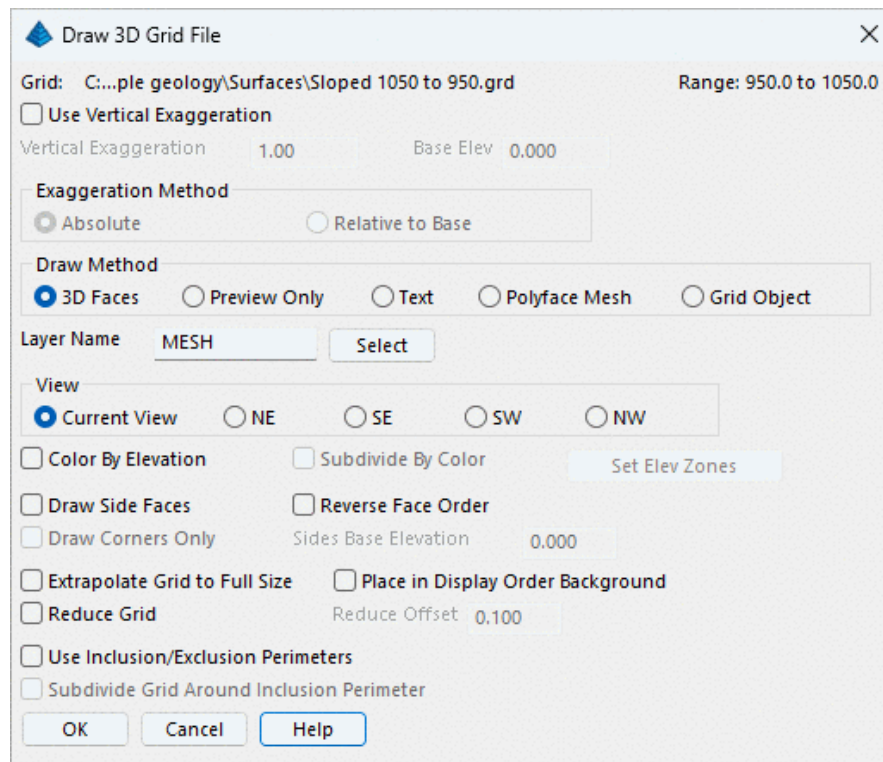
Keyboard Command: drawtri

Prerequisite: A triangulation (.flt or .tin) file

Draw 3D Grid File

This command draws the 3D grid mesh of the chosen grid (.GRD) file. Each grid cell can be drawn as a Surface Object, 3D Face entities, Polyface mesh, Text or temporary lines. 3D Faces and Polyface Meshes can be viewed/used in the following commands: *3D Viewer Window*, *Viewpoint 3D*, *Hide*, *Shade*, *3D Surface FlyOver*, and

The Surface Object is a custom object for displaying the grid in the drawing. This custom object also supports the EndPoint snap. The main advantage of the Surface Object is with displaying large grid files because the Surface Object is more efficient than drawing individual 3D Faces. The Surface Object requires Carlson to be loaded to display.



If **Use Vertical Exaggeration** is checked, grid elevations are multiplied by the value specified.

Exaggeration Method specifies whether to use an *Absolute* exaggeration method or *Relative to Base*, which uses the specified base elevation.

Specify the type of entities to draw in **Draw Method**. 3D Faces are described above. The Preview Only option draws the grid using temporary vectors. This method provides a much faster way to view the grid. However these temporary vectors are erased when the viewport is modified. This means as soon as you execute zoom, redraw, regen or plot, this grid will disappear. You can quickly redraw the grid by typing in VG for View Grid at the command prompt. Polyface Mesh is similar to 3D Faces except it is a single entity. The Text option will label the grid elevation at the grid corner. The text is placed center justified over the grid corner. To reduce clutter, there is an option to skip rows and columns.

Specify the layer for the grid entities in **Layer Name**.

Specify the initial viewing direction in **View**.

When **Color by Elevation** is checked, the grid will be colored based on a table of user-defined elevation ranges and the assigned colors. There is also an option to subdivide the grid cells at the color zone transitions, similar to the Elevation Zone Analysis command. Click the Set Elev Zones button to define ranges and colors.

When **Draw Side Faces** is checked, the program will draw vertical faces around the perimeter of the grid. The side

faces will be drawn vertically from the grid perimeter to the Sides Base Elevation. You may optionally specify the Sides Base Elevation, it defaults to 0.00.

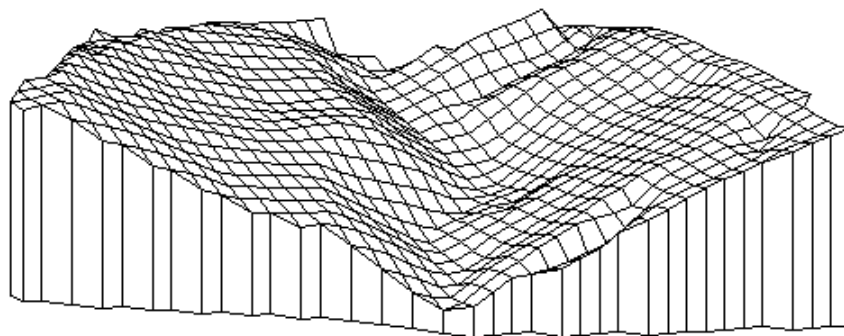
When checked, **Reverse Face Order** changes the direction of the points for a grid cell from clockwise to counter-clockwise. The order applies to shading the grid cell in 3D render viewers such as the *3D Viewer Window* command. The grid cell will only appear shaded when viewing the grid cell from the clockwise side. Viewing from the other side will show a wire frame. The default is to show the shaded side from the top-down view. This option allows you to draw the grid so that the underside of the grid is shaded.

When checked, **Draw Corners Only** will draw the side lines only at the grid corners. Otherwise side lines are drawn down each perimeter grid cell.

When checked, **Extrapolate Grid to Full Size** draws the entire rectangular surface of the grid.

When **Use Inclusion/Exclusion Perimeters** is checked, it allows you to select inclusion and exclusion areas. Only grid cells inside the inclusion polylines will be drawn. Grid cells inside the exclusion polylines will not be drawn.

When checked, **Subdivide Grid Around Inclusion Perimeter** subdivides grid cells that are partially inside and outside the perimeter into smaller resolution grid cells.



Drawn grid file using inclusion perimeter and side faces option
viewed with Viewpoint 3D

Pulldown Menu Location: Surface >> Draw Surface

Keyboard Command: plotgrid

Prerequisite: a grid (.GRD) File

Quick Draw Surface

This command draws lines for the edges of a selected triangulation model. The purpose of this routine is to quickly get a picture of the triangulation connections. The lines are drawn as temporary graphics that get cleared when the display is redrawn. To create triangulation entities, use other routines such as Draw Triangular Mesh.

Prompts

Select Triangulation File

Choose the surface to draw

Pulldown Menu Location: Surface > Draw Surface

Keyboard Command: quicksurf

Prerequisite: a surface file (.TIN, .FLT)

Quick Contours

This command draws contours for a selected triangulation model at a given contour interval. The purpose of this routine is to quickly get a picture of the surface model from the contours. The contours are drawn as temporary graphics that get cleared when the display is redrawn. To create contour entities, use other routines such as Contour From TIN File or Triangulate & Contour.

Prompts

Select Triangulation File

Choose the surface to contour

Contour interval <5.00>: *press Enter*

Pulldown Menu Location: Surface > Draw Surface

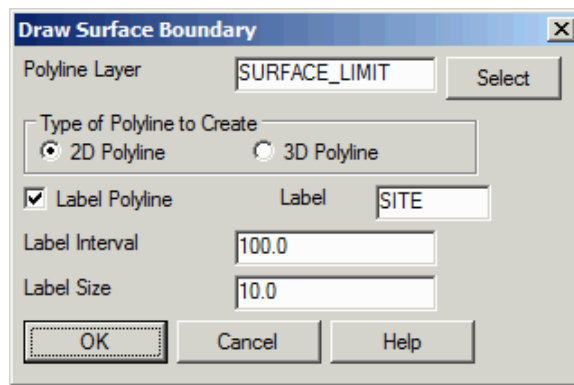
Keyboard Command: quickctr

Prerequisite: a surface file (.TIN, .FLT)

Draw Surface Boundary

This command draws the exterior perimeter of a triangulation or grid surface as a polyline. This is a simple way to show the size, shape and location of a surface without adding a lot of drawing entities and file size to the drawing.

The program prompts for the layer of the polyline to create and the type of polyline. A 2D polyline is drawn at zero elevation. A 3D polyline uses the surface elevations. There is an option for Label Polyline which adds text labels along the polyline with the specified name, size and interval. This label can be helpful to identify different surface boundaries.



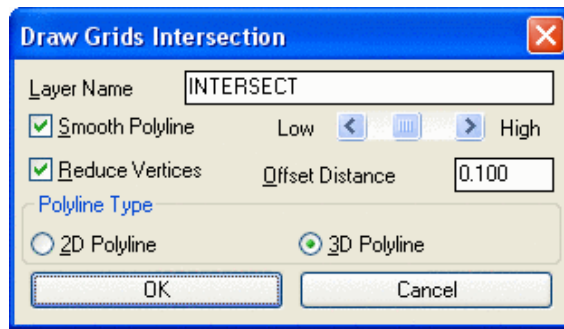
Pulldown Menu Location: Surface >> Draw Surface

Keyboard Command: grdlimit

Prerequisite: a surface file (.GRD, .TIN, .FLT)

Draw Surface Intersection

This command draws a 2D or 3D polyline at the intersection of two surfaces. In addition to this choice, the dialog also allows for the specification of the layer to draw the polylines on, whether or not to smooth them, and whether or not to reduce vertices.



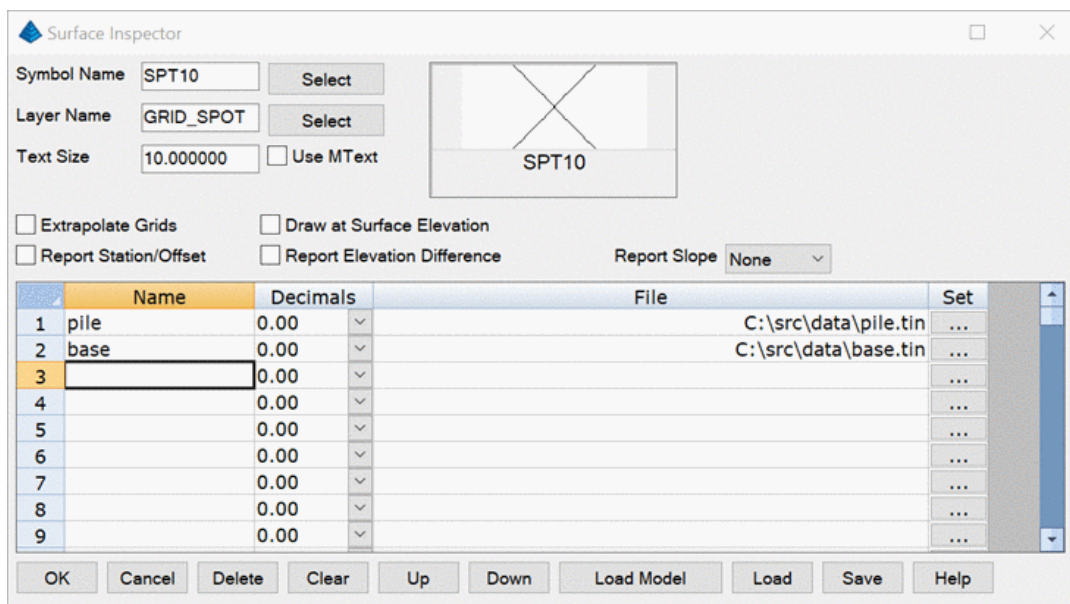
Pulldown Menu Location: Surface >> Draw Surface

Keyboard Command: grdcross

Prerequisite: two surface files (.GRD, .TIN, .FLT)

Surface Inspector

This command allows you to report and optionally label elevations for selected surface files. You can simultaneously analyze multiple surface files. Surface files can be either triangulation (.flt or .tin) files, grid (.grd) files, or any combination thereof. The following dialog opens when the command is initiated.



Symbol Name sets the symbol name to be plotted. Click **Select** to select the symbol from the symbol library.

Layer Name set the layer to use for the text labels.

Text Size sets the actual size (not scale factor) of the text label placed in the drawing. The **Use MText** controls whether to create the labels as MText or regular Text entities.

Turn the **Draw at Surface Elevation** toggle ON if you want the symbol to be located at the actual elevation of the surface.

The **Extrapolate Grids** option automatically extrapolates any unassigned grid nodes for grid surfaces.

Report Slope displays the slope of the surface along with the elevation.

Report Station/Offset prompts for a centerline and reports the station and offset in addition to the surface elevations.

When there are two surface files, the **Report Elevation Difference** option will add a third report field for the cut/fill between these two surfaces.

Name denotes the name that will be plotted when you label the elevation. The default value is the same as the name of the surface file, but you can change it.

Decimals individually sets the decimal elevation precision for each selected surface.

For **File**, either type in the surface name to use or press the **Select** button to choose the surface file from a browse window.

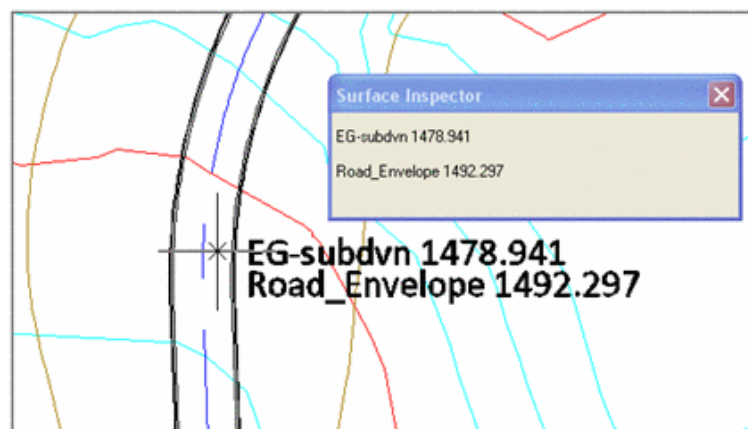
Load Model allows you to select grid files from a list of grids stored in a Geologic or Mining Model file. These files can be created in the Carlson Mining module.

Clear: Clears all values.

Load: Loads a Surface Inspector File (.SIF).

Save: Saves all settings to a Surface Inspector File (.SIF).

After you fill out the dialog box, click OK. Surface Inspector will load the surfaces and begin showing you real-time elevations for each surface as you move the cursor on the screen. If you pick a point or enter coordinates, the elevation will be labeled along with the surface name and selected symbol as shown below.



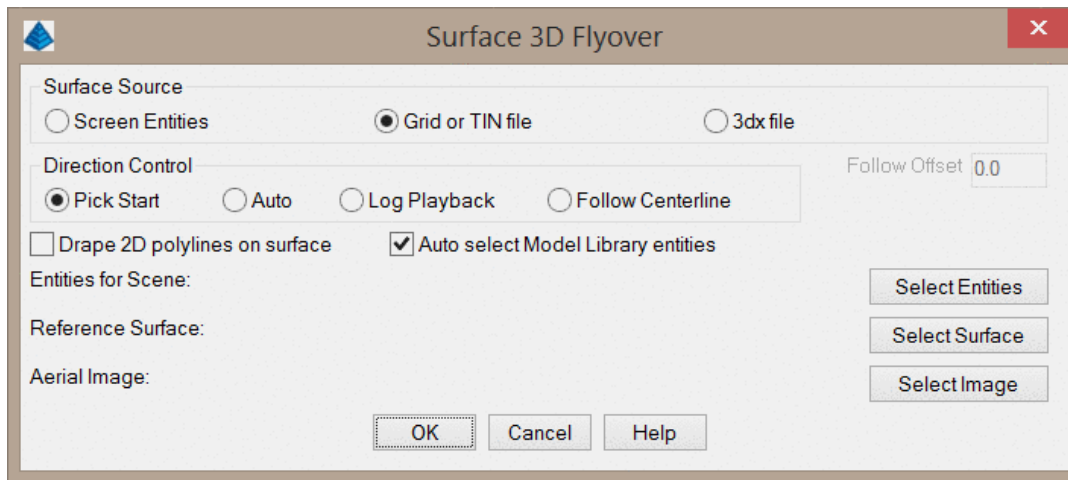
Pull-down Menu Location: Surface

Keyboard Command: surfvals

Prerequisite: Surface Model(s)

Surface 3D FlyOver

This command allows you to view a 3D surface in a simulated drive- or fly-over mode. You have the option of following a predefined path such as a road centerline or using a user-guided path (free flight). The surface to view can be defined with either screen entities, surface files, or both. The routine offers options for different types of surface shading, direction of travel, viewpoints, vehicles, reference surfaces, light position, color schemes, vertical exaggeration and more.



Surface Source: This option sets the source of the surface to view.

Screen Entities: With this option enabled, a surface will be generated from the selected 3D Faces. Other entities such as polylines will only be shown for reference. To select the screen entities, press the Select Entities button.

Grid or TIN file: With this option enabled, the surface is defined by either a triangulation file (.FLT or .TIN) or a grid file (.GRD). In addition to the surface file, screen entities may also be selected to be displayed. You will be prompted to select the surface after clicking OK.

3DX File: This method loads the scene from a 3DX file which can be saved from a previous run of Surface 3D FlyOver. Other 3D viewer commands such as the 3D Viewer Window can also save the scene to a 3DX file.

Direction Control: This option controls the method of movement through the 3D view.

Pick Start: This option allows the user to randomly navigate the site, but a starting direction must be defined by picking two points on the screen. Once travel starts, the direction can be controlled with the arrow keys on the keyboard.

Auto: This option also allows the user to randomly navigate the site, but a start point is automatically selected.

Log Playback: This option uses vehicle log files from Carlson Machine Control to animate the position of vehicles in the scene and show the surface updates.

Follow Centerline: This method prompts for a centerline file (.CL) or alignment polyline to follow. The camera position of the animation will be limited to follow this alignment. The **Follow Offset** value shifts the view position to the right (positive value) or left (negative value) of the centerline to simulate driving down one lane of the road.

Drape 2D polylines on surface: This option will drape 2D polylines onto the surface regardless of their original elevation.

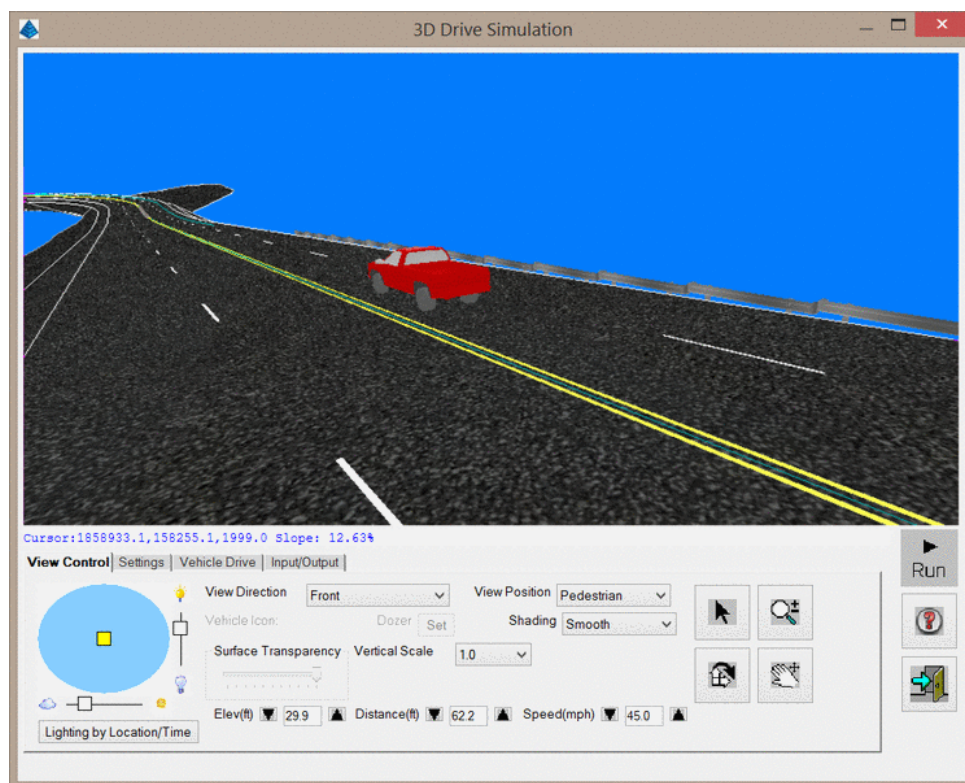
Auto select Model Library entities: When enabled, this option will automatically detect 2D symbols in the drawing and display them as 3D symbols as specified in the 3D Viewer/Model Library command.

Select Entities: This button allows you to select drawing entities to display in the 3D window.

Select Surface (Reference Surface): This button allows you to select a reference surface file. This will allow you to report the cut/fill difference between the given surfaces at the current position. This option is only available when the Surface Source is set to Grid or TIN file. Note that this reference surface will not actually be displayed in the viewer.

Select Image (Aerial Image): This button allows you to drape an aerial image on the surface. This image must be a geo-referenced TIF image that overlaps the area of the surface model. See the Create World File by Image in Drawing for additional information.

After clicking OK, the 3D graphics window will appear.



The 3D window has several mouse and keyboard controls that can assist with navigating the scene. These are listed below:

- Hold left-mouse button: Rotates/pans/zooms the scene based on the current mode. You can change the mode by clicking one of the control icons.
- Hold right-mouse button: Zooms in/out on the scene
- Hold middle-mouse button: Pans the scene
- W key: Zooms in
- S key: Zooms out

- A key: Pans left
- D key: Pans right
- Q key: Pans up
- Z key: Pans down
- E key: rotates down
- C key: rotates up
- X key: rotates left
- V key: rotates right

View Direction: This option determines the direction that you look "out of" the vehicle based on the direction of travel. This setting does not change the direction of travel. There are six different view directions: Front, Back, Left, Right, Free Rotate and Detached. For Free Rotate, you can use the view controls to position the view relative to the vehicle and hold this position relative to the vehicle when you Run the animation. For the Detached mode, the view is not connected to the vehicle during the animation run which allows you to control the view point independently as the vehicle runs.

View Position: This option determines the position of the viewpoint relative to the vehicle. There are three different view positions:

1. Driver - The Driver position puts you inside of the simulated vehicle. **NOTE:** The Vehicle Icon option is not available when Driver is used.
2. Pedestrian - The Pedestrian position puts you behind and above the vehicle when view direction is set to front, above and to the left when the view direction is right, and so on.
3. Bird - The Bird position puts you further behind and higher above the selected vehicle.

Vehicle Icon: This option determines the type of vehicle to be used in the display. There are many options available, including dozers, sport-utility vehicles, emergency vehicles and others as setup in the 3D Model Library command. You may turn off the vehicle icon on the Settings tab.

Shading: This option determines the type of shading to be applied to the surface when the surface source is from a file. This option is not active when the surface is defined by screen entities. There are several shading options, including:

- None - Does not produce shading.
- Flat - Uses one color per 3D Face.
- Smooth - Blends colors together.
- Elevation - Generates colors based on the vertical position of the surface entities.
- Normal - 3D Faces are colored according the slope and slope direction of the surface.
- Cut/Fill - Generates colors based on depth of cut or height of fill between two surfaces. This option is only available when the Surface Source is defined by a File and a Reference Surface has been specified.
 - Cut is denoted by hues of red.
 - Fill is denoted by hues of blue.

Surface Transparency: This slider bar controls the transparency of the surface. Dragging the slider to the left will make the surface more transparent.

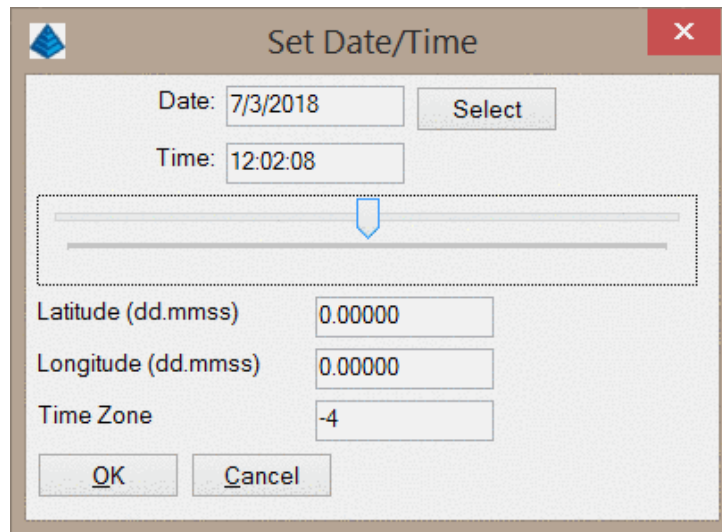
Vertical Scale: This option allows the user to specify a vertical exaggeration factor to aid in viewing relatively flat surfaces.

Elev: This value determines the height of the viewer vantage point above the surface. Clicking the up arrow will elevate further from the surface; clicking the down arrow will take you closer to the surface.

Distance: This value determines the horizontal distance from the viewers vantage point (behind) to the actual focal point on the surface. Clicking the Up arrow beside the control will increase the distance from the focal point; clicking the Down arrow will decrease the distance.

Speed: This value determines the rate of travel across the surface in miles per hour. Clicking the Up arrow beside the control will increase speed; clicking the Down arrow will decrease speed.

Lighting by Location/Time: This button will display the below dialog, which allows you to set the lighting as it would appear at a specific location at a specific time.



Date/Time: These values set the date and time for the lighting. The **Select** button will allow you to pick the date from a calendar. The horizontal slider below the time will allow you to set the time of day.

Latitude/Longitude: These values set the location of the viewpoint

Time Zone: This value adjusts for the various time zones. Values should be integers representing GMT adjustments. For example, the Eastern Standard Time (EST) in the USA will use a value of -5.

Control



Action

This control represents the position of the sun in plan view. If the yellow square is in the center of the blue circle, the sun is in a zenith. If the yellow square is near the edge of the circle, the sun is near the horizon. To move the yellow square, simply drag it to a new location, or click on the new location. The slider bars on the sides control the intensity and brightness of the display.



Switches to Pick Mode.



Switch to Dynamic Zoom mode. Click and drag to zoom in and out.



Switches to Rotation mode. When the cursor is placed near the outer edge of the view, a "Z" cursor is presented that permits rotation around the Z-axis. When the cursor is placed further into the interior of the view, an "X,Y" cursor is presented that permits the tilt angle of the view to be adjusted.



Switches to Pan mode. Click and drag to pan.



Starts (or "runs") the animation in the main window. While running, this button becomes the Stop button.



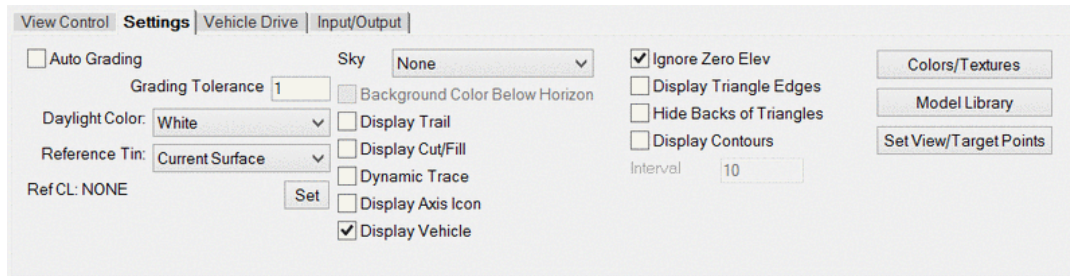
Stops the animation. When stopped, this button becomes the Run button.



Opens the help document you are currently reading.



Exits the command.



Auto Grading: This option automatically sets the Grading Tolerance based on the elevation difference with the reference TIN.

Grading Tolerance: When the cut/fill is less than this amount, the surface will be colored by the Daylight Color.

Daylight Color: The surface color for areas with cut/fill less than the Grading Tolerance.

Reference TIN: When using a Reference TIN from the first dialog, this option controls the direction of the cut to fill.

Ref CL: When a reference centerline has been selected, the current view position is reported beneath the main viewer window indicating the station/offset relative to the centerline.

Sky: This option controls the background image for the 3D viewer. By default, no background image will be displayed. When a background image is displayed, you can toggle the **Background Color Below Horizon** checkbox to hide the bottom half of the background image.

Display Trail: Displays the traveled route on the surface as a line.

Display Cut/Fill: This displays the amount of cut or fill at the location of the vehicle. This option is only available when a Reference TIN is used in the first setup dialog.

Dynamic Trace: When enabled, a position marker of the current view point is also displayed in the CAD/DWG window.

Display Axis Icon: Shows an x/y/z axis icon in the scene to help visualize the view angles.

Display Vehicle: This option toggles the visibility of the Vehicle in the 3D view.

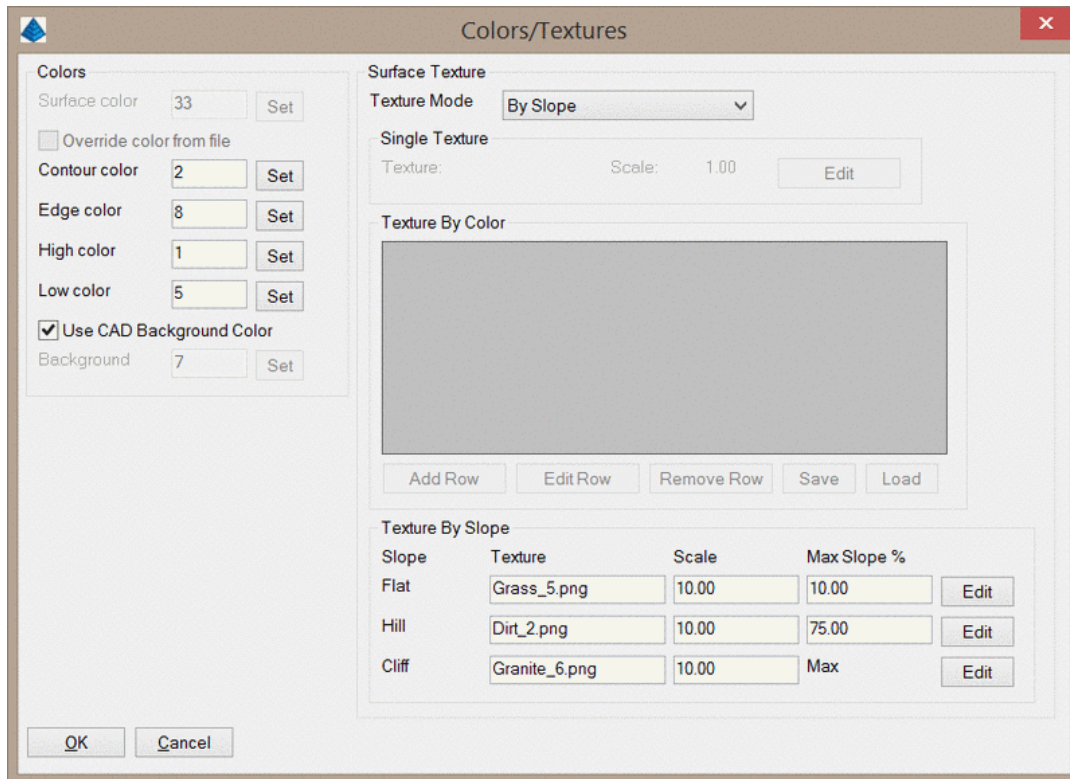
Ignore Zero Elevation: Ignores zero elevation entities in the scene.

Display Triangle Edges: Shows lines along the triangle edges in the scene.

Hide Backs of Triangles: This option toggles the display of the backs of 3D Faces. When enabled, even the edges of the triangles will be hidden.

Display Contours: Shows contours for the surface using the specified **Interval**.

Colors/Textures: This button opens the below dialog, which controls the colors and textures applied to the surface.



Surface Color: This setting will determine the color of surface entities when the shading mode is set to either flat or smooth. The color functions are only available when the Surface Source is defined by a File. If the Surface Source is defined by screen entities, color is determined by the properties of the screen entities.

Override color from file: When enabled, this option will override any color assignments and display the entire surface as a single color.

Contour/Edge color: These colors control the coloring for contours and triangle edges. Note that this does not affect the coloring of contour polylines that are loaded into the viewer, but only contours that are generated within the viewer.

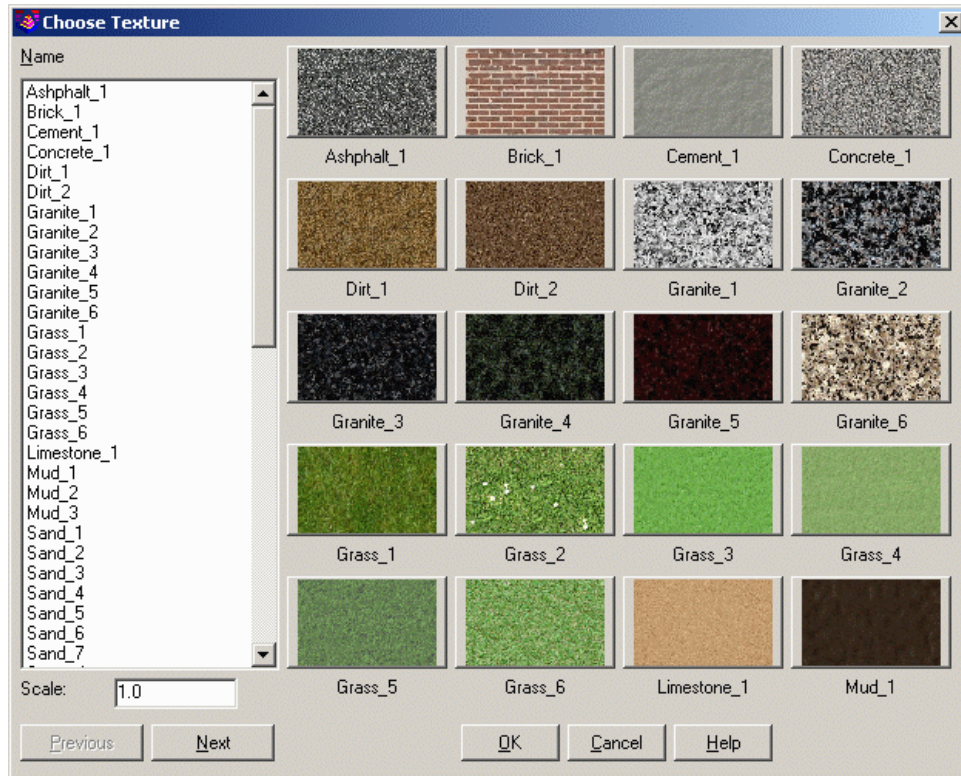
High/Low Color: When using the "Elevation" mode of shading, these values set the color gradient of surface entities that are in the higher elevation ranges of the surface.

Use CAD Background Color: When enabled, the background color of the 3D viewer will match the CAD background color. If this option is disabled, you can set a different background color.

Surface Texture: This function controls the texture for the surface. There are five Texture Modes.

None: No textures will be used.

Single Texture: Sets one texture for the entire surface. Clicking the **Edit** button will display the below dialog, allowing you to select a texture and a scale for the image.



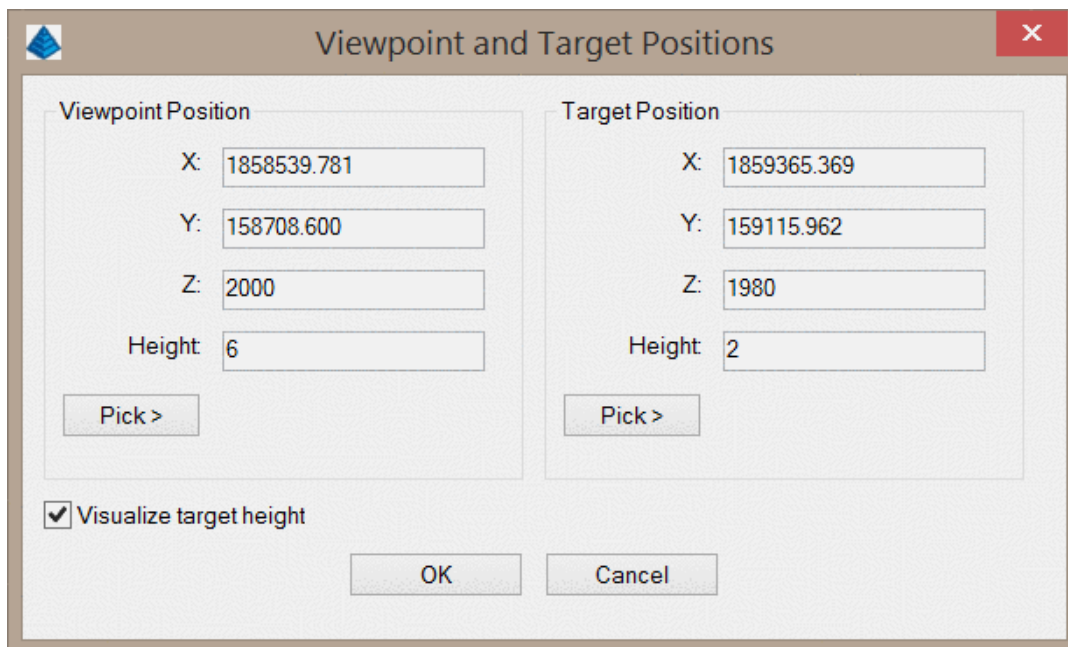
By Color: Sets a texture for each different color in the surface. The TIN file can have different colors for different areas by using the Road Network output TIN options to color by grade or by using Triangulation File Utilities. When this option is used, you will be able to add colors to the Texture by Color table.

By Color, Single as Default: This is the same as the By Color option except the Single Texture is used for any TIN colors not set in the lookup list.

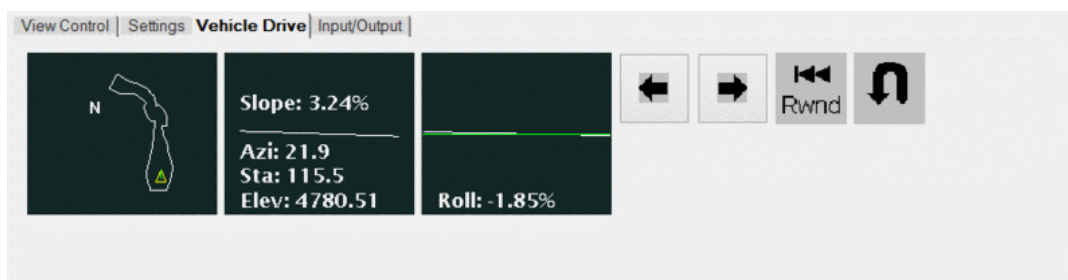
By Slope: This option sets the texture of the surface according to the slope of the surface. The cutoffs for the three slope groups are set with the **Max Slope %** value for that field. The textures for each slope group may be set by clicking the **Edit** button.

Model Library: This button opens the 3D Model Library.

Set View/Target Points: This button opens the below dialog, which lets you set the view position and target position by specifying the coordinates of each.



The positions can be entered in the edit boxes or you can use the respective **Pick >** button to pick a point in the drawing. The program will pick up the elevation of the surface for picked points and then the height above the position can be entered. For example, to check sight distance the view position could be a point on a road at the driver's eye height and the target position could be the object to check.



The window on the left shows the overall plan view and the location of the vehicle of the surface. The middle window displays the current station (when a polyline is used for direction control), elevation, slope (in relation to the direction of travel) and azimuth. The right window indicates the amount of roll or cross slope (in relation to the direction of travel) at your current position.

Control



Action

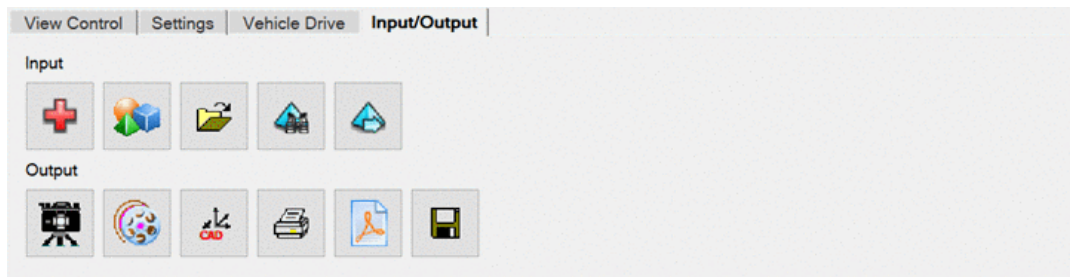
When using "Free Flight", this icon turns the direction of travel to the left.

When using "Free Flight", this icon turns the direction of travel to the right.

When using a 3D polyline for the travel direction, this button returns you back to the original starting position. The simulation must be in the Stopped mode for this to be active.

When using a 3D polyline for the travel direction, this button will reverse the direction of travel at the current position. The simulation must be in the Stopped mode for this to be active.

Additional Visualization Controls



Control



Action

Adds a 3D Model to the viewer.

Loads an MDL solid file.

Loads a 3DX file.

Sets the 3D Viewer viewpoint to match the current CAD viewpoint.

Reloads the data in the 3D Viewer.

Save a screenshot of the 3D Viewer.

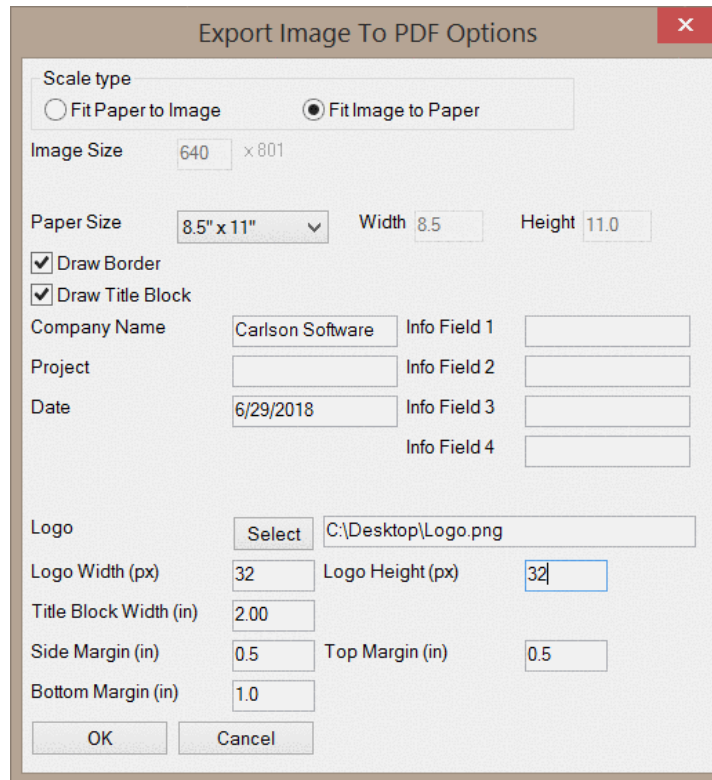
Records a movie of the 3D Viewer. The video will be saved to a .avi format. You will also be able to select from a variety of compression methods. After clicking OK, the recording will begin. You may click the icon again to stop the recording.

Sets the CAD viewpoint to match the current viewpoint of the 3D Viewer window.

Exports the 3D Viewer to a PDF report with project information. The default project information is set under Carlson Configure > General Settings. The dialog that appears for this command is shown below.

Exports the 3D Viewer to a 3D PDF. This function is further described in the to the 3D Viewer to 3D PDF help article. Note that you must have access to the CADNET module in order to use this command.

Save the entities in the 3D Viewer to a 3DX file.



Scale Type: This option controls how the image and the paper will be scaled to match one another. When the *Fit Paper to Image* option is selected, you will be able to manually set the **Image Size**.

Paper Size: This option sets the size of the PDF.

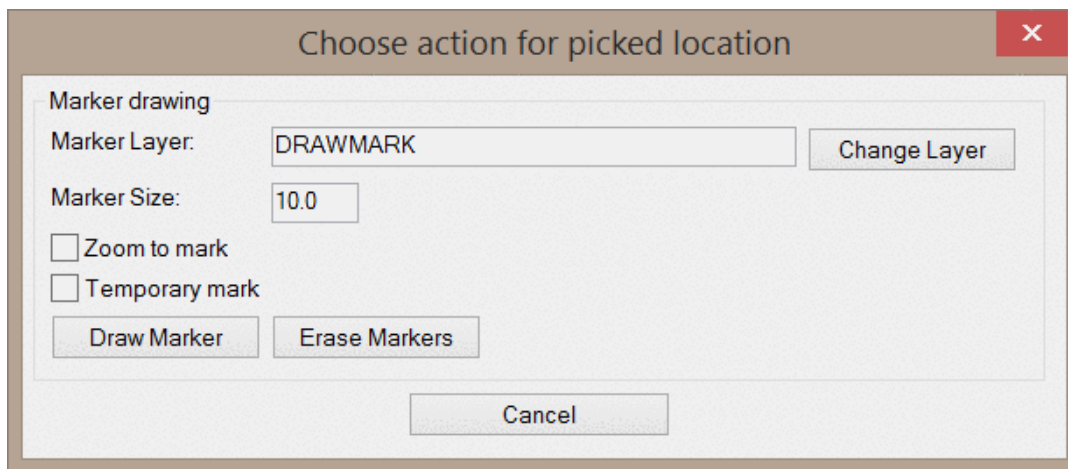
Draw Border: This option toggles the addition of a border around the PDF.

Draw Title Block: This option toggles the addition of a title block in the PDF. Information such as the **Company Name**, **Project**, **Date**, and **Logo** will be included in the title block. The logo may be selected by clicking the **Select** button and the dimensions may be specified in pixel dimensions.

Top/Side/Bottom Margin: These values set the margins around the PDF in inches.

This tab shows a project tree of the types of items in the scene. Right-click on items in the project tree to change the display properties.

When a location is "picked" through a double-click while in pick mode, the following dialog box displays with additional controls about how the location should be processed:



Marker Layer: Indicate a desired layer name or use the Change Layer button to select an existing layer for the position of an eventual marker.

Marker Size: Indicate the desired radius of a circle that will serve as the marker location.

Zoom to Mark: When enabled, the DWG will be zoomed to the location of the marker so it can be quickly found in the drawing.

Temporary Mark: When enabled, the marker will only be displayed in the drawing while the display remains static. As soon as a display update is summoned (via REDRAW, REGEN, ZOOM or PAN), the marker will no longer display on the screen. Disabling this toggle commits a CIRCLE entity into the drawing.

Draw Marker: Draws a marker into the drawing at the coordinates of the picked location and with the display characteristics defined by the Temporary Mark toggle.

Erase Markers: Erases any markers previously placed with the Draw Marker command.

- To entities within the DWG file (and optionally, Carlson surface files), use the 3D Viewer Window command.
- To quickly view just a surface file, use the Surface 3D Viewer command.

Pulldown Menu Location(s): Civil > Surface > 3D Views, Construction > Takeoff

Keyboard Command: flyby,(tk_flyby, tk_flyby2)

Prerequisite: A Carlson surface model (screen entities or file) and optionally a 3D polyline

Surface 3D Viewer

This command is identical to the 3D Viewer Window, except that this one loads a Carlson Grid GRD, TIN or FLT file. After the surface file is selected, the 3D viewer is shown.

On the Settings tab, there is an option to show contours.

On View Control, the larger pointer button puts the viewer in TIN edit mode. In this mode, the TIN Edit function list is enabled.

Swap TIN Edge: Double-click on a TIN edge to swap the connection points between two triangles.

Add Breakline: Double-click on a TIN point to set the first breakline point and then double-click on the second TIN point to create a breakline in the TIN.

Remove Point: Double-click on a TIN point to remove it from the surface.

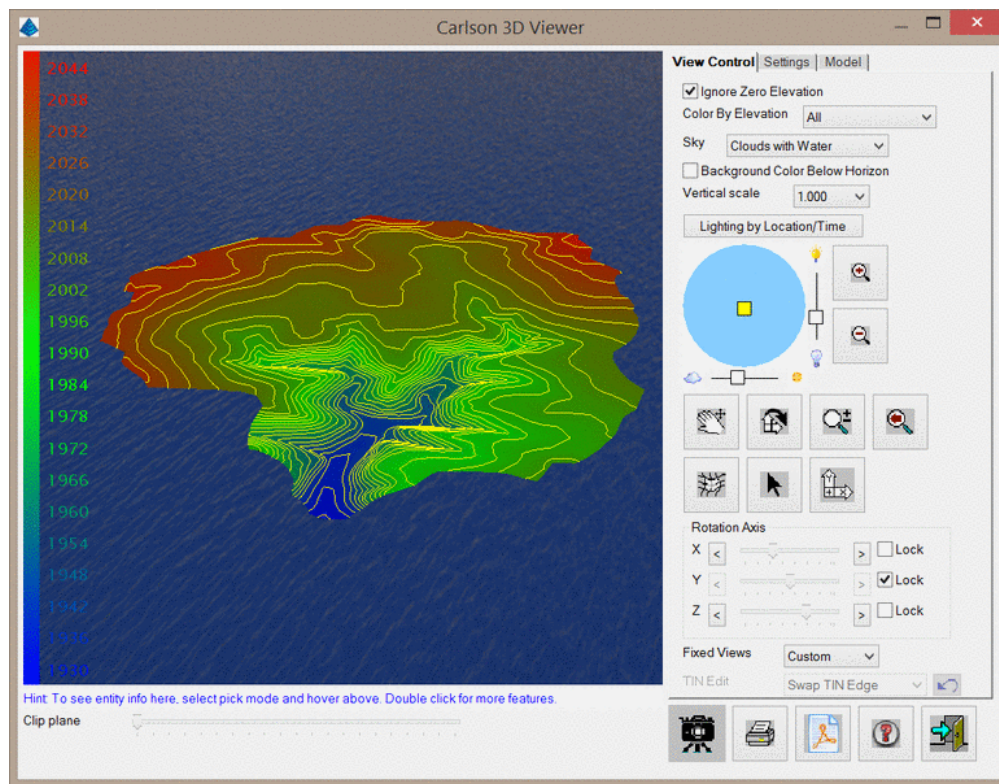
Remove Triangle: Double-click inside a triangle to remove it from the surface.

Set Point Elevation: Double-click on a TIN point and the program will prompt to enter the elevation.

Undo: The undo button next to the TIN Edit command list allows for undoing the last TIN edit.

When the TIN is edited, the program will prompt for whether to save the TIN when you exit the viewer.

The rest of the functions in this viewer are the same as documented in the 3D Viewer Window command.



- To entities within the DWG file (and optionally, Carlson surface files), use the 3D Viewer Window command.
- To visualize a site with animated vehicle controls, use the Surface 3D Fly-Over command.

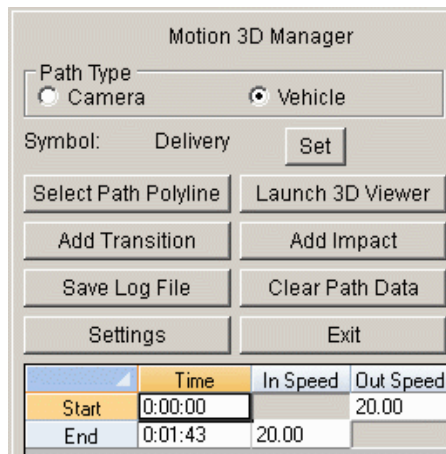
Pulldown Menu Location(s): View, Civil > Surface > 3D Views

Keyboard Command: cube_surface

Prerequisite: Carlson surface (TIN, FLT, GRD) to display

Motion 3D Viewer

This command creates moving vehicles for the Surface 3D FlyOver viewer. The vehicle paths are set by assigning a vehicle icon and speed to polylines. This command docks a dialog on the left of the drawing.



Use the Select Path Polyline function to tag a polyline as a path polyline. After selecting the polyline from the drawing, the dialog shows a spreadsheet of the transition points along the polyline. The speed is set at each point. When running the animation, the vehicle speed will be interpolated between these transition points as the vehicle moves along the polyline. At a minimum, there are two transition points for the start and end of the polyline.

The direction of the polyline in the drawing controls the direction of the movement in the viewer. When the polyline is 3D, the viewer will use the polyline elevations. When the polyline is 2D, the viewer will drape the polyline on the surface to set the elevations.

When the path polyline is selected, the Path Type chooses whether to use the polyline for the camera view position in the viewer or for drawing a vehicle icon.

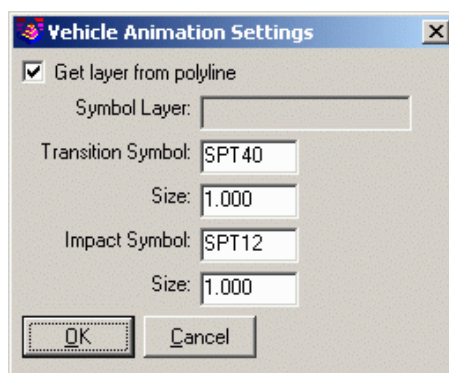
Use the Set button to select a Symbol for the vehicle icon. The list of available symbols comes from the 3D Model Library command.

Use the Add Transition function to add a speed transition point on the polyline path. The Add Impact creates a special transition point that represents where a vehicle had a collision. For this impact transition point, there is both a speed in and out to represent the sudden change in speed.

The Save Log File option creates a log playback file (.pos) which can be used by Surface 3D FlyOver.

The Clear Path Data function untags the path data from the selected polylines.

The Settings function show a dialog with settings for the temporary symbols that are shown in the drawing to indicate the transition points along the polyline.



When the path polylines are all set, use the Launch 3D Viewer command to display the scene with the vehicle

animation. The view controls are the same as the Surface 3D FlyOver command. Pick the Run button to put the vehicles in motion.

Pulldown Menu Location: Surface > 3D Views

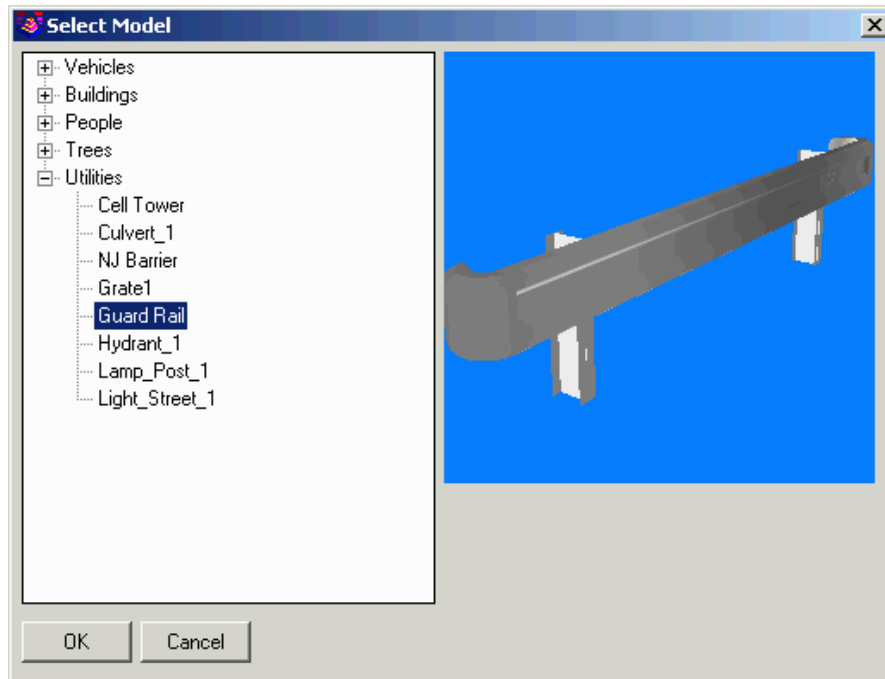
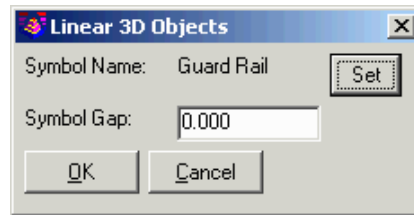
Keyboard Command: cube_motion

Prerequisite: Polyline

Linear 3D Object Polylines

These commands manage alignment polylines for 3D linear features such as fences or guard rails. These 3D features are used in the scene for 3D viewer commands such as Surface 3D FlyOver. When the polylines are 3D, the 3D viewer will use the polyline elevations. When the polylines are 2D, the 3D viewer will use the surface model to elevate the 3D objects.

This command assigns a 3D object to the selected polylines. This command prompts to select the polylines to assign and then prompts for the 3D object and Symbol Gap which controls the spacing of the 3D objects along the polyline. This gap is useful for 3D objects that occur at an interval such as street lights. For continuous objects like a fence, this gap should be set to zero. The available 3D objects to select from are defined in the 3D Model Library command.



This command creates a report of all the polylines in the drawing that are tagged as Linear 3D Object Polylines. The report includes the name of the 3D object, polyline layer and polyline starting coordinate. This report is useful to see how many and where these polylines are located in the drawing.

This command removes the 3D object assignments from the selected polylines.

Pulldown Menu Location: Surface > 3D Views

Keyboard Command: tag_linear3d, report_linear3d, untag_linear3d

Prerequisite: Polylines

Render, Shade, and Hide Commands

The *Render*, *Shade*, and *Hide* commands located under the Surface >> 3D Views menu simply execute the appropriate standard AutoCAD commands.

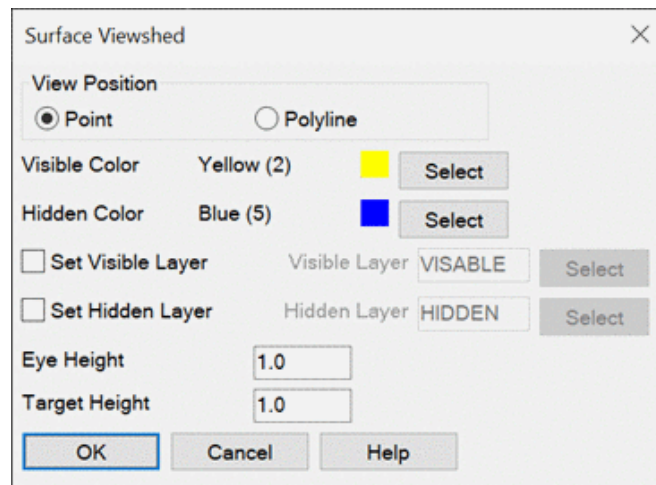
Pulldown Menu Location: Surface > 3D Views

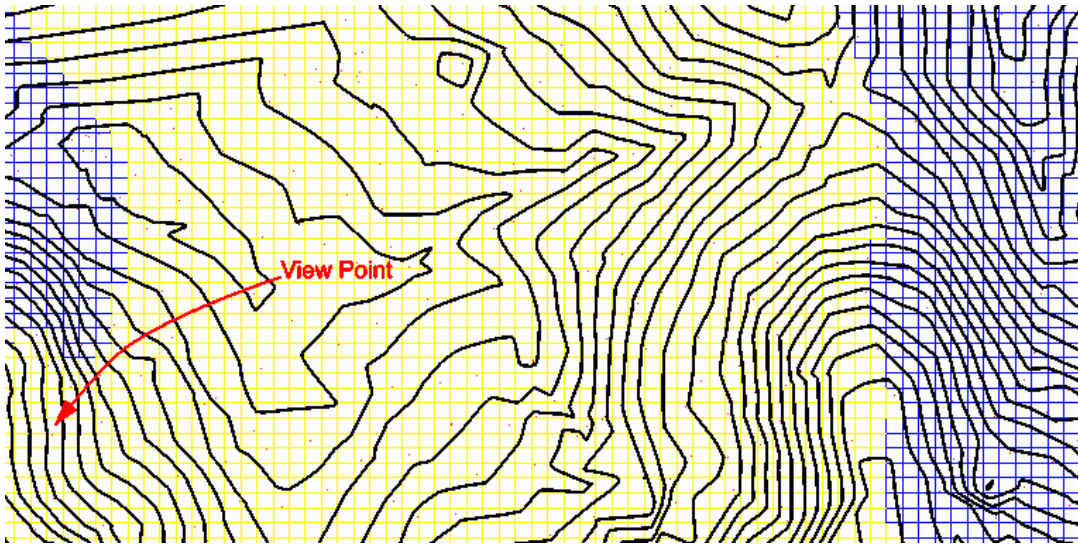
Keyboard Command: render, shade, or hide

Prerequisite: none

Line of Sight Viewshed

This command colors 3D faces for a surface for whether the 3D face is either visible or hidden from a specified view point. The 3D faces need to be already drawn before running this command. The 3D faces can come from either a grid or triangulation surface. The colors for visible and hidden are set in the dialog. There are also options to change the layer of the 3D faces for visible or hidden status. The Eye Height is added to the elevation at the View Point to set the elevation of the viewer. The Target Height sets how high of an object the viewer can see for the visible areas. The View Position by Point prompts for a single 3D point to calculate the viewshed from. The Polyline method prompts for a 3D polyline and checks multiple view positions at an interval along the polyline.





Graphic shows position of view point, surface contours and 3D faces colored with yellow for visible surface and blue for hidden surface.

Prompts

Pick view point: *pick point*

Select 3D Faces to Analyze...

Select objects: *pick 3D faces*

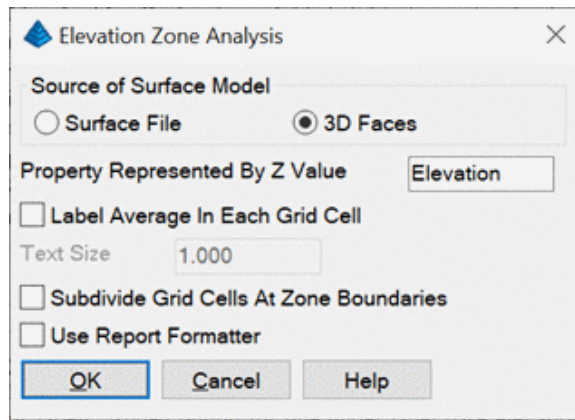
Pulldown Menu Location: Surface

Keyboard Command: viewshed

Prerequisite: 3D faces

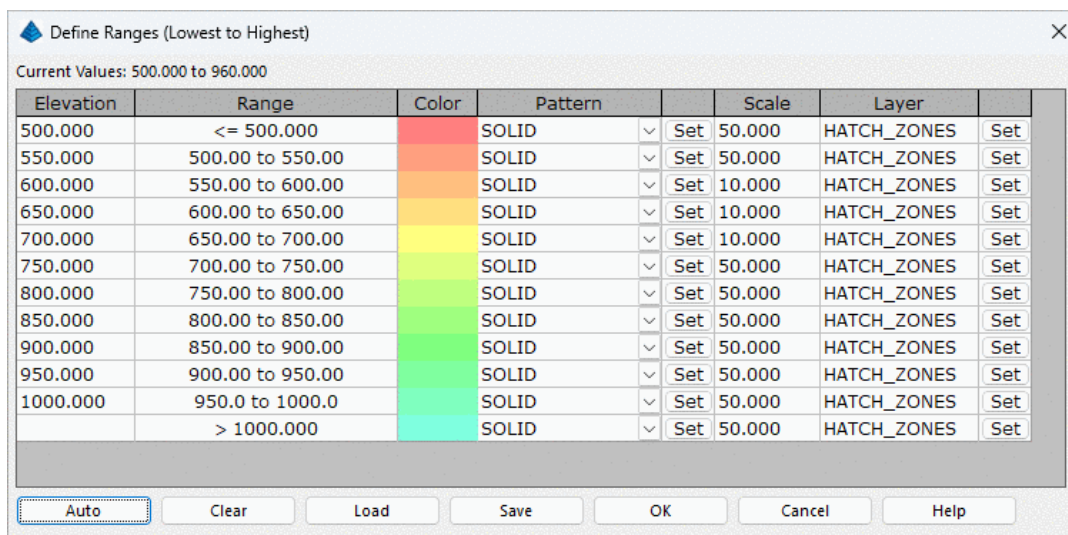
Elevation Zone Analysis

This command can be used to calculate the surface area of a surface in different elevation zone ranges and to analyze a surface by ranges or "zones" of elevation. The program requires either a triangulation surface file (TIN) or 3D Face entities. The 3D Face entities can be generated by the *Draw 3D Grid File* or *Draw Triangular Mesh* commands. The *Draw Triangulation Faces* option in *Triangulate & Contour* can also be used to create triangular 3D Faces. For each elevation zone, the 3D Faces can be hatched with a hatching pattern, solid filled with the SOLID pattern, or left empty with the NONE pattern. The 3D Faces are also placed in a separate layer for each zone. When using a surface file for input, the TIN file colors are set to the elevation zone colors which can then be rendered in the 3D Viewer commands.



In the options dialog, the **Property Represented By Z Value** sets the name used in the reports for the type of surface model. **Label Average In Each Grid Cell** creates text labels in the center of each 3D Face of the surface with the average surface value. **Subdivide Grid Cells at Zone Boundaries** gives higher resolution at the transition between zones. **Use Report Formatter** allows for customized report and different output formats.

There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the area within the inclusion polyline is calculated. Areas within an exclusion polyline are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine. Without inclusion and exclusion polylines, all the area of each selected 3D Face is used.



Prompts

Elevation Zone Analysis dialog

Select 3D Faces to Analyze...

Select objects: *pick the 3D Faces to process*

Define Ranges (Lowest to Highest) Dialog

Specify the elevation ranges, colors and patterns.

Select the Inclusion perimeter polylines or ENTER for none:

Select objects: *pick a closed polyline for the limits of disturbed area*

Select objects: *press Enter*

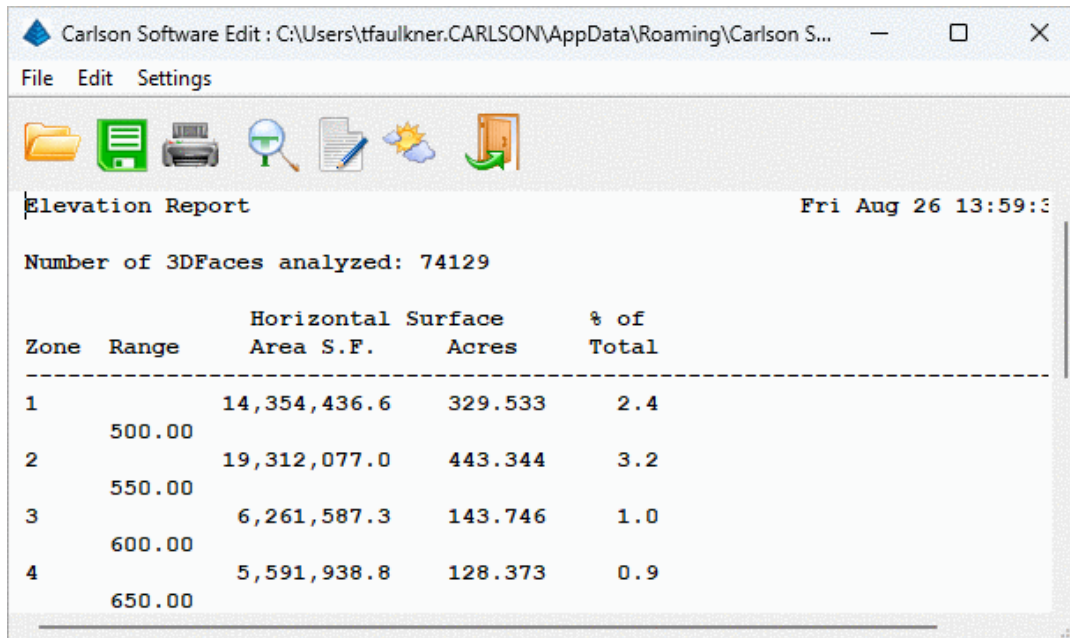
Select the Exclusion perimeter polylines or ENTER for none:

Select objects: *press Enter*

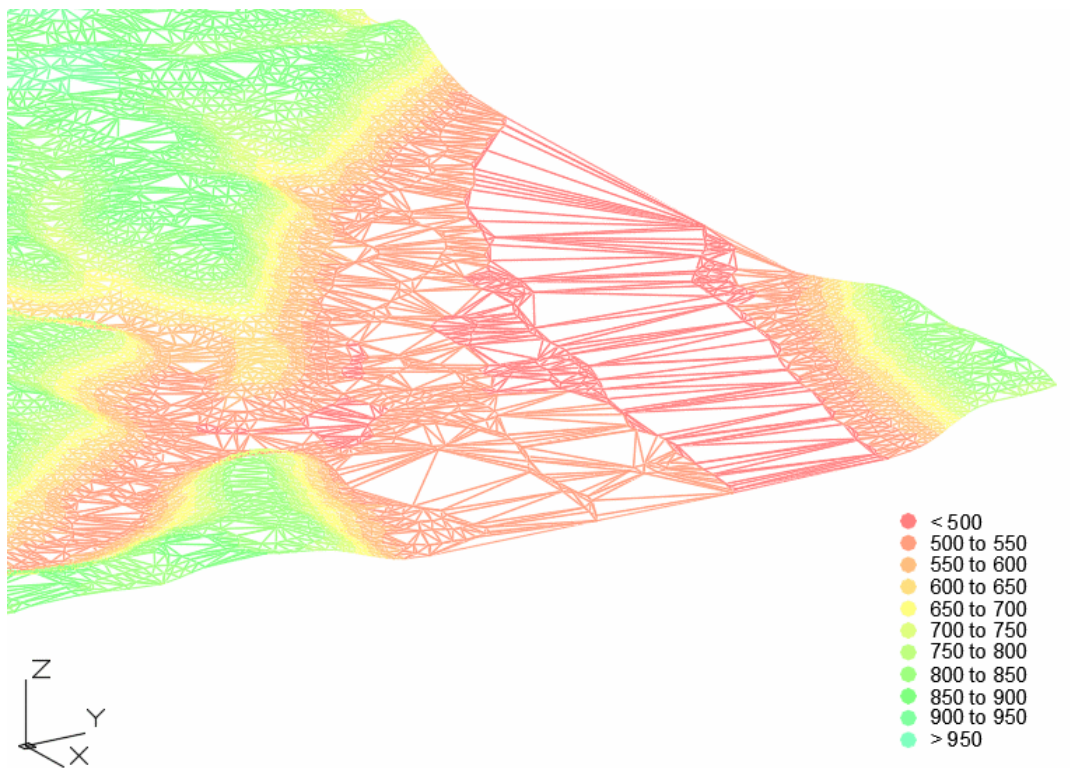
Select point for color legend (Enter for none): *pick a point*

If a point is picked, a legend showing the color of each range is drawn. The legend is drawn aligned to the current view *UCS*. For this reason it is best to have the mesh at the *Vpoint* at which it will be plotted before executing the analysis program.

A report is also generated in the standard report viewer.



Zone	Range	Horizontal Surface Area S.F.	Acres	% of Total
1	500.00	14,354,436.6	329.533	2.4
2	550.00	19,312,077.0	443.344	3.2
3	600.00	6,261,587.3	143.746	1.0
4	650.00	5,591,938.8	128.373	0.9



Result of Elevation Zone Analysis viewed in 3D and shaded

Pulldown Menu Location: Surface

Keyboard Command: elvzone

Prerequisite: displayed 3D Face entities.

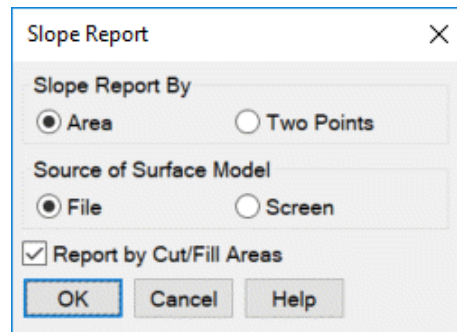
Slope Report

This command calculates the sloped surface area, average slope and average elevation on a site. The surface can be defined by a surface model file, (.GRD, .TIN or .FLT), or generated from 3D entities on the screen. Sloped area information is useful to compute seeding quantities for hillsides, for example.

For the screen method, the surface is modeled from the user-selected entities such as contour polylines. Besides the surface entities, a perimeter polyline is used as the inclusion area for the slope report. If the perimeter polyline is on the PERIMETER layer, the command will locate it automatically.

For area reports, there are options to specify inclusion and exclusion perimeters. When inclusion perimeters are specified, only the area within the inclusion perimeters is calculated. The area within exclusion perimeters is not included in the calculations. Inclusion and exclusion perimeters are represented by closed polylines and must be drawn prior to running this routine.

The option to Report by Cut/Fill Areas will report the slope areas separately for cut and fill areas. When this option is on, the program prompts for a second surface to use as a reference to determine the cut/fill areas.



Prompts

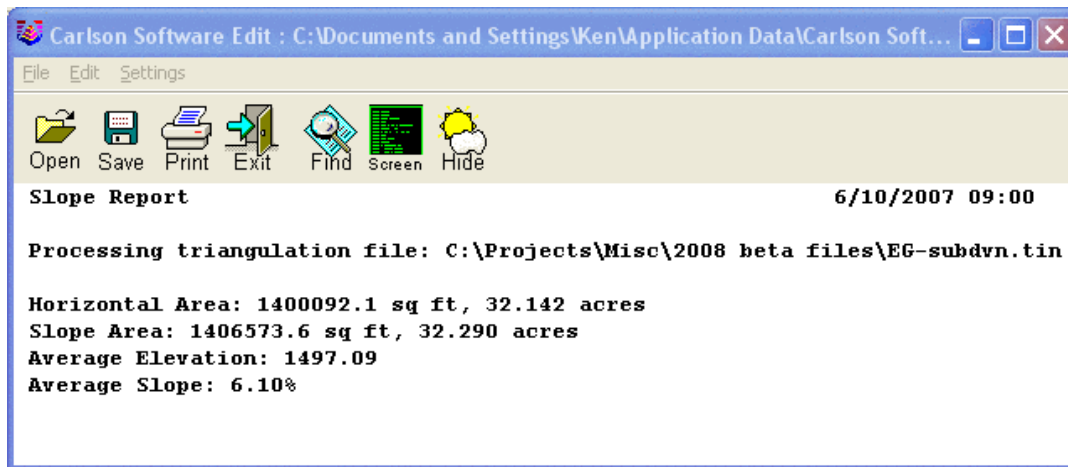
For Area report using a File:

Select surface model file.

Select the Inclusion perimeter polylines or ENTER for none: *pick any inclusion polylines*

Select the Exclusion perimeter polylines or ENTER for none: *pick any exclusion polylines*

Note: If the surface model file is a grid file (.GRD), you are prompted whether to extrapolate the grid to full grid size.



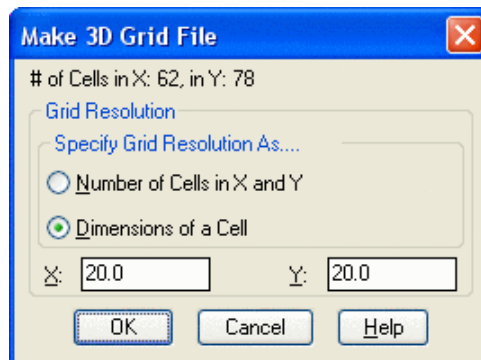
For Area report by *Screen* method:

Ignore zero elevations (<Yes>/No)? *press Enter*

Select surface entities and perimeter.

Select objects: *pick the objects*

If no polyline is found on layer PERIMETER, you are prompted to: Select Pond/Pit perimeter polyline. The Make 3D Grid File dialog is presented. Pick *OK*.



Select the Inclusion perimeter polylines or ENTER for none: *pick any inclusion polylines*

Select the Exclusion perimeter polylines or ENTER for none: *pick any exclusion polylines*

For *Points* method:

Select surface model file.

Pick first point:

Pick Second point:

The slope report is displayed on the command line for the 3D vector, projected on the surface, defined by those 2 picks.

Point 1: 5119.646,5640.322,98.979

Point 2: 4951.964,6022.419,135.546

Horiz Dist: 417.27 Slope Dist: 418.87 Elv Diff: 36.57

Slope: 8.76 Ratio: 11.41:1

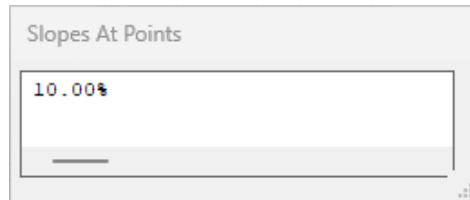
Pulldown Menu Location: Surface >> Slope Analysis

Keyboard Command: sarea

Prerequisite: A surface file or screen entities of the surface.

Slope At Points

This command reports and labels the slope between two points or at points on a surface model (triangulation or grid). The **Input Method** for **Points** prompts you to pick two points and calculates the slope between these points. The points can be either picked in the drawing or entered by point number from the current coordinate file. The Input Method for **Surface Triangles** creates a label at center of each triangle face in the surface which can work well on grid files but is typically too much information for triangulated surface files. The Input Method for **Pick on Surface** prompts to pick the points where to calculate the slope. For the Pick method, the Define Slope setting allows a couple ways to pick. The **At Single Point** option calculates the slope from the surface model at the point. The **Between Two Points** method calculates the slope using the surface model elevations at two picked points and the horizontal distance between these points. As the crosshairs are moved across the surface, the slope at the current position is displayed in a floating dialog box.

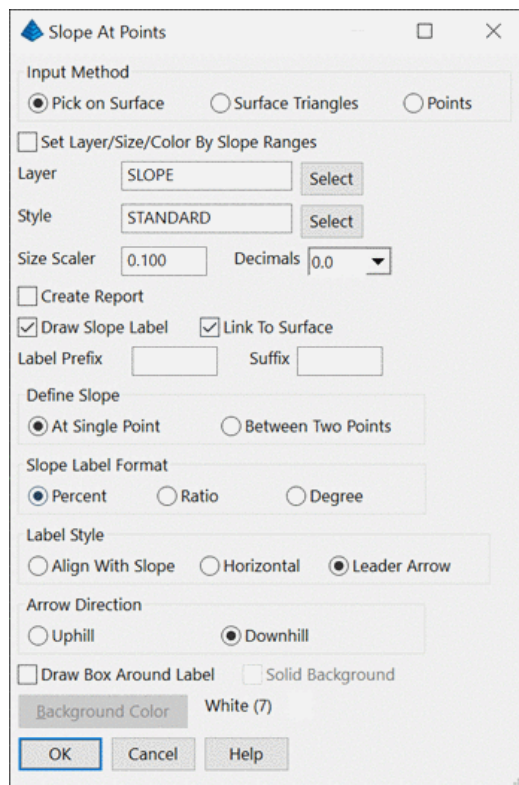


For the slope labels, the dialog has settings for the **Layer, Style, Prefix, Suffix, Decimals, Slope Label Format** and **Size Scaler**. The **Size Scaler** is multiplied by the drawing horizontal scale to calculate the text size in drawing units. The Create Report option uses the **Report Formatter** to make a report of the coordinates, slope amount and slope direction.

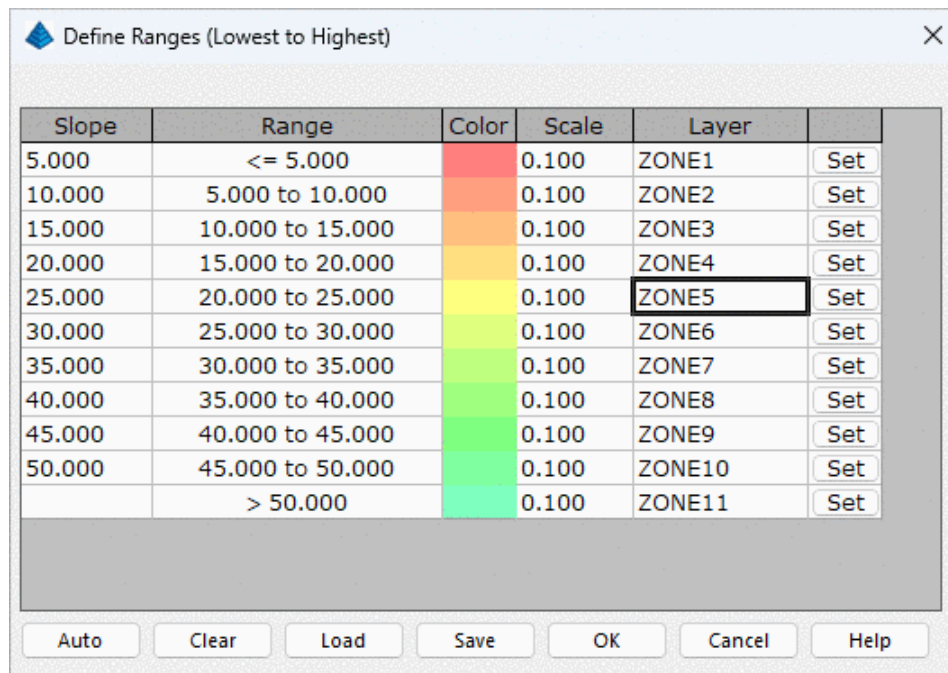
The **Label Style** of **Align With Slope** rotates the slope label in the slope direction. The **Horizontal** option creates a horizontal label. The Leader Arrow rotates the slope label in the slope direction and draws a leader arrow for the slope direction either facing uphill or downhill depending on the **Arrow Direction** option.

The **Link To Surface** option will update the slope label when the triangulation surface model is modified.

The **Draw Box Around Label** option draws a polyline box around the slope labels and the **Solid Background** option fills in this box with the specified color.



The **Set Layer/Size/Color By Slope Ranges** option invokes the Define Ranges dialog box. Enter slope values in the first column of boxes to set the Ranges.



Prompts

Slope At Points dialog box

Adjust settings as desired. Pick OK.

Select Surface Model.

Pick Points to label slope.



Draw grid file and Slope At Point labels using Surface Points

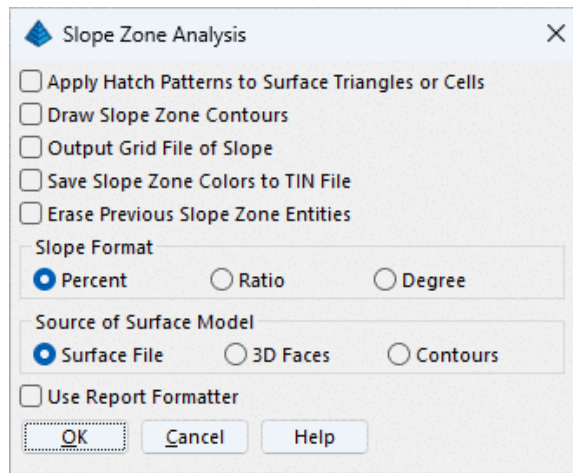
Pulldown Menu Location: Surface >> Slope Analysis

Keyboard Command: ptslope

Prerequisite: A surface model file (.TIN, .GRD, or .FLT)

Slope Zone Analysis

This command calculates the surface area of a site in different slope zone ranges. This command can use either a surface model file, (.TIN, .GRD, or .FLT), contour polylines, or 3D Face drawing entities which can be generated by the *Plot 3D Grid File* command, the *Draw Triangular Mesh* command, or the *Draw Triangulation Faces* option of *Triangulate & Contour*. For each slope zone, the 3D Faces can be hatched with any hatch pattern, including the SOLID pattern, or left empty with the NONE pattern. The command reports the area for each slope zone. When using a triangulation surface file, the *Save Slope Zone Colors* option sets the triangle colors in the TIN file for the slope zones. Then commands like *3D Viewer Surface File* can visualize the colored surface. When using the *Contours* method, the program prompts to select the contour polylines along with optional points to picking up high and low points. Using the *Contours* method along with the option to *Draw Slope Zone Contours with Hatch Slope Zones*, the hatch zone boundaries will follow the contours. The *Use Report Formatter* option allows for customized reports and different output formats.



The Draw Slope Zone Contours option generates contours of the slope zones based on the calculated slope at each point of the 3D Faces. The Output Grid File of Slope option creates a grid file of the slope values.

There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the slope area within the inclusion polyline is calculated. Slope area within an exclusion polyline are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine. Without inclusion and exclusion polylines, all the slope area of each selected 3D Face is used.

Prompts

Slope Zone Options dialog box. Choose whether to Draw Slope Zone Contours, whether to Output Grid File of Slope, and Slope Format. Pick *OK*

Select surface model file.

Define Ranges dialog. Specify the slope zones, colors and patterns from lowest to highest. Pick *OK*.

Select the Inclusion perimeter polylines or ENTER for none: *select perimeter(s) or press Enter*

Select the Exclusion perimeter polylines or ENTER for none: *select perimeter(s) or press Enter*

Report is generated.

Slope Report

Surface File: C:\scdev\data\SIMO2.grd

Average Slope: 21.6%

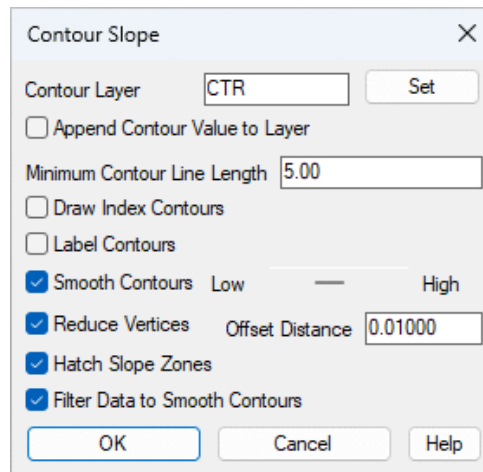
Minimum Slope: 0.0%

Maximum Slope: 81.7%

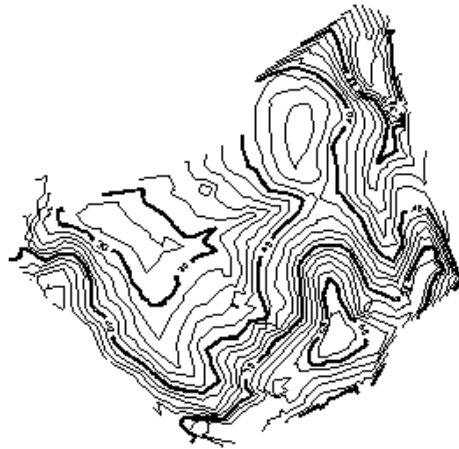
Zone	Range	Horizontal Surface Area S.F.	Acres	Slope Surface Area S.F.	Acres	% of Total	Average Slope %
1	20.00%	769,137.1	17.657	775,379.4	17.800	51.8	11.8
2	40.00%	583,761.4	13.401	607,257.9	13.941	39.3	28.2
3	60.00%	113,978.1	2.617	125,967.2	2.892	7.7	46.8
4		18,494.6	0.425	22,407.7	0.514	1.2	68.3
Total		1,485,371.1	34.099	1,531,012.2	35.147		

If you choose to draw Slope Zone Contours, the Contour Options dialog box is presented. There are settings for the contour layer and whether to label, smooth and reduce the contours. The Hatch Slope Zones option fills in the slope zone areas with colored hatching. The Filter Data to Smooth Contours option applies a filtering algorithm that reduces the noise. The slopes can vary greatly between neighboring points. When contoured directly, these slope

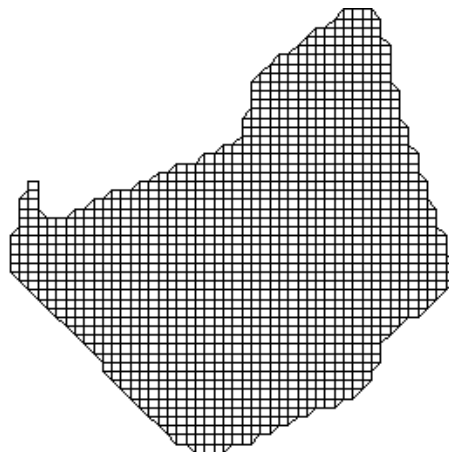
data points can produce incoherent contours with the smoothing.



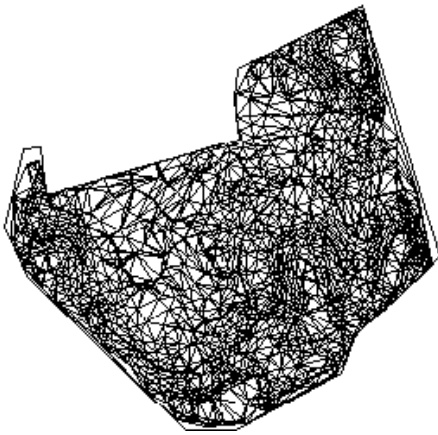
Note: If you choose to use Screen entities instead of a surface model file, you are prompted whether to: **Freeze grid layer after processing [Yes/<No>]?**



Surface contours



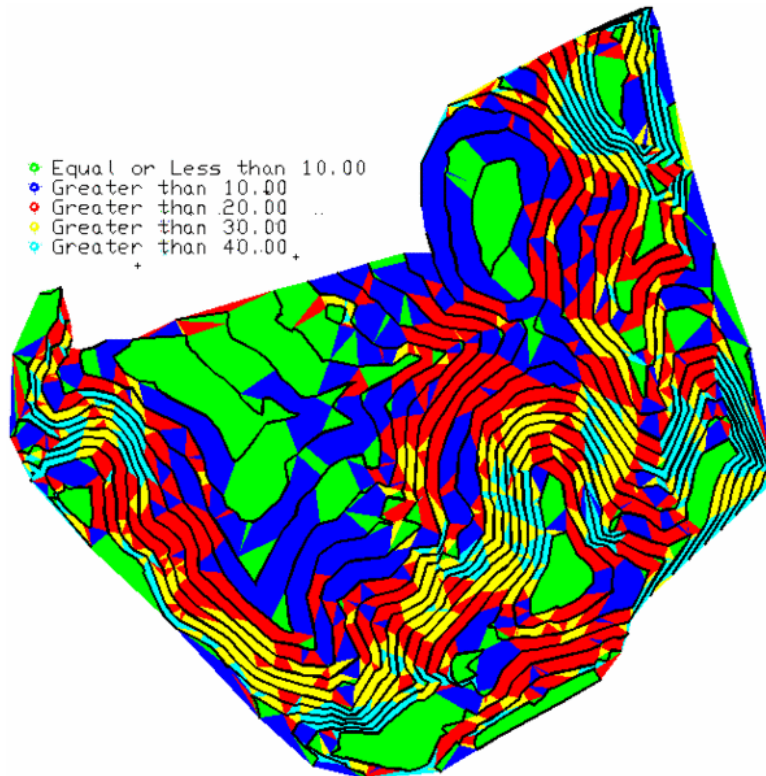
3D Faces from a grid surface model



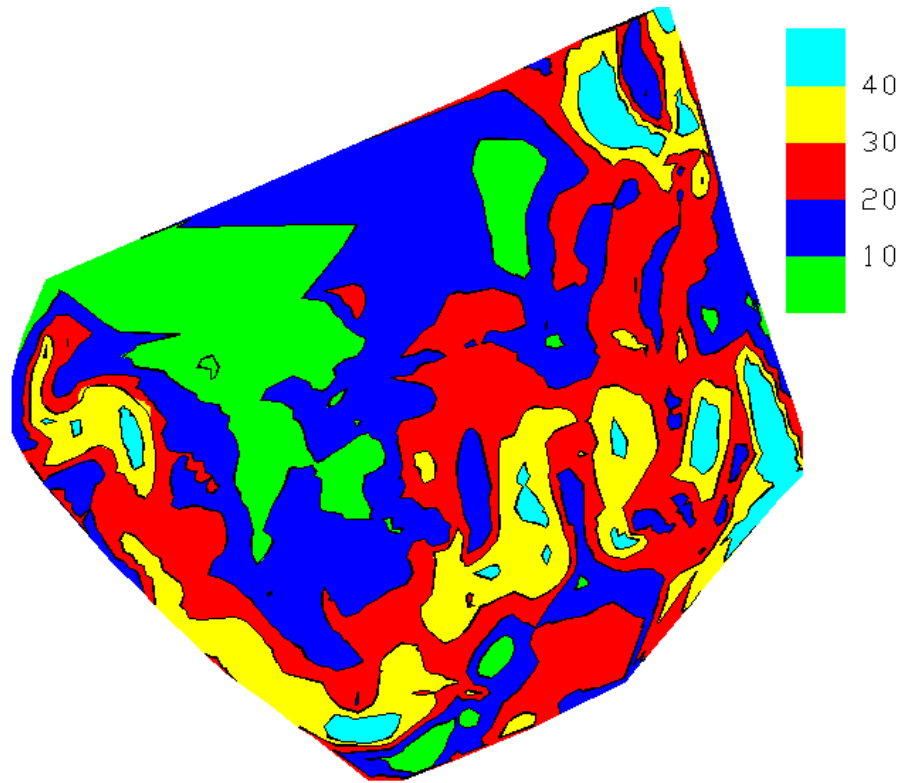
3D Faces created by *Triangulate & Contour* with the Draw Triangulation Faces option



Slope zone contours



Slope zones that follow the surface contours using the triangulation 3D Faces



Hatched slope zone contours created from the grid 3D Fac

Pulldown Menu Location: Surface >> Slope Analysis

Keyboard Command: szone

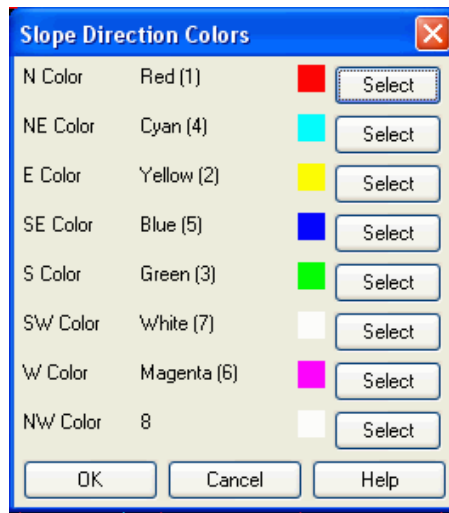
Prerequisite: Surface model file (.TIN, .GRD, or .FLT), or 3D Faces entities

Slope Direction Analysis

This command categorizes the slope direction as either N, NE, E, SE, S, SW, W or NW. The program requires 3D Face entities that can be generated by the *Draw 3D Grid File* command, the *Draw Triangular Mesh* command, or the Draw Triangulation Faces option of *Triangulate & Contour*. Each 3D Face is colored by the slope direction zone and a report of the area for each zone is generated. A pinwheel color legend can also be drawn.

There are options to specify inclusion and exclusion areas. When inclusion areas are specified, only the slope area within the inclusion polyline is calculated. Slope area within an exclusion polyline are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine. Without inclusion and exclusion polylines, all the slope area of each selected 3D Face is used.

Prompts



Select 3D Faces to Analyze ...

Select objects: *pick the 3D faces*

Slope Direction Colors Choose a color for each direction.

Select the Inclusion perimeter polylines or ENTER for none:

Select objects: *Pick a closed polyline if needed*

Select the Exclusion perimeter polylines or ENTER for none:

Select objects: *Pick a closed polyline if needed*

Select point for color legend: *pick a point*

Carlson Software Edit : C:\carlson2007b4\USER\sc...

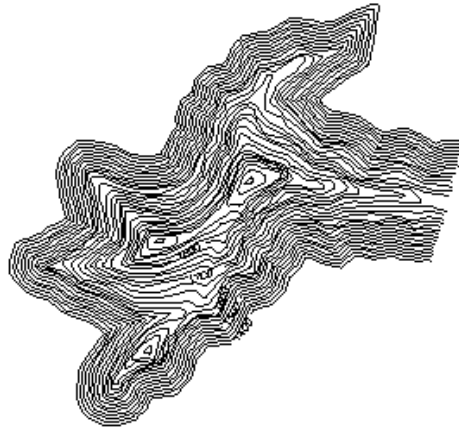
File Edit Settings

Open Save Print Exit Find Screen Hide

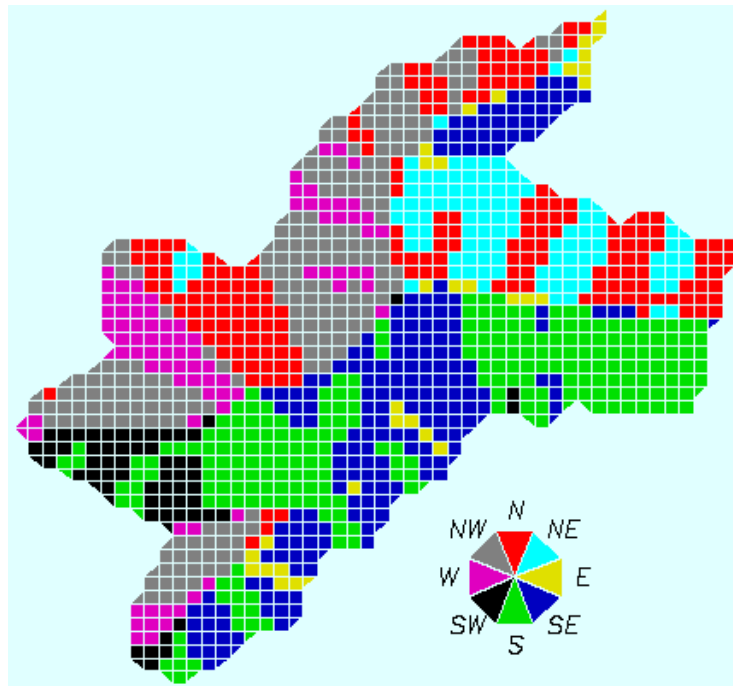
Slope Direction Report

Number of 3DFaces analyzed> 4728

Zone	Horizontal Surface Area S.F.	Acres	% of Total
N	315566.25	7.24	7.80
NE	503189.14	11.55	12.45
E	550047.67	12.63	13.60
SE	548616.28	12.59	13.57
S	581046.30	13.34	14.37
SW	476954.75	10.95	11.80
W	615927.54	14.14	15.23
NW	451803.73	10.37	11.17
Total Hz Surface Area>		4043151.66	92.82



Original Contours



Surface Colored by Slope Direction

Pulldown Menu Location: Surface >> Slope Analysis

Keyboard Command: dzone

Prerequisite: 3D Face entities

Convert LDD Contours

This command allows you to convert Autodesk Land Desktop contours (known as AECC_CONTOUR objects) into polylines. You must have the AEC Object Enabler installed before using this command. If you do not have the object enabler installed, download the latest version from <http://www.autodesk.com>.

Note: If no object enabler is installed, opening a Land Desktop drawing with contours will display large boxes for each contour, essentially outlining the extents of each one. In this case you will need to download the object enabler. If the object enabler is installed, contours will appear normally, and you can use this command to convert them to standard lwpolylines or you can use the *Explode* command. The Carlson *Convert LDD Contours* command is preferable only in the fact that it will search the drawing for AECC_CONTOUR objects and convert only those, while an *Explode* command could inadvertently explode other entities that you do not wish to be exploded.

You can use the *List* command to determine if contours are polylines or AECC_Contour objects. Here is an example listing:

AECC_CONTOUR Layer: "CONT-MJR"

Space: Model space

Handle = 429

Major Contour Interval

Elevation: 1005.00

Smoothing: None

Number of Vertices: 48

Open

Length: 560.25

Constant width: 0.00

Style Name: Standard

Prompts

Select AEC Contours to convert

Select objects: *pick the AEC contour entities*

Pulldown Menu Location: Surface >> Import/Export

Keyboard Command: aec_cntr

Prerequisite: AEC Contours to convert

Convert LDT/Civil3D Surface Drawing

This command allows you to convert Autodesk Civil3D or Land Desktop custom surface objects into standard AutoCAD entities. For example, this command will convert AECC_CONTOUR objects into polylines with elevation. The conversion is done on the currently opened drawing. Besides converting the drawing entities, the routine checks for triangulation surface definitions within the drawing and prompts whether to save these surfaces to Carlson .TIN files. The conversion routine was developed in cooperation with the Open Design Alliance (ODA) and does not use object enablers from Autodesk.

If you have Civil 3D, another way to make a drawing with standard AutoCAD entities is to use the aectoacad command in Civil 3D which converts the custom objects into standard entities.

Pulldown Menu Location: Surface >> Import/Export

Keyboard Command: c3d_tin

Prerequisite: AEC surface objects to convert

Import NASA World Elevation Dataset

This command creates a grid surface file (.GRD) using a world elevation dataset from the NASA Space Shuttle program. The data covers all land areas up to latitude 60. The data was recorded in 2000 by the Shuttle Radar Topography Mission (SRTM). The data is at an interval of 1 arc-second or about 100 feet.

Before running this command, the grid coordinate system must be setup in Settings > Drawing Setup. The program first prompts to pick a window in the drawing for the surface area. The first time that you access data in an area, the program will download source data from the Carlson Software server.

Prompts

Pick first corner of bounding box: Identify one corner of a drawing window for the surface area

Pick second corner of bounding box: *Identify the opposite corner*

Surface File To Write

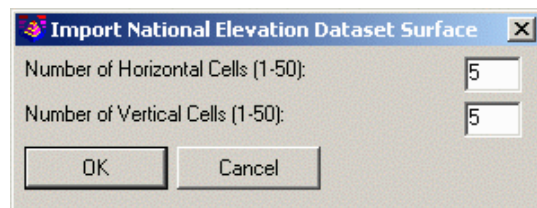
Pulldown Menu Location: Civil > Surface > Import/Export Surface, Survey > Surface > Import/Export Surface, Construction > Import/Export

Keyboard Command: nasa_grd

Prerequisite: Coordinate projection system, Internet connection

Import National Elevation Dataset

This command creates a grid or triangulation surface file (GRD or TIN) using the National Elevation Dataset (NED) from the USGS. The NED source data is at a resolution of 1/3 arc second. Before running this command, the grid coordinate system must be setup in Settings > Drawing Setup. The program first prompts to pick a window in the drawing for the surface area. Then there is a dialog to set the number of sample points. Currently the NED service is slow. So the number of samples is limited to 50 in X and Y directions. Also the NED data has limited grid resolution and does not interpolate. So if your sample spacing is smaller than the NED resolution, then samples within the same NED grid cell will have the same elevation.



Prompts

Pick first corner of bounding box: Identify one corner of a drawing window for the surface area

Pick second corner of bounding box: *Identify the opposite corner*

Options dialog

Surface File To Write

Pulldown Menu Location: Civil > Surface > Import/Export Surface, Survey > Surface > Import/Export Surface, Construction > Import/Export

Keyboard Command: getneddata

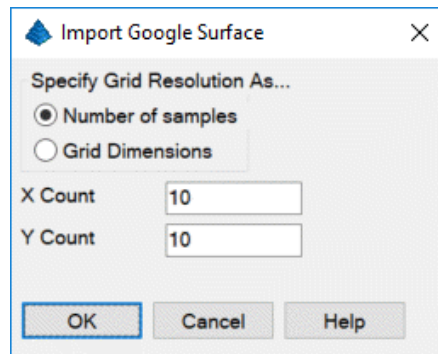
Prerequisite:Coordinate projection system, Internet connection

Import Google Earth Surface

This command creates a Carlson Triangulation file (TIN) by downloading surface data from Google Earth. While the elevation accuracy of the Google Earth surface should be considered coarse, it might be suitable for large-scale watershed modeling studies, preliminary land-planning studies or "proof-of-concept" preliminary designs.

Before running this command, the grid projection system for the drawing must be defined in Settings > Drawing Setup.

After selecting the TIN file to output, the program prompts to pick the rectangular area in the drawing for where to create the surface file. Then there is a dialog to set the grid resolution of the data points. You can either specify the number of rows and columns for the grid or define the size of the grid cells.



Note:

- In an effort to protect their servers from abuse, Google limits the number of data points.
- The Import Google Earth Surface routine fetches terrain data in real-time from the Google servers and requires an Internet connection to proceed.
- It bears repeating that the terrain data returned by Google Earth should only be used for illustrative or proof-of-concept purposes only!
- To import a Google Earth image into your drawing, use the Place Google Earth Image command.
- To import KML content into your drawing, use the Import Google Earth File command.
- To export content from your drawing to a KML file, use the Export Google Earth File command.

Prompts

Select Output File: *Choose a TIN file to create.*

Pick first corner of bounding box: *Identify one corner of a drawing window that should be used to set the Google Earth display*

Pick second corner of bounding box: *Identify the opposite corner of a drawing window that should be used to set the Google Earth display*

Import Google Surface dialog

Pulldown Menu Location: Civil > Surface > Import/Export Surface, Survey > Surface > Import/Export Surface, Takeoff > Tools > Import/Export, Construction > Import/Export

Keyboard Command: gesurface

Prerequisite: Coordinate projection system defined in Drawing Setup and Internet connection

Import/Export Trimble TTM File

These commands convert between Trimble TTM format triangulation files and Carlson format. First you select the source file to read and then the destination file to write.

Pulldown Menu Location: Surface->Import/Export Surface

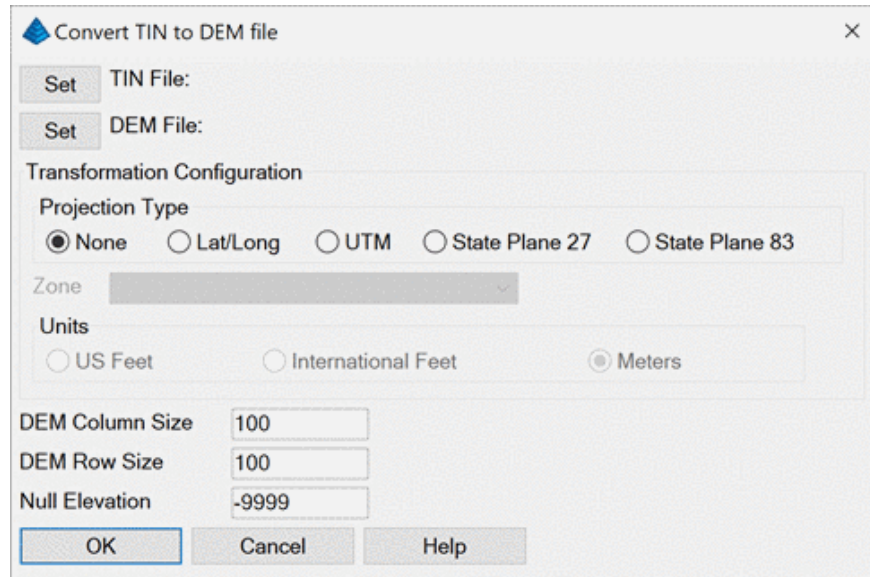
Keyboard Command: ttm2tin, tin2ttm

Prerequisite: File to convert

Export DEM TIF File

This command creates a GeoTIFF DEM (Digital Elevation Model) file from a TIN file. The GeoTIFF DEM file is a geolocated TIFF file with one raster band representing elevation. The GeoTIFF DEM file is created by first creating a grid of "DEM Column Size" columns and "DEM Row Size" rows across the extents of the TIN (based on the TIN's x,y bounds). The center of each grid cell is sampled, and the elevation of the TIN at that sample location is used to produce a corresponding pixel in the output GeoTIFF DEM file. Geolocation information for the output GeoTIFF DEM file is based on the extents of the input TIN file projected according to the "Transformation Configuration" options.

Convert TIN to DEM file dialog:



Input/Output Options: The "TIN File" "Set" button allows the user to select the input TIN file. The "DEM File" "Set" button allows the user to select the output GeoTIFF DEM file.

Transformation Configuration: This column provides options for setting the projection that will be applied when geolocating the output GeoTIFF DEM file. This projection specifies how to interpret the extents of the TIN file.

- **Projection Type:** Specifies how to interpret/project the extents of the TIN when geolocating the output GeoTIFF DEM file.
 - **None:** No projection. The extents of the TIN will be used to geolocate the output GeoTIFF DEM file with no projection.
 - **Lat/Lon:** Use Lat/Long projection. The extents of the TIN will be interpreted as Lat/Long values.
 - **UTM:** Use Universal Trans Mercator (UTM) projection. The extents of the TIN will be interpreted as UTM values. UTM Zone can be set from the "Zone" pop list.
 - **State Plane 27:** Use State Plane 27 projection. The extents of the TIN will be interpreted as State Plane 27 values. State Plane 27 Zone can be set from the "Zone" pop list.
 - **State Plane 83:** Use State Plane 83 projection. The extents of the TIN will be interpreted as State Plane 83 values. State Plane 83 Zone can be set from the "Zone" pop list.
- **Zone:** Specifies the UTM, State Plane 27, or State Plane 83 zone depending on "Projection Type" settings.
- **Units:** Specifies the units for the output GeoTIFF DEM file.

Elevation Sampling Options:

The "DEM Column Size," "DEM Row Size," and "Null Elevation" options specify how the "Export DEM TIF File" routine should go about sampling the input TIN file.

- **DEM Column Size:** Specifies the number of columns that will be included in the sampling grid.
- **DEM Row Size:** Specifies the number of rows that will be included in the sampling grid.
- **Null Elevation:** Specifies the pixel value to be used in the output GeoTIFF DEM file if no elevation information exists at a geolocated pixel's sampling position on the TIN.

Pulldown Menu Location: Surface > Import/Export Surface

Keyboard Command: tin2dem

Prerequisite: A triangulation file

Export Topcon TIN File

The Export Topcon TIN File command writes a Topcon TIN file (.TN3) from a Carlson triangulation file (.TIN, .FLT). The routine first prompts for the Carlson file and then the Topcon file.

The Import Topcon TIN File command creates a Carlson Tin file (.TIN, .FLT) from a Topcon triangulation file (.TN3). The routine first prompts for the Topcon file and then the Carlson file.

The units (Feet or Meters) for the triangulation file are the current units set in Drawing Setup.

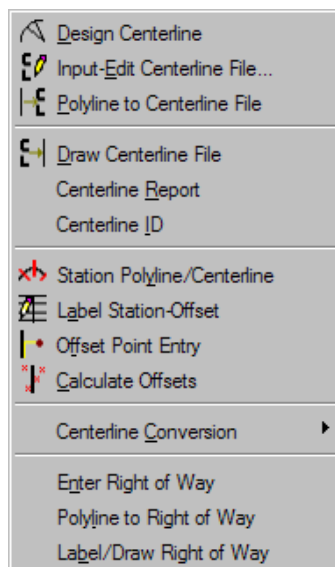
Pulldown Menu Location: Surface > Import/Export Surface

Keyboard Command: topcon_tin, tn3_to_tin

Prerequisite: A triangulation file

Centerline Menu

The Centerline menu provides commands for designing and editing centerlines and centerline files. Tools for stationing, labeling and offsetting centerlines, along with Right of Way features, are also provided in this menu. Additionally, there are many import and export conversion options to select from when you pick *Centerline Conversion*.



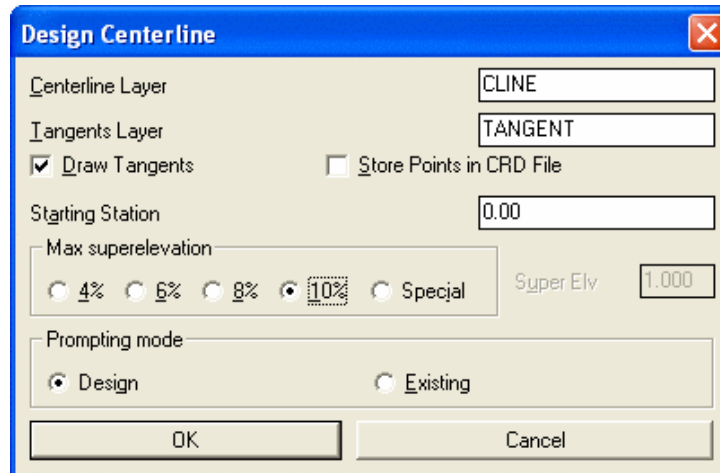
Design Centerline

This command draws a centerline polyline and writes the centerline data in a centerline file. The first step is to specify a centerline (.CL) file name. Next in the Design Centerline dialog you can specify several options. Centerline Layer is the layer name for the polyline. Tangents Layer is the layer name for the tangent lines drawn

from the centerline to the curve center. Max superelevation is used for determining the minimum recommended radius. Setting the Prompting mode to Existing skips design questions such as design speed.

After the Design Centerline dialog, the program cycles through curve prompting until End is selected. There are PC and PI modes for curve entry. In PC mode the arc's PC points are entered followed by the curve data. The PC points can be specified by either picking the point, entering a distance or entering a station. In PI mode, the arc's PI points are entered. Once the PI points determine two tangents, the program prompts for curve data for the previous PI. Spirals can only be entered in PI mode. You can switch between arc and PI mode between curves on the polyline. The arc curvature can be specified by degree of curve or radius. The minimum recommend radius is based on AASHTO. The arc length can be specified by PT station, tangent length or arc length.

The Store Points in CRD File will create points in the current coordinate file for each design point on the centerline. This option is also used for creating the SMI chain file within Centerline Utilities, since the SMI chain file requires point numbers. To specify the coordinate file, choose *Set Coordinate File* in the Points menu.



Prompts

Centerline file to design Enter the .CL file name to create.

Design Centerline Dialog Choose your options and click OK.

Pick Point or Point number: *pick a starting point or enter the starting point coordinates*

For PC mode design:

Bearing/PI/End/Undo/<Pick Point or Point number>: *pick the PC point*

Bearing/PC/PI/End/Undo/<Pick Point or Point number>: *PC*

Enter Design Speed for curve <55.00>: *40*

Minimum Recommended Radius = 426.67

View/Point/Degree of Curve/<Radius>: *500*

Curve direction (Left/<Right>)? *press Enter for right*

Length to use (Station/Tangent/<Arc>)? *press Enter for arc*

Point/Station/Tangent/<Arc Length>: *200*

Reverse/Compound Curve (Yes/<No>)? *press Enter*

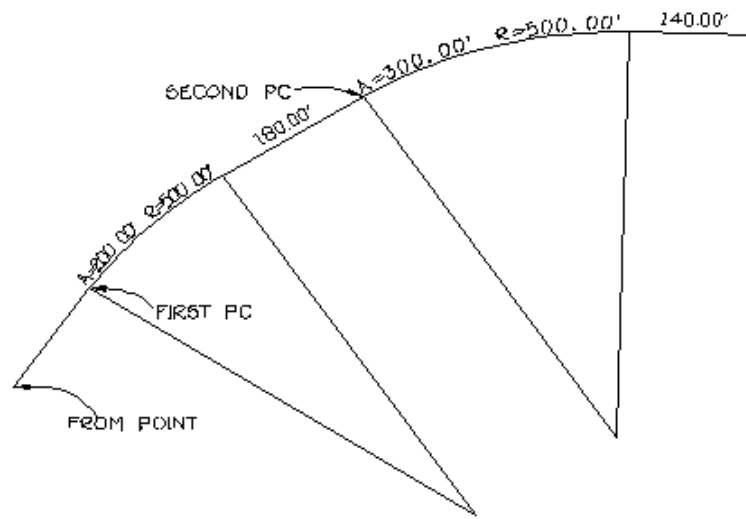
PI/Distance/Station/<Pick PC or Point number>: *D for distance*

Point/Enter Distance: *180*

Bearing/Line/Undo/End/<Continue PC>: *press Enter*

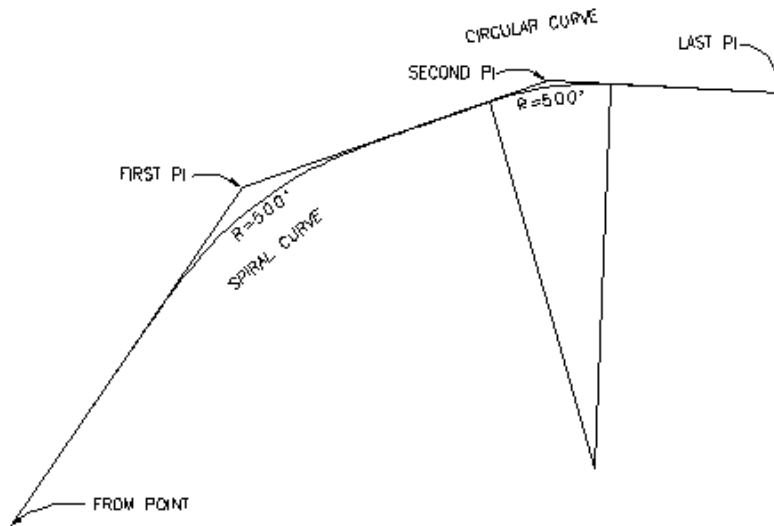
Enter Design Speed for curve <40.00>: *press Enter*

Minimum Recommended Radius = 426.67



Example of PC mode centerline design

View/Point/Degree of Curve/<Radius>: 500
Curve direction (Left/<Right>)? *press Enter*
Point/Station/Tangent/<Arc length>? 300
Reverse/Compound Curve (Yes/<No>)? *press Enter*
PI/Distance/Station/<Pick point or Point number>: D for distance
Point/Enter Distance: 140
Bearing/Line/Undo/End/<Continue PC>: E to end
 For PI mode design:
Bearing/PI/End/Undo/<Pick Point or Point number>: pi
Pick Point or Point number (PI)<5098.50,3509.11>: pick the first PI point
Type of curve [Spiral/<Circular>]? S for spiral
Enter Design Speed for curve <55.00>: 40
Minimum Recommended Radius = 426.67
View/Point/Degree of Curve/<Radius>: 500
Enter Number of Lanes <2>:
View/Enter Spiral Length In <204.8000>: 210
View/Enter Spiral Length Out <210.0000>: press Enter
Bearing/Pick next Point or Point number (PI): pick the next PI point
 TS: 1+33.280
 SC: 3+43.280
 CS: 6+39.364
 ST: 8+49.364
Bearing/Line/PC/Undo/End/<Continue PI>: press Enter
Type of curve [Spiral/<Circular>]? press Enter for circular
Enter Design Speed for curve <40.00>: press Enter



Example of PI mode centerline design

Minimum Recommended Radius = 426.67

View/Point/Degree of Curve/<Radius>: 500

Bearing/Pick next Point or Point number (PI): *pick the last PI*

PC : 9+35.900

PT : 16+34.283

Reverse/Compound Curve [Yes/<No>]? *press Enter*

Bearing/Line/PC/Undo/End/<Continue PI>: *E to end*

EndPoint : 18+37.121

Stations are printed for every PC, PT and end point in the design process.

Pulldown Menu Location: Centerline

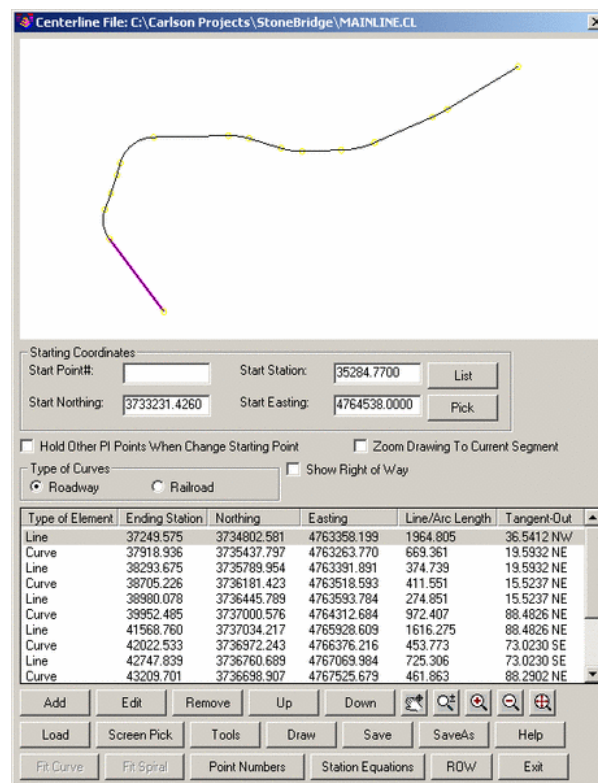
Keyboard Command: centerln

Prerequisite: None

Centerline Editor

This command can be used to input a new centerline or edit an existing centerline (.CL) file. It is a dialog-based alternative to Design Centerline and has the advantage of accepting whatever information you have on your centerlines (coordinates, stationing, length of tangents and arcs, *etc*). For creating a new centerline, it is ideal for entering data straight from highway design plans. For editing, this command allows you to change any of the geometric properties of any of the elements of the centerline (lines, curves, spiral-only and symmetrical spiral-curve-spiral elements), including the starting coordinates and station.

Starting this command launches the Centerline Editor main dialog box. To edit an existing Centerline, you can either pick the Load button and pick the .CL file, or pick the Screen Pick button and pick the polyline in the drawing that represents the Centerline. The Centerline is then displayed in the graphics window of the dialog box. The highlighted segment in the text window is also highlighted in the graphics window.



Drag Action (Zoom and Pan): In the graphics window, hold the left mouse button down and move mouse to Pan, roll the wheel to Zoom.

Zoom Drawing To Current Segment: This option zooms the drawing graphics to center on the centerline segment currently highlighted in the dialog.

Hold Other PI Points When Change Starting Point: With this option active, all the existing PI's are held when the starting coordinate is moved. Otherwise, all the PI's are moved by the same amount that the starting point is moved.

Show Right of Way: This option shows any ROW's defined in the centerline in the graphic preview window.

Type of Curves: This setting chooses between roadway and railroad definitions for curve lengths.

Add: Adds a new element after the highlighted element. Prompts you for the type of the element to be added, Line, Curve, Spiral-Only or Spiral-Curve-Spiral.

Edit: Allows you to edit the highlighted segment.

Remove: Removes the highlighted element from the centerline.

Up/Down: Moves elements in the table Up and Down in the list. For example, if this centerline ended with a tangential line from the last curve, then was followed by a non-tangential line at 45d NE, moving the last element up would create a line at 45d after the curve (non-tangential), and the formerly tangential line will remain tangential and therefore continue at NE 45d.

Load: Loads an existing centerline (.CL) file for review or editing. After loading a centerline, the listbox in the dialog shows a list of all the elements in the centerline, identifying them as either a line, curve, spiral only or full spiral-curve-spiral element and reporting the ending station, northing and easting of the element.

Screen Pick: Allows user to pick a CL off the screen in the drawing to load into the editor.

Tools > Reverse: Reverses direction of Centerline.

Tools > Rotate: Rotates the centerline by the specified rotation angle and around the specified pivot point.

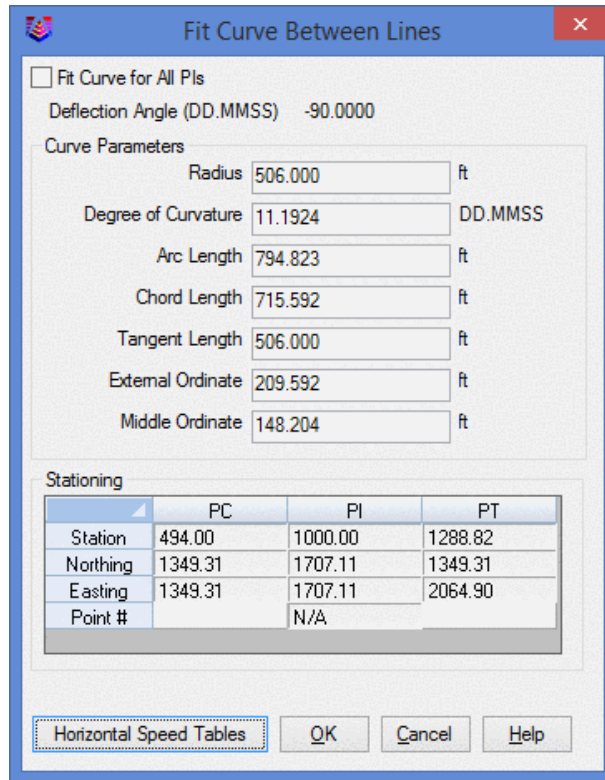
Tools > Scale: Scales the centerline which can be used for changing units between metric and English.

Draw: This button draws the centerline in the drawing on the specified layer.

Save: Saves the currently loaded centerline to a file, or will prompt you for a name if no name has been set.

SaveAs: Prompts you for a file name for the saved file.

Fit Curve: Fits a circular curve element into the centerline after the line element that is currently selected. When all the elements are lines, the program allows you to fit curve for all PIs. The program checks if the radius fits all PIs and will prompt the maximum radius that works for all PIs if the current radius is too big. The Horizontal Speed Tables allows you to pick a speed and a super elevation rate to the minimum radius.

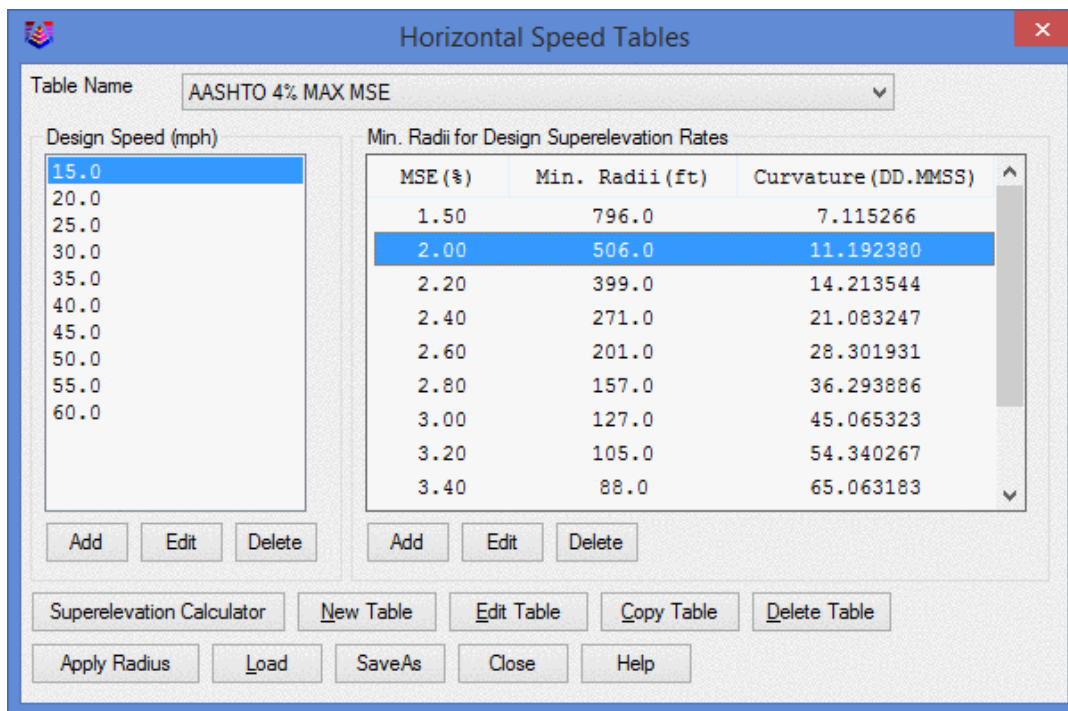


The dialog box titled "Fit Curve Between Lines" contains the following elements:

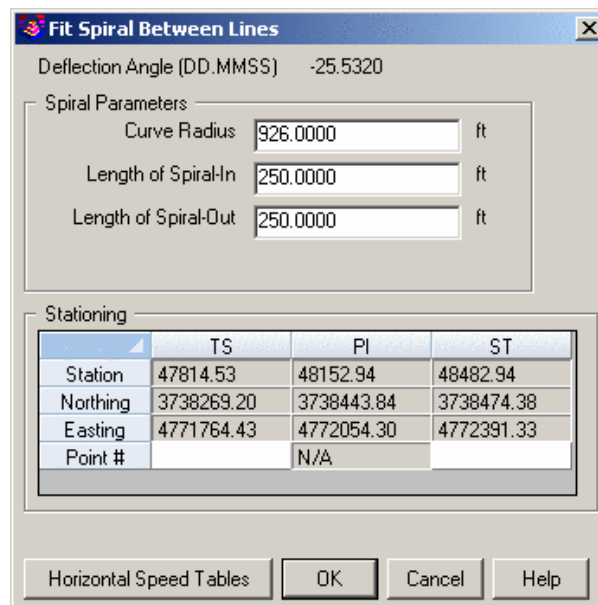
- Fit Curve for All PIs
- Deflection Angle (DD.MMSS) -90.0000
- Curve Parameters section with input fields:
 - Radius: 506.000 ft
 - Degree of Curvature: 11.1924 DD.MMSS
 - Arc Length: 794.823 ft
 - Chord Length: 715.592 ft
 - Tangent Length: 506.000 ft
 - External Ordinate: 209.592 ft
 - Middle Ordinate: 148.204 ft
- Stationing table:

	PC	PI	PT
Station	494.00	1000.00	1288.82
Northing	1349.31	1707.11	1349.31
Easting	1349.31	1707.11	2064.90
Point #		N/A	

Buttons at the bottom: Horizontal Speed Tables, OK, Cancel, Help.

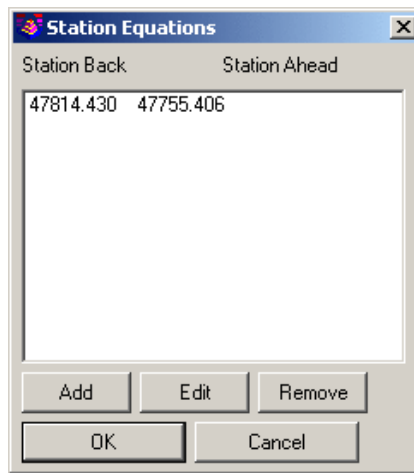


Fit Spiral: Fits a spiral curve element into the centerline after the line element that is currently selected.



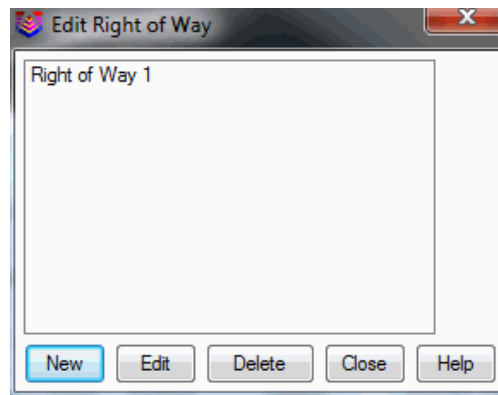
Point Numbers: This will create Carlson points along the elements of the centerline and store them to the current CRD file. The new points will be numbered in sequence beginning with the first available point number in the CRD file.

Station Equations: At any number of locations on a centerline, you can set the back station and forward station for the re-stationing of the centerline. The station equation dialog appears below:

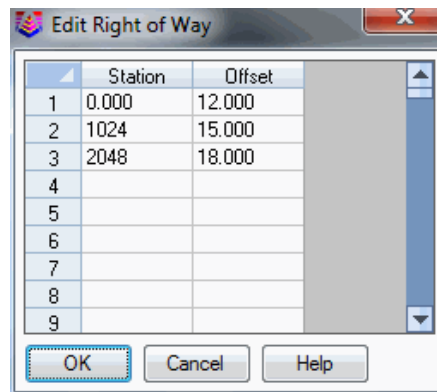


If the Station Back is lower than the Station Ahead, then a "gap" is inserted in the centerline, where the stations jump forward. If the Station Ahead is less than the Station Back, then an overlap occurs, where the common station range is repeated.

ROW: This function edits the right-of-way definitions associated with the centerline. There can be multiple ROW's assigned to the centerline for left and right sides as well as multiple on the same side. The function first shows a list of ROW's for the centerline where you can add, edit or delete.



When you add or edit a ROW, there is a second dialog for entering the stations and offsets that define the ROW relative to the centerline. Use negative offsets for left and positive for right.



Alternatively, the **Enter Right of Way** and **Polyline to Right of Way** commands are other ways to define the ROW's for a centerline.

Exit: Exits this routine, prompting to save changes if necessary.

The dialog for every type of element shows the point ID, the northing, easting and station of the start point of the

element. It then allows the user to modify or define the parameters specific to the type of element. The following are some of the things to remember about data entry in the centerline editor. These are valid for lines, curves and spirals.

- Wherever length of the element is to be entered, entering an expression of the type $123.5 - 93.7$ would evaluate the difference of the values. This is particularly convenient where only the stations of the start and end points of the element are known.
- When the station is specified, the program takes the length of the element as the difference between the station of the start point of the element and the station specified.
- All bearings should be specified by entering the angle between 0 and 90 degrees (in dd.mmss format) and selecting the quadrant.
- When entering the delta angle of a curve, only the absolute value (between 0 and 360 degrees) is to be entered. The direction of the curve is to be explicitly set as right or left, the default being left. All angles are entered in (dd.mmss) format.
- Point numbers, when used, access their coordinates in the current .CRD file. If the point number specified has no coordinates stored in the coordinate file, the point number is remembered for that particular location (say the radius point of a curve or the SC point of a spiral). Then, when the .CL file is saved, the program creates points for that location and stores them to the .CRD file with the specified point number.

The dialog for a Line allows the user to specify the line primarily by its length or station and its bearing. The line can also be defined by its end point number or its coordinates. The bearing of a line can be changed if the Tangential to the Previous Element toggle is not checked. By default, any line which follows a curve element is defaulted to be tangential to it. To use a bearing different than that of the previous element, uncheck this toggle and enter the bearing.

The dialog for the Curve allows the user to define the curve primarily by its radius and delta angle or arc length. The other parameters of the curve that can be edited are the bearing of tangent-out and the "Station to", which also defines the arc length. The curve can also be specified by entering the coordinates or point numbers of its end point (PT) and the radius point. Another way to specify the curve would be to enter the chord length or PT point station and chord bearing. If the central PI point and a point on the forward tangent are known, then the curve can be defined by entering both of these points and at least one other property of the curve (like radius, arc length, delta angle). The point on the forward tangent can be any point that defines the tangent out direction including the next PI point. If only the central PI point is known, then the tangent-out can be entered by bearing instead of by forward tangent point. Central PI and forward tangent points are not displayed from the .CL file. They have to be entered by the user and are valid only for that particular edit session; that is, they are not remembered the next time the file is loaded. Curves are assumed to be tangent to the last element unless the Tangential to the Previous Element checkbox is cleared.

The Curve Edit Mode option defines how the curve is accepted in the centerline. If the Hold PC point is checked on, the radius is taken as fixed and the delta angle of the curve is calculated based on some additional parameter. Hence, the extent of the curve is unlimited. However, if the Hold PI points option is checked on, the bearing of tangent-out of the curve is taken as fixed and the radius is calculated based on some other parameter. In this case, the curve is completely restricted within the central PI point and the bearing of tangent out. Hence, when the Hold PI points option is checked on, the above parameters should also be defined to carry out the calculations.

The dialog for the Spiral-Curve-Spiral element allows the user to define the spiral by entering either the various parameters of the spiral (like the angles and lengths) or the coordinates or point numbers of its defining points: the TS (Tangent-to-Spiral), SC (Spiral-to-Curve), Radius point, CS (Curve-to-Spiral), ST (Spiral-to-Tangent) and end point (optional). While defining the spiral by its geometric properties, the program will accept the data even if the information for the simple curve is given with zero spiral lengths. In this method, however, the central PI point of the spiral MUST be specified (that is, it is always in Hold PI Points mode). The tangent out can be defined by entering bearing or by specifying a point on the forward tangent. This forward tangent point can be the next PI coordinates. The direction of the spiral-in and spiral-out elements would be the same as the direction of the simple curve (left or right). The Spiral Definition setting chooses between Arc definition for clothoid spirals and Chord for 10-chord spirals.

The spiral can be defined by several different parameters and the order that you enter data into the spiral dialog can

be important. There are two main sequences for entering data. The method to use depends on the spiral data that you have. The first method is to enter the radius of the simple curve, the spiral in and out lengths, the tangent bearing out and the PI station. The second method is to make a Line segment coming up to the TS (tangent to spiral) point. This Line segment should be added before creating the Spiral element. Then with the Spiral In point set to the TS point, enter the radius of the simple curve, the spiral in and out lengths, the curve direction (left or right) and the arc length of the simple curve. Then the rest of the spiral points will be calculated.

The Spiral Only element allows for flexible transitions from curve to spiral to curve or line to spiral to curve or between any combination of curve and line elements. The Spiral-Curve-Spiral element, for example, can be entered as Line, Spiral Only, Curve, Spiral Only and Line, producing the same results. You can spiral from tangent to curve, curve to tangent and curve of one radius to curve of another radius. You can also spiral from one endpoint to another endpoint. To define the spiral by sweep angle, use the Delta Angle field. To define the spiral by length, use the Spiral Length field. To define the spiral by end point, fill in the min and max radius fields and then enter either the End Point Pnt# or coordinates and the program will calculate the radius and spiral length to fit that point.

The screenshot shows the 'Spiral-Only Element' dialog box with the following values and settings:

- Spiral-In Northing: 682243.3051
- Spiral-In Easting: 2054761.0326
- Spiral-In Station: 1337.1826
- Spiral Length: 175.0000
- Delta (Sweep) Angle (DD.MMSS): 6.160021
- Spiral Direction: Left, Right
- Element At Spiral Out: Line, Curve
- Curve Radius At Spiral Out: 800.000
- Min. Radius: 600
- Max. Radius: 1100
- End Point#:
 - Pnt#: (empty)
 - Northing: 682308.5742
 - Easting: 2054598.7600
- Tangential to Previous Element
- Angle Format: NE, SE, SW, NW, AZ, Gon
- Bearing(dd.mmss): 66.000139
- Deflection Angle(DD.MMSS): 0.000000

Once all the elements of the centerline are defined, the file can be saved and then plotted using the *Draw Centerline File* command.

Here is an example of a highway interchange ramp that involves a starting tangent and a spiral curve that goes abruptly into a simple curve and then a final tangent. Start by entering a starting Northing and Easting and starting Station. The Start Point# is optional. Then the concept is that you click Add to add each subsequent element (line, curve, spiral-curve-spiral or spiral only):

Starting Coordinates

Start Point#: Start Station:

Start Northing: Start Easting:

Hold Other PI Points When Change Starting Point Zoom Drawing To Current Segment

Type of Curves

Roadway Railroad Show Right of Way

Type of Element	Ending Station	Northing	Easting	Line/Arc Length	Tangent-Out

Line (Tangent) Segment: We want to enter the tangent segment length up to the TS (tangent to spiral). Enter in the length (200.0), bearing (88.0732) and then the bearing quadrant (NW). Since the next spiral-curve-spiral element can be based on a PI station, it is not necessary for this line segment to go up to the TS point. The purpose of this line segment is to establish the tangent-in direction.

Line Element [X]

Start Point#:
 Start Point Easting: 4768889.0000
 Start Point Northing: 3736808.0000
 Start Point Station: 1200.0000

End Point#:

End Point Northing:

End Point Easting:

Length:

Station to:

Tangential to Previous Element

Angle Format

NE SE SW NW AZ

Bearing(dd.mmss):

Deflection Angle(DD.MMSS):

When OK is clicked, the routine will add the Line element as the first in the list of complete centerline elements. Next up is Curve-Spiral-Curve. Click Add.

Spiral-Curve-Spiral Element

Spiral In Point#: Spiral In Point Nothing: 3736814.5419
Spiral In Point Station: 1400.0000 Spiral In Point Easting: 4768683.1070

Spiral Definition:
 Arc Chord Hold PI Point

Data for Simple Curve:
Radius: Curve Direction: Left Right
Degree of Curve (DD.MMSS): Delta Angle:
Arc Length:

Central PI Point:
Point#: Length of Spiral-In: Delta (Sweep) Angle (DD.MMSS):
Northing: Length of Spiral-Out: Delta (Sweep) Angle (DD.MMSS):
Easting: Point on Forward Tangent: Tangent-Out: Angle Format: NE SE SW NW AZ
Station: Point#: Northing: Easting: Bearing(DD.MMSS):
Length: Length:

Tangent-to-Spiral Pt Spiral-to-Curve Pt Curve-to-Spiral Pt
Pnt#: Sta:1400.000 Pnt#: Sta:1475.000 Pnt#: Sta:1577.421
Northing: Northing: Northing:
Easting: Easting: Easting:

Spiral-to-Tangent Pt Simple Curve Radius Pt End Point
Pnt#: Sta:1652.421 Pnt#: Pnt#:
Northing: Northing: Northing:
Easting: Easting: Easting:

Horizontal Speed Table OK Cancel Help

Spiral Segment: Though the dialog is complex (for total flexibility), the key on a typical symmetrical spiral curve is to enter four things: (1) the radius of the simple curve, (2) the spiral in and out lengths, and (3) the tangent-out bearing. Everything else will calculate when you press Enter for the PI station.

Curve Segment: Add the next element and select curve. The Curve dialog appears. The key is to enter the Radius Length (255), the Arc Length (150) and the Curve Direction. Everything else will calculate.

Curve Element

PC Station: 1652.4212 Northing: 3736960.5769 Easting: 4768665.1318

Curve Edit Mode:
 Hold PC Point Hold PI Points Curve Direction: Left Right

Tangential to Previous Element

Radius: Station To:
Degree of Curve (DD.MMSS): Chord Length:
Delta Ang.(DD.MMSS): Chord Brg(DD.MMSS):
Arc Length: Format: Use Radial Angle

PT Point Radius Point
Point#: Point#:
Northing: Northing:
Easting: Easting:

Central PI Point Point on Forward Tangent
Point#: List Pick Point#:
Northing: Northing:
Easting: Easting:
Length (from PC): Length (from PT):

Bearing of Tangent-Out(DD.MMSS): Angle Format:

Horizontal Speed Table OK Cancel Help

Final Line Segment: All you need to enter in the final dialog for the line (tangent) segment is its length. All other items will calculate when you press Enter.

Line Element

Start Point#:
 Start Point Easting: 4768812.6754
 Start Point Northing: 3736970.0390
 Start Point Station: 1802.4212

End Point#: List Pick

End Point Northing:

End Point Easting:

Length:

Station to:

Tangential to Previous Element

Angle Format
 NE SE SW NW AZ

Bearing(dd.mmss):

Deflection Angle(DD,MMSS):

OK Cancel Help

The completed centerline will appear as shown in the dialog and each element can be edited. Pick the Save button to store this centerline data to a .CL file.

Centerline File: C:\Carlson projects\Stonebridge\Ramp.cl

Starting Coordinates
 Start Point#: Start Station: List
 Start Northing: Start Easting: Pick

Hold Other PI Points When Change Starting Point Zoom Drawing To Current Segment

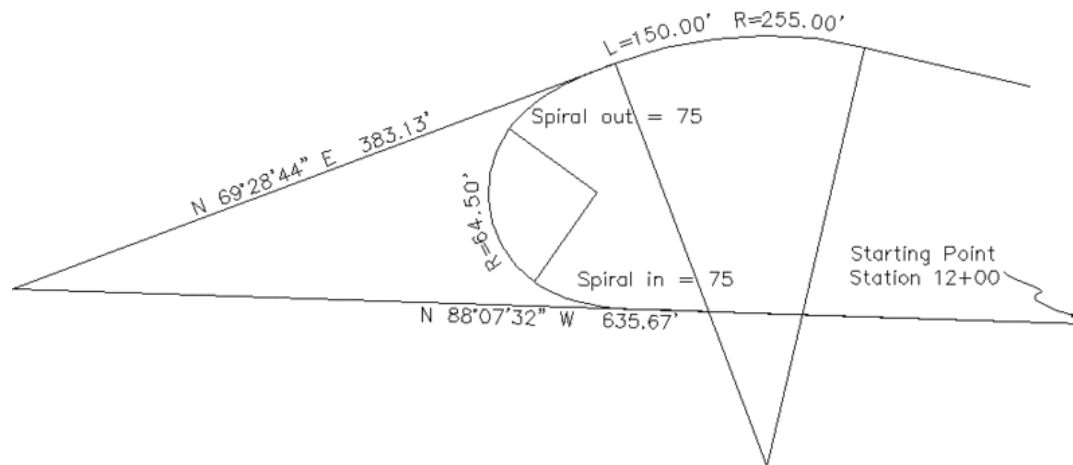
Type of Curves
 Roadway Railroad Show Right of Way

Type of Element	Ending Station	Northing	Easting	Line/Arc Length	Tangent-Out
Line	1400.000	3736814.542	4768889.107	200.000	88.0732 NW
Spiral-Curve-Spiral	1652.421	3736960.577	4768865.132	102.421	69.2844 NE
Curve	1802.421	3736970.039	4768812.675	150.000	76.4904 SE
Line	1902.421	3736947.234	4768910.040	100.000	76.4904 SE

Add Edit Remove Up Down

Load Screen Pick Tools Draw Save SaveAs Help

Fit Curve Fit Spiral Point Numbers Station Equations ROW Exit



Pulldown Menu Location(s): Centerline (Survey, Civil), Roads (Construction, Takeoff)

Keyboard Command: cledit

Prerequisite: None

Polyline to Centerline File

This command writes a centerline (.CL) file from a polyline in the direction the polyline was drawn. The Northing and Easting for each vertex of the polyline is written to the centerline file and each arc in the polyline becomes a circular curve. After selecting the polyline, the program shows the direction by drawing temporary arrows along the polyline. To reverse the direction of the polyline, there is a keyword option R for Reverse at the Command line. Also, the Reverse Polyline command can be used to switch the direction of a polyline.

For stationing the centerline, there is a Command line prompt for entering the station at the beginning of the polyline and then using the polyline segment lengths for the rest of the centerline stations. Alternatively, there is a keyword option E for Ending to specify the station at the end of the polyline and then back calculating the centerline stations to the beginning using the polyline lengths. Also, the keyword P for Point prompts to pick a reference point along the polyline and enter the station at this reference point.

In addition to being used as roadway/corridor "baselines," a .CL file can also be used as the horizontal control for a Template Point Centerline.

Note: To convert lines and/or arcs into a polyline, use the Entities to Polylines command or the Join Nearest command

Prompts

Centerline file to Write dialog *Enter the .CL file name to create*

Centerline station [Reverse/Ending/Point/<Beginning: 0+00>]: *Press Enter to accept the default station value specified or Type in the beginning station then press Enter*

Select polyline that represents centerline: *Pick the polyline that represents your centerline*

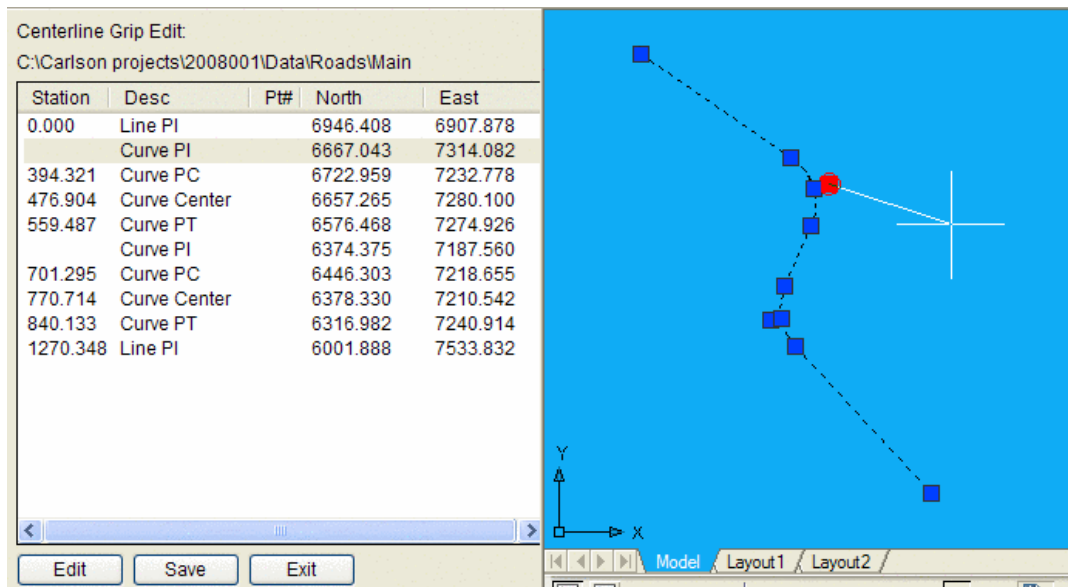
Pulldown Menu Location(s): Civil > Centerline, Survey > Centerline, Field > Roads

Keyboard Command: cpline

Prerequisite: A polyline drawn in the direction of increasing station values

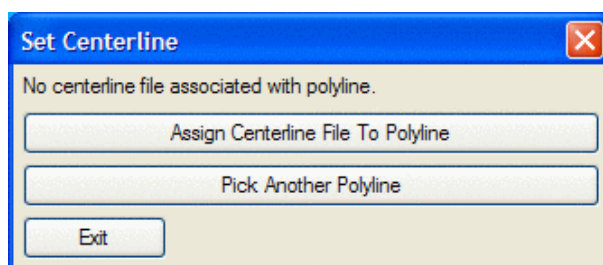
Edit Centerline On-Screen

This command allows the graphical editing of a centerline on the screen through a combination of grip editing and data editing in a docked dialog.



The command initially prompts you to select a polyline on the screen.

If you select a polyline that is not yet associated with a centerline file (.CL), you are prompted to assign a centerline file (.CL) to the polyline or to select another polyline.



Once the docked dialog appears, any component of the centerline geometry can be selected in the list and its data edited with the Edit button. However, the key points on the centerline can also simply be grip edited in the drawing, resulting in changes in the tabular data displayed in the docked dialog. Tangency between centerline components is maintained.

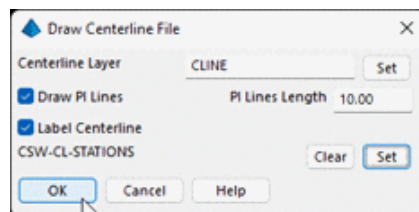
Pulldown Menu Location: Centerline

Keyboard Command: clgrip

Prerequisite: A polyline on the screen, which can either be already associated with a centerline file (.CL) or associated after picking

Draw Centerline File

This command reads a centerline (.CL) file and plots it as a 2D polyline in the drawing at the proper coordinates.



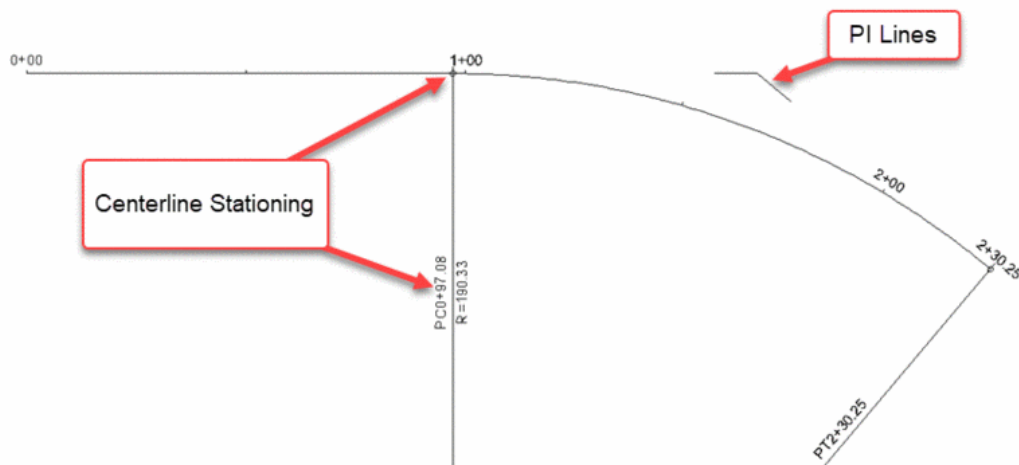
Centerline Layer is the layer name for the polyline to be drawn upon. The **Set** button can be used if you desire to select an existing layer from the current active drawing.

Draw PI Lines draws the straight line segments entering and exiting the PI. The length of the lines is controlled by entering a numerical value in the **PI Lines Length** option.

Label Centerline option draws station labels using a .STA settings file created by the Save Settings function in the Station Polyline/Centerline command.

Clear clears the current .STA file selection

When you click **<OK>**, you are prompted for the file name of the centerline(s) to plot. You can select single centerline (.cl) files individually or, by holding down the **Shift** or **Ctrl** keys, select multiple files at once. All centerline files are drawn with the same settings entered in the above dialog box.



The .CL file can be made with the following commands on the Design menu: *Polyline to CL File*, *Centerline Editor* or *Design Centerline*. Drawing the centerline file is a way to check the .CL file data graphically for correctness. If a spiral exists in the .CL file, the spiral will be represented by polyline segments.

Prompts

Draw Centerline Options dialog

Centerline File to Draw file selection dialog Select the .CL file name(s) to read and plot. Use the CTRL and/or SHIFT keys to select multiple Centerline file(s).

Pulldown Menu Location: Centerline

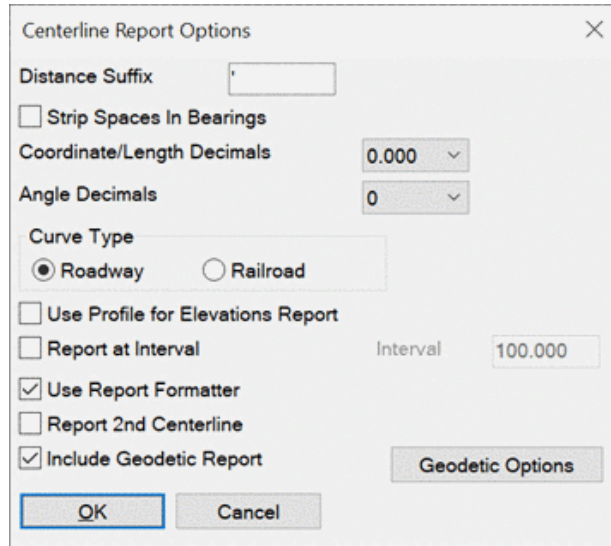
Keyboard Command: c12pline

Prerequisite: a centerline file

Centerline Report

This command reads a centerline file and creates a report in the standard report viewer which can be written to a file, a printer, or to your drawing. If the centerline file contains point numbers, then the report will include these

point numbers. If station equations are found, they are noted at the top of the report. The options dialog has settings for the report format and type of the centerline. The Use Profile for Elevations Report option will prompt you for a profile (.pro) file to add elevations to the report. The Report At Interval option will report stations, northing and easting at the specified station interval. The Use Report Formatter option lets you choose the report format and has output options for Excel. Also, the Report Formatter enables an expanded report with options for Report 2nd Centerline and Include Geodetic Report. The Report 2nd Centerline uses the second centerline for reporting offsets. The Geodetic Report adds geodetic distances and bearings to the report. It requires the grid projection to be defined in Drawing Setup.



Here is an example report:

```
Centerline Report
Centerline File: C:\sample\setback_3.cl

Station Northing Easting Bearing Distance
0+00.000 4033.165 4379.271
N 13°07'20'' W 92.076'
0+92.076 4122.836 4358.367 PC
Radius: 4196.621 4674.880 Radius Length: 325.000'
PI: 4159.044 4349.926 1+29.254 Tangent: 37.178'
Arc Len: 74.035' Delta: 13°03'07'' Right Degree: 17°37'46''
Chord Len: 73.875' Chord Brg: N 06°35'47'' W
Radial-In: N 76°52'40'' E Radial-Out: N 89°55'47'' E
Tangential-In Tangential-Out
1+66.110 4196.222 4349.881 PT
```

Pulldown Menu Location: Centerline
Keyboard Command: clreport
Prerequisite: A centerline (.CL) file

Centerline ID

Centerline ID reports the centerline file name and location that is associated with an alignment polyline. The subject polyline must have been created with either Design Centerline, Input/Edit Centerline, or Polyline to Centerline File.

When the routine is initiated and an alignment polyline is selected, the file associated with that polyline is reported at the command line. Additional alignment polylines may be selected without re-entering the command, or Enter may be pressed to exit the command.

Prompts

Select centerline polyline to identify: *pick the polyline*

Centerline Name: D:\SAMPLE.CL

Select centerline polyline to identify (Enter to end): *press Enter*

Pulldown Menu Location: Centerline

Keyboard Command: CL_ID

Prerequisite: A polyline created from a Design Centerline, Input/Edit Centerline, Polyline to Centerline File, or Centerline File to Polyline.

Station Polyline/Centerline

This command will station a polyline or centerline file at a given interval distance. The options for this command are set in the dialog shown below. After setting the options, click OK on the dialog and then pick the polyline or select the centerline file. All settings can be saved as (.STA) files and loaded for reuse, and for storing multiple stationing schemes. Polyline/Centerline station labels are also dynamic, and so will update when changes are made in the geometry.

Station Polyline/Centerline dialog box showing various settings for stationing, including Distance For Stations (100.000), Distance For Intermediate Stations (50.000), Beginning Station (0.000), and options for Locate, Label Station Text, and Station Type.

Distance for Stations is the primary interval for stationing. On Curve allows for a different interval for curve segments versus line segments.

Distance for Intermediate Stations is the intermediate interval for stationing. On Curve allows for a different interval for curve segments versus line segments.

Beginning Station is the beginning station of the centerline for stationing.

Locate Even Stations labels the stations at the distance interval (i.e. 2+00, 3+00, etc.).

Locate Odd Stations labels the non-interval stations at the polyline/centerline end points and PC and PT points.

Locate User-Entered prompts you for individual stations to label.

Locate Start/End Stations labels the Start and End Station as specified when the option Specify Start/End Stations is enabled.

Without the **Increment Station Labels from Beginning** Station option, the program increments the station labels from zero. For example, if the station interval is 100 and the polyline starting station is 145, then the program will label 2+00, 3+00, etc. With this option active, the station labels are incremented from the starting station. In this example, the program would then label 2+45, 3+45, etc.

Label Deflection Angles adds deflection angles to centerlines without arcs. Settings for this are specified in the **Label Deflections Setup**, accessed by the Deflections Setup button.

The screenshot shows the 'Label Deflections Setup' dialog box. It features a title bar with a close button. The main area contains the following controls:

- Include Station:** A dropdown menu currently set to 'Off'.
- Station Prefix:** An empty text box.
- Suffix:** An empty text box.
- Use MText:** A checked checkbox.
- Fields on separate rows:** An unchecked checkbox.
- Label North/East:** An unchecked checkbox.
- Northing Prefix:** A text box containing 'NORTH:'.
- Northing Suffix:** An empty text box.
- Easting Prefix:** A text box containing 'EAST:'.
- Easting Suffix:** An empty text box.
- Use Symbol for Delta Angle Label:** An unchecked checkbox.
- Deflection Left Prefix:** A text box containing 'DL='.
- Deflection Right Prefix:** A text box containing 'DR='.
- Suffix (for both DL and DR):** Two empty text boxes.
- Deflection Label Position:** A group box containing three radio buttons: 'Perpendicular' (selected), 'Horizontal', and 'Parallel'.
- Deflection Angle Precision:** A dropdown menu set to 'ddd.mmss'.

At the bottom of the dialog are 'OK' and 'Cancel' buttons.

Include Station sets the station name as None, Prefix or Suffix to the deflection angle.

Station Prefix and **Suffix** are added to the Station Label.

Use MText option will create the deflection angle labels as Mtext.

Fields on separate rows will place each line of text on a separate layer.

Label North/East option adds a Northing and Easting coordinate to the PI location.

Use Symbol for Delta Angle Label adds a triangular shaped symbol as a prefix to the deflection angle.

Deflection Label Position controls the position of the label as Perpendicular, Horizontal or Parallel.

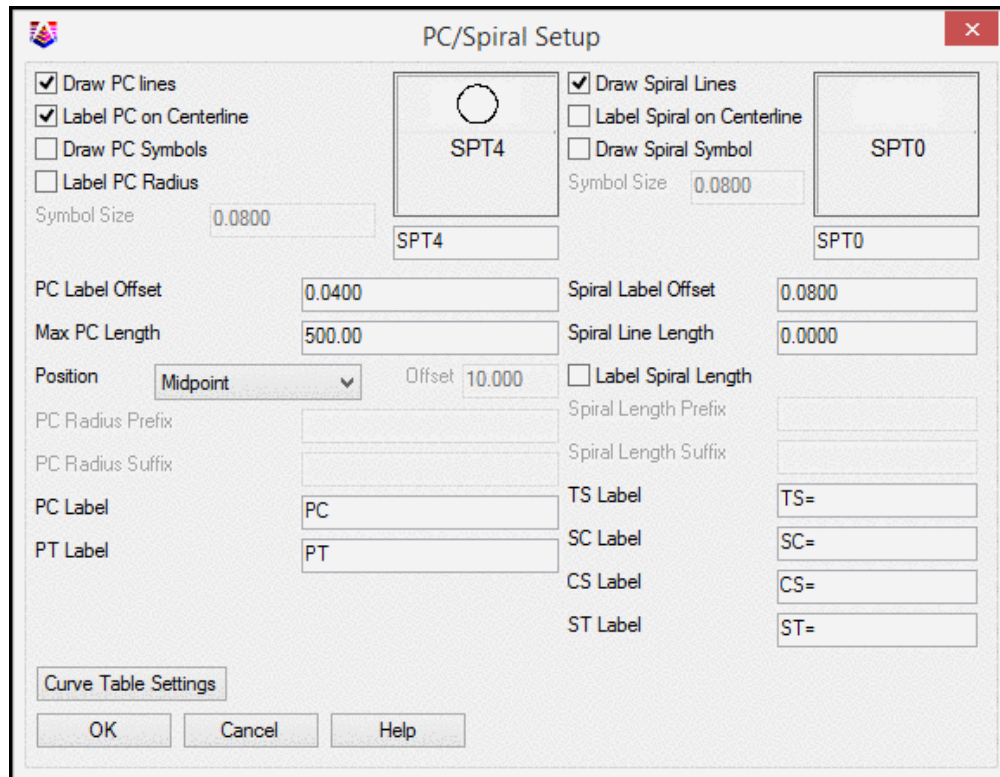
Deflection Label Alignment sets the text justification for the labels.

Deflection Angle Precision controls the format of the angle.

When **Specify Start/End Stations** is checked, only the stations between and including the specified starting and ending stations will be labeled. If locate centerline points and offset points are toggled on, only points within the specified stations will be located.

When **Erase Previous Station Labels** is checked, previous station labels are erased when new ones are generated.

The **PC/Spiral Setup PC** button accesses the **PC/Spiral Setup** dialog, where settings are controlled for lines and/or symbols and/or labels at the starting and ending (PC and PT) stations of an arc of the centerline as well as for the spiral special stations (TS, SC, CS, ST).



Label on Radial Lines controls whether to draw a perpendicular line and label the station.

Label On Centerline when checked, the station of the PC and PT will be labeled on the centerline as well as the PC and PT lines. When not checked only the PC and PT lines will be labeled.

Draw PC Symbols controls whether symbols are placed at these locations. If checked, the desired symbol is selected by picking on the symbol box.

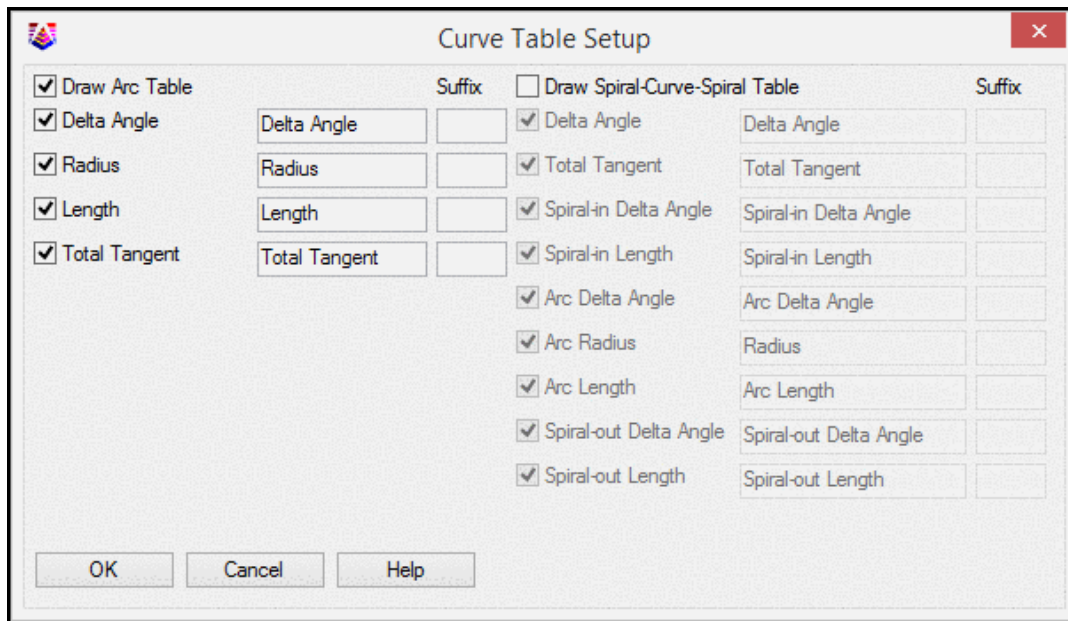
Label PC Radius controls whether this point is labeled.

Max PC Length controls the maximum length for the PC lines to be drawn described above.

Position controls the placement of the PC and PT labels as either the midpoint of the PC line or at a user defined offset from the centerline.

Prefix/Suffix: Each special station has a prefix/suffix string that can be added to the station label.

Curve Table Settings controls which elements will be labeled or placed in a data table for both arcs and spirals as shown in the **Curve Table Setup** dialog box.



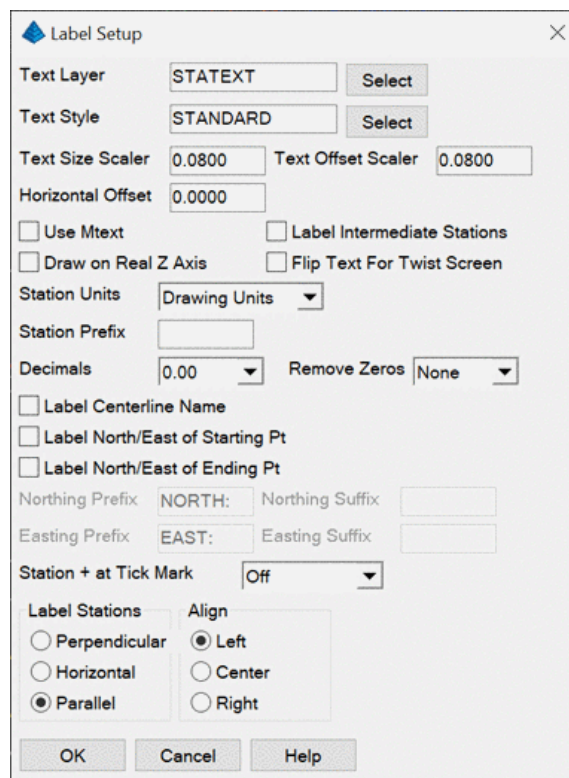
Draw Arc Table when enabled, creates and draws a table containing the arc information selected. **Delta Angle**, **Radius**, **Length** and **Total Tangent**.

When **Label PI Stations** is checked, the PI station is labeled at the PI point.

When **Locate PI Points** is checked a point will be created at the PI of a horizontal curve graphically and written to the active coordinate file.

When **Label Station Text** is checked, this command places station text along the polyline at the angle of the corresponding segment. After toggling this option on, the Label Setup button will become available for selection.

The **Label Setup** controls the placement, precision, font and size of station labels along the centerline.



Text Layer is the user-specified layer for text labels to be drawn on.

Text Style is the user-specified text style for labels.

Text Size Scaler determines the size of the station labels. This value multiplied by the horizontal scale setting in Drawing Setup results in the size of the label. For example, if the horizontal scale is set to 100 and the text size scaler is set to 0.10, the station labels will be 10 units.

Text Offset Scaler works like text size scaler above controlling the distance the text labels will be offset from the centerline.

Station + at Tick Mark labels the station text along the polyline with the '+' of the station text at the station's location on the polyline. See Marker Set up for marker size manipulation settings. The **On With +** option draws the label with the "+". The **On Without +** option draws the label without the "+".

Horizontal Offset shifts the station label along the centerline.

Use **MText** controls whether to create the labels as MText or regular Text entities.

Draw on Real Z Axis creates the labels and tick marks at elevation when using a 3D Polyline for the centerline.

Label Intermediate Stations: If the intermediate distance is the same as the station distance then no intermediate station ticks or labels will be drawn. For example, with the above entries and 0+00 for the first station the stations will be labeled with descriptions as follows: 0+00 0+50 1+00 1+50, etc.

If the **Flip Text For Twist Screen** setting is checked and the drawing has been twisted using the twist screen command, the label text will be flipped to read in the proper direction of the stationing.

Station Units when set to Miles divides the station values by 5280 for creating the station labels. When set to Kilometers, it divides the station values by 1000 for the labels.

Station Prefix adds to the front of the station labels.

Remove Zeros removes the specified number of least significant digits from the station label if these digits are all zero.

Label Centerline Name creates a label of the centerline name at the start of the centerline when using a CL file as the input.

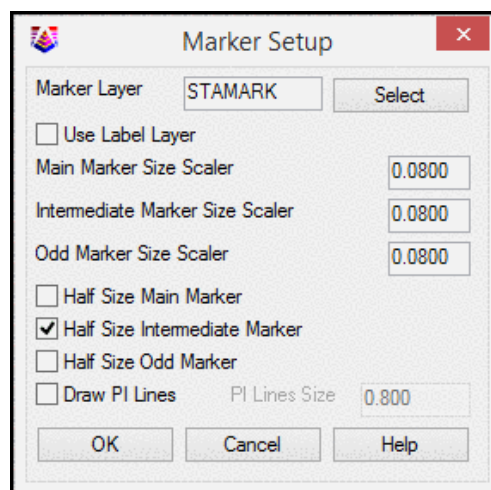
Label Northing/Easting toggles add coordinate labels at the start and/or end of the centerline including prefixes and suffixes as specified.

Decimals determines the number of decimal places of the stationing labels to be drawn for the odd stations and user entered stations only.

Use **Label Stations** to specify whether to label the stations perpendicular or parallel to the centerline.

Align determines the alignment of the station label, either left or centerline, centered along the centerline or to the right of the centerline. This option is only available when using the perpendicular option for station labels.

The **Marker Setup** options control the size of markers for different station types as well as the layer the markers will be drawn on.



The **Marker Layer** specifies which layer the station marks will be placed.

Use **Label Layer** option sets the label for the markers to be the same as that of the labels as set above.

The **Half Size Main Marker**, **Half Size Intermediate Marker** and **Half Size Odd Marker** options draw a perpendicular tick mark on only one side of the centerline. Otherwise a full marker is drawn that goes on both sides of the centerline.

Draw PI Lines option will draw a line in the direction of both tangents, in and out of the PI at the size designated in the **PI Lines Size**.

Specify whether to define the **Centerline By** picking a 2D polyline or 3D polyline in the drawing or selecting a centerline (.CL) file.

- Using a **2D Polyline** will result in horizontal distance stationing along the polyline.
- Using a **3D Polyline** will result in the slope distance stationing along the polyline.
- Using a **CL File** will result in horizontal distance stations as with the 2D Polyline option only a prompt for the centerline to use will display.

Use **Station Type** to specify the stationing format to use.

Use **Type of Curves** to specify whether you are labeling a roadway curve (arc definition) or railroad curve (chord definition).

Locate Centerline Points will locate points and store them in the current CoORDinate file.

Locate Radius Points will locate the radius points of any arc segments.

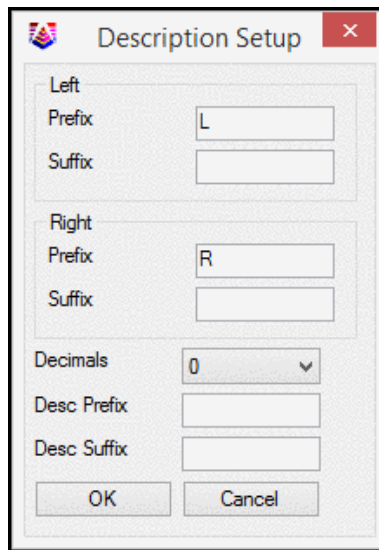
Starting Point Number determines the starting point number for the points to be located.

Vertical Exaggeration applies to Profile Polyline mode. This factor is the ratio between the horizontal and vertical scales on the profile grid.

There are two ways to **Set Elevations** for the centerline points and offset points to be created.

- The **3D Polyline** option gets the elevation of the point from a specified 3D Polyline within the drawing.
- The **Profile** option will determine the elevation of the point based upon the same station in the profile file. You will be prompted for the profile file to read for the elevation reference.
- With the **None** option selected, no elevations will be determined for the points.

When **Include Station in Description** is checked, the station along the centerline will be included in the resulting offset point description field.



Left Prefix or Suffix is added to the left offset label

Right Prefix or Suffix is added to the right offset label

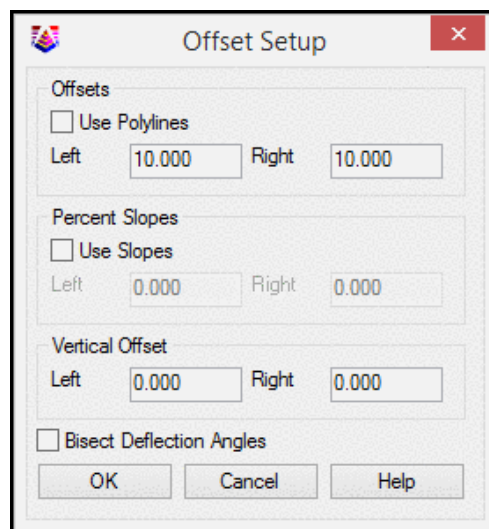
Decimals controls the label precision

Description Prefix is an optional user-specified prefix to be added to the point description.

Description Suffix is an optional user-specified suffix to be added to the point description.

When **Label Sta Equations** is checked on any station equation, contained in a centerline (*.cl) file will be labeled. This option is only available when stationing a centerline file (*.cl).

Locate Offset Points will create points at the specified left and right offset distances from the centerline. Options for setting the elevations and descriptions of the points are available from the Offset Setup dialog.



Use Polylines allows you to select an offset reference polyline.

Locate Odd creates offset points at vertices of reference offset polylines.

Left and **Right** offsets allow a user defined offset amount.

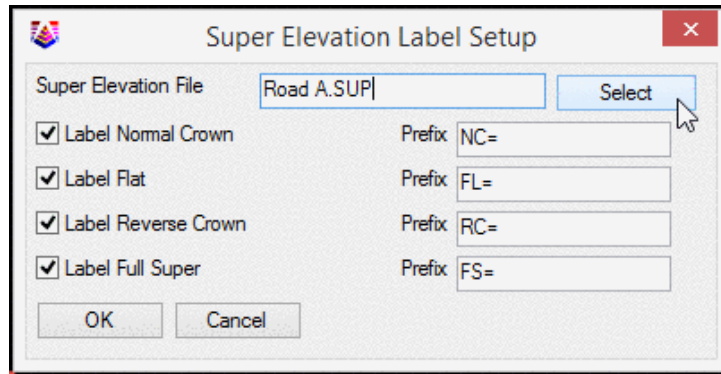
Percent Slopes allows you to project an offset grade based on a slope from the corresponding centerline points.

Vertical Offset allows you to define a separate vertical offset for the left and right horizontal offsets.

Bisect Deflection Angles controls how the offsets are located at angle points. When enabled, two offset points both

90 degrees from their respect centerlines are located.

Label Super Elevation: This option labels the super elevation transition stations as defined in the specified .SUP file. You can choose which types of the transition stations to label and set the prefix for each type.



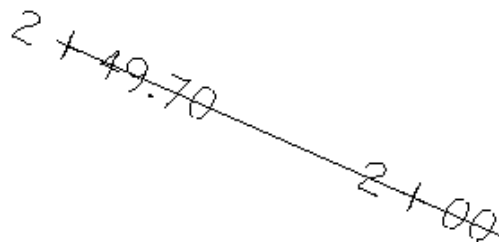
Use **Select** to select a super elevation file (.SUP). This file is created as part of the Roads Menu contained in the Civil Module.

Prompts

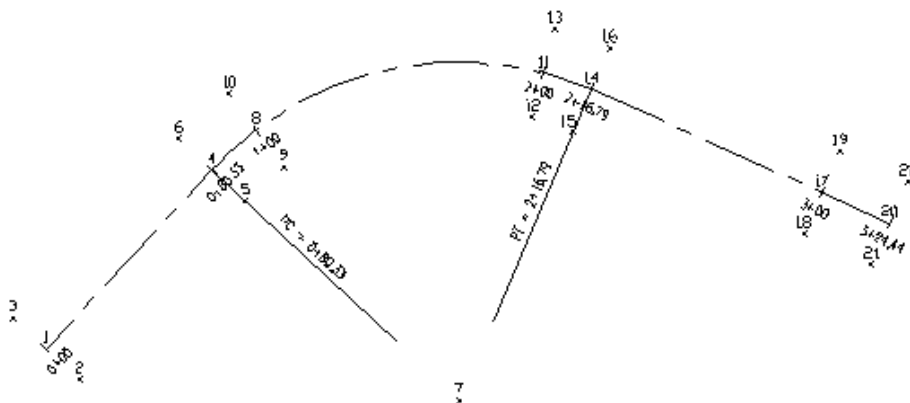
Station Polyline Dialog

Polyline should have been drawn in direction of increasing stations.

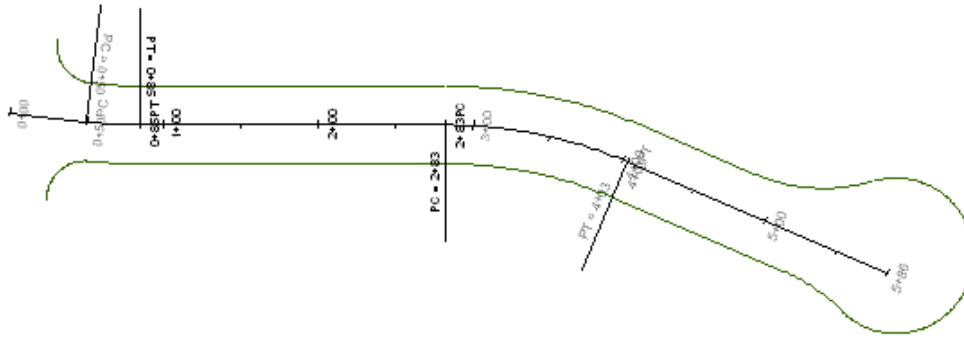
Select polyline that represents centerline: *select a polyline*



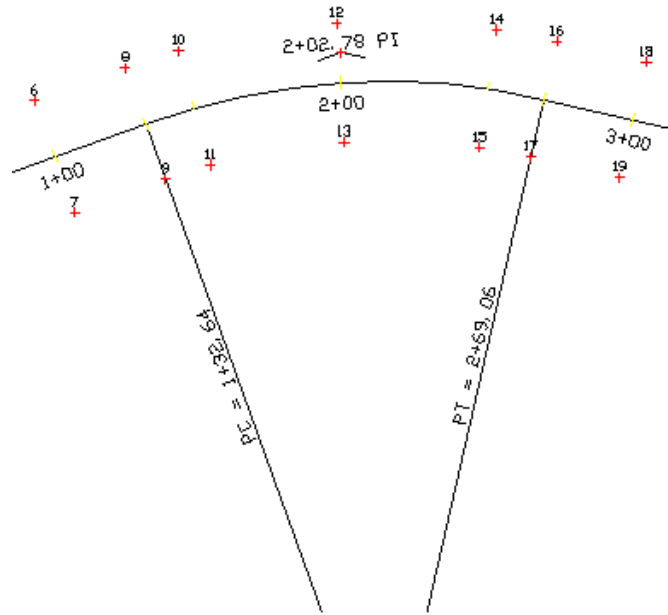
Closeup of Station + at Tick Mark option



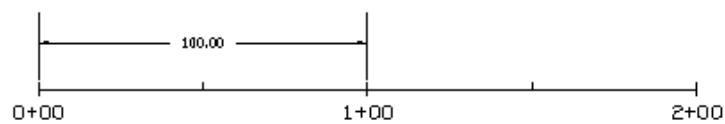
Labels with Label PC on Centerline checked on



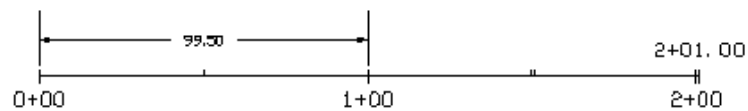
Labels set to perpendicular and Max Length of PC lines set to 75.0



Labels with Draw PI Lines, Label PI Stations and Locate PI Points all checked on



Labels using Centerline By 2D Polyline (Horizontal Station)



Labels using Centerline By 3D Polyline (Slope Station)

Pulldown Menu Location: Centerline

Keyboard Command: stapl

Prerequisite: A polyline or CL file

Label Station-Offset

This command will compute and label the stations, offsets and elevations of selected points or entities. Additional labels for the names of the reference alignments and point data (point name, northing, easting, latitude, longitude, description) can also be specified and placed.

A common usage for using dual alignments and profiles typically involves the alignment and profile of a road coupled with the alignment and profile of a pipe/utility.

1st/2nd Alignment: Specify the criteria for either one or two alignments that will be used for the label(s) that will be placed into the drawing.

Use 2nd Alignment: Enable this toggle if multiple alignments are to be used for the label(s) that will be placed into the drawing.

Name: Supply a label-friendly value for the name of the alignment (*e.g.* "King Street" or "Water Main"). The value(s) specified get assigned to the **Alignment** Label Field.

Centerline: Indicate the source (Polyline or Centerline File) for the reference alignment. If the Polyline option is selected, you will be prompted to select the polyline(s) after the OK button is pressed. If CL File option is selected, supply a valid path and file name for the centerline file or navigate to the file using the "File Picker" button. The Beginning Station will be determined from the selected Centerline File.

Beginning Station: Specify the beginning station of the centerline. The polyline should be drawn in the order of increasing stations. This control is not used when you use a centerline (.CL) file to define the centerline as the starting station of the centerline is stored in the .CL file.

Vertical Reference: Indicate the source (3D Polyline, Profile File, Road Network or Surface File) for the reference elevation. With a Vertical Reference, there are label fields to label the Elevation Reference and Cut/Fill. With 3D Polyline, there will be an additional Slope Station available under the Label Fields in addition to the regular

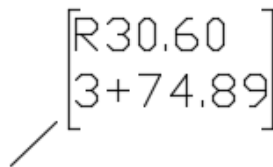
horizontal distance station. If the Profile option is selected, supply a valid path and filename for the profile file or navigate to the file using the "File Picker" button shown above. For the Road Network, specify the road network (.rdn) file with the "File Picker". With the Road Network method, the program will find the road design surface elevation for the specified points using all the road network design files including profiles, templates and transitions. For Surface File, the program will prompt for selecting a triangulation or grid surface model.

Cross Slope (%): Indicate the slope as a percentage to "travel" from the **Vertical Reference**. A value of 0 (zero) will not apply any cross slope from the reference elevation. Positive values will decrease the calculated elevation(s) and negative values will increase the calculated elevation(s).

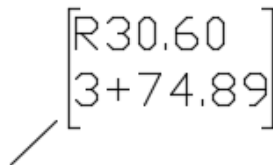
Vertical Adjustment: Indicate the desired amount of vertical displacement that should applied to the calculated elevation. This is useful when deriving elevations for back or face of curb.

Label Alignment: Specify whether the labels should be Horizontal on the screen, Vertical on the screen, Parallel to the Centerline, Perpendicular to the Centerline, or user-specified by Picking.

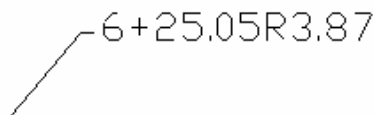
Label Brackets: Options to draw brackets around the labels with different styles including Square, Parenthesis, Arrow and Curly. The Single, Double and Triple options are for how many brackets to draw.



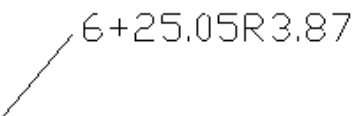
Text Placement Options: Controls how the leader is drawn with the label. The Above/Below Leader method draws the leader along the label. The After Leader draws the leader up to the label.



Above/Below Leader



After Leader with Tick



After Leader No Tick

Text Size Scaler: Determines the size of the labels. This value multiplied by the horizontal scale setting in *Drawing Setup* results in the size of the label. For example, if the horizontal scale is set to 100 and the text size scalar is set to 0.10, the labels will be 10 units.

Text Offset Scaler: Controls the distance between the leader and the label.

Text Offset Scaler: Determines the scalar to offset top, and or bottom label(s) from the label line base point (i.e. the end of the leader line). This value is multiplied by the horizontal scale setting in *Drawing Setup* to determine the on screen label(s)' text offset up/down. (NOTE: Only available when top, and or bottom label(s) text is offset up, and or down. Not available when drawing labels as MLeader objects)

Text Style: Specify the desired text style for the label.

Use MLeader: Creates an MLeader which combines the leader with the label.

Leader Segments: Specify the desired number of leader segments that should be allowed when constructing the label.

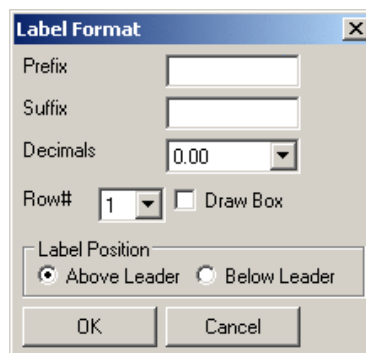
Use Relative Leader: Indicate whether successive labels placed into the drawing should re-use the geometry of the initial leader placed with the command.

Draw Leader Arrow: Indicate whether to draw an arrowhead on the leaders.

Draw Line From Centerline: This option draws a perpendicular line between the point and the centerline. The Setback Offset option shortens the line and makes gaps at the centerline and point ends. The Set Label Adjacent option places the label along this line instead of at the leader endpoint.

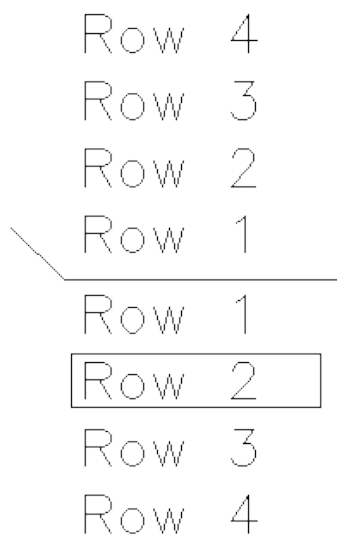
Draw At Fixed Position: After you pick the first label position, the rest of the labels will be placed at this same level. This option applies to the Vertical and Horizontal Label Alignment methods.

Label Fields: Use the green arrow buttons to specify the items that are to appear in the labels. As labels are "moved" from Available to Used, a Label Format dialog box particular to the label will appear that will allow for more precise display control. To subsequently edit each item, use the Format Editor button as shown below.



Note:

- The Row Number value is specified as the row starting closest to the leader with subsequent rows moving further from the leader as shown in the figure below. Row 2 below the leader has been illustrated with the Draw Box option enabled.



Layers: Specify the layer of each item that comprises the label.

Max Offset to Calc: Specify the maximum offset to calculate.

Truncate Station at +: Removes the digits before the + in the station labels.

Station Type: Specify the stationing format to use.

Station Units: When set to Miles divides the station values by 5280 for creating the station labels. When set to Kilometers, it divides the station values by 1000 for the labels.

Add to Existing Point Description: When picking points to label by point #, this option appends the label to point description instead of creating a text label. The description is updated both in the coordinate file and for the point description attribute in the drawing.

Flip Text for Twist Screen: When this option is enabled, the label(s) text will be flipped as necessary to adjust for the use of Twist Screen. When this option is disabled the normal, and forward direction of the Centerline will be used to determine an up direction for drawing the label(s) text. Applicable to Perpendicular, Parallel, and Pick Label Alignment.

Offset Tolerance: For points with an offset greater than the specified tolerance, this option puts the labels in a separate layer. This option is a way to highlight points with offsets more than the tolerance. The separate Offset Tolerance layer can be setup to use a different color.

Type of Curve: Specify whether the centerline is for a roadway or railroad. Stationing for Roadway Curves is measured along the curve length itself whereas stationing for Railroad Curves is measured along chord segments.

Save: Allows the current settings to be saved to a Station-Offset Settings (*.sos) file.

Load: Allows settings from a previously saved Station-Offset Settings (*.sos) file to be recalled for use.

Prompts

Polyline should have been drawn in direction of increasing stations.

Select Polyline Centerline (Alignment-1): *Pick the polyline centerline* This prompt will not appear if the Centerline File option was specified.

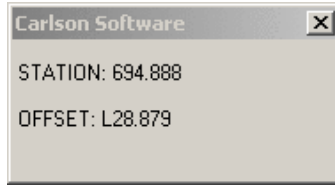
Select 3D Polyline Profile (Alignment-2): *Pick the polyline profile* This prompt will not appear if the Profile File option was specified.

Pick point or point numbers (SS for Selection Set,G for Group,Enter to End): Pick a point

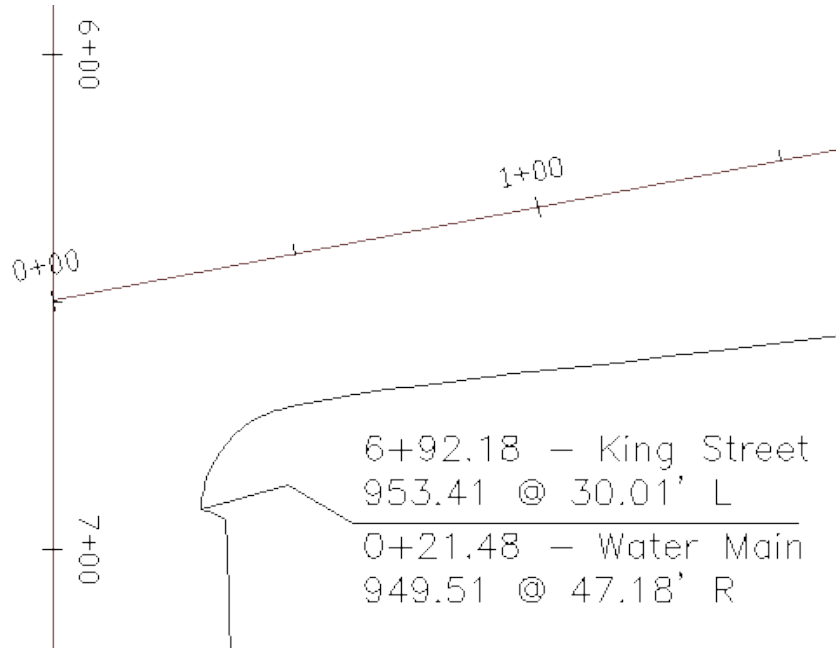
Pick point to label: *Pick a leader vertex point*

Pick label alignment: *Pick angle for the label* This prompt will only appear if the Pick option was specified.

Pick point or point numbers (SS for Selection Set,G for Group,Enter to End): Press Enter



Real-time display of Station and Offset as you move the cursor.



A sample label with a 2-segment leader.

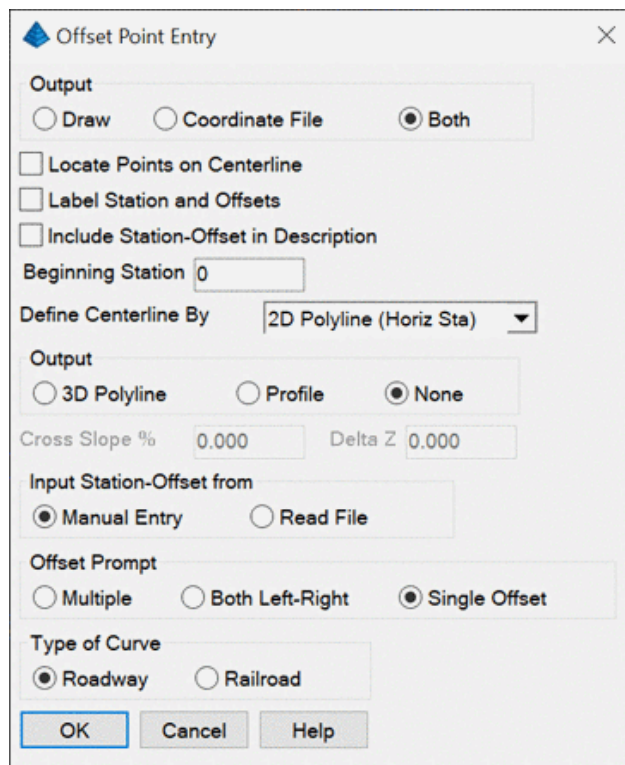
Pulldown Menu Location: Centerline

Keyboard Command: offsta

Prerequisite: A polyline or centerline file

Offset Point Entry

This command creates points along a centerline at specified stations and left and right offsets. The centerline can be defined by a polyline, centerline (.CL) file or two points.



The **Output** option to **Coordinate File** will store any points the current coordinate (.CRD) file. This includes centerline points and offset points. The option to **Draw** will draw the new points using the settings from Points > Point Defaults.

When **Locate Points on Centerline** is checked, the program will locate points along the centerline, otherwise just the offset points will be created.

When **Label Stations & Offsets** is checked, the program will label the station-offset as the point description attribute.

The **Include Station-Offset In Description** option will add the station and offset of the point into the point description.

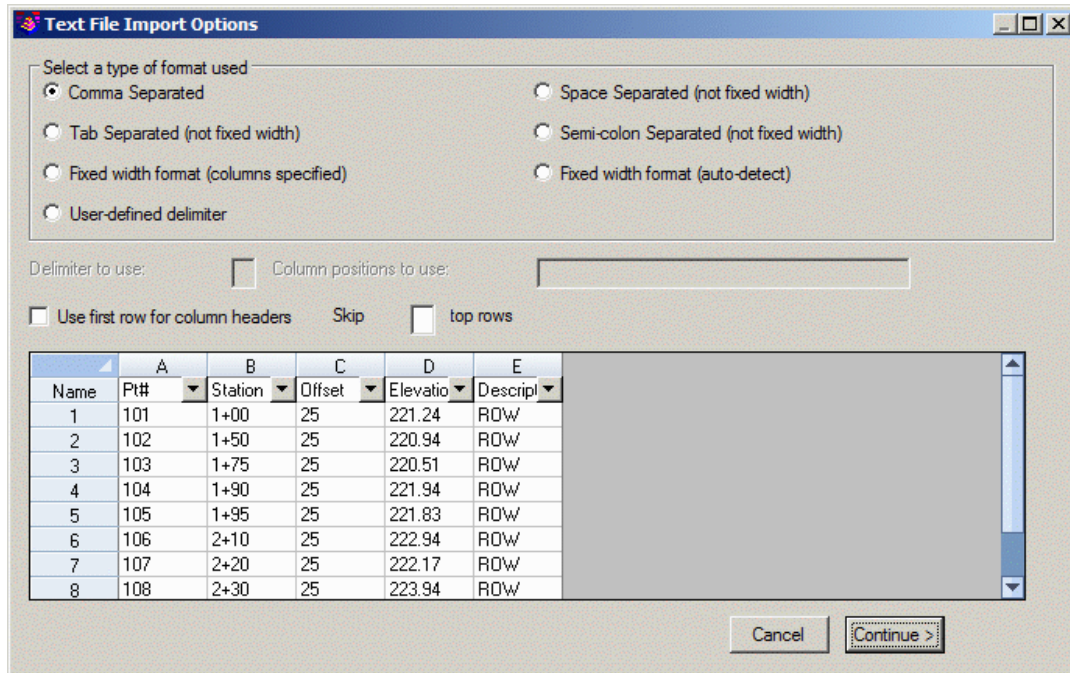
Beginning Station: Enter the Beginning Station of the Centerline.

Use **Define Centerline By** to specify whether to define the centerline by picking a polyline in the drawing, selecting a centerline (.CL) file, or using 2 points. For polylines, the 2D Polyline method works with horizontal distance stations and ignores any polyline elevations, and the 3D Polyline method works with slope distance stations using the polyline elevations.

Use **Reference Elevation** to assign elevations to the points created when locating points on the centerline of offset points. When using a 3D Polyline for the elevation reference, points will be created at the station entered and the offsets specified with the elevation of the same station along the 3D polyline. The Profile option will do the same as the 3D Polyline option only it will use a profile file for the elevation reference. You will be prompted for the profile to use for the elevation reference. None simply creates 2d point data on elevation zero. The Reference Elevation option is good for creating points along the centerline for final grade elevation points. *Profile to 3D polyline* can be used to transfer the profile data to the polyline before calculating the final grade points.

Cross Slope %: This option is used to alter the elevations of the new points by applying either a Cross Slope calculation or a Delta Z variable.

The Manual Entry option in **Input Station-Offset from** will prompt for the station and offset distances. The **Read File** option will read the stations and offsets from a text file. The text file format with point number, station, offset, elevation and description. The program handles station formats with or without the '+' (i.e. either 250 or 2+50). The elevation and description are optional. The Read File option is a quick routine to convert a station-offset data file into coordinates. The delimiter for the text file and the order of the fields are set in the dialog shown here.



When **Offset Prompt** is set to Both Left-Right, the program will prompt for left and right offsets. If you respond to an offset prompt with zero (0), no offset point is created. The Single Offset option will prompt for one offset per station. Enter a right offset with a positive value and a left offset as a negative value. The Multiple option keeps prompting for offsets at the current station until you enter a blank offset. This option applies when you have more than one offset to create on the left or right side.

Use **Station Type** to specify the stationing format to use.

Use **Type of Curve** to specify whether the curves are for a roadway or railroad.

Prompts

Offset Point Settings Dialog

Polyline should have been drawn in direction of increasing stations.

Select Polyline near endpoint which defines first station.

[nea on] Select Polyline to Station-Measure: *select a polyline*

(5309.0 4845.0) Station: 0.00

(5526.0 4917.0) Station: 228.63

Distance from beginning station along centerline (Enter to end): 110

Starting Segment Station: 0.0 Ending Segment Station: 228.633

Working Line segment...(5413.4 4879.64 0.0)

Left offset distance <10.0>: 15

Right offset distance <15.0>: 20

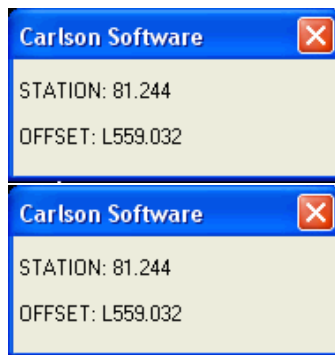
Distance from beginning station along centerline (Enter to end): *press Enter*

Keyboard Command: offpts

Prerequisite: A centerline (.CL) file, polyline, or two points

Calculate Offsets

This command calculates the station and offsets of point coordinates relative to a centerline. The points to calculate can be stored in a coordinate (.CRD) file or picked on the screen. As the crosshairs are moved, the station and offset of the current position are displayed in real-time in a small window (see example). After entering the points to calculate, the program prompts for selecting another centerline which is optional in case you want a report for multiple centerlines.



Beginning Station: Specify the beginning station of the centerline. The polyline should be drawn in the order of increasing stations. Not available when you use a centerline (.CL) file to define the centerline.

Maximum Offset to Calc: This is the maximum distance from the Centerline for which offsets are calculated.

Report Offsets Ahead/Behind Centerline: When checked, this option shows offsets for points or picked points located before the beginning station and after the ending station of the centerline.

Label Station and Offsets: When checked, the station offsets will be labeled in the drawing.

Label Cut/Fill: When using a reference elevation from the Report Grade Elevation setting, this option labels the cut/fill between the reference elevation and the points.

Cut/Fill In Inches: Sets the precision for reporting cut/fill when using inches or whether to report in decimals.

Sort Report by Stations: When checked, this option will report the station-offsets in station order no matter what order the points were calculated.

Report Point Coordinates: When checked, this option will include the point northing and easting in the report.

Report Point Notes: When checked point notes will be included on the calculate offset report.

Create Point Notes: When checked, the station and offset of the offset point will be created as notes and written to a note file (*.not). This note file will have the same name as the crd file.

Use Report Formatter: When checked, the output of this command is directed to the Report Formatter which allows you to customize the layout of the report fields and can be used to output the data to Microsoft Excel or Microsoft Access. You must check this option on in order to use the Report Grade Elevation From option.

Round Stations: When checked, this option will round the stations for the selected points on the report to the Rounding Interval specified. For example if an offset point is located at station 1+01, and the rounding interval is set to 10, then the report will show the offset point at station 1+00.

Left/Right Prefix/Suffix: These settings are used with the offset label.

Store Station Text to CRD File: When checked, the station offset text is appended to point numbers that are selected.

Report Grade Elevation From: When checked, this option will calculate an elevation for each point from a 3D polyline, grid file (.grd) or triangulation (.flt) file. To Use this option, the *Report Formatter* must be toggled on. The grade elevation is reported and compared with the point elevation to report the cut/fill. For the 3D polyline option, the grade elevation is calculated by finding the elevation at the point on the 3D polyline that is the nearest perpendicular position from the offset point. The 3D polyline that is used for elevations does not need to be the same polyline that is used as the centerline for the station-offset calculations.

Define Centerline by: Specify whether to define the centerline by picking a polyline in the drawing, selecting a centerline (.CL) file, by a point and direction angle, or using 2 points. The polyline mode can be either 2D or 3D for horizontal or slope distance stationing.

Station Type: Specify the stationing format to use.

Decimals: Specify the display precision for the stations and offsets.

Type of Curve: Specify whether the curves are for a roadway or railroad.

Prompts

Calculate Offset Settings Dialog

Polyline should have been drawn in direction of increasing stations.

Select Polyline near endpoint which defines first station.

[nea on] **Select Polyline Centerline:** *select polyline centerline*

(5309.0 4845.0) **Station: 0.00**

(5526.0 4917.0) **Station: 228.63**

PtNo. North(y) East(x) Elev(z) Description

140 4889.13 5410.25 0.00 1+10.00L10.00

Station on Line> 1+10.00 Offset> 10.00 Left

PtNo. North(y) East(x) Elev(z) Description

141 4870.15 5416.55 0.00 1+10.00R10.00

Station on Line> 1+10.00 Offset> 10.00 Right

+ before station denotes point is ahead of line segment, - denotes beyond.

Pick point or point numbers (Enter to End): 22-28

Station	Offset	Description	Elev	Pt#	North	East
4+95.89L	15.48	Catch Basin	0.00	22	4811.00	4454.00
5+78.43L	58.18	Power Pole	0.00	23	4839.00	4548.00
6+77.26L	57.28	Power Pole	0.00	24	4868.00	4656.00
9+01.55R	16.81	Catch Basin	0.00	25	4745.00	4887.00
10+50.51L	25.39	Traffic Sign	0.00	27	4872.00	5043.00
4+03.48R	22.15	Light Pole	0.00	28	4657.00	4454.00

Pick point or point numbers (Enter to End): *press Enter*

Pulldown Menu Location: Centerline

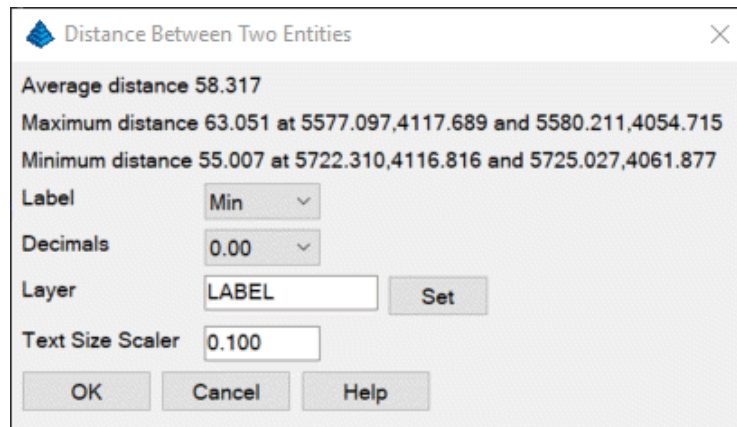
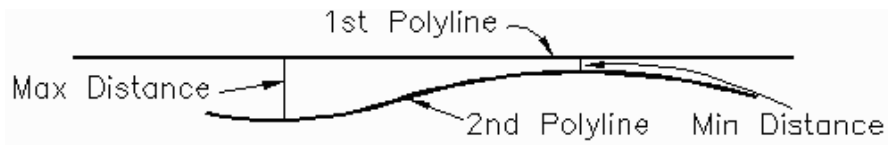
Keyboard Command: calcoff

Prerequisite: A centerline (.CL) file, polyline or two points

Distance Between Two Entities

This command reports the average, minimum and maximum distances between two entities. For example, this command can be used to find the minimum distance between a right-of-way polyline and a property perimeter

polyline. The supported entities include polylines, lines and arcs. The reports the coordinates along the two entities at the minimum and maximum distances. There is an option to dimension the minimum or maximum distances.



Prompts

Select first polyline, line or arc: *pick a polyline*

Select second polyline, line or arc: *pick a polyline*

Distance Between Two Entities results dialog

Pulldown Menu Location: Centerline

Keyboard Command: minmax2

Prerequisite: Two entities

Centerline Conversions

There are many import routines available under Centerline Conversion to convert other applications' centerline files to Carlson centerline files (.CL), and many export routines to convert Carlson centerline files (.CL) to other applications' formats. Each Import option prompts for the file to convert and the name of the new .CL file to create. Each Export option prompts for .CL file to convert and a file name for the new file. The import formats include C&G Point Group .PTS, InRoads .ASC, Geodimeter .ARE/.GEO/.RAW, GeoPak .OSD, Leica .GSI, MOSS .INP, Novapoint .TIT, SDMS .ALI/.PRJ, Softdesk, Sokkia .SDR, ISPOL .ALI, CLIP .PLA, TDS .RD5 and Terramodel .RLN/.ALN. The export formats include C&G Point Group .PTS, Leica .GSI, SMI .CH, Softdesk, Sokkia .SDR, Topcon .RD3, Trimble .DC, TDS .RD5 and TDS .PL5.

For the TDS RD5, there is an option to include a profile along with the centerline. Also, there is an option to include sections. When sections are included, the station data is included in the RD5 and the section grades are output to TP5 files where each station has a separate file for the left and right sides. The TP5 files are created in the same folder as the RD5.

For the Trimble DC, there are options to include a profile and sections along with the centerline.

For Civil 3D, the import works differently because the centerline definitions are stored within the drawing instead of external file. So this import does not prompt for a file but looks for Civil 3D centerlines in the current drawing. The Civil 3D centerline custom objects are converted to standard AutoCAD entities and there are prompts to save the centerline definitions to Carlson .CL files.

Pulldown Menu Location: Centerline > Centerline Conversion

Keyboard Command: c3d_cl, inroads2cl, geod2cl, geopak2cl, geopak2rd, wildcl2, moss2cl, sdms2cl, dcacl2, sdr2cl, ali_to_cl, pla_to_cl, importrd5, tm2cl, wildcl1, smicl1, dcacl1, cl2sdr, cl_to_rd3, export_rd5, tdscl1, export_dc, nova2cl

Enter Right of Way

This command adds right of way information to a centerline file which must be created before running this command. The right of way is created by entering station-offset points or picking points. A right of way polyline is drawn through the points and each point is labeled with the station and offset. Besides drawing the right of way, this data can also be used in *Process Road Design* to limit the cut/fill slopes.

Prompts

Choose Centerline to Process Specify a centerline file.

Layer name for labels <ROW>: *press Enter*

Number of decimal places for labels <2>: *press Enter*

Side for right of way (Left/<Right>)? *press Enter*

Starting station of centerline: 0.000

Enter station or pick a point ('U' to Undo, Enter to End): 0

Enter offset: 35

Enter station or pick a point ('U' to Undo, Enter to End): 200

Enter offset: 35

Enter station or pick a point ('U' to Undo, Enter to End): 250

Enter offset: 50

Enter station or pick a point ('U' to Undo, Enter to End): 300

Enter offset: 50

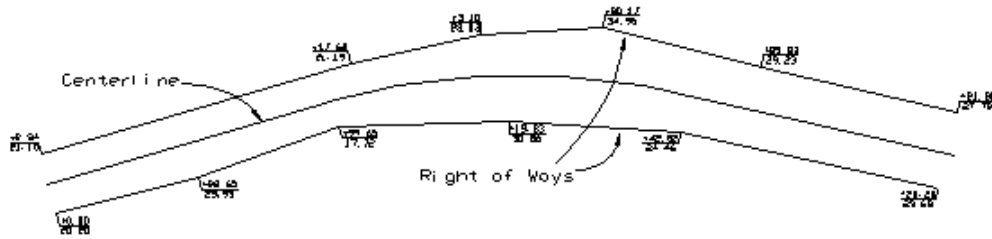
Enter station or pick a point ('U' to Undo, Enter to End): *pick a point*

Enter station or pick a point ('U' to Undo, Enter to End): *press Enter*

The end result is a new polyline and a fully annotated ROW line plot. The Enter ROW command can be used to create new polylines that can be applied to templates using the command Template Point Centerline.

Draw right of way polylines (Yes/<No>)? *press Enter*

Number of decimal places <2>: *press Enter*



Right of Way polylines and labels along a centerline

Pulldown Menu Location: Centerline

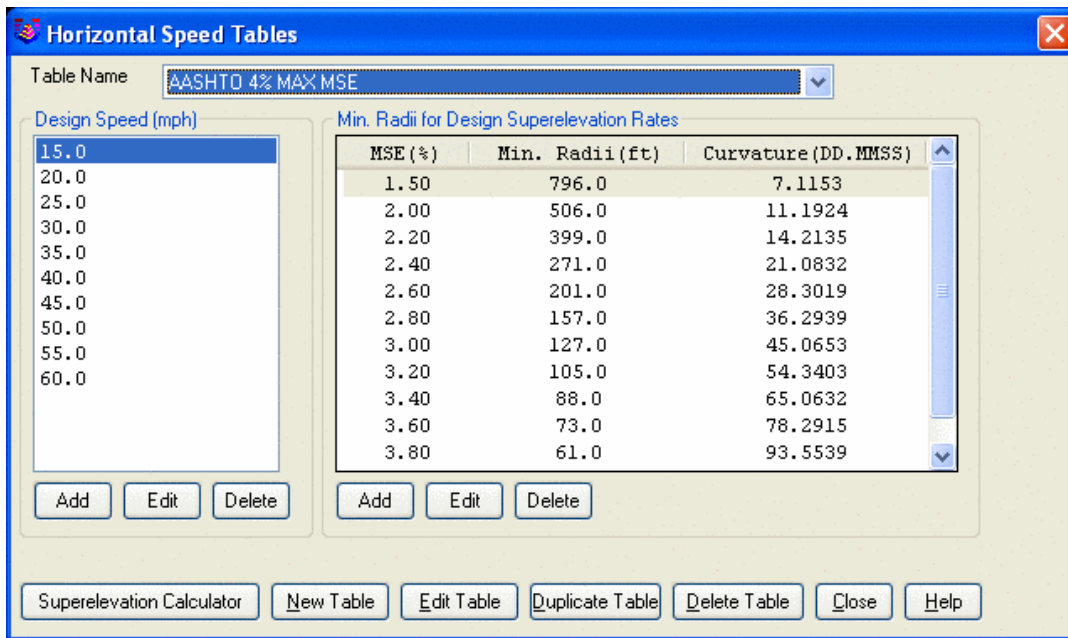
Keyboard Command: drwrow

Prerequisite: A centerline file with right of way data

Horizontal Speed Table

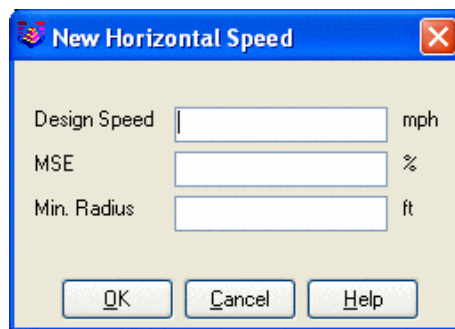
In the design of curve and spiral-curve-spiral centerlines, it's very important to determine the curve radii and superelevation rates, which can significantly affect the design speed of roads. The Horizontal Speed Table function provides a few of speed tables, which utilize AASHTO's speed table data and offer the recommendations for design speeds and curve parameters. Please refer to AASHTO A Policy on Geometric Design of Highways and Streets 2004 (pp 167-174) for details.

The Horizontal Speed Table function is integrated into the Centerline Editor command. To access the speed tables, in the curve or spiral centerline design dialogs, click on the Horizontal Speed Table button to open the speed table dialog shown as below. The Table Name list contains the names of all speed tables that has been defined. There are five default speed tables: AASHTO 4% MAX MSE, AASHTO 6% MAX MSE, AASHTO 8% MAX MSE, AASHTO 10% MAX MSE and AASHTO 12% MAX MSE. You can add, edit and delete any speed tables. On the left of the dialog, the Design Speed box lists all design speeds. On the right there is a table that lists all the curve and superelevation data of the highlighted speed. The curve and superelevation data table will change when you highlight different speeds. All speed table files are in the ...\\USER folder and are available for all projects.

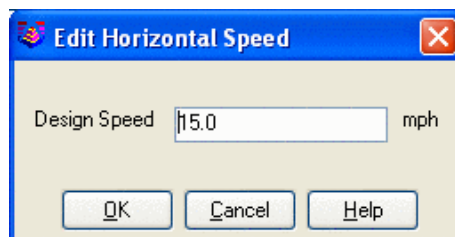


Horizontal Speed Table

There are three buttons under the Design Speed list. Add button adds a new design speed to the current speed table. The New Speed dialog opens for you to enter the Design Speed, and MSE and Minimum Radius for the speed. Edit button allows you to edit the highlighted design speed. Delete button will delete the highlighted speed and all the curve and superelevation data associate to the speed.

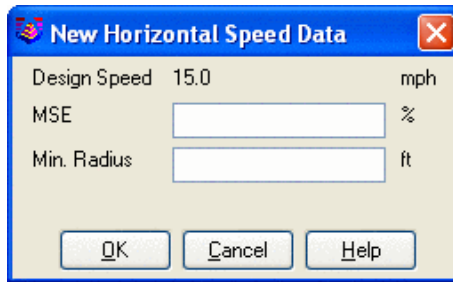


New Design Speed



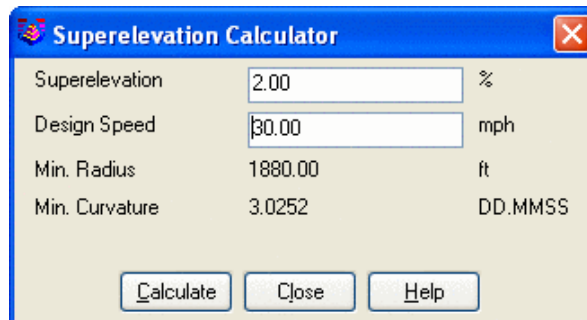
Edit Design Speed

Under the table of the curve and superelevation data, there are Add, Edit and Delete buttons which allow you to add, edit and delete the data entry for the highlighted design speed.



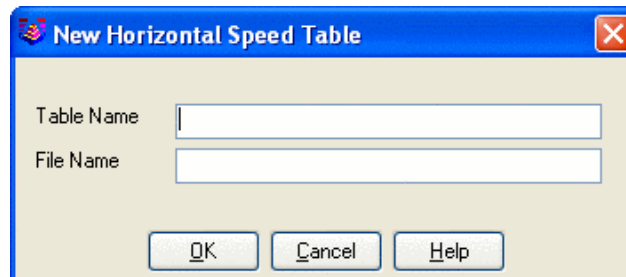
New Curve and Superelevation Data

If you have a special design speed or superelevation rate that is different than any data entry in the speed table, or if you don't want to look up the tables for a curve data, the Superelevation Calculator button is here to help you to get the curve data. Click on the button to open the Superelevation Calculator dialog. Enter the values in the Superelevation and Design Speed boxes, the Minimum Radius and Curvature will be calculated and displayed.



Superelevation Calculator

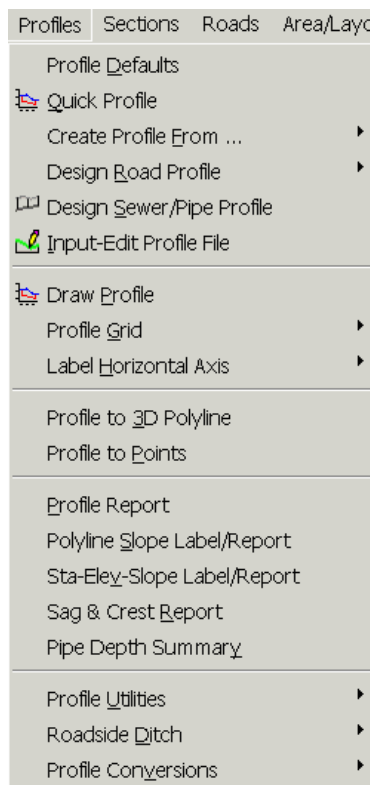
New Table button creates a new speed table, Edit Table button is used to modify the name of current table or the file that stores the speed data, Duplicate Table button makes a new speed table that contains the same data as current table, and Delete Table button removes current table completely.



New Horizontal Speed Table

Profile Menu

The Profile menu shown below has commands for creating, drawing and reporting profiles.



Quick Profile

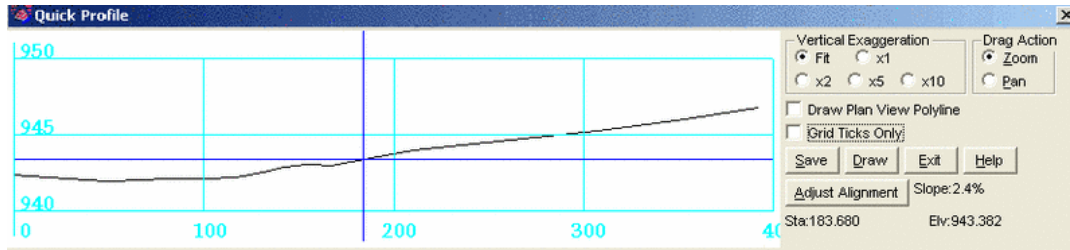
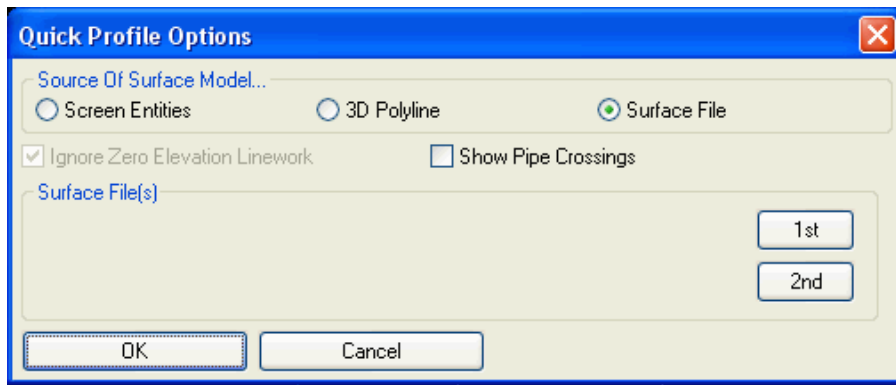
This command allows you to create a profile in one step. The alignment for the profile can be defined using picked points, a centerline file or a polyline. The surface for the profile can be defined by 3D screen entities, 3D polyline or surface files (grid or triangulation).

Screen Entities: The program creates the profile by finding the intersections of the centerline with 3D linework entities in the drawing. There's an option for whether to ignore entities at zero elevation.

3D Polyline: Creates a profile using a selected 3D polyline. The polyline vertex elevations are used for the profile elevations and the profile stations are from the lengths of the polyline segments.

Surface File: This option allows you to use one or two grid or triangulation surfaces. There's also an option to Show Pipe Crossings which will find and display pipe crossings from sewer networks and 3D polylines tagged as pipes. The sewer network can be created in the Hydrology module. To tag a 3D polyline as a pipe, use the Assign Pipe Data To Polyline command.

Since picked points are the default for the horizontal alignment, the command is as quick as select surface type (screen or file), then *Pick, Pick, Enter* and view. The resulting profile is displayed in a graphic dialog box with real time data reporting. As the crosshairs are moved across the profile in the window, the station, elevation and slope data corresponding to the current crosshair location appear in the lower right of the window. A second crosshair on the plan view corresponds to crosshair movement along the profile so the user knows exactly where the current profile point is on the plan view. Also the Adjust Alignment function allows you to drag a horizontal alignment point and update the profile in real-time.



Vertical Exaggeration: Determines the amount of vertical exaggeration for the profile in the window.

Drag Action: Determines whether the right mouse button functions as "Zoom" or "Pan" in the profile window.

Grid Ticks Only: Instead of the full graph as shown above, Grid Ticks only plots only ticks along the horizontal and vertical axis near the station and elevation text.

Colors: Set the profile colors for the graph.

Adjust Alignment: Allows you to pick a horizontal alignment point and while moving it, the profiles are updated in real-time. You can also select a horizontal alignment segment and move the whole alignment position. The Adjust Alignment function is only available when surface files are used as the source of the surface model. When you adjust the alignment, the program prompts on exit whether to save the adjusted alignment to a centerline file (CL).

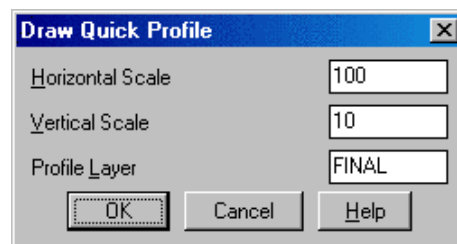
Save: Writes the current profile data to a .PRO file.

Draw: This draws the profile with grid in the drawing. The user has options for horizontal and vertical scales and the layer of the profile. The Draw Profile command includes more options for drawing the profile. In order to use this command, you must first create a .PRO file using the Save command described above.

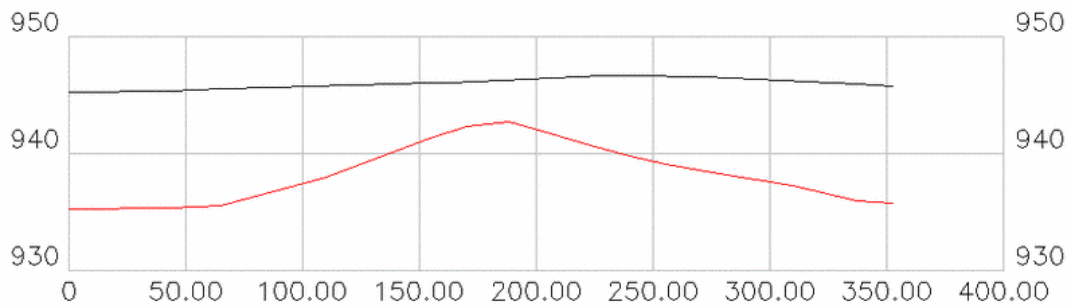
Print: This makes a graphic report of the profile in either PDF or DWF format as selected under Settings->Configure.

Exit: Exits this command.

Help: Opens on-line help.



Note that the Draw option will exit the Quick Profile command after the drawing is complete. A typical completed drawing, in this case with two surfaces, is shown below. Note also that the horizontal stationing text offset follows the setting in the Draw Profile command itself.



Prompts

Pick starting point (CL-Centerline, P-Polyline): *screen pick alignment points for profile*

Pick second point: *pick next point*

Pick next point (Enter to end): *press enter to end*

Tested 58 of 58 Entities Intersects found > 33

Dialog Box

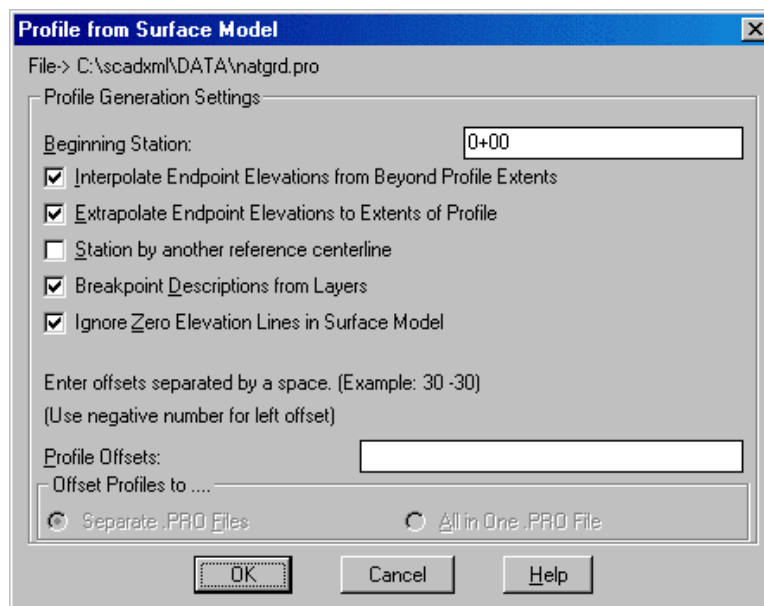
Pulldown Menu Location: Profiles

Keyboard Command: quickpro

Prerequisite: 3D screen entities or surface file

Profile from Surface Entities

Profile from Surface Entities creates a profile from contours, triangular mesh, and other 3D drawing entities. The method is to draw a polyline as the profile centerline. Then the profile is derived from the intersections of this polyline with the 3D entities. For added accuracy in pulling the profile, include the triangular mesh as well as the contours.



File: Displays the name of profile to be created.

Beginning Station: Specify the beginning station for the profile.

Interpolate Endpoint Elevations from Beyond Profile Extents: When checked, the program will look past the ends of the centerline for additional intersections with 3D entities. These additional intersections will then be used to interpolate the elevation at the starting and ending station of the centerline.

Extrapolate Endpoint Elevations to Extents of Profile: This option uses the slope of the last two elevation points of the profile and calculates the elevation of the endpoint from this slope.

Station by another reference centerline: When checked, the program will prompt you to pick another centerline polyline. The intersection points along the first centerline are then projected onto the second centerline. The profile then stores the elevation of the intersection with the station along the second centerline.

Breakpoint Descriptions from Layers: When checked, breakpoint descriptions are assigned based on layer name of surface entities. These descriptions are used in routines such as *Profile Editor* and *Profile Report*.

Ignore Zero Elevation Lines in Surface Model: When checked, any zero elevations selected in the surface model are ignored.

Profile Offsets: Specify optional offset profiles. Enter offsets separated by a space. Example: 30 -30 (to create 30' left and 30' right offset profiles). After entering the offset values, press TAB to select file options described below.

Offset Profiles to: Specify whether offsets profiles should be created as separate profile (.PRO) files, or included in a single profile (.PRO) file. Only available if you specify Profile Offsets above. Offset profiles are automatically named by combining the profile name and the offset. For example, if the profile is named NATGRD.PRO and you create a 30' right offset profile, it will be named NATGRD30.PRO.

Prompts

Profile File to Write dialog Specify a new profile file (.PRO) name to create.

Profile from Surface Model dialog Make choices, click OK.

Polyline should be drawn in direction of increasing stations.

CL File/<select polyline which represents the profile centerline>: *pick the centerline* (Do not press Enter.)

Select Lines, PLines, and/or 3DFaces that define the surface for profiling.

Select objects: *C* (for crossing and window everything the centerline crosses) or *All* (to select all objects on the drawing)

Keyboard Command: prosm

Prerequisite: A polyline centerline and surface lines and polylines.

Profile from Grid or Triangulation Surface

This command creates a profile (.PRO file) from a centerline polyline and a surface model stored in a 3D grid file (.GRD) or triangulation file (.TIN or .FLT). The polyline defines the alignment of the profile and the grid defines the surface.

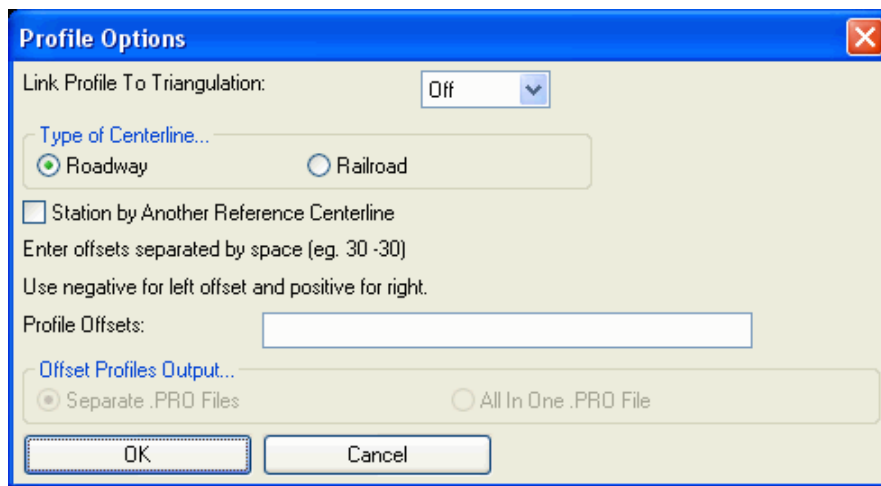
After selecting the reference surface file, there is a Profile Options dialog with these options:

Link Profile To Triangulation: This option will update the profile whenever the reference triangulation is modified.

Type of Centerline: This setting chooses the type of stationing for centerline curves.

Station by Another Reference Centerline: This option uses a second reference centerline for the stationing of the profile. The main centerline is used to find the elevations on the surface and then these main centerline positions are projected onto the reference centerline to get the stationing. The reference centerline needs to extend along the full range of the picked polyline in order to project correctly and capture offsets along the entire length of the picked centerline.

Profile Offsets: In addition to creating the profile along the centerline, you can also create profiles offset left and right.



Prompts

Choose Grid or Triangulation file to process Select existing .GRD, .TIN, or .FLT file.

Profile Options dialog.

Choose PROfile file to Write dialog Enter a profile file (.PRO) name to write.

Polyline should have been drawn in direction of increasing stations.

CL File/<Select polyline that represents centerline>: *select a polyline*

Polyline should have been drawn in direction of increasing stations.

CL File/<Select Reference centerline polyline>: *select a polyline*

CL File/<Select Reference centerline polyline>: *press Enter*

Reference CL starting station <0.0>: *press enter*

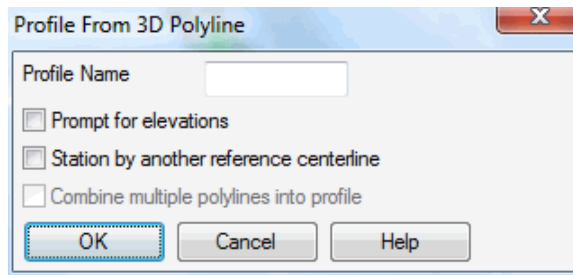
Pulldown Menu Location: Profiles > Create Profile From ...

Keyboard Command: progrid

Prerequisite: A .GRD grid file, .TIN, or .FLT tmesh file

Profile from 3D Polyline

To create a profile (.PRO), Profile from 3D Polyline uses X-Y distances between the points of a 3D polyline for sequential stations and the Z values at these points for profile elevations. In the options dialog, Profile Name is an optional description for the profile. The Prompt For Elevations option will prompt for the elevation at each polyline vertex to use for the profile instead of using the polyline elevations. The Station By Another Reference Centerline method locates the station for each polyline vertex along a reference centerline and uses this reference station instead of the polyline distance for the profile stationing. The reference centerline can be defined by another polyline or centerline file (.CL). When using the reference centerline, the Combine Multiple Polylines Into Profile option allows you to select multiple 3D polylines and put the data into a single profile. For example, you can use these two options to create a profile of curb elevations with road centerline stationing by selecting multiple 3D curb polylines and the road centerline as the stationing reference.



After the options dialog, the program prompts for the .PRO file to create and then the 3D polyline to process.

Prompts

Profile From 3D Polyline dialog

Profile File to Write dialog Specify a profile file name to create

Select polyline to profile: *pick a 3D polyline*

Created 72 data points for profile C:\sample\abc.pro

The new profile is then stored.

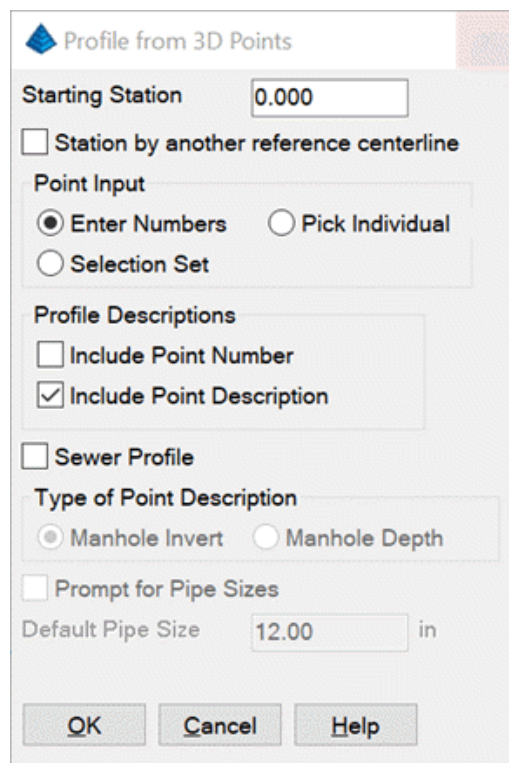
Pulldown Menu Location: Profiles

Keyboard Command: pro3dpl

Prerequisite: A 3D polyline

Profile from 3D Points

This command creates a .PRO file using the X-Y distances between user-specified points for sequential stations and the Z values at these points for profile elevations. Unlike many of the Carlson profile routines, this routine does not require a horizontal alignment. The points can be specified by point numbers from the current coordinate file or by picking Carlson points on the screen.



The Starting Station sets the profile station for the first point. The Station By Another Reference Centerline option

projects the points onto a separate centerline to calculate the stations for the profile. The Profile Description options allow storing the point number and description to the profile descriptions. The Sewer Profile option applies when the points are at manholes. In this mode, the program creates a sewer type profile and prompts for pipe sizes.

Prompts

Profile From 3D Points dialog

Pick Point Entity: 34

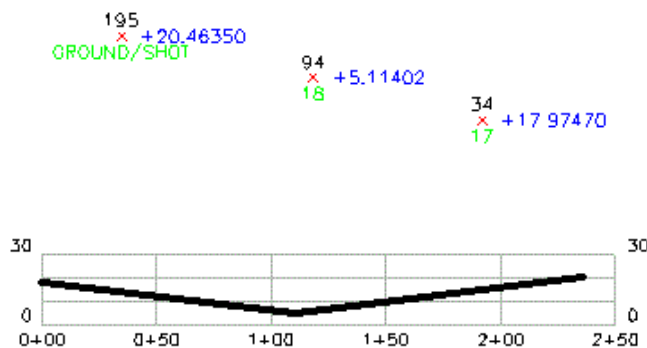
Pick Point Entity (Enter to end): 94

Pick Point Entity (Enter to end): 195

Pick Point Entity (Enter to end): *press Enter*

Select Profile File to Write Choose a .pro file name

In this graphic showing the points 34, 94 and 195. Note that the "+" in front of elevations higher than 0 is an option within Draw-Locate Points.



Pulldown Menu Location: Profiles > Create Profile From

Keyboard Command: pro3dpt

Prerequisite: Plot points with real Z axis elevations or a coordinate file.

Profile from Section File

This command creates a .PRO file from user-specified offsets or from specified descriptions on cross sections contained in a .SCT file. The elevations of the profile are derived from the elevations of the cross sections at the offset and the stationing for the profile matches the stations of the cross sections. There is an option to extend cross section elevations to reach the offset for the profile when necessary. If, for example, a road design has "SH" at some offset on each cross section, both left side and right side, you can pull the profile from the "SH" descriptions by specifying left or right side. So you can profile the shoulder, or the "TIE" point to existing ground, etc. Alternately, if the sections extend from offset 100 left to offset 100 right, you can pull a profile at offset -30, or 30, in which case the program will create a profile by interpolating the elevation from the sections for each station in the cross section file.

Prompts

Choose SCT file to read dialog

Select the existing cross section file.

Profile of offset or template description [<Offset>/Desc]? *press Enter*

Enter the offset to profile (left offsets as negative) <0.0>: *press Enter*

Extrapolate sections to this offset (<Yes>/No)? *press Enter* This prompt appears if the program detects that

some or all sections do not extend to the requested offset to profile.

If you chose Desc above, one of the advantages of the new file loading dialog is that you can review the ASCII descriptions in the section file, as shown below:

Enter description to profile: *EP*

Template side to process [Left/<Right>]? *L*

PROfile file to Write dialog box Enter a profile file name to write.

Create another profile (Yes/<No>)? *press Enter*

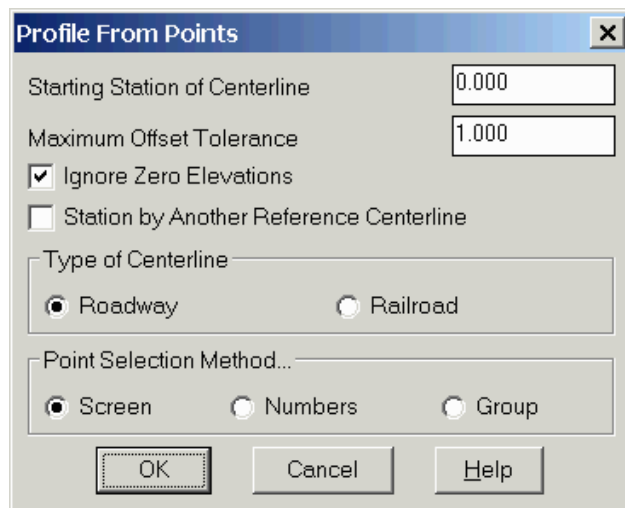
Pulldown Menu Location: Profiles > Profile from ...

Keyboard Command: profsct

Prerequisite: Cross sections in a .SCT file

Profile from Points on Centerline

This command creates a .PRO file from points and a centerline that is represented by a polyline or centerline file. The elevations of the profile are derived from the elevation of the points and the stationing for these profile points is calculated from the distance along the centerline. The points must be within the offset distance from the polyline in order to be included in the profile. The profile is created by projecting the points perpendicular onto the alignment to determine the station and the elevation comes from the point elevation. The polyline or centerline should be drawn (or defined) in the direction of increasing stations. The points can be selected from point entities in the drawing (Screen), by point numbers from the current coordinate file (Numbers), or by point group as defined by the Point Group Manager (Group).



Prompts

PROfile file to Write dialog box: Enter a new profile file name to write.

CL File/<Select polyline that represents centerline>: *pick a polyline or choose C for Centerline*

Select Centerline file if Centerline option is used. If the desired points are further from the centerline, enter a larger maximum offset tolerance.

Note: for all selected points, the points should be located on the real Z axis.

Select the Carlson points along the centerline.

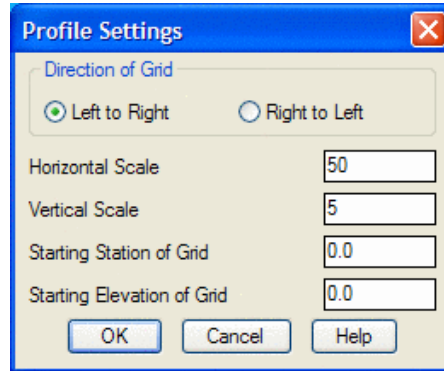
Select objects: Select the point entities.

Keyboard Command: profpts

Prerequisite: A polyline centerline and points

Profile from Polyline on Profile Grid

This command allows you to convert a polyline that is drawn on a profile grid into a profile (.PRO) file. The polyline must be drawn in the direction of stationing. Vertical curves, which are typically parabolas, will be captured as a series of vertices. The polyline can either be used to create a new profile (.PRO) file, or can be appended to an existing profile (.PRO) file. When the profile grid has equal horizontal and vertical scale for a vertical exaggeration scale of 1, and the polyline contains arcs, then the program creates a profile with circular curves.



Prompts

File Selection Dialog Box Specify the profile (.PRO) file to create, or an existing profile (.PRO) file to append to.

Profile Settings Dialog Set these parameters to match the dimensions of the grid for the profile plot.

Pick Lower Left Grid Corner: *pick the grid corner* Endpoint snap is set on.

Profile number <1>: *press Enter* This is an optional profile name used for multiple profiles.

Select the polyline to write profile from:

Select object: *pick the 2D polyline in the grid*

A station and elevation report is produced.

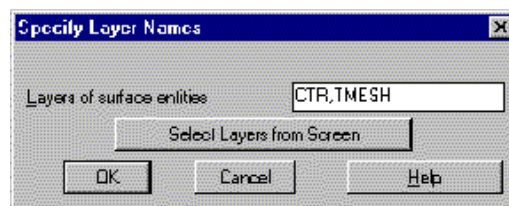
Pulldown Menu Location: Profiles > Create Profile From...

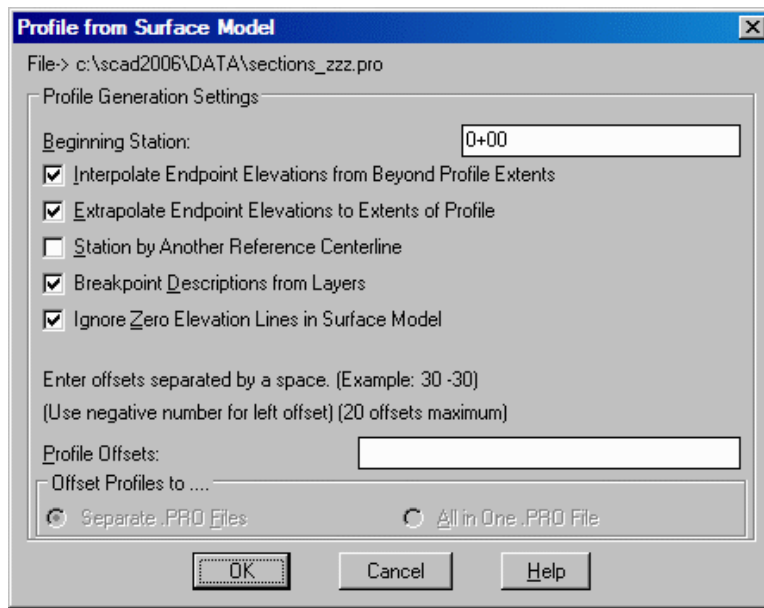
Keyboard Command: pro2dpl

Prerequisite: Drawn polyline which represents profile, existing drawn profile grid

Profile from Layers

This command creates a profile from surface entities with one of the specified layers. The surface entities can be contours, triangular mesh, and other 3D drawing entities. This command is the same as Profile from Surface Entities with the addition of the layer filtering. The method is to draw a polyline as the profile centerline. Then run Profile by Layers and specify the layer names of the surface entities to include in the profile. For example, the layer names CTR and TMESH could be entered to use only the contour polylines and triangulation mesh on these layers. Entities on all other layers would be ignored. The profile is derived from the intersections of this polyline with the 3D entities on the specified layers.





Prompts

Specify Layers Names Dialog Specify layers, click OK

Profile File to Write Dialog Specify a profile file name (.PRO) to create

Profile from Surface Model dialog box

Polyline should be drawn in direction of increasing stations.

CL File/<select polyline which represents the profile centerline>: *Pick the centerline*

Select surface entities on corresponding layers.

Select objects: *C* For crossing and window everything the centerline crosses.

Pulldown Menu Location: Profiles > Profile from ...

Keyboard Command: prolayer

Prerequisite: A polyline centerline and surface lines and polylines.

Profile from Pipe Polylines

This command creates a profile that contains the station, elevation and pipe width of pipes that cross the centerline. This type of profile is called a Crossing profile and Draw Profile treats it differently. Instead of connecting the station-elevation points with a polyline, Draw Profile draws each station-elevation as a circle with a radius of the pipe width. When there is vertical exaggeration in the drawn profile, the pipe circles are drawn as ellipses.

This routine uses a polyline that represents the centerline. The pipe polylines are 3D polylines with an assigned pipe width. One way to create them is to use the command *Draw Pipe 3D Polyline* in the Profile Utilities sub-menu. To attach the pipe width value to a polyline, use the *Assign Pipe Width to Pline* command also in the Profile Utilities sub-menu. The program then finds the intersections of the polyline centerline with the pipe polylines and stores the station of the intersection along the centerline with the elevation and pipe width of the pipe polyline. There is also a prompt to whether the pipe position is at the top, bottom or middle of the 3D pipe polylines.

Prompts

Profile File to Write Dialog Enter new .PRO file name.

Polyline should have been drawn in direction of increasing stations.

CL File/<Select polyline that represents centerline>: *pick a polyline*

Enter the starting station <0.0>: *press Enter*

Select the pipe polylines crossing the centerline.

Select objects: *pick pipe polylines*

Position of pipe polylines on pipe [Top/Center/<Bottom>]? press Enter

Found 2 crossing pipe polylines.

The command Draw Profile would then interpret this profile as a pipe profile, and plot it as needed.

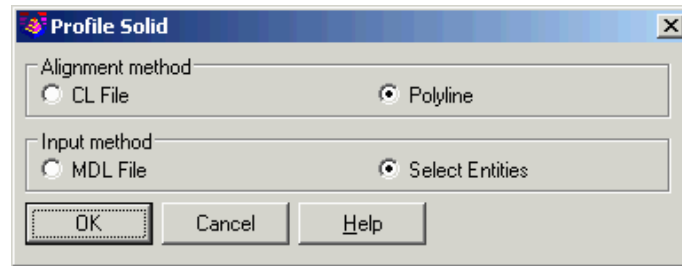
Pulldown Menu Location: Profiles > Profile from ...

Keyboard Command: propipe

Prerequisite: A polyline centerline and pipe polylines

Profile from 3D Solids

This command creates a profile of slices through 3D solids. The alignment for the profile can be defined either by centerline file or by a polyline. The solid model can be defined either by a Carlson 3D model file (MDL) or by 3D Solid entities in the drawing.



Prompts

Profile Solid dialog

Profile to Write *Specify a file name*

Select entities for model.

FILter/<Select entities>: *pick 3D solids*

Select polyline centerline: *pick a polyline*

Pulldown Menu Location: Profiles > Create Profile From

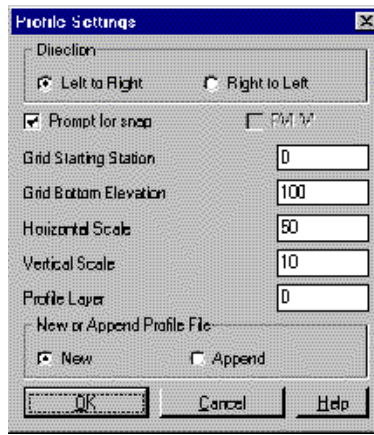
Keyboard Command: pro_solid

Prerequisite: CL file or polyline alignment, and 3D Solid or MDL file

Enter Profile On-Screen

This command allows you to create profile files and is similar to *Design Road Profile*. The only difference is that Enter Existing Profile does not ask for vertical curves. The procedure is to first specify the on-screen grid and then enter or pick the stations and elevations. The profile is drawn as it is entered.

Notice that the station, elevation, and slope at the current position of your cursor crosshairs is displayed at the bottom of the side-bar menu. These values will update whenever the crosshairs move except after selecting either the side-bar or top menu.



Prompts

Profile Settings dialog

Profile File to Write dialog Specify a profile file (.PRO) to create.

Station of first PVI or pick a point: 0

Elevation of PVI: 565

Second station or pick a point (U, E, D, Help): 200 'U' is undo, 'E' ends the routine, 'D' is incremental distance to the next station, 'H' brings up an explanation of these items on-screen.

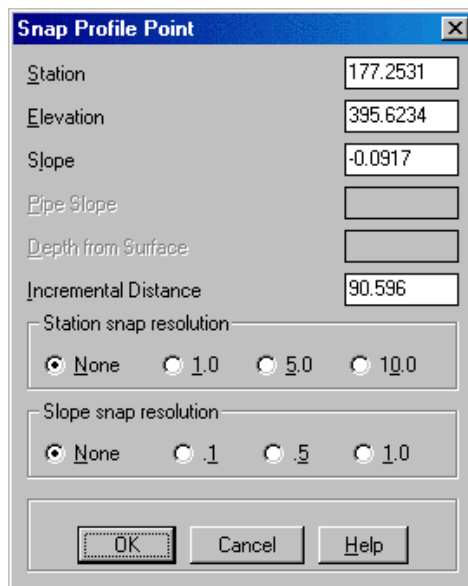
Percent grade entry/Ratio/<Elevation of PVI>: 575

Station of next PVI or pick a point ('U' to Undo, Enter to End): *pick a point*

Snap PVI dialog

This dialog box appears when you pick a point and the Prompt for Snap option in the Profile Settings dialog is selected. The station and slope may be changed to the nearest snap value. The elevation is the free variable and it will change to compensate for any snap. To change the elevation, select the elevation edit box and enter the new value.

Station of next PVI or pick a point ('U' to Undo, Enter to End): *press Enter*



Pulldown Menu Location: Profiles

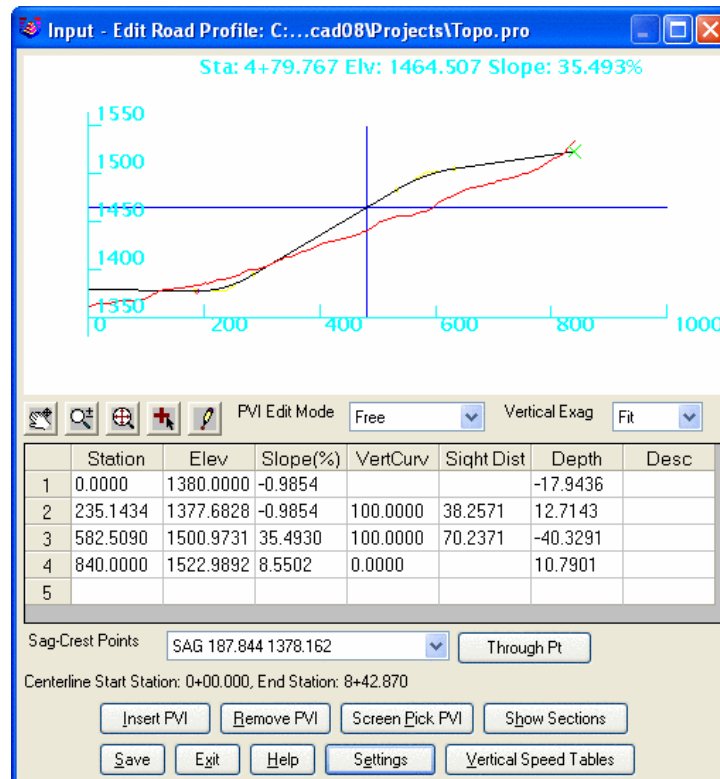
Keyboard Command: makeprof

Prerequisite: A profile grid drawn on-screen

Road Profile Editor

This command opens the Road Profile Editor dialog, showing the profile graph and a spreadsheet table containing the profile data. With this dialog, you can enter and edit road profile files (.PRO), not only by specifying the values in the spreadsheet, but also by editing the PVI points on the profile graph directly. The updates in the spreadsheet and the graphic box are synchronized.

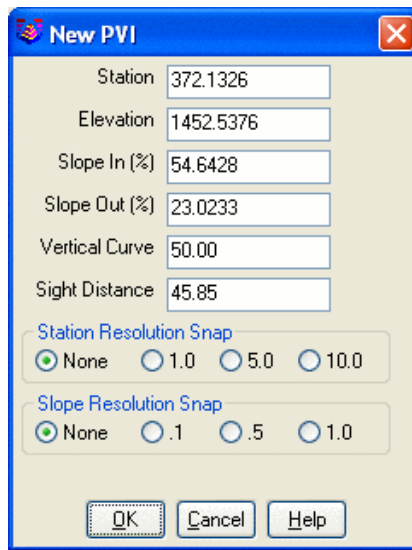
From the Profile menu in the Civil Design Module, choose Road Profile Editor. The program reads a road profile file (.PRO), a road centerline file (.CL) and a surface file (.TIN or .FLT). If you design a new road profile, just enter the new road profile file name. If you open an existing profile to edit, the profile graph is shown in the graphic box on the top, and the spreadsheet is filled with profile data. The buttons and lists between the graphic box and the spreadsheet provides the abilities to input and edit the road profile in graphic.



Road Profile Editor

1. Functions Editing Profile in Graphic:

Switch to pan mode button: Switch the cursor to PAN mode. **Switch to dynamic zoom mode button:** Switch the cursor to ZOOM mode. **Zoom Extents button:** Zoom the graphic window to show the complete graph. **Add PVI button:** Allow you to add a new PVI point by picking at any locations inside the graphic box. The program will extract the station and elevation of the point and display them in the New PVI dialog, from where you can modify the station and elevation directly in the Station and Elevation boxes, or by modifying the Slope In and Slope Out values. You can also specify the vertical curve length or sight distance to define the vertical curve of current PVI. The resolution snap for the station and slope round up the station and slope values. Click on OK button to save the new PVI data. Below is an example of the dialog.



New PVI dialog

Edit PVI button: Pick an existing PVI point on the profile graph and drag it around to change the station and elevation.

PVI Edit Mode list: This list has five options: Free, Hold Slope In, Hold Slope Out, Hold Station and Hold Elevation, which controls the movement of the PVI that is being edited by the Edit PVI button command.

Vertical list: This list determines the vertical exaggeration of the profile graph.

2. Spreadsheet Editor:

The spreadsheet editor allows you to enter and modify data cell by cell. The profile graph will be updated automatically after any changes of the profile data. **Insert PVI** button inserts a row in front of the highlighted row to create a new PVI, **Remove PVI** button deletes the highlighted row as well as the corresponding PVI, and **Screen Pick PVI** button allows you to pick a point on screen and insert it into the spreadsheet. In the **Sag-Crest Points** list, the coordinates of all Sag and Crest Points are listed.

3. Settings Dialog

Click on the Settings button, the settings dialog displays.

Hold Current Elevation: When you change a PVI's station or elevation, if this toggle is on, its slope out will be changed and the elevation of the next PVI is held, otherwise its slope out is held and the elevation of the next PVI will be changed.

User K-Value: Toggles between displaying K-Value and Sight Distance in the fifth column of the spreadsheet.

Grid Ticks Only: Toggles between displaying the grid and grid ticks in the graphic box.

Show Slope When Zoom In: This option allows to display the slopes on the long enough profile segments when zoom in.

Show Reference Surface: An option to show the reference surface profile along with the road profile in the graphic box.

Show Reference Surface At Left Offset: An option to show the reference surface profile at a user-specified left offset of the road centerline.

Show Reference Surface At Right Offset: An option to show the reference surface profile at a user-specified right offset of the road centerline.

Show Centerline Special Stations: When this toggle is on, the points at centerline special stations such as PC, SC, ST, TS and SP are shown in the graphic box.

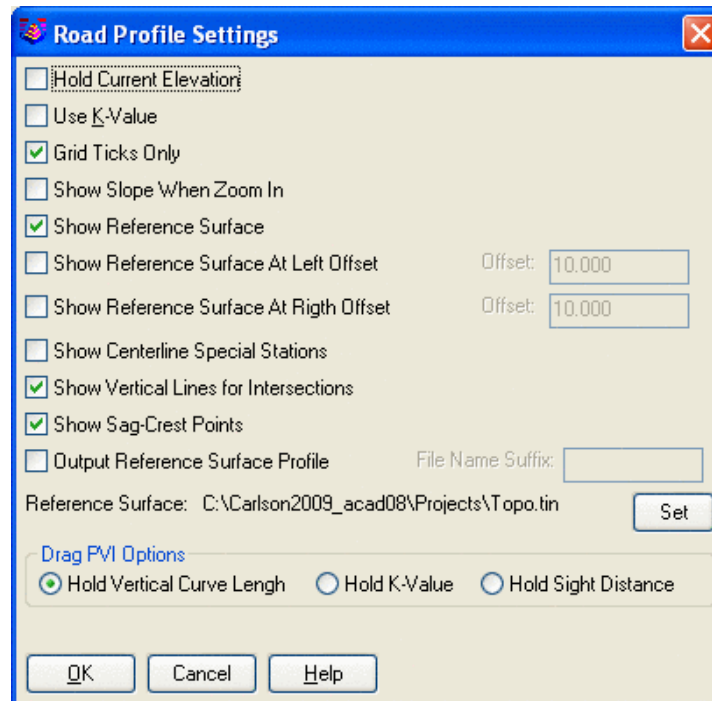
Show Vertical Lines for Intersections: When this toggle is on, vertical lines represent intersections of two road profiles are shown.

Show Sag-Crest Points: An option to draw the sag and crest points in the graphic box.

Output Reference Surface Profile: An option to output the surface profile to a file whose name has a suffix of the current road profile file name.

Set button: Set the current Reference Surface file to another one.

Drag PVI Options: When pick an existing PVI point on the profile graph and drag it around, you may choose to hold either the vertical curve length, or the K-value, or the sight distance.



Road Profile Settings

4. Show Sections

This function applies the design template at the road profile to get the road section file, computes the outslopes and earthworks relative to the reference surface section file, and displays both road and surface sections in a graphic dialog box. Click on the **Show Sections** button, the Road Design Templates dialog displays. The last 4 input items are strictly optional design files.

Design Template: Specify a template file (.TPL) or template series file (.TSF) that defines the final grade offsets and elevations and the cut/fill slopes.

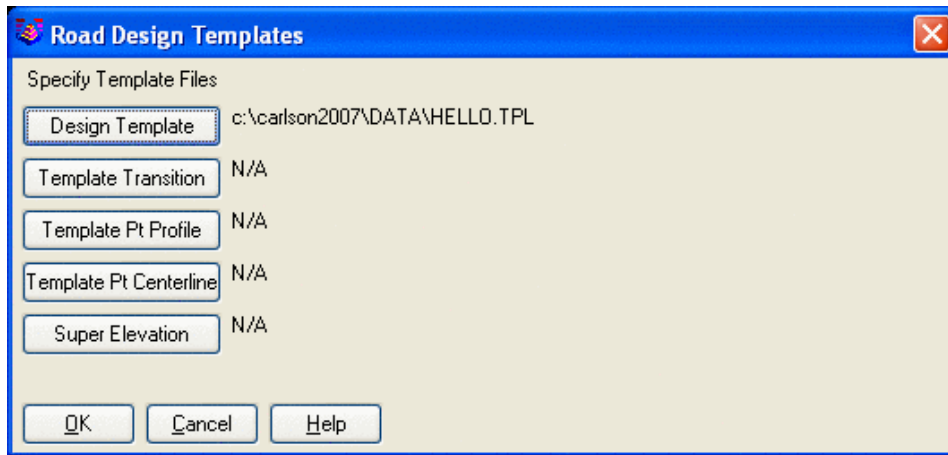
Template Transition: Specify a .TPT file, which allows modified template files to be applied at different ranges of stations on a project.

Template Point Profile: This option lets you have separate profiles for template points that are independent of the centerline file.

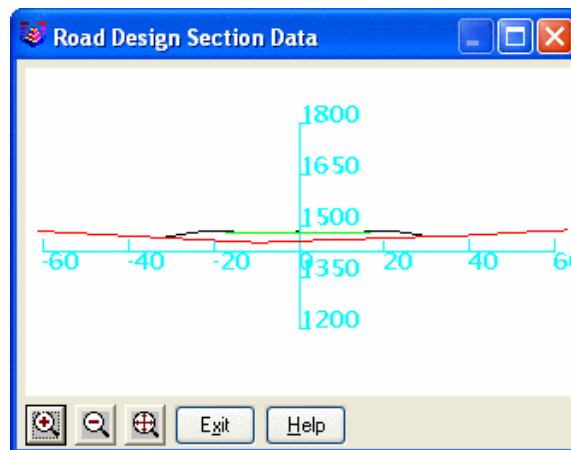
Template Point Centerline: This option lets you have separate centerlines for template points that are independent of the main centerline.

Super Elevation: This option is used to specify a super elevation file (.SUP) that defines the super elevation transition stations on a project.

After specifying the design template file(s), click on OK button to display the section graph. In the section dialog, the graph is automatically updated when you move your cursor along the road profile graph to change stations.



Show Sections: Choose Templates



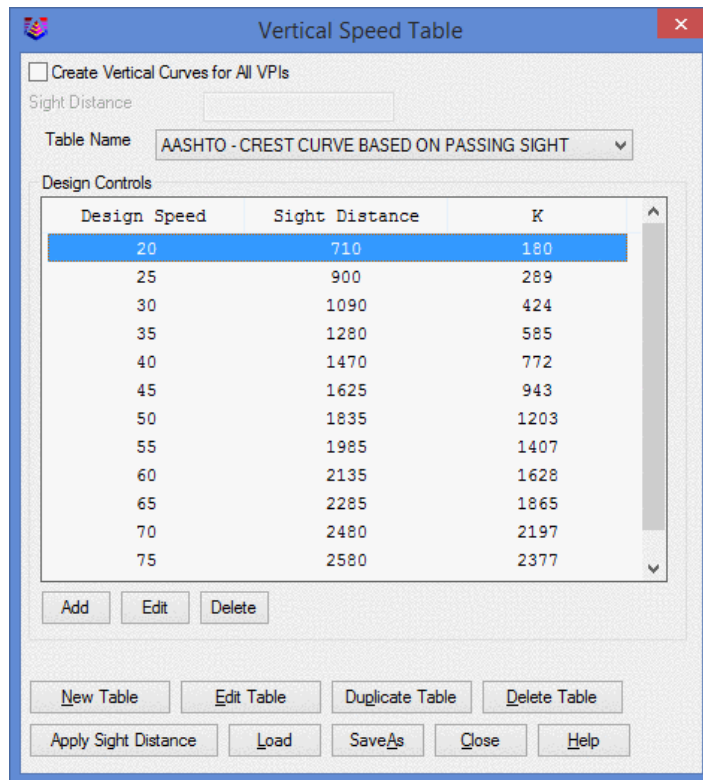
Road Design Section Dialog

5. Vertical Speed Tables

The Vertical Speed Table function provides a few of speed tables, which utilize AASHTO's speed table data and offer the recommendations for design speeds and curve parameters. Please refer to AASHTO A Policy on Geometric Design of Highways and Streets 2004 (pp 265-280) for details.

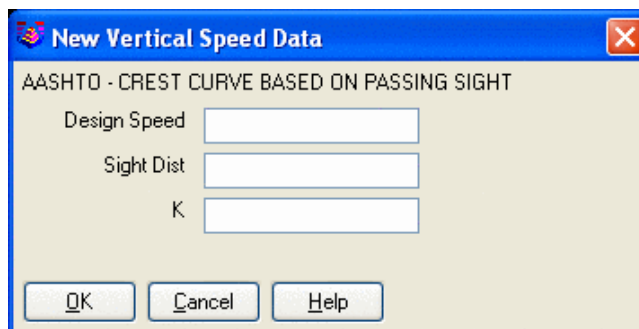
Click on the Vertical Speed Tables button to open the speed table dialog shown as below. The **Table Name** list contains the names of all speed tables that have been defined. There are five default speed tables: AASHTO - Crest Curve Based On Passing Sight, AASHTO - Crest Curve Based On Stopping Sight, AASHTO - Sag Curve Based On Stopping Sight, METRIC- AASHTO - Crest Curve Based On Passing Sight, METRIC- AASHTO - Crest Curve Based On Stopping Sight and METRIC- AASHTO - Sag Curve Based On Stopping Sight. You can add, edit and delete any speed tables. All speed table files are in the ... \USER folder and are available for all projects.

After highlighting a Design Speed and click Apply Sight Distance button, the Sight Distance value will be applied and the Vertical Curve is calculated and filled in the spreadsheet. When there's no vertical curve, you are allowed to create Vertical Curves for all VPIs and the program will check if the vertical curve value fits all the VPIs.

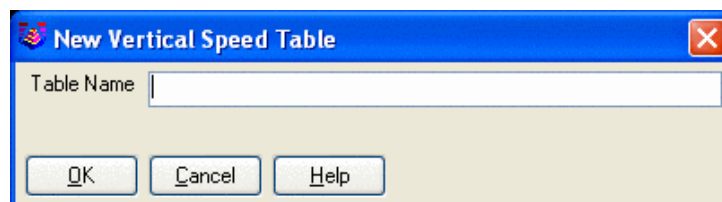


Vertical Speed Table

Click on the Add button, the New Vertical Speed Data dialog displays. Enter values in the Design Speed, Sight Distance and K boxes. Click on OK button to commit the new speed entry. Edit button allows you to modify design speed, sight distance and K values of the highlighted speed entry, and Delete button deletes the highlighted entry from current table. New Table button creates a new speed table, Edit Table button is used to modify the name of current table, Duplicate Table button makes a new speed table that contains the same data as current table, and Delete Table button removes current table completely.



New Speed Entry



New Vertical Speed Table

Prompts

Road Profile Editor dialog: Fill in values.

Pulldown Menu Location: Profiles > Design Road Profile

Keyboard Command: roadpro

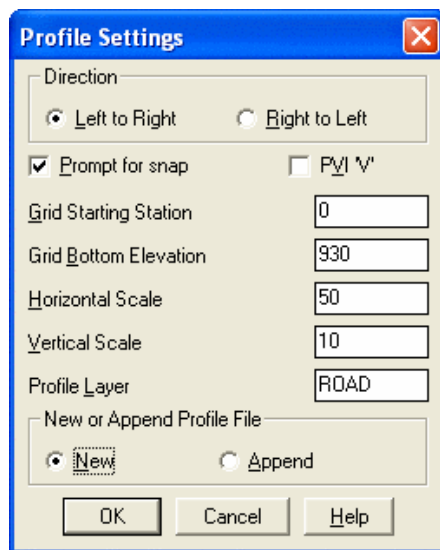
Prerequisite: a road profile file (.PRO), a road centerline file (.CL), a surface file (.TIN, .FLT)

Design Road Profile

This command is for simultaneously creating a .pro file and drawing the road profile. It is typically used when designing a road profile on top of a plotted existing grade profile, where the goal is to minimize cut and fill and keep to a minimum the number for vertical curves and avoid excessively steep grades. It is often necessary to match the starting and ending elevations of existing roads or features. For example, a side road will contact the main road at a fixed, given elevation. One concept to remember is that it may be best to favor a little more fill than cut in the design profile, because if your design template for the road involves ditches, a little bit of cut can lead to significant extra cut volumes due to the ditch placements. The *Design Road Profile* command works fine when overlaying on profile plots with either matching horizontal and vertical scales or exaggerated vertical scales (e.g. 50 H and 5 V). Just be sure to specify the correct scale settings in the Profile Settings dialog. The procedure is to first specify the on-screen grid and then enter or pick the stations and elevations.

Once two segments have been entered, you will be prompted for the vertical curve length. The vertical curve is a parabola, the typical form used in the United States. If you don't want a vertical curve, enter 0. Otherwise you can directly enter the vertical curve, or enter the sight distance or the K-value from which the vertical curve is calculated. The vertical curve can also be specified to pass through a point or do a best fit through multiple points. This through point option would be useful for hitting an existing feature such as a driveway on the vertical curve. Unequal vertical curves is another option where the vertical curve length going into the PVI differs from the length leaving the PVI. Before using your entry, the vertical curve, sight distance, and K-value are displayed. Object height and eye height are two variables that effect the vertical curve. Their values can be set using the command *Profile Defaults*.

Notice that the station, elevation and slope at the current position of your cursor crosshairs are displayed in real-time in a small dialog.



Prompts

Profile Settings dialog

Profile to Write Dialog Note that you can choose to append to an existing road profile, which allows you to continue design work in different work sessions. If Append is selected, the cursor will default to the end point of the selected profile, which will be treated as a 'PVI' point, so that you will be prompted for a vertical curve length after your very next picked point.

Pick Lower Left Grid Corner <5000.08,3211.24>[endp on]: Pick a lower left corner for the plotted grid on the screen. If you have just finished plotting the existing profile, the program will remember your lower left coordinates, and you just hit Enter to accept the default values.

Enter station or pick a point (Enter to End): 0

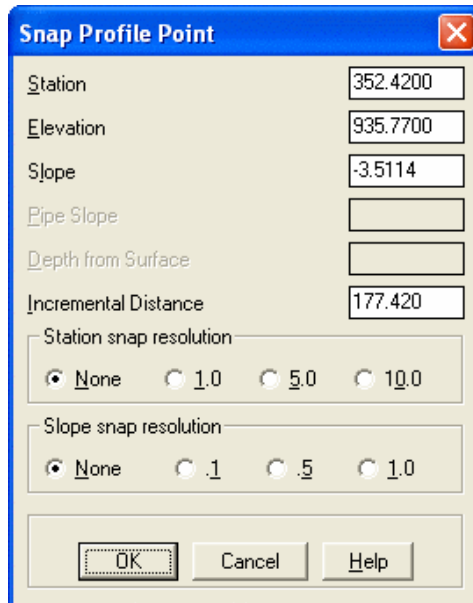
Elevation of PVI: 932.5

Station of second PVI or pick a point (U,E,D,Help): 175

Percent grade entry/Ratio/<Elevation of PVI>: 942

Station of next PVI or pick a point ('U' to Undo, Enter to End): *pick a point*

Snap PVI dialog



The Snap Profile Point dialog box is a standard Windows-style window with a blue title bar and a close button (X) in the top right corner. It contains several input fields and two groups of radio buttons. The fields are: Station (352.4200), Elevation (935.7700), Slope (-3.5114), Pipe Slope (empty), Depth from Surface (empty), and Incremental Distance (177.420). The Station snap resolution group has radio buttons for None (selected), 1.0, 5.0, and 10.0. The Slope snap resolution group has radio buttons for None (selected), .1, .5, and 1.0. At the bottom are three buttons: OK, Cancel, and Help.

The Snap PVI dialog box appears when you pick a point (if the Prompt for Snap option in the Profile Settings dialog is selected). The station and slope may be changed to the nearest snap value. The elevation is the free variable and it will change to compensate for any snap. To change the elevation, select the elevation edit box and enter the new value. In this example, you might choose a slope snap of 0.1 and if the station was flexible (not fixed, such as the end of the road), you could choose a station snap of 10.

View Table/Unequal/Through pt/Sight Distance/K-value/<Length of Vertical Curve>: 100

For Crest with Sight Distance>VC and Vertical Curve => 100.00

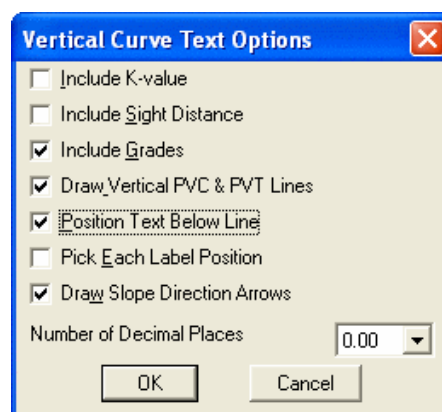
Sight Distance => 124.43, **K-value =>** 11.2

Use these values (<Y>/N)? *press Enter*

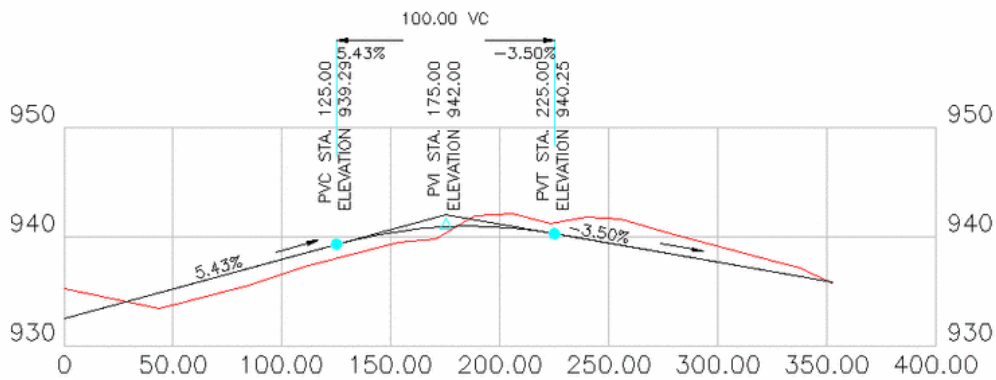
Station of next PVI or pick a point ('U' to Undo, Enter to End): *press Enter*

Vertical Curve Text Options dialog box

Pick vertical position for VC text: *Pick a position above the profile grid.* The final plot is shown below:



The Vertical Curve Text Options dialog box is a standard Windows-style window with a blue title bar and a close button (X) in the top right corner. It contains a list of checkboxes and a dropdown menu. The checkboxes are: Include K-value (unchecked), Include Sight Distance (unchecked), Include Grades (checked), Draw Vertical PVC & PVT Lines (checked), Position Text Below Line (checked), Pick Each Label Position (unchecked), and Draw Slope Direction Arrows (checked). The Number of Decimal Places dropdown menu is set to 0.00. At the bottom are two buttons: OK and Cancel.



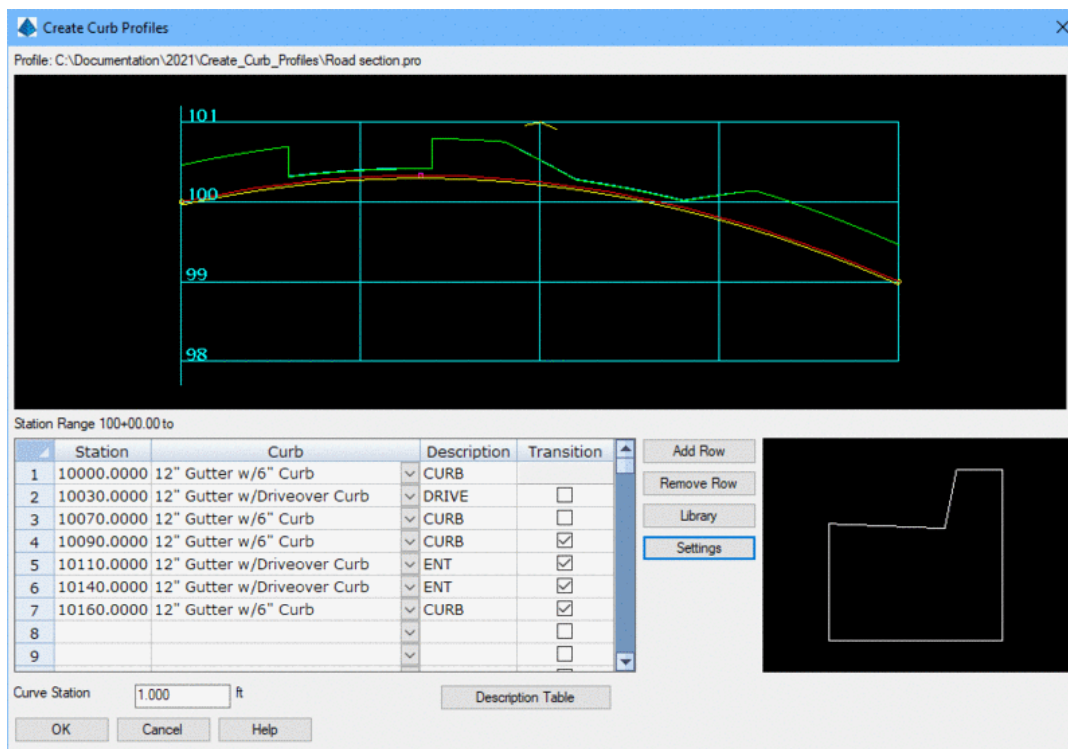
Pulldown Menu Location: Profiles

Keyboard Command: road

Prerequisite: A profile grid

Create Curb Profiles

This command allows you to build separate profiles (*.PRO) representing each daylighting portion of a curb section such as edge of pavement, gutter, face of curb and back of curb. These can then be drawn and plotted along side other profiles such as edge of pavement, centerline, ditch and sewer profiles.



Station determines which Curb Section is applied.

Curb displays Curb Section to be applied at station.

Description Used to a place label along profile if Description matches Description from the Description Table.

Transition determines if Curb Section will hold Curb Section geometry or perform a linear transition to next Curb Section station.

Add Row adds a Curb Section to the list of stations.

Remove Row deletes the highlighted Curb Section from the list of stations.

Library allows you to select a Curb Section already defined by Carlson or one you have created in a previous project using the Curb Section Library.

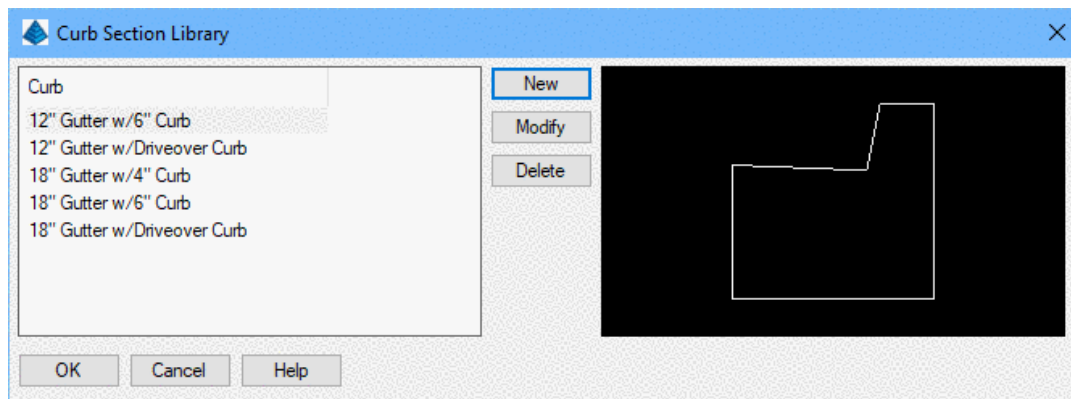
Settings to help you specify which of the profiles are created; the prefix that is applied; the color in the dialog box and whether or not these profiles are linked to the road profile.

Curve Station specifies an interval or 'densification' along vertical curves for each curb profile such that they appear curved.

Description Table allows you to automatically label the curb profiles. Enter a Description for the curb section and enter the same description in the Description Table and the desired label to be placed along the profile. You can also specify a symbol to be placed at that location.

Curb Section Library

Select a curb already defined by Carlson or one you have created in a previous project using the Curb Section Library.



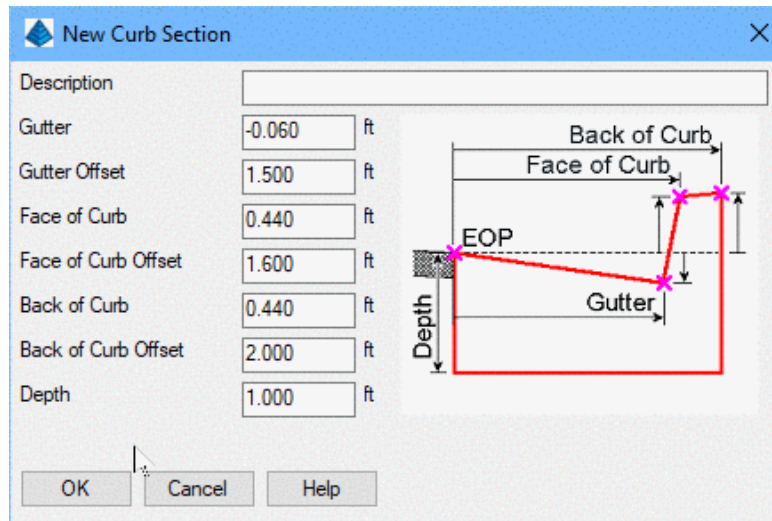
New Curb Section creates a new Curb Section.

Modify Curb Section modifies the highlighted Curb Section.

Delete Curb Section removes Curb Section from the Library.

New / Modify Curb Section

Create or modify a new curb section calculating profiles. Using the illustration as a guide on the New/Modify dialog boxes, enter the dimensions of the curb section using the Edge of Pavement (EOP) as the 'origin' or baseline for entering horizontal (Offsets) and vertical dimensions. For example, use positive offset values for the Gutter Offset; Face of Curb Offset and Back of Curb Offset as measured from the EOP toward the right. For heights above the edge of pavement such as Face of Curb and Back of Curb, use positive values above the Edge of pavement, and negative for Gutter if below the EOP. For the initial Depth of the Curb at the EOP, enter a positive value for the Depth even though this is below the EOP. NOTE: The Description entered here appears under the Curb heading on the main Curb Profiles dialog box and is not used for the purposes of labelling using the Description Table.



Description used to identify the Curb Section in the Library.

Gutter is the depth of gutter from the EOP, usually displayed as a negative value.

Gutter Offset is the bottom of gutter measured horizontally from the EOP.

Face of Curb is the height of the face of curb measured above the EOP.

Face of Curb Offset is measured horizontally from the EOP.

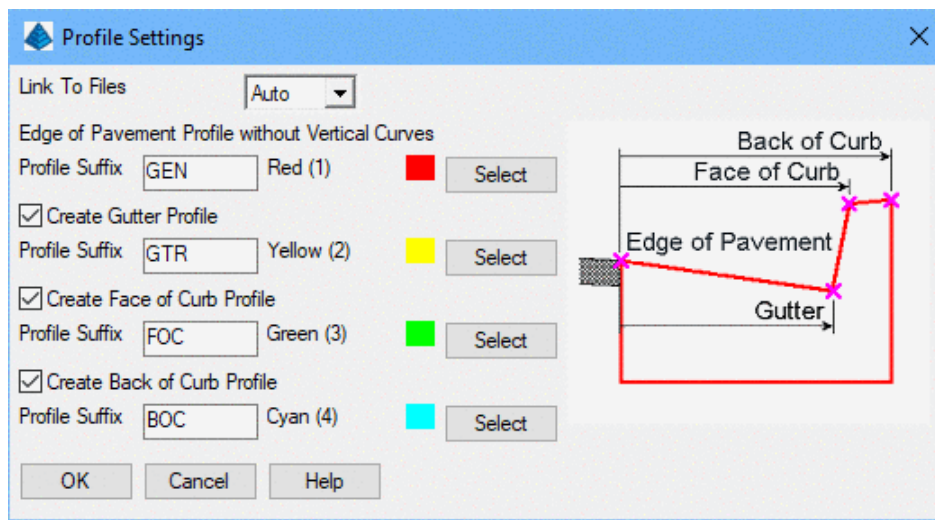
Back of Curb is the height of the back of curb measured above the EOP.

Back of Curb Offset is measured horizontally from the EOP.

Depth displays the depth of Curb Section at the EOP and entered as a positive value.

Settings

Specify which of the profiles are created; the prefix that is applied; the color in the dialog box and whether or not these profiles are linked to the road profile.



Link To Files determines how changes to the main road profile file affect curb profiles.

- **Off** indicates no changes to the curb profiles once initially created until Curb Profiles is run again.
- **Prompt** indicates changes to the road profile file trigger a prompt to the drawn curb profiles should they be updated.
- **Auto** indicates changes to the road profile file result in an automatic change to the drawn curb profiles.

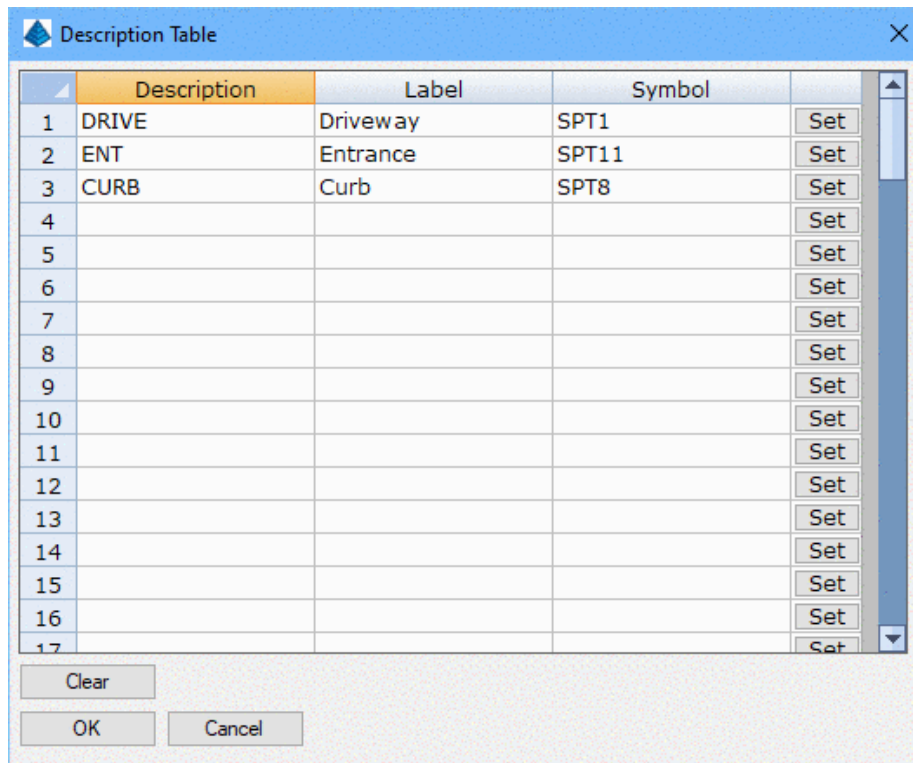
Profile Suffix is the suffix applied to the named curb profile (*.PRO) of the EOP, Gutter, Face and Back of Curb profiles.

Select allows you to set the color of the profile as displayed in the Curb Profiles dialog box. NOTE: This does not affect the color of the drawn profile in CAD.

Create toggles which profiles you are creating for all the curb profiles.

Description Table

Automatically label the curb profiles based on the Draw Stacked Labels using the Draw Profile command. The label is selected based on the description of the curb station. Additionally you can specify a symbol to be placed at the desired curb profile station.



Description matches the description of the Curb Section station. Used to determine what label and/or symbol can be drawn along the profile.

Label text placed at the profile station. You can use this in conjunction with the Draw Profile > Stacked Label to identify the Curb Section.

Symbol draws a symbol along the profile at the Curb Section station.

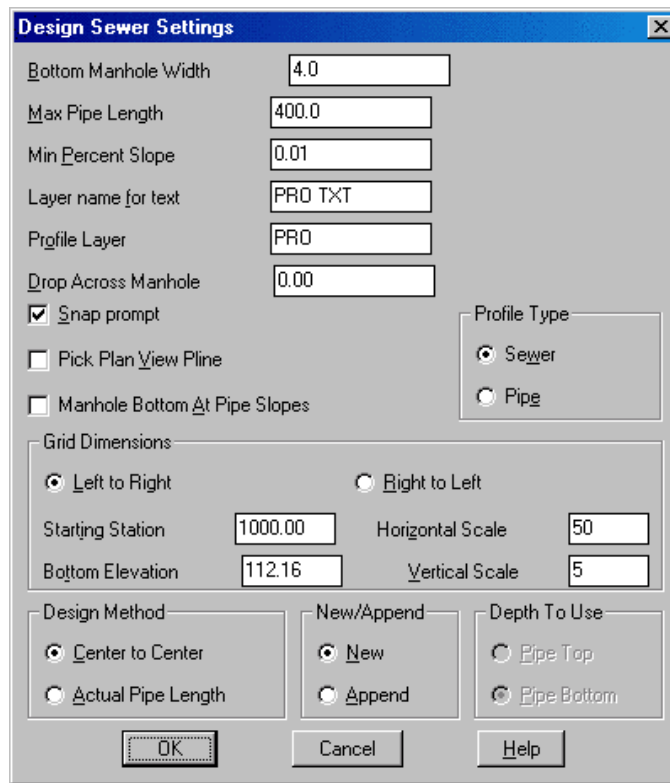
Pulldown Menu Location: Profiles > Design Road Profile

Keyboard Command: CURBPROF

Prerequisite: Design profile

Design Sewer/Pipe Profile

This command creates a sewer profile (.PRO) file with manholes, or will create a pipe profile (no manholes, no manhole width), and draws it on the screen. It requires that a grid is already drawn. It begins with the Design Sewer Settings dialog box.



Bottom Manhole Width: Specify the size for the bottom of manholes. Not available when Profile Type is set to pipe.

Max Pipe Length: Specify the maximum limit for the distance between manholes.

Min Percent Slope: Specify the minimum slope (absolute value) between manholes.

Layer name for text: Specify the layer name for annotation. If you enter a layer that does not exist, it will be created.

Profile Layer: Specify the layer name for pipes and manholes. If you enter a layer that does not exist, it will be created.

Drop Across Manhole: Specify the amount the elevation drop across the manhole in the direction of the profile. Will accept a negative a value. Not available when Profile Type is set to pipe.

Snap Prompt: Activates the PVI Snap dialog box. See below for description.

Pick Plan View Polyline: Allows you to select a polyline from plan view that represents the sewer centerline. This leads to the plotting of manhole symbols on the plan view and also creates default manhole-to-manhole stations.

Manhole Bottom At Pipe Slopes: When checked, the manhole bottom will be drawn level with the pipe slope.

Profile Type: Choose between Sewer profile or Pipe profile. Pipe profile do not include manholes.

Grid Dimensions: Specify the grid dimensions on which the sewer will be designed.

Design Method: Choose whether distances specified are center or manhole to center of manhole or actual pipe length. Not available when Profile Type is set to pipe.

New/Append: Choose between creating a new profile (.PRO) file or appending an existing file.

Depth to Use: Choose between specifying pipe top or pipe bottom elevations. Not available when Profile Type is set to sewer.

Prompts

File Selection dialog

Choose a new profile file name to create.

Pick Lower Left Grid Corner <5000.0,5000.0>[endp on]: *pick the corner*

Select existing ground polyline or ENTER for none: You may optionally pick a polyline to use for calculating the depth from the surface as the sewer stations are entered.

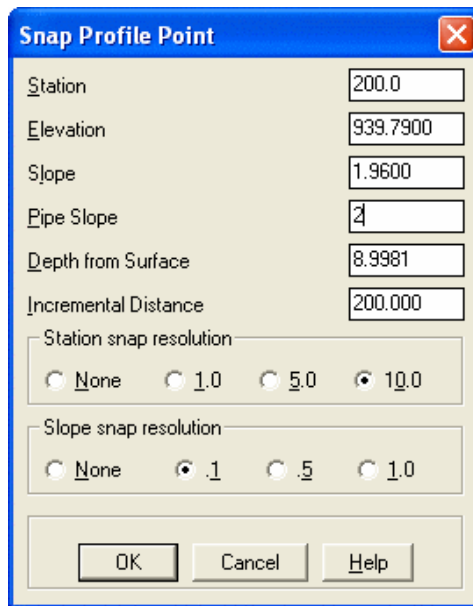
Enter station or pick a point (Enter to End): 0

Depth from Surface/<Elevation of manhole>: 935.7

Enter the step up/down in feet <0.00>: *press Enter*

Station of second MH or pick point (U,E,D,Help): *pick a point*

If the Pick Plan View Polyline option has been chosen, the program will default to the station of the next vertex in the selected polyline. If the Prompt for Snap option was selected in the main dialog, then the Snap Profile Point dialog appears here. The station and slope may be changed to the nearest snap value. The elevation is the free variable and it will change to compensate for any snap. To change the elevation, select the elevation edit box and enter the new value.



The image shows a dialog box titled "Snap Profile Point" with a close button (X) in the top right corner. The dialog contains several input fields and two groups of radio buttons. The fields are: Station (200.0), Elevation (939.7900), Slope (1.9600), Pipe Slope (2), Depth from Surface (8.9981), and Incremental Distance (200.000). The "Station snap resolution" group has radio buttons for None, 1.0, 5.0, and 10.0, with 10.0 selected. The "Slope snap resolution" group has radio buttons for None, .1, .5, and 1.0, with .1 selected. At the bottom are three buttons: OK, Cancel, and Help.

Enter the step up/down in feet <0.00>: *press Enter* Enter 0.1 if pipe drops one tenth into manhole and you are designing in upstream direction.

If you enter a station for the next manhole rather than picking a point on the screen, then you will be prompted as follows:

Depth/Percent grade/Min grade/<Elevation of manhole>: 939.79

Size of pipe in inches <10.0>: 8.0

Station of next manhole or pick a point (U,E,D,Help): *press Enter*

If you picked a plan view polyline, you will be asked:

Draw manholes on centerline [Yes/<No>]?: Y Then you will be prompted for the default manhole symbol to use.

Profile Sewer Settings dialog

Sewer Label Options dialog

Sewer Annotation Options dialog (Displayed by pressing the Annotation Options button.)

Select existing ground polyline: *pick a polyline or press Enter to be prompted for each manhole surface elevation*

This prompt only appears if no ground polyline was selected above.

Manhole No. 1 label [MH #1]: *press Enter*

Manhole No. 2 label [MH #2]: *press Enter*

Draw Sewer Options [X]

General | Manhole | Pipe

Draw Horiz Axis Annotations
 Draw Annotations At Manholes
 Tick Mark for Station
 Label Invert Elev w/ Vert. Line
 Station Manholes By Another Centerline

Rim Label Position ...
 None
 Label with Leader
 Above Manhole

Draw Sump Sump Height: 2.00
 Draw Base Base Height: 1.00
 Label Precision

Sewer Label Position
 Pick Row Above Manhole

Draw Station Label: STATION
 Draw Reference CL Offset Label: OFFSET
 Draw Surface Label: SURFACE
 Draw Invert In Label: INVERT IN
 Draw Invert Out Label: INVERT OUT
 Draw Description Label: DESC

OK Cancel

Draw Sewer Options [X]

General | Manhole | Pipe

Manhole Name in Circle
 Manhole Name in Hexagon
 Draw Manhole Sides Down To Invert
 Manhole Rim Offset Prompt
 Manhole Width Prompt
 Manhole Bottom At Pipe Slopes

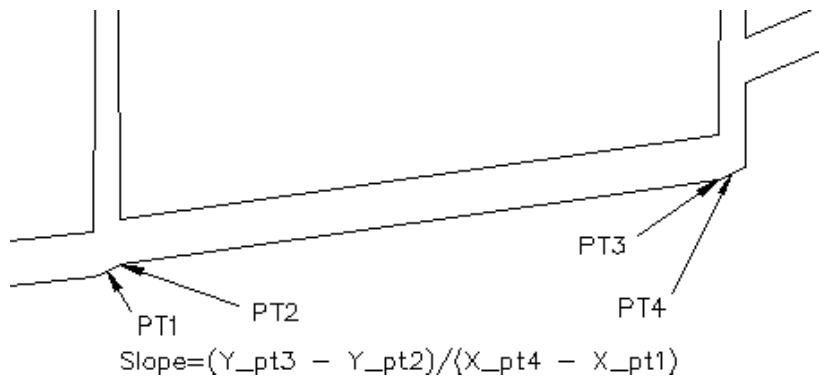
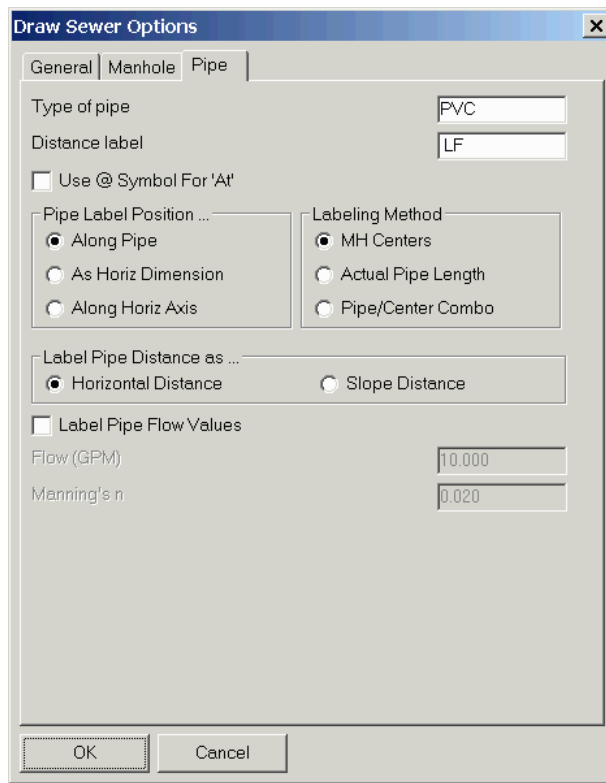
Drop Across Manhole: 0.00

Horiz Axis Text Orientation
 Horizontal Vertical

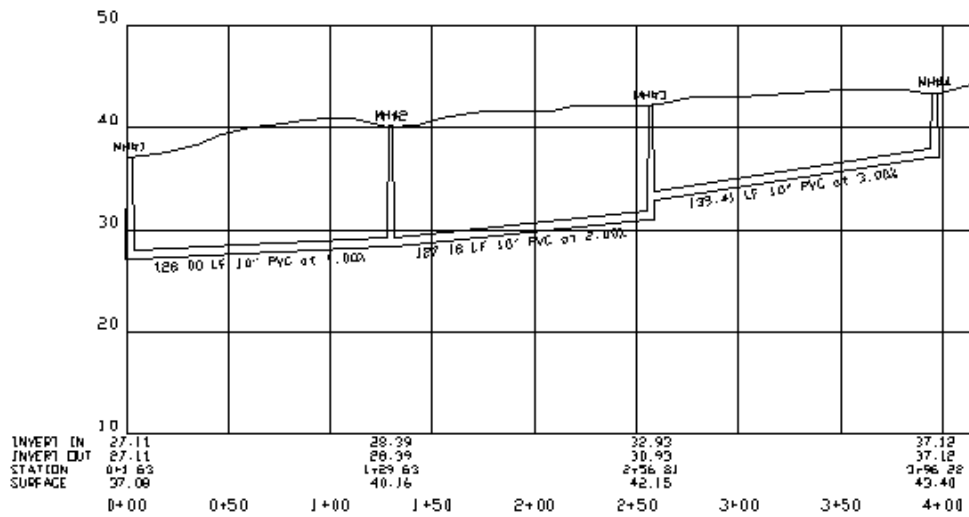
Taper Format
 Symmetric Taper Left Taper Right

Manhole Dimensions
 Manhole Top Width: 2.00
 Manhole Bottom Width: 4.00
 Top Taper Offset: 100.00
 Fixed Taper Height: 0.00

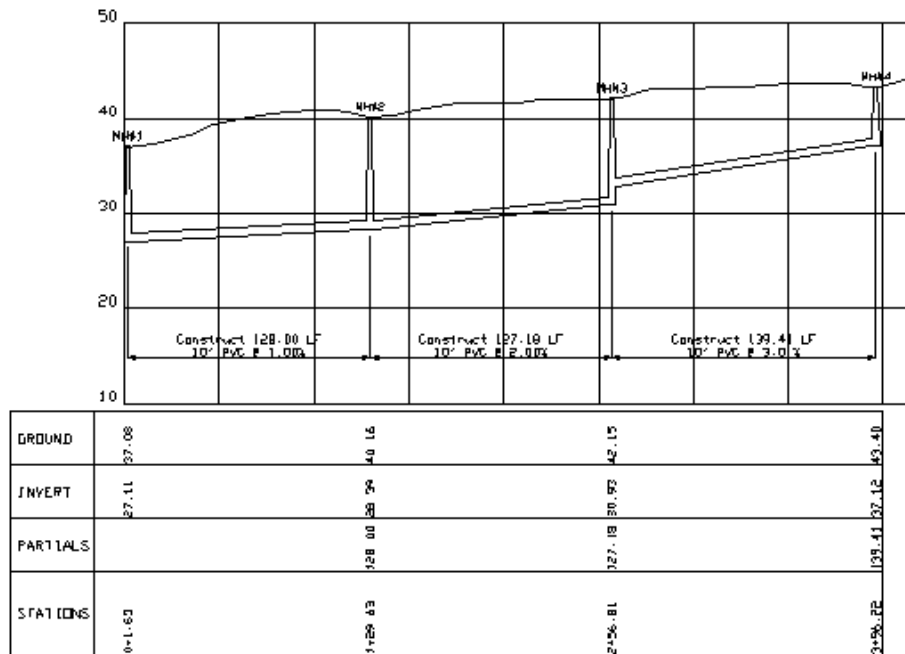
OK Cancel



Pipe/Center Combo Labeling Method calculates the slope as the elevation difference from the edge of the pipe, divided by the distance between the manhole centers.



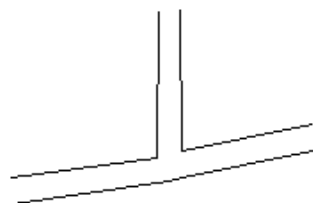
Example of sewer profile and surface profile



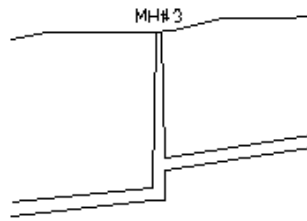
Example of sewer profile using Horizontal Axis Text Orientation as Vertical and Pipe Label Position as Horizontal Dimension



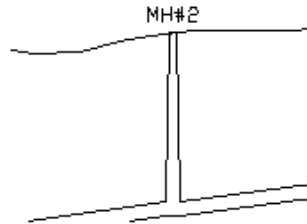
Detail of manhole bottom at pipe slope



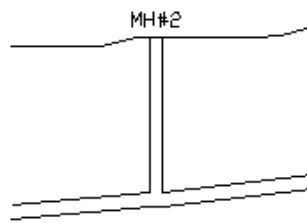
Detail of drop across manhole of 0.2



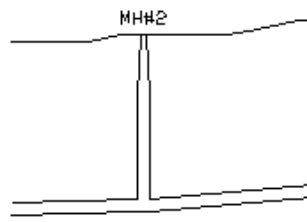
Detail of step up



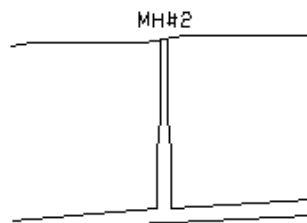
Top=2, Bottom=4, Offset=100



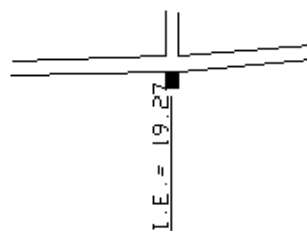
Top=4, Bottom=4



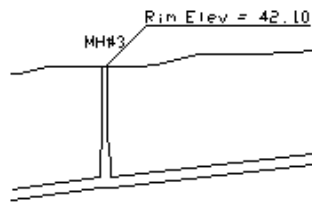
Top=2, Bottom=4, Offset=4, Fixed=0



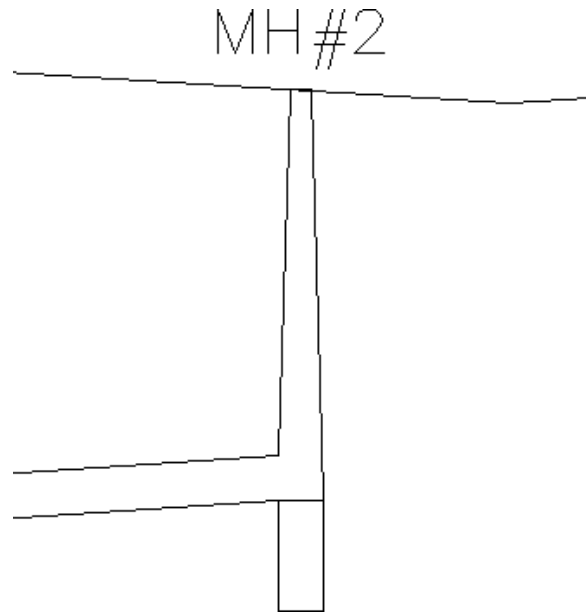
Top=2, Bottom=4, Offset=4, Fixed=2



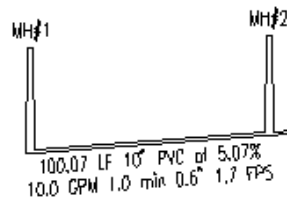
Detail of Draw Manhole Base and Label Invert Elevation with Vertical Line



Detail of Label Rim Elevation at Manhole



Manhole with the Draw Sump option



Label Pipe Flow Values option shows flow rate, travel time, depth and velocity

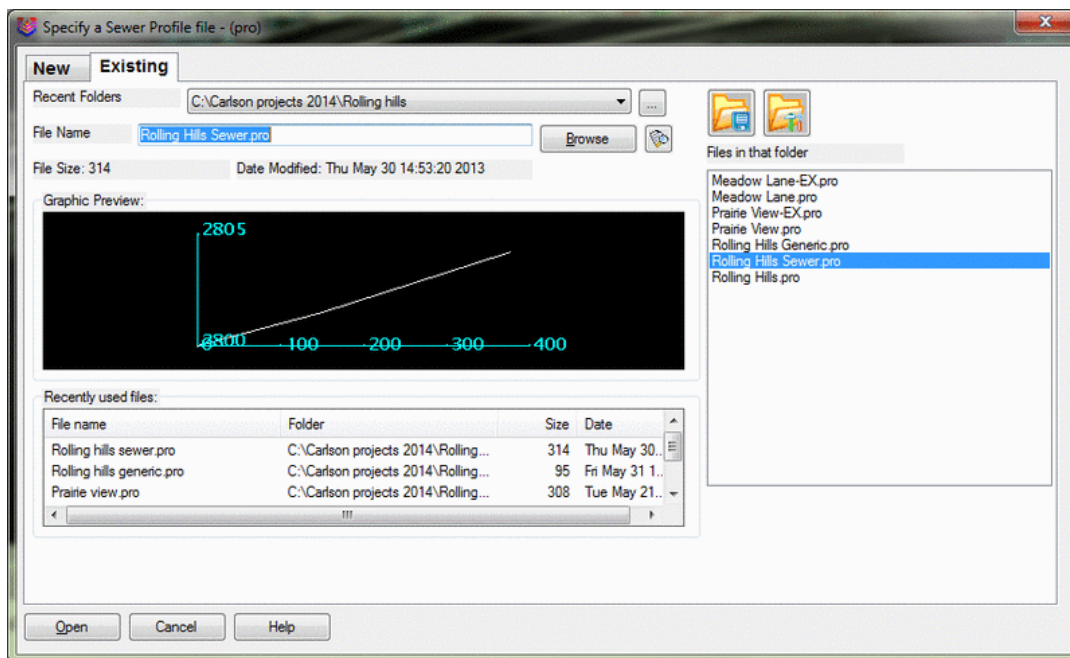
Pulldown Menu Location: Profiles

Keyboard Command: sewer

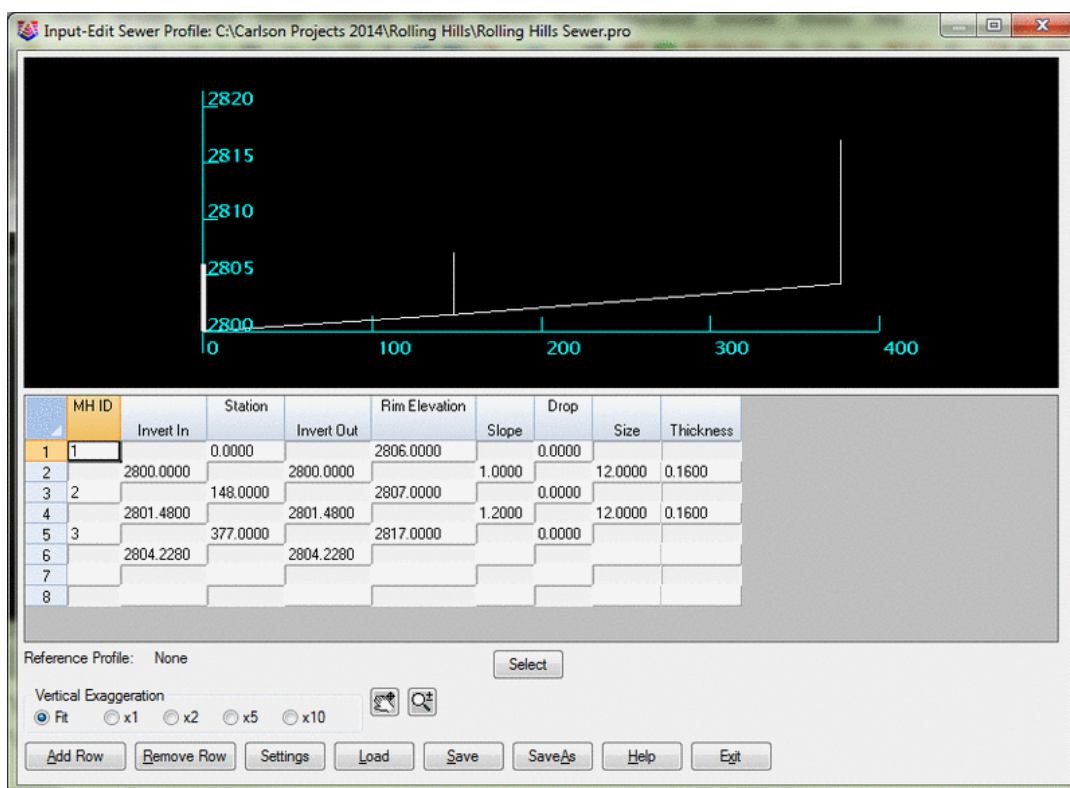
Prerequisite: A profile grid

Sewer/Pipe Profile Editor

This command allows for a way to enter and edit sewer profiles in a spreadsheet type editor formatted specifically for this purpose. In order to start a new profile must be created or an existing profile selected.



The dialog is staggered to make it clear how the sewer profile is structured. It provides for the ability to specify inverts in and out, rim elevations, slopes of the pipes, drops across the manhole and sizes and thicknesses for the pipes. The preview pane has standard controls for panning, zooming and setting the exaggeration for easy viewing.



Reference Profile: To show a reference profile choose the Select button and select a profile to use.

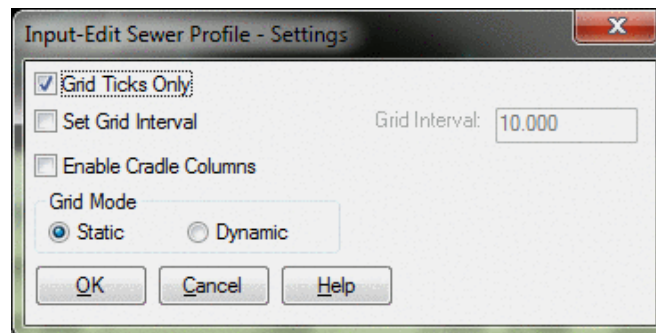
Vertical Exaggeration: Use the toggles here to fit or set the horizontal versus vertical exaggeration.

Pan and Zoom: Use the Pan and Zoom buttons to change the operation of the mouse in the preview window.

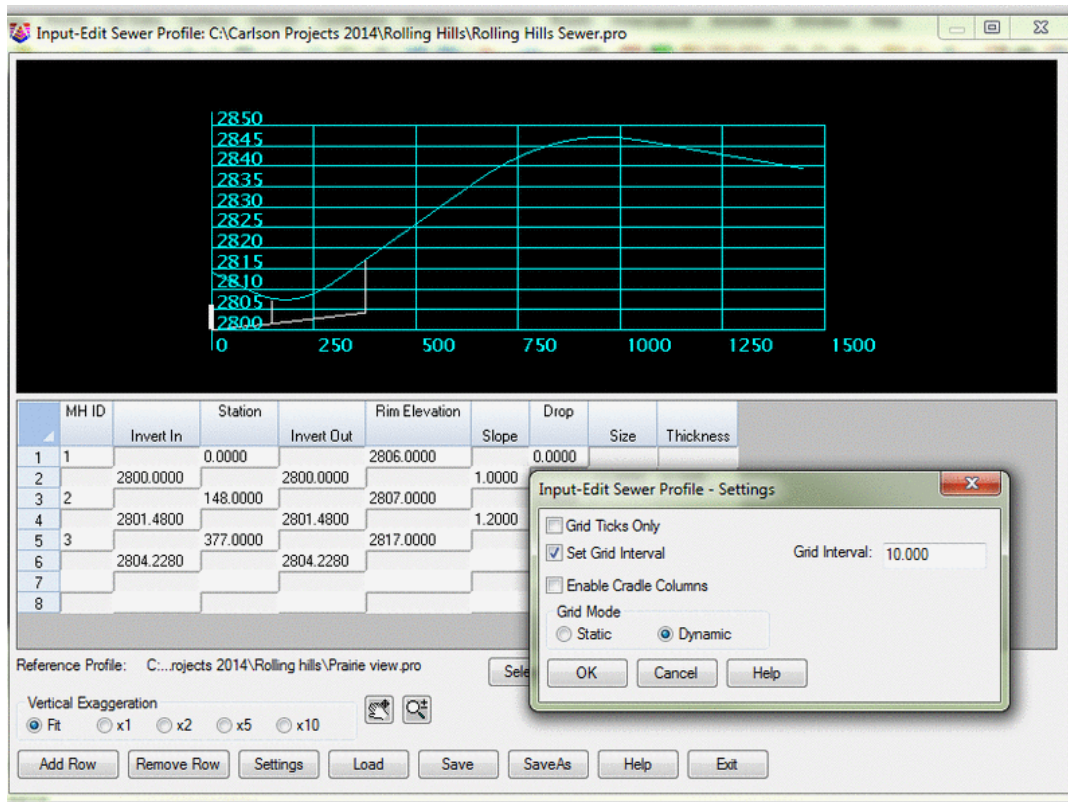
Add Row: Use the Add Row button to add a row. Highlight a row and then choose the Add Row button.

Remove Row: Select the row to remove and then select the Remove Row button.

Settings: Choose either Grid Ticks only or Set Grid Interval to change the preview display of the grid. Choose Enable Cradle Columns to add input for Cradles. Set static or dynamic to change the horizontal spacing of the grid lines.



Results of altered settings and a reference profile:



Load: Use the Load button to load a different profile.

Save: Use the Save button to save the profile.

SaveAs: The SaveAs button allows the saving of the profile with a different name.

Help: Launch help for this command with the Help button.

Exit: Use the Exit button to exit the command a prompt may ask for saving or not saving changes.

Pulldown Menu Location: Profiles

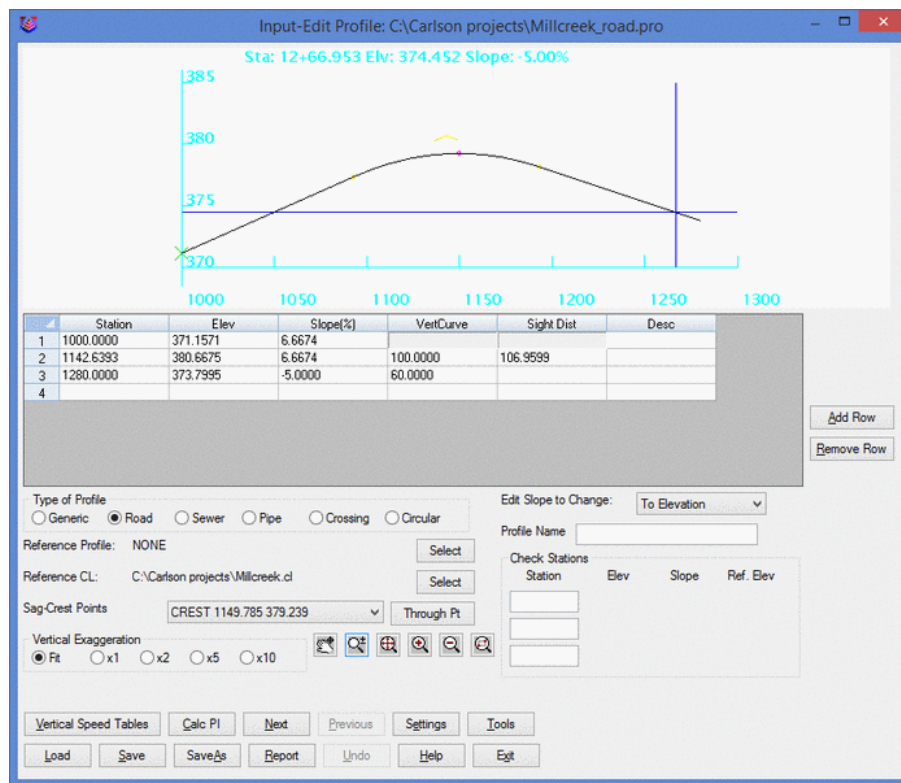
Keyboard Command: swrprofedit

Profile Editor

Similar to the Road Profile Editor command, this command features a spreadsheet type editor and handles a variety of profile (.PRO) configurations. Besides editing a profile, this routine can be used to just view the contents of a profile.

The command starts by prompting for the profile file to edit. Alternately, you can run Profile Editor by double-clicking on a profile polyline that is drawn on a profile grid.

The opening dialog below shows the layout of this editor. At the top of the dialog, you can dynamically see the profile and vary its appearance by using zoom and pan. The station, elevation and slopes are also shown at the lower left of the dialog which update/track with the movement of the cursor. There are between five and nine possible data fields in a profile depending on the type of profile that has been selected.



Profile Name: This name is optional and often used when multiple profiles are stored in a profile (.PRO) file and graphically generated using the Draw Profile command.

Add Row: Adds a new row into the profile after the current row.

Remove Row: Removes the current row.

Type of Profile: There are 6 types of .pro files and the spreadsheet columns will change to match the data fields for the selected profile type:

- *Generic* - Generic profiles have station, elevation and description fields.
- *Road* - Road profiles include the Generic controls and adds a vertical curve field. For an asymmetrical vertical curve, enter the left and right side values separated by a dash in the spreadsheet cell. For example, a 200' vertical curve with 50' to the left of PVI and 150' to the right would be entered as "50-150".
- *Sewer* - Sewer profiles include the Generic controls and adds step up, pipe size, pipe thickness, manhole elevation and manhole ID fields.
- *Pipe* - Pipe profiles include the Generic controls and adds a pipe size field.
- *Crossing* - Crossing profiles are for pipe crossings along the centerline. Besides station and elevation, the crossing data points also have the pipe size. The crossing elevation is for the bottom elevation of the pipe. The crossing profile data points are not connected.
- *Circular* - Circular profiles are the same as Road profiles except the vertical curve is circular instead of parabolic.

Edit Slope To Change: This setting controls which field to update when the slope is modified in the spreadsheet.

Reference Profile: Selects a reference profile and displays it in the profile graphic view. Also a Depth column is added to the spreadsheet which you can edit to set the profile elevation by depth from the reference profile.

Reference CL: Selects a reference centerline which is used for station equations.

Sag-Crest Points: When editing a road profile, its sag/crest points are shown here.

Through Pt: This button lets user to make the road profile pass through a certain point.

Vertical Exaggeration: Changes the look of the profile.

Edit Slope to Change: When edit the slope value, you can choose a value to change from the following selection: the next elevation, the previous elevation, the next station or the previous station.

Check Stations: Reports profile information at the specified stations. The Check Stations are not stored in the profile; they are merely used as a design/analysis tool for viewing the elevations at certain stations while adjusting the profile data. The **Offset** and **Slope** adjust the profile elevation and report the elevation at the offset.

Speed Tables: This button is enabled only when you edit a road profile. Please refer to the documentation on Road Profile Editor for the information on Vertical Speed Tables.

Next: Used for navigation when editing a .PRO file containing multiple profiles, loads the next profile.

Previous: Used for navigation when editing a .PRO file containing multiple profiles, loads the previous profile.

Load: Used for loading another, existing .PRO file for editing.

Save: Saves the profile using the current profile file name. The current profile file name is displayed in the top title bar of the dialog box.

SaveAs: Allows you to save the profile under a different profile file name.

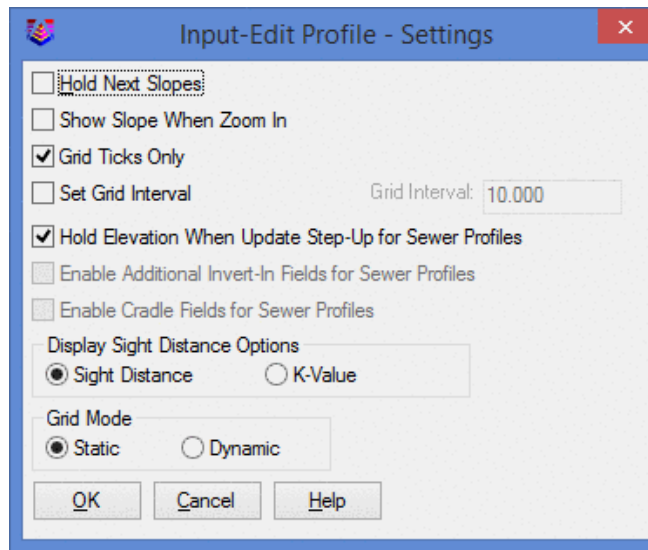
Calc PI: This function calculates a station/elevation point given two existing station/elevation points and slopes from them. The values are entered in this dialog. When you pick Calculate, the program finds the intersection of the grade lines. Then pick OK and the calculated PVI is added to the profile.

Calculate Intersection Point	
Intersection at Station: 1630.000 Elevation: 997.200	
Starting Station	790
Starting Elevation	993
Forward Station	1850
Forward Elevation	999.4
Slope From Starting Position to PI (percent)	0.5
Slope From Forward Position to PI (percent)	-1.0
[Calculate] [OK] [Cancel]	

Report: Creates a report of current profile.

Undo: Reverts the last action in the editor.

Settings: Opens the settings dialog.



Hold Next Slopes: When editing a profile elevation, this option will maintain all the slopes after the edit point by adjusting the elevations. Otherwise, the elevations for the rest of the profile points are held and the slope from the edit profile point to the next profile point is adjusted.

Show Slope When Zoom In: This option allows to display the slopes on the long enough profile segments when zoom in.

Grid Ticks Only: Toggles between displaying the grid and grid ticks in the graphic box.

Set Grid Interval: This option allows you to control the elevation grid spacing in the graphic preview. When this option is off, the program automatically figures the elevation grid interval.

Hold Elevation When Update Step-Up for Sewer Profiles: Toggles between holding the elevation or the pipe slope

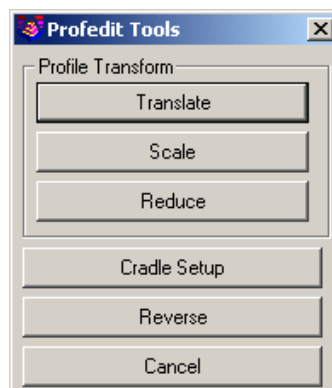
Enable Additional Invert-In Fields for Sewer Profiles: When editing a sewer profile, this option allows you to display an extra invert-in column for in-coming pipes. The invert-in elevations are separated by commas.

Enable Cradle Fields for Sewer Profiles: When editing a sewer profile, this option allows to display cradle above and cradle below columns.

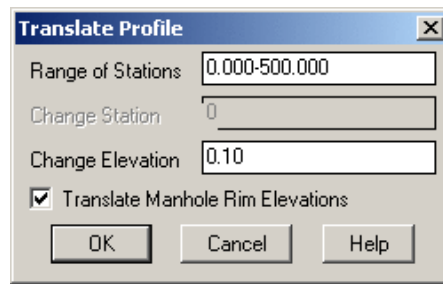
Display Sight Distance Options: Display either K-Value or Sight Distance in the fifth column for road profiles.

Grid Mode: The **Dynamic** option will update the grid interval labels when you zoom in or out of the profile image. The **Static** option will keep the grid interval labels static.

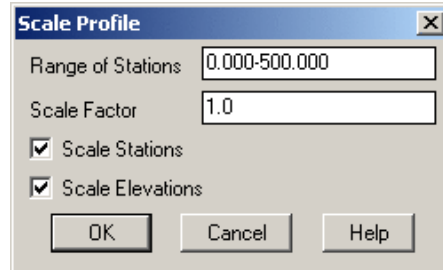
Tools: Opens the Tools dialog.



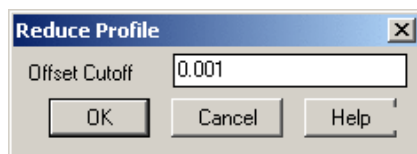
Translate: Globally adds or subtracts value to stations and/or elevations within the specified range of stations. while Scale will apply the specified scale factor to stations and/or elevations within the specified range of stations.



Scale: Applies the specified scale factor to stations and/or elevations within the specified range of stations.

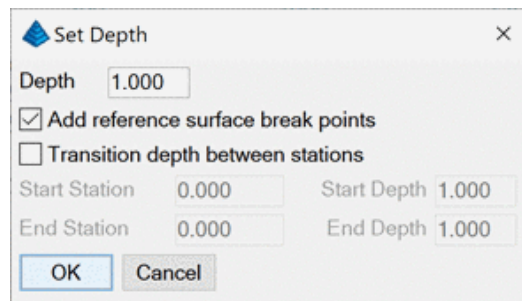


Reduce: Reduces the profile points by the Offset Cutoff value.

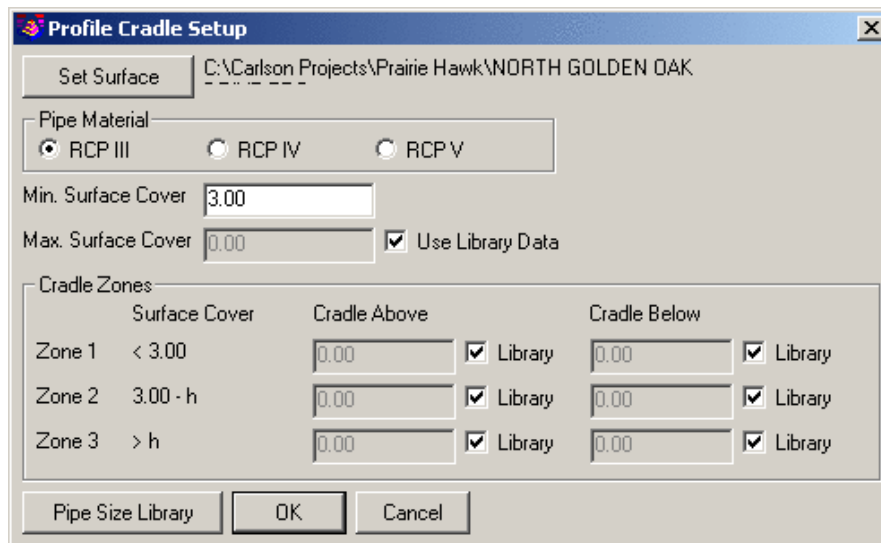


Reverse: Reverses the direction of the stationing for the profile.

Set Depth: When using a Reference Profile, this function sets the profile elevations by a specified depth from the reference profile.



Cradle Setup: Sets up the cradles for sewer profiles. The cradle parameters are different with different pipe sizes and are defined in the Pipe Size Library. You can either use library data or specify new values here.



Pulldown Menu Location: Profiles

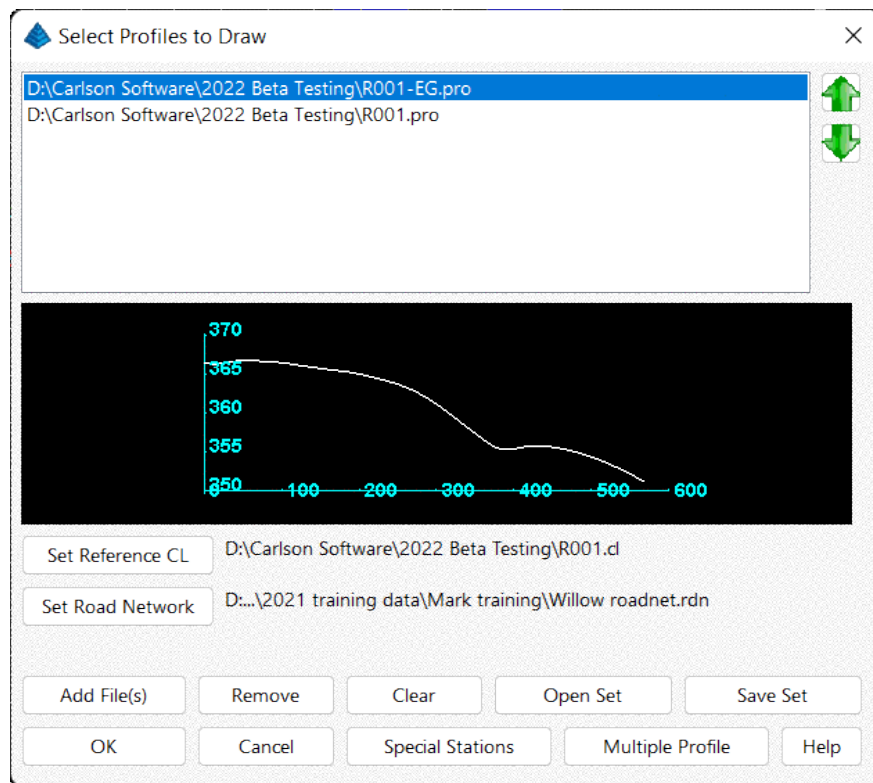
Keyboard Command: profedit

Prerequisite: None

Draw Profile

Draw Profile is a flexible routine for drawing a profile anywhere in the drawing. The profile can be placed in a layout tab or in the model space of the drawing. It may be drawn with or without a grid or with just tick marks. The vertical curve annotations for a road profile and manhole annotations for a sewer profile, may also be drawn. Draw Profile uses the profile information that is stored in .PRO files. It is also able to use a reference centerline stored in a .CL file. Once the profile is drawn using Draw Profile, the design and labeling routines of the Profiles dropdown are applicable to the profile. Please note, several of the options presented in the following dialogs will depend on the type of unit system being used, metric or English. Options such as text sizes, sheet dimensions, and scaling factors may vary from the examples shown here.

Select Profiles to Draw



Reference CL: When using Draw Profile to create Plan & Profile Sheets, a reference centerline will be needed to properly establish the "Plan" portion of the sheet. Use the **Set** button to locate and specify the desired Reference Centerline.

Set Road Network: Use this to set the current Road Network File (.RDN)

Add File(s): Use this option to add desired Profile (*.PRO) files one at a time into the dialog box.

Remove: Use this option to remove the currently selected profile the dialog box.

Clear: Use this option to remove all profiles from the dialog box.

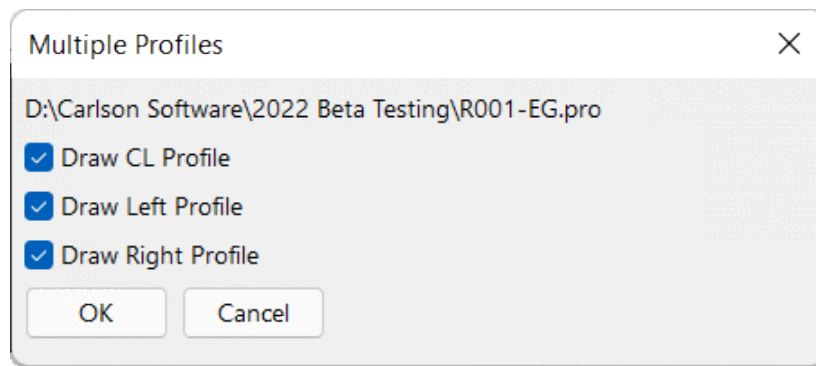
Open Set: Opens/loads a previously saved set of Profile files as stored in a *.PST file.

Save Set: Permits the current collection of profiles to be saved to a *.PST file for later recall.

Special Stations: Use this option to to label stations that are significant for the project.

Multiple Profiles: Use this option to add desired Profile (*.PRO) files (one or several at a time via standard Windows Shift+click and/or Ctrl+click functionality) into the dialog box.

Multiple Profile: For *.PRO files that have multiple profiles within the same file (*e.g.* those produced by the Profile Editor command that might have profile data at the centerline and also at offsets of -12 and +12), use this button to select which of the profiles to draw:



Another type of multiple profile is a profile with breaks such as a curb profile with breaks across intersections. This is built-in to Road Network (*aka* RoadNet). In RoadNet Settings (specifically, the Output Options tab), there's **Output EOP Profiles**. RoadNet will then create profiles with descriptions at the intersections and Draw Profile will break the profile across the intersections. You can also manually put in these descriptions in Profile Editor to set the breaks. Use **EP_FL:@E** to stop the profile and **EP_FL:@S** to resume the profile.

When the Select Profiles To Draw dialog has been set up, select the OK button to move on to the *Draw Profile* dialog box.

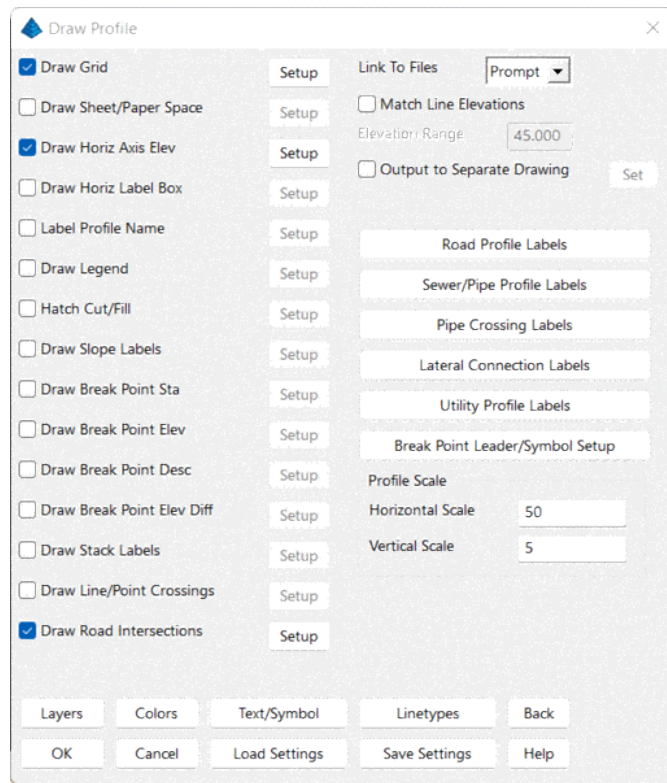
Draw Profile

Within the Draw Profile dialog box are numerous sub-commands that permit great flexibility in how the resulting profile(s) are drawn. Use the table below to access additional information about each of these sub-commands:

Draw Grid	Draw Sheet/Paper Space Label Profile Name	Draw Horiz Axis Elev
Draw Horiz Label Box		Draw Legend
Hatch Cut/Fill	Draw Slope Labels	Draw Break Point Sta
Draw Break Point Elev	Draw Break Point Desc	Draw Break Point Elev Diff
Draw Stack Labels	Draw Line/Point Crossings	Draw Road Intersections
Road Profile Labels	Sewer/Pipe Profile Labels	Pipe Crossing Labels
Lateral Connection Labels	Utility Profile Labels	
Layers/Colors	Text/Symbol	Draw Break Leader/Symbol Setup Linetypes

Draw Profile "Sub-Commands"

For each of the sub-commands above that have a check-box toggle, enable the toggle and use the appropriate Setup button to further access the option.



Link To Files: This setting controls the linkage of the plotted profile(s) to the actual profile file(s) (.PRO), determining how changes to the file affect the plotted profile(s):

- **Off:** Changes to an underlying profile file do not trigger a change to its drawn profile.
- **Prompt:** Changes to an underlying profile file trigger a prompt if its drawn profile should be updated.
- **Auto:** Changes to an underlying profile file result in an automatic change to its drawn profile.

Match Line Elevations: For high relief profiles that might otherwise exceed the profile limits of the sheet (typically dependent on the Vertical Scale), the Match Line Elevations option can be used to "break" (or vertically split) the profile and redraw the remaining portion (again, shifted vertically) to remain in the profile portion of the sheet.

Elevation Range: This is the range of elevations that is used in conjunction with the Match Line Elevations option. If the range is exceeded (that is, if the range is greater than 80), the program will break the profile and draw the remainder with a separate vertical axis range.

Output to Separate Drawing: When enabled, this option draws the profile(s) to a separate drawing. Use the **Set** button to specify the name/location of the external drawing. Suggested uses for this feature are when profile-only sheets need to be generated and provided to others for detailing or construction purposes.

Horizontal Scale: Indicate a real-world distance that should be used within the span of a horizontal grid cell.

Vertical Scale: Indicate a real-world elevation amount that should be used within the span of a vertical grid cell.

Layers

The Layers button permits layer names to be associated with items including the profiles, profile grid and general labels.

Sewer Label Layer Use Profile Layer option sets the profile layer name as a prefix or suffix to the sewer profiles and text layers

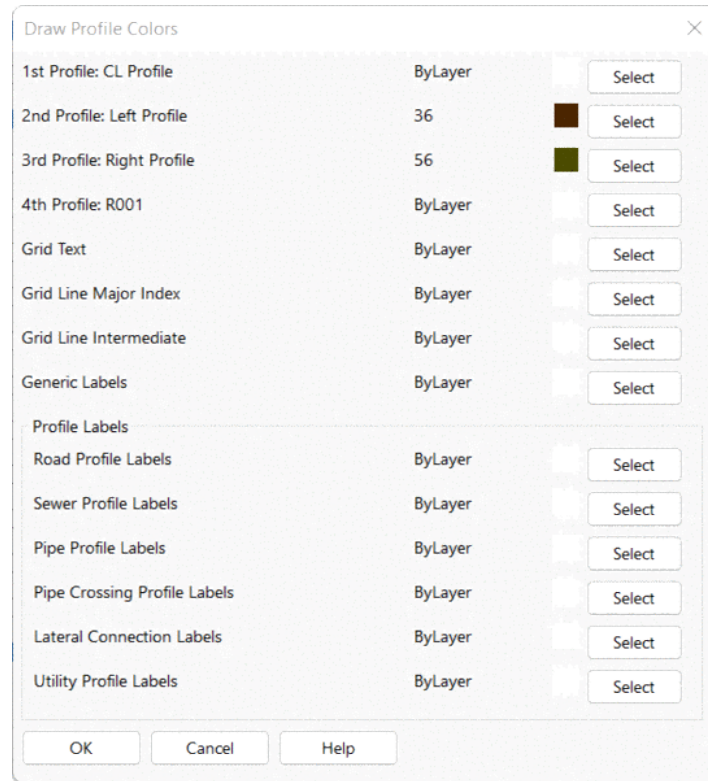
Set Level by Reference CL sets the level of the profile entities based on the name of the profile which can be seen in the Level Manager under the View menu.

The Prefix and Suffix options allow the user to add a prefix or suffix in various ways to the default layer names shown above.

Profile Line Layers displays the names of the profiles selected and allow the user to control which layer each profile will be placed on. If more profiles are selected, they will be displayed.

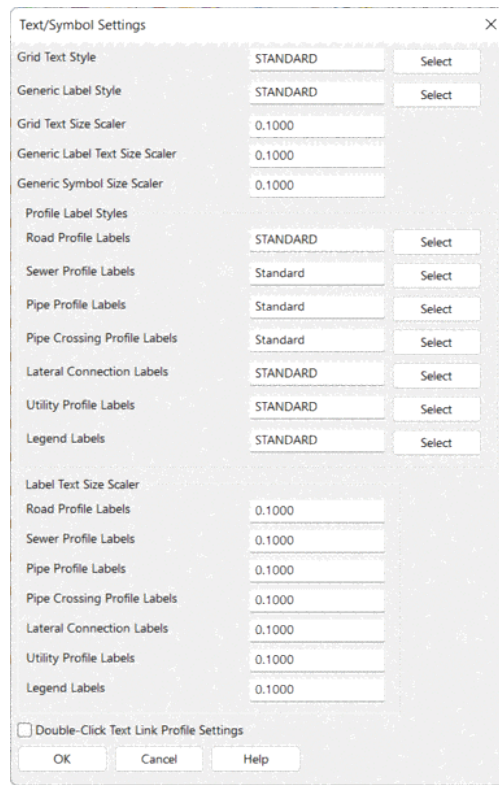
Colors

The Colors button provides the ability to control the color associated for items such as the profiles, profile grid and general labels. Any color selected will override the default ByLayer.



Text/Symbol Settings

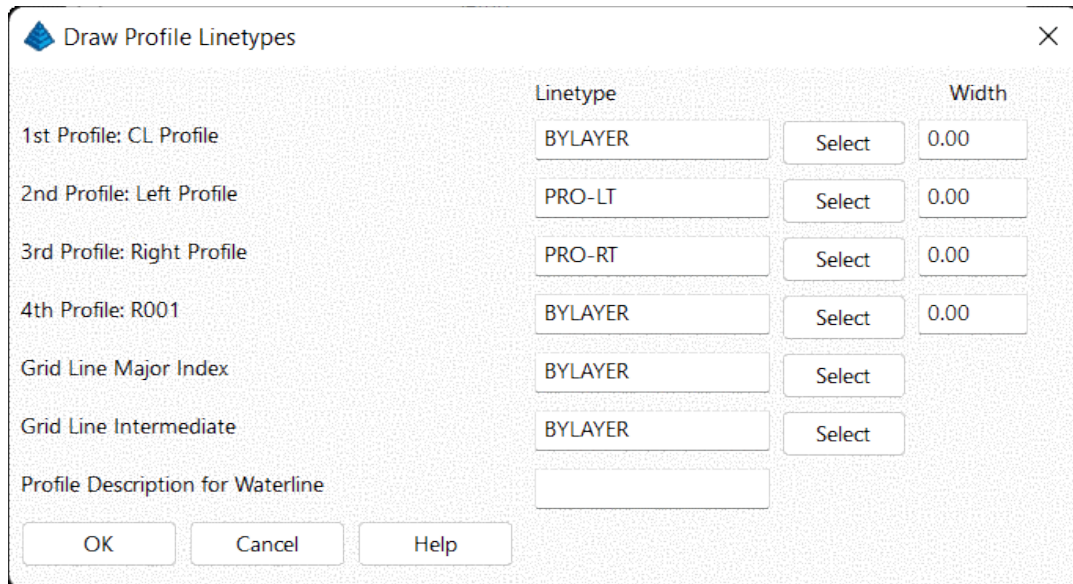
The Text/Symbol button has the text style and size scalars for profile grid and general labels. The size scalars are multiplied by the profile Horizontal Scale to determine the text size in drawing units.



Double-Click Text Link Profile Settings: When enabled, double-clicking a drawn profile text item launches the Draw Profile Settings for that text item. When disabled, double-clicking the text item launches the native CAD command (*e.g.* Edit Text).

Linetypes

The Linetypes button has lincotype settings for the profile line and profile grid. There are also settings for drawing a profile legend of the profile lines.



Label Profile Name on Profiles adds the name of the profile on the profile in a user defined location.

Profile Description for Waterline will draw a horizontal blue line at the profile elevation for the profile point with a description that matches this setting.

Draw Legend when enabled, will draw a legend of linetypes on the sheet(s) created when the **Draw Sheet** option is selected.

Draw Border with Wipeout places a wipeout component under the legend.

Line Length defines the length of the box that is drawn around the legend.

The **Horizontal** and **Vertical Offsets** control the placement of the legend relative to the sheet border.

Back: Allows you to return to the previous dialog box to alter or adjust the information it provides.

Load Settings: Loads a saved collection of Draw Profile settings, saved in a (.PFS) file.

Save Settings: Saves all Draw Profile settings in a (.PFS) file. Use this to be able to use all the settings on a future project.

Draw Grid

The Draw Grid option controls whether or not the grid and axis elevations for the profile are drawn. The **Setup** button launches the Grid Setup dialog. Here you can control numerous settings for how the grid, general text and stationing are drawn.

The screenshot shows the 'Grid Setup' dialog box with the following settings:

- Intervals:** Horizontal Grid: 25, Vertical Grid: 1, Horizontal Major Grid: 50, Vertical Major Grid: 5, Station Text: 50, Elevation Text: 5.
- Vertical Grid Adder:** Top: 10,000, Bottom: 15,000.
- Horizontal Grid Adder:** Left: 50,000, Right: 50,000.
- Grid Style:** Grid Lines, Grid Direction: Left to Right.
- Draw Vertical Bar on Right:** unchecked, Draw Vertical Bar separately: unchecked, Offset: 0,500.
- Tick Size:** Major, Minor.
- Label Stations:** checked, Label Station Equations: checked, Label Base Elev: unchecked, Label Scale: checked.
- Grid Title:** unchecked.
- Station Type:** 1+00.
- Station Text Orientation:** Horizontal, Station Text Position: Bottom.
- Draw Grid Line at PC/PT Station:** unchecked, Use Partial Labels for Intermediate Stations: checked, Increment Station Text from Beginning Station: unchecked.
- Label Elevations:** checked, Suffix: , Elev Text Vertical Justify: Bottom.
- Draw Elevation Labels Only On Left Side:** unchecked, Draw Grid Line Under Elevation Labels: unchecked.
- Elevation Text Offset Left:** Offset Scale: 0,100, Pick.
- Elevation Text Offset Right:** Offset Scale: 0,100, Pick.
- Offset Station Text:** checked, Offset Scale: 0,800, Pick.
- Stack Profile Grids:** unchecked, Setup.
- Draw Elevation Bar:** unchecked, Skip Grid Range Dialog: unchecked.

Horizontal Grid: Enter a value of how often grid lines should be displayed to coincide with the station values along the horizontal axis of the grid.

Horizontal Major Grid: Enter a value of how often major (or "heavy") grid lines should be displayed to coincide with the station values along the horizontal axis of the grid.

Station Text: Indicate how often station text labels should appear along the horizontal axis of the grid.

Vertical Grid: Enter a value of how often grid lines should be displayed to coincide with the elevation values along the vertical axis of the grid.

Vertical Major Grid: Enter a value of how often major (or "heavy") grid lines should be displayed to coincide with the elevation values along the vertical axis of the grid.

Elevation Text: Indicate how often elevation text labels should appear along the vertical axis of the grid.

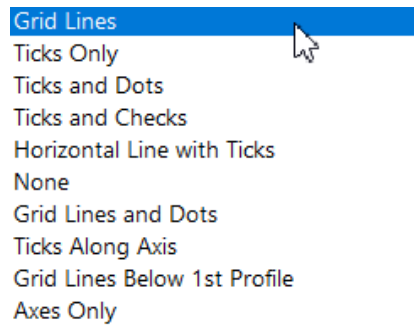
Vertical Grid Adder to Top: This adds the specified amount of grid to the top of the profile.

Vertical Grid Adder Bottom: This adds the specified amount of grid to the bottom of the profile.

Horizontal Grid Adder to Left: This adds the specified amount of grid to the left of the profile.

Horizontal Grid Adder to Right: This adds the specified amount of grid to the right of the profile.

Grid Style: This selects the type of Grid to generate. The choices are Grid Lines, Ticks Only, Ticks and Dots, Ticks and Checks, Horizontal Lines with Ticks, None, Grid Lines and Dots, Ticks Along Axis, and Grid Lines Below 1st Profile



Grid Direction: Profiles can be drawn Left to Right (the default) or Right to Left. Although most profiles are drawn left to right, if you have a road that runs East to West and you wish to draw the profile stationing beneath the actual road stationing, then choosing a Right to Left profile may be appropriate.

Draw Vertical Bar on Right: This option places the vertical label bar on the right of the grid, as opposed to the left.

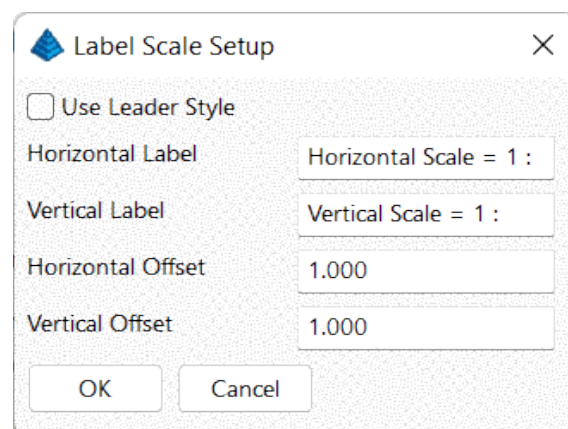
Draw Vertical Bar Separately: Draws a vertical bar with elevation labels separate from the profile grid. The **Offset** value controls how far away from the grid it will be drawn.

Label Stations: Disable this option if you do not want station labels to be placed along the grid.

Label Station Equations: Disable this option if you do not want station equation labels to be placed along the grid.

Label Base Elevation: Labels the low elevation on the sheet. The setup dialog allows for prefix and suffix for the text, size of text, layer and number of decimals to display.

Label Scale: Click on this option and you obtain a scale drawn at the lower left corner of the profile. Click the **Setup** button to establish the desired Scale labels and placement values.



Grid Title: Click on this option to place a Grid Title. Specify from the available items such as Horizontal Scale, Vertical Scale, Base Elevation and Profile Name. Configure the layer, text style, text scale and location offset.

The screenshot shows the 'Grid Title Setup' dialog box. It features a title bar with a blue diamond icon and the text 'Grid Title Setup' and a close button. The dialog contains several input fields: 'Layer' with 'PROTXT' and a 'Select' button; 'Text Style' with 'STANDARD' and a 'Select' button; 'Text Scale' with '0.100'; and 'Location Offset' with '0.100'. Below these are two list boxes: 'Available' containing 'Horizontal Scale', 'Vertical Scale', and 'Base Elevation'; and 'Used' containing 'Profile Name'. Between the lists are buttons for 'Add >>', '<< Remove', and 'Edit'. To the right of the 'Used' list are two green arrow buttons (up and down). At the bottom are 'OK', 'Cancel', and 'Help' buttons.

Station Type: Indicated the preferred style of station formatting.

Station Text Orientation: This option allows you to specify the orientation of the station text Horizontal or Vertical shown along the profile.

Station Text Position: Indicate if the station text should appear along the top of the profile grid or along the bottom of the profile grid.

Draw Grid Line at PC/PT Station: Places a separate grid line on PC and PT stations of curves from the reference centerline.

Use Partial Labels for Intermediate Stations: Enable this toggle if the "full station" content to the left of the "+" symbol should be omitted at intermediate stations. This is useful for large station values where intermediate station labels are desired. When enabled (assuming 100' station values), an intermediate station such as 1023+50 would simply be annotated as +50.

Increment Station Text from Beginning Station: Enable this option if you wish to have the station text labels be relative to the starting station value. For example, if the starting station value is 0+23.68 and the *Station Text* interval is 50, station labels of 0+73.68, 1+23.68, 1+73.68, *etc.*, would be generated.

Label Elevations: Disable this option if you do not want elevation labels to be placed along the grid.

Elev Text Vertical Justify: Indicate vertical justification for the elevation labels.

Draw Elevation Labels Only On Left Side: Enabling this option eliminates elevation labels on the right side of the profile.

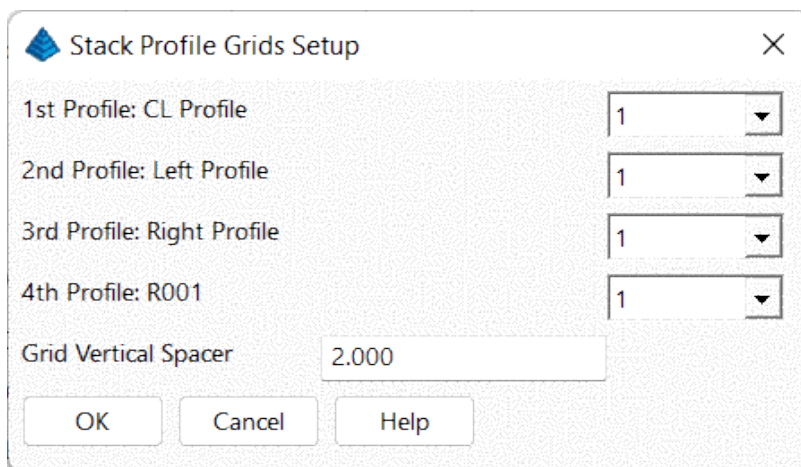
Draw Grid Line Under Elevation Labels: Enabling this option extends the grid lines underneath the elevation labels.

Offset Elevation Text: This option offsets the left-side vertical axis text using the specified Offset Scale.

Offset Station Text: This option offsets the horizontal axis Station text by the specified Offset Scale, allowing the insertion of elevation or other information above the stationing. It is often used in conjunction with the Label Horizontal Axis options.

Stack Profile Grids: This option allows you to stack profile grids for multiple profiles. In the **Setup** dialog, all profiles in the Multiple Profile file are listed and you can choose which one goes to the first grid, which one is second, and so on.

Grid Vertical Spacer: Indicate the amount of vertical space between successive grids.

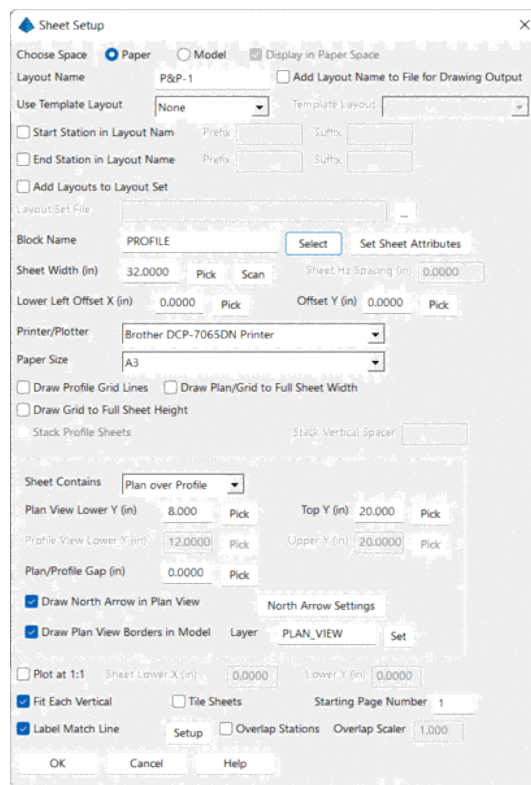


Draw Elevation Bar: Click on this option if you desire to have a vertical barscale displayed. It will run up and along the left-most vertical grid line of the profile, unless the Draw Vertical Bar on Right option is selected.

Skip Grid Range Dialog: This option automatically uses the calculated grid station and elevation ranges after picking OK from the main dialog instead of showing the dialog to set these ranges.

Draw Sheet/Paper Space

Plan Only, Profile Only, or Plan and Profile sheets can be created. The options within Sheet Setup become available when this toggle is checked on. Select Setup to access the Sheet Setup dialog.



Choose Space: Indicate whether sheets are to be drawn to **Paper** Space (also known as a Layout) or to **Model** Space. When the Model Space option is selected a toggle for how the sheets are created is enabled. They can be displayed in Paper Space or Model Space. If the Paper Space toggle is used the sheet will be drawn in Model Space but with Paper Space units.

Layout Name: Enter a name for the paper space "tabs" to be assigned to each layout for each sheet. The program will automatically divide the plan view and the profile view into sheet layouts, and if the length of the profile extends beyond a single sheet, then multiple layouts are created, with the layout name ID incremented by 1.

Note:

- The "Tile Sheets" toggle needs to be disabled for the auto-incrementing functionality.
- If either the *Start Station in Layout Name* or the *End Station in Layout Name* options are enabled, the Layout Name field will be disabled as the Layouts will get named automatically.

If you enter "ms" to go to model space within a Layout tab, you can pan to alter the plan view position. However, it is best to zoom in/out and edit within the Model tab. The Layout tabs appear at the bottom of the screen, along with the "Model space" tab to go back to standard plan view:

Add Layout Name to File for Drawing Output: When the option to Output To Separate Drawing is on, this option will create a separate DWG file for each layout by adding the layout name to the main DWG file name.

Use Template Layout: This option allows you to use a layout that exists in the current drawing as the template for layouts created by the command. This option only works for paper space sheet creation.

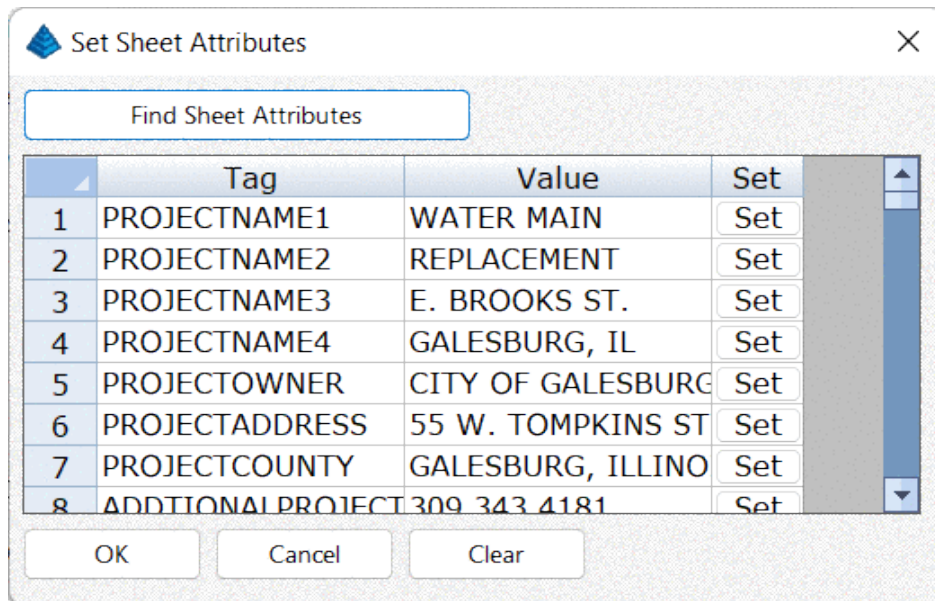
Start/End Station in Layout Name: These options allow you to include starting and ending station in the Layout Names.

Add Layouts to Layout Set: This option allows you to add the layouts created to an existing layout set that was previously generated using the Layout Set Manager. You will need to specify the name of the layout set.

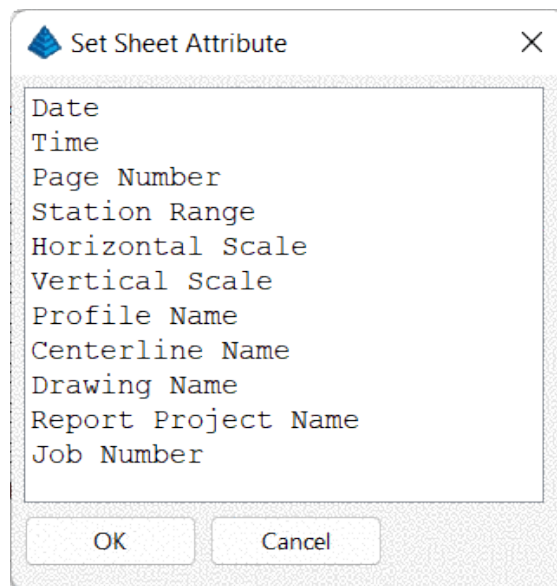
Block Name: This is the drawing name for the plan and profile sheet to be inserted. The Set button can be used to change the block name. Carlson provides a standard plan and profile border in the form of profile.dwg located in the working folder of %AppData%\Carlson Software\...\Sup\. You may wish to revise profile.dwg and add your company logo, and re-save it as profile1.dwg. Alternatively, you could add your own complete version of a Plan

and Profile sheet block/border. Be aware that the *Draw Right to Left* option in Draw Grid is superseded when Draw Sheet is enabled. Note that the Sheet mode will re-orient the centerline left to right, which may cause text (such as the stationing) to plot upside down, until you use the Flip Text command.

Set Sheet Attributes: This button allows you to specify the values used by any attribute definitions associated with the sheet Block Name. These can be entered manually in the Set Sheet Attributes dialog.



You can use the Set button to the right of any field to set that field to a preset value pulled from the drawing information.



Sheet Width: This is the profile width in the units specified on the sheet.

Lower Left Offset X/Y: Indicate the offset value(s) for the insertion point of the sheet in CAD units. This option allows user-defined Block Names to be properly positioned relative to the remainder of entities placed through the Draw Profile command.

Printer/Plotter: Indicate the desired output device.

Paper Size: Indicate the desired paper size.

Draw Profile Grid Lines: Enable this option if your Block Name does not contain profile grid lines and if you want profile grid lines to appear on the sheet.

Draw Plan/Grid to Full Sheet Width: Enable this option if you want to have what would otherwise be "partial" sheets (typically found at the end of a Plan & Profile Sheet run) occupy the full width of the sheet.

Draw Grid to Full Sheet Height: Enable this option if you want to have the profile grid drawn to the height of the sheet.

Stack Profile Sheets: When enabled (and when the Choose Space option is set to **Model**), this option permits the profiles to be stacked on top of one another.

Stack Vertical Spacer: Specify the amount of space that should be utilized when **Stack Profile Sheets** is enabled.

Sheet Contains: This drop list allows the selection of which type of sheet to generate.

Plan View Lower Y: This sets the lower position of the paper space window for the plan view.

Top Y: This sets the upper limit for the plan view window.

Profile View Lower Y: This sets the lower position of the paper space window for the profile view.

Upper Y: This sets the upper limit for the profile view window.

Plan/Profile Gap: Indicate the amount of vertical separation between the Plan portion of the sheet and the Profile portion of the sheet.

Draw North Arrow in Plan View: This draws a North Arrow in plan view. Click the **North Arrow Settings** button to establish the desired North arrow and placement information.

Draw Plan View Borders in Model Space: This draws the borders in Model Space which can be useful for orienting text and other labels to the orientation of the sheet. When this option is selected, use the Layer text box or Set button to choose the layer on which the borders will be drawn.

Plot at 1:1: With this clicked on, the sheet will be paper size, designed to be plotted at 1:1. A 30-inch profile sheet will measure 30 units, even though the centerline and profile may be 1500 feet in length. If the Scale 1:1 option is turned on, then you cannot check the distances of features using commands such as Bearing and Distance on the Inquiry menu, because the distances will be scaled down by a factor equal to the drawing scale (for example, at 1"=50', the reduction in scale factor is 1/50 or 0.02). You can set the absolute starting coordinate for the 1:1 scaled plot by setting the **Sheet Lower X** and **Sheet Lower Y** values. With this clicked off, the profile will draw full size, with a 1500-foot profile measuring 1500 feet.

Fit Each Vertical: With this option turned on, the program will size the profile grid to fit within the vertical space on the profile sheet. With this option off, the profile grid is sized to fit the elevation range of the profile.

Tile Sheets: If clicked on, only one Layout is created in paper space, and all sheets appear in this single Layout as tiles of individual sheets, much like the tiles mode of viewing files within Windows Explorer.

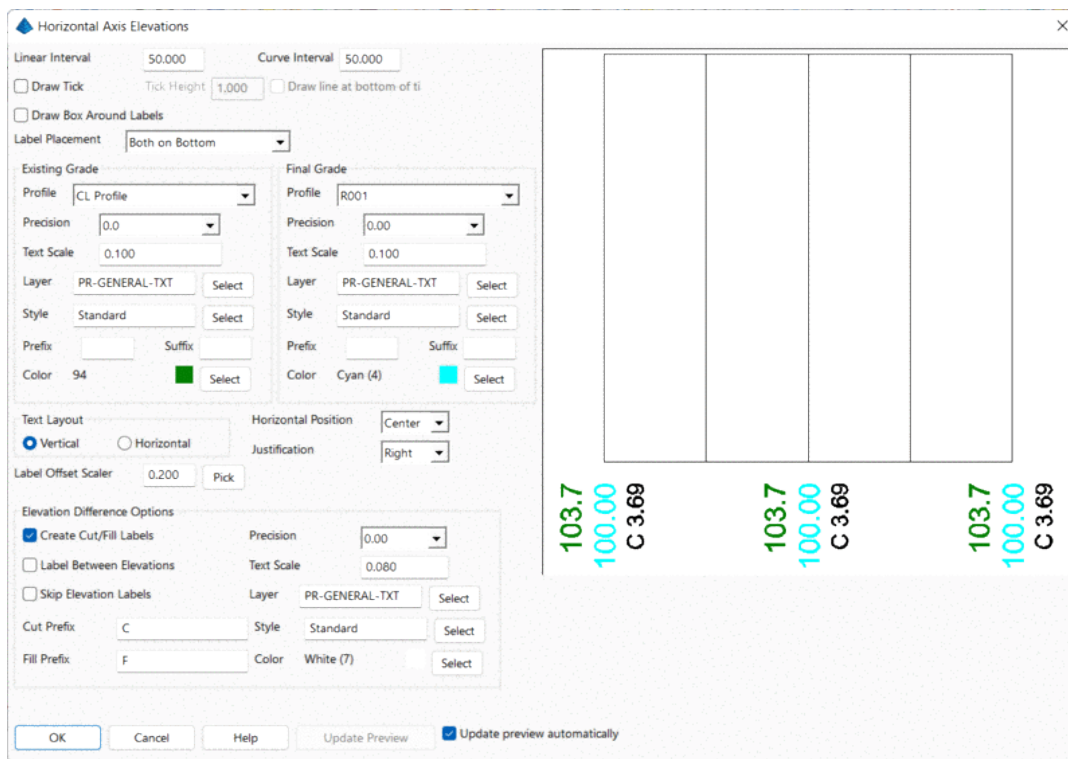
Starting Page Number: Specify the desired page number for the sheet(s) about to be cut. This is commonly used with the Set Sheet Attributes option.

Label Match Line: When clicked on and multiple sheets are plotted with plan view option on, a match line will plot in the plan view.

Overlap Stations: In multiple plan and profile sheet plotting, after the first sheet, all subsequent sheets will have the first 2 stations in common with the last 2 stations on the previous sheet, if the Overlap Station option is turned on. For example, if the last 2 stations are 3+10 and 3+20 on sheet 1, then sheet 2 will start with 3+10, then 3+20, with this option turned on. With this option turned off, if the first sheet ends with 3+20, then the second sheet would begin with 3+20.

Draw Horizontal Axis Elevations

This option creates elevation labels along the horizontal axis. Pick **Setup** to access the Horizontal Axis Elevations settings dialog. A preview of the labels will be shown to the right of the settings. If the preview does not match the settings, click the Update Preview button.



Linear and Curve Interval: Indicate how often the profile elevation labels should be placed along the horizontal axis of the sheet. The Curve Interval applies within vertical curves and the linear applies everywhere else.

Draw Tick and Tick Height: This option draws a line at the specified height at each station for the elevation labels.

Draw Box Below the Labels: When enabled, a box is drawn around the labels.

Place Existing Grade Profile Elevation on Top: When enabled, places the existing grades above the profile grid as opposed to below.

Existing/Final Grade: Indicate the appropriate profile, precision, text scale, layer, style, prefix, suffix and color for the text labels.

Text Layout: Indicate whether the text labels should be oriented vertically or horizontally.

Label Offset Scale: Indicate the distance from the horizontal axis for the labels. If the value is negative, the labels are placed above the horizontal axis.

Justification: Allows you to justify the elevation text about the left, center, or right side of the interval insertion.

Elevation Difference Options: If both existing grade and final grade are to be drawn, you may choose to also label the Cut/Fill depth values with the appropriate precision, text scale, layer, style, prefix, suffix and color for the text labels that separates the existing and final profiles at each station.

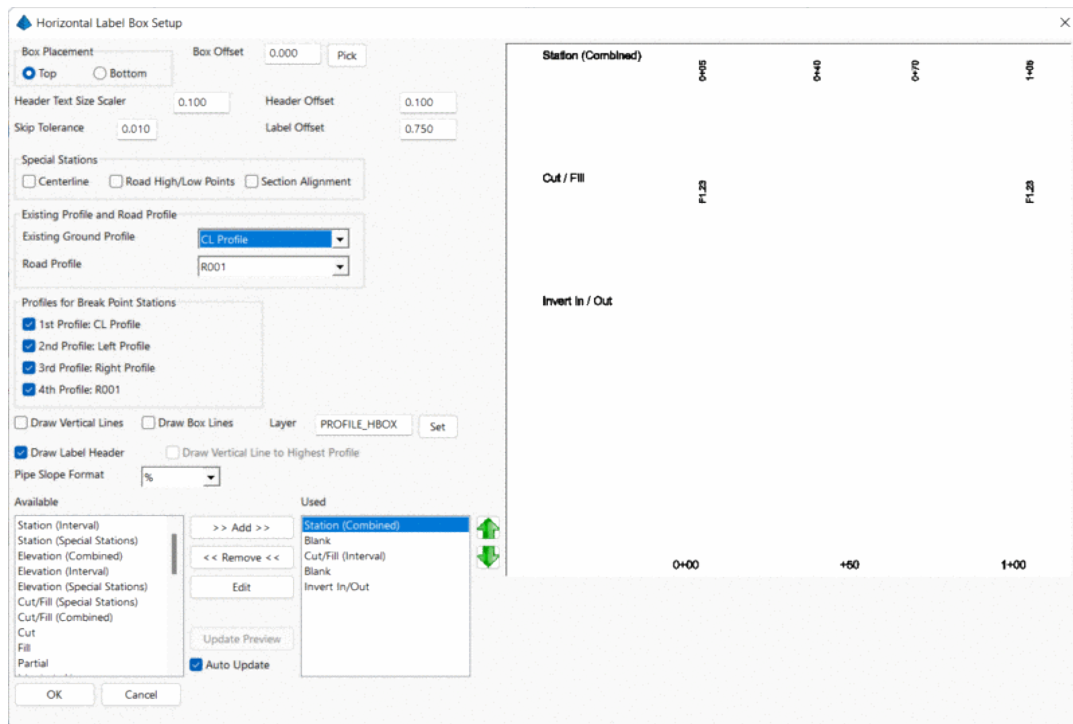
Label Between Elevations: When enabled (and for a **Text Layout** of Vertical), the label is centered on the various elevations and Cut/Fill Difference label.

Skip Elevation Labels: When enabled, only the Cut/Fill Difference label is generated.

Draw Horizontal Label Box

This option draws a boxed area either above or below the profile. It is best used in standard Draw Grid mode, with Draw Sheets clicked off. Pick Setup to access the Horizontal Label Box Setup dialog which has a list of available

fields to label. To label a field, highlight the field from the Available list and pick the Add button. Then use the Up/Down buttons to order the fields in the used list.



Box Placement: Set this to either Top or Bottom.

Box Offset: This controls how far to offset the box from the profile.

Header Text Size Scaler: Indicate a relative scale size for the header text values. This value is a scaler that is multiplied by the profile horizontal scale.

Header/Label Offset: This controls how far to offset the Header Text/Label Text box from the profile.

Skip Tolerance: Indicate a threshold in which labels should not be placed in an effort to reduce annotation clutter.

Special Stations: This controls whether special stations on the Centerline, high and low points on the profile and odd stations that exist in a Section Alignment are labelled.

Existing and Road Profile: Set which profile is existing and which is the finished road.

Profiles for Break Point Stations: Toggle on which profiles should be used for break point stations.

Draw Vertical Lines: This option draws lines from the data point on the profile to the label in the box.

Draw Box Lines: This option draws the row and column lines for the label box.

Layer: Set the layer for the elements to be drawn on.

Draw Label Header: Indicate whether or not the label header(s) should be drawn.

Draw Vertical Line to Highest Profile: This option is only available when the Draw Vertical Lines toggle is on. When on the vertical lines will be drawn to the highest profile.

Pipe Slope Format: Use this drop down to set the slope format between unit per unit or expressed as a percent.

Pipe Distance Format: Use this drop down to set whether the pipe length is shown as the horizontal distance or the slope distance.

Add and Remove: Use the >>Add>> button to add items to the list for annotation. Use the <<Remove<< to remove items from the list.

Edit: Use the Edit button to set parameters for the label in the box. The Label 2nd Row option creates another row for the field. You can also edit a blank row and specify the row height scaler.

Station (Combined) Label Box Fi... X

Label: Station (Combined)

Label 2nd Row:

Decimals: 0.00

Text Size Scaler: 0.0800

Draw Vertical Line: None

Row Height Scaler: 0.750

Layer: PR-GENERAL-TXT Select

Style: STANDARD Select

Color: ByLayer Select

Highlight Major Interval

Text Size Scaler: 0.08

Decimals: 0

Horizontal Major Interval Label

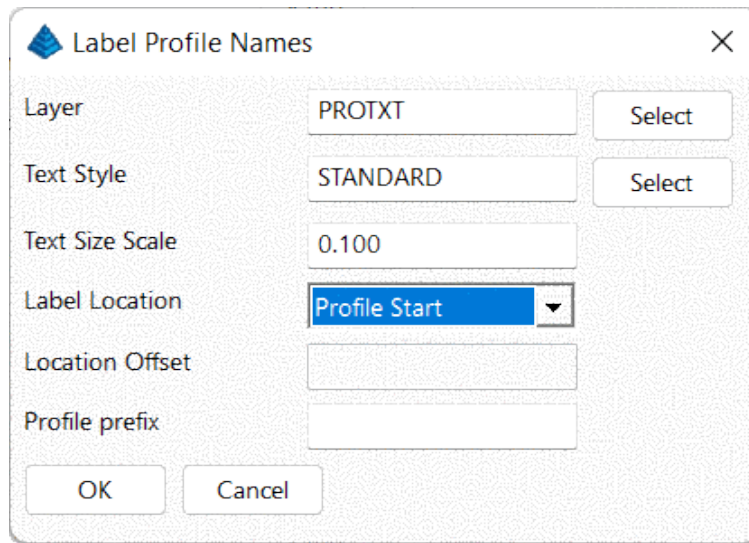
OK Cancel

Use the Update Preview to see the results of your changes before committing them to a sheet. An example of the resulting plot is shown here:

Station (Combined)	0+00	0+05	0+40	0+70
Elev.	105.00	102.00	108.00	
Cut / Fill	F1.23	F1.23	F1.23	

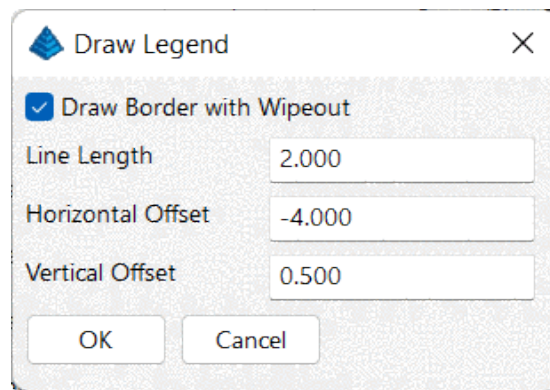
Label Profile Name

Use the Label Profile Names to place the profile names on the grid. Specify the Layer, Text Style, Text Size, Profile Prefix as well as the Label Location and the offset.



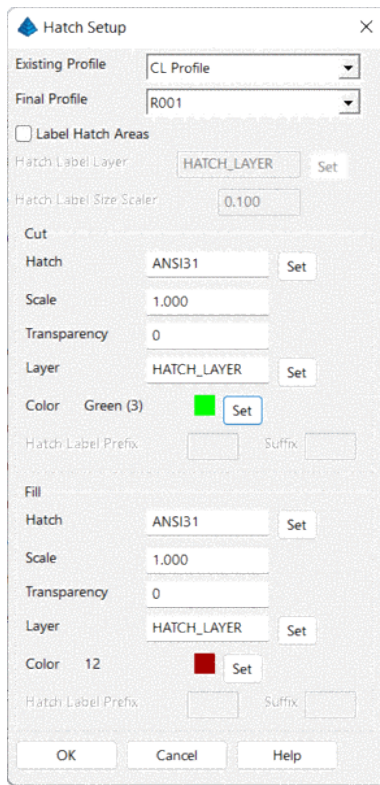
Draw Legend

Use the Draw Profile Legend setup to place a legend on the grid. Choose the line length, offset and if there is a wipeout placed under the legend.



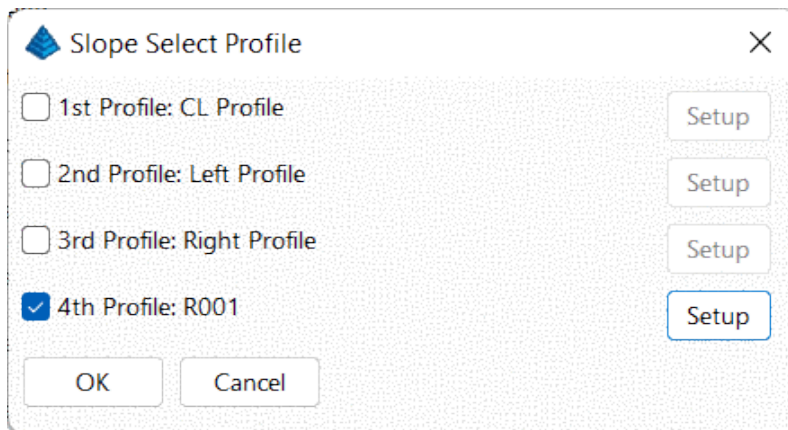
Hatch Cut / Fill

Use the Hatch Cut/Fill toggle to enable Hatch Setup. Choose the hatch pattern you would like to use for Cut and Fill as well as the scale, layer, transparency and color for each. There is an option to label the hatch areas with separate prefix/suffix labels for cut/fill.

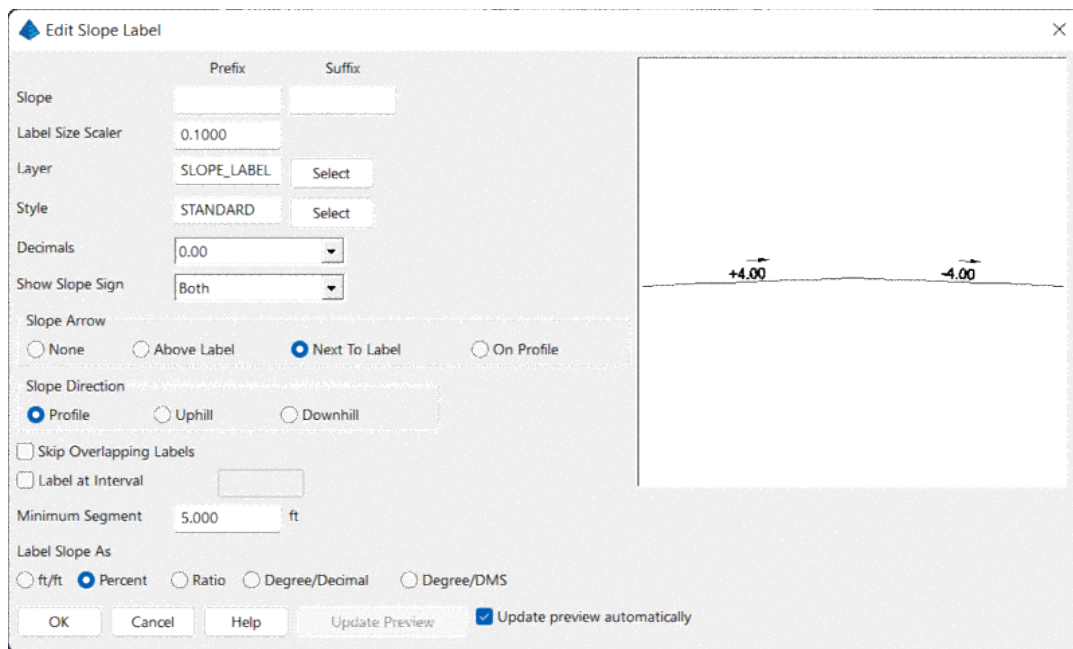


Draw Slope Labels

When enabled, this option allows you to detail additional slope information onto selected profiles.



Indicate the desired profile(s) whose slope annotation you'd like to control and click on the Setup button for expanded criteria.



Prefix/Suffix: Enter a prefix or suffix on the slopes as desired. Be aware that you may want to add spaces after the prefix or before the suffix.

Label Size Scaler: Set the label size scaler appropriate to your sheet setup.

Layer and Style: Set the layer and style for the text.

Decimals: Set the number of decimal places for the annotation.

Show Slope Sign: Indicate the preferred method for distinguishing positive *vs.* negative slopes.

Slope Arrow: If you would like a slope arrow to help understand the direction of slope, use one of the radio buttons for Above, Next to or On the profile.

Skip Overlapping Labels: If the labels would overlap and you do not want that to happen, use the Skip Overlapping Labels toggle.

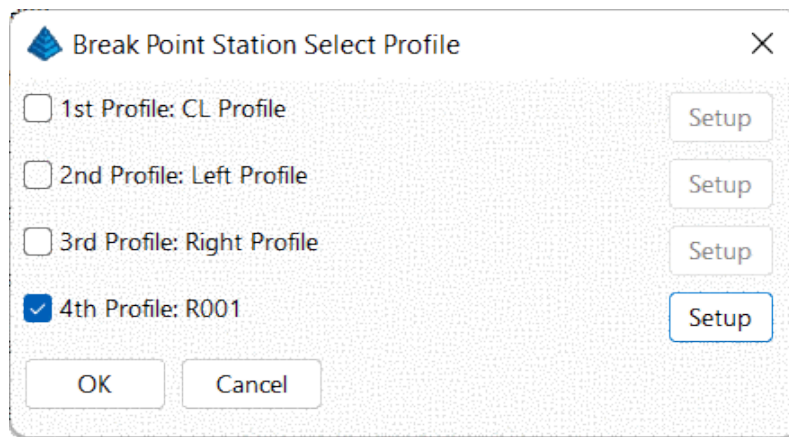
Label at Interval: Draws slope labels at the specified station interval.

Minimum Segment Size: You may not want to annotate shorter segments. If not, set a minimum segment size.

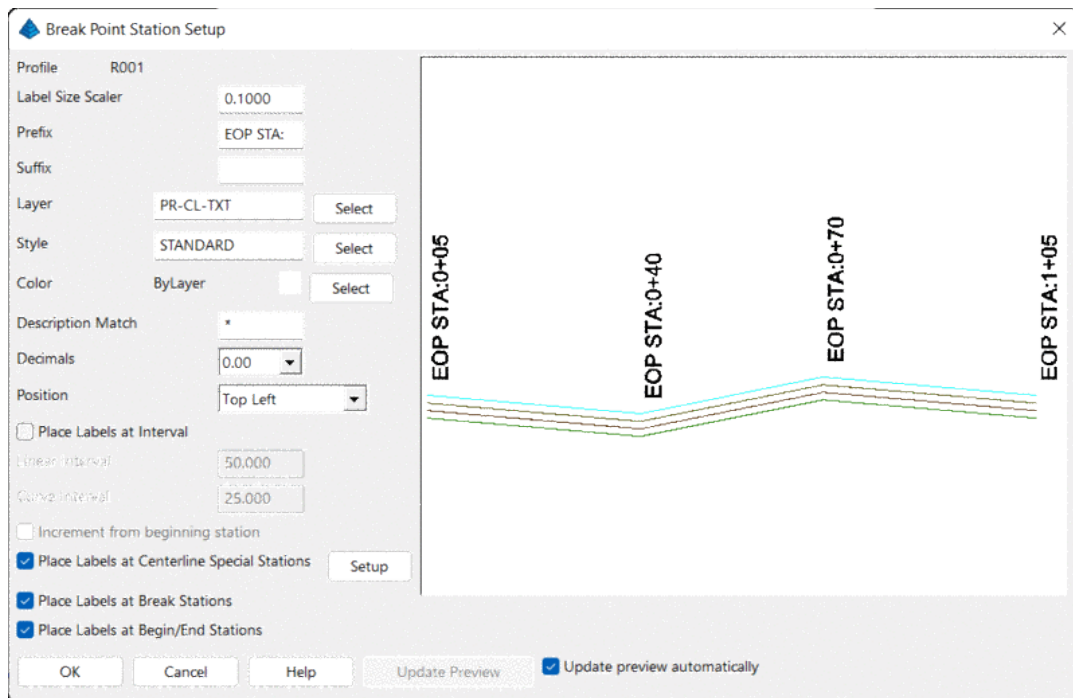
Label Slope As: You have the option of labeling the slope express as unit per unit, as a percent or as a ratio.

Update Preview: Use the update Preview button to see what your changes would look like before committing them to sheets.

Draw Break Point Station



When enabled, this option will label station values along the profile line above each break point in the profile. Pick **Setup** to access the **Break Point Station Setup** dialog.



Label Size Scaler: Set the label size scaler appropriate to your sheet setup.

Prefix / Suffix: Enter a prefix or suffix on the slopes as desired. Be aware that you may want to add spaces after the prefix or before the suffix.

Layer, Style and Color: Set the layer, style and color for the text.

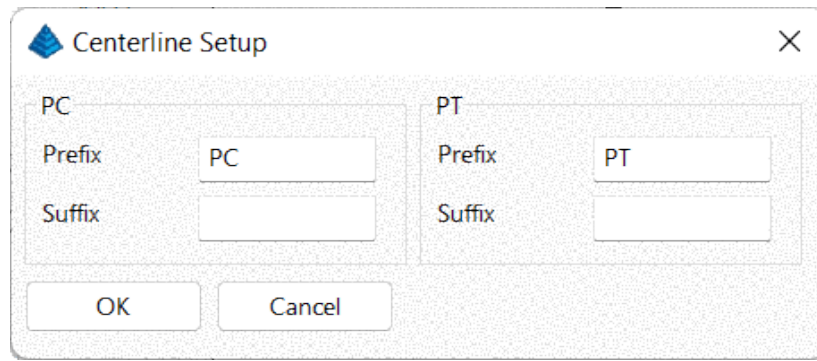
Description Match: Set the description you are looking to match.

Decimals: Set the number of decimal places for the annotation.

Position: Set the position for the text top or bottom and left, center or right.

Place Labels at Interval: Set up linear and curve intervals for the text.

Place Labels at Centerline Special Stations: Turn the toggle on and select the Setup button to change the settings.

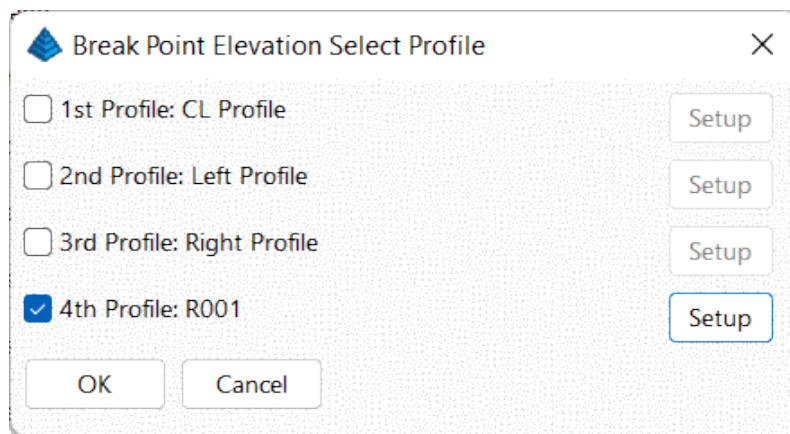


Place Labels at Break Stations: Toggle this on to place labels at break stations.

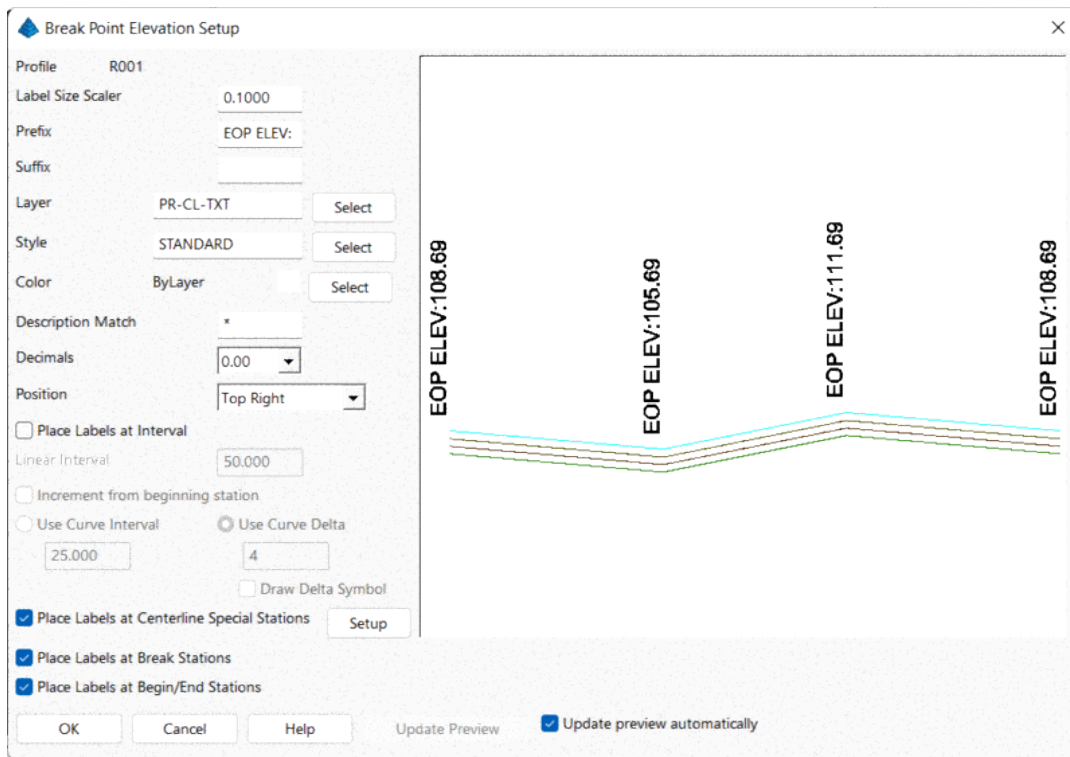
Place Labels at Begin/End Stations: Toggle this on to place labels at the beginning and ending stations.

Update Preview: Use the update preview button to check your changes before committing them to a drawing.

Draw Break Point Elevation



When enabled, this option will label elevation values along the profile line at each break point in the profile. Pick **Setup** to access the Break Point Elevation Setup dialog.



Label Size Scaler: Set the label size scaler appropriate to your sheet setup.

Prefix / Suffix: Enter a prefix or suffix on the slopes as desired. Be aware that you may want to add spaces after the prefix or before the suffix.

Layer, Style and Color: Set the layer, style and color for the text.

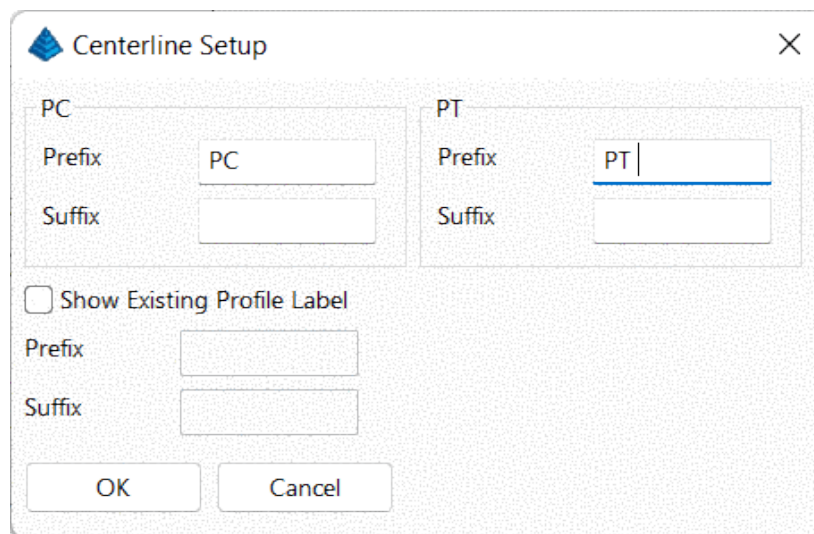
Description Match: Set the description you are looking to match.

Decimals: Set the number of decimal places for the annotation.

Position: Set the position for the text top or bottom and left, center or right.

Place Labels at Interval: Set up linear and curve intervals for the text.

Place Labels at Centerline Special Stations: Turn the toggle on and select the Setup button to change the settings.

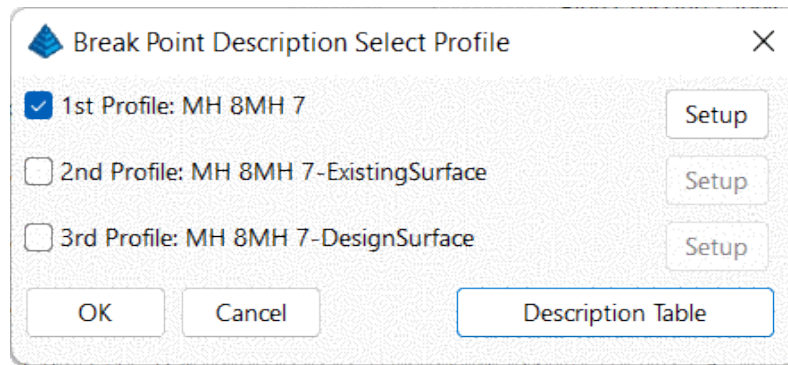


Place Labels at Break Stations: Toggle this on to place labels at break stations.

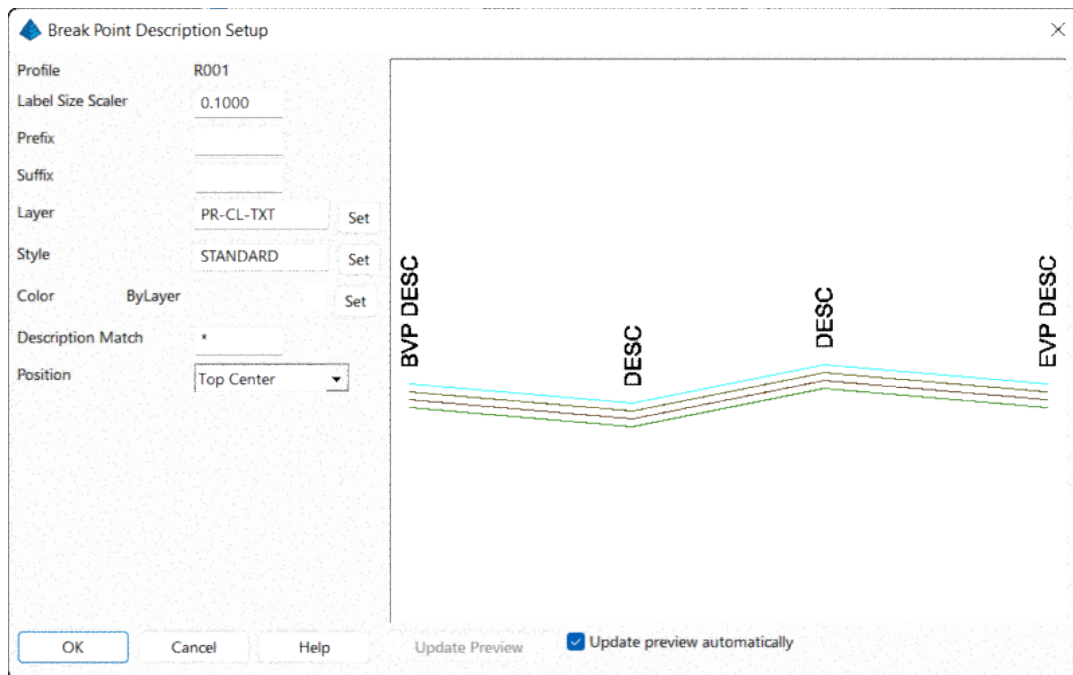
Place Labels at Begin/End Stations: Toggle this on to place labels at the beginning and ending stations.

Update Preview: Use the update preview button to check your changes before committing them to a drawing.

Draw Break Point Description



When enabled, this option will label descriptions along the profile line at each break point in the profile. Pick Setup to access the Break Point Description Setup dialog.



Label Size Scaler: Set the label size scaler appropriate to your sheet setup.

Prefix / Suffix: Enter a prefix or suffix on the slopes as desired. Be aware that you may want to add spaces after the prefix or before the suffix.

Layer, Style and Color: Set the layer, style and color for the text.

Description Match: Set the description you are looking to match.

Position: Set the position for the text top or bottom and left, center or right.

Update Preview: Use the update preview button to check your changes before committing them to a drawing.

The **Description Table** button on the Select Profile dialog for Draw Break Point Desc brings up the spreadsheet

dialog shown below. This table sets up translations of profile descriptions into labels and symbols to draw. For example, for a profile description of "WL", you can label the profile as "WATERLINE" and draw a symbol at this point. If you only want to change the label, then only fill out the label and leave the symbol blank.

	Description	Label	Symbol	
1	WL	WATERLINE	SPT7	Set
2	DK	DOCK	SPT8	Set
3				Set
4				Set
5				Set
6				Set
7				Set
8				Set
9				Set
10				Set
11				Set
12				Set
13				Set
14				Set
15				Set
16				Set
17				Set
18				Set
19				Set
20				Set

Buttons: Clear, Load, Save, OK, Cancel

Draw Break Point Elevation Difference

Break Point Elevation Difference Select Profile

1st Profile: CL Profile Setup

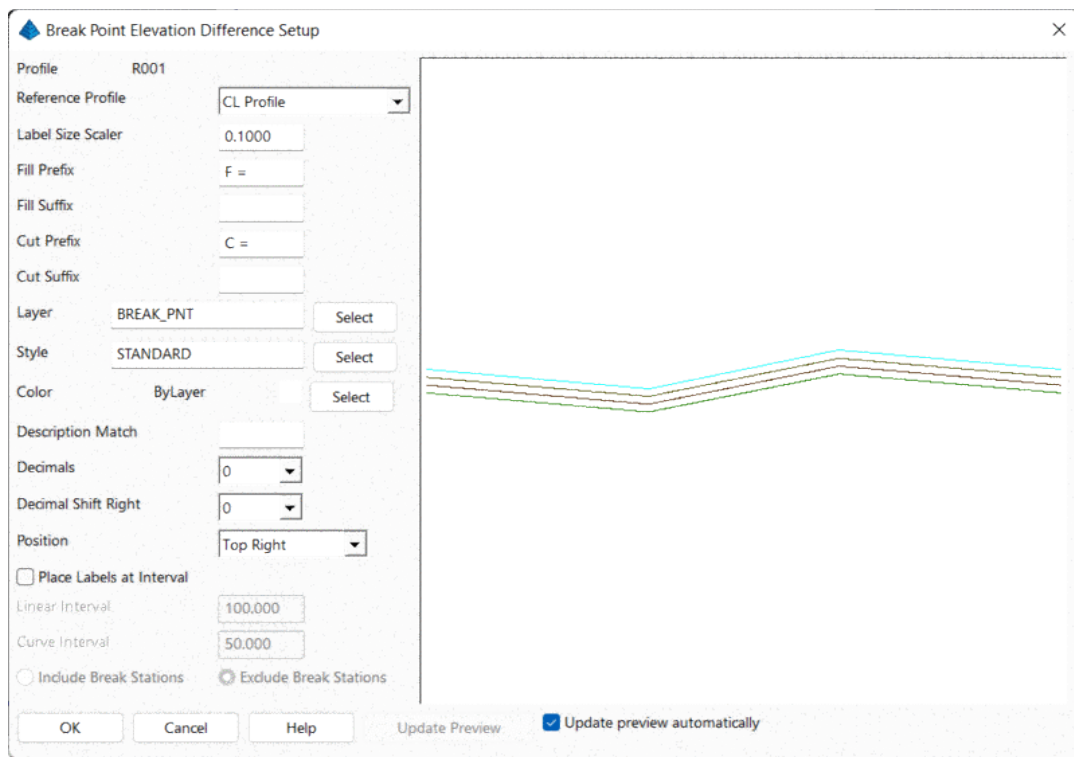
2nd Profile: Left Profile Setup

3rd Profile: Right Profile Setup

4th Profile: R001 Setup

OK Cancel

When enabled, this option will label elevation difference values along the profile line at each break point in the profile relative to a reference profile (*e.g.* existing grade). Pick **Setup** to access the Break Point Elevation Difference Setup dialog.



Reference Profile: Indicate the profile that should be used as the point of comparison for the break point locations.

Label Size Scaler: Set the label size scaler appropriate to your sheet setup.

Prefix/Suffix: Enter a prefix or suffix on the slopes as desired. Be aware that you may want to add spaces after the prefix or before the suffix.

Layer, Style and Color: Set the layer, style and color for the text.

Description Match: Set the description you are looking to match.

Decimals: Set the number of decimal places for the annotation.

Decimal Shift Right: Indicate the number of places to shift the decimal point to the right. For example, if a traditional elevation difference was calculated to be 1.234 and the Decimal Right Shift value is set to 1 (a factor of 10), the reported elevation difference would be shown as 12.34.

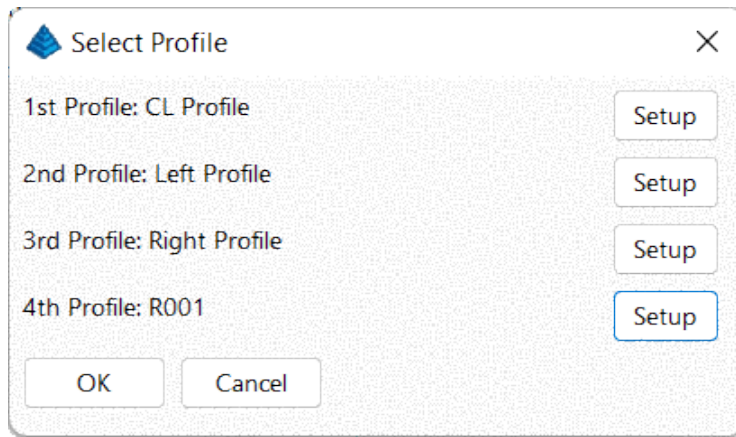
Position: Set the position for the text top or bottom and left, center or right.

Place Labels at Interval: Set up linear and curve intervals for the text.

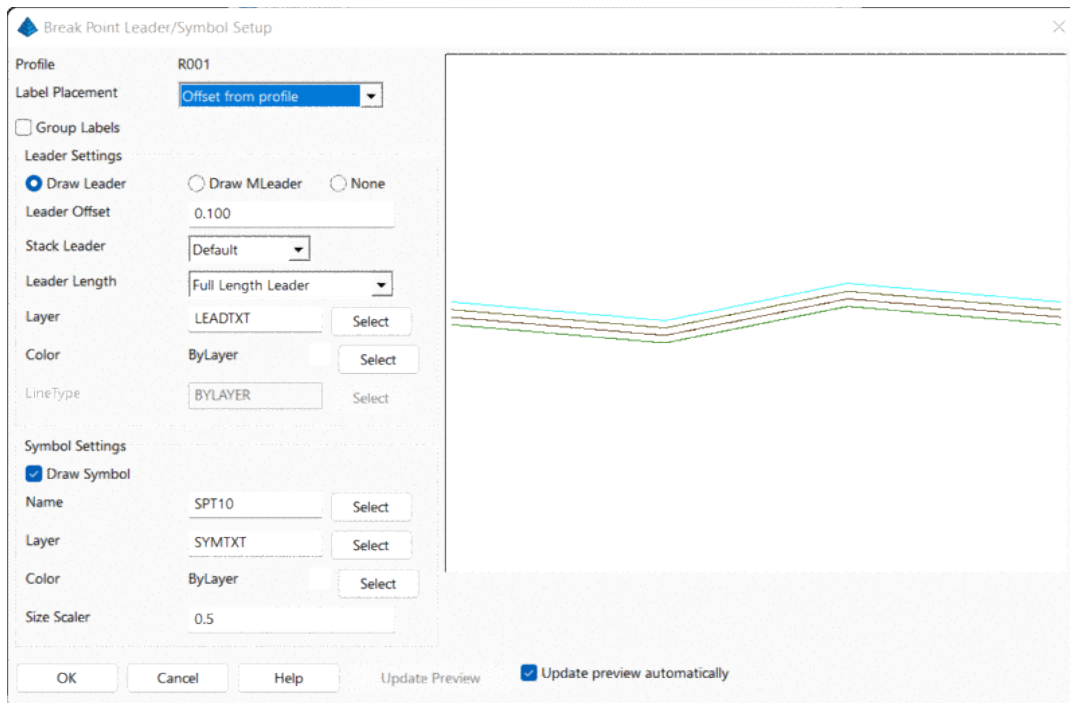
Include / Exclude Break Stations: Use the radio buttons to toggle between including or excluding break stations.

Update Preview: Use the update preview button to check your changes before committing them to a drawing.

Break Point Leader / Symbol Setup



Click this button to establish if it desirable to have a leader and/or break point symbol used in conjunction with the Draw Break Point Sta and/or Draw Break Point Elev options.



Label Placement: Use the label placement settings to control where this annotation will be placed.

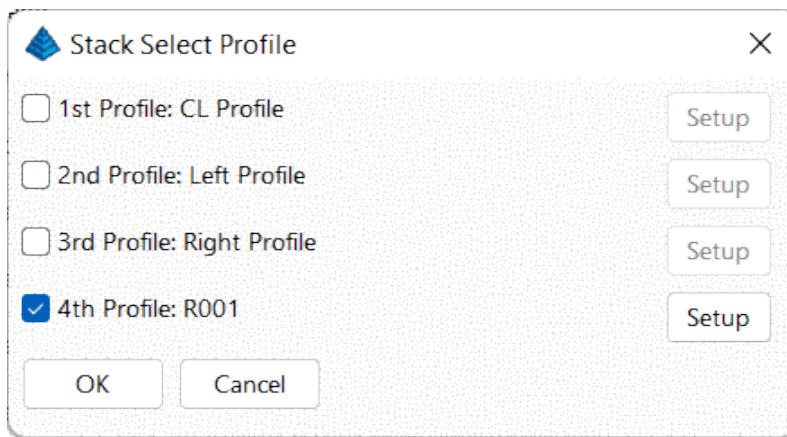
Offset Scaler: Set the label size scaler appropriate to your sheet setup.

Leader Settings: Set one of the two options for how the leader is drawn.

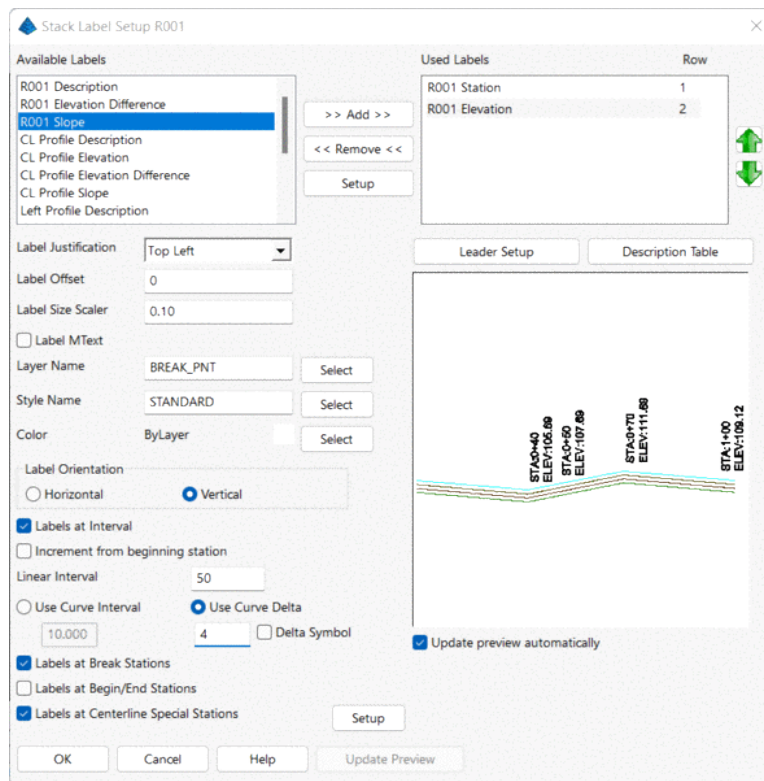
Symbol Settings: Set and select the symbol, layer, color and scaler for the symbol.

Update Preview: Use the update preview button to check your changes before committing them to a drawing.

Draw Stack Labels



When enabled, this option will label station values, elevation and elevation differences along the profile line at breaks, fixed intervals along curves/tangents, begin/end station, and Special Centerline stations along the profile. Pick Setup to access the **Stack Label Setup** dialog.



Add and Remove: Use the >>Add>> button to add items to the list of used labels to annotate. Use the <<Remove<< to remove items from the list of used labels.

Setup: Highlight a Used Label and then click the Setup button to set parameters for the label in the box. The Label on New Row option creates another row for the label.

Label Justification: Specify the justification location of the label (Top or bottom left, center or right).

Label Offset: Set the label offset from the profile line in paper units.

Label Size Scaler: Set the label size scaler appropriate to your sheet setup.

Label MText: Creates MTEXT entity when toggled ON.

Layer Name: Set the label layer.

Style Name: Set the label text style.

Color: Set the label color.

Label Orientation: Label may be oriented horizontally or vertically along the profile.

Labels at Interval: Toggle this ON to place the stacked labels at regular intervals. You can specify the starting station and identify different intervals along vertical curves.

Increment from beginning station: Set the station from which to begin the interval.

Linear Interval: Set the distance along the profile to place the stacked labels.

Use Curve Interval: Set the distance interval along curves to place the stacked labels. This is addition to the linear interval already specified.

Use Curve Delta: Specify the number of segments along a given curve portions to place the stacked labels. Short and long curves will have the same number of stacked labels using this option. You may also place the Delta symbol at those locations.

Labels at Break Stations: Places a stacked label at every change in grade along the profile.

Labels at Begin/End Stations: Places a stacked label at the beginning and ending of the profile.

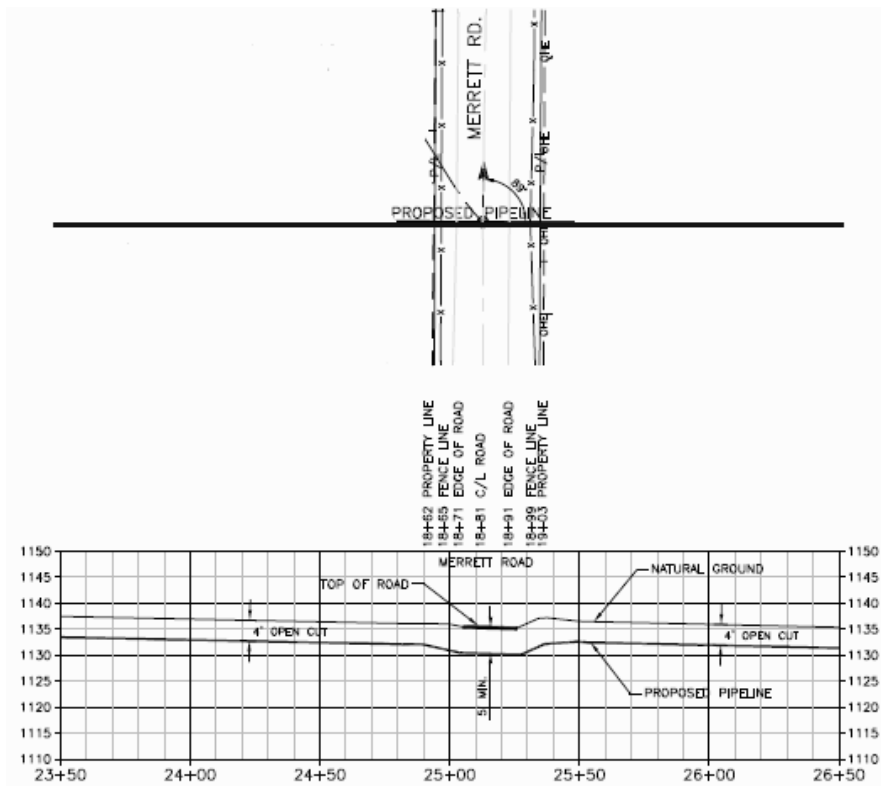
Label at Centerline Special Stations: Places a stacked label at the PC/PT from the referenced horizontal centerline file along the profile.

Leader Setup: Displays the Breakpoint Leader/Symbol setup.

Description Table: Displays the current Description Table.

Draw Line / Point Crossings

This option draws labels for linework that crosses the reference centerline. The reference centerline is set in the first Draw Profile dialog where the profiles to draw are selected. The **Setup** dialog has a list of layers. The program will find intersections between the reference centerline and linework on these specified layers. For each layer, there is a Description which is used for the label on the profile. Besides labeling these descriptions for the crossings, the program includes the station along the reference centerline at the crossing. In the options dialog, there are settings to control the layer, style, color, size, decimal places for the station label, label position and whether to draw a vertical line from the label to the profile.



Linework Crossings Label Setup

Linework Crossings **Point Crossings** CL Deflection General Settings

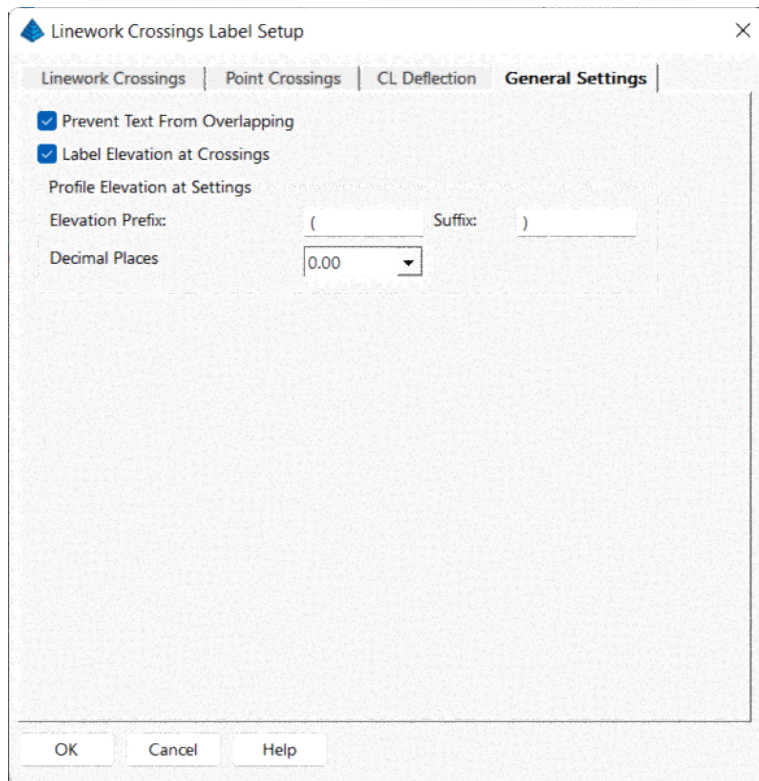
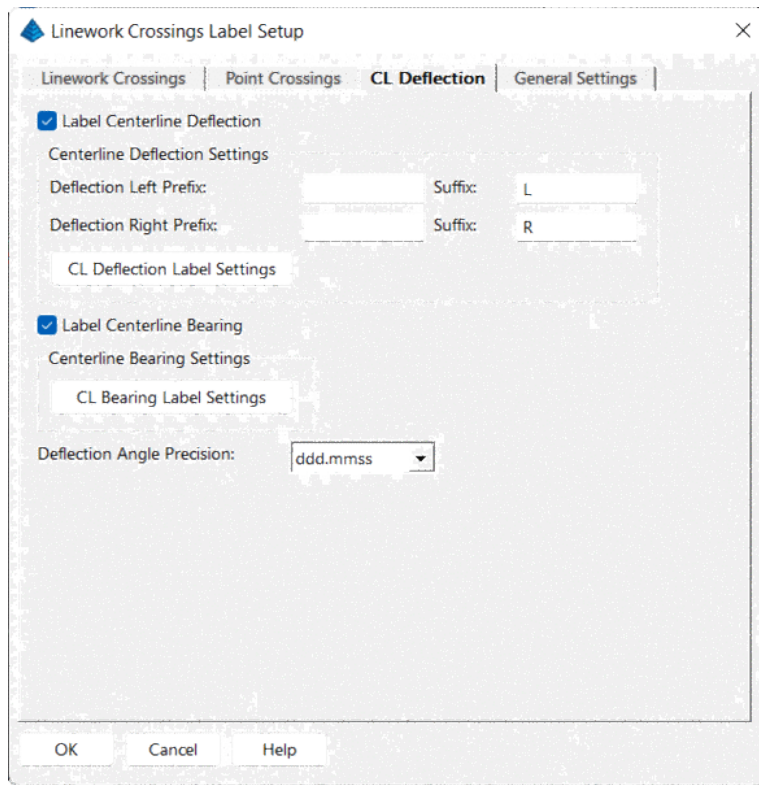
	Description	Label
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		

Remove Description Clear

Point Tolerance 1.000 (ft)

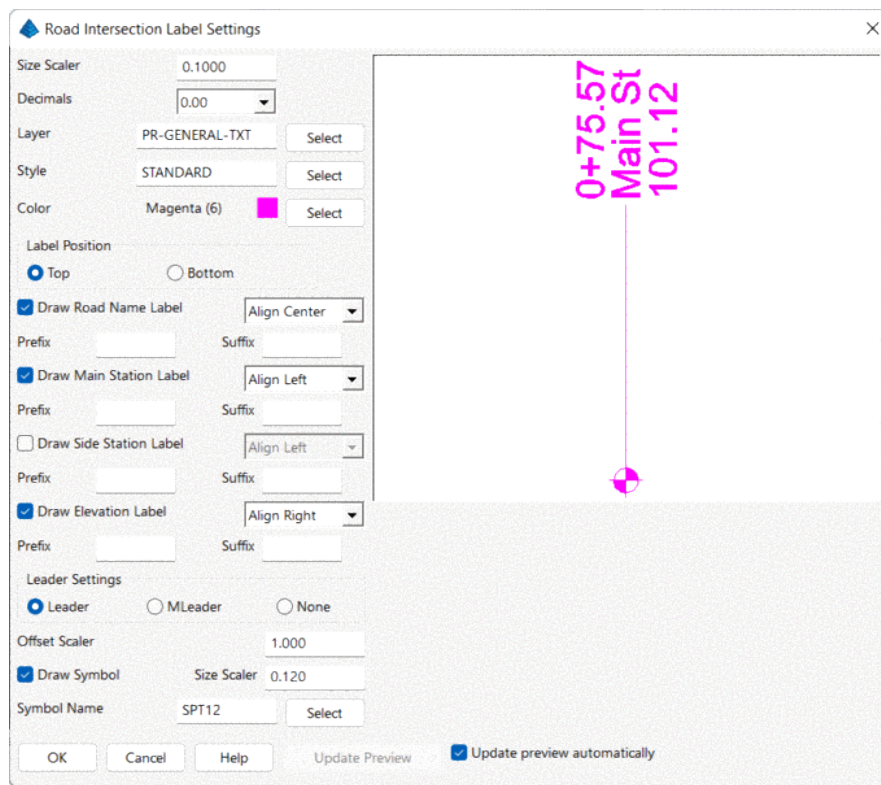
Point Label Settings

OK Cancel Help



Draw Road Intersections

When enabled, this option will label the location(s) of any road(s) from an identified Road Network that intersect the main road.



Size Scaler: Set the label size scaler appropriate to your sheet setup.

Decimals: Set the number of decimal places for the annotation.

Layer, Style and Color: Set the layer, style and color for the text.

Label Position: Use the label placement settings to control where this annotation will be placed.

Draw Road Name Label, Draw Main Station Label, Draw Side Station Label, Draw Elevation Label: Toggle on to draw the this annotation and set the location.

Leader Settings: Indicate the type of leader to be drawn.

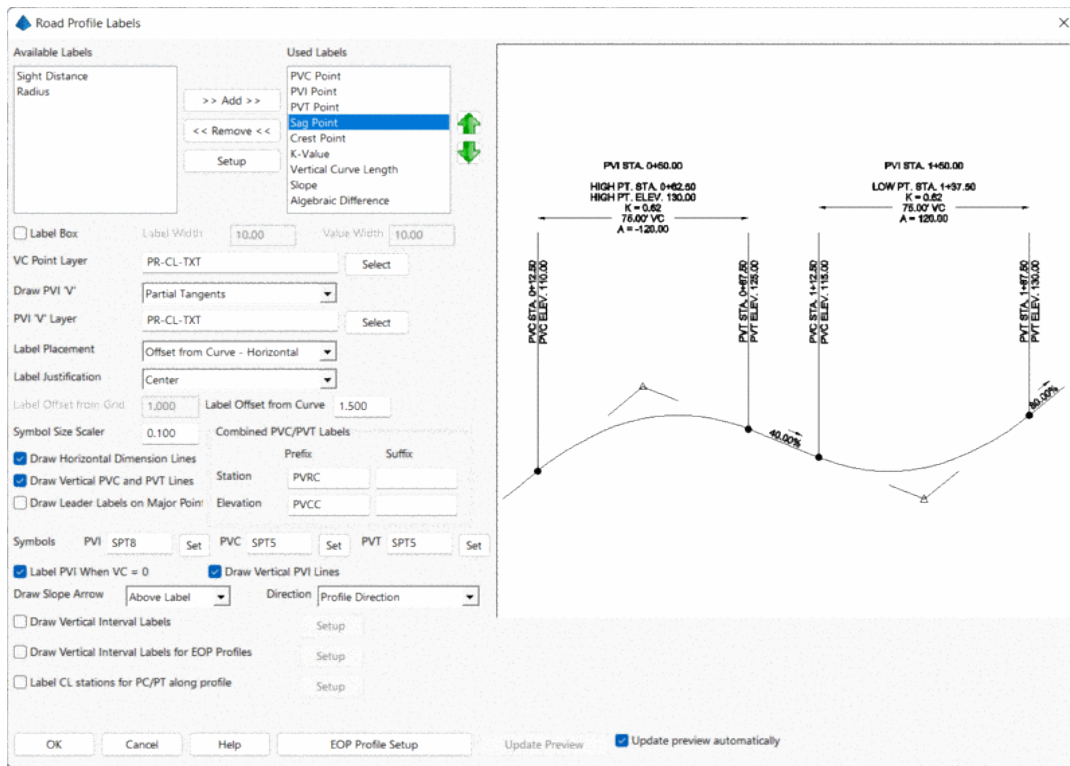
Offset Scaler: Indicate the plotted distance between the symbol and the text label(s) to establish the length of the leader.

Draw Symbol: Toggle this on to set a symbol and the size of the symbol.

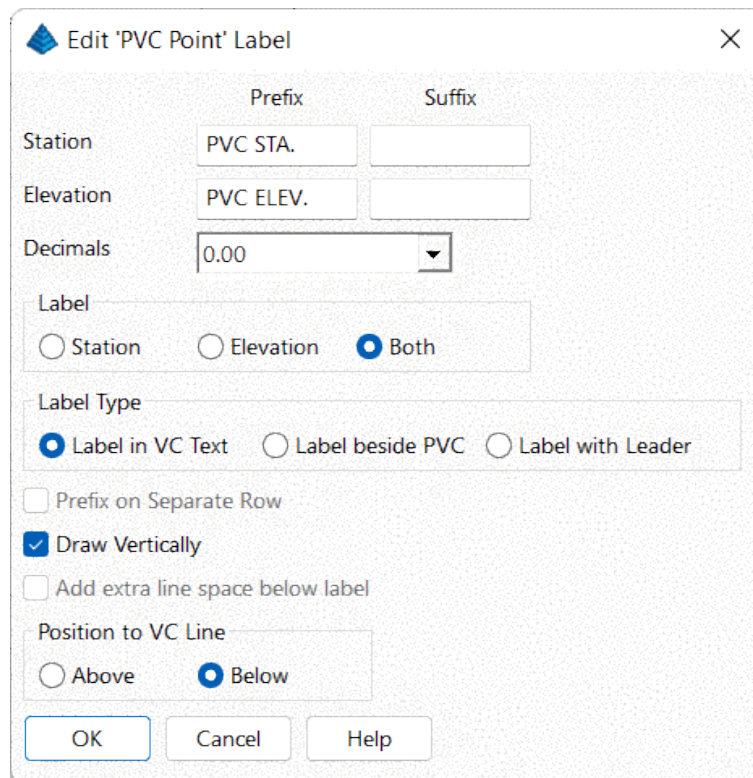
Update Preview: Use the update preview button to check your changes before committing them to a drawing.

Road Labels

This button opens Vertical/Circular Curve Settings dialog. From a wide variety of available labels, you are able to create your own label selections very conveniently.



Each label can be edited individually through the **Setup** button. You can specify the prefix, suffix, symbol style, decimal places, text orientation and position, *etc.*, in the Edit Label dialog. For the PVC/PVI/PVT labels, the **Label Beside** option applies when the Draw Horizontal Dimension Lines is off and it controls whether to place the labels along the profile at the station, or grouped with the rest of the road labels.



VC Point Layer sets the layer for the the point symbols on the vertical curve.

Draw PVI 'V': You can choose to draw either a full tangents style PVI 'V' point, or a partial tangents style, or nothing.

PVI "V" Layer specifies the layer that the tangent lines will be drawn on

Label Placement: This setting determines where to place the vertical curve labels. There are six options: Pick Single Row, Pick Individual Position, Auto Place Above Highest PVI Point, Specify Offset from Grid Top, Offset from Curve - Aligned, Offset from Curve - Horizontal.

Label Justification sets the text justification to Left or Center

Label Offset from Grid/Curve: Indicate the distance from the Grid or Curve when the **Label Placement** option is set to *Offset from Grid Top* or *Offset from Curve*, respectively.

Symbol Size Scaler sets the size for the symbols at the vertical curve points

Draw Horizontal Dimension Lines: This option draws horizontal lines connecting the PVC and PVT of all vertical curves.

Draw Vertical PVC & PVT Lines: This option draws vertical lines emanating from the PVC and PVT of all vertical curves.

Combined PVC/PVT Labels: sets the prefix and suffix for the stations and elevations of the vertical curve components

Symbols: for the PVI, PVC and PVT can all be selected from the symbol library.

Label PVI When VC=0: When vertical curve length is 0, no label is created unless you choose this option and then the PVI label would be shown.

Draw Slope Direction Arrow: Draws an arrow to indicate slope direction.

Arrow Direction: You can choose from Profile Direction, Uphill Slope Direction and Downhill Slope Direction.

Draw Vertical Interval Labels: This option labels the intervals of the vertical curve section. In its setup dialog, you can specify the intervals, distance from the vertical curve to put the labels, decimal places to display the interval stations and elevations, symbol settings and label settings.

Vertical Interval Labels [X]

Interval for Labels: 50.00

Offset Factor from VC: 1.00

Decimals: 0.00

Draw Symbol at Intervals

Symbol Name: SPT5 [Select]

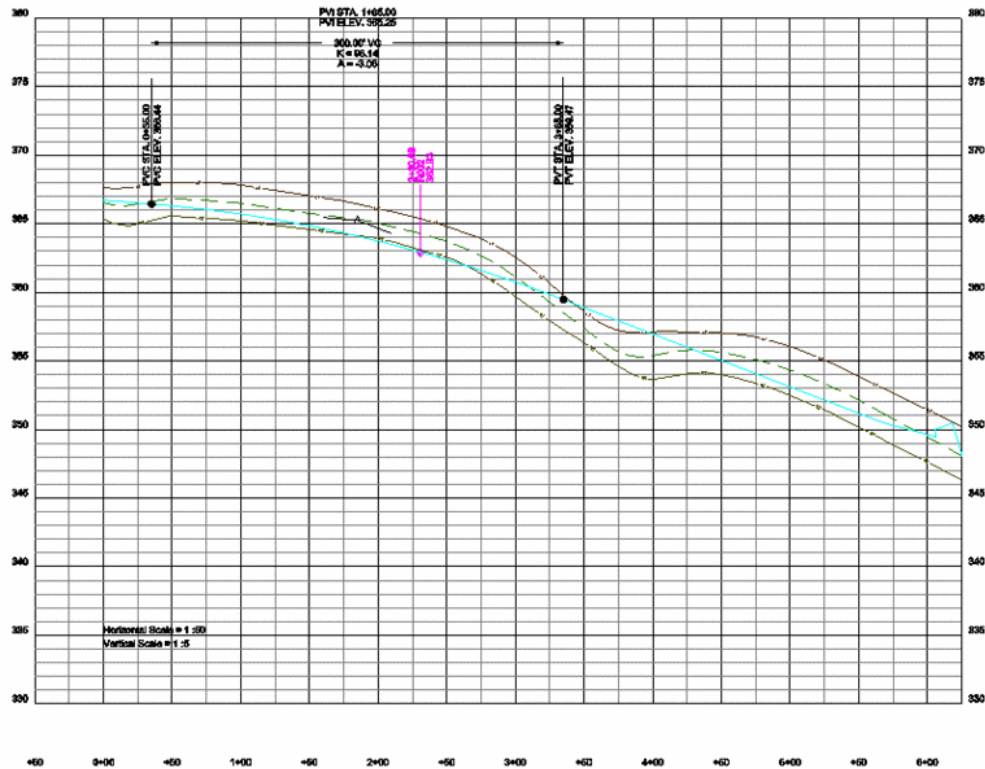
Label Options: Station Elevation Both

Station Prefix: STA Suffix: []

Elevation Prefix: ELEV Suffix: []

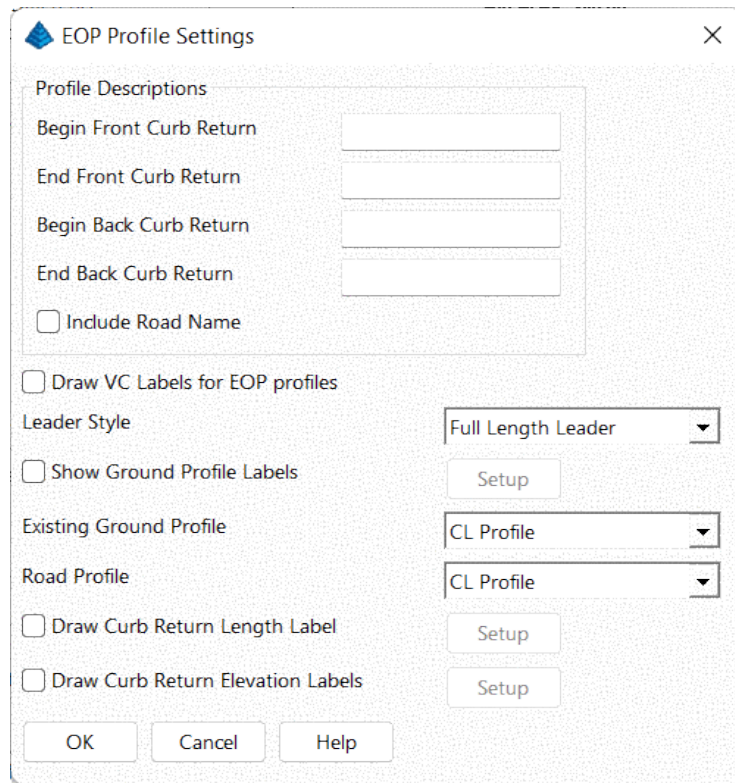
[OK] [Cancel] [Help]

Here is an example of a road profile.



EOP Profile Setup

This button allows you to establish the criteria for drawing and labeling Edge of Pavement (EOP) profiles:



Begin/End Front Curb Return: Enter a description for the front curb return.

Begin/End Back Curb Return: Enter a description for the back curb return.

Include Road Name: Enable this control if you'd like the road name included with the edge of pavement profile.

Draw VC Labels for EOP Profiles: When enabled, this option will label vertical curves found in edge of pavement profiles.

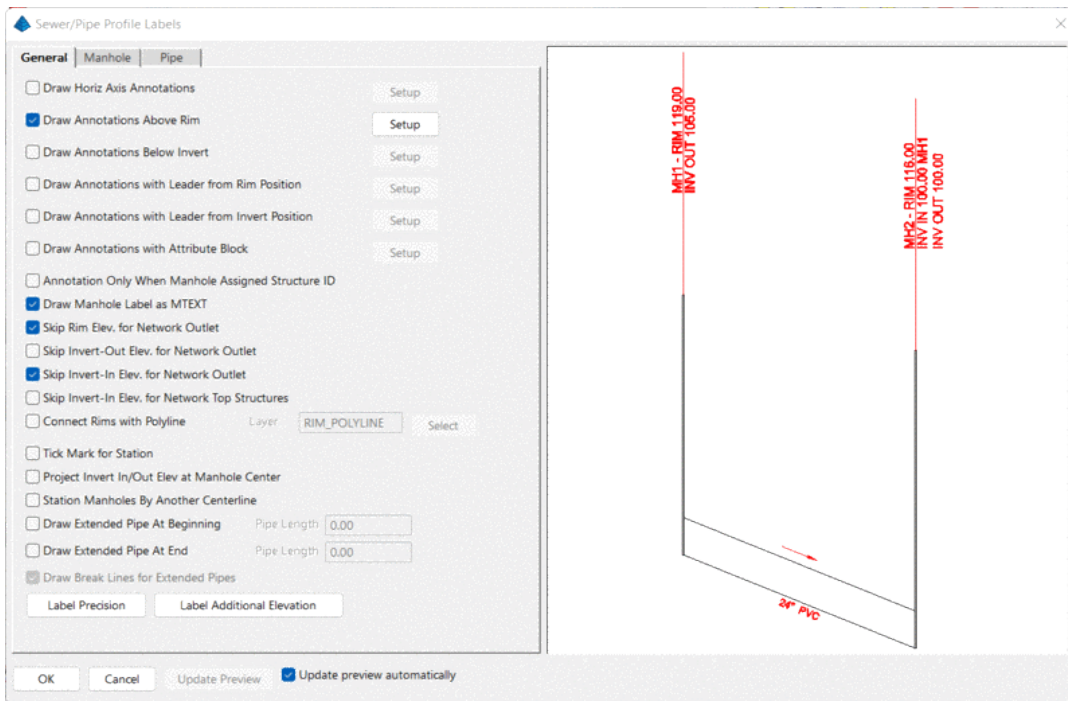
Draw Curb Return Length Label: When enabled, this option will label the length of curb returns. Use the **Setup** button to specify and control the display settings.

Draw Curb Return Elevation Labels: When enabled, this option will label the elevations of curb returns. Use the **Setup** button to specify and control the placement and display settings.

Sewer/Pipe Labels

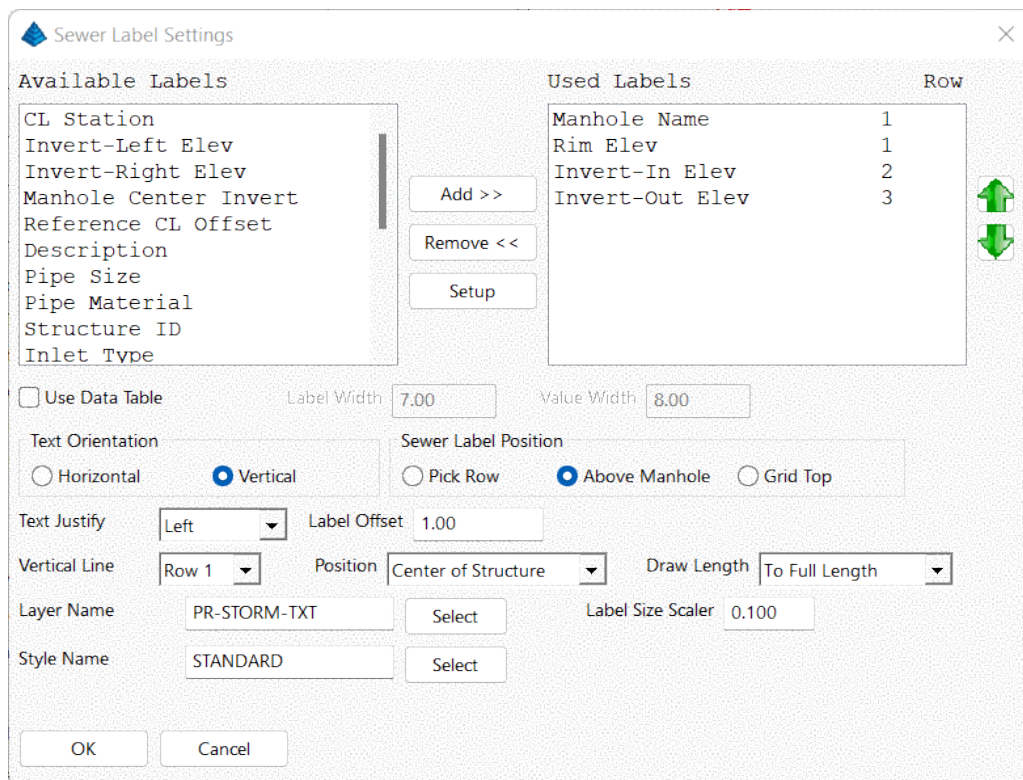
This button opens Draw Sewer/Pipe Options dialog.

General Tab



Each style has a **Setup** dialog to specify which labels are to be created and in what order. For labels with leaders, you can setup the leader styles

Example of Settings for Draw Annotations above Rim



Note:

In addition to the numerous annotation options found on the General Tab, many of the options will also contain an **Equation** option that permits user-defined equations using numbers and/or Rim, Invert Out or Step Up for even additional control. For example, to label rims with 2 elevations for 2 datums, one might specify an equation of **[RIM] + 1.0** (or similar).

Draw Horiz Axis Annotations: Labels structure or pipe profile along the horizontal axis.

Draw Annotations Above Rim: Creates structure or pipe profile labels above the rim of manholes.

Draw Annotations Below Invert: Creates structure or pipe profile labels below the rim of manholes.

Draw Annotations with Leader from Rim Position: Creates structure or pipe profile labels with a leader from manhole's rim position.

The screenshot shows the "Sewer Label Settings" dialog box. It is divided into two main sections: "Available Labels" and "Used Labels".

Available Labels: A list of labels that can be added to the project. The labels are: Invert-Left Elev, Invert-Right Elev, Manhole Center Invert, Reference CL Offset, Manhole Name, Description, Pipe Size, Pipe Material, Structure ID, and Inlet Type.

Used Labels: A table showing the labels currently in use, with a "Row" column indicating their order. The labels and their rows are: CL Station (1), Invert-In Elev (2), Invert-Out Elev (3), and Rim Elev (4). There are green up and down arrow buttons to the right of this table for reordering.

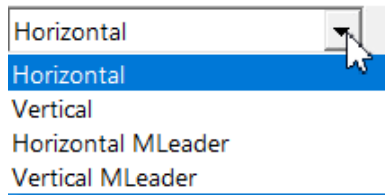
Buttons: Between the two lists are "Add >>", "Remove <<", and "Setup" buttons. At the bottom left are "OK" and "Cancel" buttons. At the bottom right are "Leader Setup", "Layer Name" (with a "Select" button), "Style Name" (with a "Select" button), "Label Size Scaler" (set to 0.100), and "Label Offset" (set to 0.500).

The screenshot shows the "Leader Settings" dialog box. It contains several settings for leader lines:

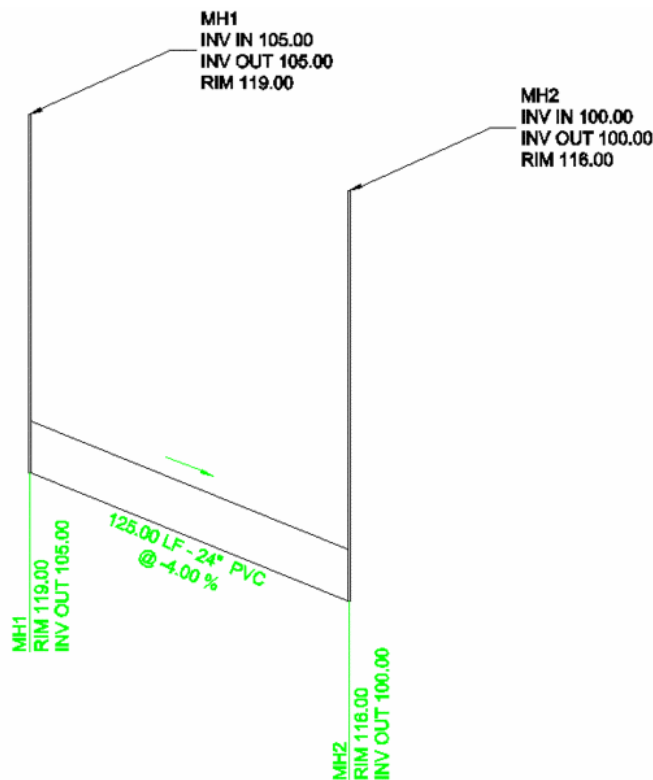
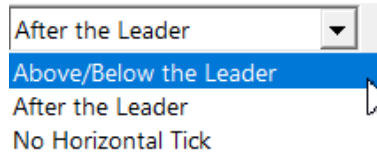
- Leader Style:** A dropdown menu set to "Horizontal".
- Text Placement:** A dropdown menu set to "After the Leader".
- Leader Line:** A dropdown menu set to "ALL Above Leader".
- Leader Offset:** A text input field set to "1.000".
- Linetype:** A text input field set to "BYLAYER" with a "Select" button next to it.
- Draw Arrow Head:** A checked checkbox.
- Arrow Size:** A text input field set to "0.120".

At the bottom are "OK", "Cancel", and "Help" buttons.

Leader Styles



Text Placement



Draw Annotations with Leader from Rim Position Example

Note: Using the the Network > Sewer Labels > Move Profile Pipe Label will allow for movements of labels and will hold the text in new location if redraw of the profile is required.

Draw Annotations with Leader from Invert Position: Creates structure or pipe profile labels with a leader from manhole's invert position.

Draw Annotations with Attribute Block: Inserts blocks with attributes for the structure or pipe labels.

Draw Manhole Label as MTEXT: When enabled, the manhole label will be placed as an MTEXT entity.

Skip Rim Elevation/Invert Out Elevation/Invert In Elevation for Outlet: When enabled, the cited component(s) will not be placed for the special Outlet structures.

Connect Rims with Polyline: When enabled, a polyline representing the average grade between each pair of rim elevations will be drawn to the specified layer.

Tick Mark for Station: Draws a tick mark at every station.

Project Invert In/Out Elev at Manhole Center: The Invert In/Out elevations are not the actual values, but are projected elevations to the manhole center.

Station Manholes by Another Centerline: This option will make new station for each manhole by referencing the profile to another centerline, for example a road centerline.

Draw Sump: When enabled, specify the height of the sump to be drawn into the sewer profile.

Draw Base: When enabled, specify the base height to be drawn into the sewer profile.

Draw Extended Pipe At Beginning/End: Draws the pipe beyond the beginning or the end by a specified length.

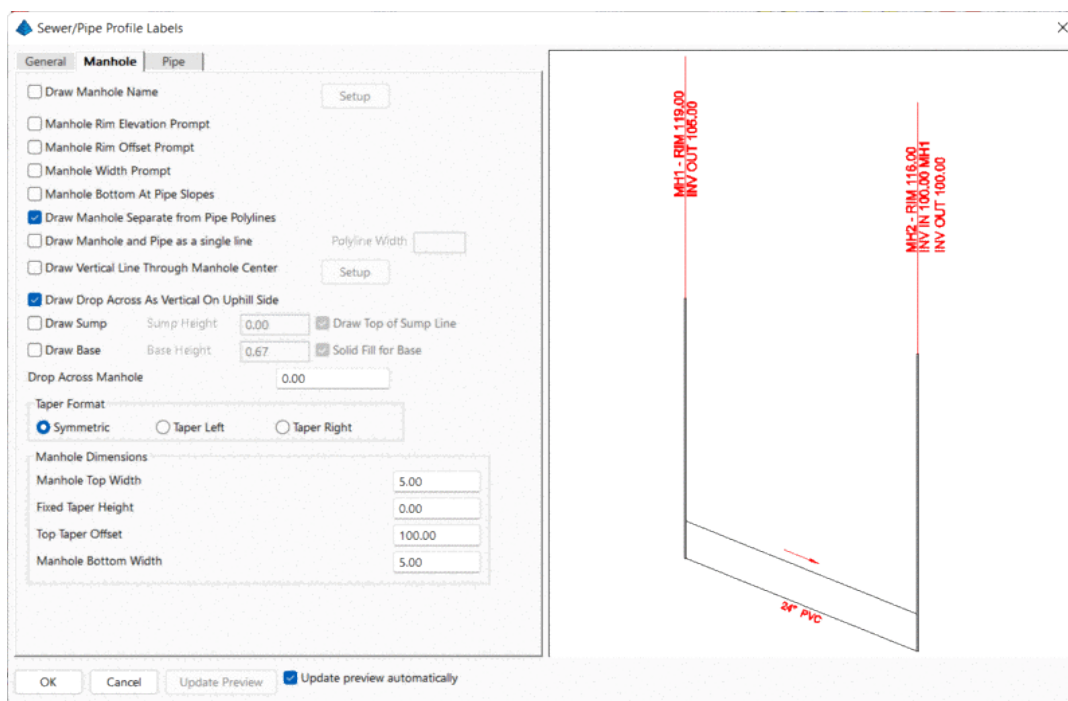
Draw Break Lines for Extended Pipes: When enabled (and requires either **Draw Extended Pipe At Beginning** or **Draw Extended Pipe At End** to be enabled), a "squiggle" is drawn at the end of the extended pipe to indicate that it is a partial pipe. Otherwise, the pipe end is left "open".

Label Precision: Click on the Label Precision button to set the amount of precision used for sewer station, elevation, length and slope labels.

Label Additional Elevation: This option may be utilized to apply a scale factor (*e.g.* Imperial Units to SI Metric Units) for plans that require alternate units to be cited.

Manhole Tab

On this tab, you are able to specify how to label the manhole name and how to draw the manholes.



Draw Manhole Name: Enable this option and select the desired geometric shape that shall circumscribe the manhole name. If selected, enter any desired prefix or suffix for the labels.

Manhole Name Setup

Symbol: Hexagon

Layer: MANHOLE_LABEL [Select]

Text Style: STANDARD [Select]

Text Size Scaler: 0.1000

Label Position: Above Manhole [Offset: 1.0000]

Prefix: [] Prefix on a new row

Suffix: [] Suffix on a new row

Draw Manhole Name as MText

OK Cancel Help

Draw Manhole Sides Down To Invert: Closes the manhole at pipes.

Manhole Rim Elevation Prompt: Ignores the manhole's rim elevation and prompts to enter new values.

Manhole Rim Offset Prompt: Prompts to enter the offset value and adds the offset to the manhole's rim elevation.

Manhole Width Prompt: Enable this option to prompt for the top width of the manhole.

Manhole Bottom at Pipe Slopes: Enable this option to prompt for the bottom width of the manhole.

Draw Manhole Separate from Pipe Polyline: Enable this option to draw the shape of the manhole as a separate polyline from that of the pipe.

Draw Vertical Line Through Manhole Center: Draws a vertical line through the manhole center from rim to bottom of profile grid.

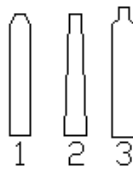
Draw Drop Across As Vertical On Uphill Side: If a step up is used, draws this as a vertical line on the higher side of the structure.

Draw Sump: When enabled, specify the height of the sump to be drawn into the sewer profile.

Draw Base: When enabled, specify the base height to be drawn into the sewer profile.

Drop Across Manhole: Adds a step up to the invert-in elevation.

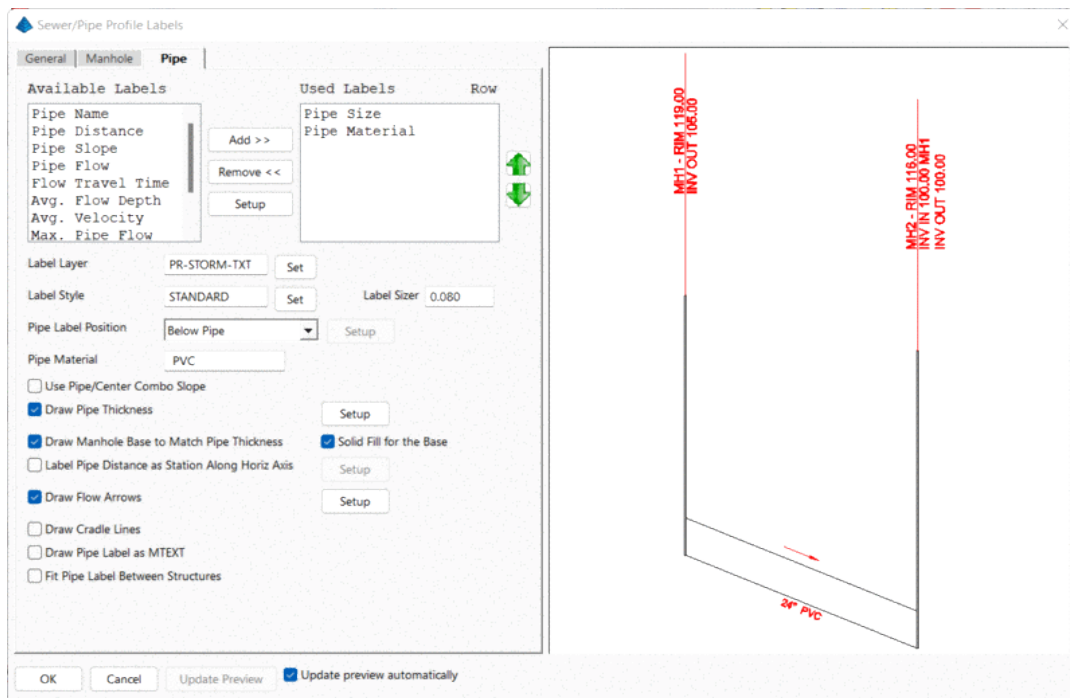
Taper Format, Manhole Dimensions: When drawing from a profile file created with the Design Sewer/Pipe Profile command, these parameters are used to define the manhole shape and dimension. When drawing from a profile created from Network in the Hydrology module with commands such as Export to Profiles, these Draw Profile settings are ignored and the dimensions come from the Network instead. The taper settings are used for transitioning between different manhole top and bottom widths. The Top Taper Offset sets the distance from the top of the manhole to the point that the taper will end. The Fixed Taper Height determines the overall length of the tapered section.



In this example image, all the manholes have Top Width of 2 and Bottom Width of 4. Manhole #1 has Top Taper Offset of 2 and Fixed Taper Height of 0. Manhole #2 has Top Taper Offset of 100 and Fixed Taper Height of 0. This large Top Taper Offset is greater than the manhole depth so that the taper runs the full length of the manhole. Manhole #3 has Top Taper Offset of 3 and Fixed Taper Height of 1.

Pipe Tab

Here you can choose to label pipe in a very flexible order. Each label has a setup function which specifies the label prefix and suffix, decimal places, row number and *etc.*



Label Layer: Set the layer for the labels.

Label Style: Set the text style and size of the text.

Pipe Distance Method: Label either the center to center distance or the actual distance of the pipe.

Pipe Slope Method: Choose from three common methods of calculating pipe slope.

Pipe Label Position: Choose from along the pipe, along the axis or inside the pipe.

Pipe Material: Indicate the type of material used for the pipe.

Draw Pipe Thickness: When selected, draws pipes in profile as double lines indicating the thickness of the pipe. This option also allows for cross hatching of the double lines.

Draw Manhole Base to Match Pipe Thickness: When selected, this option draws the base of the manhole to the same depth relative to the pipe thickness.

Label Pipe Distance as Station Along Horiz Axis: This option creates pipe distance labels as the station style along the horizontal axis. Click the **Setup** button to access the labeling method and style.

Label Pipe Distance as Station

Label Direction

Downstream to Upstream Upstream to Downstream

Layer Name: PIPE_LEN [Select]

Label Style: STANDARD [Select]

Label Offset: 0.10

Marker Height: 1.00

Label Decimals: 0.00

OK Cancel Help

Draw Flow Arrows: Indicate if arrows should be drawn illustrating the direction of flow.

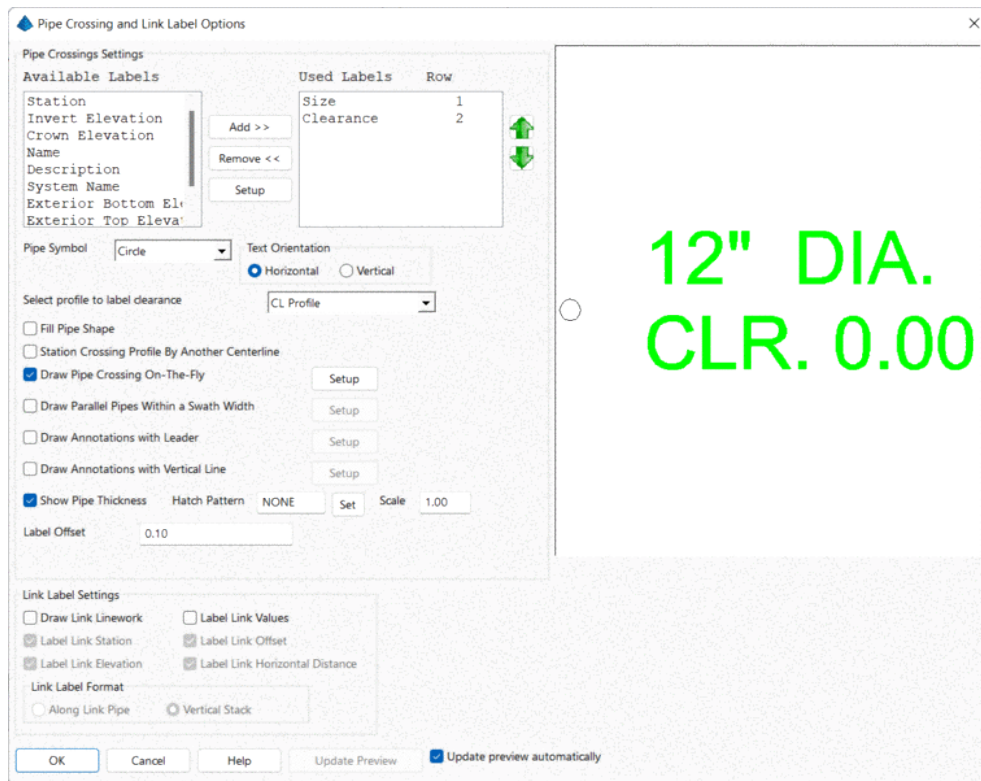
Draw Cradle Lines: If the sewer profile contains cradle data, this option would draw cradle lines above and below the pipe segments.

Draw Pipe Label as MTEXT: When enabled, text labels will be drawn as a *multiline text* (MTEXT) entity.

Fit Pipe Label Between Structures: When enabled, this option will ensure that pipe labels will fall within a structure-to-structure distance.

Pipe Crossing Labels

This button opens Pipe Crossing and Link Label Options dialog, which contains all the settings for drawing a pipe crossing type or profile, or the pipe crossings when pipes or sewer networks in the drawing are intercepted by a profile to be drawn.



(Setup) Label Prefix/Suffix: Indicate labels that should precede and/or follow the pipe information.

(Setup) Label Precision: Decimal places of the labels.

(Setup) Label Station/Elevation/Size/Name/System Name: Options to label the parameters or not.

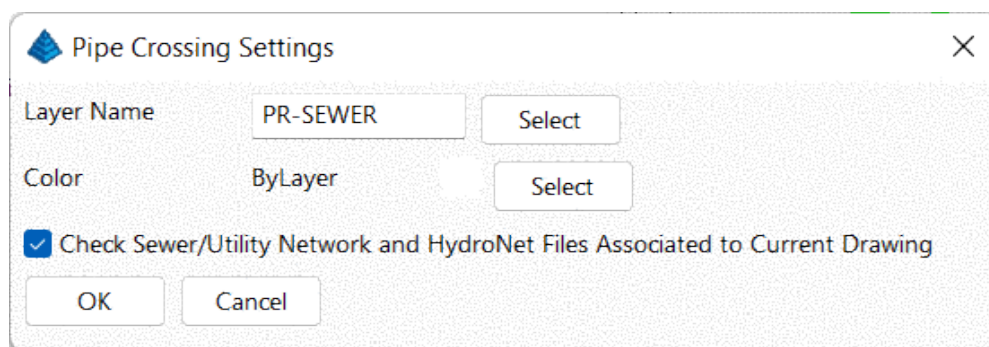
Pipe Symbol: Options to show pipe crossing in circle, square, or based on the pipe shape.

Text Rotation: Labels can be drawn either horizontally or vertically. This option becomes disabled when the *Draw Annotations with Leader* option is enabled.

Fill Pipe Shape: When enabled, the cross-section area of crossing pipes will be hatched for visual distinction.

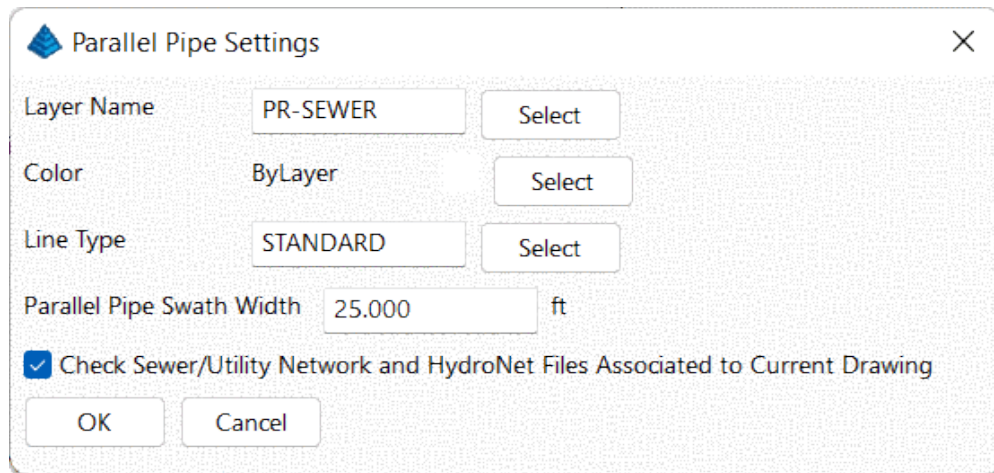
Station Crossing Profile By Another Centerline: This option will make new stations by referencing the profile to another centerline, for example a road centerline.

Draw Pipe Crossing On-The-Fly: When this option is chosen and there are pipes or sewer networks drawn in the drawing, the program will prompt to select a reference centerline that represents one of the profiles to be drawn to detect the pipe crossings. Any pipe crossings found would be drawn with other profiles.



Draw Parallel Pipes Within a Swath Width: When this option is chosen and there are pipes or sewer networks drawn in the drawing, the program will prompt to select a reference centerline that represents one of the profiles to

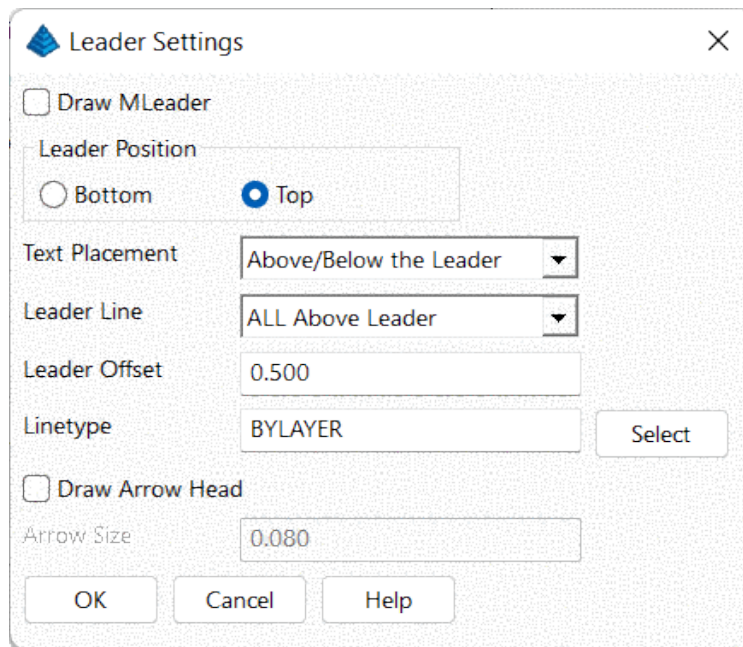
be drawn to detect if there's any pipe segments that are within a swath width along the profile. Any pipe segments found would be drawn with other profiles.



The screenshot shows the 'Parallel Pipe Settings' dialog box. It has a title bar with a blue diamond icon and a close button (X). The settings are as follows:

- Layer Name: PR-SEWER (with a 'Select' button)
- Color: ByLayer (with a 'Select' button)
- Line Type: STANDARD (with a 'Select' button)
- Parallel Pipe Swath Width: 25.000 ft
- Check Sewer/Utility Network and HydroNet Files Associated to Current Drawing
- Buttons: OK, Cancel

Draw Annotations with Leader: When enabled, this uses a leader in conjunction with pipe labels.



The screenshot shows the 'Leader Settings' dialog box. It has a title bar with a blue diamond icon and a close button (X). The settings are as follows:

- Draw MLeader
- Leader Position: Bottom, Top
- Text Placement: Above/Below the Leader (dropdown menu)
- Leader Line: ALL Above Leader (dropdown menu)
- Leader Offset: 0.500
- Linetype: BYLAYER (with a 'Select' button)
- Draw Arrow Head
- Arrow Size: 0.080
- Buttons: OK, Cancel, Help

Draw Annotations with Vertical Line: When enabled, this uses a vertical line and orientation to indicate the location of the pipe crossing being labeled.

Vertical Line Settings [X]

Layer Name: XPIPE_PROFILE [Select]

Color: ByLayer [Select]

Line Type: STANDARD [Select]

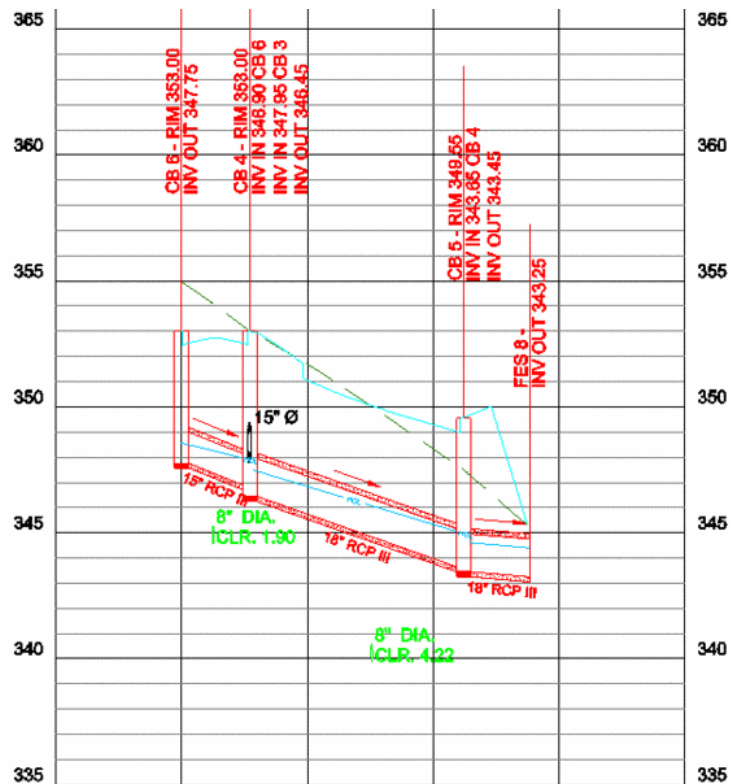
Line Position: Row 1 [v]

Line Location: Grid Bottom [v]

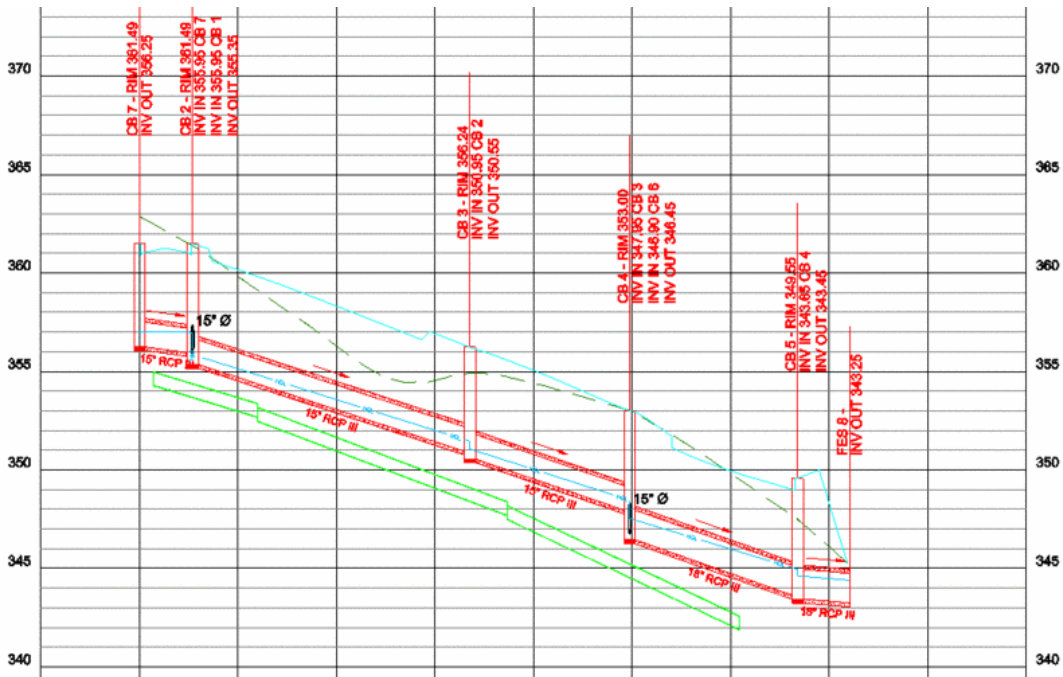
[OK] [Cancel]

Show Pipe Thickness: When enabled, this draws the pipes in profile using double lines to indicate the thickness of the pipe. The area between the lines can be cross-hatched.

Link Label Settings: Settings to determine how to draw link labels.



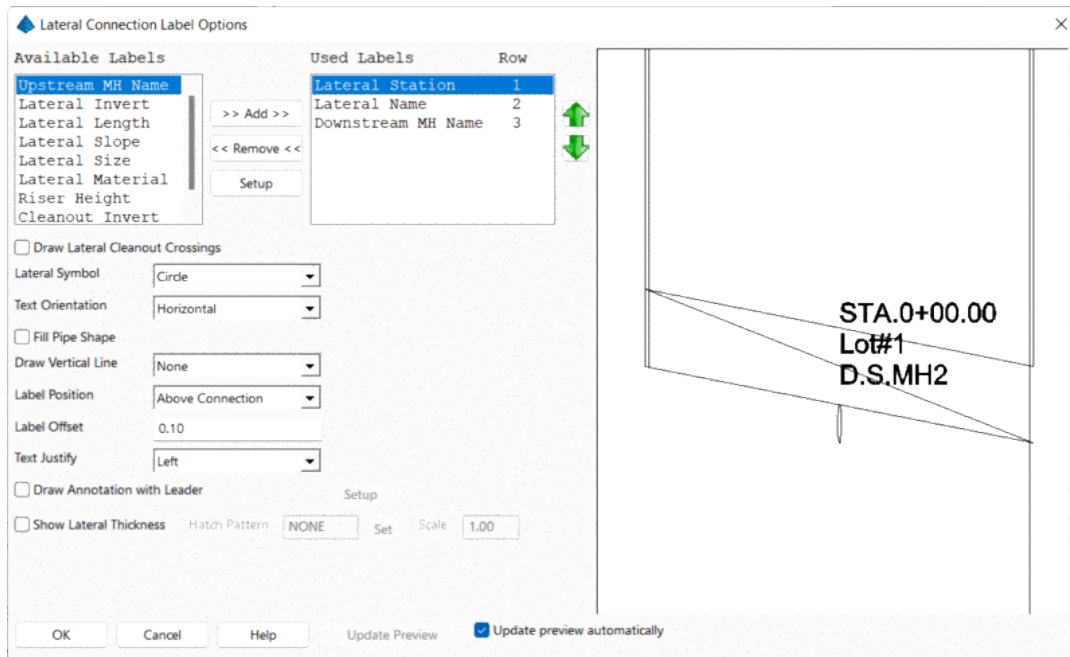
An Example of Pipe Crossings On-The-Fly (Showned in Green)



An Example of Parallel Pipes Within a Swath Width (Shown in Green)

Lateral Connection Labels

These settings apply for profiles create from a Sewer Network from the Hydrology module that contains lateral structures. There are several lateral data fields available for labeling. Use the Add/Remove buttons to make the list of fields to label. Use the Setup button to set the prefix and suffix for each field, and control whether the field is labeled on a separate row. There are settings to choose the symbol on the pipe at the lateral station, the text orientation, whether to draw a vertical line at the lateral station, set the label position and offset, set the text justification and whether to draw a leader from the pipe to the label.



Lateral Symbol: Set this to circle, square or pipe shape as appropriate.

Text Orientation: Use this to control whether the text is oriented horizontally or vertically.

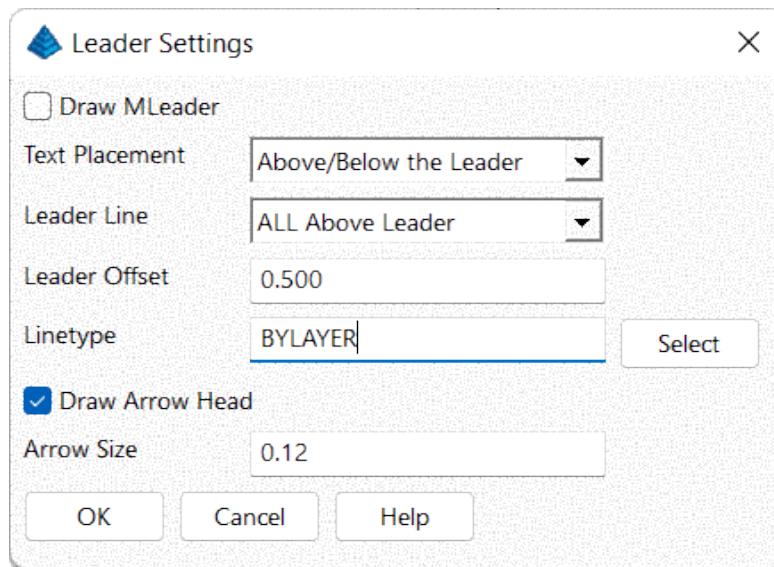
Draw Vertical Line: Set the vertical line to none or a number of rows.

Label Position: Use this to set the label position relative to the Right-of-Way, Connection or Mainline.

Label Offset: Set the offset of the label relative to its reference point.

Text Justify: Use this to justify the text either center, left or right.

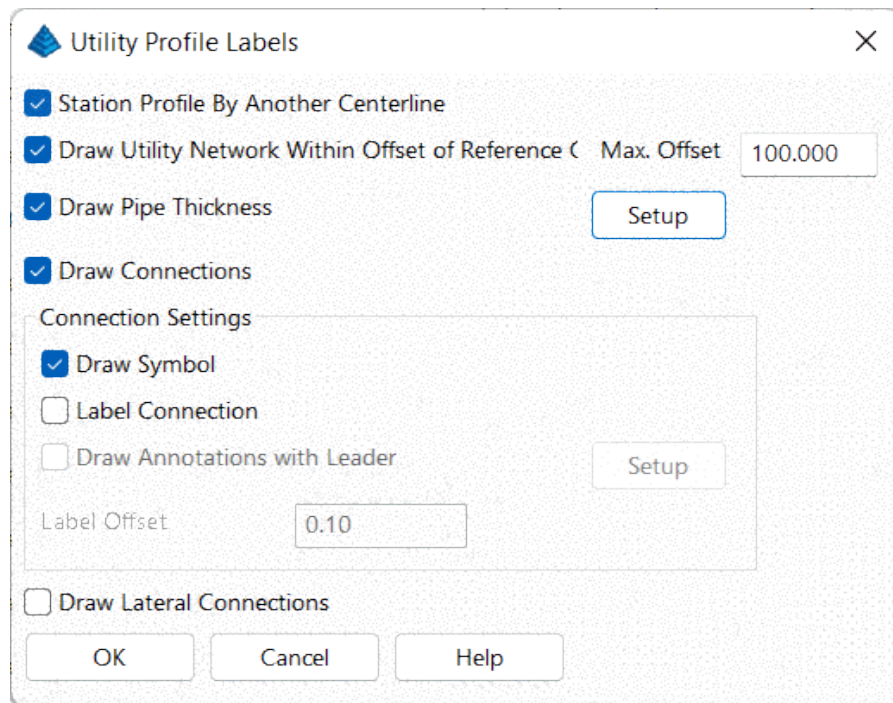
Draw Annotation with Leader: If you want to use a leader, enable this option.



Show Lateral Thickness: Use this setting to show the thickness of a lateral.

Update Preview: Use this button to update the preview window and review your changes before committing them to a drawing.

Utility Profile Labels



Station Profile By Another Centerline: This option will derive stations by referencing the utility to another centerline, for example a road centerline.

Draw Utility Network Within Offset of Reference: Set the Max. Offset for utility networks to draw.

Draw Pipe Thickness: If a pipe size thickness is specified within the Pipe Size Library the program will also draw wall thickness.

Draw Connection: When enabled, and for Utility Networks that have been assigned Connections, the connection point(s) is (are) drawn into the profile.

Label Connection: When enabled, and for Utility Networks that have been assigned Connections, the label(s) for the connection point(s) is (are) drawn into the profile.

Draw Annotations with Leader: This option will permit the Utility label to be placed with a leader with further control via the **Setup** option.

Label Offset: Indicate the plotted distance the Utility label should be drawn away from the Utility entity.

Prompts (may vary based on Settings)

Profile Grid Range

Profile Station Range: 0+00.00 to 5+68.19

Profile Elevation Range: 349.25 to 368.04

Grid Starting Station: -50.000

Grid Ending Station: 625.000

Grid Top Elevation: 380.000

Grid Bottom Elevation: 330.000

Use Fixed Grid Top/Bottom Elevations

Fixed Grid Top Elevation: 385.000

Fixed Grid Bottom Elevation: 305.000

OK Cancel

Grid Starting Station: Indicate the desired starting station for the profile grid.

Grid Ending Station: Indicate the desired ending station for the profile grid.

Grid Top Elevation: Indicate the bounding upper elevation for the profile.

Grid Bottom Elevation: Indicate the bounding lower elevation for the profile.

Use Fixed Grid Top / Bottom Elevations: Allows user to set fixed parameters for profile top and bottom elevations for all profiles.

Polyline should be drawn in direction of increasing stations.

CL File/ <Select pipe crossings on-the-fly or parallel pipes centerline>: Pick a polyline upon which to base the stationing or *Type C to select an existing Centerline .CL file and then press Enter*

Centerline Starting Station <0.0>: *Press Enter to accept the default station value specified or Type in the beginning station then press Enter*

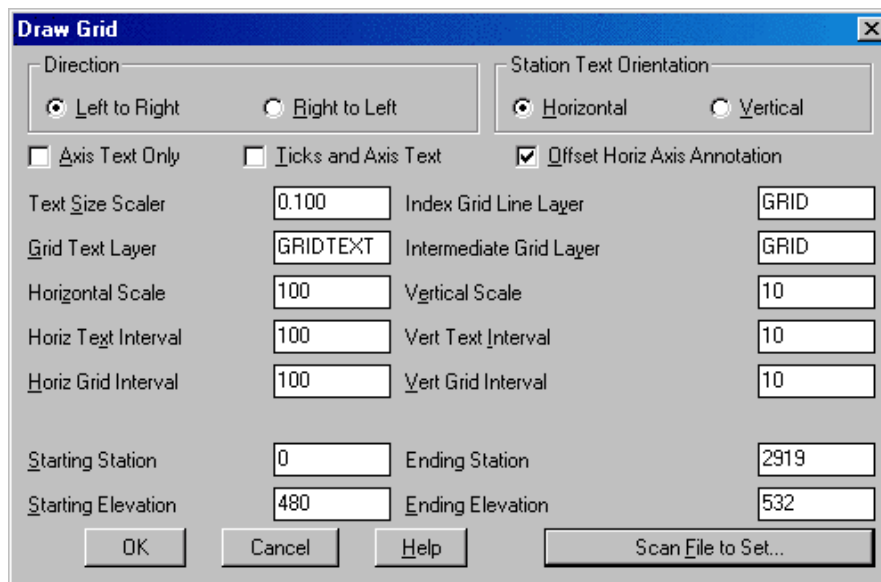
Pulldown Menu Location(s): Civil > Profiles, Survey > Surface, Hydrology > Network, Construction/Takeoff > Roads, Field > Roads

Keyboard Command: drawprof

Prerequisite: A profile .PRO file

Draw Profile Grid

This command plots a grid and labels the grid lines with stations and elevations. Profile grids can also be plotted along with the profile when using the command *Draw Profile*. Use this command to draw only the grid. The following dialog box appears:



Direction: Choose grid direction, either left to right or right to left.

Station Text Orientation: Specify whether the station text should be plotted horizontal or vertical.

Axis Text Only: When checked, grid lines are not drawn.

Ticks and Axis Text: When checked, one horizontal and vertical grid line as well as the annotations will be drawn.

Offset Horizontal Axis Annotation: When checked, additional space is added between the bottom horizontal grid line and the station labels in order to leave room for Horizontal Axis Elevations and sewer profile annotations.

Text Size Scaler: This sets the size of text used for annotation. This value is multiplied by the horizontal scale to obtain actual text size.

Index Grid Line Layer: Specify the layer name for index grid lines.

Grid Text Layer: Specify the layer name for text annotation along the horizontal and vertical axis.

Intermediate Grid Layer: Specify the layer name for intermediate grid lines.

Horizontal Scale: This sets the horizontal scale for the profile grid.

Vertical Scale: This scale sets the vertical exaggeration of the profile grid.

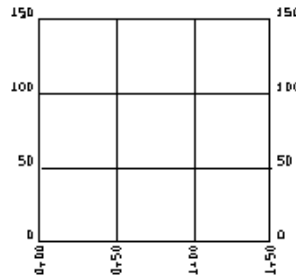
Horiz. Text Interval: This sets the spacing of the stationing text that appears along the horizontal axis. If you use a large Text Size Scaler such as 0.2 in English units, it is best to set the horizontal text interval to twice the horizontal scale, so that the larger text will not overlap along the horizontal axis.

Vert. Text Interval: This sets the spacing of the elevation text that appears along the vertical axes.

Horiz. Grid Interval: This sets the spacing of the grids that run vertically from the horizontal scale.

Vert. Grid Interval: This sets the spacing of the grids that run horizontally between the vertical axes on the left and right side of the profile.

Scan File to Set: Prompts to select a profile (.PRO) file which it reads to set the values for starting and ending stations and elevations.



Draw Grid with Station Text Vertical and no Offset Horizontal Axis

Prompts

Draw Grid dialog box

Pick Starting Point for Grid <0.0 , 500.0>: *pick a point*

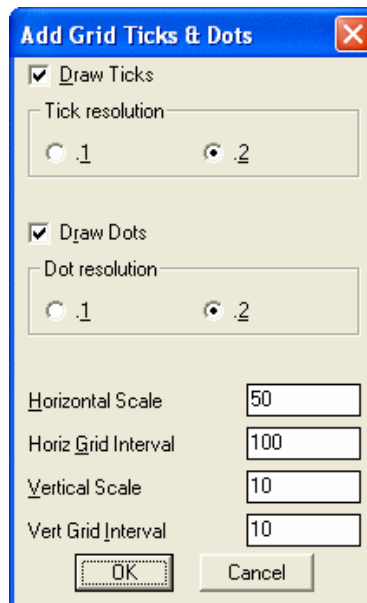
Pulldown Menu Location: Profiles > Profile Grid

Keyboard Command: drawgrid

Prerequisite: None

Add Grid Ticks and Dots

This routine draws ticks on the axis and/or interval dots on an existing grid.



Prompts

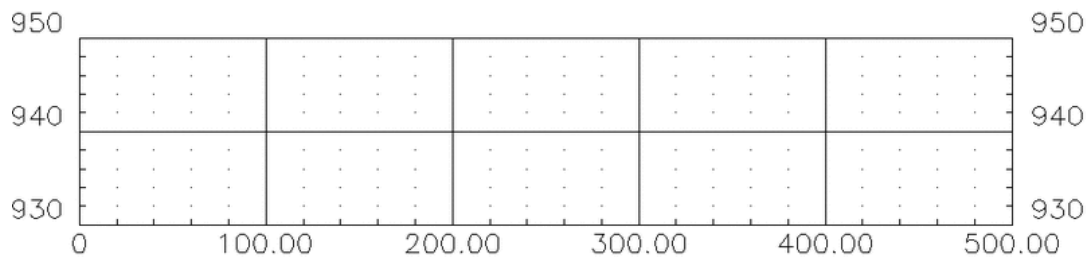
Add Grid Ticks & Dots dialog

Specify whether to draw the ticks and/or dots, and choose their resolutions (.1 or .2). Also, make sure the grid

parameters match the grid that you're working on.

Pick Lower Left Corner of Grid: *pick the corner* (endpoint snap is on)

Pick Upper Right Corner of Grid: *pick the corner* (endpoint snap is on)



Grid ticks and dots with metric stationing (no '+') as set in Profile Defaults

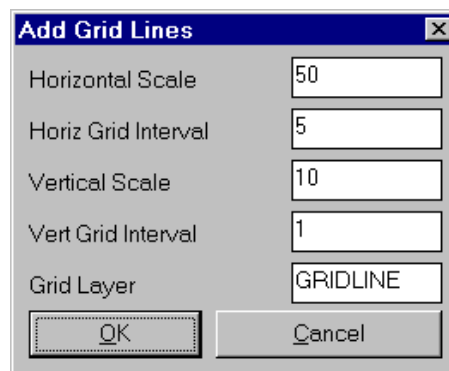
Pulldown Menu Location: Profiles > Profile Grid

Keyboard Command: tickdot

Prerequisite: A profile grid

Add Grid Lines

This routine draws grid lines at the specified scale and interval between the picked lower left and upper right grid corners.



Prompts

Add Grid Lines dialog

Make sure the grid parameters match the grid that you're working on.

Pick Lower Left Corner of Grid: *pick the corner* (endpoint snap is on)

Pick Upper Right Corner of Grid: *pick the corner* (endpoint snap is on)

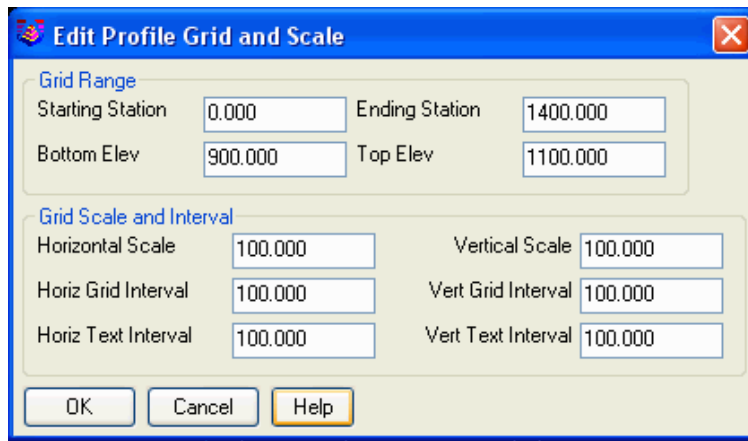
Pulldown Menu Location: Profiles > Profile Grid

Keyboard Command: gridline

Prerequisite: A profile grid

Adjust Profile Grid

This command allows you to modify the profile grid parameters for a profile that is already drawn. First you pick on any entity that is part of a drawn profile. Then change any of the settings in the dialog shown here and the program will update the profile.



Prompts

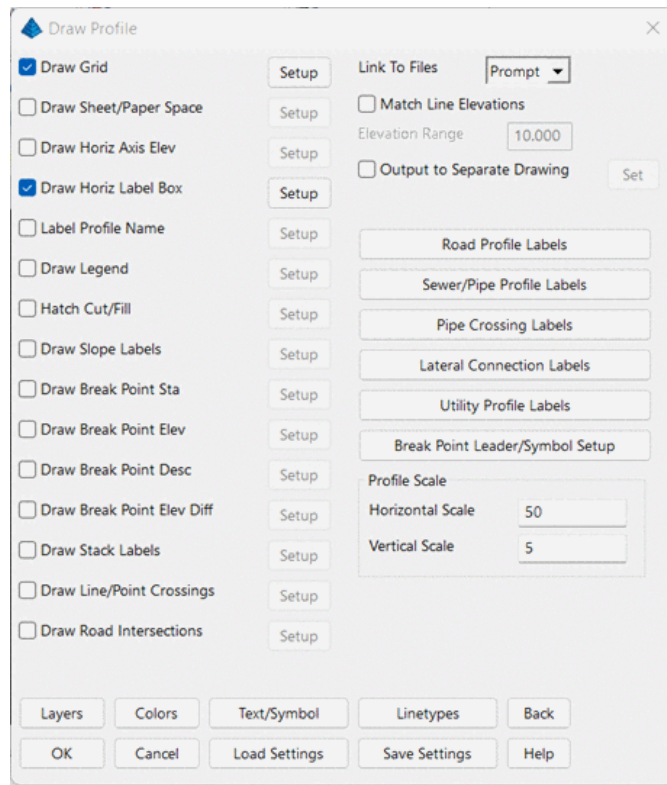
Pick profile to edit: *pick any entity from a drawn profile*
Edit Profile Grid and Scale dialog

Pulldown Menu Location: Profiles > Profile Grid/Sheet
Keyboard Command: edit_drawprof
Prerequisite: A drawn profile

Adjust Draw Profile Settings

This command allows changes to be made to an existing profile, either on the Model tab or on a layout tab. These changes include all of the options that were used to create the profile originally using the Draw Profile command.

The first step in Adjust Draw Profile Settings is to choose the profile you want to change by picking any entity on the profile drawing. Once you have selected a profile in the drawing, the **Draw Profile** dialog box appears, and contains all of the settings for creating the profile. Please see the Draw Profile section of this manual for a description of these settings.



When OK is clicked at the base of the dialog box, the profile is updated in its current location.

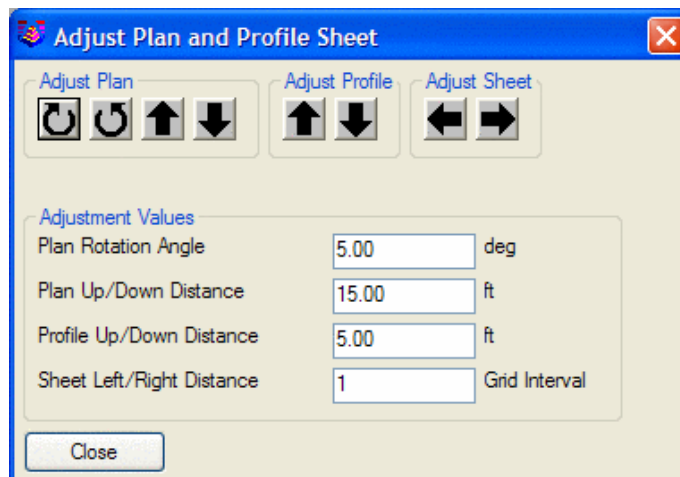
Pulldown Menu Location: Profile Grid/Sheet > Adjust Draw Profile Settings

Keyboard Command: reset_drawprof

Prerequisite: A profile in the drawing

Adjust Plan/Profile Sheet

This command is used on a Plan & Profile sheet generated by the Draw Profile command with the Draw Sheet option selected. Adjustments can be made to the plan view, profile, or sheet itself. The command is run entirely from the Adjust Plan and Profile Sheet dialog box. Adjustment settings are defined, and then all adjustments are accomplished simply by picking the appropriate icons.



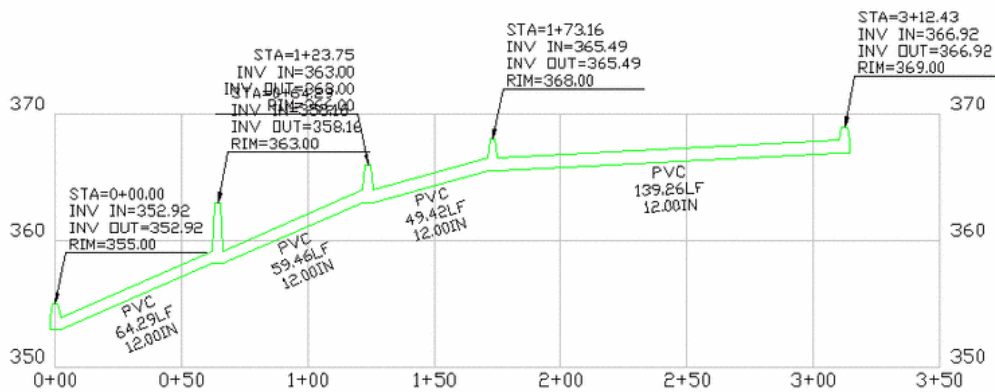
Pulldown Menu Location: Profiles > Profile Grid/Sheet

Keyboard Command: ppsheet

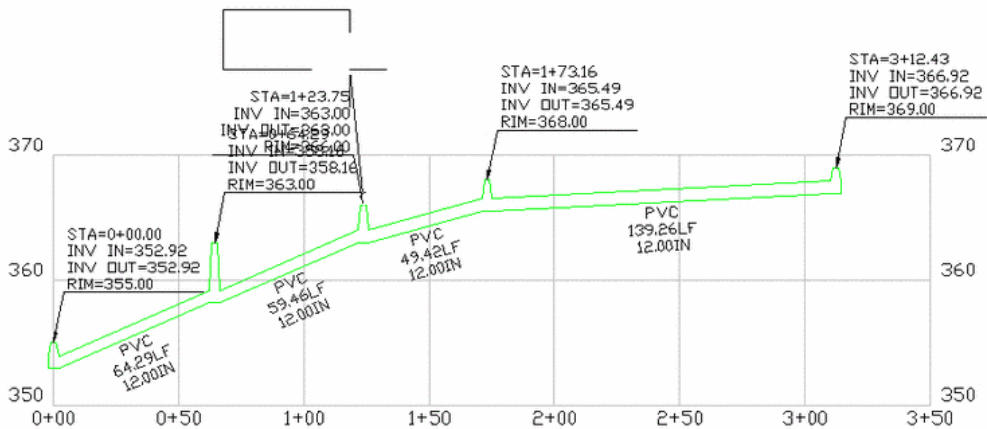
Prerequisite: A Plan & Profile paper space layout generated with Draw Profile command

Move Sewer Profile Labels

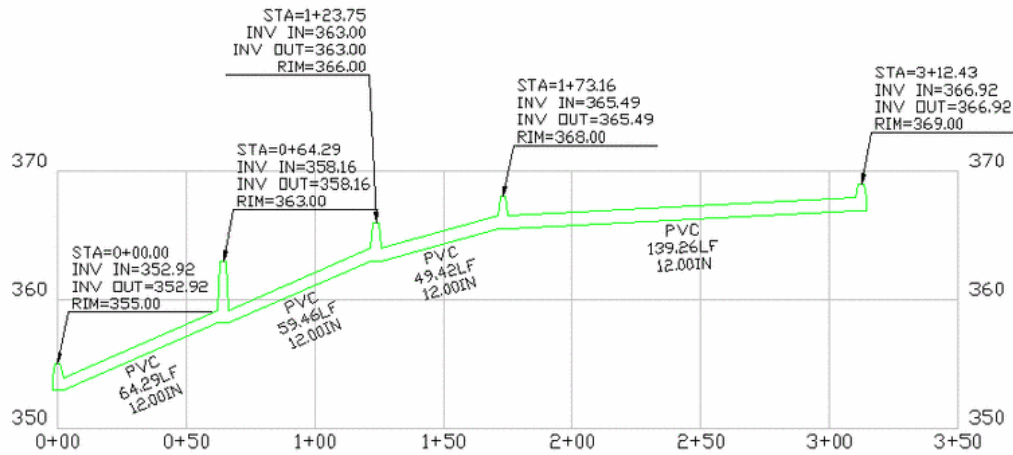
This command moves the selected stack profile labels and sewer profile labels with a leader. The purpose is to clean up label overlaps. To move a label, pick any one of the text labels and the program will pick up all the other associated labels. Then pick the new location and while the pointer is moved, the program shows an outline of the label area. The program remembers the moved locations for each label so that when the sewer profile labels are redrawn, the moved locations are retained. The Restore function puts the labels back to their default position. The following graphics show the sewer profile labels before and after Move Sewer Profile Labels was used to clean up the label overlaps.



Sewer Profile Labels with Overlap



Move a label away



Sewer Profile Labels without Overlap

Prompts

Select sewer profile label to move (R for Restore): *Pick a sewer profile label text with leader entity*

Pick label position: *pick a point*

Select sewer profile label to move (R for Restore, Enter to end): *press Enter to end*

Pulldown Menu Location: Profiles > Profile Grid/Sheet

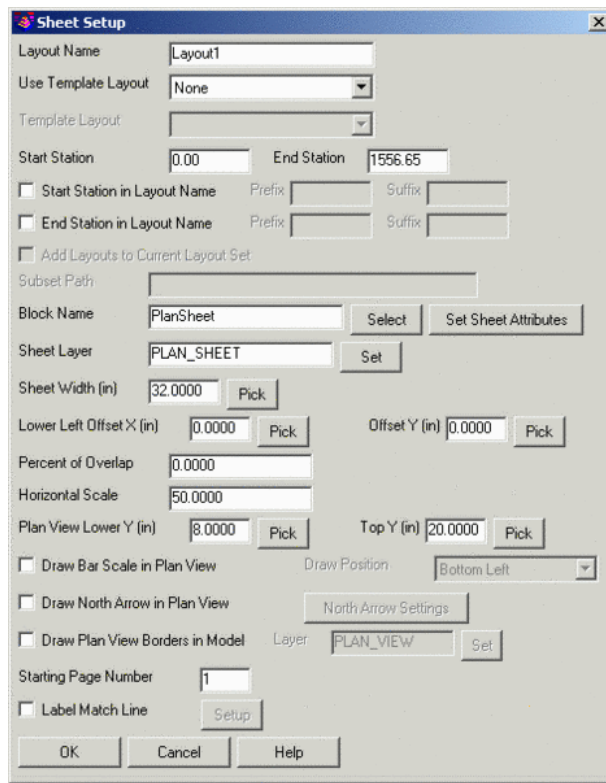
Keyboard Command: move_swrprof_label

Prerequisite: sewer profile labels with leader

Draw Plan View Sheets

Draw Plan View Sheets creates layout tabs with viewports for plotting an existing centerline. These plan view sheets are sized and oriented automatically based on the centerline design.

The first step in Draw Plan View Sheets is to choose the centerline to be plotted. This may be selected from the drawing, or you may select an existing centerline (.CL) file. Once you have selected a centerline, the **Sheet Setup** dialog box appears, and contains all of the settings for creating the layout tabs and sheets.

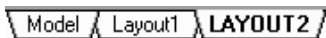


Layout Name: Enter a name for the paper space "tabs" to be assigned to each layout for each sheet. The program will automatically divide the centerline plan view sheet layouts, and if the length of the centerline extends beyond a single sheet, then multiple layouts are created, with the layout name ID incremented by 1.

Note:

- If either the *Start Station in Layout Name* or the *End Station in Layout Name* options are enabled, the Layout Name field will be disabled as the Layouts will get named automatically.

If you enter "ms" to go to model space within a Layout tab, you can pan to alter the plan view position. However, it is best to zoom in/out and edit within the Model tab. The Layout tabs appear at the bottom of the screen, along with the "Model space" tab to go back to standard plan view:



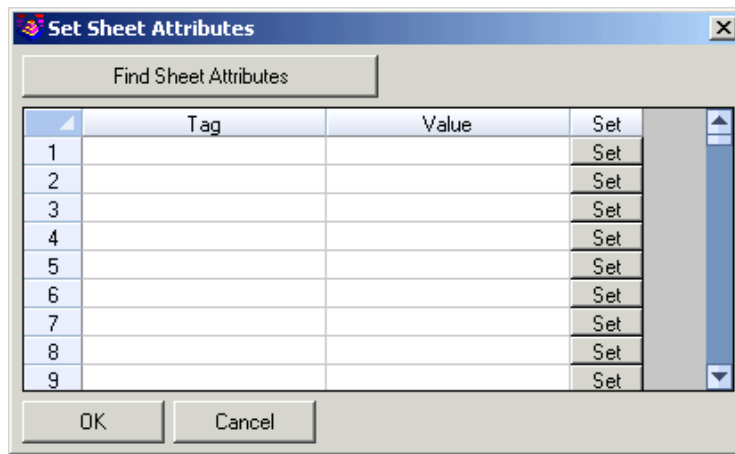
Start/End Stations: These values control the station range to draw.

Start/End Station in Layout Name: These options allow you to include starting and ending station in the Layout Names.

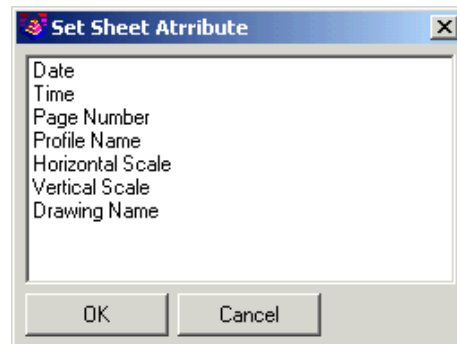
Add Layouts to Current Layout Set: This option allows you to add the layouts created to an existing layout set that was previously generated using the Layout Set Manager. You will need to specify the name of the layout set.

Block Name: This is the drawing name for the sheet to be inserted. The Set button can be used to change the block name. Carlson provides a standard plan and profile border in the form of profile.dwg located in the working folder of %AppData%\Carlson Software\...\Sup\. You may wish to revise profile.dwg to remove the grid, resize the plan viewport, and add your company logo, and re-save it as plan.dwg. Alternatively, you could add your own complete version of a sheet block/border.

Set Sheet Attributes: This button allows you to specify the values used by any attributes associated with the sheet block. These can be entered manually in the Set Sheet Attributes dialog.



You can use the Set button to the right of any field to set that field to a preset value pulled from the drawing information.



Sheet Width: This is the profile width, in inches, on the sheet.

Lower Left Offset X/Y: Indicate the offset value(s) for the insertion point of the sheet in CAD units. This option allows user-defined Block Names to be properly positioned relative to the remainder of entities placed through the Draw Profile command.

Percent of Overlap: Use this to set the amount of centerline shown beyond the match line on each sheet. This can make it easier to piece together sections of the centerline on separate sheets.

Horizontal/Vertical Scale: Set the scales to be used for the new viewport(s).

Plan View Lower Y: This sets the lower position of the paper space window for the plan view. With Lower Y set to 9 (inches above the base of the sheet) and Top Y set to 21, there is a 12 inch vertical window, running the full Sheet Width (typically 30 to 32). This window for the plan view can be expanded or reduced with these settings.

Top Y: This sets the top vertical limit for the plan view window, measured in inches from the bottom of the plan and profile sheet.

Draw Scale Bar in Plan View: Adds a scale bar to the finished sheet. If this option is selected, the scale bar can be placed in any corner of the sheet using the Draw Position drop down.

Draw North Arrow in Plan View: This draws a North Arrow in plan view. Click the **North Arrow Settings** button to establish the desired North arrow and placement information.

Draw Plan View Borders in Model Space: This draws the borders in Model Space which can be useful for orienting text and other labels to the orientation of the sheet. When this option is selected, use the Layer text box or Set button to choose the layer on which the borders will be drawn.

Label Match Line: When clicked on and multiple sheets are plotted with plan view option on, a match line will plot in the plan view.

When OK is clicked at the base of the dialog box, the layout tabs are created if necessary, and the sheets are drawn.

Pulldown Menu Location: Profiles

Keyboard Command: plansh

Prerequisite: None

Horizontal Axis Elevations

This command labels the elevations of a profile along the bottom horizontal axis at a user-specified interval. It requires an existing grid and profile. The profile can be read from either a .PRO file or from a profile polyline on the grid. This polyline must be drawn in the direction of the grid. There are more labeling options when using the screen polyline method.

In the dialog, you can set the layer name, style, size, color and decimal places for the labels. Two profiles can be labeled at once to handle existing and final profiles in one step (see graphic). When labeling two profiles with the "File" method (recalling a profile), use the "L" justification for the first set of horizontal axis elevations, and use the "R" justification for the second set. One convention is to label the existing profile to one decimal place and the final profile to two decimal places. When labeling only one profile, use the center justification. When using two profiles from the "Screen" selection method, there is an option to also label the elevation difference between the profiles. The Label Between Elevations option chooses between labeling the values in the order of existing elevation, cut/fill and final elevation or in the order of existing elevation, final elevation then cut/fill. The Skip Elevation Labels option will label only the cut/fill and not the elevations. The Save and Load functions are to save and recall settings from this dialog using a .HOR file.

Horizontal Axis Elevations

Direction of Grid
 Left to Right Right to Left

Horizontal Scale: 50.0
Horizontal Text Interval: 50.0
Vertical Scale: 5.0

Grid Starting Station: 0.0 Elevation: 0.0

Existing Grade
Precision: 0.0
Text Height: 4.00
Layer: PROTXT Set
Style: ROMANS Set
Color: ByLayer

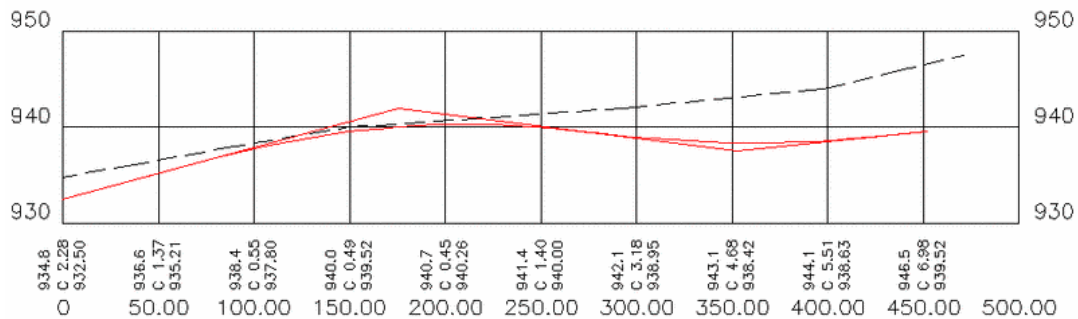
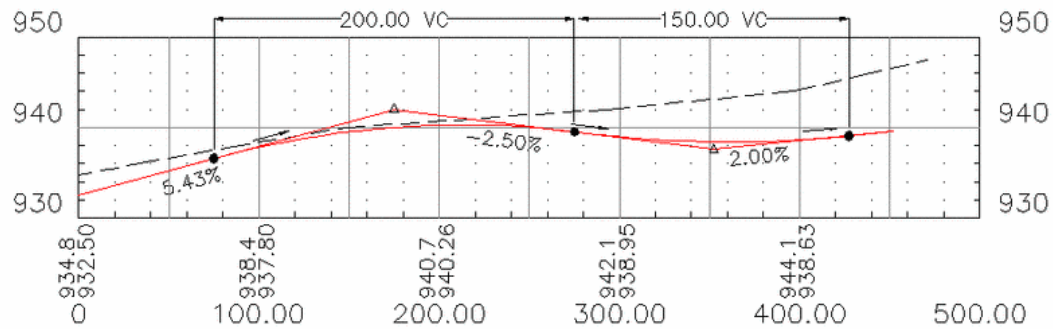
Final Grade
Precision: 0.00
Text Height: 4.00
Layer: PROTXT2 Set
Style: ROMANS Set
Color: ByLayer

Text Layout
 Vertical Horizontal
Text Offset Scaler: 7.00

Elevation Difference Options
 Create Cut/Fill Labels Precision: 0.00
 Label Between Elevations Text Height: 4.00
 Skip Elevation Labels Layer: PROTXT3 Set Style: ROMANS Set

Use Design Box Vertical Lines

OK Cancel Help Save Load



Prompts

Read Profile from a File or from the Screen (File/<Screen>): *press Enter*

Plot Elevations on Horiz Axis dialog

Make sure the grid starting station and elevation match the grid that you're working on.

Pick the existing grade (Enter for none): *pick a profile polyline on the grid*

Pick the final grade (Enter for none): *press Enter*

Alignment of text (<Left>/Center/Right)? *C* This prompt occurs only in the "File" selection method.

Pick Lower Left Grid Corner: *pick the corner* (endpoint snap is on)

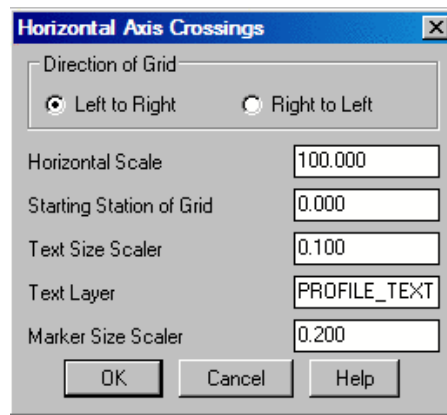
Pulldown Menu Location: Profiles > Label Horizontal Axis

Keyboard Command: horelev

Prerequisite: Profile grid with a profile polyline

Horizontal Axis Crossings

This purpose of this command is to draw ticks on the horizontal axis of the profile at station locations where the centerline intersects selected plan view polylines. It requires a grid, profile and an existing CL file, as well as user-specified values entered into the dialog. The profile can be read from either a .PRO file or from a profile polyline on the grid. This polyline must be drawn in the direction of the grid. In the dialog, you can set the direction of the grid, the horizontal scale and the starting station of the grid. You can also determine the Text Size Scaler, Text Layer name and the Marker Size Scaler. The command line offers the option to choose the existing centerline (.CL) file. You enter "C" and a dialog appears where you may select the file.

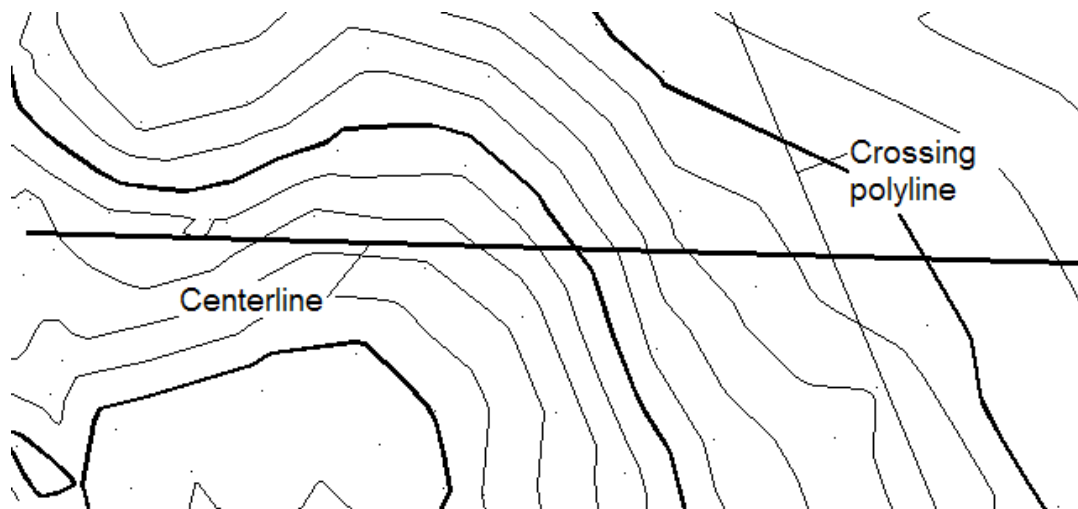


Prompts

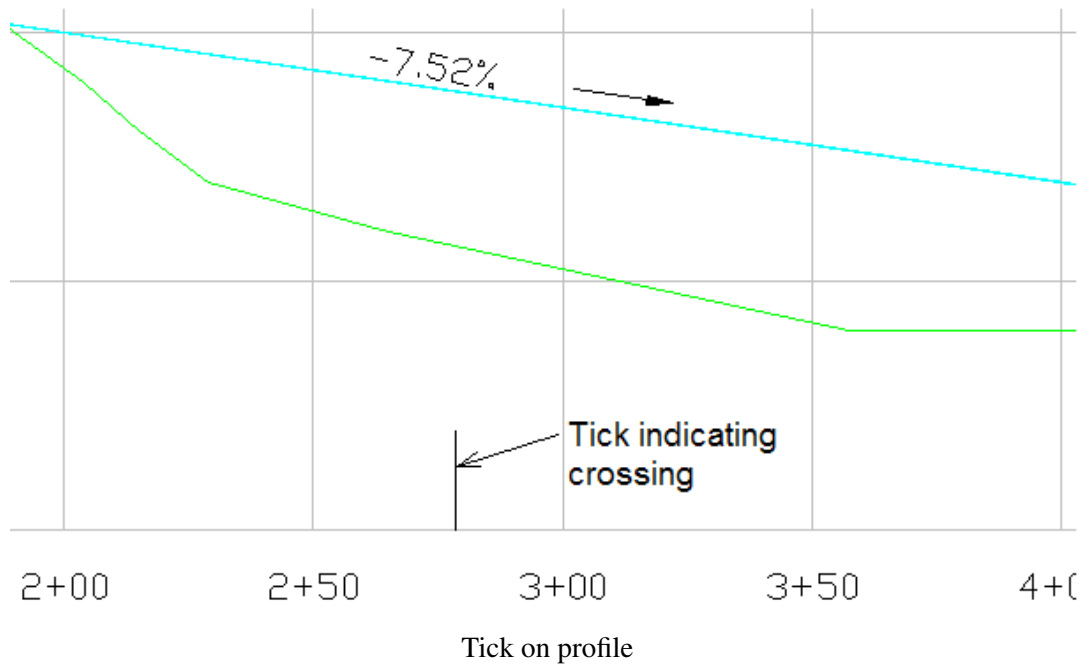
Horizontal Axis Crossings dialog Fill in values.

Polyline should have been drawn in direction of increasing stations.

CL File/ <Select polyline that represents centerline>: *pick polyline*



Plan view showing crossing



Pulldown Menu Location: Profiles > Label Horizontal Axis

Keyboard Command: horcross

Prerequisite: Profile, profile centerline polyline and a .CL file

Profile to 3D Polyline

This command converts a 2D polyline centerline into a 3D polyline that follows the elevations of the profile. Horizontal and vertical curves are represented as a series of polyline segments since 3D polylines cannot contain arcs. Profile to 3D Polyline can be combined with other commands for plan-view road design as follows:

1. Draw 2D polyline centerline.
2. Profile from Surface Model - to create existing surface profile.
3. Design Road Profile - to design the final profile with vertical curves.
4. Profile to 3D Polyline - create a 3D polyline of the road centerline.
5. Offset 3D Polyline - offset the 3D polyline centerline left and right by the horizontal and vertical distances.
6. Design Pad Template - run twice for left and right polylines of road to tie into surface at specified cut and fill slopes. This creates the limits of the disturbed area. Or use Join Nearest, Direct Connect Endpoints, to create a closed loop pad with one run of Design Pad Template for simple ramps, driveways and access roads.
7. Triangulate & Contour - draw final contours using road 3D polylines.
8. Volumes - use any of the volumes commands to calculate cut and fill volumes.

Prompts

Layer Name for 3D Polyline <3DPROF>: *press Enter*

Select profile centerline polyline: *pick a polyline*

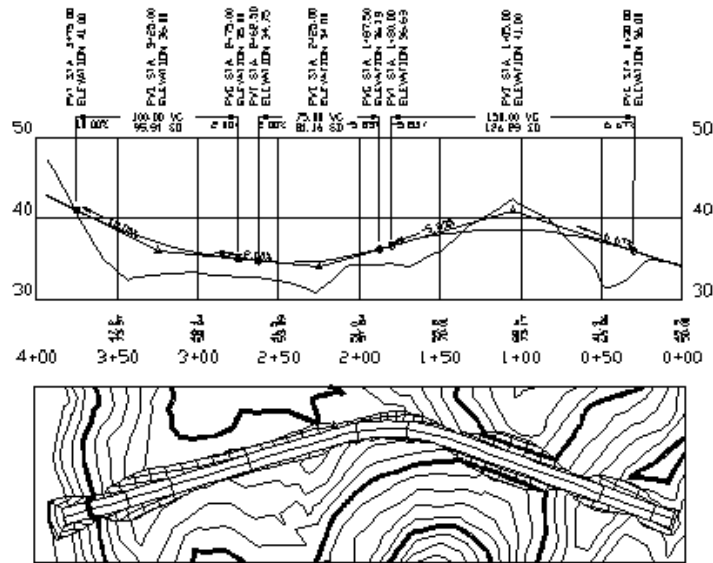
Station by another reference centerline [Yes/<No>]? *N* for no. This option will prompt for a second centerline to use for stationing.

Enter the starting station <0.0>: *press Enter*

Select Profile File

Starting station of centerline <0.0>: *press Enter*

Erase centerline (Yes/<No>)? *Y* This option will erase the original 2D polyline centerline.



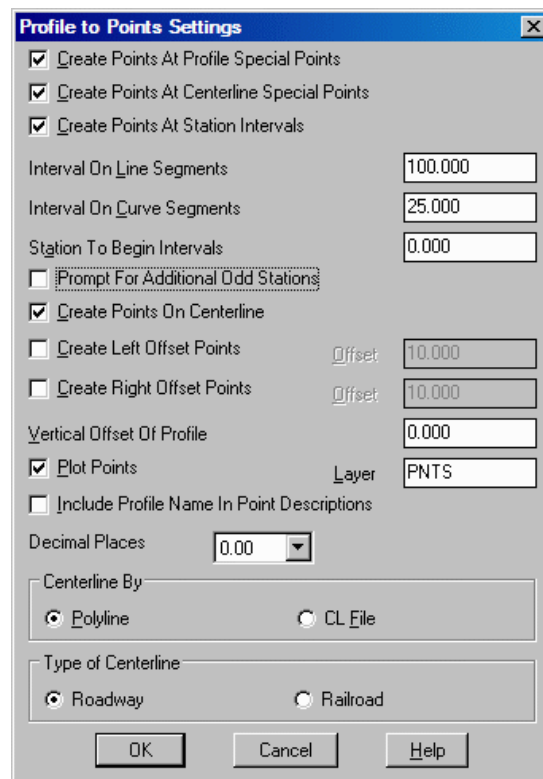
Example of road design in plan-view with Profile to 3D Polyline

Keyboard Command: proto3dp

Prerequisite: A .PRO file and a centerline polyline

Profile To Points

This command creates Carlson points along a horizontal alignment polyline using a profile file to compute the point elevations. The created points are stored in a coordinate (.CRD) file and can also be drawn on screen in the layer specified by the user. Station text, profile name, and special points (vertical and horizontal PC's and PT's) can be stored in the point description depending on user settings.



Create points at Profile special points: Includes vertical PC and PT points.

Create points at Centerline special points: Includes horizontal PC and PT points.

Create points at Station Intervals: Allows you to specify intervals for point creation.

Interval On Line Segments: Specify station interval for line segments.

Interval On Curve Segments: Specify station interval for curve segments.

Station to Begin Intervals: Specify station to start intervals.

Prompt For Additional Odd Stations: Any station can be entered to create additional points with elevations derived from the profile.

Create Points on Centerline: When checked, points will be created on the centerline.

Create Left Offset Points: When checked, left offset points will be created. Specify the offset in the edit box.

Create Right Offset Points: When checked, right offset points will be created. Specify the offset in the edit box.

Vertical Offset of Profile: Specify the vertical offset. Enter zero for no vertical offset.

Plot Points: When checked, points will be plotted in the drawing, otherwise points are only added to the current coordinate (.CRD) file.

Include profile name in point descriptions: When checked, the profile name will be used as the prefix on the point description. For example, if the profile name is DESIGN.PRO, then the point description might be DESIGN 0+63.37.

Decimal Places: Specify the display precision for points that are plotted in the drawing. This setting does not affect the coordinates stored in the CRD file.

Centerline by: Click either Polyline or CL File.

Type of Centerline: Click either Roadway or Railroad.

OK: Specify files.

Prompts

Select Coordinate File to Process

If the current coordinate is set, it is used automatically without this prompt.

Select profile centerline polyline: *pick a polyline*

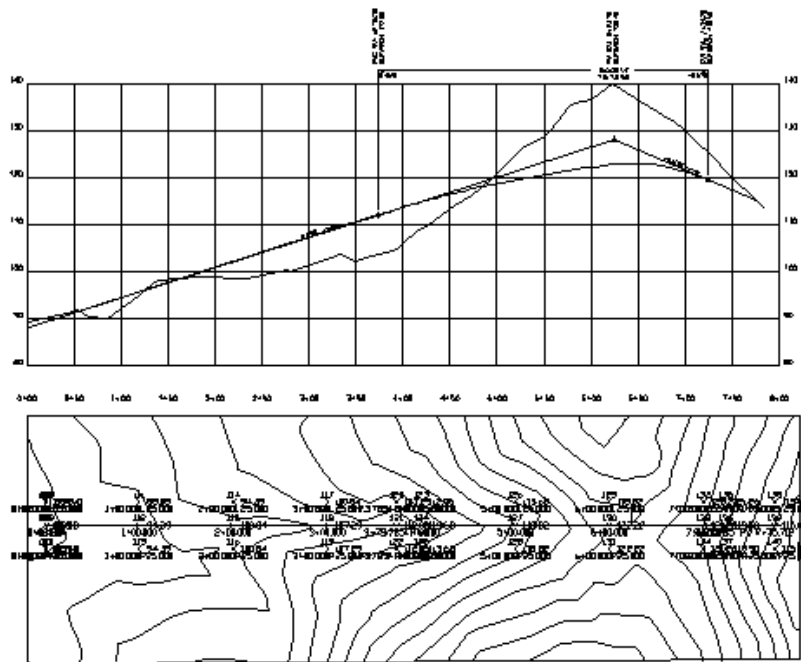
Starting station of centerline <0.0>: *press Enter*

Station by another reference centerline [Yes/<No>]? *N for no.* This option will prompt for a second centerline to use for stationing. With this option, the first centerline is used for locating the points and the second reference centerline is used for locating the profile stations. So the first centerline represents where the points are created (ie. the edge of pavement) and the second centerline represents the profile location (ie. the road CL).

Choose Profile to Process dialog Specify a profile name.

Starting point number <1>: *press Enter* This defaults to the point number after the highest one currently in the CRD file.

Station for additional point (ENTER to end): *press Enter* This option will create a point at the specified station. Prompt occurs only if option is specified in dialog.



Points created along profile centerline using elevations from the above road profile

Keyboard Command: pro2pts

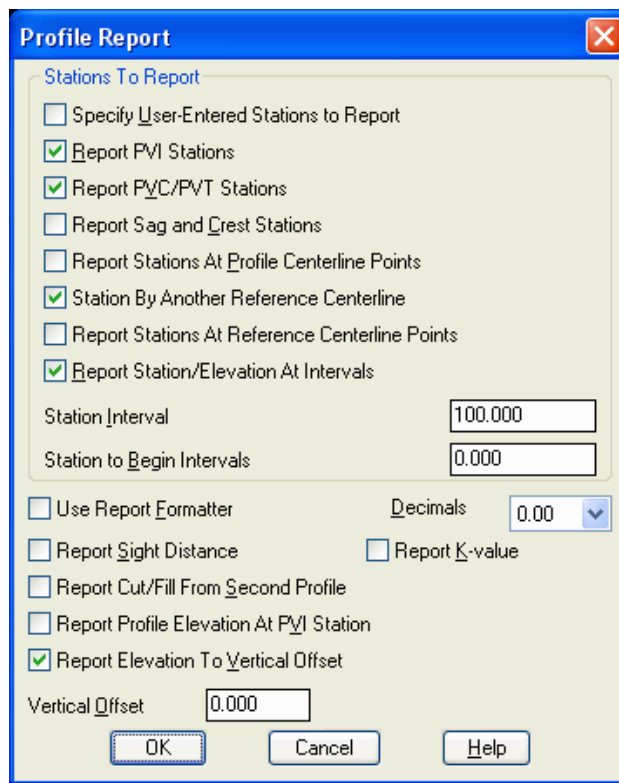
Prerequisite: A .PRO file and a centerline polyline

Profile Report

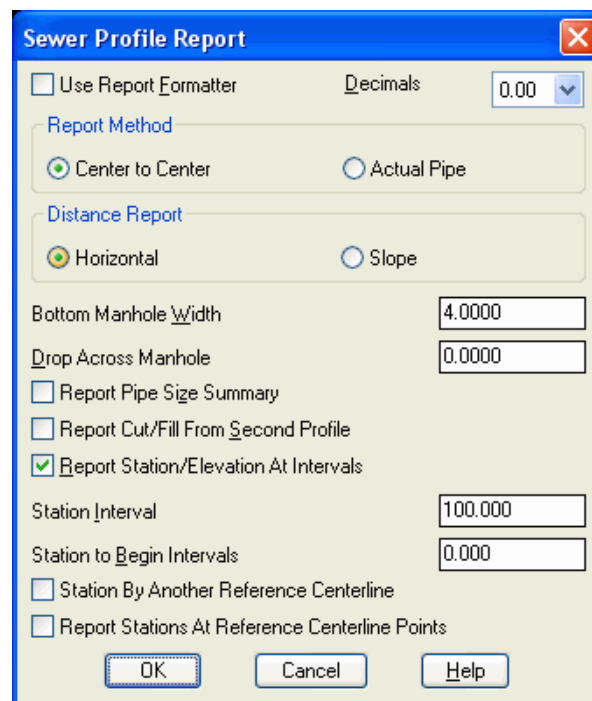
This command creates a summary report of generic, road, crossing, pipe and sewer profiles using a profile file (.PRO file). The report is generated in the standard report viewer which can print the report, save it to a file or draw it on the screen. The different types of profiles have different report options.

To report stations adjusted by station equations, first run Centerline Editor and enter the station equations. Then run Profile Editor and assign the centerline as the Reference Centerline for the profile.

For roadway profiles, Report Sag and Crest Stations will calculate and report sag and crest stations and elevations. Report Stations at Centerline Points will prompt the user for a centerline file (.cl file) and report stations and elevations at horizontal PC and PT points. Report Cut/Fill from Second Profile will compute and report the elevation difference between the subject profile and a second reference profile. Report Min/Max Cut/Fill reports the stations and amounts for the min and max cut and fill between the road and reference profiles. Report Station/Elevation at Interval will calculate and report stations at the specified interval in addition to other points. Report Elevation to Vertical Offset creates an additional elevation column in the report. The differential amount for this column is specified by the user in the Vertical Offset window. The Use Report Formatter option runs the report through the report formatter where you can choose which fields to report and the report order as well as output to Excel or databases.



For sewer profiles, the Report Method chooses between reporting the slopes and pipe distances between manhole centers or from the outside manhole edges for the actual pipe dimensions. The Report Pipe Size Summary option reports the total pipe length for each different pipe size. The Station By Another Reference Centerline option reports the sewer stations using a reference alignment besides the sewer alignment. For example, this option can be used to report the sewer stations based on the road centerline. When this option is on, the program will prompt for both the sewer centerline and the reference centerline. The program first finds the position of the sewer station along the sewer centerline and then finds the station of the nearest perpendicular offset along the reference centerline.



Prompts

Specify a Profile File dialog Choose the .PRO file.

Profile Report dialog Make selections, click OK.

If a vertical offset is entered, a second column of elevations is reported.

Sample Profile Report:

```
Profile Report
Road Profile
Station Elevation Type VertCurve Distance Slope Desc
0+00.00 88.08          0.00
1+00.00 94.39          6.45%
2+00.00 100.84         6.45%
3+00.00 107.29         6.45%
3+73.78 112.05         PVC          371.48      6.45%
4+00.00 113.68          6.00%
5+00.00 118.82          4.27%
6+00.00 122.22          2.54%
6+23.78 128.18         PI    350.00    250.00      6.45%
7+00.00 121.26          -6.10%
7+23.78 119.50         PVT          100.00     -8.67%
7+75.71 115.00          0.00         51.93      -8.67%
```

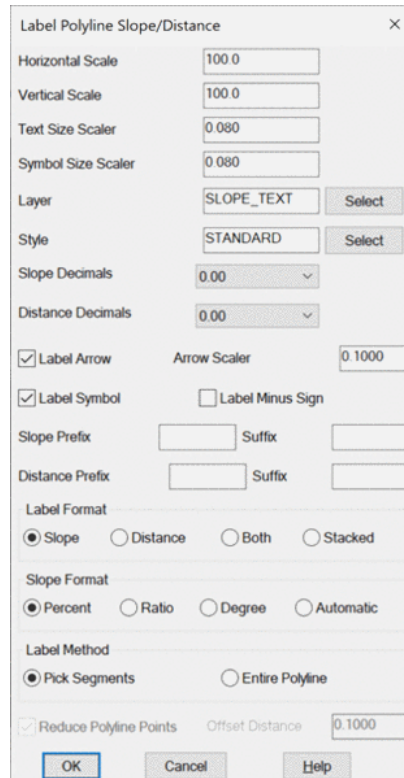
Pulldown Menu Location: Profiles

Keyboard Command: preport

Prerequisite: A .PRO file

Polyline Slope Report

This command calculates the slope and distance of a line, polyline segment, an entire polyline, or pair of points, as drawn on a profile. There are options to label the slope and distance. If the labels are turned off, this command can be used to just report the profile slope and distance. The distance is the horizontal distance between the profile stations.



Horizontal Scale: Specify the horizontal scale of the profile grid.

Vertical Scale: Specify the vertical scale of the profile grid.

Text Size Scaler: Specify the text size scaler which is multiplied by the Horizontal Scale.

Symbol Size Scaler: Specify the symbol size scaler which is multiplied by the Horizontal Scale.

Layer: Layer name for the labels.

Style: Text style for the labels.

Slope and Distance Decimals: Specify the display precision for the slope labels and distance labels.

Label Symbol: When checked, the degree symbol or percent sign will be used in the label.

Label Arrow: When checked, a slope direction arrow will be included.

Arrow Scaler: Specify the arrowhead size scaler which is multiplied by the Horizontal Scale.

Label Minus Sign: Will label a minus sign on negative slopes.

Slope Prefix/Suffix: Strings to add to the slope value for the slope label.

Distance Prefix/Suffix: Strings to add to the distance value for the distance label.

Label Format: Choose whether to label slope and distance.

Slope Format: Specify how to label the profile slopes. The automatic settings means to use a percent label for any slope less than 10%. and a ratio for any slope greater than 10%.

Label Method: Choose to label the entire profile at once or to pick individual segments.

Reduce Profile Points: When checked, the number of labels created on the profile will be reduced based on the Offset Distance value. Applies only to the Entire Polyline selection option..

Offset Distance: Specify maximum offset between profile vertices. Only available when Reduce Profile Points toggle is checked on.

Prompts

Slope Report Options dialog box

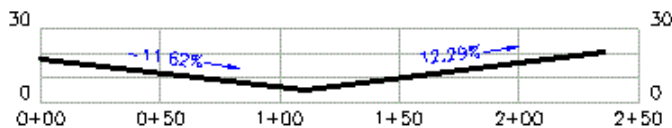
Points/<Select line or polyline to list-label>: *pick a polyline*

Slope Distance> 600.33 Horizontal Distance> 600.00

Elevation Difference: 20.00 Slope Ratio: 30.00:1 Slope Percent: 3.33

Starting point of label ([Enter] for none): *pick a point*

Points/<Select line or polyline to list-label>: *press Enter* If you choose P for points, you go into the Points mode and can label the slope of any pair of screen picks on the profile.



Pulldown Menu Location: Profiles

Keyboard Command: llg

Prerequisite: A profile grid and profile polyline

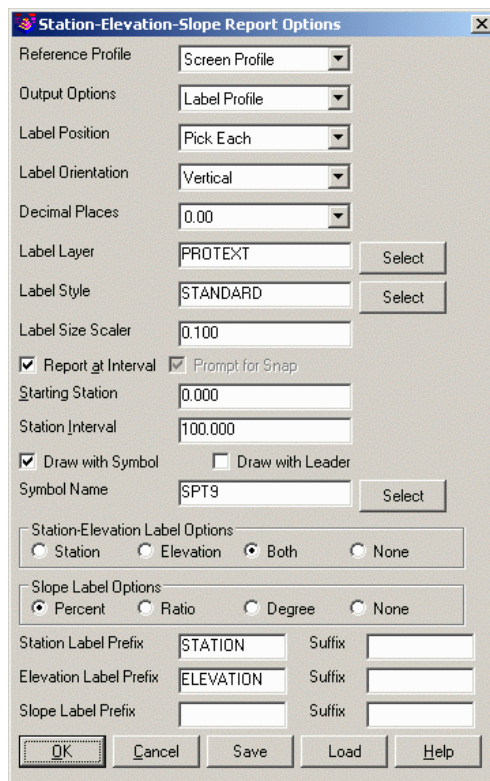
Station-Elevation-Slope Report

This command calculates the elevation and slope along a profile at user specified stations or intervals. The routine allows three types of **Reference Profile** input options: Profile File (an existing .PRO file), Screen Profile (existing grid and polyline profile), or None (allows you to specify station-elevation points without referencing a profile). If the Screen Profile option is used, the profile polyline direction must match that of the stationing on the grid.

There are two **Output Options**: Report and Label Profile. The Report option will send the output data to the standard report viewer, which can then be printed, saved to a file or plotted in the drawing. The Label Profile option will create text on the existing grid and polyline profile. With either option, the user will be prompted to enter or pick the station to report unless the **Report at Interval** option is checked on. In this case, the reporting will be done automatically at the interval specified. With the Label Profile option, the user has the additional options for defining the data to be labeled(Station, Elevation, Both or None), the slope format and the vertical position of the text on the

grid. The **Label Orientation** choose between horizontal and vertical labels.

This command can also be used as a profile inspector. As you move the cursor around, the station, elevation and profile grade are displayed in a real-time window, unless you specify the more automatic "report at interval" method. If Prompt for snap is set on (available in non-interval mode), then when a point on the profile is picked, you have the opportunity to snap to an even 1, 5 or 10 stations.



Prompts

Station-Elevation-Slope Report Options dialog

Profile Settings dialog Check that these parameters match the grid.

Pick Lower Left Grid Conner <5177.48,5034.10>[end on]: *pick the corner* (endpoint snap is on)

Pick profile polyline: *pick the profile polyline*

Range of Stations > 0.0 - 312.43

Station> 0+00.00 **Elevation**> 364.00 **Slope**> 4.00%

Pick the vertical position for the text: *pick a point to place the text*

Station> 0+50.00 **Elevation**> 366.00 **Slope**> 4.00%

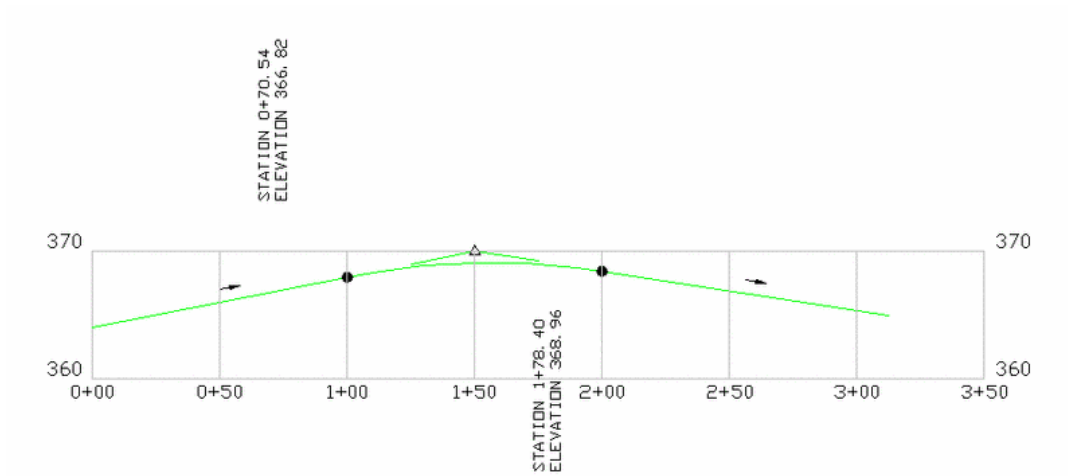
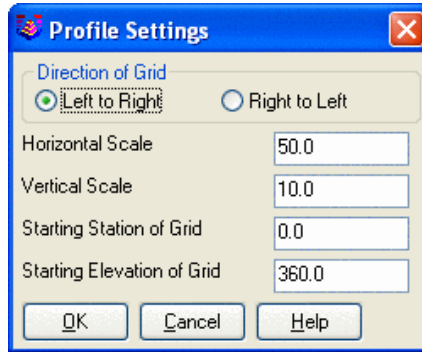
Station> 1+00.00 **Elevation**> 368.00 **Slope**> 4.00%

Station> 1+50.00 **Elevation**> 369.12 **Slope**> 0.50%

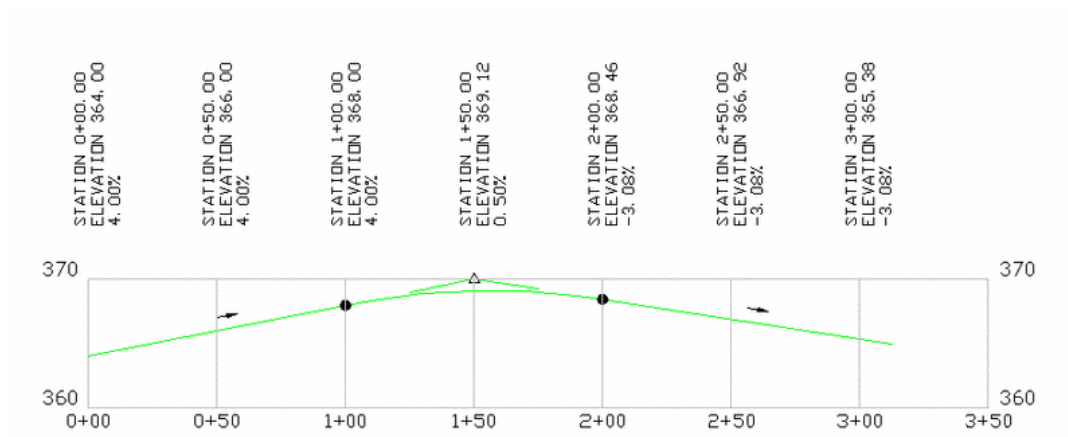
Station> 2+00.00 **Elevation**> 368.46 **Slope**> -3.08%

Station> 2+50.00 **Elevation**> 366.92 **Slope**> -3.08%

Station> 3+00.00 **Elevation**> 365.38 **Slope**> -3.08%



Picked method with Slope set to None



Interval method with Slope in Percent

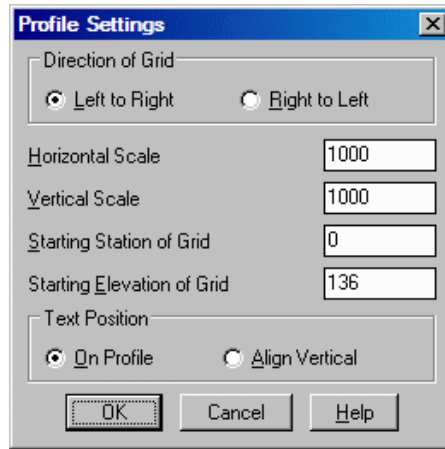
Pulldown Menu Location: Profiles

Keyboard Command: staelv

Prerequisite: Profile grid with profile polyline or .PRO file

Sag & Crest Report

This command will calculate the high and low point (sag and crest) on the vertical curves defined in the specified road (.PRO) profile file. Plotting the calculations in the drawing is optional. A profile grid must already be drawn to use the plotting option. The sag and crest are only labeled if the respective low and high points occur on a vertical curve.



Prompts

Report only/ <Plot calculations>: *press Enter*

Profile Settings dialog If you're using the plot option, make sure these parameters match your grid.

[end on]Pick Lower Left Grid Corner <0.00,0.00>: *pick this point*

Number of decimal Places <2>: *press Enter*

Sag & Crest Report

SAG Station > 3+71.80 Elevation > 1000.00

CREST Station > 9+40.20 Elevation > 1027.19

Pulldown Menu Location: Profiles

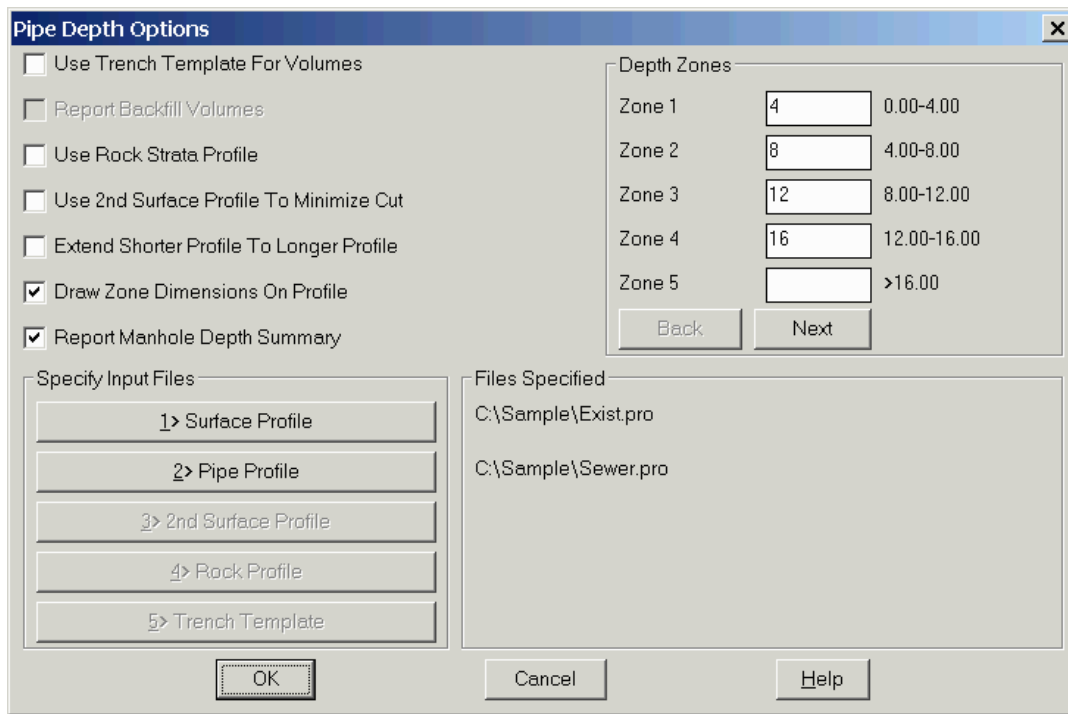
Keyboard Command: sagcrest

Prerequisite: A road profile

Pipe Depth Summary

This command reports the horizontal distances for the range of depths comparing a surface profile to a trench, pipe or sewer profile. There is an option to use two surface profiles and the program will use the minimum of the two depths. In addition to the report, the depth ranges can be labeled along the profile in the drawing.

The simplest of applications of this command, comparing a sewer profile to a surface profile and reporting the depth summary according to the specified Depth Zones, is shown below.



Use Trench Template for Volumes: Trench templates are made using the command Trench Template Editor within the Profile Utilities "flyout". Trench earthwork volumes are then computed.

Report Backfill Volumes: Available if trench templates is clicked on.

Use Rock Strata Profile: If clicked on, the Rock Profile can be entered in the lower portion of the dialog, and if the pipe invert is below rock surfaces along any segment, rock linear feet will be reported, in the same depth categories as used for trench depths. In the example shown below, if rock depth is uniformly 5 feet below surface elevation, in the form of a rock profile, rock quantities are 348 feet of 0-2 feet depth of rock trenching.

Use 2nd Surface Profile to Minimize Cut: If the final grade is below existing grade, in those areas, it saves trenching work to first do the cut to final grade, prior to filling over existing grade in areas of fill. Then trench depths are minimized. This option, if clicked on, computes trench depths to the minimum of the two specified surfaces, and activates the 2nd Surface Profile option in the lower portion of the dialog.

Extend Shorter Profile to Longer Profile: This option will extrapolate the starting and ending stations of the shorter profile to match the longer profile.

Draw Zone Dimensions on Profile: The depth zones will be annotated along the horizontal axis of a profile drawing with this option.

Report Manhole Depth Summary: This leads to the depth summary report.

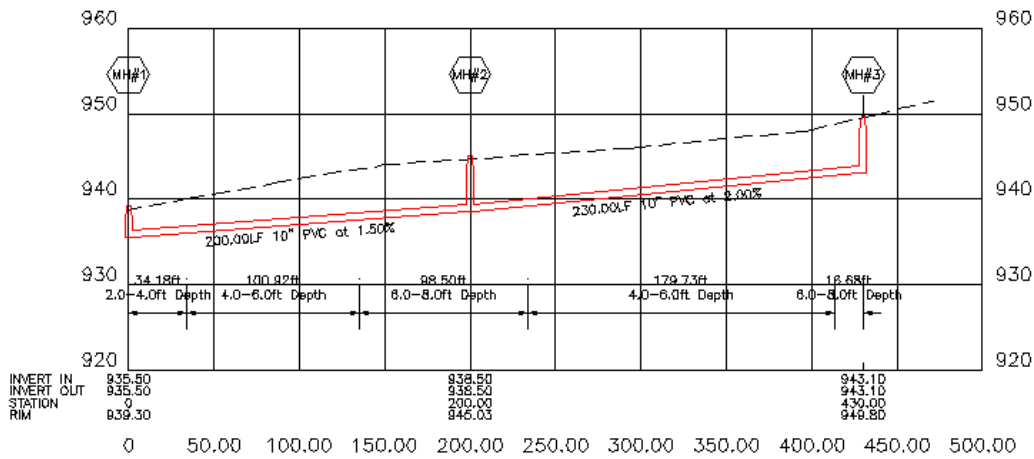
Depth Zones: These zones are for reporting the pipe range of depth. The depths should be entered in lowest to highest order. Use the Next and Back buttons to move between the 20 possible depth values.

Prompts

Pipe Depth Options dialog

Pick lower left grid corner [int on]: *pick the profile grid corner*

Pick vertical position for dimensions: *pick a point below the profile grid*



Pipe Depth Summary

Surface Profile: C:\SCAD2005\DATA\SURFACE.PRO

Pipe profile: C:\SCAD2005\DATA\SEWER.PRO

Depth	Manholes	Linear Ft	% of Total
0-2.0	0	0.0	0.0
2.0-4.0	1	34.2	7.9
4.0-6.0	0	280.6	65.3
6.0-8.0	2	115.2	26.8
8.0-10.0	0	0.0	0.0
>10.0	0	0.0	0.0
Total	3	430.0	

Manhole	Depth
MH#3	6.45
MH#2	6.21
MH#1	3.28
Total:	15.94

Pulldown Menu Location: Profiles

Keyboard Command: pipedeeep

Prerequisite: Two profiles, one for the surface and one for the pipe invert elevation

Label Profile On Centerline

This command labels a road profile on a centerline in the plan view. First you are asked to select a road profile and a centerline. Then on the Label Profile On Centerline Settings dialog, you need to choose which labels to draw. The left list shows the available labels to draw, the right list shows the label that have been chosen to draw. **Add** button moves the highlighted label on the left list to the right list, **Remove** button moves the highlighted label on the right list to the left list, and **Setup** button allows you to edit the options of how to display the highlighted label in the plan view.

The available labels include:

PVC, PVI, PVT, High and Low Points: To label the station and/or elevation of the profile at these points.

Slope: To label each different slope in the profile.

Station At Interval: To label the stations along the centerline at a station interval.

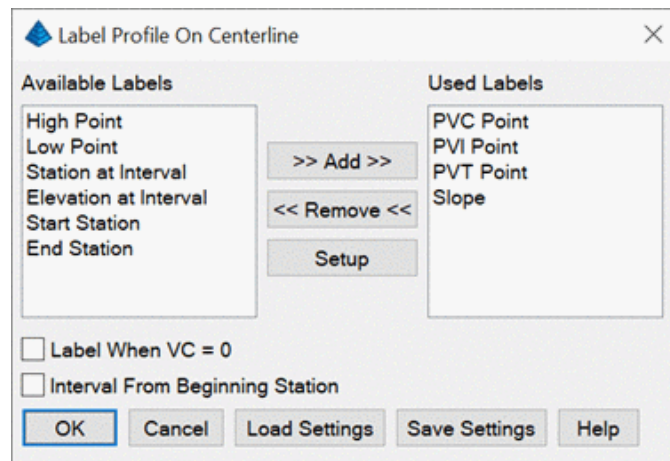
Elevation At Interval: To label the profile elevations at a station interval.

Start Station: To label the profile starting station.

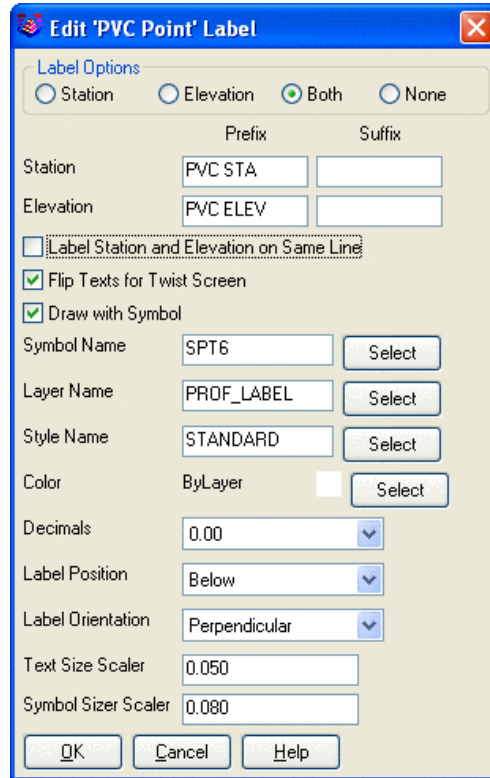
End Station: To label the profile ending station.

On the point label edit dialog, you may choose to display either the station or elevation, or both, or neither of them, and specify their prefix or suffix. If you choose to label both the station and the elevation, the labels are drawn in two lines by default, unless you elect the option to **Label Station and Elevation on Same Line**. When the screen is twisted, some of the labels would possibly be drawn upside down, the **Flip Texts for Twist Screen** option flips the text to the proper direction. **Draw with Symbol** setting draws a symbol at the position of the label. Next, you would select from a list the **Symbol Name**, **Layer Name**, **Style Name** and **Color** for displaying the label. The **Label Position** can be set to **Above** or **Below** the centerline or to the **Left** or **Right** of the centerline. There are three **Label Orientation** settings: **Horizontal**, **Parallel** and **Perpendicular**. **Text Size Scaler** and **Symbol Size Scaler** determine the size of the text label and symbol label respectively in plan view. The **Interval From Beginning Station** setting controls whether the station interval increments from the starting station of the centerline or from zero. The **Label When VC = 0** option will label profile stations that have a vertical curve of zero.

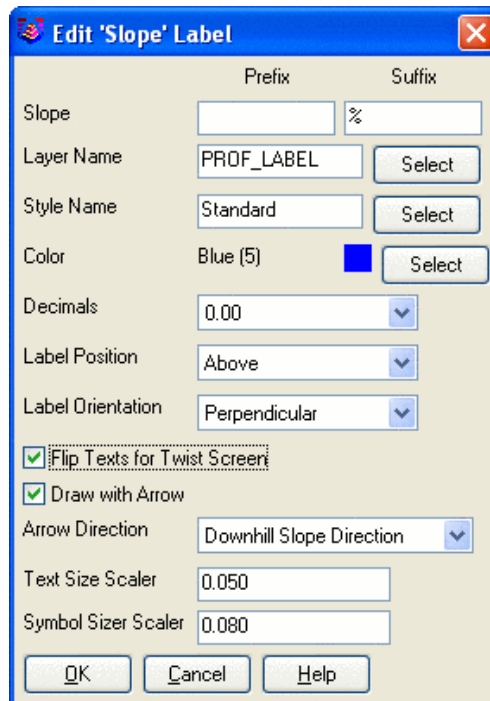
The slope label edit dialog is very similar to the point label edit dialog, the big difference is the **Arrow Direction**. There are four arrow directions: **Centerline Direction**, **Uphill Slope Direction**, **Downhill Slope Direction** and **Away from PVI**.



Label profile On Centerline Settings Dialog

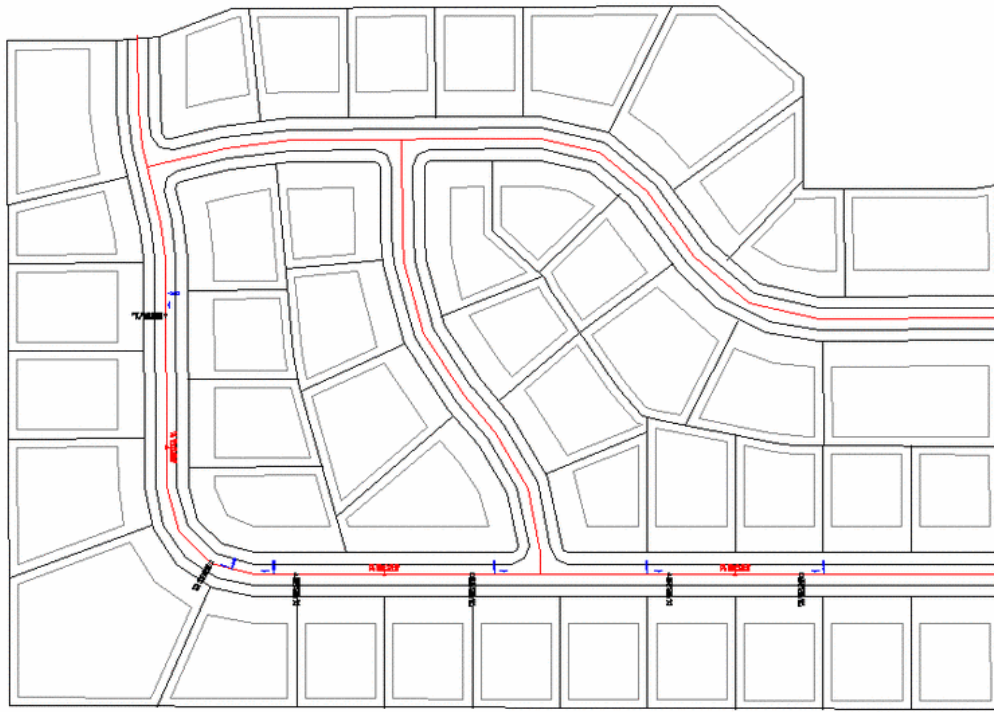


An example dialog for editing PVC Point Label

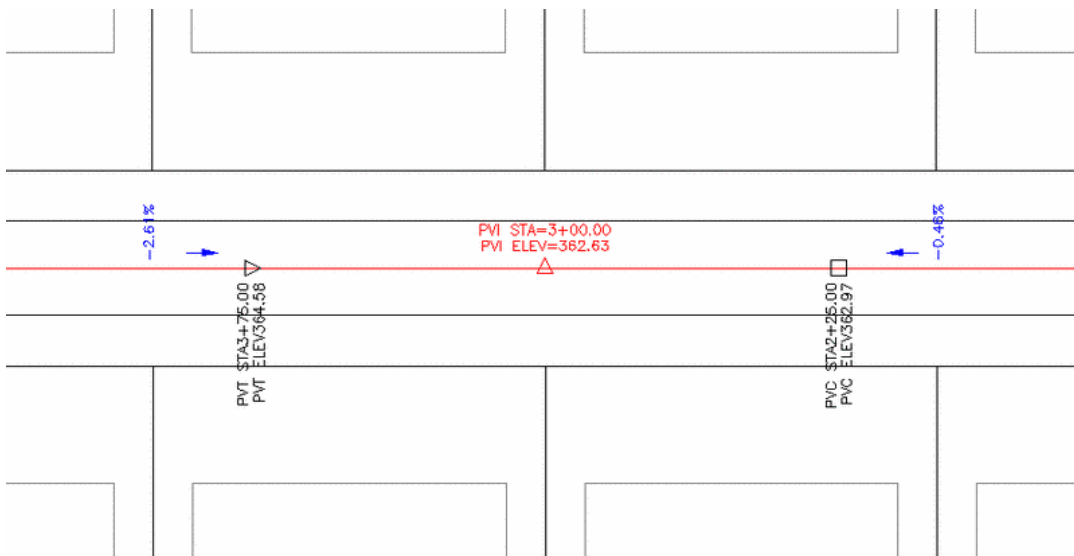


An example dialog for editing Slope Label

Here's a site design example shown below, where a profile is labeled on the centerline. The PVI labels are above and parallel to the centerline, the PVC and PVT labels are below and perpendicular to the centerline, and the slopes are above and perpendicular to the centerline, their arrows are in the downhill slope direction.

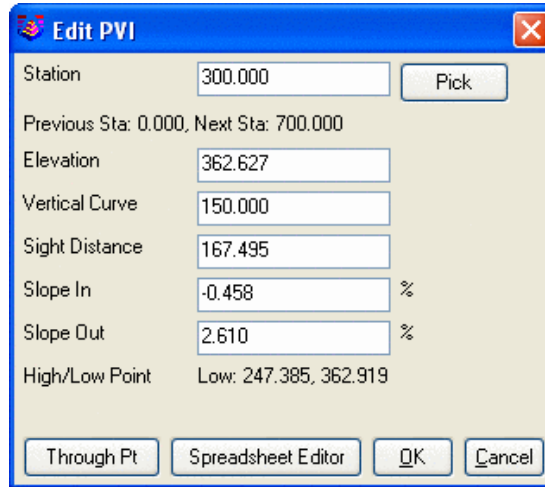


Site Design Example

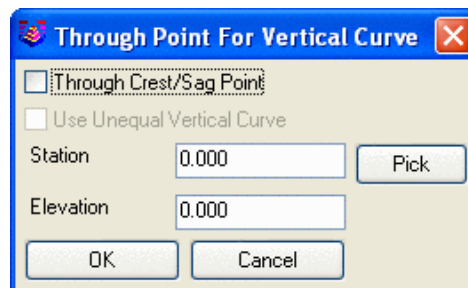


A closer look of profile labels

If the road profile is modified by a Carlson program, the profile labels will be updated automatically. When you double-click on one of the labels, a dialog of the labeled PVI section will pop up for editing. In the sample dialog show below, you can modify the station, elevation, and other parameters of the PVI point. **Pick** button allows you to pick on the screen to get the station. **Through Pt** button opens a dialog where you can define a point on the profile to pass through the fixed station and elevation. When click on **Spreadsheet Editor** button, the road profile editor is opened for you to edit road profile.



Edit PVI Dialog



Through Point Dialog

Pulldown Menu Location: Profiles > Label Profile On Centerline

Keyboard Command: proflabel

Prerequisite: a road profile file (.PRO), a centerline in the plan view or a centerline file (.CL)

Profile ID

This command reports the profile file name, horizontal scale and vertical scale that was used to draw the selected drawing entity. Simply pick a profile entity in the drawing and the profile file name is reported in the command text window. The profile must be drawn in Carlson by completing the Draw Profile command (without aborting the command by pressing Esc).

Prompts

Select profile entity to identify: *pick an entity*

Horizontal Scale: 50.0 **Vertical Scale:** 10.0

Profile Name: sewer.pro

Select profile entity to identify: *press Enter to end*

Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: profid

Prerequisite: Profiles drawn on the screen

Review Profile Links

This command shows a list of all the profile links that the program knows about in the current drawing. These links are between the profile files and the drawn profiles in the drawing. You can use the Remove button to remove links for any obsolete profiles or if you don't want to link a certain profile.

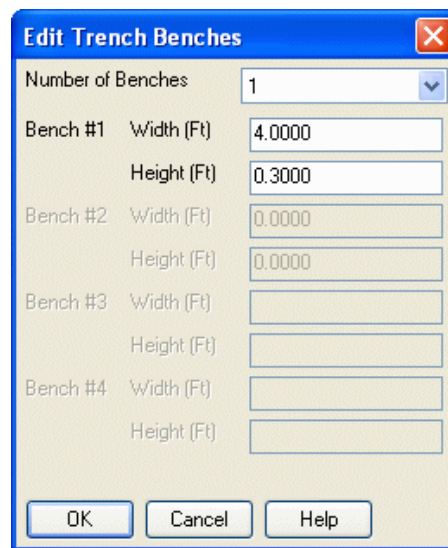
Pulldown Menu Location: Profile->Profile Utilities

Keyboard Command: profdict

Prerequisite: None

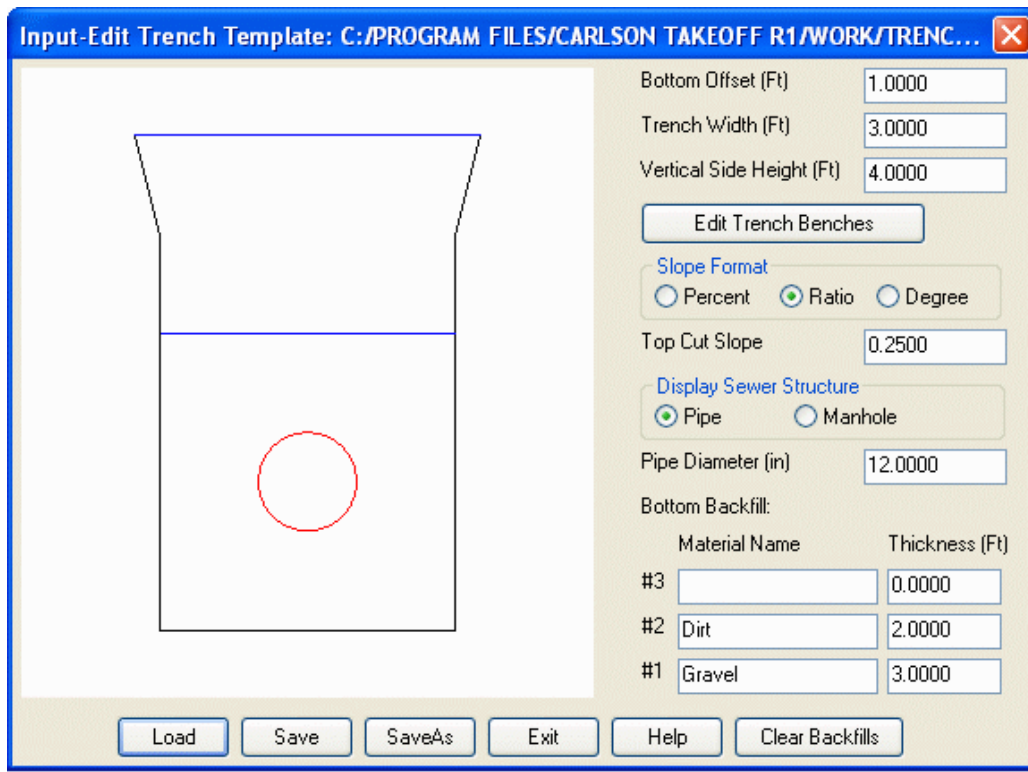
Trench Template Editor

This command lets you create a new trench template or modify an existing trench template. It prompts you the **Trench Template Editor Dialog**. If you are modifying a trench template, click the **Load** button on the dialog to open a trench template file and display the template data on the dialog. Enter the dimensions of the trench: bottom offset, trench width and vertical side height. The Edit Trench Benches button will bring up the below dialog, and allows you to enter in up to four benches into your trench.



Number of Benches		1
Bench #1	Width (Ft)	4.0000
	Height (Ft)	0.3000
Bench #2	Width (Ft)	0.0000
	Height (Ft)	0.0000
Bench #3	Width (Ft)	
	Height (Ft)	
Bench #4	Width (Ft)	
	Height (Ft)	

There are three methods for entering the cut slope, Percent, Ratio and Degree. Choose one of the methods and enter the slope value. There are three trench bottom backfill layers that can be defined. Enter the layer label in the material name field, the depth of the layer in the thickness field. Click Save or SaveAs to save the template information in a .tch file, and Click Exit to quit this command.



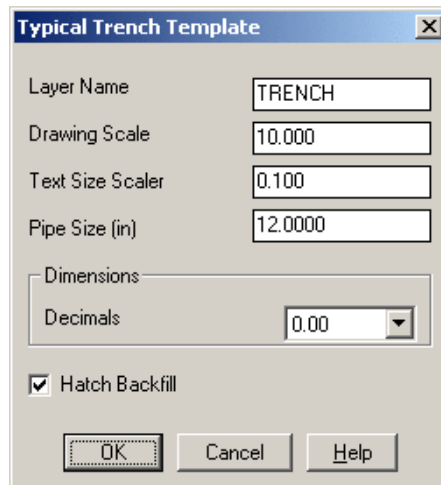
Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: make_trench_tpl

Prerequisite: None

Draw Typical Trench Template

This command draws a trench template on the screen. After you select a trench template file (.tch) to draw, a **Typical Trench Template Dialog** is prompted for entering the layer name, drawing scale, text size scalar, pipe size and selecting how many decimal points you want. You can also hatch the backfill on the drawing. Click OK to draw the template at the position that you pick on the screen.



Prompts

Pick position to draw template: *pick a position on the screen*

Pulldown Menu Location: Profiles > Profile Utilities

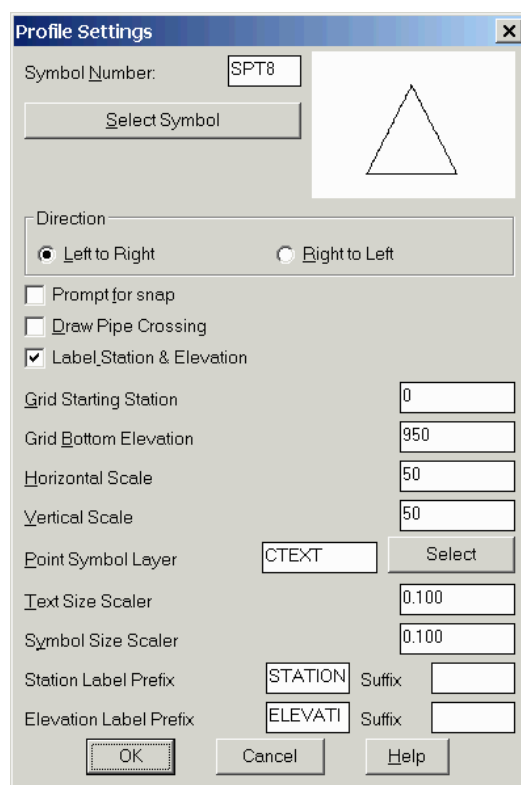
Keyboard Command: draw_trench_tpl

Prerequisite: None

Point Placement on Profile

This command has two methods for placing points on a profile. One method places symbols on an existing profile at picked points or at entered stations and elevations. The station and elevation of the current position of the crosshairs is displayed in the lower right of the screen menu. The symbols can be any of the point symbols or a special pipe crossing circle that will become an ellipse to represent any vertical exaggeration.

The other method will draw an entirely new profile based on points that are defined in a coordinate (.CRD) file. The elevations of the profile come from the elevation of the points and the stations come from the station value in the description field of the point. Points without the station value in the description field and points with a zero elevation are ignored. The station text in the point descriptions can be generated with the *Calculate Offsets* command in the Centerline menu. Using SurvCE, the Carlson data collection program, you can gather points in the field and store their station as the beginning of the description, and these points would then plot as a profile using this command.



Prompts

Place points from CRD file or pick points (File/<Points>)? *pick*

Profile Settings dialog

Pick lower left grid corner <5000.0,5000.0>: *pick the grid corner*

Enter station or pick a point (Enter to End): *75*

Elevation of point: *565*

For CRD File option:

Select CoorD File to Read

This is the source file that contains the profile information.

Range of Point Numbers to use (A for All) <A>: *press Enter*

Wildcard match of point description <*>: *press Enter*

Plot Full or Abbreviated text (Full/<Abbrev)? *Full*

Range of stations: <134.41 - 938.31>

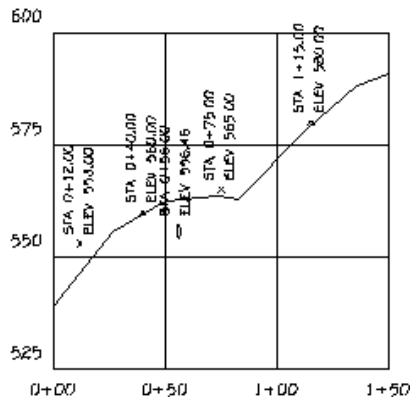
Starting Station <134.41>: *press Enter*

Ending Station <938.31>: *press Enter*

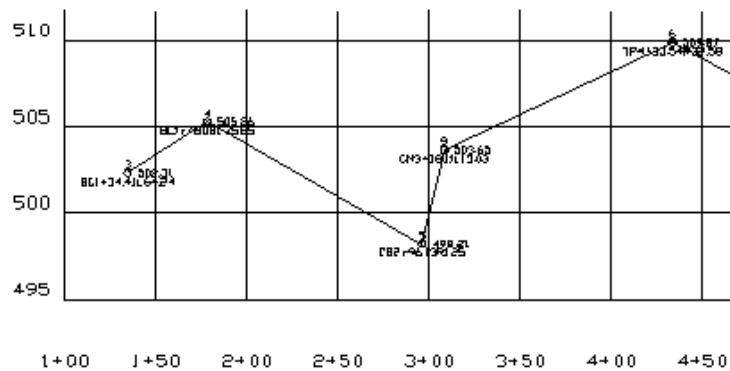
Profiles Settings Dialog

Starting/Datum Elevation of Profile <495.0>: *press Enter*

Pick Starting Point For Axis <100.0,495.0>: *press Enter*



Points placed on profile with a pipe crossing at station 56



Grid, profile, and points drawn by Point Placement on Profile

Sample contents of .CRD file:

- 3, 4368.47, 4250.26, 502.31, BC1+34.41L64.24
- 4, 4406.95, 4273.95, 505.26, BC1+78.08L75.85
- 5, 4427.67, 4416.11, 498.21, CB2+96.13R1.25
- 6, 4436.55, 4549.39, 509.87, TP4+33.54R32.58
- 7, 4566.77, 4795.20, 515.50, MH7+07.05L55.04
- 8, 4572.69, 4996.60, 520.14, MH8+88.55R43.18
- 9, 4446.17, 4419.49, 503.65, CM3+08.11L13.03
- 10, 4506.57, 4814.72, 505.00, EPL7+04.50R8.18

Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: *ptpro*

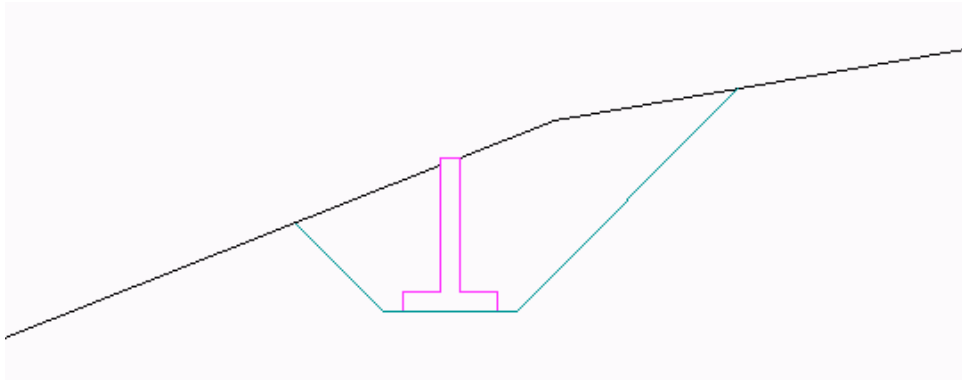
Prerequisite: A .CRD file including points with elevations and station information in the description field

Restricted Grade Design

This command draws a retaining wall symbol in profile view and draws the excavation tie slopes from the base of the wall to the ground profile.

The program first prompts to select the retaining wall section symbol. To add custom symbols, create a wallXX.dwg in the Carlson SUP folder where XX is the wall number. For example, create a drawing wall3.dwg in the %appdata%\Carlson Software\Carlson2014\r18.2_x64\sup folder. This dwg should have one polyline drawn left to right to outline the wall section. The polyline should be open along the bottom. The wall1.dwg and wall2.dwg are examples of how to draw the wall section polyline.

After selecting the wall section, the program prompts for the tie slope and the base placement offset which is the offset from the base of the retaining wall to the bottom of the tie slope. Next there are prompts for the horizontal and vertical scales of the profiles which set the vertical exaggeration for the drawing. Then the program prompts for the insertion point for the base of the wall. Finally there is a prompt for selecting a ground polyline on the profile to intersect with the tie slopes.



Prompts

Tie slope ratio <1.000>: *press Enter*

Base placement offset <1.000>: *press Enter*

Horizontal Scale <50>: *press Enter*

Vertical Scale <50>: *press Enter*

Pick insertion point: *pick a point*

Select ground polyline (ENTER for None): *pick polyline*

Total Wall Area: 12.00, Above Ground: 0.22, Below Ground: 11.78

Do another retaining wall [*<Yes>/No*]? *No*

Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: rwallsym

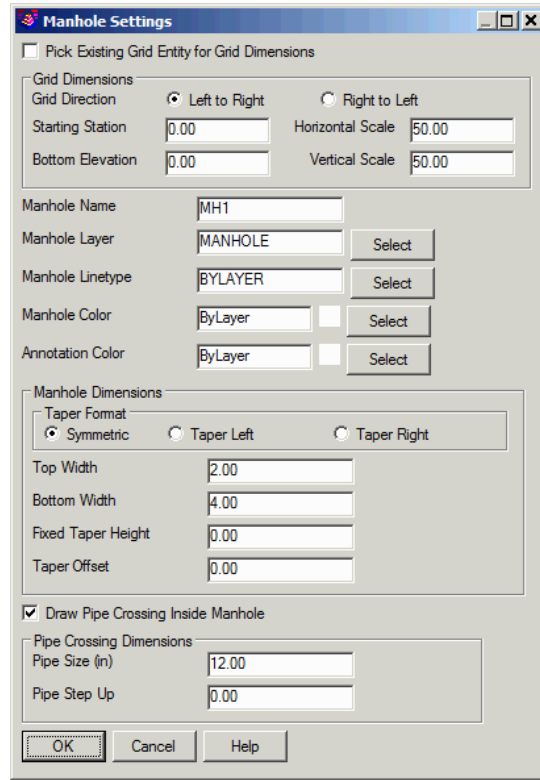
Prerequisite: None

Draw Single Manhole

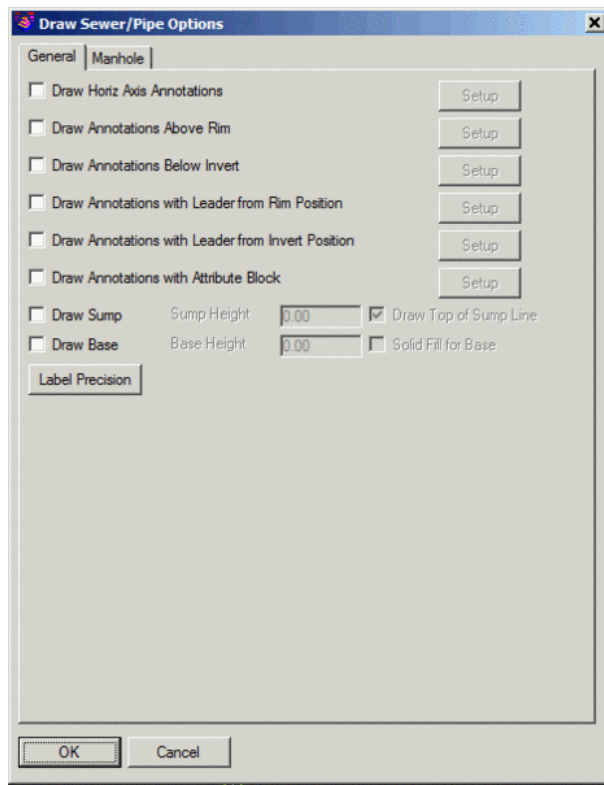
This command draws and labels a single manhole on a profile grid from user selected points at the desired bottom and top locations of the manhole. The user specifies the horizontal and vertical scales, layer and drop across manhole. Other options include the manhole top and bottom width, top taper offset and the fixed taper height. The Top Taper

Offset sets the distance from the top of the manhole to the point that the taper will end. A 2-foot offset on a 7-foot manhole is shown below. The Fixed Taper Height determines the overall length of the tapered section.

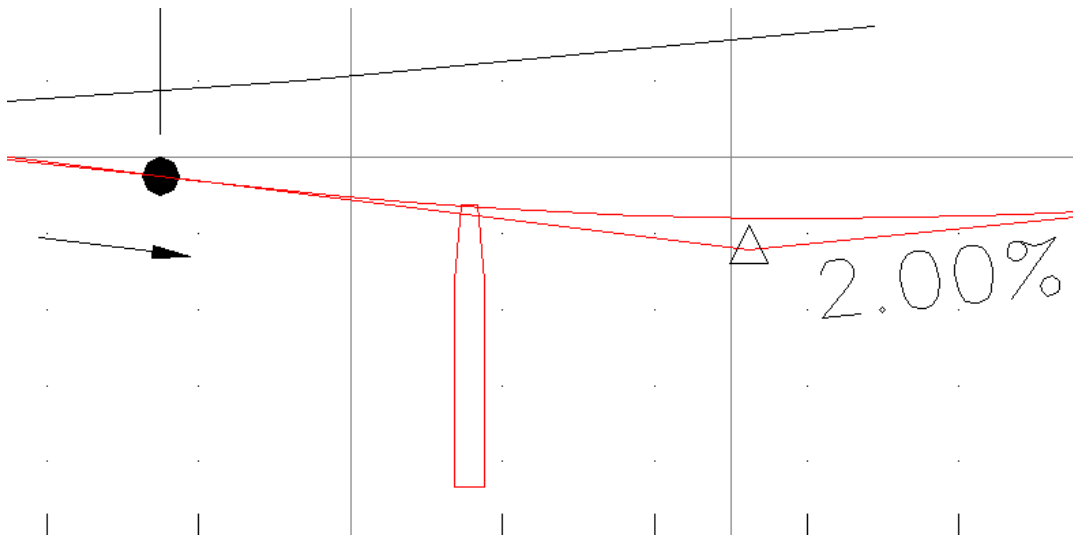
The manhole station, manhole elevation and rim elevation can be screen picked points on the profile or these values can be typed in at the command line. When using screen picks, the program shows the values of the picked positions in a dialog which allows for rounding or editing of the values.



The first dialog has the settings for the profile grid and the manhole parameters.



The second dialog has the label settings. These settings are the same as Draw Profile. See the Draw Profile topic in the manual for information on these settings.



Prompts

Manhole Settings dialog

Pick Lower Left Grid Corner <0.00,0.00>[end on]: *pick corner of profile grid*

Draw Options dialog

Enter manhole station or pick a point (Enter to End): *pick the invert elevation point*

Enter manhole rim elevation or pick a point (Enter to End): *pick the surface point*

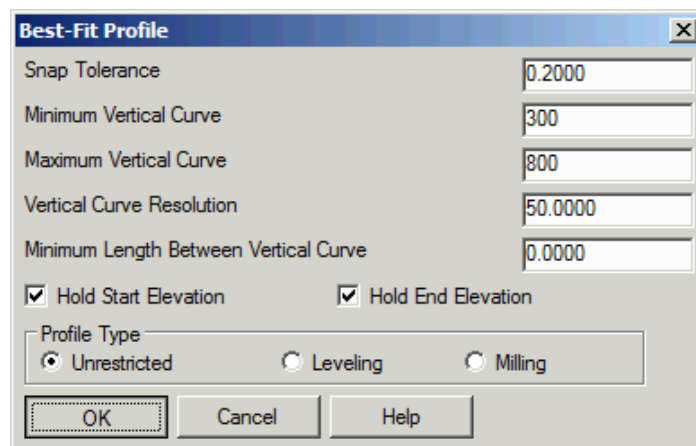
Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: manhole

Prerequisite: Profile grid in drawing

Best Fit Profile

This command processes an input profile that has data points at an interval and creates a best-fit profile with tangents and vertical curves. Each tangent segment in the profile is calculated by the best-fit line least-squares method and each vertical curve is determined by calculating vertical curve lengths between the specified Minimum and Maximum Vertical Curves at the Vertical Curve Resolution and choosing the length with the smallest residuals. The input profile represents the surveyed of the existing profile. One method to create this input profile from survey point data is to use Triangulate & Contour to make a TIN surface from the points and then use Create Profile From Triangulation Surface.

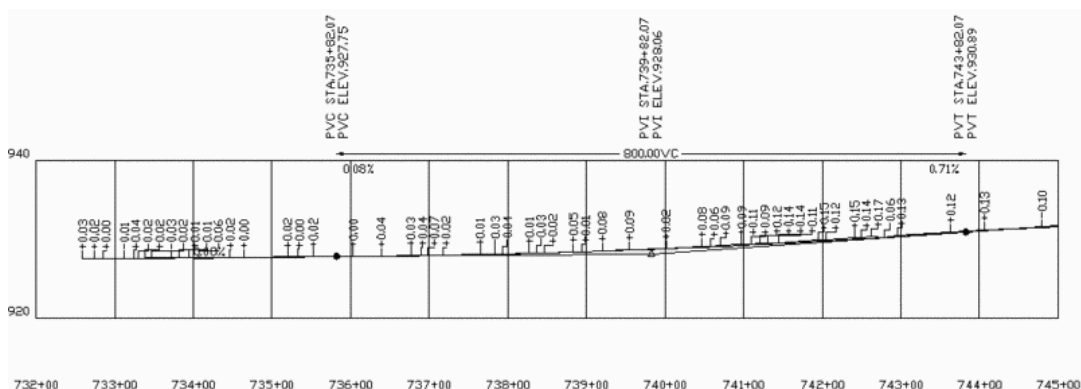


In the process options dialog, the Snap Tolerance is the max offset from the point to the profile which is used for finding the best-fit tangent segments. The Minimum and Maximum Vertical Curve values control the range of possible vertical curve lengths. The Vertical Curve Resolution is used to round the resulting length to this value. The Hold Start and End Elevation options keep the original profile elevations for matching an existing road. For Profile Type, Leveling keeps the best-fit profile from going any lower than the original profile, and Milling keeps the best-fit profile from going any higher than the original profile.

The residual for each point is the elevation difference from the point to the best-fit profile. The results are shown in a dialog and you can toggle each point for whether to include in the calculations. Points that are toggled off are not used for calculating the profile but are still used in the residual report. The Remove function removes the point from both calculation and residual reporting. After picking OK on the results dialog, the program prompts for the profile to create with the best-fit results.

Station	Elevation	Residual	Process
732+58.955	927.519	0.026	Yes
732+74.094	927.528	0.023	Yes
732+84.984	927.509	-0.005	Yes
733+12.021	927.528	-0.006	Yes
733+24.312	927.579	0.035	Yes
733+30.257	927.525	-0.024	Yes
733+38.175	927.570	0.015	Yes
733+47.014	927.534	-0.028	Yes
733+72.478	927.558	-0.024	Yes
733+82.343	927.599	0.009	Yes
733+94.652	927.591	-0.009	Yes

This example Draw Profile shows the input profile and the best-fit profile along with elevation difference labels between the input profile break points and the best-fit profile.



Prompts

Select Profile to Process Pick the input profile

Best-Fit Profile Options Dialog Set processing options

Best-Fit Profile Results Dialog Review results and toggle points on/off

Select Profile to Write Specify the output profile

Pulldown Menu Location: Profiles->Profile Utilities

Keyboard Command: bestpro

Prerequisite: Profile to process

Merge Profiles

This command combines a range of stations of one profile and a range of stations of a second profile. The stations and elevations in these two ranges can be stored in new file or overwrite an existing profile. Both profiles must be the same type: generic, road, pipe, or sewer.

Prompts

First Profile to Merge Select a profile.

Second Profile to Merge Select a profile.

Range of first profile stations to use <0.0 - 400.0>: *press Enter*

Range of second profile stations to use <400.0 - 800.0>: *press Enter*

Profile file to Save dialog box

Ranges can overlap, as shown below:

Range of first profile stations to use <0.000 - 471.214>:

Range of second profile stations to use <450.000 - 480.000>:

In the case of overlap, all non-matching stations and elevations in the two sets of profile ranges will be used in the final profile. If matching stations are found, the elevations of the first and second profiles will be averaged.

Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: mergepro

Prerequisite: Two profiles

Average Profiles

This command averages up to four profiles and stores the resulting profile into a user-specified file name. Profiles that don't share the exact profile range will be projected to match the low and high stations in the selected profiles, after which the averaging will be computed.

Prompts

1st Profile file to Average dialog Specify a profile file.

2nd Profile file to Average dialog Specify a profile file.

3rd Profile file to Average dialog Hit Cancel to stop selecting profiles.

Choose Profile to Write Specify a profile file.

Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: avgpro

Prerequisite: Two or more profile files

Draw Pipe 3D Polyline

This command creates a 3D polyline that represents a pipe. The points can be either picked on screen or specified by point number in the current coordinate file. This command is a convenient way to make 3D polylines that can become "pipe polylines" used for capturing their profile positions, leading to circular or elliptical or even square plots of the pipes or culverts within Draw Profile. However, this command is not required nor sufficient to make a pipe polyline useful in the *Draw Profile* command. Pipe polylines are made only by converting 3D polylines into pipe polylines using the adjacent command, *Assign Pipe Width to Pline*.

Prompts

Layer Name for 3DPoly <PIPE>: *press Enter*

Prompt for elevations (.XY filter) (Yes/<No>)? *Y for yes*

Undo/<Pick point or point numbers>: *pick a point*

Elevation <0.0>: *554.12*

Undo/<Pick point or point numbers>: *pick a point*

Percent slope/Ratio slope/Elevation <0.0>: *553.72*

Undo/Close/<Pick point or point numbers>: *press Enter*

Draw another 3D polyline (Yes/<No>)? *press Enter*

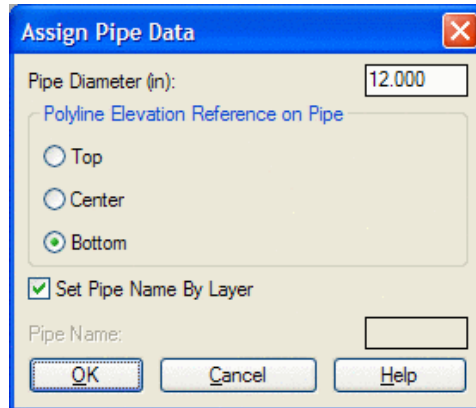
Pulldown Menu Location: Profile->Profile Utilities

Keyboard Command: drwpipe

Prerequisite: None

Assign Pipe Width to Polyline

This command attaches a pipe width to one or more polylines. Any polyline can be used, but it should be a 3D polyline that represents the elevations of the pipe. Pipe width is used in commands such as *Profile from Pipe Polylines* and *Section Points from Pipes* commands.



Prompts

Select polyline: *pick a polyline*

Enter pipe width (in): 18

Set pipe width for 1 polylines.

Select polyline (Enter to End): *press Enter*

Pulldown Menu Location: Profile->Profile Utilities

Keyboard Command: plwidth

Prerequisite: A polyline

Identify Pipe Polylines

This command reports pipe data tagged to polylines by the Draw Pipe 3D Polyline or Assign Pipe Data To Polyline commands. There are two methods. The Pick method prompts to select a polyline and reports its data. The Search method checks all the polylines in the current drawing and reports all the pipe data from the drawing and highlights the pipe polylines.

Prompts

Pick polylines to check or search drawing [<Pick>/Search]? *press Enter*

Select pipe polyline: *pick a polyline*

Polyline is set as pipe. Size: 12.0" Reference: Bottom

Select pipe polyline (Enter to end): *press Enter*

Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: id_pl_pipe

Prerequisite: Pipe polylines

Label Pipe Polylines

This command labels pipe values in plan view for 3D polylines that are tagged with pipe data. See the Plan View Label Settings command in Hydrology > Network for information on the label settings.

	Position	Prefix	Suffix	Sequence
<input checked="" type="checkbox"/> Label Pipe Length	Above Pipe			1
<input checked="" type="checkbox"/> Label Pipe Size	Below Pipe		"	1
<input checked="" type="checkbox"/> Label Pipe Slope	Above Pipe		%	1
<input checked="" type="checkbox"/> Label Pipe Name	Above Pipe			1

Slope Format
 Percent Ft/Ft

Pipe Direction Label: Arrow On Pipe

Slope Decimals: 0.00 Length Decimals: 0.00

Text Size Scaler: 0.100 Arrow Size Scaler: 0.100

Label Layer: SWRNET Set Label Style: ROMANS Set

OK Cancel Load Save Help

Prompts

Label Pipe dialog

Select pipe lines to label.

Select objects: pick 3D polylines with pipe data

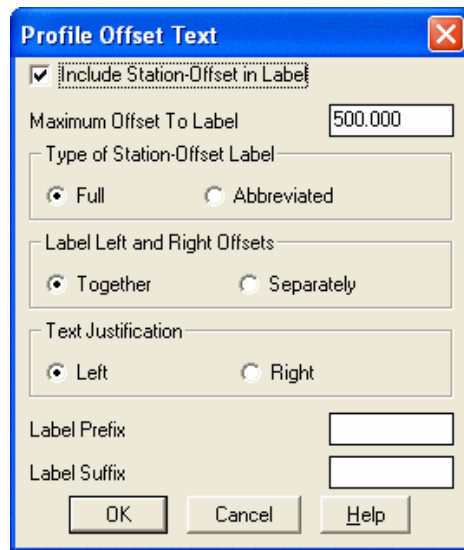
Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: labpipe

Prerequisite: 3D polyline with pipe data

Profile Offset Text

This command draws station/offset and description text for points along a centerline polyline at a picked vertical position on the screen. It works well when used on combined Plan and Profile sheets, where the offset text can be plotted in the profile portion. The text is drawn vertically and is positioned horizontally at the station of the centerline. The station and offset of the point can optionally be included in the text. The points can either be picked or specified by point number. After picking the point, a text editor allows you to type in additional text for the label. For centerlines that are not roughly East-West, use Twist Screen under the View pulldown to re-orient the centerline to a near horizontal position on the screen. Only the "pick point or point number" option will display the edit box for the description.



Include station-offset in label: When clicked on, the calculated station and offset text is plotted.

Full or Abbreviated: The abbreviated form leaves off the even 100 feet in front of the stationing, and saves some space. Station 14+50.23 would plot as +50.23.

Label Left and Right Offsets (Together or Separately): The "Separately" option will ask for a horizontal alignment point for left and another for right offsets. Otherwise offsets will be labeled along one horizontal alignment based on one pick.

Text justification (Left or Right): Left plots down the screen and right will plot up the screen.

Label Prefix: Will place this prefix in front of the station and offset or entered text for the picked position. For example, the word "Sta." could be added as a prefix, leading to a plot such as Sta. 14+50.23.

Label Suffix: Will append this suffix to all text for each picked position.

Prompts

Profile Offset Text Dialog *make choices, click OK*

Polyline should have been drawn in direction of increasing stations.

CL File/<Select polyline that represents centerline>: *pick the centerline*

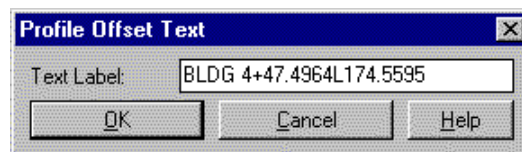
Starting station of centerline <0.0>: *press Enter*

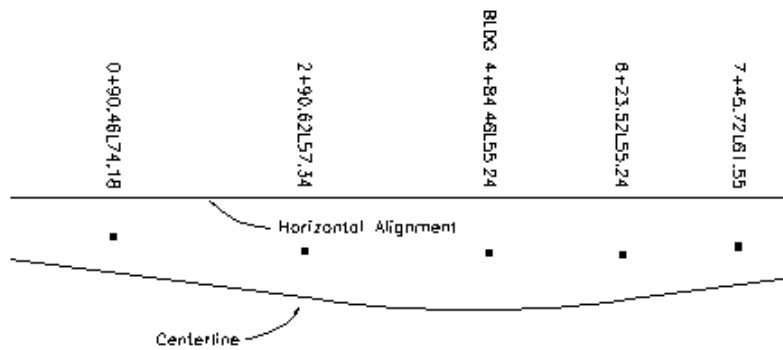
Pick horizontal alignment for text: *pick a point*

Pick point or point number (SS for Selection Set, Enter to End): *pick a point*

Profile Offset Text dialog

Pick point or point number (Enter to End): *press Enter*





Profile Offset Text along a centerline

Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: protext

Prerequisite: A centerline polyline

Label Profile Differentials

This command labels the elevation difference between two profiles at specified stations. The text label is drawn along the polyline centerline in plan view. Cut is labeled as negative and fill as positive.

Prompts

Existing Ground Profile dialog Specify a profile file to read.

Final Surface Profile dialog Specify an existing profile file to read.

Select centerline polyline: Pick a polyline. This represents the centerline and should be drawn in the direction of increasing stations.

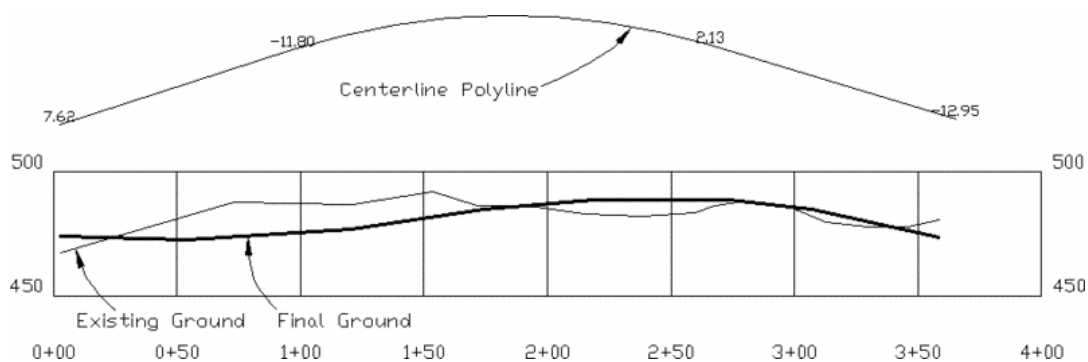
Starting station of polyline <0.0>: *press Enter*

Label all polyline vertices (<Yes>/<No>)? *press Enter* This option will label the elevation difference at the stations of each point on the polyline centerline.

Pick points to label (Yes/<No>)? *press Enter* This option allows you to pick points along the centerline to label.

Prompt for text position (Yes/<No>)? *press Enter* This option allows you to pick the position of each elevation difference label. Otherwise the text is automatically centered at the point on the polyline.

Text size <4.00>: *press Enter*



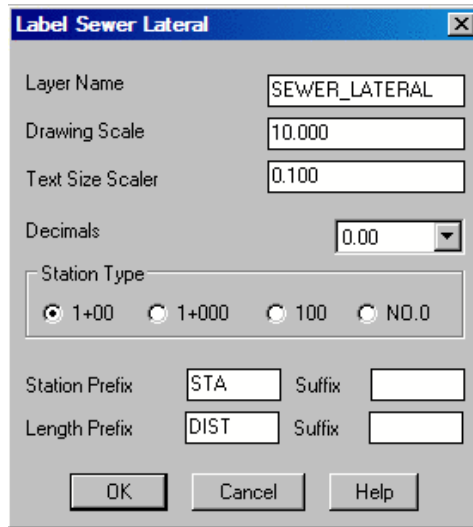
Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: prodiff

Prerequisite: Two profile files

Label Sewer Laterals

This is a command to label, in plan view, the sewer laterals in linear feet. This includes a station distance from a known starting station on the main line, as well as the length of the lateral itself from the main sewer line to the property. You may optionally include a prefix and/or suffix for both the station and lateral labels.



Prompts

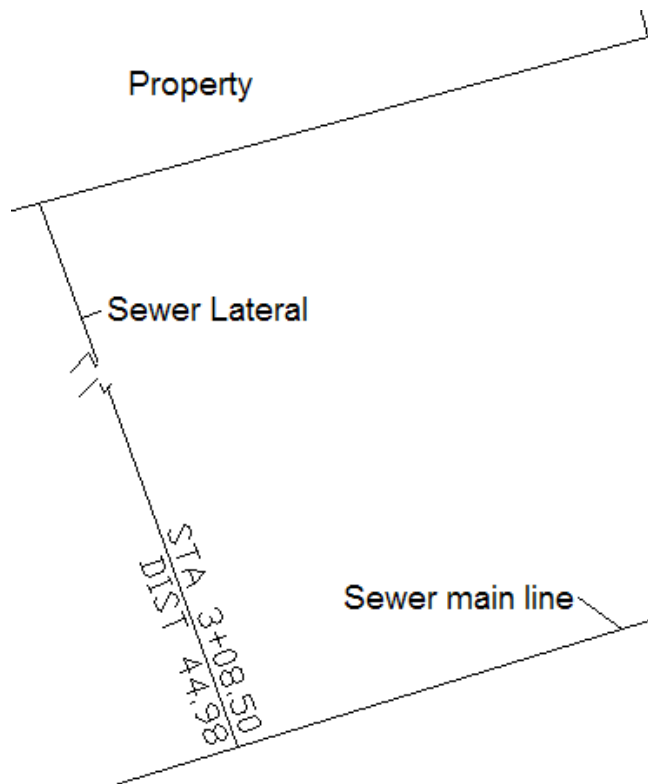
Label Sewer Lateral dialog Specify your preferred values.

Pick centerline/polyline that represents a sewer: *pick an entity*

Starting Station of the sewer <0.0>: *press Enter*

Pick a lateral intersection point (Enter to end): *pick an intersection point*

Pick a lateral to label: *pick entity*



Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: label_sewer_lateral

Prerequisite: Sewer line with lateral(s)

International Roughness Index

This command calculates and reports the international roughness index for a profile. The command starts by prompting for the profile .PRO file.

The dialog box 'Calculate International Roughness Index' contains the following fields and buttons:

- Profile: C:\Carlson\Work\Manatts\E-49 Tama.pro (Set button)
- Reference CL: C:\Carlson\work\Manatts\E-49 Tama CL.cl (Set button)
- Profile Offset: 3 ft (Draw button)
- Starting Station: 1407+50.00
- Ending Station: 1688+00.00
- Overall IRI: 6.24 in/mi
- Overall Length: 27842.13 ft
- Interval: 528.00 ft (Calculate button)

Starting Station	Ending Station	IRI (in/mi)	Classification
1407+50.00	1412+78.00	7.76	Extreme
1412+78.00	1418+06.00	7.85	Extreme
1418+06.00	1423+34.00	3.43	Rough
1423+34.00	1428+62.00	17.20	Extreme
1428+62.00	1433+90.00	15.12	Extreme
1433+90.00	1439+18.00	8.42	Extreme
1439+18.00	1444+46.00	0.98	Medium
1444+46.00	1449+74.00	2.53	Rough
1449+74.00	1455+02.00	8.91	Extreme
1455+02.00	1460+30.00	2.81	Rough
1460+30.00	1465+58.00	4.89	Rough

Buttons: Setup Classification, Report, OK, Cancel, Help

The command opens to the Calculate International Roughness Index dialog and is initially set to the Starting and Ending Station values of the selected profile .PRO file. The profile .PRO file can be edited with the edit control or selected by pressing the Set button. The Starting and Ending Stations can be adjusted to isolate a specific area of interest along the profile. The IRI is calculated and output for the length of profile's station range. An Interval can be applied to calculate the IRI on an interval along the profile's station range. Use the Interval toggle to supply the interval distance and then press the Calculate button to run the calculation on the interval.

Additionally, a Reference Centerline can be set by pressing the Reference CL Set button. When a reference centerline is supplied, there is additional ability to output the International Roughness Index classifications to the drawing. Use the Profile Offset to supply the track that the profile is offset from the centerline and press the Draw button to output the classifications to the drawing.

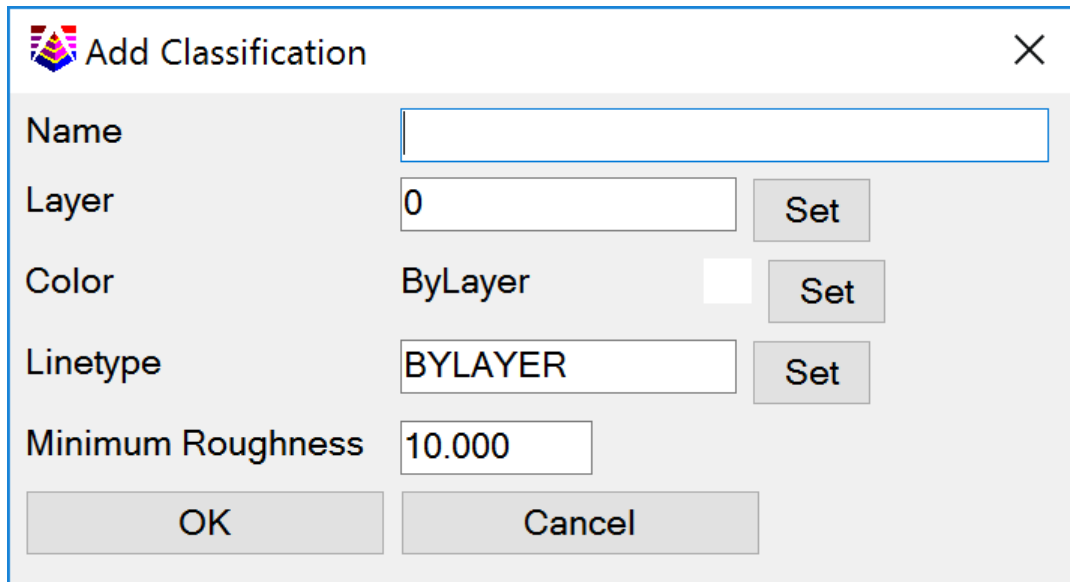
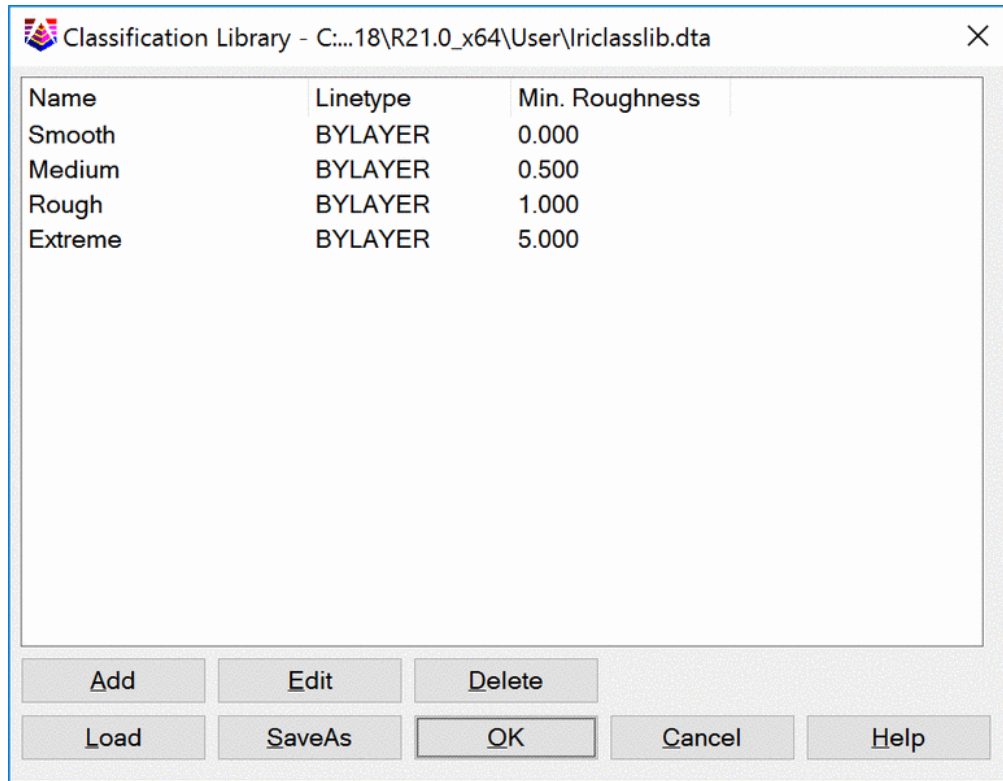
Setup Classification

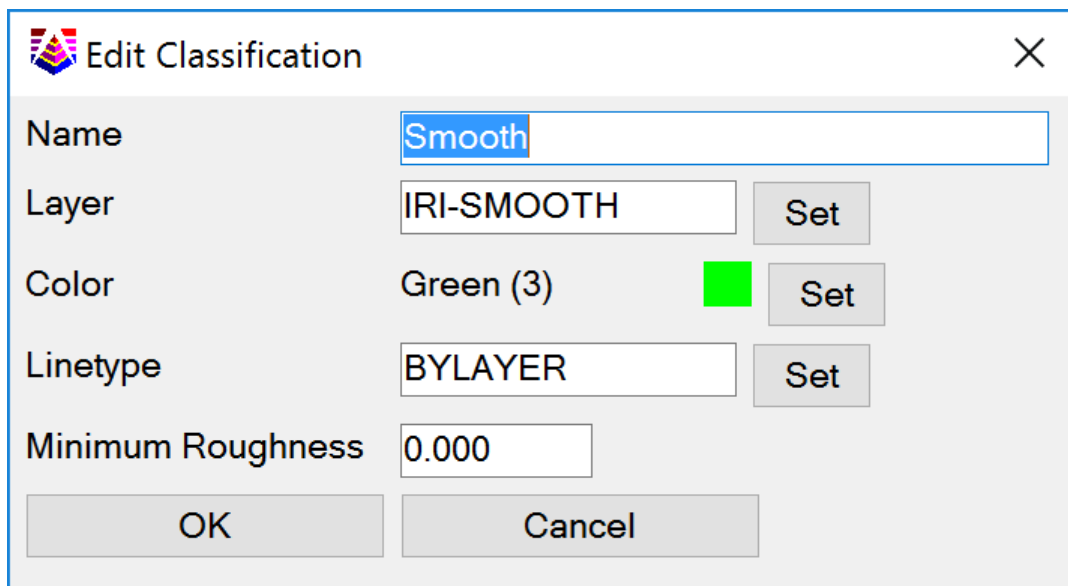
This defines a collection of roughness classifications. Each classification has a Name for a description, Layer, Color, Linetype and Minimum Roughness. Use the Add, Edit and Delete buttons to manage the list of classifications.

When the classifications are defined in the library, use the Draw Profile Offset to create the classifications in the drawing.

The current Classification Library is stored in the current USER folder under Documents & Settings in a file called

iriclasslib.dta. The Load and SaveAs functions can be used to store and recall the classification library to a .IRL file for archiving or sharing.





Report

Use the Report button to customize or "user define" the reporting options. The Report Formatter Options dialog box offers a variety of output options. You can choose the fields to report from the Available list and put them in report order under the Used list.

Selecting the Display option shows the report in the standard Carlson report viewer. Reports can be exported to an Excel spreadsheet as well.

Pulldown Menu Location: Profile > Profile Utilities

Keyboard Command: calciri

Prerequisite: .pro file

Restricted Grade Design

This command calculates the length of vertical curve required if the grade change is restricted to a rate of change per 100 feet/units.

Prompts

Percent of Grade + for uphill - for downhill.

Line select/<Percent of grade in <2.0>>: L

[nea on] select Line that defines grade in: *pick the line*

Slope Distance> 900.89 Horizontal Distance> 900.00

Elevation Difference: -40.00 Slope Ratio: -22.50:1 Slope Percent: -4.44

Line select/<Percent of grade out <-2.0>>: 2.5

Percent of grade change restriction per 100 linear units <2.0>: *press Enter*

Required length of vertical curve> 347.22

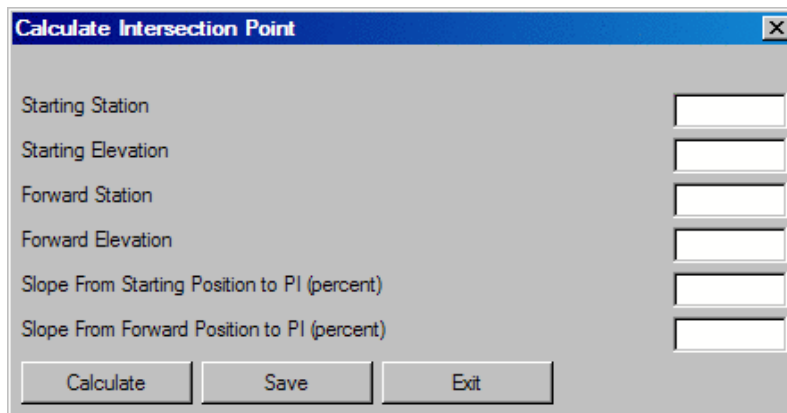
Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: vcrj

Prerequisite: None

Calculate Intersection Point

This command is a profile utility that is used to find the intersection from two points and given slopes.



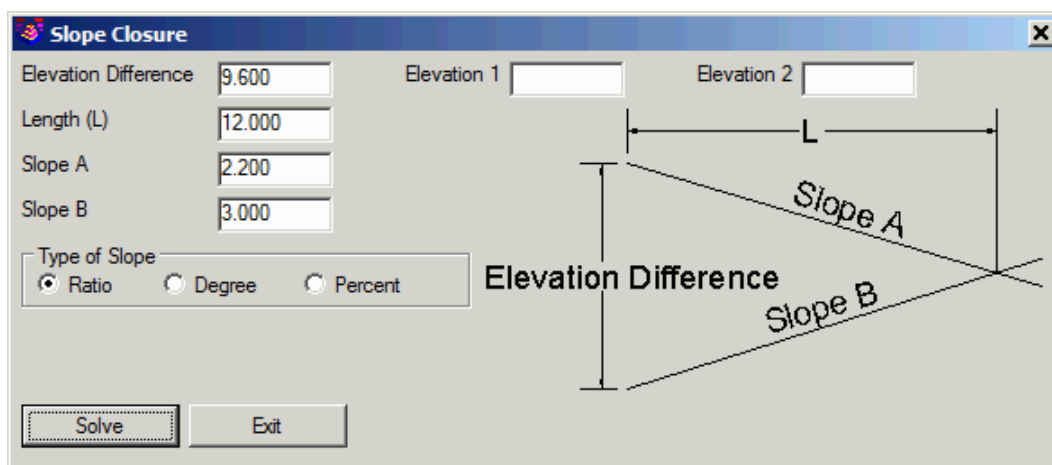
Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: calc_pro_pi

Prerequisite: None

Slope/Length/Elevation Calculator

This command calculates length given two slopes and elevation difference or elevation difference given two slopes and length. The elevation difference can be entered directly or enter two elevations to have the program calculate the elevation difference. To run, fill in the input values and pick the Solve button.



Pulldown Menu Location: Profiles > Profile Utilities

Keyboard Command: slope_calc

Prerequisite: None

Sight Distance Design

This command computes the length of vertical curve required for a user-specified sight distance, grade in, and grade out. The object height and eye height may be set using the Settings > Carlson Configure > Section-Profile Settings command.

Prompts

Percent of Grade + for uphill - for downhill.

Line select/<Percent of grade in <2.0>>: L

Select Line that defines grade in: *pick the line*
Slope Distance> 600.33 Horizontal Distance> 600.00
Elevation Difference: 20.00 Slope Ratio: 30.00:1 Slope Percent: 3.33
Line select/<Percent of grade out <-20.0>>: -2.22
View Table/<Required Sight distance <450.0>>: 450
With SD<VC, Required Length of Crest Vertical Curve> 846.41
K Value> 152.35

Pull-down Menu Location: Profiles > Profile Utilities

Keyboard Command: vcsd

Prerequisite: None

Plot VC from Tangents

This command plots a vertical curve by selecting the tangent grade line in, and then selecting the tangent line out. The vertical curve is drawn in the current layer.

Prompts

Select Line or polyline that defines grade in: *pick the line*
Slope Distance> 600.33 Horizontal Distance> 600.00
Elevation Difference: 20.00 Slope Ratio: 30.00:1 Slope Percent: 3.33
Select Line or polyline that defines grade out: *pick the line*
Slope Distance> 900.22 Horizontal Distance> 900.00
Elevation Difference: -20.00 Slope Ratio: -45.00:1 Slope Percent: -2.22
View Table/K value/Sight distance/<Length of Vertical Curve <450.000>>: 350
With SD<VC Length, Crest Sight Distance> 289.37 K Value>63.00

Pull-down Menu Location: Profiles > Profile Utilities

Keyboard Command: vctan

Prerequisite: None

Enter Roadside Ditch

With this command you enter station and elevation profile data and pipe data which is stored in a file and can then be used by Draw Roadside Ditch. The roadside ditch consists of ditch, pipe and road profiles for the left and right side of a road centerline.

The program first asks whether to read an existing file. This option allows you to add data to an existing file so that you don't have to enter all the data in one run. Before entering the profile data, the program asks for the names for the profiles on the left and right side such as shoulder and edge of road. There can be any number of named profiles. Ditch and pipe profiles do not need to be named because the program always includes these. Along with the profile name, the program also asks for a layer name to use with each profile. The layer name is used in Draw Roadside Ditch to put the profiles in different layers that can give the profiles different colors and linetypes.

Then the program begins a cycle of asking for stations and elevations. First you need to specify whether the next data is for the left or right side by entering *L* or *R*. Then enter the station followed by the profile type (either Ditch, Pipe or Roads). The ditch profile type prompts for the ditch elevation. The pipe profile type prompts for pipe elevation, ending pipe station and elevation, pipe size, pipe type, and data about what the pipe is going under. The roads profile type prompts for the elevations for each of the named profiles. If there is no elevation for one of them, just press Enter. Once all the station and elevation information is entered, enter *End* at the next station prompt. Then specify a file to save the roadside ditch data. The file has a .RDS extension and is a text file that can be edited if necessary.

Prompts

Add to existing roadside file (Yes/<No>)? *press Enter*
Left profile name or Enter for none: *EDGE OF ROAD*
Layer name for EDGE OF ROAD <ROAD1>: *press Enter*
Additional Left profile name or Enter to continue: *LT. SD. SHLDR.*
Layer name for LT. SD. SHLDR. <ROAD2>: *press Enter*
Additional Left profile name or Enter to continue: *press Enter*
Right profile name or Enter for none: *CL OF ROAD*
Layer name for CL OF ROAD <ROAD1>: *CLINE*
Additional Right profile name or Enter to continue: *EDGE OF ROAD*
Layer name for EDGE OF ROAD <ROAD2>: *ROAD1*
Additional Right profile name or Enter to continue: *RT. SD. SHLDR.*
Layer name for RT. SD. SHLDR. <ROAD3>: *ROAD2*
Additional Right profile name or Enter to continue: *press Enter*
Right/Enter Left station <0.00>: *1305*
Assign (<Ditch>/Pipe/Road)? *R*
EDGE OF ROAD elevation or Enter for none: *31.2*
LT. SD. SHLDR. elevation or Enter for none: *30.8*
End/Undo/Right/Enter Left station <1305.00>: *press Enter*
Assign (<Ditch>/Pipe/Road)? *press Enter*
Enter ditch elevation: *29.2*
End/Undo/Right/Enter Left station <1305.00>: *1307*
Assign (<Ditch>/Pipe/Road)? *P*
Enter pipe elevation: *28.48*
Enter ending pipe station: *1318*
Enter ending pipe elevation: *28.45*
Pipe size in inches <15.0>: *press Enter*
Pipe name <RCP>: *press Enter*
Enter flare width: *2*
Enter flare length: *10*
Drive type <TOP CONC. DWY.>: *press Enter*
Drive id <100>: *211*
Drive elevation: *30.79*
Drive starting station <1307.00>: *press Enter*
Drive ending station <1318.00>: *press Enter*
End/Undo/Right/Enter Left station <1307.00>: *R*
End/Undo/Left/Enter Right station <1307.00>: *1321*
Assign profile (<Ditch>/Pipe/Road)? *press Enter*
Enter ditch elevation: *30.2*
End/Undo/Left/Enter Right station <1321.00>: *E*
Dialog to Specify a File to save the entered data.
Pulldown Menu Location: Profiles > Profile Utilities
Keyboard Command: rside1
Prerequisite: None

Draw Roadside Ditch

This command draws the left and right side profiles for a road as entered and stored in a file made by Enter Roadside Ditch. The profiles are drawn on a special profile sheet. To customize this profile sheet, use AutoCAD to modify the drawing ROADSIDE.DWG in the Carlson\SUP directory.

Prompts

Choose a Roadside Ditch file to draw

Horizontal scale <10.0>: 5
Vertical scale <1.0>: *press Enter*
Grid horizontal label interval <50.0>: *press Enter*
Starting station to draw <1026.00>: *press Enter*
Ending station to draw <1320.00>: *press Enter*
Pick grid location: *pick a point*

Pulldown Menu Location: Profiles > Profile Utilities
Keyboard Command: rside2
Prerequisite: A Roadside Ditch file (.RDS)

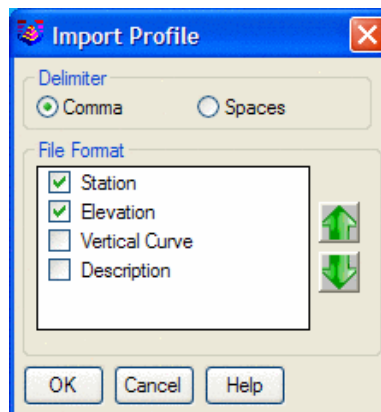
Profile Conversions

There are eleven Profile Conversion commands, all of which are listed below. The first nine in the list are Import Profile commands. These commands allow you to convert a single profile file from their respective program to the Carlson profile (.PRO) format. For each, you are prompted to select the file to be imported, then provide a Carlson profile file name. Underneath each of the nine brief descriptions shown are, in bold, the prompts that you see in dialog box form and/or on the command line.

The last two commands listed below are Export Profile commands. They allow you to convert a single Carlson profile (.PRO) file to Softdesk (.TXT) format, or a single Carlson profile (.PRO) file to Leica (.GSI) format. You are prompted to select the Carlson profile file, then provide a name for the Softdesk or Leica file.

Import Columnar Text

Allows you to Import a comma or space delimited text file to create a profile (.PRO) file.



Import CAiCE Profile

Allows you to convert a single CAiCE (.KCP) profile file to the Carlson profile (.PRO) format. You are prompted to select the CAiCE file, then provide a Carlson profile file name.

Pulldown Menu Location: Profiles > Profile Conversions
Keyboard Command: caice2pro

Import Leica Profile

Allows you to convert a single Leica profile (.GSI) file to the Carlson profile (.PRO) format. You are prompted to select the Leica file then provide a Carlson profile file name.

Choose Leica/Wild File to Read dialog Select existing file.
Choose Profile to Write dialog Select file name.

Pull-down Menu Location: Profiles > Profile Conversions

Keyboard Command: wildpro2

Import MOSS Profile

Allows you to convert a single MOSS profile (.INP) file to the Carlson profile (.PRO) format. You are prompted to select the MOSS file then provide a Carlson profile file name.

Choose MOSS Profile File to Read dialog Select existing file.

Choose Profile to Write dialog Select file name.

Pull-down Menu Location: Profiles > Profile Conversions

Keyboard Command: moss2pro

Import Novapoint Profile

Allows you to convert a Novapoint profile (.NYL) file to the Carlson profile (.PRO) format. You are prompted to select the Novapoint file then provide a Carlson profile file name.

Choose Novapoint Profile File to Read dialog Select existing file.

Choose Profile to Write dialog Select file name.

Pull-down Menu Location: Profiles > Profile Conversions

Keyboard Command: nova2pro

Import Softdesk Profile

Allows you to convert a single Softdesk profile (.TXT) file to the Carlson profile (.PRO) format. You are prompted to select the Softdesk file then provide a Carlson profile file name.

Pull-down Menu Location: Profiles > Profile Conversions

Keyboard Command: dcapro2

Import Sokkia/SDR Profile

Allows you to convert a single Sokkia/SDR (.SDR or .RAW) profile file to the Carlson profile (.PRO) format. You are prompted to select the Sokkia/SDR file, then provide a Carlson profile file name.

Pull-down Menu Location: Profiles > Profile Conversions

Keyboard Command: sdr2pro

Import Spanish ALZ Profile

Allows you to convert a single Spanish ALZ profile (.INP) file to the Carlson profile (.PRO) format. You are prompted to select the Spanish ALZ file and then provide a Carlson profile file name.

Choose CLIP File to Read dialog Select existing .ALZ file.

Choose Profile to Write dialog Select file name.

Pull-down Menu Location: Profiles > Profile Conversions

Keyboard Command: alz_to_pro

Import Spanish RAS Profile

Allows you to convert a single Spanish RAS profile (.RAS) file to the Carlson profile (.PRO) format. You are prompted to select the Spanish RAS file and then provide a Carlson profile file name.

ISPOL File to Read dialog Select existing .RAS file.

Choose Profile to Write dialog Select file name.

Pulldown Menu Location: Profiles > Profile Conversions

Keyboard Command: ras_to_pro

Import Terramodel Profile

Allows you to convert a single Terramodel (.RLN) profile file to the Carlson profile (.PRO) format. You are prompted to select the Terramodel file, then provide a Carlson profile file name.

Pulldown Menu Location: Profiles > Profile Conversions

Keyboard Command: tm2pro

Export Softdesk Profile

Choose Profile File to Read dialog Select existing .PRO file.

Choose Softdesk File to Write dialog Enter new Softdesk file name.

Pulldown Menu Location: Profiles > Profile Conversions

Keyboard Command: dcapro1

Export Leica Profile

Choose Profile File to Read dialog Select existing .PRO file.

Choose Wild File to Write dialog Enter new .GSI file name.

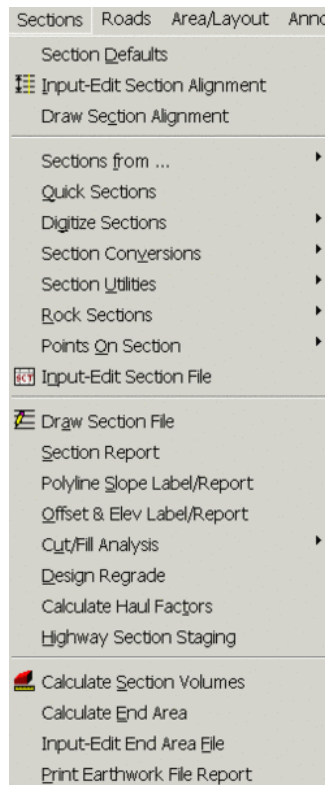
GSI file format [<8>/16]? *press Enter*

Pulldown Menu Location: Profiles > Profile Conversions

Keyboard Command: wildpro1

Section Menu

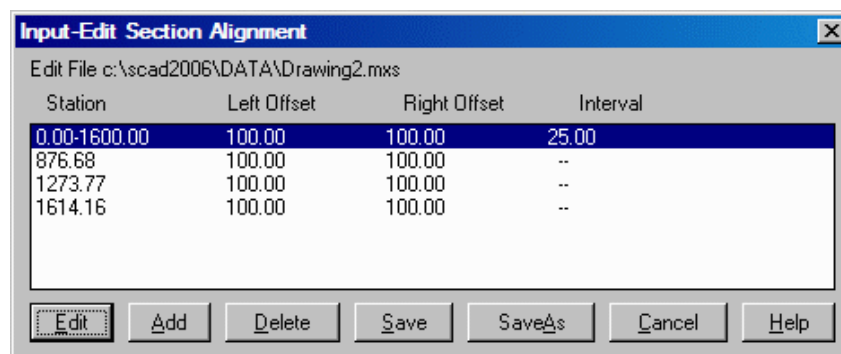
The Sections menu shown below has commands for creating, drawing and reporting sections. All commands are described in this section except for Polyline Slope Label/Report which is described under the Profiles section.



Section Alignment Editor

This command will create or append to a section alignment file which is stored as a Multiple Cross Sections (.MXS) file. This file contains the coordinates that define the center and endpoints of section lines and is a requirement of many section commands such as *Sections from Surface Entities* and *Sections to 3D Polyline*. The section alignment defines the stations along a centerline and how far left and right to create cross sections. This routine starts by asking for a new or existing .MXS file name. Then the centerline is specified by either by choosing a centerline file (.CL file) or selecting a polyline that represents the centerline. Next, the program prompts for the starting station of the centerline. If this is a new section alignment, the Make MXS File Settings dialog appears.

The Section Alignment Editor dialog lists all the section stations and offsets in the alignment of an existing .MXS file.



Dialog if using an existing .MXS file

Edit: Allows you to edit the currently highlighted row.

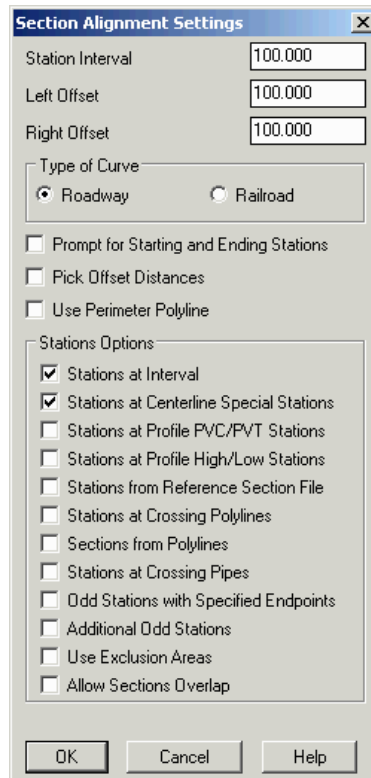
Add: Allows you to add more sections by displaying the Make MXS File Settings dialog (shown below).

Delete: Deletes the currently highlighted row.

Save: Saves the MXS file, exits this dialog and draws the section alignment on the screen using temporary vectors

(yellow for left offsets, magenta for right offsets). Any viewport change such as *Redraw* or *Zoom* will cause these vectors to disappear. The draw the section lines with Line entities, use the *Draw Section Alignment* command.

SaveAs: Saves a new MXS file with a user-specified name.



Dialog used for a new section alignment

Station Interval: Enter the station interval for sections.

Right Offset: Enter the width for the sections, right of the centerline. Not available if Pick Offset Distances is checked.

Left Offset: Enter the width for the sections, left of the centerline. Not available if Pick Offset Distances is checked.

Type of Curve: Specify either Roadway or Railroad curve to account for the differences in stationing curves.

Prompt for Starting and Ending Stations: This option allows you to specify the range of stations to process. Otherwise the program will use the full station range of the centerline.

Pick Offset Distances: Allows you to specify the offsets by using the distance between two picked points in the drawing.

Use Perimeter Polyline: Allows you to specify a closed polyline that will be used as the limit of the cross sections. The offsets will be contained within this closed polyline.

Station By Another Reference Centerline: Prompts for a separate CL to use for stationing. The program sets for station to store for the section alignment by locating the position of the original station on the section alignment centerline and then finding the station of this position along the reference centerline.

Stations at Interval: Creates cross sections at the specified interval such as every 25 feet. If the Prompt for Starting and Ending Stations is on, then the program will apply the station interval to the user-specified range of stations. Otherwise the station interval is used along the entire centerline.

Stations at Centerline Special Stations: Creates cross sections at every transition point in the centerline such as the PC, PT, spiral points and end points.

Stations at Profile PVC/PVT Stations: Creates cross sections at profile vertical curve transitions stations. When active, the program will prompt for the profile to process.

Stations at Profile High/Low Stations: Creates cross sections at profile vertical curve high and/or low stations. When active, the program will prompt for the profile to process.

Stations from Reference Section File: Creates cross sections at stations contained in the reference section file.

When active, the program will prompt for the section file to process.

Stations at Crossing Polyline: Allows you to select polylines that cross the centerline and creates cross sections at the intersections of these polylines with the centerline.

Sections from Polyline: This method prompts to select polylines that define the section alignments. This method is handy when the section alignments have multiple bends such as for HEC-RAS.

Stations at Crossing Pipes: This method creates cross sections at the stations where pipes intersect.

Odd Stations with Specified Endpoints: Creates cross sections at stations that are entered or at picked points along the centerline. This option also allows you to pick the left and right offset points which do not have to be perpendicular to the centerline.

Additional Odd Stations: Creates cross sections at the specified stations but the offsets are always perpendicular to the centerline with the user-defined default offset distances.

Use Exclusion Areas: This option prompts for selecting closed polylines to use as exclusion areas which are areas to skip for the section surface. The stations and offsets for the exclusion areas are stored in the section alignment file. Then routines like Calculate Section Volumes and Draw Sections will skip over these areas and not calculate volumes or draw section lines in these areas.

Allow Sections Overlap: On the inside of a centerline curve, section lines will cross when the section length is longer than the radius. This option controls whether to shorten the section lines on the inside of the curve or to keep the regular length and let the section lines cross.

Prompts

Specify an MXS file dialog Choose new or existing.

Polyline should have been drawn in direction of increasing stations.

CL File/<Select polyline that represents centerline>: *pick centerline*

Enter Beginning Station of Alignment <0.00>: *press Enter*

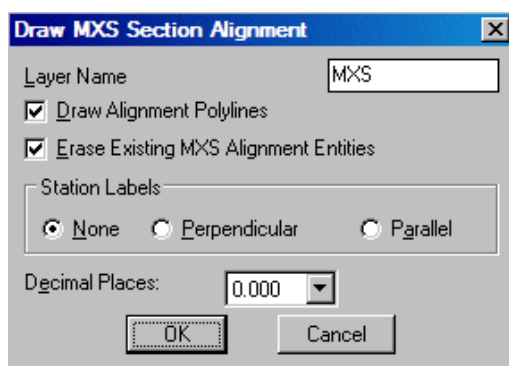
Pulldown Menu Location: Sections

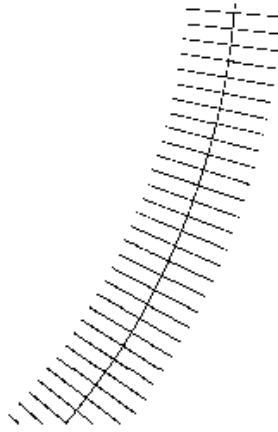
Keyboard Command: editmxs

Prerequisite: A polyline centerline or a centerline .CL file

Draw Section Alignment

This command will draw the location of the cross sections contained in an existing .MXS file. The cross sections stations can also be labeled Perpendicular or Parallel. The main purpose of this routine is to allow you to graphically view the location of the cross sections.





A drawn .MXS file along with the centerline

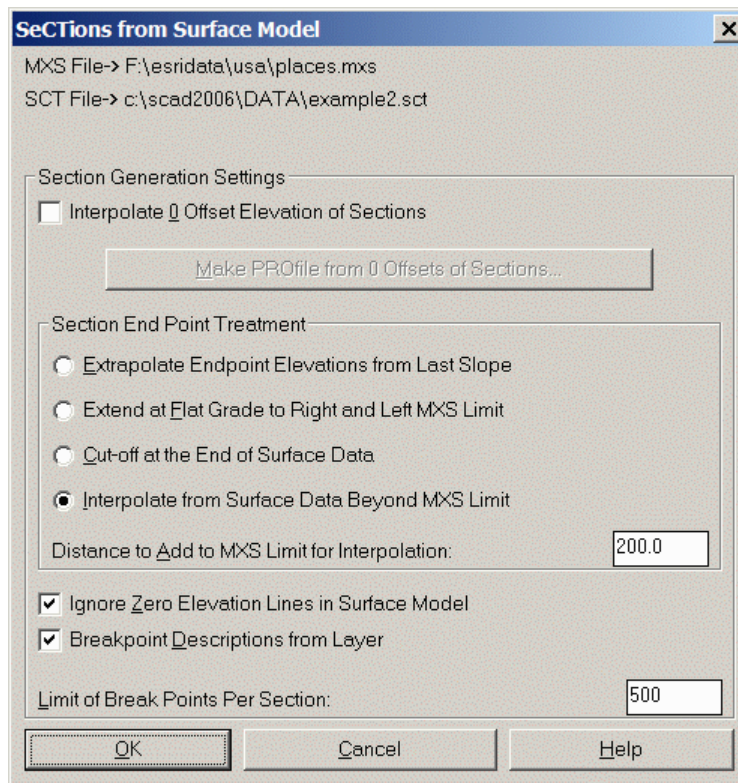
Pulldown Menu Location: Sections

Keyboard Command: drawmxs

Prerequisite: An existing .MXS file

Sections from Surface Entities

This command allows you to create cross sections from a surface model. The stations for the sections, and the left and right offset distances, are defined in the MXS file. This file must be created before running this routine by using the *Section Alignment Editor* command. The surface model is defined by lines or polylines with elevation. The polylines with elevation could be a contour drawing file from a photogrammetry firm, or it can be created from survey points with the *Triangulate & Contour* command. When using *Triangulate & Contour* it is useful to use the Draw Triangulation Lines option because the 3D triangulation lines represent all the breaklines in the surface which increases the accuracy of the cross section verses just using the contours. Breaklines or 3D polylines can also be used to represent ridges and valleys. The program samples the selected lines, polylines and 3DFace entities and calculates the intersections of these segments with any of the cross sections. The station, offset and elevation of these intersections make up the data in the section file. This section (.SCT) file can be reviewed or edited with the *Section File Editor* command. Also, the section file can be plotted with the *Draw Section File* command or used in the by the *Process Road Design* command to calculate volumes.



Interpolate 0 Offset Elevation of Sections: When checked, this option will add a data point at offset zero for every station with an elevation that is interpolated from existing offsets.

Make Profile from 0 Offsets of Sections: Allows you to specify a .PRO file name to create from the 0 offsets of the sections.

Section End Point Treatment: The section end points are the left and right furthest offsets such as left and right 100 feet. When calculating sections based on the intersections with surface entities, there usually isn't an intersection exactly at the end points. For example, there could be contours at offsets right 87.31 and 105.43 but no intersection exactly at 100. There are four methods for determining the elevation for these end points.

Extrapolate Endpoint Elevation from Last Slope: This option calculates the slope from the last two offset-elevation points and calculates the elevation at the endpoint from this slope. For example, given offsets at 80 with elevation 100, and 90 with elevation 101, the elevation at offset 100 would be 102.

Extend at Flat Grade to Right and Left MXS Limit: This option uses the last offset elevation as the end point elevation. For example, if the last offset were 85 with elevation 102, the program would add an offset at 100 with elevation 102.

Cut-off at the End of Surface Data: This option does not add offsets at the end points. The sections will end at the last offset found in the surface model.

Interpolate from Surface Data Beyond MXS Limit: This option looks beyond the offset limit for more intersections with surface entities. The endpoint elevation is then interpolated between the offsets above and below the endpoint. For example, given offsets at 90 with elevation 101, and at 110 with elevation 103, the endpoint offset at 100 would have elevation 102. If this option is selected, the Distance to Add to MXS Limit for Interpolation field becomes available.

Distance to Add to MXS Limit for Interpolation: Enter distance.

Ignore Zero Elevation Lines in Surface Model: When checked, all zero elevations will be ignored.

Breakpoint Descriptions from Layer: When checked, this option will store the layer name of the surface entity as the description for the offset-elevation point in the section file.

Limit of Break Points Per Section: Specify the maximum number of break points per section. Default value can be set using the *Section Defaults* command.

Prompts

MXS File to Process *Select the section alignment .MXS file*

Section File to Write *Specify the .SCT file*

New or Append *Choose whether to create a new .SCT section file, or add to an existing .SCT section file*

Sections from Surface Model dialog *Make selections*

Select Lines, PLines, and/or 3DFaces that define the surface.

Select objects: *Pick the surface entities*

Pulldown Menu Location: Sections

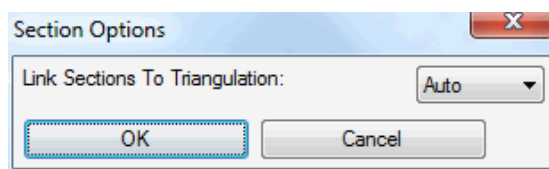
Keyboard Command: setsm

Prerequisite: Constructed surface model (.MXS file) to be sampled

Sections from Grid or Triangulation Surface

This command creates a cross section file (.SCT file) from a surface model that is defined by a 3D rectangular grid file (.GRD file) or a triangulation file (.FLT, .TIN). The grid file can be created in the Civil Design module with the *Make 3D Grid File* routine. The triangulation file can be created with the Write Triangulation File option in the *Triangulate & Contour* command. This command also requires a Section Alignment (.MXS) file to define the alignment and stations of the sections. The number of section points created is displayed at the end of the routine.

When using a triangulation file, there is an option for whether to link the sections to the triangulation. With the link, the section file will get updated in case the triangulation file is updated. When the link option is set to Auto, the update is done automatically. When the link option is set to Prompt, the program will prompt with a dialog for whether to update the sections when a triangulation change is detected.



Prompts

Choose Grid or Triangulation File to process *choose existing .GRD, .FLT, or .TIN file*

Choose MXS File to Process *choose existing .MXS*

Choose Section file to write *enter new file name*

Found 1410 section points.

Pulldown Menu Location: Sections

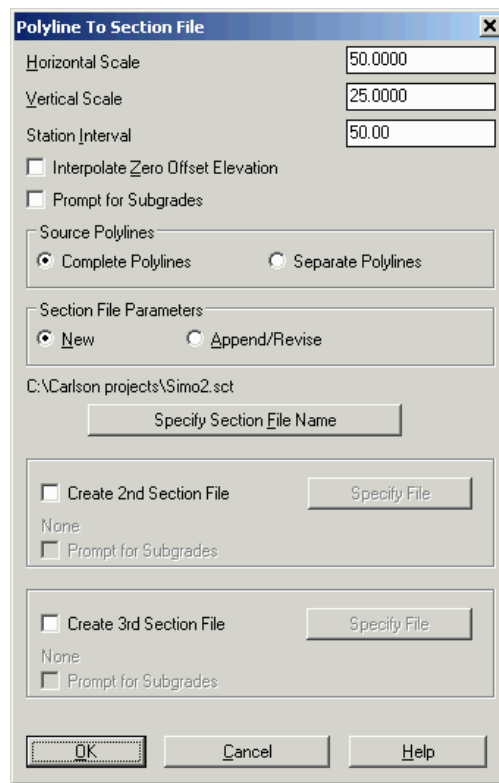
Keyboard Command: setgrid

Prerequisite: Grid (.GRD) or triangulation (.FLT or .TIN) file, and a cross sections .MXS alignment file

Sections from Polylines

This command allows the user to select a polyline that represents a section in cross section view and writes it to a .SCT file. This can be useful for revising sections or for defining a new one. For example, let's say you have extracted

sections from a surface model of the existing ground on a site, and have plotted them using the *Draw Section File* command. Next, the *Polyline by Slope Ratio* command is used to draw the proposed or final grade sections. Now use this command to send the sections to a Section file and compute the earthworks using the *Calculate Sections Volume* command. After selecting the command, the Polyline to Section File dialog appears.



The first time this command is selected, the output Section file is set to the same name as the current drawing. Select the Specify Section File Name button to specify a different name. The Station Interval edit box allows you to specify the amount that the default station number will be incremented as the station prompt shown below appears. The Interpolate Zero Offset toggle, if on, causes the program to output the elevation of the zero offset to the output .SCT file. A second and a third section file can be specified to process three sections at a time for each station. This allows you to handle both existing and final grades at once. The Prompt for Subgrades option will prompt for selecting subgrade polylines after the surface polyline for that section. For Source Polylines, the Complete Polylines method works with a single polyline that completely define the section for the station. The Separate Polylines methods allows you to select multiple polylines per section in case the section polylines are broken up into separate polyline entities. After selecting the OK button, the prompts below appear.

Prompts

Exit/Pick text/ <Station <0.00>>: *press Enter*

Exit/Pick text/ <Starting elevation of grid <100.00>> *440* This supplies the drawing coordinate to translate the grid from.

[int on] Pick point at starting elevation and zero offset of section ([Enter] for none): *press Enter*

Select station 0.00 1st section polyline: *select a polyline*

Select station 0.00 2nd section polyline: *select another polyline*

Exit/Pick text/ <Station <0.00>>: *E*

Pulldown Menu Location: Sections

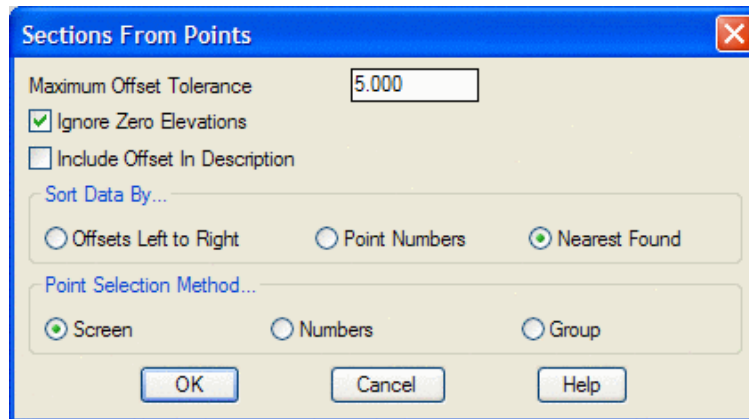
Keyboard Command: sctfpl

Prerequisite: Plot the section or profile to write to the .SCT file.

Sections from Points

This command creates an .SCT file from Carlson points in the drawing. An .MXS file is needed to define the centerline and the stations of the cross sections. The offsets for the cross section points are derived from the perpendicular distance between the centerline and the Carlson points. The cross section elevations come directly from the elevations of the points. In order to be included in a cross section, a Carlson point must be within the offset tolerance distance of the cross section line.

The order that the points within the Offset Tolerance at each station are used will of course determine the shape of the cross section. There are three ways for the collected points to be sorted. The Offsets Left to Right option sorts by the distance of each point from the CL. The Point Numbers option ignores that data, and instead sorts the points by their numbers. The Nearest Found option ignores both distance from the CL and point numbers and instead checks the horizontal and vertical proximity of the points to each other and sorts them based on this data. A powerful application of this method would be a survey of a tunnel where the points collected at each station were collected in a random order.



Prompts

Choose MXS File to Process *select file*

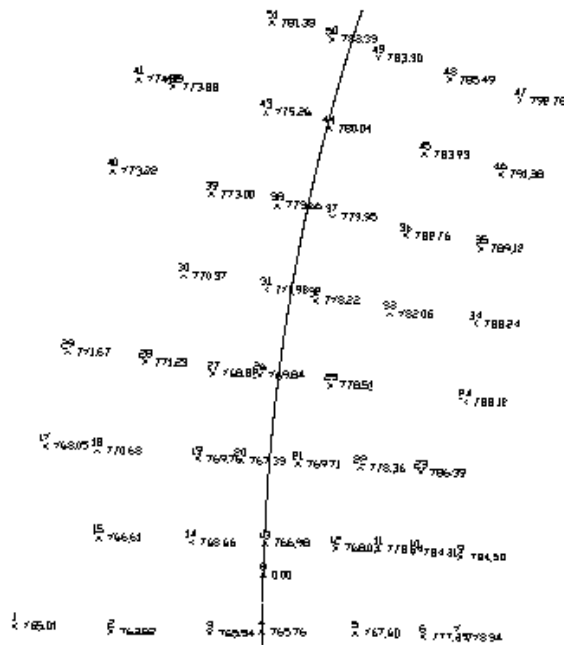
Choose SCT file to Append/Write *select file*

Enter the maximum offset tolerance <1.0>: *press Enter*

Ignore Zero Elevations (<Yes>/No)? *press Enter* This option filters out all Carlson points that have a zero elevation.

Select points along the sections.

Select objects: *pick the Carlson points*



Carlson points for use in creating Section file

Pulldown Menu Location: Sections

Keyboard Command: scpts

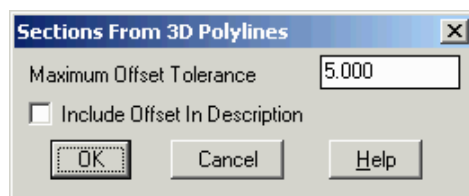
Prerequisite: Carlson points and an .MXS file

Sections from 3D Polylines

This command creates a cross section file (.SCT) from 3D polylines. The 3D polylines should be drawn perpendicular to the centerline. A section alignment file (.MXS) from the Section Alignment Editor command is needed to define the centerline and the stations of the cross sections.

The elevations for the cross section points come from the 3D polyline vertice elevations. The offsets for the cross section points are derived from the perpendicular distance between the centerline and the polyline vertices. In order to be included in a cross section, a polyline vertice must be within the **Maximum Offset Tolerance** distance of the cross section line.

When using 3D Polylines drawn in cross section, they should be drawn from left to right across the centerline. Use the Reverse Polyline command if needed to change the 3D polyline direction.



Prompts

Choose MXS File to Process *select file*

Choose SCT file to Append/Write *select file*

Sections From 3D Polylines dialog

Select polylines for sections.

Select objects: *pick polylines*

Pulldown Menu Location: Sections

Keyboard Command: sct3dp

Prerequisite: 3D Polylines and section alignment file (.MXS)

Sections from Coordinate File

This command will read a Carlson coordinate file, and, if the proper point descriptions are found, convert it to a .SCT file (stations, offsets, & elevations). This command offers great utility when combined with an electronic data collector. Sections can be surveyed and then compiled directly to cross sections, without having to extract them from a triangular mesh surface model. In other words, from the same file we can derive both plan and cross section views from one survey file. The .CRD file should have a point with a centerline description, followed by points that describe left and right offset points. For example, lets say we have a .CRD file with the following data in it.

Prompts

Coordinate File to Process dialog *choose the .CRD file*

Section File to Write *choose the .SCT file to create*

Sections From Coordinate File dialog *put in range*

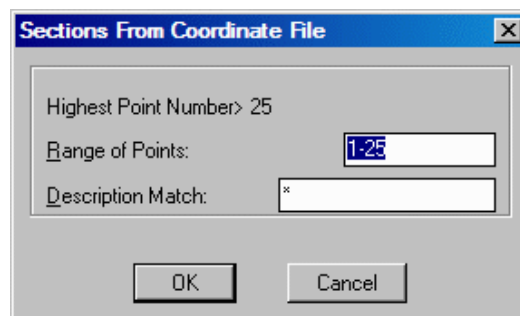
Station Center Point Description <SC>: *press Enter* You can use any set of characters that you want to use as the zero offset description code, although SC is recommended because it is the default.

Station Left Point Description <SL>: *press Enter* You can use any set of characters that you want to use as the left offset description code, although SL is recommended because it is the default.

Station Right Point Description <SR>: *press Enter* You can use any set of characters that you want to use as the right offset description code, although SR is recommended because it is the default.

Point#	North	East	Elevation	Description	
1	1000	1000	1050	CL/SC1+50	(centerline descriptor)
2	1007	1000	1049	EP/SL1+50	(left offset descriptor)
3	1008	1000	1048.2	ES/SL1+50	
4	1010	1000	1048	TD/SL1+50	
5	1012	1000	1046	BD/SL1+50	
6	1014	1000	1047.6	TD/SL1+50	
7	1016	1002	1047.8	PP	(would be ignored and not used in *.SCT file)
8	993	1000	1049	EP/SR1+50	(right offset descriptor)
9	992	1000	1048.2	ES/SR1+50	
10	990	1000	1048	TD/SR1+50	
11	988	1000	1046	BD/SR1+50	
12	986	1000	1047.6	TD/SR1+50	

Notice that SC is the zero offset/centerline description code, SL is the offset left description, and SR is the offset right description. The station number should immediately follow and be the last characters in the description field. The station number may or may not contain a plus sign. This example would produce an .SCT file that contains the station number 150 and 10 offsets and elevations.



Pulldown Menu Location: Sections

Keyboard Command: textstc

Prerequisite: A coordinate file with the correct descriptions.

Sections from Profile

This command creates or adds data to a .SCT file from the station-elevation data in a profile file (.PRO). For every matching station in the profile and section file, the profile elevation is applied to a specified offset in the section file. For example, consider a profile for the edge of pavement that is a constant 12 to the right side of the centerline. The elevations from this profile could be added to offset 12 in the section file. The program will assign the elevation at station 1+00 from the profile to the elevation at station 1+00 and offset 12 in the section file. When creating a new section file, an .MXS file is needed to define the station interval.

Prompts

Section offset of profile <0.0>: *enter the offset Negative for left, positive for right.*

Choose SCT File to Append/Write dialog *choose new or append existing .SCT file*

Choose MXS file to process dialog *choose existing*

Choose Profile file to process dialog *choose existing .PRO file*

Created 11 section points.

Pulldown Menu Location: Sections

Keyboard Command: pro2sct

Prerequisite: A profile .PRO file and .MXS file

Sections from Layers

This command creates cross sections from surface entities on one of the specified layers. The surface entities can be contours, triangular mesh, and other 3D drawing entities. This command is the same as *Sections from Surface Entities* with the addition of the layer filtering. Specify the layer names of the surface entities to include in the sections. For example, the layer names CTR and TMESH could be entered to use only the contour polylines and triangulation mesh on these layers. Entities on all other layers would be ignored. See *Sections from Surface Entities* for more details

Prompts

Specify Layer Names Dialog *pick Select Layers from Screen button, choose, then OK*

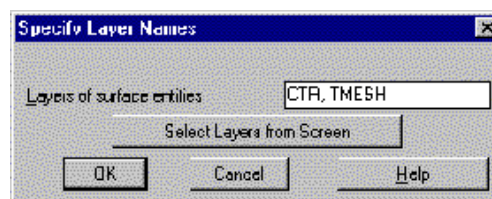
MXS File to Process dialog *choose existing .MXS alignment file*

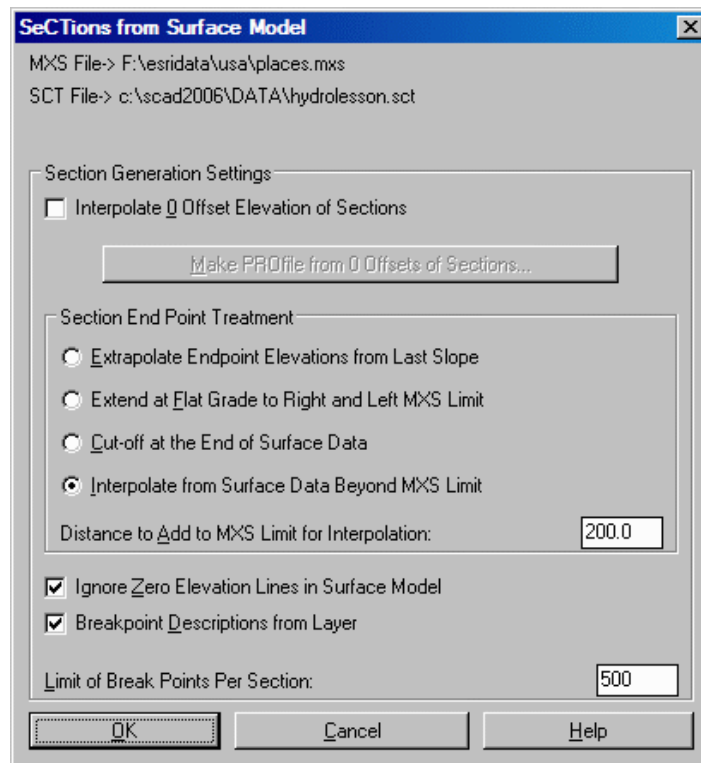
Section File to Write dialog *specify a new .SCT section file to create or append existing*

Sections from Surface Model dialog *Make selections*

Select surface entities on corresponding layers.

Select objects: *pick the linework to process*





Pulldown Menu Location: Sections

Keyboard Command: sctlayer

Prerequisite: A cross sections alignment file (.MXS file), surface lines and polylines.

Sections Points from Surface Entities

This command creates a section .SCT file in a similar way as *Sections from Surface Entities*. The difference is that these section points are not connected by a polyline as a surface in *Draw Section File*. Instead, a point symbol is drawn for each offset-elevation point. First this command prompts for the symbol to use. Next the program asks for the section alignment (.MXS) file to use, then the section (.SCT) file to make or append. Then you select polylines to create the section points from. The intersections of these polylines with the cross sections are calculated and the resulting station-offset-elevations are stored in the section file. The description field for these section points identify them as individual points instead of a surface. The description starts with POINT- followed by the symbol name (e.g. POINT-SPT5).

Prompts

Select Symbol dialog

MXS File to Process Select an existing section alignment .MXS file

Section File to Write dialog

New or Append Choose between creating a new .SCT file or add to an existing section file

Select surface polylines.

Select objects: *pick the polylines*

Added 21 points to section file.

Writing section file> C:\scad2006\data\horn.sct

Pulldown Menu Location: Sections > Points on Section

Keyboard Command: sctpts2

Prerequisite: Surface polylines and an MXS file

Sections Points from Right of Way

This command is the same as *Sections Points from Surface Entities* except that the symbol for the right of way is automatically drawn as a downward pointing arrow. The description field for these section points identify them as right of way points. The description is POINT-ROW. A new .SCT file is created or an existing one appended to.

Prompts

MXS File to Process Select a section alignment file

Section File to Write dialog

New or Append Choose between creating a new .SCT file or add to an existing file.

Select right of way polylines.

Select objects: *pick the polylines*

Added 21 right of way points to section file.

Writing section file> C:\scad2006\data\horn.sct

Pulldown Menu Location: Sections > Points on Section

Keyboard Command: sctrow

Prerequisite: Right of way polylines and an MXS file

Sections Points from Pipes

This command is the same as *Section Points from Surface Entities*, except that the symbols for the pipes are automatically drawn as a circles with a radius set to the pipe width. The profile equivalent of this command is *Profile from Pipe Polyines*. When *Draw Section File* has vertical exaggeration, the pipe is drawn as an ellipse. The description field for these section points identify them as pipe points. The description is POINT-PIPE followed by the pipe size in feet (e.g. POINT-PIPE-1.500). The pipe polyines used to derive the pipe section points can be created with the *Draw Pipe 3D Polyline* and *Assign Pipe Width to Pline* commands in the Sections->Section Utilities menu. Also, the position of the pipe polyines on the pipe can be selected. The choices are Top, Center and Bottom.

Prompts

MXS File to Process Select a section alignment file

Section File to Write

New or Append Choose between creating a new .SCT file or add to an existing file.

Select pipe polylines.

Select objects: *pick the polylines*

Position of pipe polyines on pipe [Top/Center/<Bottom>]? Enter

Added 46 pipe points to section file.

Writing section file> C:\scad2006\data\pipe.sct

Pulldown Menu Location: Sections > Points on Section

Keyboard Command: sctpipe

Prerequisite: Right of way polylines and an .MXS file

Quick Section

This command creates section files in one step. The horizontal alignment for the sections can be defined by using picked points, a centerline file or a polyline. A section alignment (.MXS) file is not required for this routine. 3D screen entities or surface files (.GRD, .FLT, or .TIN) are used to define the vertical alignment.

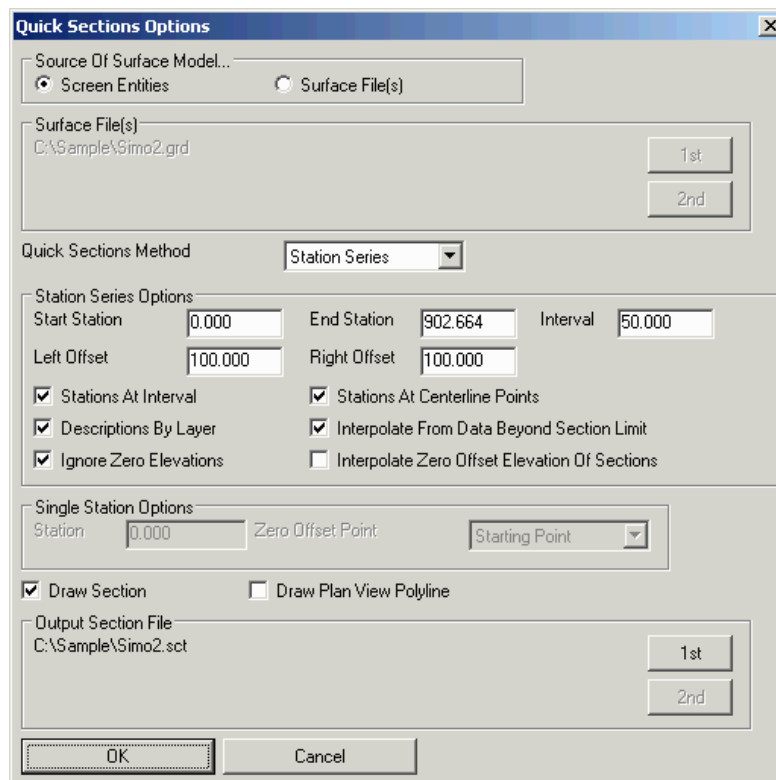
There are two options under Quick Section Methods. The Station Series method creates sections perpendicular from the horizontal alignment at a station interval. In this case, the horizontal alignment represents the centerline. The Single Station method creates one section along the horizontal alignment appends this section to the output section file. In this case, the horizontal alignment represents the alignment of the section.

For the Station Series method, there are settings for the Start Station of the horizontal alignment, the End Station to stop creating sections, the Interval for the stations, and the Left and Right Offsets to define the section width. There are also options to control the section stations to create. The Stations At Interval option will create sections at the specified station interval. The Stations At Centerline Points option will create sections at the special stations of the centerline for the centerline transitions such as PC, PT points

For the Single Station method, the Station value is assigned to this section. The Zero Offset Point chooses between using the starting point of the horizontal alignment as the zero offset or selecting a point along the alignment as the zero offset.

With the Source Of Surface Model set to Surface Files, the program prompts for up to two surface files so that up to two section files can be generated at a time. When the Surface Model is set to Screen Entities, only one section file is created from the screen entities. With Screen Entities, there are a few more options. The Descriptions By Layer option will use the layers of the screen entities as the descriptions for the section points. The Interpolate From Data Beyond Section Limit will check for intersections with the section line and the screen entities beyond the left/right offsets to interpolate the elevations at the left/right offset extents. The Ignore Zero Elevations will filter out screen entities that are at zero elevation. The Interpolate Zero Offset Elevation Of Sections will create a section point at offset zero by interpolating between the nearest section points.

The program requires an output section file to store the results. There is an output option to draw the sections which calls the Draw Section File command. Finally, the option to Draw Plan View Polyline will draw the horizontal alignment as a polyline which is especially useful if the method to define the alignment by picked points was used.



Prompts

Pick starting point (CL-Centerline,P-Polyline): *select a point*

Pick second point: *select second point*

Pick next point (Enter to end): *press Enter*

Quick Section Options dialog

Choose Source of Surface Model, Screen Entities or Surface File, and make other selections. Click OK.

Keyboard Command: quickset

Prerequisite: 3D Screen entities or surface files

Tablet Calibrate

This command executes the routine to perform calibration of the digitizer tablet to the drawing. There are two methods of calibration: Known Reference Points, and Drawing Scale with New Reference Points, which are explained in detail below. The Calibrate routine must be used prior to using the Digitize Contours command.

Please refer to Configure , General Settings and Digitizer Puck Layout for selection of the correct puck layout before proceeding.

Tablet Calibration

Known Reference Points uses two known coordinates for reference points on the drawing. When this option is selected, the fields for coordinate info activate. Enter the known northing and easting values for the reference points from the info on the drawings in the appropriate fields and press ok. The command line will prompt for the selection of each point from the drawing on the tablet. Furthermore, Carlson saves the coordinates of the two reference points for future calibrations and displays them on the **Tablet Calibration Dialog** the next time it is accessed, so if you are working in the same drawing, you can use the **Known Reference Points** method with the saved coordinates to digitize back to your previous coordinates. For greater calibration accuracy, choose two points that are farther apart rather than closer together.

Tablet Calibration

Calibration Methods

Known Reference Points

Drawing Scale with New Reference Points

Draw Reference Points

Known Reference Points

Point 1: Northing Easting

Point 2: Northing Easting

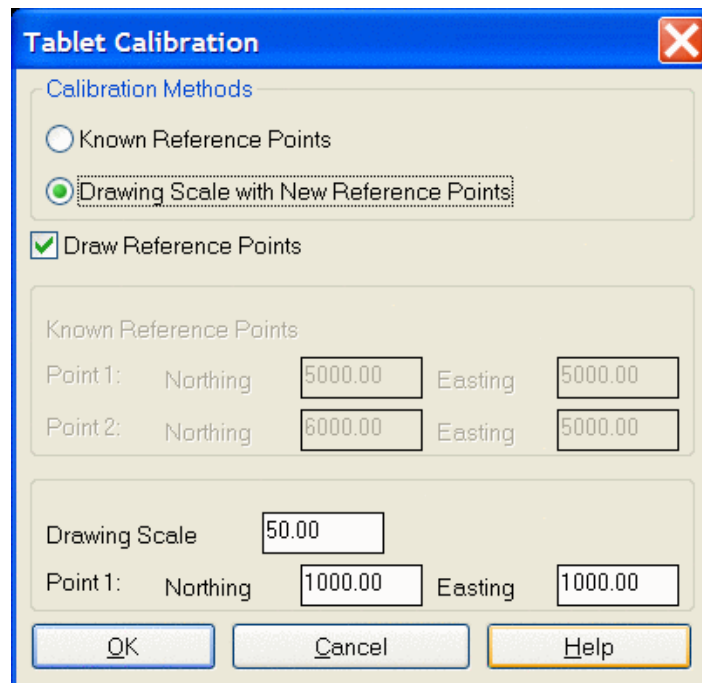
Drawing Scale

Point 1: Northing Easting

OK Cancel Help

Drawing Scale with New Reference Points is very convenient when you don't know the precise coordinates of the entities on your drawing. The user must specify the drawing scale from your plan, this method can establish a coordinate system relative to the position of the plan on the digitizer board. In addition to the drawing scale, you

are required to enter a random coordinate for the first reference point, the default coordinate is (1000,1000). Takeoff would computer the coordinate of the second reference point that you pick based on the first point. The coordinates of these two reference points would be saved and will be display on the **Tablet Calibration Dialog** next time when you calibrate the tablet, so you can digitize back to the previous coordinates using **Know Reference Points** method if you are working on the same drawing, though you might have moved or rotated your drawing on the digitize board option allows the user to specify the drawing scale of the plans be digitized and to assign an assumed northing and easting for a base point. When selected, Drawing Scale and Northing and Easting for Point 1 activate. Press ok. The command line will prompt for a pick of the first point.



Prompts

Tablet Calibration Dialog

Specify the Calibration Methods. If you select Drawing Scale method, enter the drawing scale and the coordinate of the first reference point. Otherwise enter the exact coordinates of the first and second reference points.

Pick first reference point: *pick a point*

Pick second reference point: *pick another point*

Pulldown Menu Locations: Contour in Civil Design, Sections in Civil Design, Digitize in Takeoff

Keyboard Command: digsetup

Prerequisite: Affix a drawing to your digitizer tablet. Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Select the puck layout in Configure .

Digitize Sections Plan

This command allows you to digitize cross sections from a contour map. This is useful for pulling cross sections and earthworks from existing contour maps made by aerial photography, USGS or other engineering firms.

Prompts

Use **TABLET CALibrate** command to set scale prior to using this routine.

Contour Increment <1>: 2 The contour increment/interval of the map to be digitized.

Pick Zero Offset Station point: *pick a point*

Zero Offset Elevation: 1122.56

Starting Contour elevation <1122>: 1124 This is the elevation of the first contour to digitize.

Next Point Up Right: *pick a point*

If the first contour line is moving up in elevation and to the right of the zero offset point, then pick a point on the first line with elevation 1124. If this is not the case, then review the options below to change the prompt mode.

Press digitizer/mouse buttons:

- 1 - To pick next point on contour line
- 2 - To change to UP mode
- 3 - To change to DOWN mode
- 4 - Prompt for new elevation (this elevation is applied to the next point picked)
- 5 - To change to RIGHT mode
- 6 - To change to LEFT mode

or Press Keys:

- [X] - To end point prompting
- [U] - To change to UP mode
- [D] - To change to DOWN mode
- [N] - Prompt for new elevation (this elevation is applied to the next point picked)
- [E] - Erase/Delete the last point picked
- [R] or [+] - To change to RIGHT mode
- [L] or [-] - To change to LEFT mode

Press one of the keys, buttons or select from the side bar screen menu to change prompt to appropriate mode. When you have finished picking points press the [X] key to end the point prompting. The program then prompts:

Send Section to a file (Y/N) <Y>: *press Enter*

Name of Section File to write </sc/data/example.sct>: *press Enter* If the file already exists the user is asked to Overwrite or Append to the file.

Section Station Number: 100 The station, offsets and elevations are then written to the section (.SCT) file.

Pulldown Menu Location: Sections > Digitize Sections

Keyboard Command: digsec

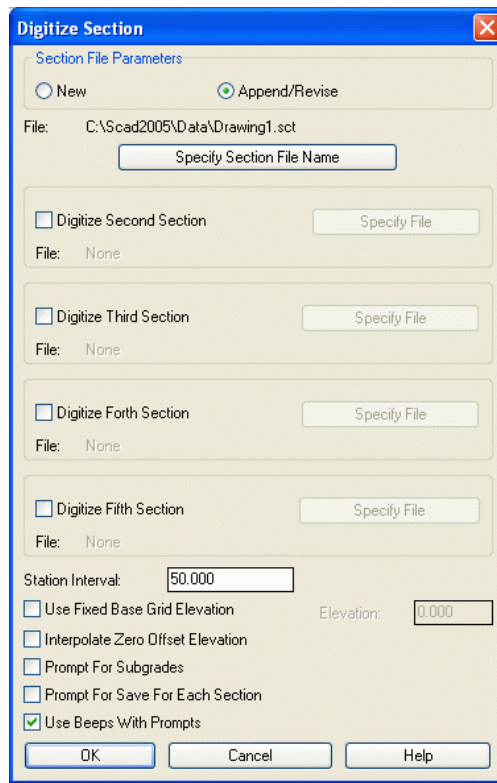
Prerequisite: An existing plan view contour drawing on screen or digitizing tablet. If digitizing a map on your tablet use the *Tablet Calibrate* command to calibrate your digitizer tablet to the scale of the drawing.

Digitize Sections XSec

This command creates a section file (.sct) by digitizing a section drawing. The command starts with the dialog shown below where you specify the section file name to create. The station interval is used to automatically default to the next station value when digitizing a series of stations. The Interpolate Zero Offset option will interpolate an elevation at the exact zero offset.

After the dialog, the program will prompt to pick three reference points on the section. These points should have known offsets and elevations. Additional sections can be aligned by a single point. Corners on the section grid can be used for these reference points. The reference points and the user-entered offset and elevations for them sets up the program for the section. Now you can start picking the section grade points.

You can also digitize existing and final surfaces back to back, and there is an undo function that will allow undo while digitizing points. As the section is digitized, it is shown in a real-time graphics window. Holding down the right mouse button acts as a zoom function, while holding down the mouse scroll button acts as a pan. The puck keys can be used to enter all the input data.



Prompts

Digitize Section dialog

Calibrate section sheet:

Pick First section sheet reference point: *pick a point on the section grid*

Enter offset <0.0>: -50

Enter elevation: 200

Pick Second section reference point: *pick another point on the section grid*

Enter offset: 50

Enter elevation: 210

Pick Third section reference point:

Enter offset: 50

Enter elevation: 230

Section station to digitize <0.000>: 133.63

Digitize break point for SAMPLE GRID section 133.630 (Enter to end): *pick a point on the section starting at the left and working right*

Digitize break point for SAMPLE GRID section 133.630 (Enter to end): *pick a point on the section*

Digitize break point for SAMPLE GRID section 133.630 (Enter to end): *pick a point on the section*

Save changes to SAMPLE GRID section 133.630 [<Yes>/No]? *press Enter*

Digitize break point for SAMPLE GRID FINAL section 133.630 (Enter to end): *pick a point on the section starting at the left and working right*

Digitize break point for SAMPLE GRID FINAL section 133.630 (Enter to end): *pick a point on the section*

Digitize break point for SAMPLE GRID FINAL section 133.630 (Enter to end): *pick a point on the section*

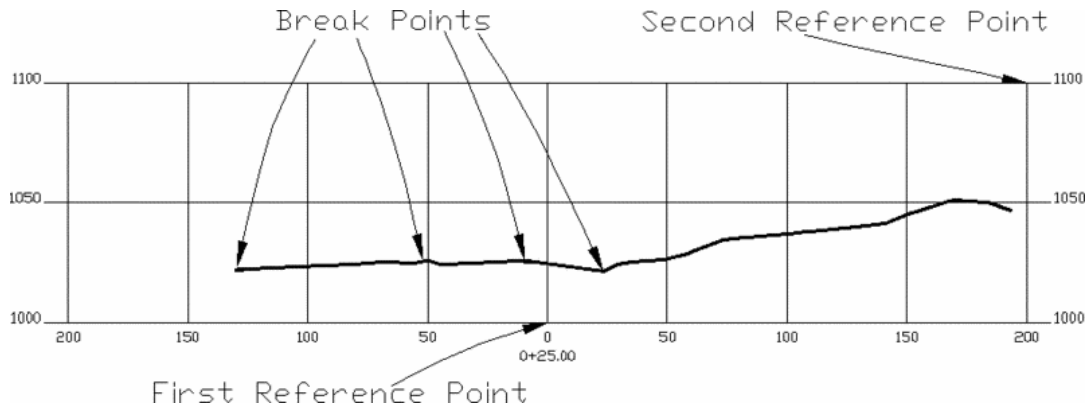
Save changes to SAMPLE GRID FINAL section 133.630 [<Yes>/No]? *press Enter*

Digitize another station [<Yes>/No]? *press Enter*

Calibrate next section:

Pick section reference point: *pick a point on the section grid*

Enter offset <-50.00>: 0
 Enter elevation <200.00>: 200
 Section station to digitize <233.630>: 200
Digitize break point for SAMPLE GRID section 200.000 (Enter to end): *pick a point* on the section
Digitize break point for SAMPLE GRID section 200.000 (Enter to end): *pick a point* on the section
 Save changes to SAMPLE GRID section 200.000 [<Yes>/No]? *press Enter*
Digitize break point for SAMPLE GRID FINAL section 200.000 (Enter to end): *pick a point* on the section starting at the left and working right
Digitize break point for SAMPLE GRID FINAL section 200.000 (Enter to end): *pick a point* on the section
Digitize break point for SAMPLE GRID FINAL section 200.000 (Enter to end): *pick a point* on the section
 Save changes to SAMPLE GRID FINAL section 200.000 [<Yes>/No]? *press Enter*
 Digitize another station [<Yes>/No]? *N*



Pulldown Menu Location: Sections > Digitize Sections
Keyboard Command: digxsec
Prerequisite: Affix cross section on digitizing tablet

Digitize End Areas

This command writes an earthwork (.EW) file that can be used by the *Print Earthwork File Report* command and print an earthworks and volumes report. It is the users responsibility to record the sections in the proper consecutive sequence. The earthwork (.EW) file written by this command can be edited in any ASCII text editor.

Prompts

Datum elevation <0.0>: *100* Enter the datum elevation that you calibrated the tablet with.
Horizontal Scale <20.0>: *press Enter*
Vertical Scale <20.0>: *10*
Digitize cut area (Enter to end): *pick a point* Starting at either end of the section, digitize break points of cut area.
Digitize cut area (Enter to end): *pick a point*
Digitize cut area (Enter to end): *pick a point*
Digitize cut area (Enter to end): *press Enter* Press Enter to end prompting of break points. The end area is then displayed.
More cut areas (Y/N) <N>: **[Enter]**
 Respond with Y if you have more cut areas.
Digitize fill area (Enter to end): *pick a point*
Digitize fill area (Enter to end): *pick a point*
Digitize fill area (Enter to end): *pick a point* Press Enter to end prompting of break points.
More fill areas (Y/N) <N>: *press Enter* Respond with Y if you have more fill areas.
Send data to file (Yes No) <Y>: *press Enter* If you made no errors respond with Y to save data in the file.

End Area File to write <c:\scad2006\data\quan.ew>: *press Enter*

Station Number: 150 This would enter a station of 1+50.

Pulldown Menu Location: Sections > Digitize Sections

Keyboard Command: digendar

Prerequisite: An existing cross section on digitizing tablet. If digitizing a map on your tablet use the Tablet Calibrate command to calibrate your digitizer tablet to the scale of the drawing.

Section Conversions

All Import commands in this submenu are designed to convert other section file formats to the Carlson section (.SCT) file format. The Import Columnar Text has some options to make the program match the import data. This routine can be used for section text files that have station, offset, elevation and optionally description separated by spaces or commas. All the other Import routines read specific formats from other software. The Export commands are designed to convert the Carlson section (.SCT) file format to other section file formats. You will be prompted to specify the file name to convert, then specify a section (.SCT) file name.

Note: The Import/Export LandXML Files routine in the File menu supports section data as well as other survey and civil data types.

Another Note: The Section Report routine can be used to Export section data from Carlson and this command includes an option to use the Report Formatter which allows you to select the fields to export and their order. Plus the Section Report report formatter has functions to export to Excel and databases.

Prompts

Prompts and commands vary for importing and exporting section file data.

Importing:

Import Columnar Text

Type of delimiter [<Space>/<Comma>]? *C for comma.* Choose the type of separator between fields in the import file.

Section data contains description field [Yes/<No>]? *N for no.* This option specifies whether the import file contains descriptions for the section points.

Add description to section data [Yes/<No>]? *Y for yes.* This option will assign a specified description to the section points.

Description for section data: *TOPO*

Import Agtek Reads .ROG and .RDS format section files (ASCII only).

Import Arkansas DOT Imports Level Note File

Import C&G Reads C&G .CEW section files.

Import CAICE Earthworks Reads .ERP files.

Import Ceal Reads CEAL section files.

Import EMXS Reads section data from .XNG files.

Import GEOPAK Reads .XRS, .XSR, and .TXT format section files (ASCII only).

Import Georgia DOT Reads .END files.

Import IGRDS Reads .LIS, .RDS, and .TXT files.

Import InRoads Reads .TXT files.

Import MicroStation Reads InRoads .GEN files.

Import Moss Reads MOSS section files.

Import NC DOT Reads .ORI and .TXT files.

Import Pizer Reads .TXT files.

Import RoadCalc Reads RoadCalc (Eagle Point) sections files.

Import SMI Reads .CUT format section files (ASCII only).
Import Softdesk Reads .SEC format section files (ASCII only).
Import Spanish SC1 Reads ISPOL .SC1 section files.
Import Spanish TRV Reads CLIP .TRV section files.
Import TDS Reads .RD5 and .TP5 files.
Import Terramodel Reads .XSC files.

Exporting:

Export C&G Converts Carlson .SCT file to .CEW format.
Export GEOPAK Converts Carlson .SCT files to .TXT format.
Export IGRDS Converts Carlson .SCT files to .RDS format. Prompts for section surface type - original ground or final surface.
Export RoadCalc Converts Carlson .SCT files to RoadCalc (Eagle Point) format.
Export Topcon Converts Carlson .SCT files to .RD3 format.

Pulldown Menu Location: Sections > Section Conversion

Keyboard Commands xsecread, agtek, level, ceal, geopak2sct, gadot2sct, igrds2sct, moss, ncdot2sct, pizer2sct, inroadcalc, smisct, softsct, sc1_to_sct, trv_to_sct, tm2sct, sct2geopak, sct2igrds, outroadcalc, gen2sct

Prerequisite: Sections files; formats vary by command

Create Overshoot Section File

This command allows the user to create an overshoot section file using an existing .SCT section file and an existing rock section file. The existing .SCT file must be read, then an existing rock section file (.SCT) needs to be chosen. The new overshoot section file and file name will then be written. An overshoot drop depth is the key prompt in this command, along with the values for the beginning and the final stations. This will begin the processing of this routine and create the new data.

Prompts

Existing Section File to Read *select SCT file*

Rock Section File to Read *select SCT file*

Section File to Write *enter new SCT file name*

Enter the overshoot drop depth: 5

Range of stations: 0.00 to 150.00

Enter beginning station to process <0.00>:

Enter final station to process <150.00>: 1000

Last station to average <1614.160>: *press Enter*

Section File to Write Select a SCT file name and folder.

Pulldown Menu Location: Sections > Rock Sections

Keyboard Command: rockover

Prerequisite: A .SCT file

Draw SectionTemplate DWG

This command is step one in the Point on Section procedure. It inserts a section or template drawing to scale on a standard section sheet. The section or template must be an existing, separate drawing (.DWG) file. Prior to running this command, the scale and text size should be set with the *Drawing Setup* command found on the Settings menu.

Prompts

Enter the horizontal scale <1.0>: *press Enter*

Enter the vertical scale <1.0>: *press Enter*

Three file selection dialogs follow.

Choose Standard Section Sheet: *pick existing .DWG file* The standard section sheet is a drawing created at 1'=1' with its insertion point coordinates 0,0 at the lower left gridline.

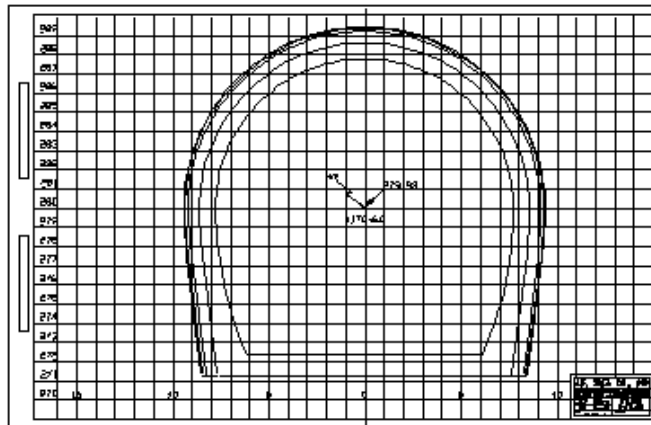
Choose Profile File: *pick existing .PRO file* The profile (.PRO) file for the vertical alignment defines the insertion elevation for the template insertion point.

Choose Tunnel Template: *pick existing .DWG file* The tunnel or section template is a drawing created at 1'=1' with the insertion point for the template at coordinates 0,0.

Enter profile station for section/template: 117060

Enter or pick section/template insertion point *pick a point or press Enter for none*

The standard section sheet with template in the center of sheet at the input horizontal and vertical scales is plotted.



Section sheet with template

Pulldown Menu Location: Sections > Points On Section

Keyboard Command: tunnel

Prerequisite: Vertical alignment in .PRO file, template or section drawings, and section sheet drawing.

Point Placement on Section

This command is step two in the Points on Section procedure. Before running this routine, the section or template sheet must be drawn on screen and there must be an existing coordinate (.CRD) file to read, with station and offset data in the description fields, as described under Points on Section. Station location points may also be picked on screen with the Points option. This command draws points on the section template from the coordinate (.CRD) file or via the Points method. If the Point option is selected, a Section Settings dialog appears, followed by a Snap Point dialog. The point elevation and the offset data in the description field are used to locate the point on the section.

Prompts

Place points from .CRD file or pick points [File/<Points>]? F

Enter the horizontal scale <1.0>: *press Enter*

Enter the vertical scale <1.0>: *press Enter*

Layer for points <PNTS>: *press Enter*

Select Coordinate File to Read Dialog *pick a file* You select the crd file to process.

Range of Point Numbers to use (A for All) <A>: *press Enter for all points to process*

Wildcard match of point description <*>: *press Enter for all points with or without descriptions*

Plot Full or Abbreviated text (Full/<Abbrev>)? F Here we used F for full description.

Range of stations: 117060.000 to 117090.000

Enter station to process: 117060

Enter search zone <1.0>: Search zone applies to survey data collected in an approximate range plus or minus a small distance on a known station.

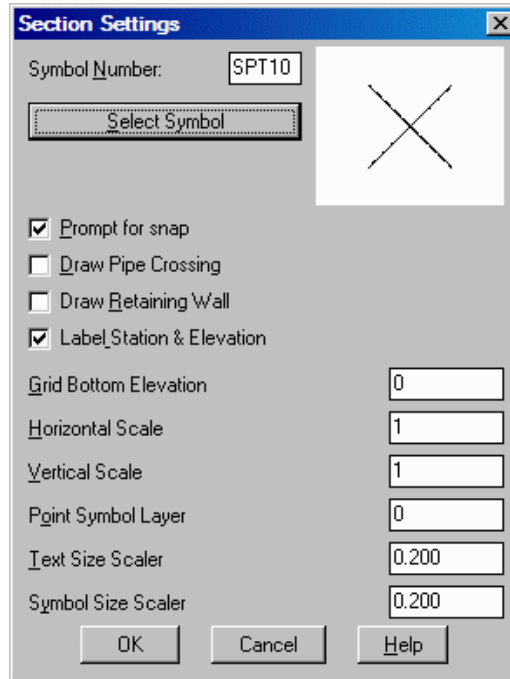
Pick Center of Grid [int on]:

Pick a known elevation on the centerline and on the next prompt enter that elevation.

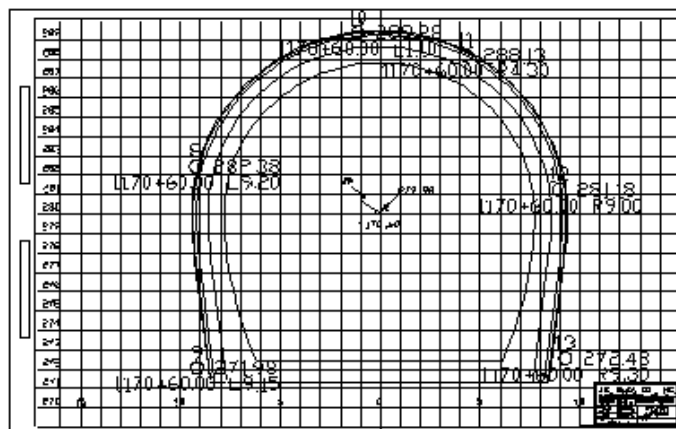
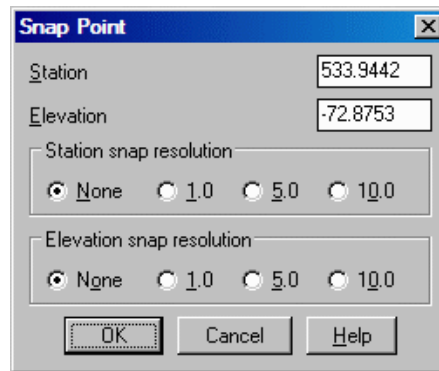
Enter base elevation of grid: 278

Enter station to process: *Enter next station or press Enter to end*

The points plot on the template or cross-section.



Point option dialog



Points plotted on template or cross-section

Pulldown Menu Location: Sections > Points On Section

Keyboard Command: ptsct

Prerequisite: Drawn section sheet and .CRD file with station and offset description field data

Point Offset Report/Plot

This command, which is step 3 in the Points on Section procedure, labels the tight/clear distances between points and a polyline on the section. Tight points are inside or to the right of the polyline while clear points are outside or to the left of the polyline. The polyline should be drawn in a clockwise direction. The points can be picked on the section or located from a .CRD file with the station and offset data in the description fields as described under Points on Section.

Prompts

Enter the horizontal scale <1.0>: *press Enter*

Enter the vertical scale <1.0>: *press Enter*

Coordinate File to Process Dialog Box *pick a file* Select the coordinate file for clear/tight report.

Write report to file (Yes/<No>)? *Y*

Enter the report file name to write: *117060.XS*

Write report to printer (Yes/<No>)? *Y* Make sure the printer is on-line and connected to the printer port.

Press Enter to continue *press Enter*

Write report into drawing (Yes/<No>)? *Y*

Enter the report title <Tight/Clear Report>: *press Enter*

Pick location for report: *pick starting location for Tight/Clear report*

Use manual or automatic label placement (Manual/<Automatic>)? *press Enter* Automatic draws the arrow leader lines perpendicular a set distance prompted next. Manual allows picking each leader lines location.

Label offset distance <1.00>: *press Enter*

Enter station to process: *117060*

Enter search zone <1.0>: Search zone applies to survey data collected in an approximate range plus or minus a small distance for a known station.

Pick Center of Grid [int on]: Pick a known elevation on the centerline and on the next prompt enter that elevation.

Enter base elevation of grid: *278*

Select polyline: *pick template polyline*

Number/<Pick Point>: *N* You can pick offsets or use point numbers.

Pick point/<point number or range>: *8-13*

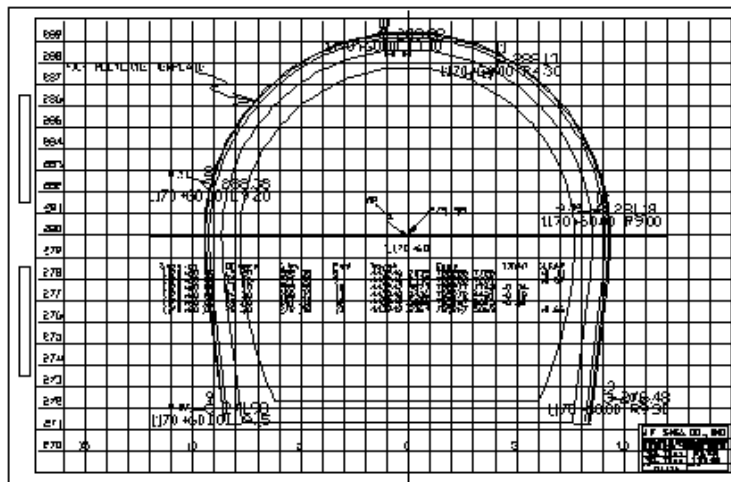
Station	Offset	Elev	Pt#	North	East	TIGHT	CLEAR
1170+60.00	L9.20	282.38	8	443246.7039	785285.7725	+0.11	
Station	Offset	Elev	Pt#	North	East	TIGHT	CLEAR
1170+60.00	L9.15	271.98	9	443246.7118	785285.7231	+0.57	
Station	Offset	Elev	Pt#	North	East	TIGHT	CLEAR
1170+60.00	L1.10	289.28	10	443247.9877	785277.7749	-0.04	
Station	Offset	Elev	Pt#	North	East	TIGHT	CLEAR
1170+60.00	R4.30	288.13	11	443248.8436	785272.4431	-0.19	
Station	Offset	Elev	Pt#	North	East	TIGHT	CLEAR
1170+60.00	R9.00	281.18	12	443249.5886	785267.8025	-0.32	
Station	Offset	Elev	Pt#	North	East	TIGHT	CLEAR
1170+60.00	R9.30	272.48	13	443249.6361	785267.5063	+0.66	

Pick point/<point number or range>: *press Enter*

Enter more point numbers or pick more offsets or press return for no more.

Enter station to process (Enter to End): Enter next station or press Enter to end.

With Point Offset Report/Plot completed the finished product with points plotted on the cross section can be plotted to a printer or plotter with Tight/Clear or Cut/Fill report included as shown.



Pull-down Menu Location: Sections > Points on Section

Keyboard Command: sctoffset

Prerequisite: Drawn section sheet

Section File Editor

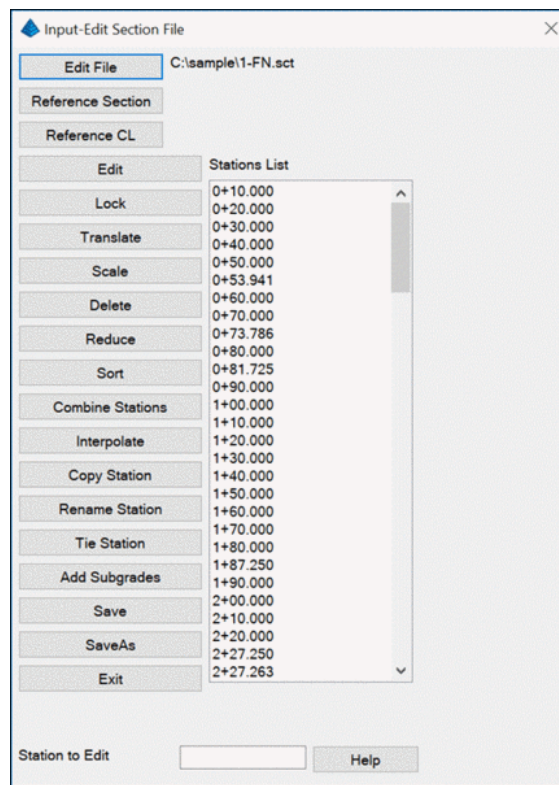
This program can be used to enter or edit data stored in a section file (.SCT file), including a real-time graphic window in the Edit mode. The section data consists of stations, offsets, elevations and descriptions. This command also has utilities for translating the offsets and elevations, deleting stations from the file, intersecting the out-slopes of one section file with another, combining multiple occurrences of the same station and sorting the stations, offsets and elevations.

While editing the section file, a second section file can be used as reference. To choose this file, pick the **Reference Section** button. For example, when editing the proposed section file, you can reference and view the ground section file as the second file. Besides showing the reference section in the graphic preview, the program also reports the end areas while editing a section station. Also, the reference section can be used to tie to the catch point. Another option is to set a **Reference CL** which is used to adjust the stations when the centerline has station equations.


The program begins by prompting for a New or Existing section .SCT file to process. The Section file to process dialog appears, allowing you to specify the file that you want to operate on. Use the New option to create a new file. Use the Existing option to edit the offsets and elevations for station/sections that you have already created, or append new stations to a file. The program defaults to a section file with the same name as the drawing or a name that you specified using another section command. You also can choose a 2nd existing .SCT file to reference. After specifying the file name(s), the program displays any stations currently in the file, in the Stations List of the Section File dialog box.

Alternately, when sections are drawn in the drawing, you can double-click on a section polyline to launch Section File for the .SCT file associated with the section polyline.

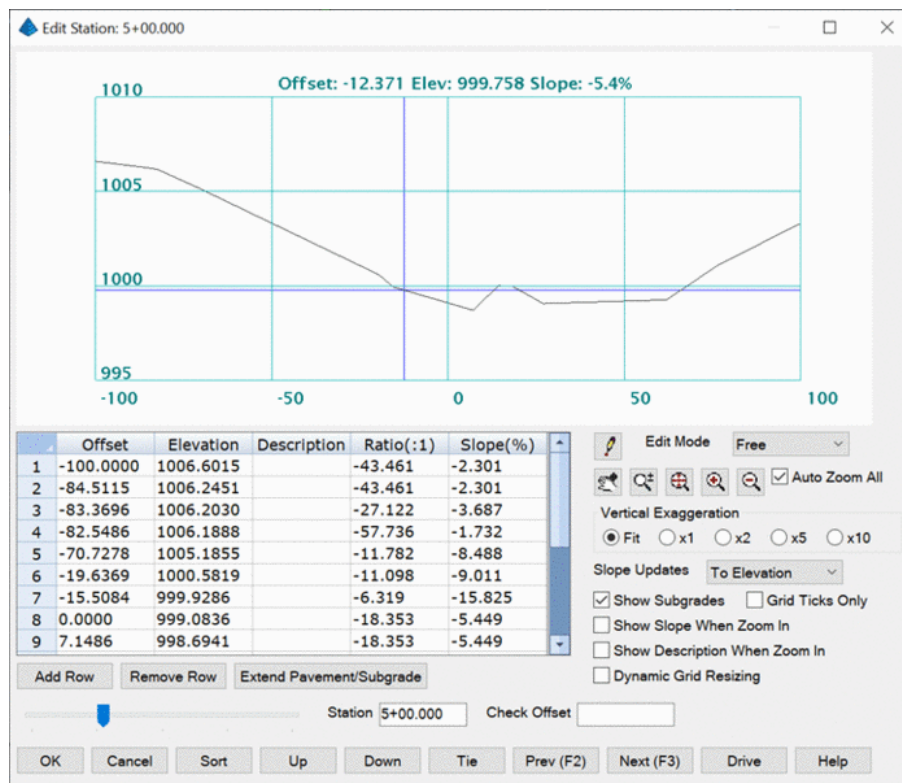
If you specified a new file, the Stations List box will be blank. To edit and display the offset and elevation data at a station, you can double click on the station in the Stations List box, or input the station in the Station to Edit edit box at the bottom of the dialog. To add a station to a new file or existing file, you must enter the station in the Station to Edit edit box. Stations will present in accordance with the Section-Profile settings in Configure under the Settings pull-down menu (eg. 10+00, 1+000, 1000).



Edit: Opens the Edit Station dialog which shows a graphic of the section on top, a list of the offset-elevation points in the middle, and the function buttons on the bottom. To add an offset point, type in the offset, elevation and optional description in the spreadsheet. Left offsets are entered as negative numbers. You can enter the slope or ratio from the last point and the program will calculate the elevation. To edit an offset point, highlight the point from the list and then edit the values in the Offset, Elev and Desc columns. The highlighted point will be marked by an X in the graphic screen. The Sort button will sort the list of offsets from lowest to highest, left to right. It is recommended that you Sort offsets before doing the Tie command, so that the left-most and right-most offsets appear first and last in the offset list. The Up button will move the highlighted offset point up in the list. Likewise the Down button moves the highlighted offset point down in the list. Prev (F2) and Next (F3) buttons move through the stations and allow you to review and edit stations in forward or reverse order. The scroll bar can also be used to quickly move through stations and then zero in with Prev (F2) or Next (F3). When you edit a slope value in the spreadsheet, then the Slope Updates setting controls which elevation value to update for the section segment.

The section data can be edited directly in the spreadsheet or graphically by picking the Edit Point button with the pencil icon.  To edit graphically, use the click-n-drag method. Start by picking the Edit Point button and then pick the section point to edit in the graphic preview and hold the mouse button down and then drag the mouse to the new position and then release the mouse button. The **Edit Mode** setting governs the click-n-drag operation. The **Free** mode allows the section point to be moved anywhere. There **Hold Slope** modes maintain the slope and moves the point along these slopes. The **Hold Offset** allows changing the section point elevation only. The **Hold Elevation** allows changing the point offset only.

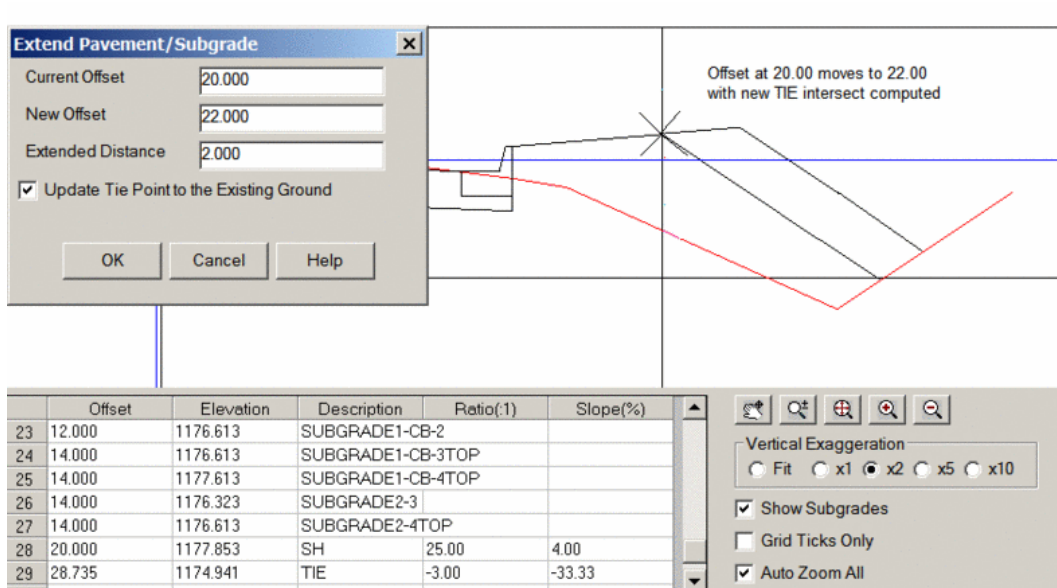
When a Reference Section is set in the main dialog, the Edit Station spreadsheet has a column for Depth. This Depth is the elevation different to the reference section. You can edit the depths to change the elevations for the section.



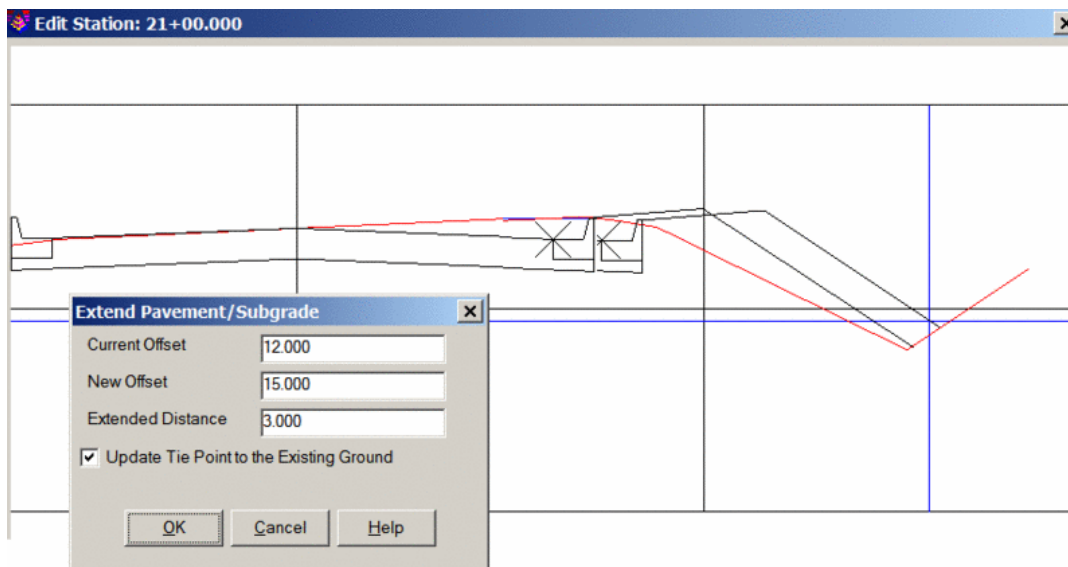
The **Add Row** button inserts an offset line above the currently highlighted row. The **Remove Row** button erases the highlighted offset and elevation from the list. After inputting or editing press the **OK** button to return to the Stations List dialog and keep any changes you have made. Select the **Cancel** button if you want to cancel changes made to the current station.

A real-time report of offset-elevation-slope now displays in the top of the graphic as you move the cursor across the section in the graphic window. The screen defaults to zoom mode where holding down the right-mouse button zooms in and out. You can also switch to pan mode. There are buttons for zoom extents, zoom in and zoom out. If your mouse has a scroll button, you can hold it down to pan and scroll it to zoom in and out. You can also set the **Vertical Exaggeration** ranging from 1X to 10X and including "Fit". **Show Subgrades** has the ability to tie a subgrade into the surface. **Grid Ticks Only** just shows the left and bottom axis lines of the grid with grid tick marks along the axes. With **Auto Zoom All** turned off, you can hold the same view position as you click **Next** and **Previous** and move through the list of stations. The **Description** is used in Draw Section File command to label under the station value and used in the Section Report command. The **Check Offset** field calculates an elevation based on an entered offset.

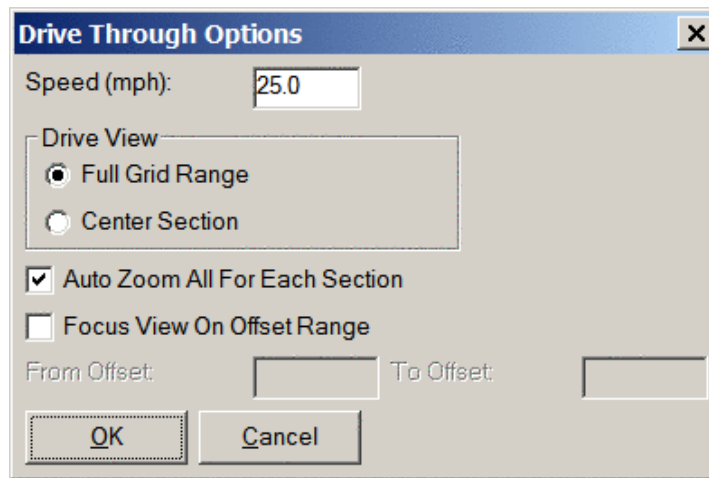
Extend Pavement/Subgrade allows you to move a surface point and shift, in parallel, the associated subgrades and tie points. One application, shown below, is to extend a shoulder point and re-computer the TIE point, all in one clean operation:



Another application of Extend Pavement/Subgrade is to move the curb position and all associated subgrades. The "inside" curb point is at 12.00 units from centerline. If the pavement is extended from 12 to 15 at this station, use of this feature will extend the subgrades, maintain all slopes and re-compute the TIE point, as shown below:

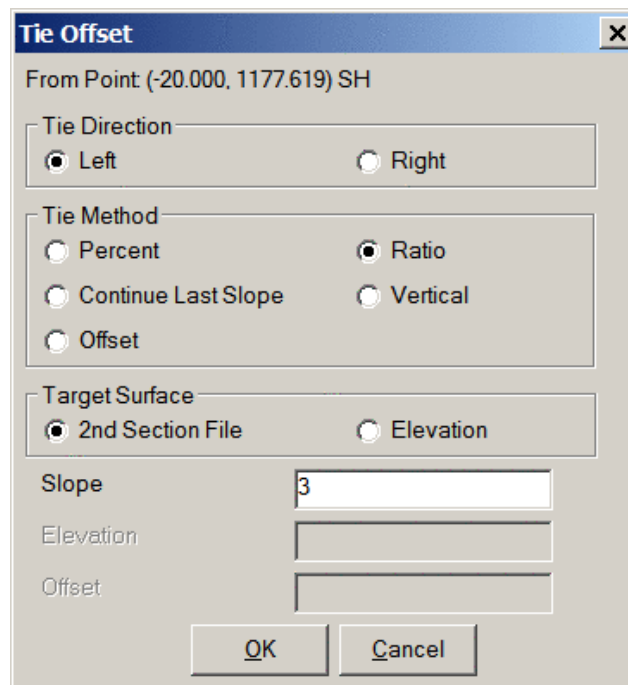


Drive (Edit Station): This function scrolls through the sections at the rate of speed specified by the user in the Speed window. The Drive View options determine whether the sections are displayed using the full width of the graphic window or centered in the window. The combination of Full Grid Range and Auto Zoom All allows the sections to rise and fall with the centerline elevations, as if you were driving an actual road. With Auto Zoom All off, and Full Grid Range on, the grid itself moves up and down at the current position of the first section, as you drive. Focus View On Offset Range allows the user to set the left and right viewing limits of the sections. Section data beyond the specified limits is not displayed.



Elevation Field (Edit Station): Equations (+, -, *, /) can be entered to calculate or adjust an elevation. For instance, to subtract 1.25' from elevation 1926.18, simply enter 1926.18-1.25 and press enter. The new elevation will be calculated and displayed in the viewer window.

Tie (Edit Station): The Tie button allows you to tie the left and right surface points of the 1st section file into the 2nd section file. It is used for classic outslope intersects from final grade to existing grade. The dialog layout includes an option to tie the section to a specified elevation, in addition to a surface (second section file). A left or right tie direction can also be selected. If a point has been tied in from SH for shoulder at offset -20 at 3:1, a new offset with the description "TIE" is created. If you try another outslope such as 4:1 from the same SH shoulder point, a new "TIE" point is created and the old TIE point is removed automatically.



Lock: This function will tag the section file as locked so that no routine can automatically overwrite this file. If a routine attempts to overwrite this section file, the program will stop, report that the file is locked and prompt whether to override the lock.

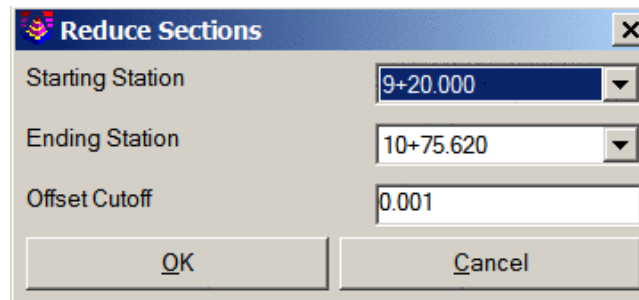
Translate: Allows you to add or subtract a distance from the offsets to adjust or shift the centerline. You can also adjust the elevations up or down. When using this option, you can choose the range of stations to operate on (starting and ending stations) and the values to adjust the offsets and elevations. If, for example, you want to shift the centerline, but not the elevations, enter the plus or minus amount you want to translate, and when prompted

for the elevation enter zero. The Station Equation function removes the station equation adjustments to the station values in order to set the station values to the true centerline distances. To use the Station Equation function, a Reference CL must be set on the main dialog. For elevations, there is an option to use a profile to define the amount of elevation change as the difference between the profile elevation and the section elevation at the zero offset.

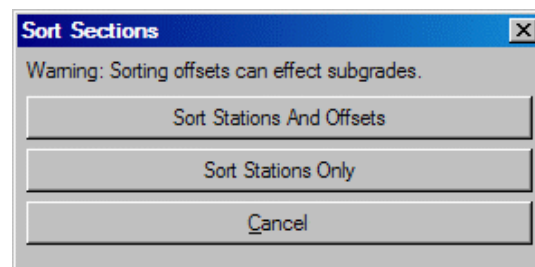
Scale: Allows you to scale the station, offsets and/or elevations by the specified scale factor. This function can be used to convert between English and metric units.

Delete: Allows you to remove a station or range of stations from the Stations List. You can delete a range of stations or an individual station. Also there are options to delete all the data for the selected stations or filter to delete only data that is outside an offset or elevation range. Since the station editor data is stored in memory, if you accidentally delete a range, Quit the editor without saving the stations to disk. Then recall the original file.

Reduce: Allows you to remove offsets from a range of stations by removing vertices in the offsets that are virtually in a straight line. Using an offset cutoff, meaning no offset and elevation moves more than the entered amount (eg. 0.01), excessive numbers of vertices can be eliminated. The command is similar to Reduce Vertices when applied to the plan view.

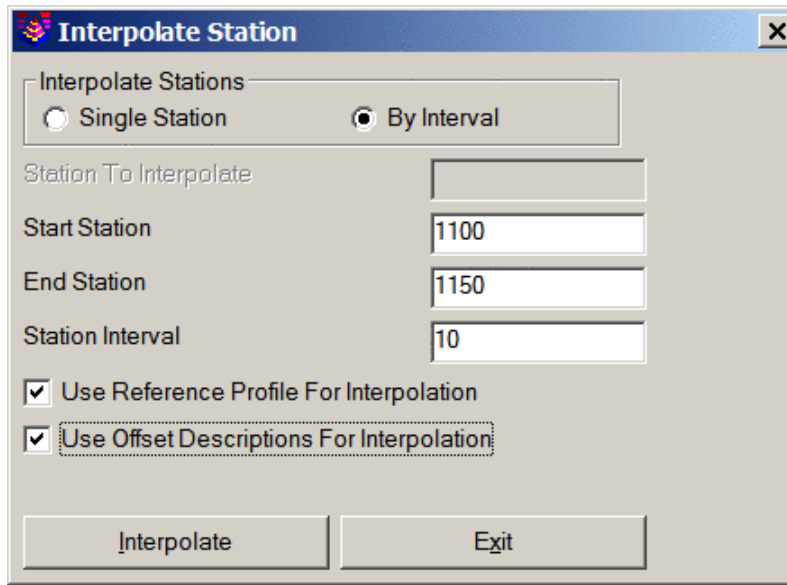


Sort: Allows you to sort the station numbers into ascending order, and sort the offsets and elevations in the individual station records (offsets are sorted from left to right). When sections are derived from the *Sections from Surface Entities* command they are already sorted, but when sections are digitized or input manually they occur in the order that you digitized them. So, for proper plotting and earthworks, you may want to run the Sort option before processing.



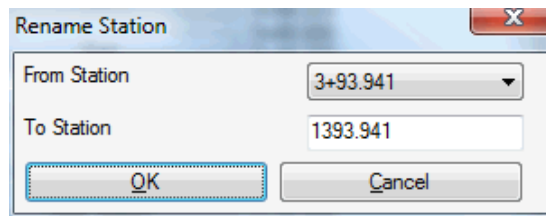
Combine Stations: Used to bring together in one record slot multiple occurrences of the same station number. This can occur when using the Digitize Sections (XSec) command and the section that you are digitizing has match/break lines which forces you to digitize the station in two or more parts.

Interpolate: Allows you to add or overwrite a station between two stations or projecting forward from two stations. You can choose to interpolate a single station or an interval of stations. Specify the two known stations in the Start Station and End Station edit boxes, as well as the interval if using the interval method. The program will do straight line, mathematical interpolations, adding offsets to the interpolated stations to match the totality of offsets in the starting and ending stations. However, if the offsets have descriptions, you can choose to interpolate by description and the program will interpolate by description (eg. EP at 12 on Station 1100 and EP at 15 at station 1150 would lead to EP at 12.6 at 1110). There is also an option to reference a profile, so if station 1100 and 1150 were on either side of a high point at 1125, the interpolated offsets would respect the profile as well as the starting and ending station. Use of this command is often critical to creating accurate digital terrain models of sites for machine control. Select the OK button to execute the function with the current settings or select the Cancel button to abort the process.

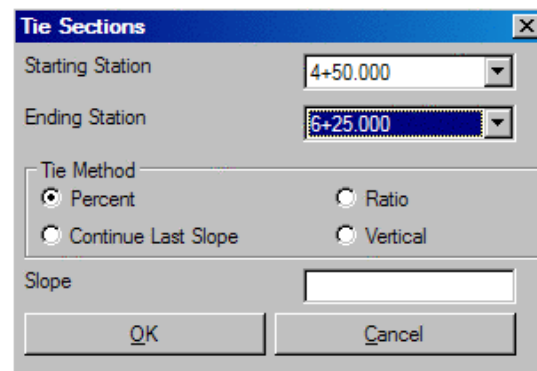


Copy Station: Allows you to copy a station that already exists to a new or existing station number. Choose the existing From Station using the edit pulldown box, then enter the new station number in the To Station edit box. Select the OK button to execute the function with the current settings, or select the Cancel button to abort the process.

Rename Station: Allows you to change the value of a station. In the dialog, select the existing station from the list and enter in the new station value.

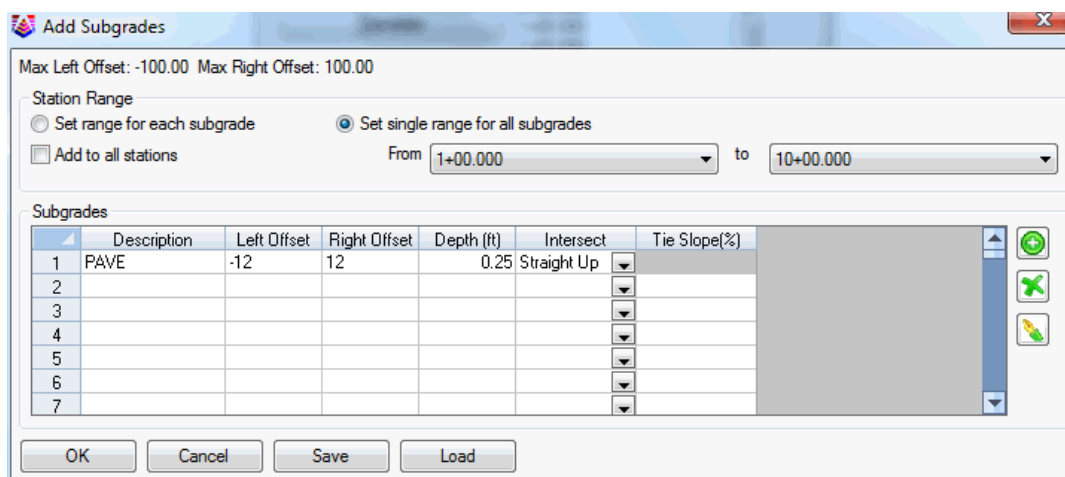


Tie Station: Allows you to tie the outslopes into the reference second section file. This routine first brings up a dialog to specify the range of stations to process. It includes a line to set the slope to tie with. The program will start from the left most offset and use this slope to find the intersection with the reference section file. Then the intersection from the right most offset is calculated with this slope. These intersection points are the tie points. The slope can be defined by percent, ratio, continue the last slope, and vertical.



Add Subgrades: Adds subgrades to the sections with specified depths and offsets. You can add multiple subgrades at a time by filling in the spreadsheet. Each row of the spreadsheet is for a separate subgrade. Each subgrade definition takes a description, left and right offsets, depth and intersection method of either straight up or at a specified slope. The subgrades are added by referencing the existing surface elevation and dropping down the specified depth. The center of the subgrade always drops down vertically. The outside of the subgrade ties in by the specified intersection method. The station range to add the subgrades can be the same of all the subgrades or

specified separately for each subgrade.



Save: Saves the currently loaded section file.

SaveAs: Allows you to save the currently loaded section file as a different file.

Exit: Allows you to exit from the section editor and return to the drawing editor. The program will warn you to save to a file if you have made changes.

Pulldown Menu Location: Sections

Keyboard Command: scted

Prerequisite: None

Draw Section File

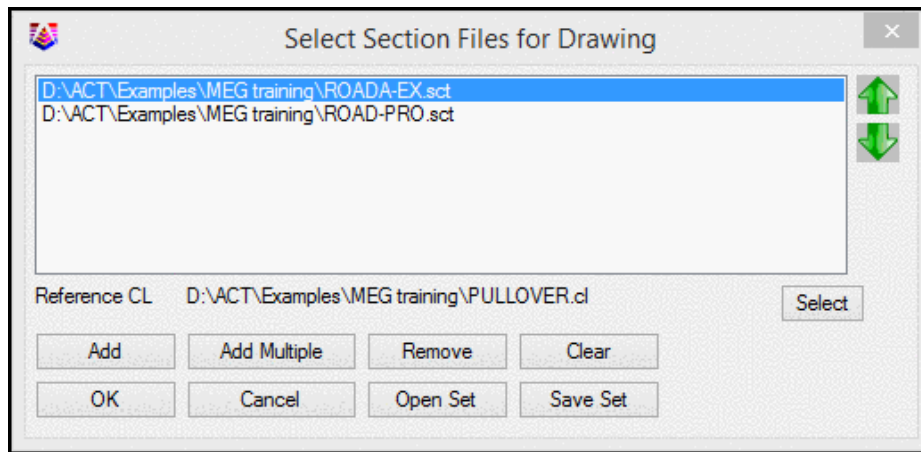
This command generates plots of cross-section data which can be used to further iterate the corridor design or used for construction documentation. The Section files drawn with this command can be created by several methods including the Section File Editor, Digitize Sections, any commands under the Create Sections from... menu, Process Road Design or Road Network commands.

For metric-based projects, please refer to the Drawing Metric Section Sheets section of this document.

The Draw Section File routine will call two primary dialog boxes:

- The first is the Section Files for Drawing dialog box that allows you to specify the Section files (.sct) to be drawn and some general sheet and layer settings.
- The second is the Draw Section File dialog box that allows you to specify various scale, layout and labeling settings.

If the *Type of Plot* option in the Draw Section File dialog box is set to "Sheet," a third Section File Sheet Drafting Parameters dialog will be displayed which provides detailed sheet layout settings.



Add: Specify an unlimited number of Section (.sct) files to plot

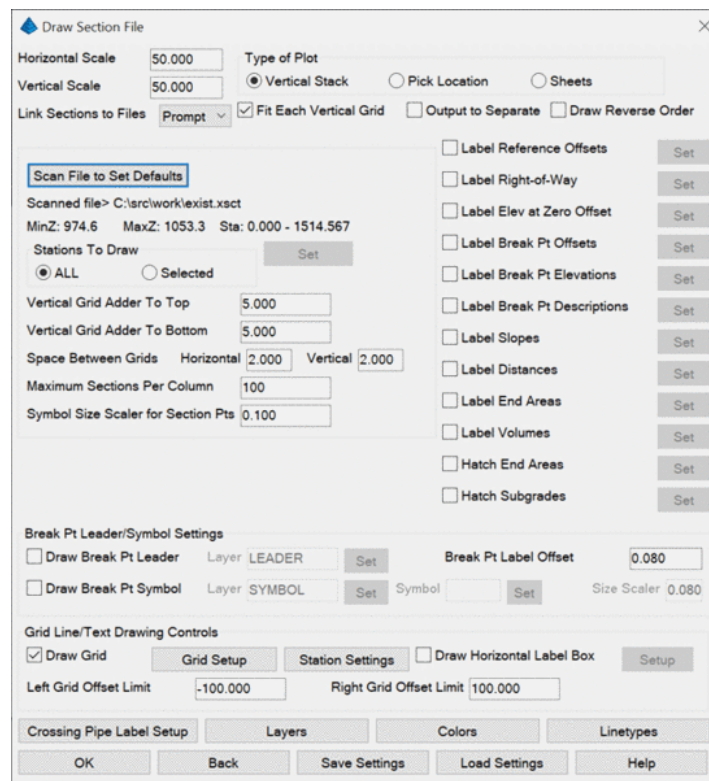
Add Multiple: Add multiple files to plot with a single selection

Remove: Highlight and remove any section file from the selection set

Clear: Clear the selection set (remove all files).

Open Set: Open a preset selection of cross sections(.XST file)

Save Set: Save the current selection set as an .XST file



After specifying the Section Files (.sct) the Draw Section File dialog box opens:

Horizontal Scale: Specify the horizontal scale.

Vertical Scale: Specify the vertical scale. The vertical scale relative to the horizontal scale determines the vertical exaggeration factor.

Link Sections to Files: This setting controls the linkage of the plotted sections to the actual section (.sct) file(s), determining how changes to the file affect the plotted sections.

Type of Plot: Specify the type of Plot you wish to create.

- **Vertical Stack:** Stacks all the cross sections in a vertical row or rows. The cross sections will be drawn in model space and will automatically stack with regard to the parameters set.
- **Pick Location:** allows the user to manually select a location in model space for each individual section.
- **Sheets:** draws the cross sections in either model space or paper space with regard to the settings in the Section File Sheet Drafting Parameters dialog box. When you pick OK on this main dialog, the Sheet Drafting Parameters dialog box appears allowing you to specify all the settings for sheet plotting.

Section File Sheet Drafting Parameters

Choose Space: Indicate whether sheets are to be drawn to **Paper** Space (also known as a Layout) or to **Model** Space. When drawing to Model Space, the Display In Paper Space option will draw the sections in model space and then create layouts with viewports to show the sections.

Layout Name: Indicate the name of the layout to which the first sheet should be drawn.

Use Template Layout: This option allows you to use a layout that exists in the current drawing as the template for layouts created by the command. This option only works for paper space sheet creation.

Add Layout Name To File For Drawing Output: When the Output To Separate Drawing option is active from the

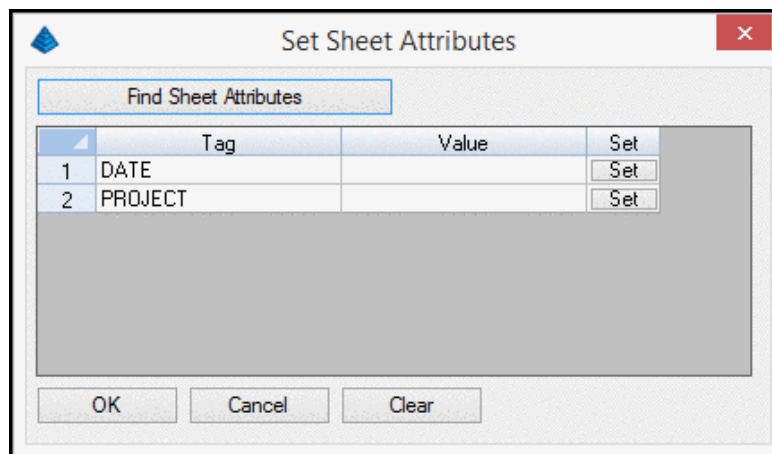
main settings dialog, this option creates separate drawing files for each sheet by appending the layout name to the drawing name.

Tile Sheets: Enabling this option places all sheets in the specified Layout Name. The result is a vertical stack of sheets in the layout. Disabling this option allows additional layouts to be created each containing one sheet. As additional layouts are created, the name of each successive layout is incremented by a value of 1.

Plot at 1:1: Enabling this option draws the sections so that one unit horizontally in the section is equivalent to one plotted unit. The ratio of the Horizontal Scale:Vertical Scale determines the amount of vertical exaggeration.

Block Name: Specify the drawing name that will be inserted for each sheet. The default is SCTSHT1 which is included with Carlson Software and is located in the `%AppData%\Carlson Software\...\Sup\` folder. You can use this or use a sheet block of your own design. The block should be drawn at a 1:1 scale since the program inserts it using the Horizontal Scale setting from the previous dialog. Click the **Set** button to browse/navigate to an alternate drawing file.

Set Sheet Attributes: For grid sheet block names that utilize attributes (useful for items such as sheet numbers, drawn date, drawn by, job name, *etc*), use this command to provide attribute values that will be placed for each sheet block:



Find Sheet Attributes: This routine will scan the Block Name for any attribute definitions and return them to the dialog box so values can be established for each attribute.

Starting Page #: Indicate the starting page number to be applied to the plots through the use of the Set Sheet Attributes command.

Scan Block for Width/Height: Use this routine to scan the specified Block Name for its width and height. These values are populated into the Sheet Width and Sheet Height controls.

Sheet Grid Interval: Indicate the spacing between the grid lines in the sheet block. The routine will not draw the grid lines and uses this information to control the placement of each section onto the sheet.

Vertical Space Between Sheets: Indicate the amount of space that should be placed between sheets when the Tile Sheets option is enabled.

Rows of Sections

Per Sheet: Specify the maximum number of sections that can be stacked on top of each other on a sheet.

Space Between: Specify how much space will be placed between the top of the last section plotted and the bottom of the next section. For U.S. Customary based units, a value of 1 would be a good starting value.

Columns of Sections

Per Sheet: Specify how many columns of sections can be placed on each sheet.

Space Between: Specify the distance between the left edge of one section column edge and the right edge of the next column. This will generally be the area where elevation labels and station circle annotation will be placed. For U.S. Customary based units, a value of 2 would be a good starting value.

Label Grid Zero Offset: Enable this toggle if the zero offset location of each section should be labeled on each section.

Column Order: Controls the station order of the sections for going top to bottom, or bottom to top.

Offset for 1st Section

Horizontal Offset: Specify how far from left edge of the sheet the first section will be placed on to the section sheet. The block SCTSHT1 has a 1" left margin.

Vertical Offset: Specify how far from bottom edge of the sheet the first section will be placed on to the section sheet. The block SCTSHT1 has a 1/2" bottom margin.

Preview: This button allows you to get an approximate idea of what the initial sheet will look like based on the current settings.

Back: This button allows you to return focus to the main dialog and make changes to any previous settings or cancel the routine.

Save Settings: This button allows you to save all the parameters settings to a file so you can easily recall them for another project.

- **Off:** A linkage between the SCT file and the graphical section entities is not formed; you will need to manually re-create section sheets after section design changes.
- **Prompt:** You will be asked whether or not to update the plotted sections when the underlying SCT file is changed.
- **Auto:** The plotted sections will automatically update when the underlying SCT file changes.

Fit Each Vertical Grid: When checked, the grid bottom elevation and grid height are set automatically and you may specify values to add to the top and bottom of each grid (see Vertical Grid Adder to Top and Vertical Grid Adder to Bottom). When not checked, you specify the elevation of the grid bottom and the grid height through the Grid Bottom Elevation and Grid Vertical Height controls, respectively.

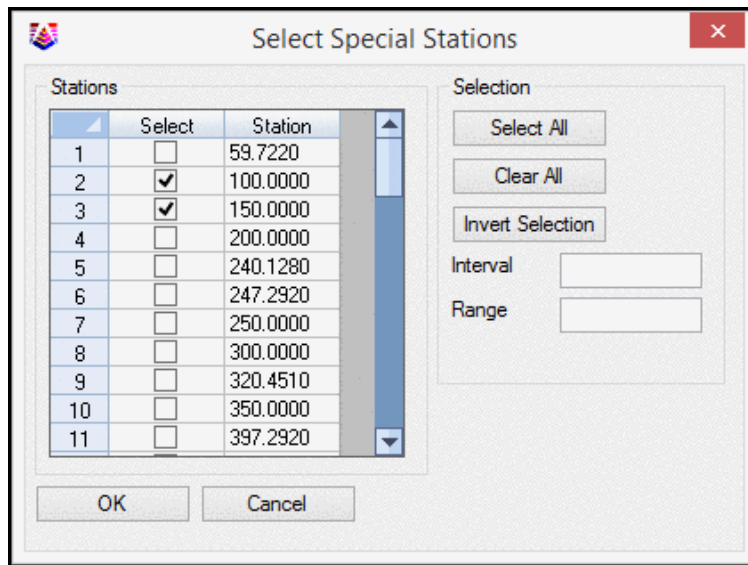
Output to Separate Drawing: When checked, this option will prompt for a New drawing name and location into which all cross sections will be drawn. When using the Vertical Stack method, the program will prompt for whether to output all the sections to the same drawing or create a separate drawing for each station.

Draw Reverse Order: When checked, this option will draw the cross sections in the order of the highest numbered station to the lowest.

Scan File to Set Defaults: This button allows the program to set the minimum and maximum parameters. If you choose this option, the program will automatically set the range of stations, vertical spacing distance, right and left grid distances and starting/datum elevation. This option writes a file called "sectsort.tmp" that is read and used to set the defaults the next time you use the program. Therefore, if you are selecting a different .SCT file to plot you should use this option to update the .TMP file.

Stations to Draw: Select eight All or Selected to specify the range of stations from the file which will be drawn.

- **All:** When selected the program will draw all of the stations in the file.
- **Selected:** This option will allow the user to specify exactly which stations will be drawn via the Set dialog box.



Interval: Specify the interval of stations to draw. For example, perhaps you sampled every 25 feet with the Sections from Surface Model command for more accurate quantities but only want to plot 50 foot stations. ALL is the default value for this field.

Range: Specify a range of stations to include

Vertical Grid Adder to Top: Specify the distance that will be added to the highest elevation of the section for the sheets and pick location options. This option is only available when Fit Each Vertical Grid is checked ON.

Vertical Grid Adder to Bottom: Specify the distance that will be subtracted from the lowest elevation of the section for the sheets and pick location options. This option is only available when Fit Each Vertical Grid is checked ON.

Grid Bottom Elevation: Specify actual bottom elevation for each section grid. This option is only available when Fit Each Vertical Grid is checked OFF.

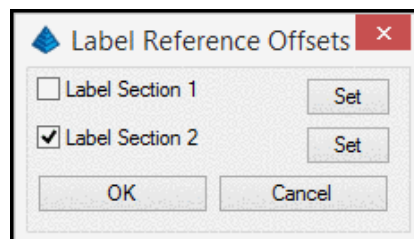
Vertical Grid Height: Specify actual grid height for each section grid. This option is only available when Fit Each Vertical Grid is checked OFF.

Space Between Grids: Specify the **Horizontal** and **Vertical** distance between the sections when they are drawn when the Vertical Stack option is specified.

Maximum Sections Per Column: Sets the maximum number of sections allowed per column when the Vertical Stack option is specified.

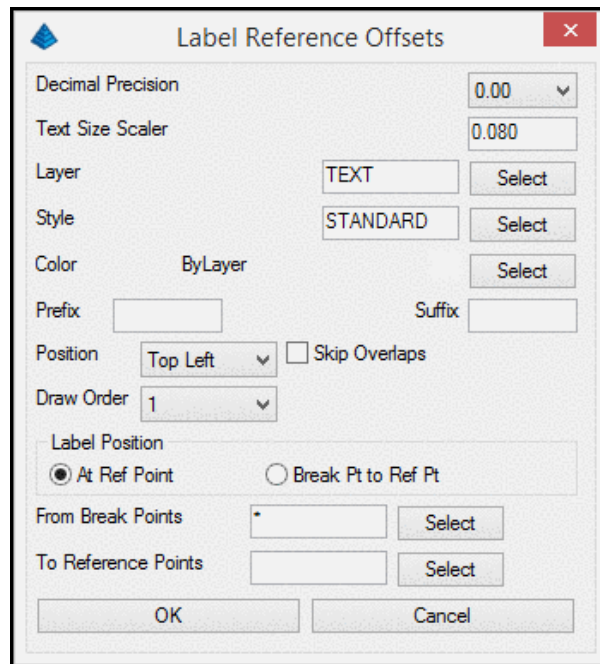
Symbol Size Scaler for Section Pts: Sets the scale as a multiplier of the overall drawing scale for the section points to be drawn.

Label Reference Offsets: When enabled, the offset from selected break points of one section file relative to the position(s) of selected points from another section file can be labeled onto the plots.



Specify which section(s) to draw

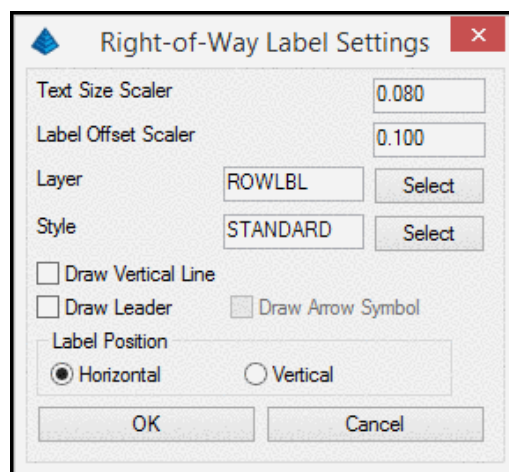
Set: Opens the dialog box for to set the display precision, text size scaler, layer, text style, color, prefix, suffix, position and draw order for these labels.



From Break Points: allows a user based filter to specify break points

To Reference Points: allows the user to control which offsets to label by specifying the reference points from a template (TPL) file.

Label Right of Way: When enabled, this option will label Right of Way points as defined using the Section Points from Right of Way command. Press the **Set** button to the right of this toggle to set the text size and label offset scalers, layer and text style settings.

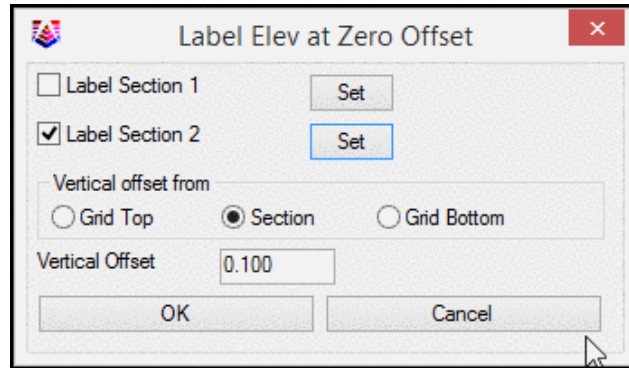


Draw Vertical Line: Places a vertical line, from top to bottom, through the Right-of-Way point.

Draw Leader/Draw Arrow Symbol: When enabled, a short vertical line is drawn, with or without, the arrowhead through the Right-of-Way point.

Label Position: Indicate the desired orientation of the "ROW" text label.

Label Elev at Zero Offset: Will label the section elevation at offset zero.

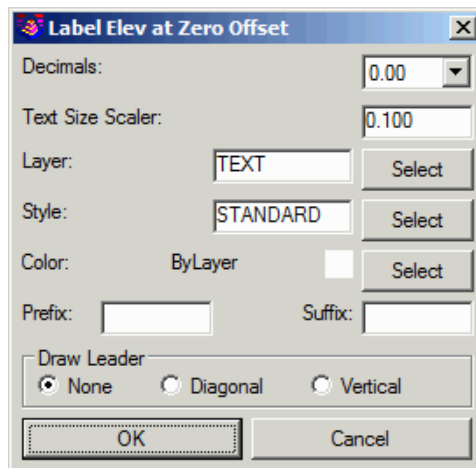


Specify which section(s) to draw

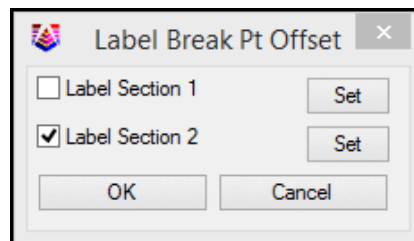
Vertical Offset from: sets the location of the Label relative to the grid or section.

Vertical Offset: controls the offset distance for the label.

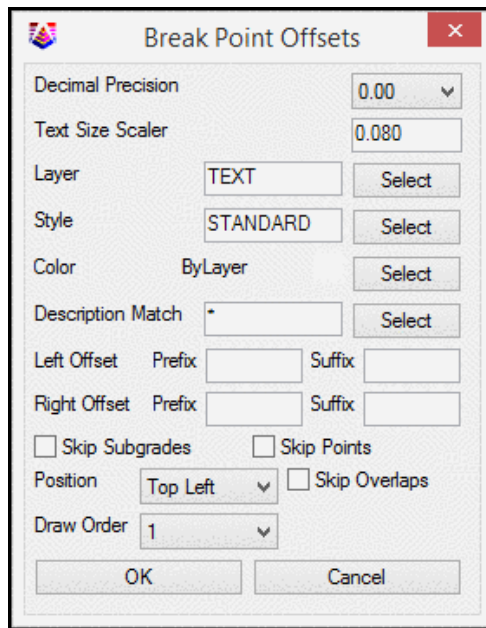
Set: opens the dialog box for to set the display precision, text size scaler, prefix, suffix, color and layer for these labels. The Draw Leader option can be set to None, Diagonal or Vertical.



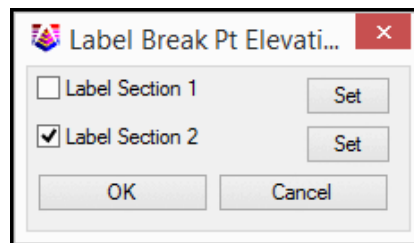
Label Break Pt Offsets: Select the Section(s) to be included in the labeling parameters



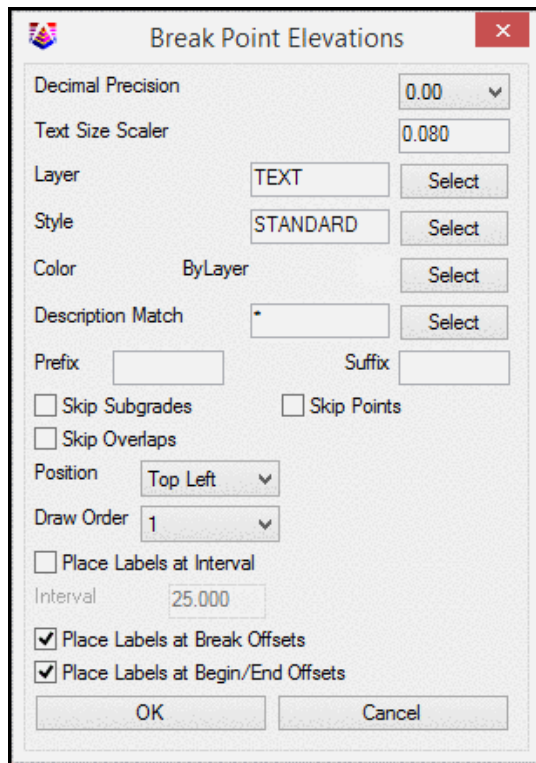
Set: Opens the dialog box for to set the display precision, text size scaler, layer, text style, color, prefix, suffix, and position for these labels. In addition the label position and draw order can be selected.



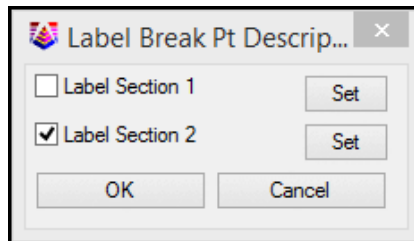
Label Break Pt Elevations: Select the Section(s) to be included in the labeling parameters.



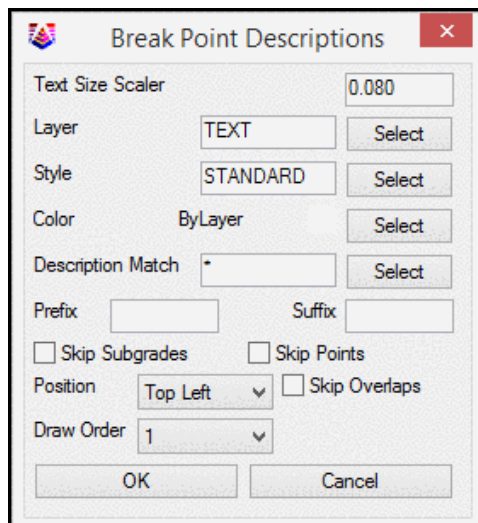
Set: Opens the dialog box to set display precision, text size scaler, Layer, style, color, prefix, suffix, position and draw order for these labels. The Description Match is a way to filter which section points to label. There are also options to toggle whether or not to place labels at the break offsets or Beginning and ending offsets.



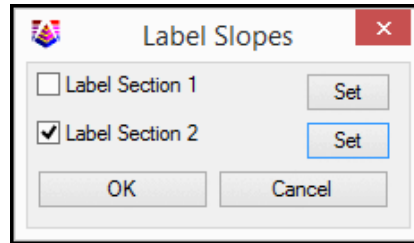
Label Break Pt Descriptions: Select the Section(s) to be included in the labeling parameters.



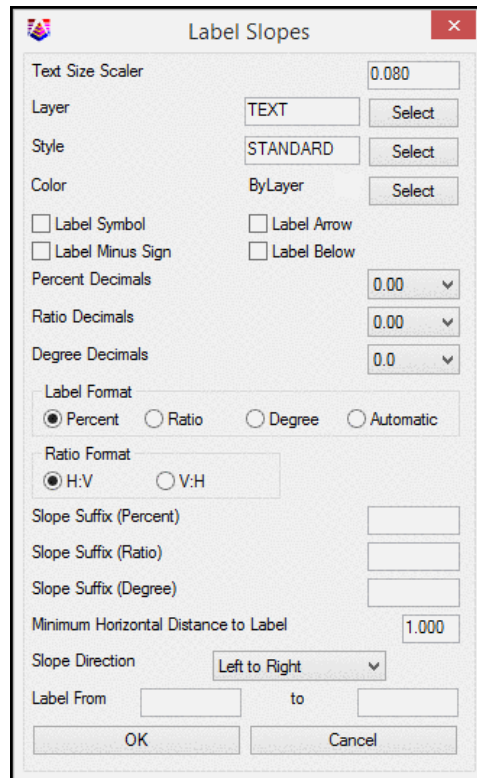
Set: opens the dialog box to set the text size scaler, layer, style, color, prefix, suffix, position and draw order for these labels. The Description Match is a way to filter which section points to label.



Label Slopes: Select the Section(s) to be included in the labeling parameters.



Set: Opens the dialog box to set the text size scaler, layer, style and color for these labels.



Label Symbol: places the % symbol after the slope label

Label Arrow: draws a slope direction arrow next to the label.

Label Minus Sign: places a - in front of negative slope values.

Label Below: places the label below the section line.

Label Format: controls which format the slopes will be labeled in.

Ration Format: Sets either *horizontal to vertical* or *vertical to horizontal* format

Slope suffix (Percent): creates a suffix to the slope label when using the Percent Label Format.

Slope suffix (Ratio): creates a suffix to the slope label when using the Ratio Label Format.

Slope suffix (Degree): creates a suffix to the slope label when using the Degree Label Format.

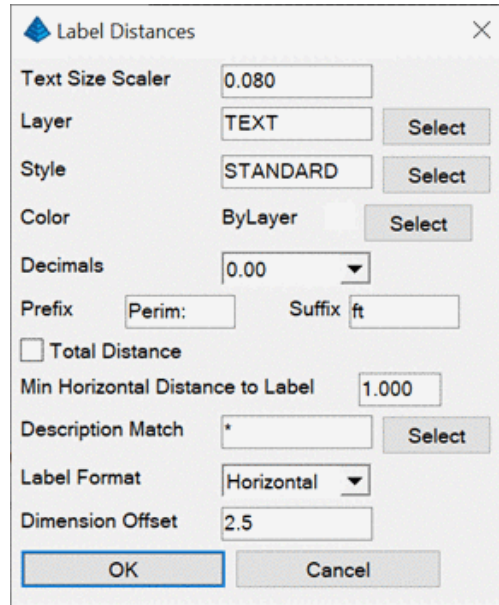
Minimum Horizontal Distance to Label: sets the minimum value of a segment length to label.

Slope Direction: can either be set *left to right* or *right to left*.

Label From: specifies the portions of the section to label a slope. for example, you could specify to only label the

slope between the SW (sidewalk) and SH (shoulder) ID points as defined in the Template file (.tpl) that was used to generate your Section file (.sct).

Label Distances: This option labels the horizontal distances between offset points. There are separate Label Distances settings for each section file. On the first dialog, toggle which section file you want to label distances. Choose the Set button to control the settings for that station. The settings dialog has settings for the distance label size, layer, style, color, decimals and suffix. The **Total Distance** option chooses between labeling just the total distance of the section or individual section segments. The **Description Match** is a way to control which section offset points to label. The **Min Horizontal Distance To Label** setting is a way to filter out short section segments and only label the longer segments. The **Label Format** sets whether to label the distances as a CAD dimension or as regular text rotated by the section slope or horizontal. The **Dimension Offset** controls the vertical distance between the section segment and the label.



Label End Areas: Will label cut and fill end areas on each section. The end areas are either between two sections (existing and design) or with a single section that creates a closed loop like for a tunnel. This dialog box sets the Decimal Precision, text size scaler, layer, style and color of the labels.

The dialog box 'End Area Settings' contains the following fields and options:

- Decimal Precision: 0.0
- Text Size Scaler: 0.080
- Layer: ENDAREA (with Select button)
- Style: STANDARD (with Select button)
- Color: ByLayer (with Select button)
- Cut: Prefix Cut, Suffix
- Fill: Prefix Fill, Suffix
- Skip Zero Labels
- Auto-Center On Section
- Offset: Horizontal 0.000, Vertical 0.000
- Vertical offset from: Top, Bottom
- Use Table
- Title Header: End Areas, Append Station
- Name Header: Material, Value Header: SF
- Include Elevation
- Ground Header: Ground, Final Header: Final
- Include Subgrades
- Subgrade 1: Asphalt, Subgrade 2: Gravel
- Subgrade 3: , Subgrade 4:
- Subgrade 5: , Subgrade 6:
- Buttons: OK, Cancel

Use Table: this option will create a table of the cut/fill values on each section.

Title Header: labels the table with this header. **Append Station** adds the station to the title header.

Cut and Fill label prefix and suffix: allows user defined prefix and suffix added to the label

Auto-Center On Section: this option will center the table or label next to the drawn cross section.

Skip Zero Labels: this option will not label cut or fill values of zero.

Include Elevation: this option will include the centerline elevations in the table. The Ground and Final Headers are user defined.

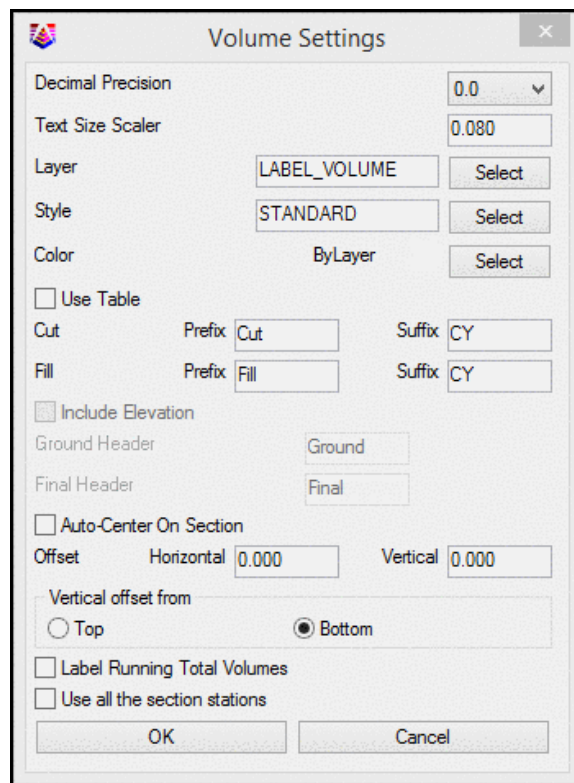
Include Subgrades: labels the end areas of the subgrades with settings for the names of up to 6 subgrades.

Offset Horizontal and Vertical: manually controls the placement of the table or label.

Vertical offset from: specifies where the offset dimensions are to be referenced to.

End Areas	
Material	SF
Cut	387.5
Fill	0.0
Asphalt	2.4
Gravel	8.0
Stone	6.7

Label Volumes: Will label cut and fill volumes on each section. The volumes are measured between the current station and previous station. There are settings to control the format and placement of the labels. This dialog box sets the Decimal Precision, text size scaler, layer, style and color of the labels.



Use Table: option will create a table of the cut/fill values on each section.

Cut and Fill label prefix and suffix: allows user defined prefix and suffix added to the label.

Include elevation: when enabled, this option will include the centerline elevations in the table. The Ground and Final Headers are user defined.

Auto-Center On Section: this option will center the table or label next to the drawn cross section.

Offset Horizontal and Vertical: manually controls the placement of the table or label.

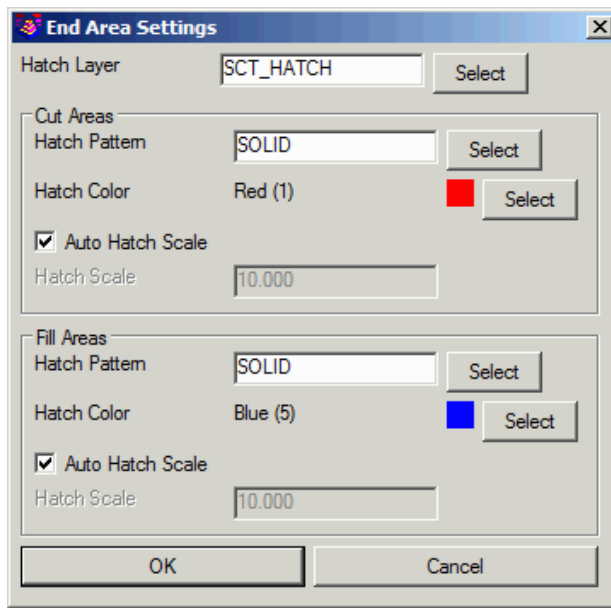
Vertical offset from: specifies where the offset dimensions are to be referenced to.

Label Running total Volumes: will label the accumulative total volumes with each station.

Use all the section stations: chooses between using all the section stations to calculate volumes or only the stations being drawn.

Hatch Subgrades: This option hatches the subgrade areas of design section using the section points with descriptions starting with SUBGRADE. There are settings for up to four different hatch patterns.

Hatch End Areas: This option hatches the cut/fill areas between the first and second section files. The program treats the first section as existing and the second as design for determining cut verses fill. There are separate hatch pattern, color and scale settings for cut and fill.



Draw Break Pt Leader: Enable this option to include a leader with the Label Break Pt Offsets, Label Break Pt Elevations or Label Break Pt Descriptions options. Click the **Set** button to specify the desired layer for the leader.

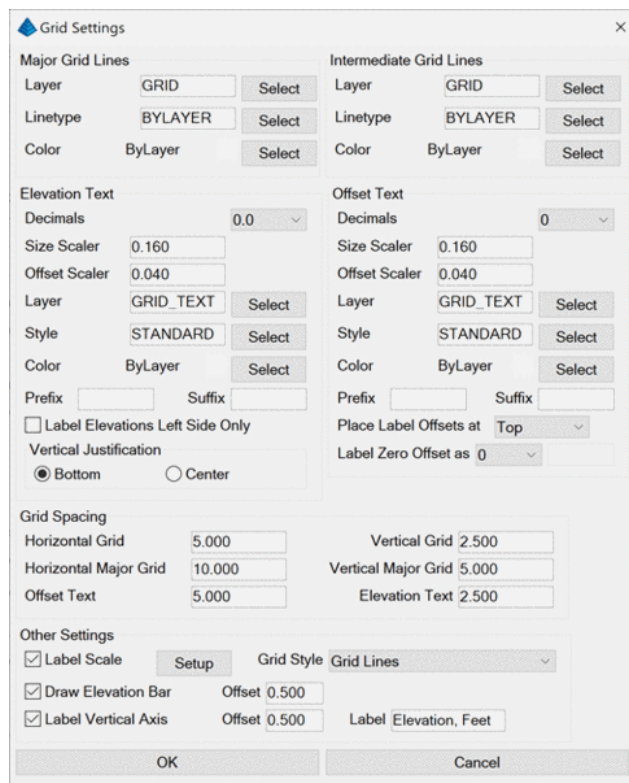
Note:

- When redrawing sections, the program retains any custom edits to label and leader positions.

Draw Break Pt Symbol: Enable this option to include a symbol with the Label Break Pt Offsets, Label Break Pt Elevations or Label Break Pt Descriptions options. Click the **Set** button (to the immediate right of the Layer control) to specify the desired layer for the symbol. Click the **Set** button (to the immediate right of the Symbol control) to specify the desired symbol and indicated the desired Size Scalar.

Break Pt Label Offset: Indicate the desired offset amount from the surface break point to its label.

Draw Grid: When enabled each cross section is drawn on a grid. The **Grid Setup** is used to customize how the grid will be drawn.



Main Grid Lines: Sets the Layer, Linetype and Color for the Major Grid Lines as set below

Intermediate Grid Lines: Sets the Layer, Linetype and Color for the Intermediate Grid Lines. (Those not defined as major)

Elevation Text: this section contains controls for setting the elevation Precision, text Size Scaler, Offset Scaler, Layer, text Style, and color, prefix and suffix for the elevation labels.

Offset Text: this section contains controls for setting the offset Precision, text Size Scaler, Offset Scaler, Layer, text Style, and color, prefix and suffix for the offset labels.

Label Elevations Left Side Only: when enabled, elevations will be labeled on the left side of the sections only.

Vertical Justification: this sets the justification of the elevation text.

Place Label Offsets: **Top** will place the labels at the top of the grid, **Bottom** will place along the bottom of the grid and **None** skips drawing the offset labels.

Grid Spacing: sets the horizontal and vertical spacing for the grid lines as well as the horizontal and vertical Major Grid lines the offset text (grid spacing) and the elevation text.

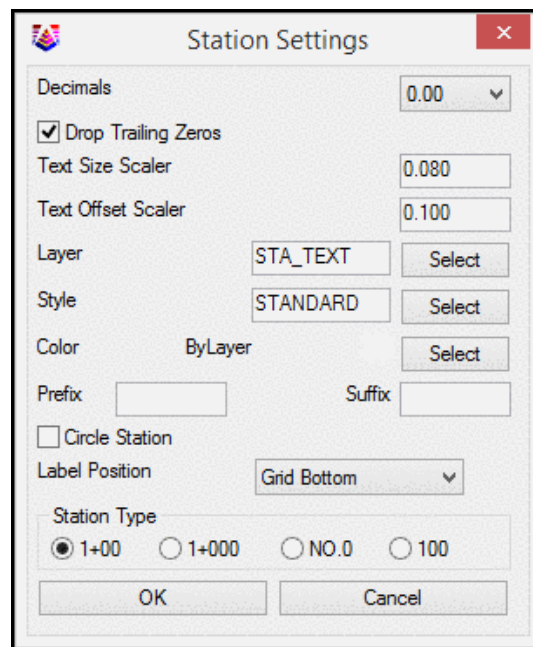
Label Scale: when enabled will label the horizontal and vertical scale.

Grid Style: sets the type of grid lines to be drawn. Grid Lines, Ticks Only, Ticks and Dots, Ticks and Checks, Text Only, Grid Lines and Dots, or Grid Lines Below 1st Section. This last option trims the grid lines by the first section so that the grid lines are drawn only below the section.

Draw Elevation Bar: places the elevations on a bar offset from the grid at a user defined location.

Label Vertical Axis: draws a label vertically along the vertical axis.

The **Station Settings** button displays another dialog box for the station label settings including decimal places, size, layer, style, color, prefix, suffix, format type and position. When the section file has descriptions assigned to the sections, the descriptions are labeled under the station position value. The **Circle Station** option will draw a circle around the station label.



Draw Horizontal Label Box: Enabling this option will draw a table with desired labeling above or below each cross-section. By picking the **Set** button to the right, you can choose the data to be placed in the table. The Elevation, Offset and Description of each point on the cross section can be added to the table. If more than one Section file (.sct) is being drawn on the cross-section, you will also have the option of displaying the elevation difference between sections.

Box Placement: Chooses between drawing the label box above or below the section grid.

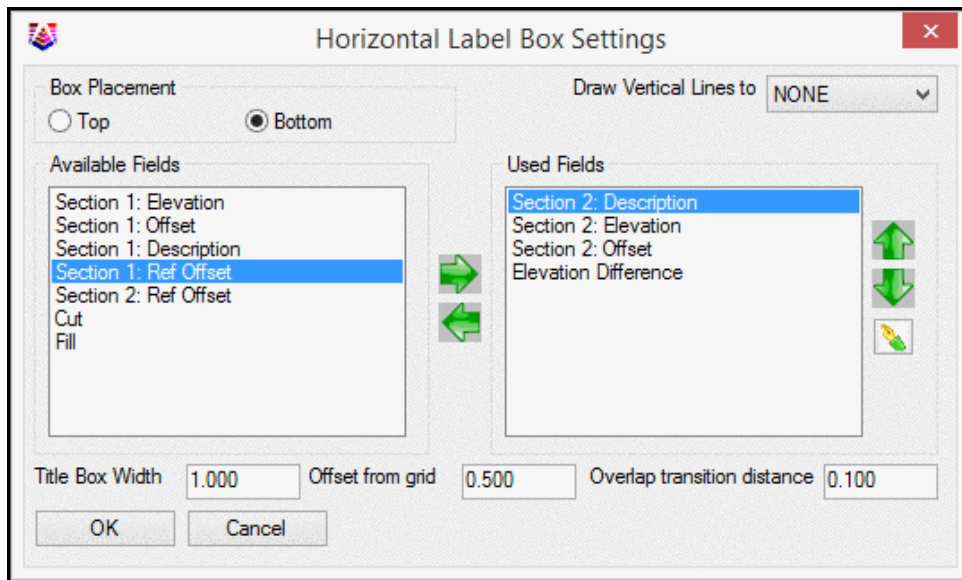
Draw Vertical Line: Sets drawing a vertical line from the section data point to the specified row in horizontal label box.

Title Box Width: Sets the size for the field name in the label box.

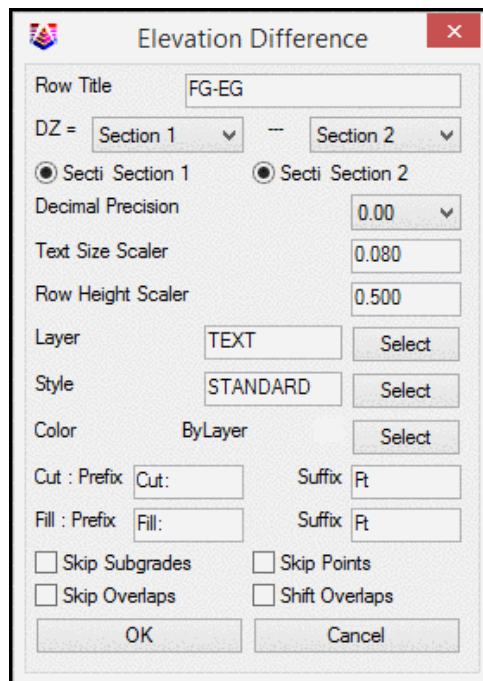
Offset From Grid: Controls the separation distance between the section grid and the label box.

Overlap Transition Distance: When drawing vertical lines and the labels are shifted to prevent overlap, this setting controls the length of the line for shifting over.

Label Elevation at Interval of Top Section: When labeling multiple values such as elevations for different sections, this option uses the stations from the section points of the top section as the stations to label for all the sections.



In the *Draw Horizontal Label Box* dialog, select from the **Available Fields** in the list on the left to populate the list of **Used Fields** on the right side. Once an item has been moved to the list of **Used Fields**, you can double-click on the Field to change settings and format for each Field. An example of the **Elevation Difference** option is shown below:



The **Row Title** for each field can be edited from the default to show a descriptive title. The **DZ** value in the Elevation Difference settings dialog allows you to specify which Section's elevations are to be subtracted from the other. This setting is critical to return the correct cut and fill depth values. In all field settings boxes, you have the ability to skip surface points in order to make the data more legible.

Place Labels at Interval: This option creates labels at the specified offset interval.

Place Labels at Break Offsets: This option creates labels at each section point.

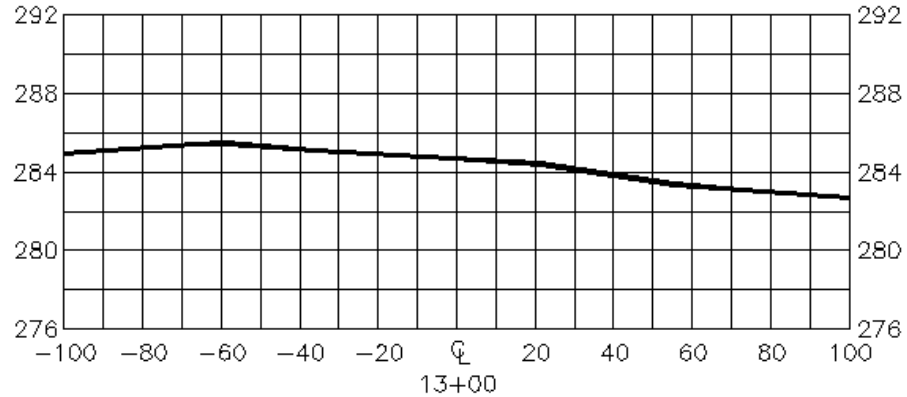
Skip Subgrades: Enable this option to skip all subgrades as may have been defined in Design Template files (.tpl).

Skip Points: Enable this option to skip points in the Section file (.sct) that were created using any of the **Create Section Points...** commands.

Skip Overlaps: Enabling this option will cause any overlapping text in the table to be skipped. Having this option enabled will disable the Shift Overlaps option.

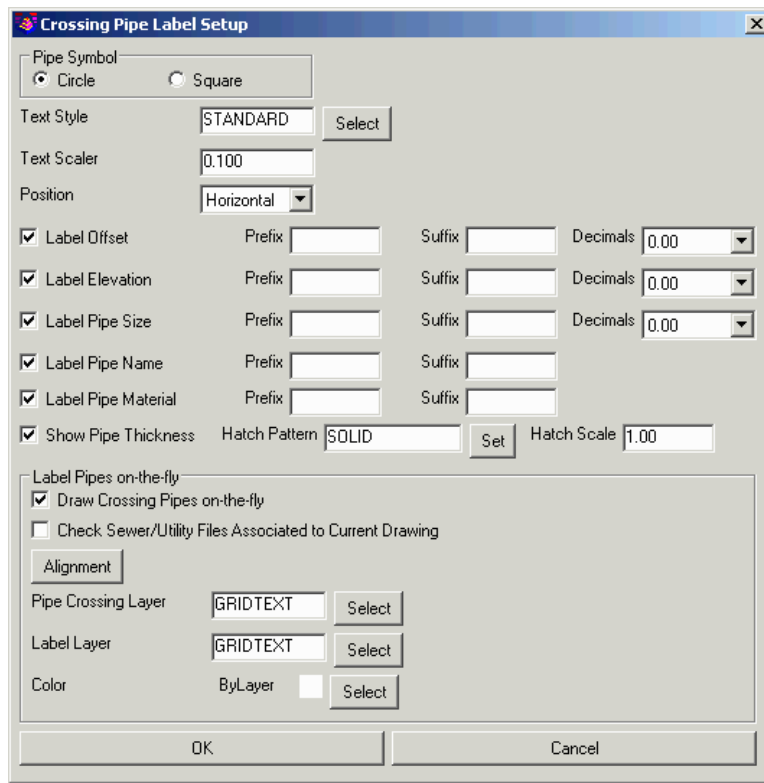
Shift Overlaps: Enabling this option will shift any text in the table to the right so that it does not overlap preceding text.

A sample cross-section with **Horizontal Label Box** is shown below:



EG : Elevation	284.92	285.42	285.48	285.08	284.70	284.42	283.37	283.07	282.70
EG : Offset	-100.00	-66.78	-58.61	-36.29	-1.24	21.45	55.51	72.66	100.00

Crossing Pipe Label Setup



Pipe Symbol: Choose whether to show the Pipe Crossing symbol as a circle or a square. When the sections are drawn with vertical exaggeration by different horizontal and vertical scales, then the circles or squares will be stretched into ellipses or rectangles.

Text Style and Text Scaler: Specify the text style and size of Pipe Crossing labels.

Position: Chooses between creating horizontal or vertical labels.

Label Offset, Label Elevation, Label Pipe Size, Label Pipe Name, Label Pipe Material: Enable any or all of these options to label the distance left or right off the alignment (Label Offset), the invert elevation, pipe size and pipe name of each crossing pipe. Use the optional settings for specifying "Prefix" or "Suffix" text and use Decimals to set precision for each label.

Show Pipe Thickness draws the pipe thickness around the crossing symbols using the specified hatch.

Draw Pipe Crossings on-the-fly: Enable this option to have Crossing Pipes that have been created using a Sewer Network file (.sew) or Draw Pipe 3D Polyline command drawn in cross sections. It is not necessary to enable this option if Pipe Crossings have been saved to a Section file (.sct) using the Section Points from Pipes command.

Check Sewer/Utility Files Associated to Current Drawing: This option looks for crossings with the pipes in .sew and .util files for the current drawing. To review and edit which files are in the current drawing, use the File > Drawing Explorer command. The program will show a list of files to choose from for checking.

Alignment: Pick this button to select either a Centerline file (.cl) or Section Alignment file (.mxs) to scan for Crossing Pipes.

Layer and Color: These settings specify the layer and color of the Pipe Crossing symbol.

Prompts

If the Pick Location option was specified, the program scans the station data and determines the minimum and maximum elevations, and proposes a datum elevation. If you have pre-plotted a grid sheet and want to reference another local grid coordinate, then change the datum elevation appropriately. The Pick Location type of plotting has the following prompts:

Station> 25.000 Min Elev> 1055.301 Max Elev> 1057.068

Change datum elev/<Select point that represents 0 offset elev 1050.0>: Pick a point

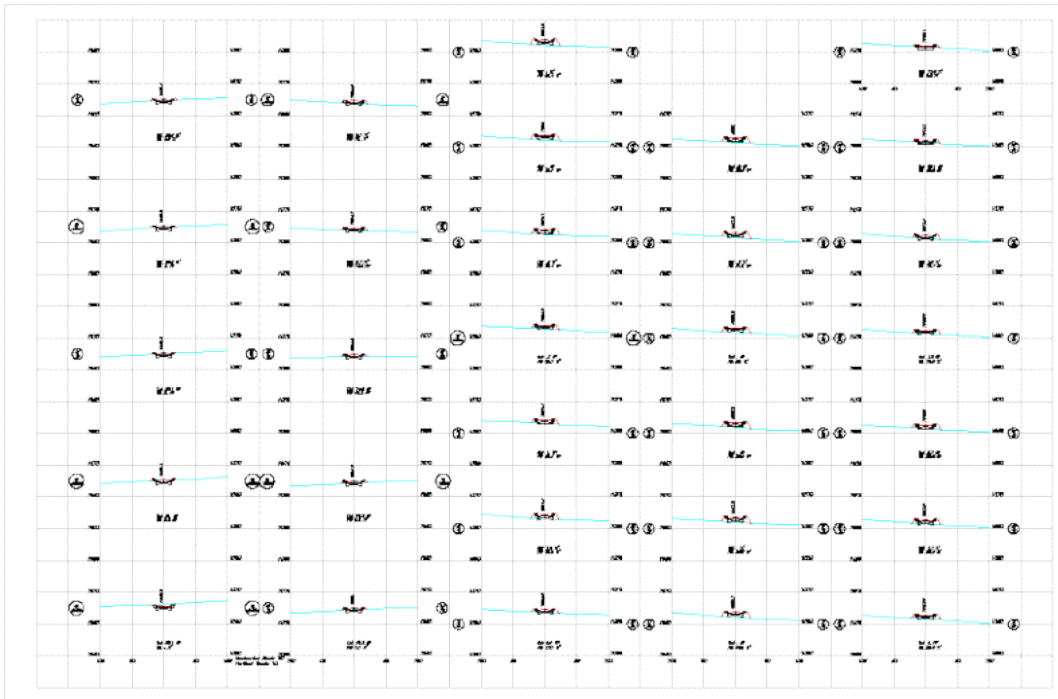
Station> 50.000 Min Elev> 1055.557 Max Elev> 1057.324

Change datum elev/<Select point that represents 0 offset elev 1050.0>: Pick a point

The program continues to prompt until the last station in the range specified is drawn. You can use the Cancel function (the *Esc* key) to stop plotting, if necessary.

If the Sheets option was specified with Model space as the destination, you can choose where to insert the sheet(s):

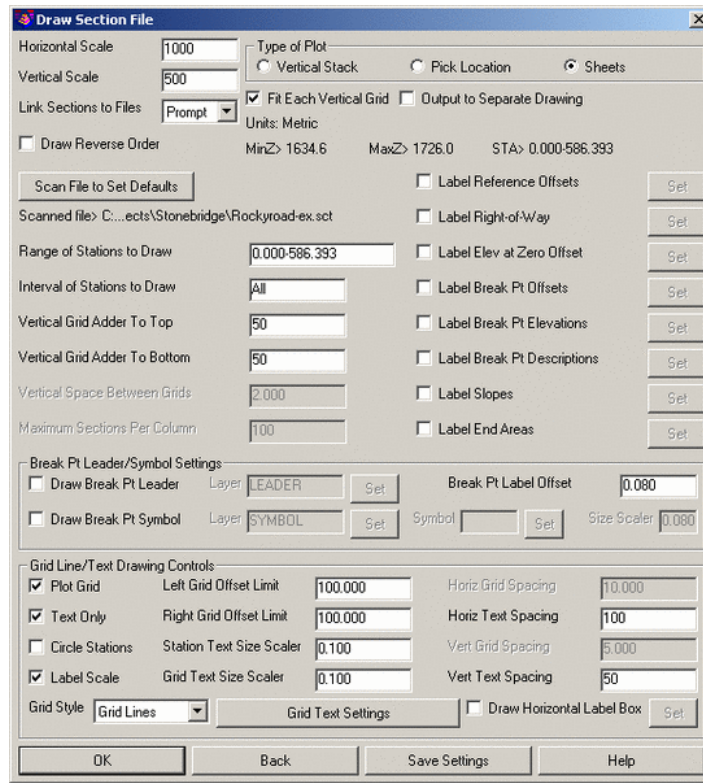
Select Starting Point for Row of Sheets <0.0,0.0>: Pick a point or press Enter to accept the default value specified



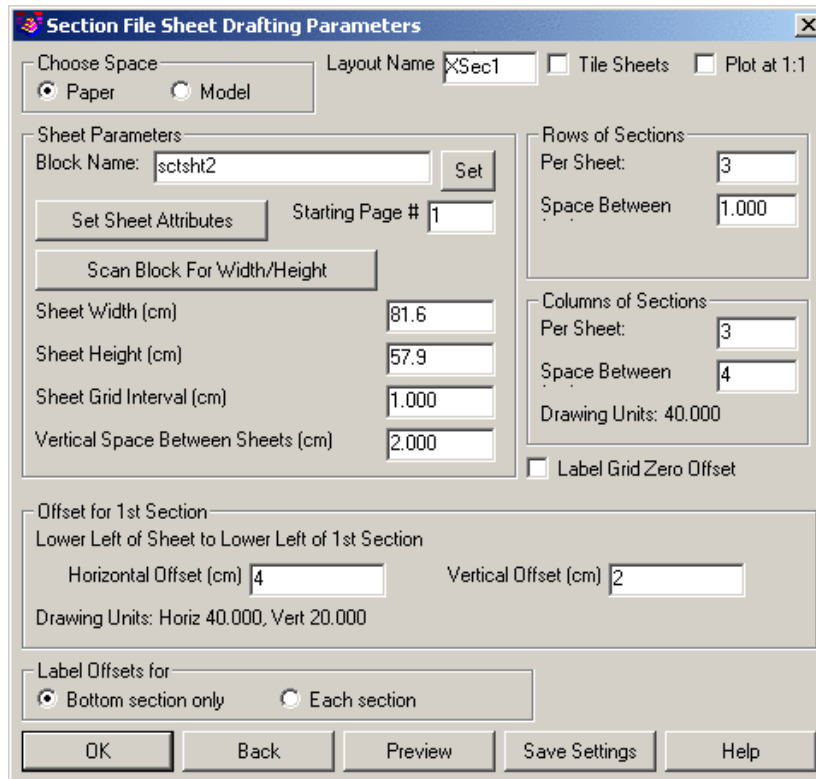
Sheet Sample

Drawing Metric Section Sheets

First, be sure that you are set to metric mode in Drawing Setup under the Settings menu. For our example, assume a 1:1000 horizontal scale. Once set, issue the Draw Section File command and click OK to reach the second dialog. There is a different block name for metric sections called schsht2.dwg which is located in the **%App-Data%\Carlson Software\...\Sup** folder. Begin by setting the parameters for the second dialog as shown.



Second dialog with metric settings

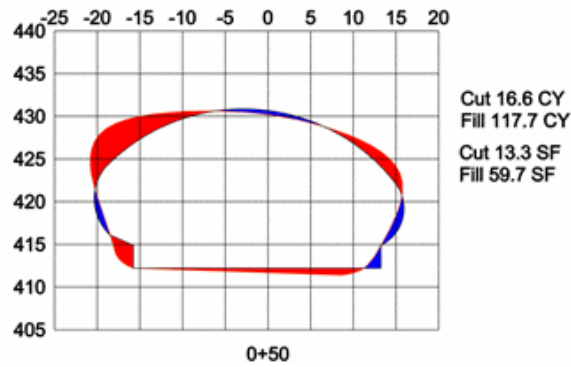


Third dialog with metric settings

Adjust settings as needed to achieve the desired look/layout.

For sections that make a closed loop such as for tunnels, the Label Volumes, Label End Areas and Hatch End Areas will use the difference between the sections. When you have one section for the tunnel design and another section

for the actual, these features will show the over cut and under cut amounts.



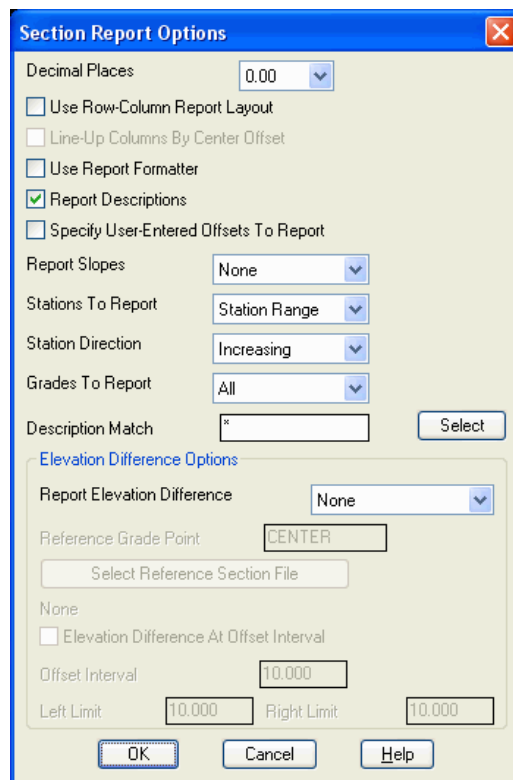
Pulldown Menu Location(s): Civil > Sections, Field > Roads

Keyboard Command: drawsct

Prerequisite: A Section (.sct) file

Section Report

This command generates a report of a section file for the specified stations. The information contained in the report is determined by the settings in the Section Report Options dialog box.



Decimal Places: Specify the display precision for stations and elevations.

Use Row-Column Report Layout: When checked, offsets are reported in columns. Example reports showing the difference are shown below. Also when active, there is an option to **Line-Up Columns By Center Offset** which makes the zero offset column line up. Otherwise, the columns are lined up by the left most offset.

Use Report Formatter: Report output is directed to the Report Formatter which allows for custom reports, as well as being able to export the report to Microsoft Excel or Access.

Report Descriptions: Controls whether the descriptions for each section point are reported.

Specify User-Entered Offsets To Report: After choosing OK from this dialog, the program will prompt for additional offsets to report with interpolated elevations. These are for offsets that don't already exist as section points in the section file.

Report Slopes: Will report the slope between section points. Specify how to report the slopes, either none, percent, ratio, or auto format. Auto format means that slopes less than 10% are reported in percent, while greater slopes are reported as ratios.

Stations to Report: Specify either a range and interval of stations to report or enter each station one at a time.

Station Direction: This setting controls the order of the stations for the report.

Grades to Report: This applies to section files that contain subgrades. For these section files, you can choose which grades to report (top surface or subgrades). All is also an option.

Description Match: This field can be used to filter the section points by their description.

Report Elevation Difference: Reports section elevations by Reference Grade Point, Section File or Surface File. The Surface File method works with a triangulation file (TIN) and can be used to compare a 3D TIN model with the design sections.

Reference Grade Point: Specify the reference grade ID. Only available if Grade Point option is selected, as mentioned above.

Select Reference Section File: Specify a reference file. Only available if Section File is chosen, as mentioned above.

Elevation Difference at Offset Interval: Used if there is an elevation difference.
The next three options only available if Elevation Difference at Offset Interval is clicked.

Offset Interval: Value required.

Left Limit/Right Limit: Values required.

Prompts

Section Report Options dialog *choose options*

Section File to Report dialog *choose existing file*

Starting station for report <0.000>: *press Enter*

Ending station for report <1147.478>: *press Enter*

Station interval (A for All) <100.0>: *press Enter*

```

=====
Row-Column Layout ON
=====
Section Report      05/15/2002 18:31
File: C:/scadxml/DATA/125.sct

STATION

16+78.12   -260.00  -259.65  -244.64  -234.32  -213.99
          1401.30  1401.30  1400.92  1400.77  1400.62
          EXTRAPOL 0          0          0          0

```

```

=====
Row-Column Layout OFF
=====
Section Report      05/15/2002 18:31
File: C:/scadxml/DATA/125.sct

STATION: 16+78.12
-260.00    1401.30    EXTRAPOLATED
-259.65    1401.30     0
-244.64    1400.92     0
-234.32    1400.77     0
-213.99    1400.62     0
-209.00    1400.71     0
-185.27    1401.44     0
-183.67    1401.49     0
-183.33    1401.50     0
-158.34    1401.37     0

```

Sample Report

Pulldown Menu Location: Sections

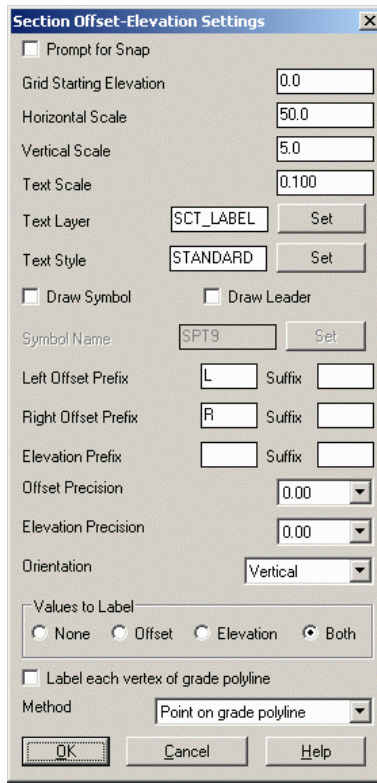
Keyboard Command: sctprnt

Prerequisite: A section file (.sct)

Offset & Elevation Report/Plot

This command calculates the offset and elevation at points along a polyline on a section grid. The results can be drawn on the grid or just displayed on the text screen. The Method setting controls whether the offset and elevation are either calculated for each vertex of the polyline, at picked points along the polyline or at user specified points on the grid. This command can also be used as a section inspector. As you move the cursor across the section, the offset, elevation and slope are reported in real-time in a pop-up window.

The Prompt For Snap toggle controls whether the command will present the snap dialog as you pick points to figure the offset and elevation at. The Grid Starting Elevation edit box allows you to input the beginning elevation of the local grid that you are designing in. Use the Scale edit boxes to set the proper horizontal and vertical scales for your design environment. The Label each vertex of grade polyline option will draw the offset-elevation label above each point in the selected polyline. There are also settings to control the layer, size, style, orientation, prefix, suffix and decimal precision for all the labels.



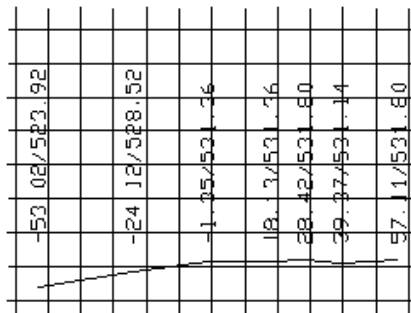
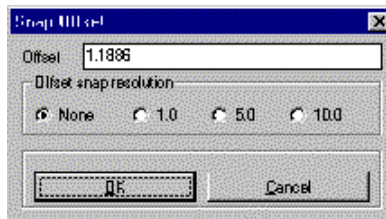
Prompts

Section Offset-Elevation Settings dialog Choose the scales and base elevation that match your section grid.

Pick center grid point [int on]: Pick the grid point at the zero offset and base elevation. The intersection osnap mode is on.

Pick grade polyline: *select polyline*

Pick vertical alignment for text: *pick point above the polyline*



Offset & elevation at each polyline vertex

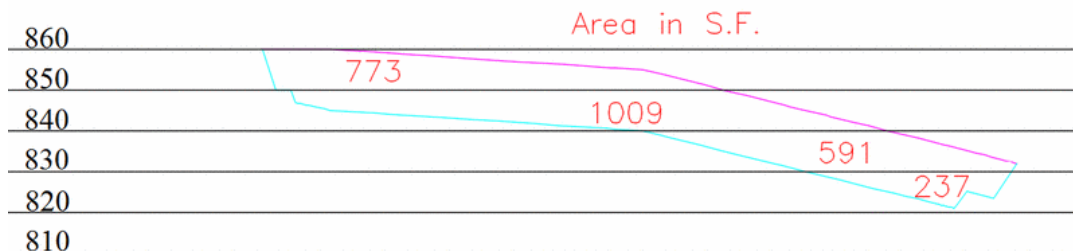
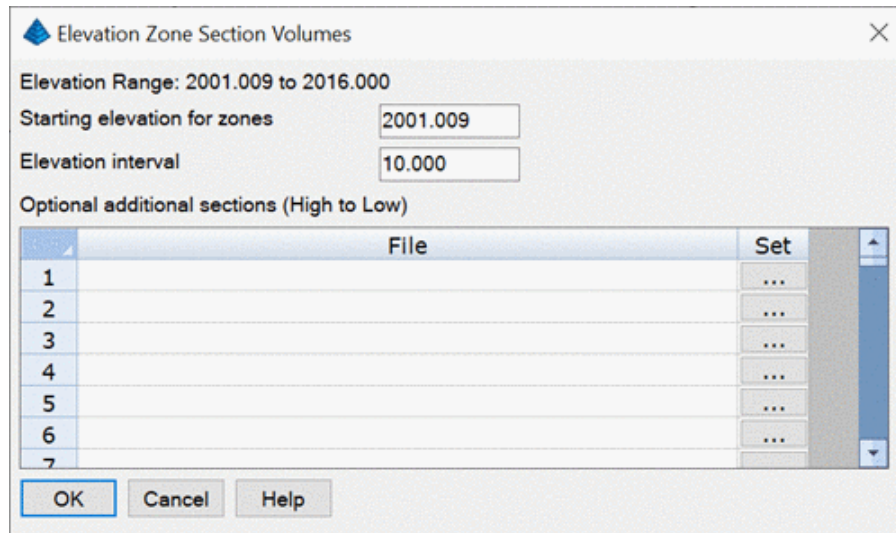
Pulldown Menu Location: Sections

Keyboard Command: offelev

Prerequisite: Must plot the polyline that represents the grade

Elevation Zone Section Volumes

This command calculates section volumes between two section files and reports the volumes broken out by elevation zone. After selecting the bottom and top section files, there's a dialog to set the starting elevation for the zones and the elevation interval. There is also the option to supply additional section files for reporting volumes for additional materials.



Station	Material	LTW	Area (S.F.)
0+000	Coastal	8200-8200	0
0+000	Coastal	8200-8200	237
0+000	Coastal	8200-8200	591
0+000	Coastal	8200-8200	1009
0+000	Coastal	8200-8200	773

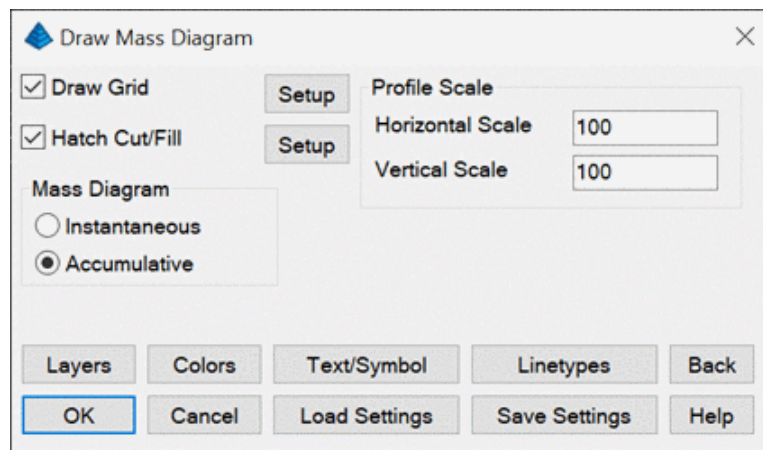
Prompts

Bottom Section File *select section file*
Top Section File *select section file*
Elevation Zone Section Volumes dialog

Pulldown Menu Location: Sections > Cut/Fill Analysis
Keyboard Command: xsctdz
Prerequisite: Existing and Design section files

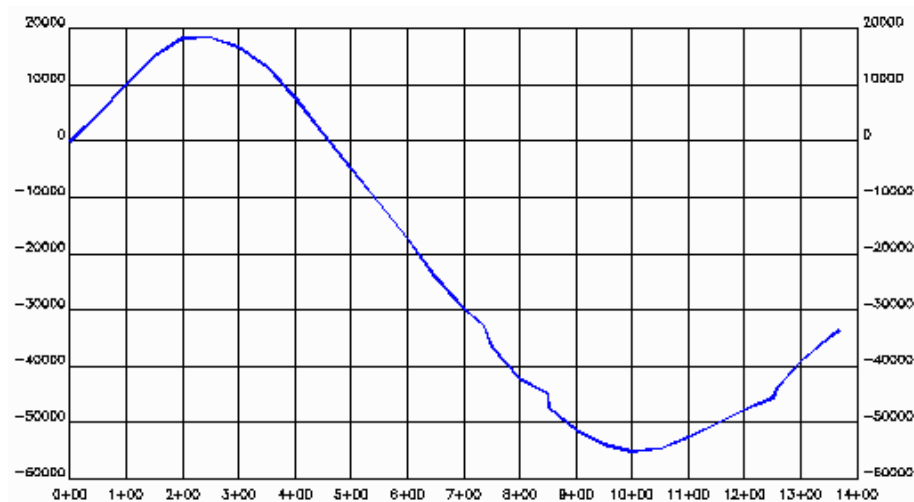
Draw Mass Diagram

This command draws a mass diagram uses data from a .MAS file created by commands such as Process Road Design, Calculate Section Volumes or Edit-Process End Areas. The diagram options are set in the dialog shown here. The Mass Diagram method of Instantaneous graphs the difference in cut/fill at each station, and the Accumulative method graphs the running total cut/fill difference starting from the first station. The other settings are a subset of the settings in the Draw Profile command. Please see the Draw Profile section of the manual for a description of these settings.



Prompts

Mass Diagram File to Read dialog *choose .MAS file*
Draw Mass Diagram dialog Make selections.



Pulldown Menu Location: Sections > Cut/Fill Analysis

Keyboard Command: drawmas

Prerequisite: .MAS file

Mass Diagram Report

This command creates a report for mass diagram data from a .MAS file created by commands such as Process Road Design, Calculate Section Volumes or Edit-Process End Areas. The Report Formatter is used to specify the layout of the report with options to output to Excel and databases. The report includes the stations and accumulated cut/fill volumes. Positive amounts indicate more fill than cut and negative is for more cut than fill.

Prompts

Mass Diagram File to Read dialog *choose .MAS file*

Number of decimal places <2>: *press Enter*

Report Formatter dialog Make selections.

Mass Diagram

File: C:\sample\simo2.mas

Station Mass

0+00.00 17.02
0+50.00 4789.41
1+00.00 10174.48
1+50.00 15215.27
2+00.00 18363.56
2+50.00 18467.06
3+00.00 16772.88
3+50.00 13227.30
4+00.00 7898.93
4+50.00 1418.09
5+00.00 -4995.74

Pulldown Menu Location: Sections > Cut/Fill Analysis

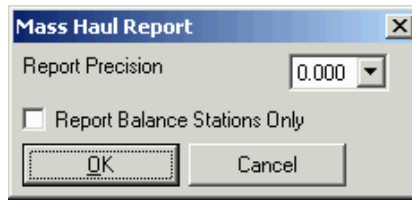
Keyboard Command: masreport

Prerequisite: .MAS file

Mass Haul Report

This command reports the mass haul amounts and stations of balance. Before running this command, the mass haul data file must be created with another routine such as Process Road Design, Calculate Section Volumes or Mass Haul Analysis. The mass haul data file is a profile (.pro) format with stations and mass haul volume instead of the stations and elevation of the typical profile.

The Mass Haul Report has the stations and accumulated cut/fill volume balance up to the stations. The Report Balance Stations Only option makes a report for the stations where the cut/fill is balanced.



Mass Haul Report

File: C:\SAMPLE\MASSHAUL.PRO

Station Balance

0+09.226 0.000

1+53.409 0.000

7+04.560 0.000

11+97.656 0.000

Pulldown Menu Location: Sections > Cut/Fill Analysis

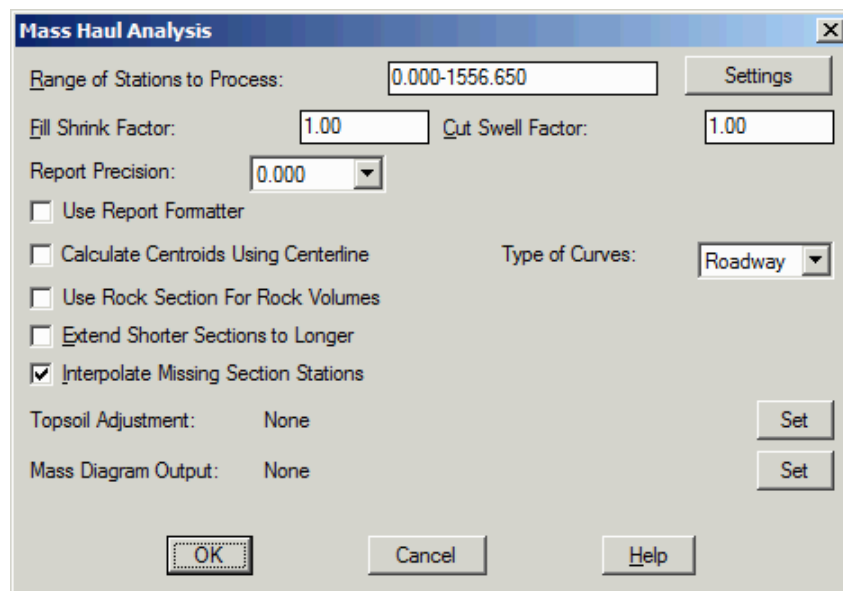
Prerequisite: Mass haul data file (.pro)

Keyboard Command: mhreport

Mass Haul Analysis

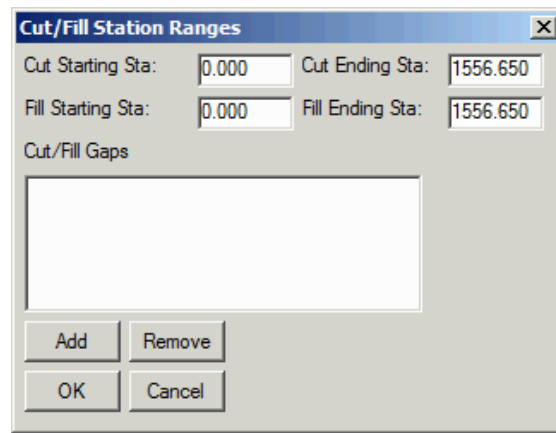
This command will determine the volume and haul distance for each group of net cut and net fill station ranges along a road. The program calculates the optimized cut to fill movements so that the total volume-distance moved is minimized.

You will first be prompted to select the Existing Ground section file and the Design Surface section file or on End Area (.ew) file. These files will be used to determine the Mass Haul quantities. If you do not have either of these files, you can create them using the different Create Sections commands under Roads. After you selected your section (.sct) files or (.ew) file, the following dialog will appear.



Range of Stations: The program will pick up the range of stations determined by your section files. In this field, you can modify the range of stations to process. The Settings button brings up another dialog with more station

options:



Cut/Fill Starting/Ending Stations: The Cut and Fill Starting and Ending Stations are for tapering the end areas at the start and end of the section range down to zero beyond the station range.

Cut/Fill Gaps: Use the Add and Remove buttons to define a series of station ranges for cut/fill gaps where the program will not calculate any volumes.

Shrink/Swell Factors: The Shrink Factor is multiplied by the fill quantities and the Swell Factor is multiplied by the cut quantities.

Report Precision: This setting controls the number of decimal places to use in the report.

Use Report Formatter: The Report Formatter will allow you to customize the information reported by the Mass Haul Analysis.

Calculate Centroids Using Centerline: This option will find the center offset for each Cut/Fill area and use a centerline to adjust the station interval along curve segments for the end area volumes.

Use Rock Section For Rock Volumes: This option will use a third section file for reporting rock cut quantities.

Extend Shorter Sections to Longer: This option will find your longest section and match the length of all your other sections to it.

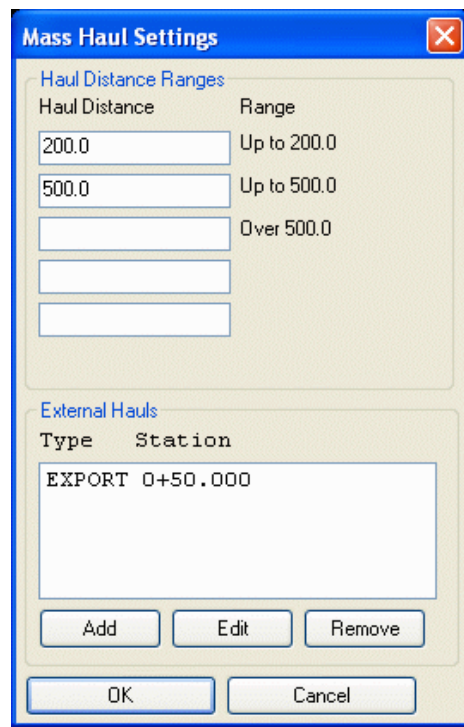
Interpolate Missing Section Stations: Toggle this on to interpolate any missing stations so that the Mass Haul report can use all the stations.

Topsoil Adjustment: This will apply a Topsoil Removal/Replacement definition from the Template Adjustments to adjust the sections.

Mass Diagram: This will create a Mass Diagram of the cut/fill balance by station. This data is stored in a profile file (.pro) format file, and you can use Draw Profile to draw it.

Mass Haul Settings

The Haul Distance ranges are for reporting the cut to fill volume movements by the different haul distance ranges. For each range of stations with a net cut volume, the report has a row for the net fill station range the cut was moved to, the amount of cut/fill, the volumes per haul distance range, the average haul distance per range and the overall haul distance average. The In Station Volume is the amount of cut and fill that occurred at the same station and doesn't have to be hauled to another station. The purpose is to evaluate how far the cut has to be moved, and the haul distance ranges can be used to separate the distances for different types of equipment.



The image shows a 'Mass Haul Settings' dialog box with a blue title bar and a close button. It is divided into two main sections: 'Haul Distance Ranges' and 'External Hauls'.

Haul Distance Ranges

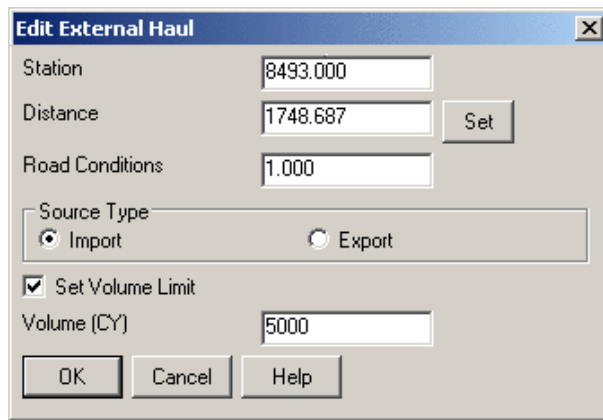
Haul Distance	Range
200.0	Up to 200.0
500.0	Up to 500.0
	Over 500.0

External Hauls

Type	Station
EXPORT	0+50.000

Buttons: Add, Edit, Remove, OK, Cancel

The External Hauls can be used to specify the stations along the road for borrow pits or dump piles. The program will use volume from these external hauls when the cut/fill of the road does not balance.



When dealing with multiple external hauls, the program will automatically minimize earth movement based on the haul distances and the available import/export (determined by "Set Volume Limit") of each haul. "Distance" sets how far a borrow pit or dump pile is from the entry station. "Road Conditions" is a multiplier against the specified distance. For example, if road conditions are twice as poor along an 2,700ft entry road as the main road, then a "Road Conditions" of 2.0 should be used. The program will then use a distance of 5,400ft (2,700x2) for that external haul when calculating optimized movement along the road.

Mass Haul Analysis

Existing Section> C:\Takeoff\Drawings\demo2-og.sct

Final Section> C:\Takeoff\Drawings\demo2-fn.sct

Volumes per Range Average Haul per Range

Net Cut Net Fill Total In Sta Haul 0 200 Over 0 200 Over

Station	Station	Cut(CY)	Import	Fill(CY)	Export	Volume	Volume	Volume	Volume	Volume	Volume	Volume	Volume	Overall Avg
0+00.000	1+00.000													
1+00.000	2+50.000	1251.729	0.000	1251.729	0.000	1251.729	1127.595	124.134	124.134	0.000	0.000	0.000	153.135	
0.000	0.000	153.135												
12+00.000	10+50.000													
13+80.000	12+00.000	887.367	0.000	887.367	0.000	887.367	239.938	647.429	542.651	104.777	0.000	0.000	164.552	
209.534	0.000	179.887												
13+70.000	15+05.340													
13+90.000	15+23.200	95.633	0.000	95.633	0.000	95.633	51.559	44.074	44.074	0.000	0.000	0.000	137.235	
137.235														
13+80.000	14+20.000													
14+20.000	14+50.000	216.434	0.000	216.434	0.000	216.434	96.872	119.563	119.563	0.000	0.000	0.000	38.560	
0.000	0.000	38.560												
14+60.000	14+50.000													
14+80.000	14+60.000	43.333	0.000	43.333	0.000	43.333	18.620	24.712	24.712	0.000	0.000	0.000	11.818	
11.818														
14+70.000	15+00.000													
15+00.000	15+14.270	82.194	0.000	82.194	0.000	82.194	29.738	52.456	52.456	0.000	0.000	0.000	24.395	
24.395														
Total:		9808.744	0.000	69788.70	0.000	74189.76	66786.40	7403.360	1288.814	930.060	783.423	127.043		

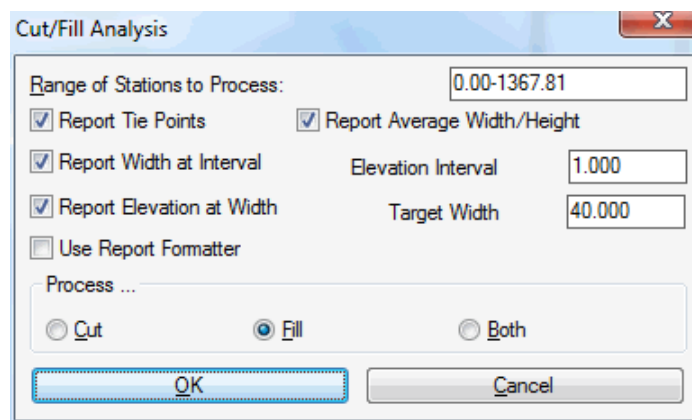
Pulldown Menu Location: Sections

Prerequisite: A Section Alignment File and Existing and Road Sections

Keyboard Command: masshaul

Cut/Fill Width Analysis

This command generates a report of the horizontal width of cut and fill areas between two cross sections. The report generated can take into account the cut, the fill or both. For example, the Process option for Cut only will report only the grinding areas as the cut end areas. The options for this command are set in the dialog shown. The **Report Width At Interval** option reports the width of each area is reported at different elevations set by the **Elevation Interval**. For the **Report Elevation At Width** option, the elevation is calculated for where the area has the **Target Width**. For the **Report Average Width/Height** option, the average width and height is calculated for each area. The **Report Tie Points** option reports the intersection points between the original ground and design sections. The **Use Report Formatter** option allows for customized reports and output to Excel and databases.



Prompts

Section File (Existing Ground) dialog choose existing .SCT file

Section File (Final Ground) dialog choose the other existing .SCT file

Cut/Fill Analysis dialog Make selections.

Cut/Fill Section Report is created.

Width Analysis Report:

```
Cut/Fill Section Report
Target Width: 20.00
Section 1: C:\data\simo2.sct
Section 2: C:\data\final.sct
Station: 0+10.000
Fill Area: 153.289
  Target Width at 108.07
  Area Above Target Width: 72.48
  Area Below Target Width: 80.81
  Average Width: 12.19
  Average Height: 2.95
  Elev: 110 Width: 25.97
  Elev: 105 Width: 10.65
  Elev: 100 Width: 2.86
  Elev: 95 Width: 0.00
```

Tie Points Report:

Section File 1: C:\sample\rehab1\exist-rd.sct

Section File 2: C:\sample\rehab1\grind.sct

Station Fill Area Tie Offset Width Tie Offset Left Tie Offset Right Tie Elev Left Tie Elev Right

732+58.510 12.941 24.000 -12.000 12.000 927.312 927.337

732+75.000 13.485 24.000 -12.000 12.000 927.381 927.288

733+00.000 13.554 24.000 -12.000 12.000 927.505 927.271

Pulldown Menu Location: Sections > Cut/Fill Analysis

Keyboard Command: cfwidth

Prerequisite: Two section .SCT files

Cut Sheet

This command compared a base grade section (.SCT) file with a final grade section (.SCT) file. It then reports the cut or fill values between these two sections files. The cut and fills are calculated at offsets in a specified range of stations. The offsets can be entered either as values (ie. 10) or by grade IDs (ie. "EP") when the section has descriptions on the section points. The report shows a column of cut and fill values for each offset as shown below. The two section .SCT files should have matching stations, because the program compares the offsets and elevations of a station in the first section file with the offsets and elevations of the same station in the second section file. The Report Formatter option allows for customized reports and output to different formats such as Excel. Otherwise the standard report shown here is generated.

Prompts

Base Grade Section File dialog *choose existing .SCT file*

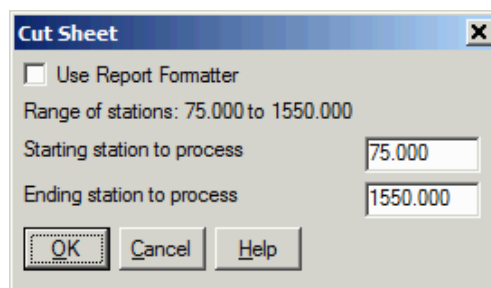
Final Grade Section File dialog *choose the other existing .SCT file*

Left offset: -10

Right offset: 10

Additional Offset (Enter to end): *press Enter*

Cut Sheet dialog



Cut Sheet Report

Station	-10.000 Offset	CL	+10.000 Offset
25.00	+4.55		+4.07
50.00	+21.10		+21.95
75.00	+24.26		+26.97
100.00	+30.63		+33.54
125.00	+41.35		+42.52

150.00	+49.61		+50.55
175.00	+49.02		+46.88
200.00	+43.65		+45.45
225.00	+24.22		+29.03
250.00	+7.44		+7.27
275.00	+41.58		+33.21

Pulldown Menu Location: Sections > Cut/Fill Analysis

Keyboard Command: cutsheet

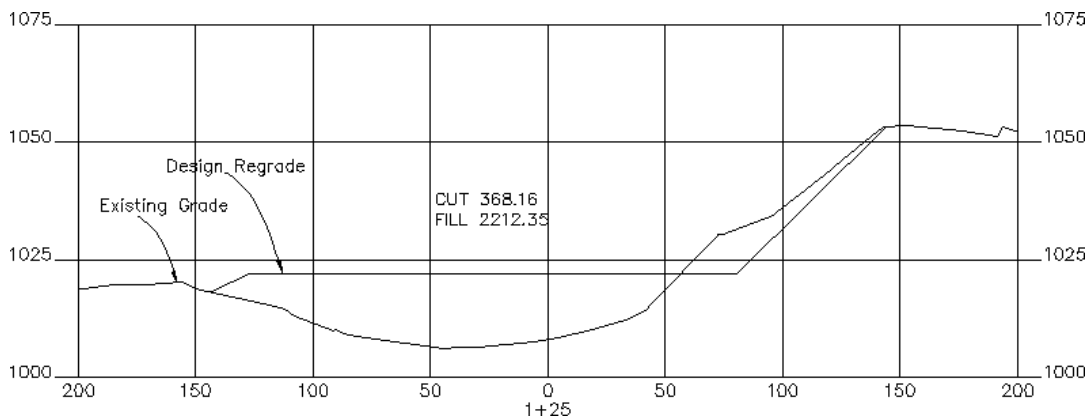
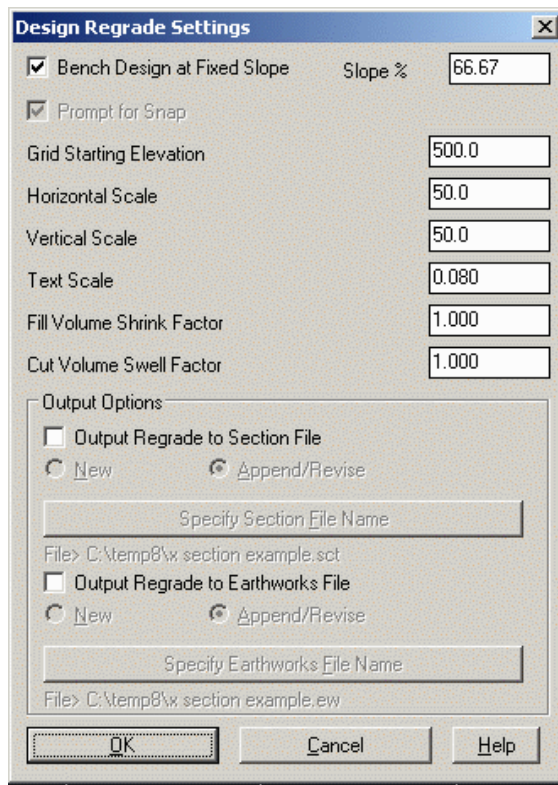
Prerequisite: Two section .SCT files

Design Regrade

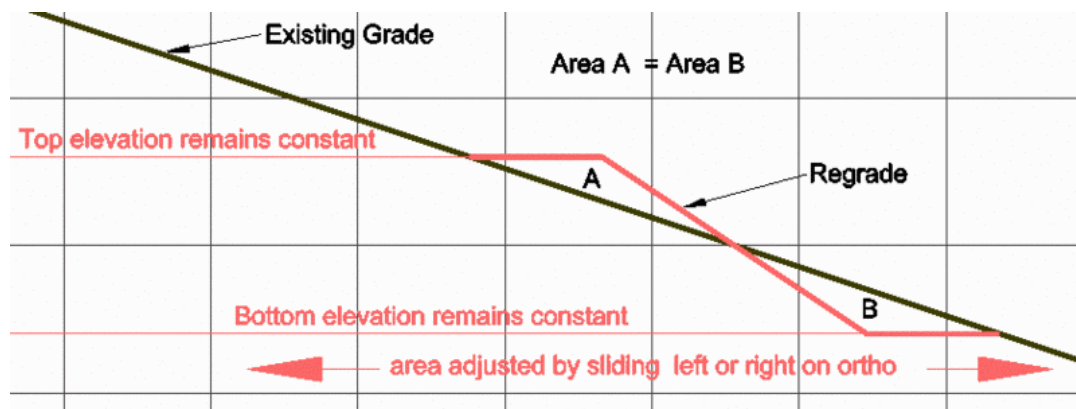
This command allows you to create final sections and have the offset, grades, and end area dynamically displayed and calculated while in the design process. This command works on a section grid that has an existing grade polyline which is used as the reference for the cut/fill calculations. The final section points can be entered as offset-elevation or picked on the screen. As you move the crosshairs on the graphics screen, the offset, elevation, slope percent, slope ratio, and end areas are displayed in a real-time pop-up window. When picking design points, you can use this real-time window to check the slope and other values and get these values close to the desired amounts. Then after the point is picked, a snap dialog appears (if the snap option is active) which allows you to set a slope or other value to a fixed amount (i.e. set the slope to 2.00% from 1.97%).

There are several options when specifying the final section points. The 'T' (Tie) option goes from the last final section point to intersect the existing grade at the specified slope. The 'F' (Force grade) option prompts for a slope percent or ratio that is used to set the slope between the last two regrade points. The program expects the final section points to be entered from left to right, but you can reverse the entry order by using the 'S' (Switch Direction) option. The 'P' (Pick existing) will use a point on the existing grade that is closest to the picked point. This is the same as the object snap *nearest*. The 'M' (Modify) option allows you to change an already placed regrade point. The program will prompt you to pick which regrade point to change. You can then pick a new position for it. This option allows the adjusting of already placed regrade points to help balance the cut/fill.

The command starts with a settings dialog. The Prompt for Snap toggle controls whether the command will present the snap dialog for each section point that is picked on the screen. The Grid Starting Elevation edit box allows you to input the beginning elevation of the local grid that you are designing in. Use the Horizontal and Vertical Scale edit boxes to set the proper horizontal and vertical scales for your design environment. The Text Scale value is multiplied by the horizontal scale to set the text size of the cut/fill end area labels. The regrade can be saved in a .sct file with the Output Regrade to Section File option. The Output Regrade to Earthworks File option stores the cut and fill values of the regrade to an earthwork (.EW) file that can be used in the *Print Earthwork File Report* command.



The **Bench Design at Fixed Slope** creates a regrade between top and bottom bench elevations. Besides the bench elevations, this method also takes input for the slope between the benches. Then the program calculates the position for the regrade by side shifting the regrade surface until the cut and fill areas balance.



Prompts

Design Regrade Settings dialog Make choices and click OK.

Pick Center Point of Grid [int on]: *pick a point* Pick the point where the local grid lines intersect at the 0 offset and datum elevation.

Select the existing grade polyline: *select polyline*

First offset or pick a point (P,Help): *pick a point* If you have the snap prompting on then each time you pick a point from the screen the dialog below will appear.

Second offset or pick a point (U,S,T,P,M,Help): *pick a point* If you make a mistake on one of the points you select than use the U option to undo or delete your last entry.

Next offset or pick a point (U,F,S,T,P,M,Help): *F* If you want to type in a specific grade use the Force grade option.

Ratio/<Percent grade between 0.0 and 30.2>: *-2*

Next offset or pick a point (U,F,S,T,P,M,Help): *pick a point*

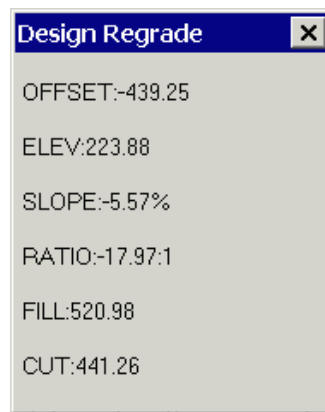
Next offset or pick a point (U,F,S,T,P,M,Help): *75*

Percent grade/Ratio of slope/<Elevation>: *445*

Next offset or pick a point (U,F,S,T,P,M,Help): *press Enter* Press Enter to end the command.

Pick point for label (Enter for none): *pick a point to label the cut/fill end areas*

Enter station of regrade: *100*



Pulldown Menu Location: Sections

Keyboard Command: regrade

Prerequisite: Must plot the polyline that represents the existing grade

Calculate Haul Factors

After creating a regrade polyline with *Design Regrade*, this command will calculate how far and how much earth must be hauled in order to obtain the regrade from the existing grade. Areas from cut will be moved into areas of fill. In order to be optimal, this command expects even amounts of cut and fill. Haul factors measure the amount of earth moved times the distance. There is also an option to label the cut and fill areas.

Prompts

Horizontal Scale <50.0>: *press Enter*

Vertical Scale <50.0>: *press Enter*

Select the existing grade polyline: *pick the polyline*

Select the regrade polyline: *pick the polyline*

Label areas (<Yes>/<No>)? *press Enter*

Text size <4.00>: *press Enter* This defaults to the horizontal scale times the text scaler.

Enter the report title <Haul report>: *press Enter*

Write report to file (Yes/<No>)? *press Enter*

Write report to printer (Yes/<No>)? *press Enter*

Haul factors for a regrade:

Haul report

Haul factor 344.43, Amt 27.32, Dist 12.61 from cut 1 to fill 1

Haul factor 15.99, Amt 1.45, Dist 11.04 from cut 1 to fill 2

Haul factor 74.78, Amt 4.51, Dist 16.60 from cut 2 to fill 2

Haul factor 2339.17, Amt 58.07, Dist 40.28 from cut 2 to fill 3

Haul factor 2945.88, Amt 62.10, Dist 47.44 from cut 3 to fill 3

Haul factor 1038.63, Amt 11.20, Dist 92.72 from cut 4 to fill 3

Haul factor 116.30, Amt 6.09, Dist 19.09 from cut 4 to fill 4

Haul factor 50.81, Amt 2.99, Dist 16.98 from cut 4 to fill 5

Total cut: 173.73, Total fill: 174.93

Left over cut: 0.00, Left over fill: 1.20

Haul factor (dist*amt): 6925.98

Haul ratio (haul factor/total amt): 39.87

Pulldown Menu Location: Sections

Keyboard Command: haulfact

Prerequisite: Existing and final grade polylines

Sections to 3D Polylines

This command creates 3D polylines from a section (.SCT) file. Besides the section file, a centerline polyline, centerline file or section alignment (.MXS) file must be specified to define the plan view location of the 3D polylines. The elevations for the 3D polylines come from the section file. These 3D polylines can then be used by other Carlson routines to create surface models.

In the options, dialog set the range of stations to draw and the Layer Name for the 3D polylines.

The **Draw Method** has four options:

Cross Section: The 3D polylines are drawn as cross-sections perpendicular to the centerline at each station.

Offsets By Description: The 3D polylines are drawn parallel to the centerline by connecting section points when the same description. To use this method, the SCT file must have descriptions on the section points. For example, if the section file has descriptions for each section point then you can draw 3D polylines for EP, SHD, TIE, etc.

Offsets By Value: The 3D polylines are drawn parallel to the centerline at a specified offset value.

Offsets By Sequence: The 3D polylines are drawn parallel to the centerline by connecting section points by their sequence in the section file. For example, a 3D polyline will connect all the 1st section points, another 3D polyline will connect all the 2nd section points, etc. This method requires the sections to have the same number of data points for each station.

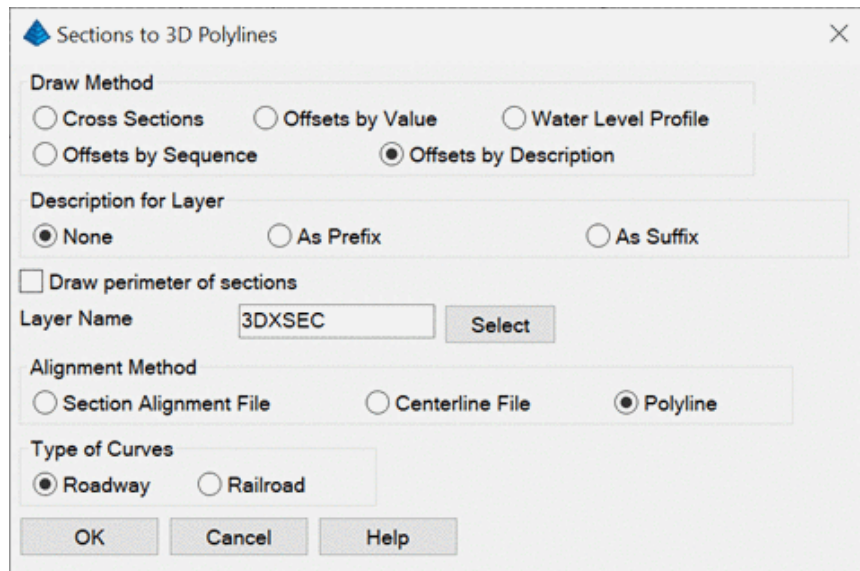
Water Level Profile: This method draws two 3D polylines for the left and right banks of a waterway. The section file represents the cross sections of the waterway. The program also prompts for a profile of the water level. At each section, the program gets the water elevation at that station from the water profile and finds the offset on the section for the left and right banks.

Description for Layer: This option uses the section point descriptions in the layer names for the 3D polylines when using the Offsets By Description method.

Draw perimeter of sections: This option will connect all the left most offsets and right most offsets together with a 3D polyline.

Alignment Method: Chooses how to define the horizontal alignment.

Type of Curves: This option chooses between roadway and railroad methods for stationing along curves.



Prompts

Choose Section File to Process Select the .sct file

Sections to 3D Polylines dialog Choose your settings

Select centerline polyline: *pick the polyline*

Enter the centerline starting station <0.0>: *press Enter*

Use reference profile to interpolate between sections [<Yes>/No]? *N for no.* This option will prompt for a profile to use for interpolating elevations along the 3D polylines between the section stations. This improves the accuracy when the profile goes through vertical curves. Without the profile, the 3D polyline elevations will be straightline interpolated between the sections.

Draw all template ids or specific ids and offsets [All/<Specific>]? *press Enter for Specific*

Enter Offset or Description to draw: *EP*

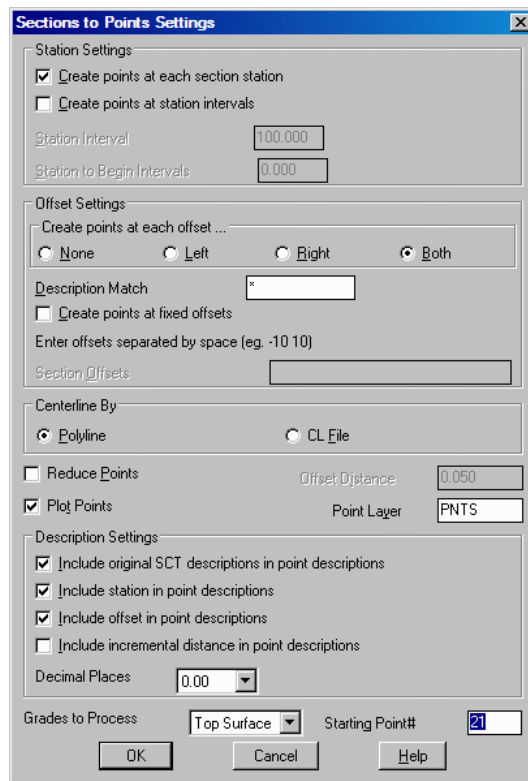
Pulldown Menu Location: Sections > Section Utilities

Keyboard Command: scto3dp

Prerequisite: A section (.SCT) file

Sections to Points

This command creates Carlson points using a section (.SCT) file to define the point elevations. The x,y position of the points are calculated based on the station and offset along a centerline polyline. These points are stored in the current coordinate (.CRD) file and can also be plotted in the drawing. Points can be created at each station in the section file or at a set station interval. The range of stations to process can also be set. The Description Match field can be used to filter the offsets and only create points with matching descriptions (e.g. only "EOP" offsets). The Create points at fixed offsets option can be used to make points at user-specified offset distances. The program will interpolate the elevations for these points by interpolating from the neighboring offsets. This is both a Centerline by Polyline or by CL File option. The CL File option will prompt for an existing centerline (.CL) file. The Reduce Points option will skip creating points for the same offset between stations if the x,y position and elevation change is less than the offset tolerance. Essentially, when a series of offsets are on a straight line (no vertical and no horizontal curve) then only the starting and ending points are needed and all the intermediate points can be skipped. For example, the Reduce Points routine will look at the left side EOP offset points at stations 1+00, 1+05 and 1+10 and if these three points make a straight line then the point for station 1+05 can be reduced. The Offset Distance is the tolerance that Reduce Points uses for testing whether the middle point (offset point at station 1+05) can be reduced. The distance for the middle point is calculated as the perpendicular distance from the middle point to the line between the two end points. Both the horizontal and vertical distances are checked.



Prompts

Sections to Points Settings dialog

Coordinate File to Process Choose a .CRD or other coordinate file to add the points to. This prompt only occurs if no coordinate file is current.

Choose SCT file to read *pick the cross section file*

Range of stations: 3.34 to 750.00

Enter the starting station to process <3.34>: *press Enter*

Enter the ending station to process <750.00>: *press Enter*

Select centerline polyline: *pick the polyline that defines the stations*

Type of centerline [<ROadway>/RAilroad]? *RO*

Enter the centerline starting station <0.0>: *press Enter*

Created 65 points.

Keyboard Command: sctopt

Prerequisite: A .set file and polyline centerline

Design Section Staging

This command takes a design cross section and splits it into two stages for cases when the design surface will be built in stages. There are two staging methods.

The Offset method splits the design section at a specified offset with the left side as one stage and the right side as the other stage. This method applies to the situation of designing a partly completed road or regrade. For example, if a four lane road will be built two lanes at a time, then the offset method can be used to split the design section with two lanes on the left side of the offset and the other two lanes on the right side. Using an existing and a final grade section file, the program will create four new sections files for the finished existing sections, finished final sections, remaining existing sections, and remaining final sections. The source existing and final section files should have matching stations. There is an option to process a range of the possible stations from the section files. The complete part of the road can be either on the left or right side. The pivot point is a cross section offset where the

completed part ends. From this point, the final grade will connect to the existing grade by a line at the specified slope.

The Description method uses a specified description from the existing ground section file plus an offset from this description. Then the existing section is overlaid onto the design section for the offset zone around this description. This method applies when a portion of the existing ground stays intact when the first stage of design is built and then this remaining portion of the design is done as the second stage. For example, this applies to improving railroads where the existing track is left undisturbed while the work for the new bed is prepared. In this case, the existing section file should have a description for the offset position of the existing track centerline. Then you specify the buffer offset around this centerline. From the resulting left and right offsets, the program ties the existing section into the design at a specified slope.

Prompts

For Offset Method:

Select Existing Sections File Choose the cross sections file.

Select Final Sections File Choose the cross sections file.

Enter slope as percent grade or slope ratio [Percent/<Ratio>]? *press Enter*

Enter the fill slope ratio <2.0>: *press Enter*

Enter the cut slope ratio <2.0>: *press Enter*

Stage by side from offset or overlay existing at description [<Offset>/Desc**]?** *press Enter*

Place road on left or right [<Left>/Right**]?** *press Enter*

Range of stations: 50.0 to 100.0

Enter the starting station to process <50.0>: *press Enter*

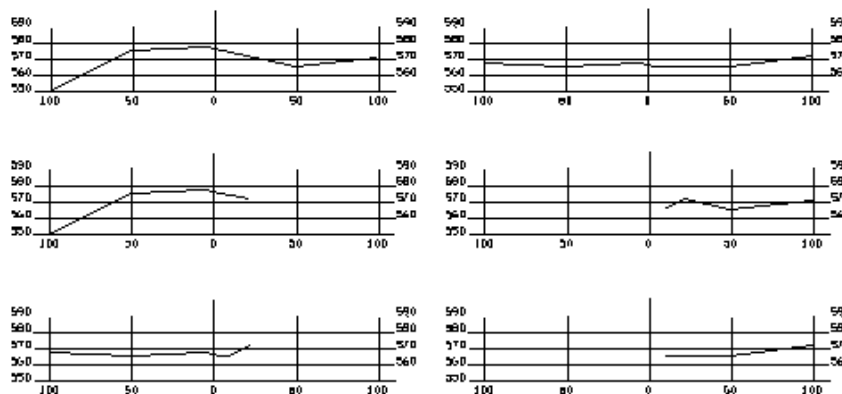
Enter the ending station to process <100.0>: *press Enter*

Apply same pivot offset to all stations [Yes/<No>]? *Y*

Enter the pivot offset (enter left offsets as negative) <0.0>: *5.0*

SCT File dialogs Enter new .SCT file names for 1) existing road .SCT file, 2) final road .SCT file, 3) remaining existing .SCT file and 4) remaining final .SCT file.

Here is an example of the Offset method showing the original existing and design sections and then the four new sections files for the finished existing sections, finished final sections, remaining existing sections, and remaining final sections that the routine creates.



For Description Method:

Enter slopes as percent grade or slope ratio [Percent/<Ratio>]? *press Enter*

Enter the fill slope ratio <2.0>: *press Enter*

Enter the cut slope ratio <2.0>: *press Enter*

Stage by side from offset or overlay existing at description [<Offset>/Desc**]?** *D for description*

Existing section target description: *CL*

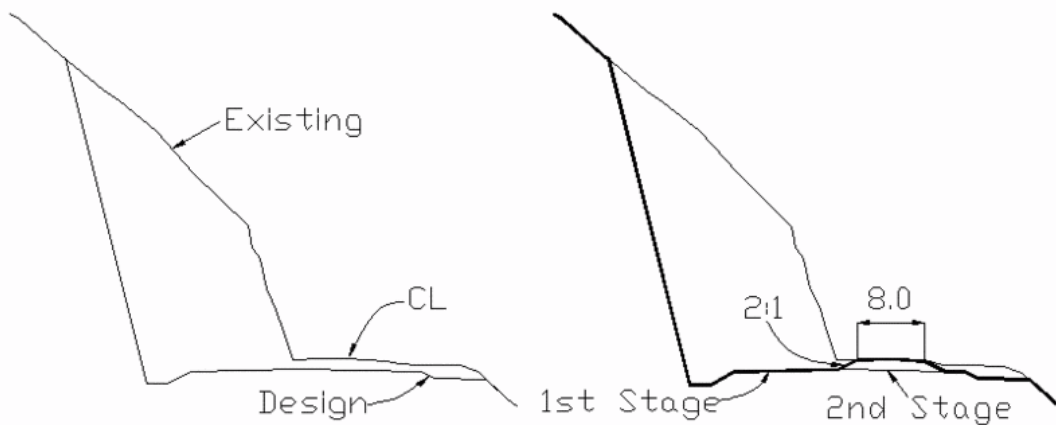
Range of stations: 100.00 to 100.00

Enter the starting station to process <100.00>: *press Enter*

Enter the ending station to process <100.00>: *press Enter*

Enter the buffer offset <0.0>: 4

Here is an example of the before and after for the Description method.



Pulldown Menu Location: Sections

Keyboard Command: sctstage

Prerequisite: Existing and final grade section files (.SCT)

Draw Pipe 3D Polyline

This command creates a 3D polyline that represents a pipe. The points can be either picked on screen or specified by point number in the current coordinate file. This command is a convenient way to make 3D polylines that can become "pipe polylines" used for capturing their profile positions, leading to circular or elliptical or even square plots of the pipes or culverts within Draw Profile. However, this command is not required nor sufficient to make a pipe polyline useful in the *Draw Profile* command. Pipe polylines are made only by converting 3D polylines into pipe polylines using the adjacent command, *Assign Pipe Width to Pline*.

Prompts

Layer Name for 3DPoly <PIPE>: *press Enter*

Prompt for elevations (.XY filter) (Yes/<No>)? *Y for yes*

Undo/<Pick point or point numbers>: *pick a point*

Elevation <0.0>: 554.12

Undo/<Pick point or point numbers>: *pick a point*

Percent slope/Ratio slope/Elevation <0.0>: 553.72

Undo/Close/<Pick point or point numbers>: *press Enter*

Draw another 3D polyline (Yes/<No>)? *press Enter*

Pulldown Menu Location: Sections

Keyboard Command: drwpipe

Prerequisite: None

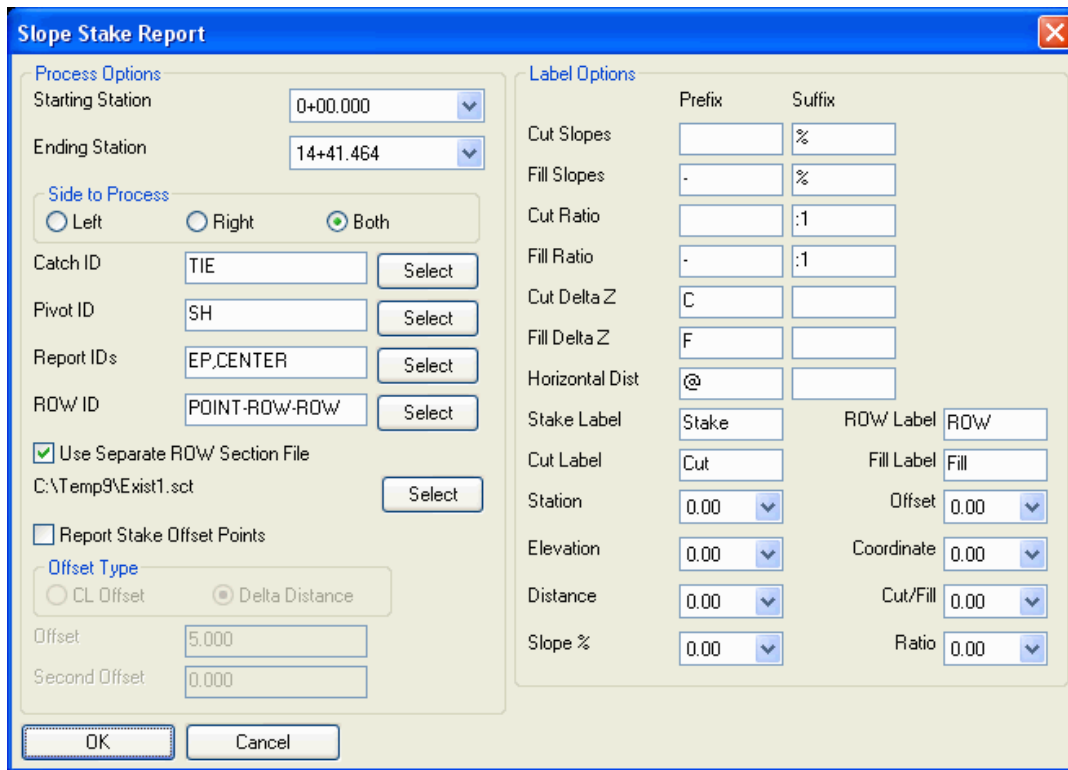
Assign Pipe Width to Polyline

This command is described in the Profiles Menu section of Help, under Profile Utilities.

Slope Stake Report

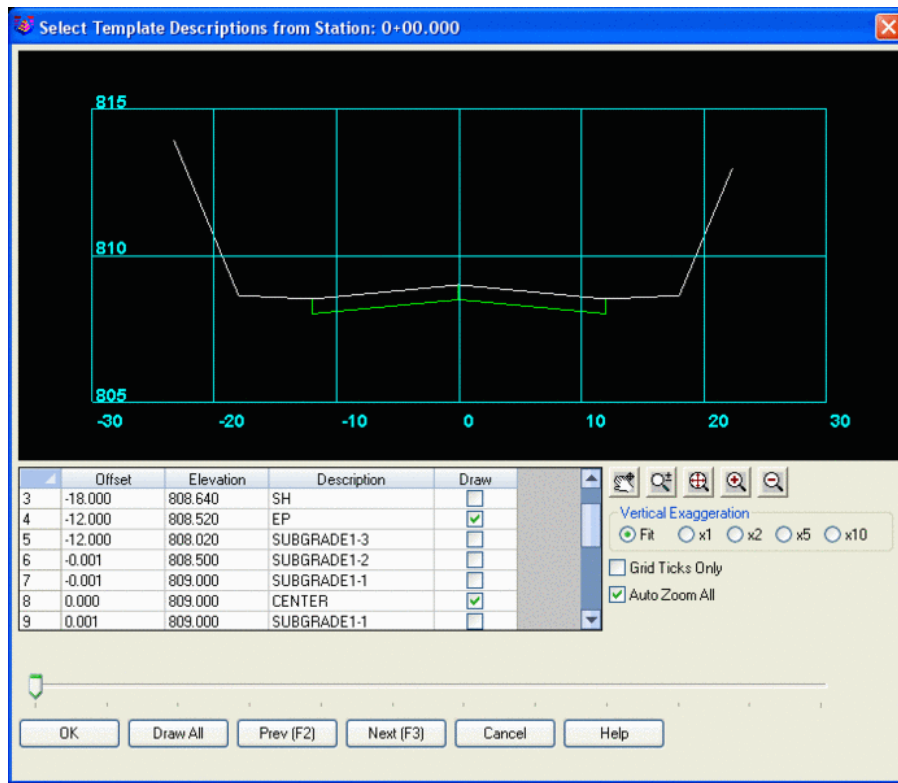
This command creates a slope stake report using cross section data with stations, offsets, elevations and descriptions. The program uses the data point descriptions to identify and catch and pivot points and other data points to report.

Besides processing a section file (.sct), the program also uses a centerline file (.cl) for reporting coordinates for the data points.



After selecting these files, there is a dialog for the report options. The Starting and Ending Stations control the range of stations to report. The Side To Process option selects whether to report the slope stake for the left or right of the centerline or both sides. The Catch ID is the section data point description for where the section ties into existing ground. The Pivot ID is the description for the hinge point at the beginning of the cut/fill slope. The Catch and Pivot ID's are required for the report. The Report ID's are optional additional section break points to include in the report. They should be entered in outside template to inside order. The Select button shows a graphic of the section data with toggles to select which section points to report. You can pick any combination of surface or subgrade points. Don't include the Catch ID or Pivot ID in the Report IDs because those IDs are already reported. The ROW ID is optional for including the ROW in the report. Typically, the ROW data is in the existing surface sections instead of the final sections. In this case, the Use Separate ROW Section File option can be used to specify the section file with the ROW data. The Report Stake Offset Points option adds an offset point that can be used for stakeout. The offset amount can be relative to the catch point or to the centerline. A second offset can be used for orientation with the first offset. The program reports the distance from the offset point to the catch point.

The Label Options section has prefix and suffix settings to add to the report values. There are also settings for the labels to use for the Stake, ROW and Cut/Fill names. There are also controls for the decimal places for the report values.



After specifying the options, the program uses the Report Formatter to generate the report. The available fields are point name, station, offset, northing, easting, elevation, cut/fill label, horizontal distance, vertical distance, slope percent and slope ratio. You can select which fields to report and their order. The field labels and decimal precision is controlled by the Report Formatter. The report can be output to the Report Viewer or Excel.

Slope Stake Report

Sections: C:\sample\road.sct
Centerline: C:\sample\demo3.cl

Station: 0+00.000

From Name	To Name	Horizontal	Vertical	Cut/Fill	Slope%	Ratio
Stake Offset	TIE	5.000	0.000	Flat	0.00	999.900
TIE	SH	5.255	5.255	Cut	100.00	1.000
SH	EP	6.000	0.120	Cut	2.00	50.000
EP	CENTER	12.000	0.480	Fill	4.00	25.000

You can also use the Mirror the columns option in the Report Formatter to layout the report like this:

Sections: C:\sample\final.sct
Centerline: C:\sample\demo.cl

Station: 0+00.00

CP	SH	EP	CENTER	EP	SH	CP
-19.53	-18.00	-12.00	0.00	12.00	18.00	19.53
1038.19	1038.57	1038.45	1038.69	1038.45	1038.57	1038.19
-25.00%	2.00%	-2.00%		-2.00%	2.00%	-25.00%
F 0.38	C 0.12	F 0.24		F 0.24	C 0.12	F 0.38
@ 1.53	@ 6.00	@ 12.00		@ 12.00	@ 6.00	@ 1.53

Prompts

Section File To Process Select a .sct file.

Centerline File To Process Select a .cl file.

Slope Stake Report dialog Choose report options.

Report Formatter dialog Configure and display report.

Pull-down Menu Location: Sections > Section Utilities

Keyboard Command: ssreport

Prerequisite: A section and a centerline file

Update Sections from Slope Stake

This command updates the offset and elevation of tie points in a road design section file using the slope stake report from SurvCE for the measured tie point locations.

Prompts

Section File to Read *select road design section file*

Slope Stake File to Read *select SurvCE slope stake file*

Section File Tie Point Description <TIE>: *enter description of tie points in the road design section file*

Pull-down Menu Location: Sections > Section Utilities

Keyboard Command: stakesct

Prerequisite: Drawn Section File and SurvCE Slope Stake File

Extend Sections to Offset Limits

This command extends the offsets to the left and right limits for each station in a section file. The left and right offset limits are defined in the section alignment (.MXS) file. The elevations for the extended offsets can be extrapolated from the last slope from the existing offsets or the elevations can be carried flat from the last offset elevation. For example, consider section station with a left most offset of -192.5 and a right most offset of 197.3. If the MXS file had offset limits of 200 for left and right, then this routine would assign offsets with elevations at offsets -200 and 200. The resulting section file can be saved to a separate section (.SCT) file or overwrite the original SCT file.

Prompts

Select Section Alignment File *select .MXS file*

Section File to Read *select .SCT file*

Extend last slope or use last elevation [<Slope>/Flat]? *press Enter*

Choose SCT file to Write *specify new .SCT file name*

Pull-down Menu Location: Sections > Section Utilities

Keyboard Command: extendsct

Prerequisite: .SCT file and .MXS file

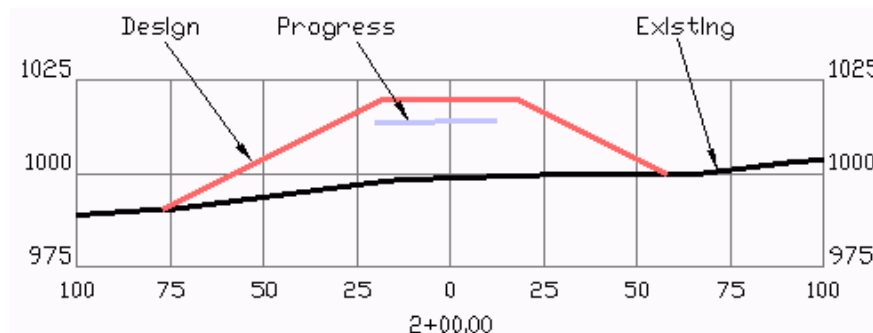
Extend Progress Sections

This command extends cross sections that partially cover the design sections to make the sections reach the full extents of the design. This routine applies to a design that is being built in stages. The purpose is to build sections for the surface at a progress stage during construction. The input progress sections for this command are updated and saved back to the progress section file. The progress sections can then be used in Calculate Sections Volumes to calculate progress volumes.

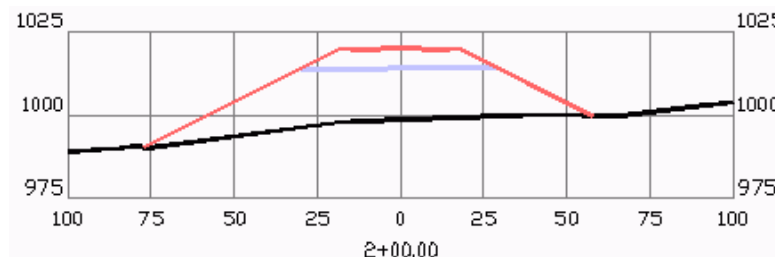
The progress sections will not go beyond the design so that extra volumes are not counted as part of the progress. For example, if a design is in fill and the input progress sections go above the design, then the output progress sections will be brought down to match the top of the design.

Besides prompting for the design sections and input progress sections, the program also prompts for the existing ground sections which are used to limit the progress sections to avoid any extra volumes. For example, if a design is in fill and the input progress sections go below the existing ground, then the output progress sections will be brought up to match the existing ground.

An example application is for road design where the progress is surveyed during construction using GPS to gather points along the road. These points are then used to create a progress section which cover a portion of the road surface but don't tie into the final design. Then this Extend Progress Sections command takes the progress sections from the points and extends the sections to tie into the design.



Input Sections



Resulting update of progress section

Prompts

Original Surface Section File *select .SCT file*

Design Surface Section File *select .SCT file*

Progress Survey Section File *select .SCT file*

Pulldown Menu Location: Sections > Section Utilities

Keyboard Command: sctprogress

Prerequisite: Existing, Design and Progress .SCT files

Section Area Report

This command reports the surface area from a cross section surface. Either the horizontal or slope area is reported. There are also prompts to limit the area report to a station range or offset range. The program prompts for the section file to process. There is an option to use a second section file for a cut/fill reference. When this reference section file is used, the program reports the area broken out by cut and fill areas.

Section Area

Section File: final.sct
Starting Station: 0+75.000
Ending Station: 15+50.000

Horizontal Area(SF)
127322.1

Prompts

Section File to Process *select .SCT file*

Use Another Section For Cut/Fill Reference dialog

Report slope or horizontal area [<Horizontal>/Slope]? *press Enter*

Starting station to process <75.000>: *press Enter*

Ending station to process <1550.000>: *press Enter*

Limit offsets to process [Yes/<No>]? *press Enter*

Pulldown Menu Location: Sections > Section Utilities

Keyboard Command: sctarea

Prerequisite: .SCT file

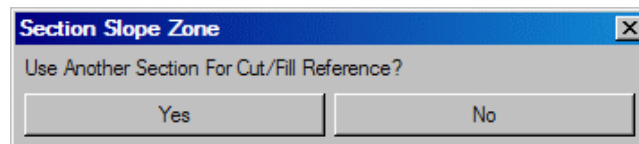
Slope Zone Section Analysis

This command reports the areas of the surface within different slope ranges. The surface is defined by a cross section file (SCT). The option to **Use Another Section For Cut/Fill Reference** reports the cut/fill areas and volumes within given ranges of slopes.

Prompts

Select Section to Process *Select .SCT file*

Select Slope Zone dialog



Report slope or horizontal area [<Horizontal>/Slope]? *S*

Slope format [<Percent>/Ratio]? *press Enter*

Greatest slope % of zone 1: *10*

Greatest slope % of zone 2: *press Enter*

Starting station to process <0.000>: *press Enter*

Ending station to process <1514.567>: *press Enter*

The Standard Report Viewer creates a report called Section Slope Zone Analysis Report.

Section Slope Zone Analysis

Section File: C:\src\work\output.xsct

Starting Station: 0+00.000

Ending Station: 15+14.567

Slope % Area(SF) % of Total

0.0 to 10.0 78757.5 61.3

> 10.0 49617.0 38.7

Total: 128374.5

Pulldown Menu Location: Sections

Keyboard Command: sctzone

Prerequisite: .SCT file

Color 3D Lines By Slope

This command sets the color of 3D lines by slope range. The slope ranges are defined in percent slope for up to ten ranges. There is an option to draw a legend of the slope colors.

Slope %	Range	Color	
5.000	<= 5.000	170	Set
10.000	5.000 to 10.000	150	Set
20.000	10.000 to 20.000	130	Set
30.000	20.000 to 30.000	110	Set
40.000	30.000 to 40.000	90	Set
50.000	40.000 to 50.000	70	Set
75.000	50.000 to 75.000	50	Set
100.000	75.000 to 100.000	30	Set
	> 100.000	10	Set

Prompts

Define Slope Ranges Dialog

Select 3D lines to color.

Select objects: *pick the entities*

Draw Legend [**<Yes>/No**]? Y

Legend size <5.000>: *press Enter*

Pick point for legend: *pick a point*

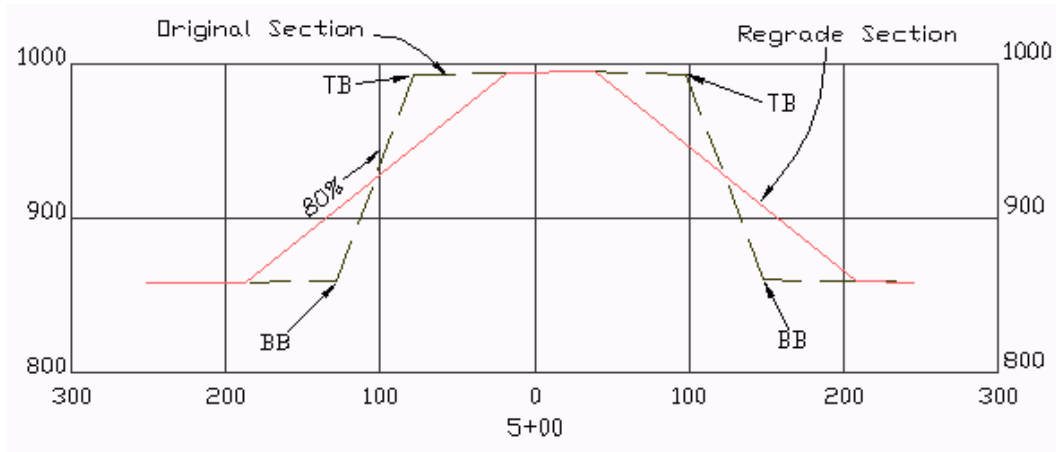
Pull-Down Menu Location: Surface > Slope Analysis

Keyboard Command: color3dline

Prerequisite: 3D Lines

Regrade Fill Slope

This command is used on stockpile projects to regrade the fill slopes of the cross sections with less steep slopes while maintaining the same end areas. The slope portion of the cross section to regrade is identified by entering the section descriptions for the top and bottom points.



Prompts

Select existing section file Pick a section file to read

Specify new section file to Output Enter a section file to create

Target Slope Percent: 80

Top of Slope Offset Description: TB

Bottom of Slope Offset Description: BB

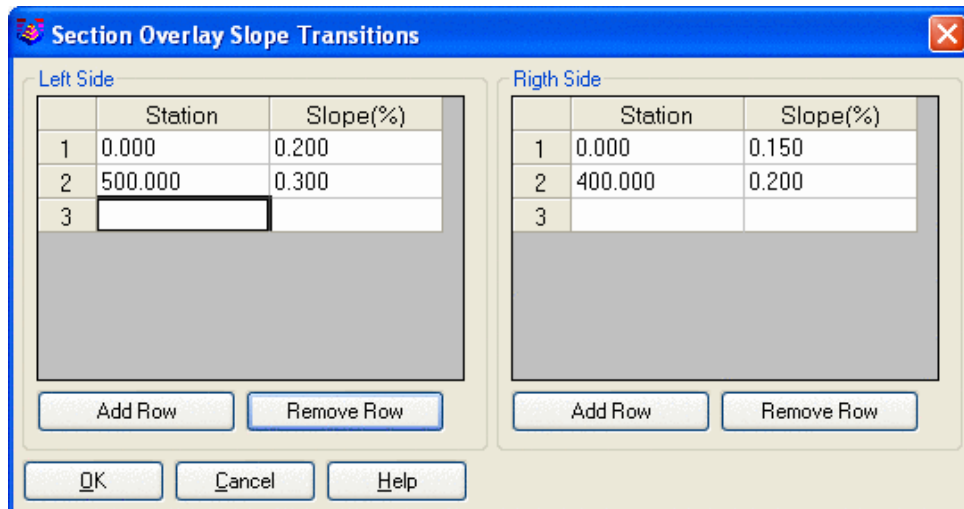
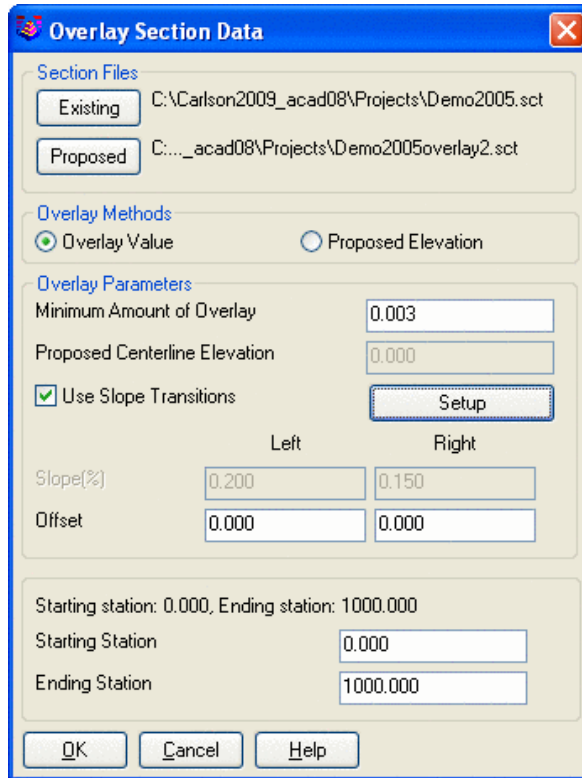
Pulldown Menu Location: Sections > Section Utilities

Keyboard Command: regrade_slope

Prerequisite: A section (.SCT) file

Overlay Section File

This command will create a section file at given cross slopes and minimum overlay from a reference section file. An existing and a proposed .SCT file must be selected, along with additional section information. A choice between Overlay Value or Proposed Elevation must be made. When the Overlay Value option is chosen, a Minimum Amount of Overlay must be entered, and the Slope values must be defined. You are allowed to set up slope transition values for certain station ranges. When the Proposed Elevation option is chosen, a Proposed Centerline Elevation value must be entered.



Prompts

Overlay Section Data dialog *Select file names and options, click OK*
 Report viewer creates a proposed section file report.

Pulldown Menu Location: Sections > Section Utilities

Keyboard Command: sct_overlay

Prerequisite: A .SCT file

Average Section Files

This command will average a section file for a given station range. A source file to process must be selected to get things started. A starting station to average and a last station to average must then be entered. A new .SCT file will

be created as a result.

Prompts

Select Source Section File to Process Select a SCT file.

Starting station to average <0.000>: *press Enter*

Last station to average <1614.160>: *press Enter*

Section File to Write Select a SCT file name and folder.

Pulldown Menu Location: Sections, Section Utilities >

Keyboard Command: avgscct

Prerequisite: A .SCT file

Merge Sections

This command combines a range of stations of one section and a range of stations of a second section. The stations, offsets and elevations in these two ranges can be stored in a new file or they can overwrite an existing profile. Two .SCT files are required.

Prompts

First Section File to Merge *select an existing .SCT section file*

Starting station to merge <0.000>: *press Enter*

Last station to merge <1614.160>: *press Enter*

Second Section File *select another existing .SCT file*

Starting station to merge <0.000>: *press Enter*

Last station to merge <1310.050>: *press Enter*

Section File to Write *Enter a new .SCT file name and choose folder*

Pulldown Menu Location: Sections > Section Utilities

Keyboard Command: mergesct

Prerequisite: Two section files

Compare Section Files

This command will report the differences between two section files. The report includes differences in offsets or elevations for the section data points. Also, the report includes stations that exist in one file and not the other. Here is a sample report.

Compare Section Files

1st Section File: C:\sample\final1.sct

2nd Section File: C:\sample\final2.sct

	1st Section File			2nd Section File		
Station	Offset	Elevation	Description	Offset	Elevation	Description
6+00.000	10.000	1007.873	EP	10.000	1008.003	EP
6+00.000	22.799	995.074	TIE	22.914	995.089	TIE
6+50.000	10.000	1008.906	EP	10.000	1009.208	EP
6+50.000	21.735	997.170	TIE	22.062	997.145	TIE

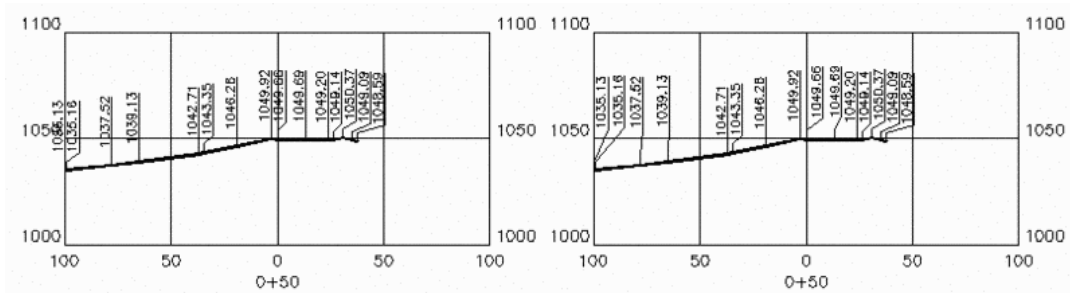
Pulldown Menu Location: Sections, Section Utilities >

Keyboard Command: comparesct

Prerequisite: A .SCT file

Move Section Leader Labels

This command custom positions section labels together with leaders. The section labels must be created with the Draw Break Point Leader option with the Draw Section File command. When the sections are redrawn, any custom positions done by this command are retained.



Before and after of Move Section Leader Labels to clean up label overlaps with grid lines

Prompts

Select section label to move: *pick a text label*

Pick label position: *pick a new position for the label*

Select section label to move (Enter to end): *press Enter*

Pulldown Menu Location: Sections > Section Utilities

Keyboard Command: sctmove

Prerequisite: section labels with leaders

Update Sections from Polylines

This command is used to update section (.SCT) files from manual changes made to polylines originally created by Draw Section.

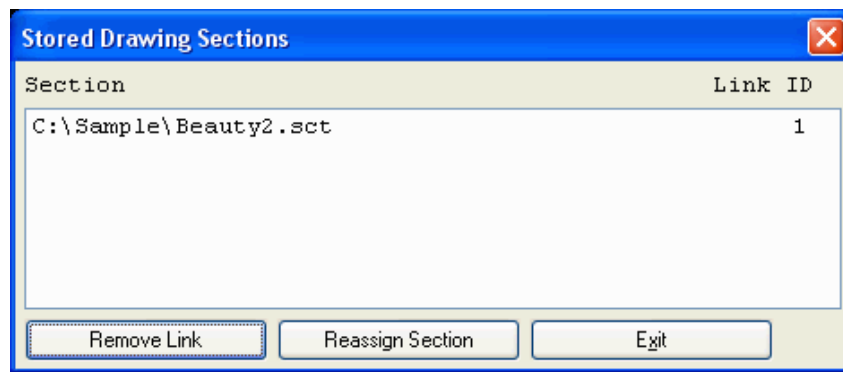
Pulldown Menu Location: Sections > Section Utilities

Keyboard Command: update_sct_from_dwg

Prerequisite: Drawn Sections

Review Section Links

This command shows a list of all the section links that the program knows about in the current drawing. These links are between the section files and the drawn sections in the drawing. These links are created by the Draw Sections command. You can use the Remove button to remove links for any obsolete sections or if you don't want to link a certain section. You can also reassign a link in case the location of the section file has changed.



Pulldown Menu Location: Sections->Section Utilities

Keyboard Command: sctdict

Prerequisite: none

Section ID

This command is used to pick a section drawing entity and then report the source section file name.

Prompts

Select section entity to identify: *select entity*

Pulldown Menu Location: Sections > Section Utilities

Keyboard Command: sctid

Prerequisite: A .SCT file

Set Descriptions By Offset

This command assigns descriptions to existing section points in a section file at specified offsets. Besides the specified offsets, the description can be set for the section point with the farthest left or farthest right offset.

Prompts

Set description for left most offset [Yes/<No>]? *Y*

Description: *TIE*

Set description for right most offset [Yes/<No>]? *Y*

Description: *TIE*

Offset to set description (Enter to end): *12*

Description: *EP*

Offset to set description (Enter to end): *press Enter*

Pulldown Menu Location: Sections > Section Utilities

Keyboard Command: sct_desc_offset

Prerequisite: A section file

Set Descriptions By Polyline

This command assigns descriptions to existing section points in an SCT file using plan view polylines. For example, you can enter a description of "EP" and then selection the edge of pavement polyline. For each station, the program calculates the offset of the selected polyline and then sets the description at that offset in the SCT file if there is a section point with the matching offset.

Prompts

Choose Section File to Process Select the .sct file

Select centerline polyline: *pick the polyline for the zero offset position of the SCT file*

Enter the centerline starting station <0.0>: *press Enter*

Description for section: *EP*

Select EP polylines.

Select objects: *pick the polylines for EP*

Description for another section (Enter to end): *press Enter*

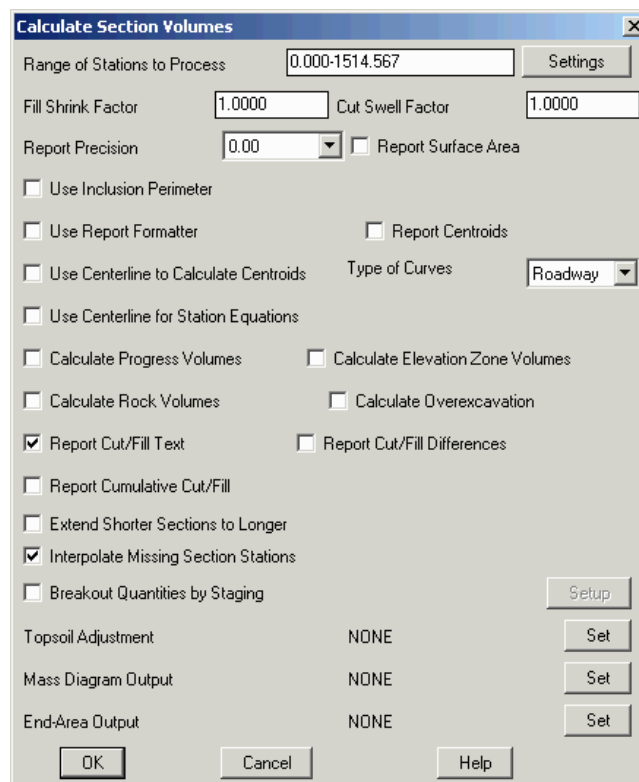
Pulldown Menu Location: Sections > Section Utilities

Keyboard Command: sct_desc_pl

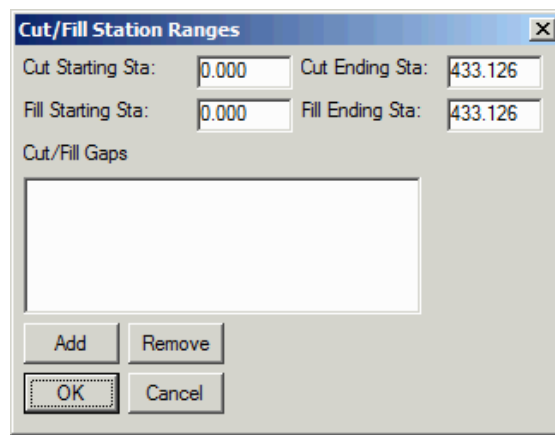
Prerequisite: A section (.SCT) file and polylines

Calculate Section Volumes

This command will read two section files and compute the cut and fill end areas and volumes. It computes the sections volume in the order they appear in the file. If you need to sort the stations in sequential order use the *Section File Editor* command. Begin by selecting the base section file then the final section file. After specifying the input files the Calculate Section Volumes dialog appears. The settings can then be chosen and customized to match your reporting needs. There is an option to apply topsoil removal/replacement adjustments, as well as support for processing sections with subgrades.



Range of Stations to Process: Specify the range of stations to process. Separate stations with a hyphen as shown. The Settings button brings up another dialog with more station options:



Cut/Fill Starting/Ending Sta.: Volumes are calculated using end areas between the range of stations. Instead of cutting off the volumes exactly at this range, the Ending and Starting Stations for Cut and Fill can be used to have the volume taper from zero at the specified Starting Station to the volume at the first station in the range. Likewise, the Ending Stations can be used to taper the volume from the last station in the range to zero at the specified Ending Station.

Cut/Fill Gaps: Use the Add and Remove buttons to define a series of station ranges for cut/fill gaps where the program will not calculate any volumes.

Fill Shrink/Cut Swell Factor: Allows you to specify a value that the volume calculated will be multiplied by.

Report Precision: Specify the decimal precision for the report.

Use Centerline to Calculate Centroids: When checked, the program will calculate the centroids using a centerline (.CL) file. You will be prompted to select the centerline file.

Use Centerline for Station Equations: This option applies to section files with stations numbered using station equations. This option will use the station equations defined in a centerline file to remove the station equations from the section file stations for calculating the true end area distances. You will be prompted to select the centerline file.

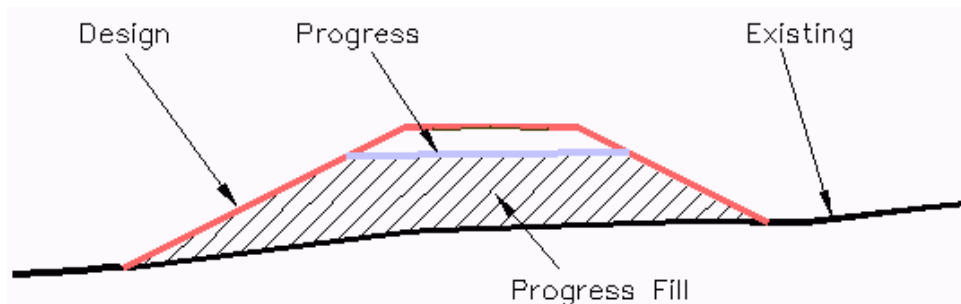
Use Inclusion Perimeter: This option prompts for a closed polyline in plan view for the limits of volumes. Without this inclusion perimeter, volumes are calculated to the extents of the sections.

Use Report Formatter: This option allows for customized report layout and contents. Otherwise a standard report is displayed.

Report Centroids: Specify whether or not to report centroids.

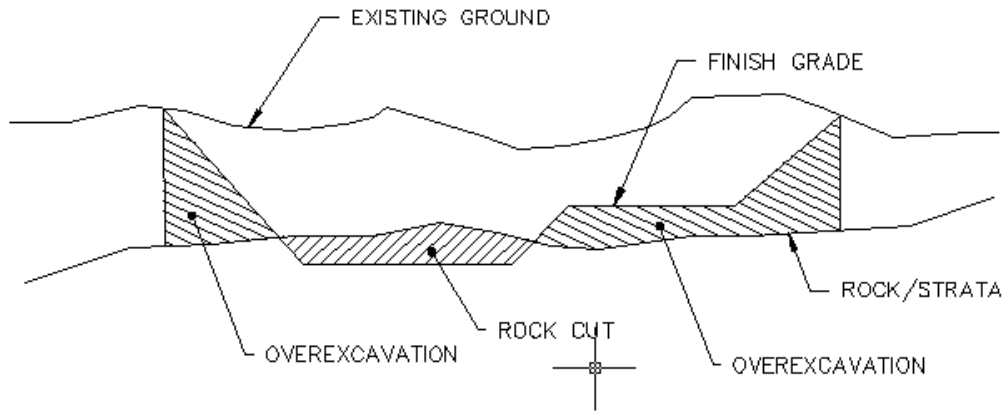
Calculate Rock Volumes: When checked, you will be prompted to select a third section (.SCT) file that will be used to calculate rock quantities.

Calculate Progress Volumes: This option prompts for a third section file that represents a stage of progress between the existing and design surfaces. The includes both the total cut/fill quantities and the subset of progress cut/fill quantities. The progress volumes are limited to be within the volumes between existing ground and design in order to avoid any extra cut or fill.



Calculate Elevation Zone Volumes: This option reports the amount of cut and fill for elevation zones. With this option, the program prompts for a starting station and elevation interval. Then starting at this elevation, the program calculates the amount of cut and fill between the section surfaces up to the next elevation. This elevation zone calculation is repeat until the highest elevation in the section files is reached.

Calculate Overexcavation: When checked, calculates volume of overexcavation. See diagram.



Report Cut/Fill Text: Specify whether or not to report cut/fill at each station.

Report Cut/Fill Differences: Adds a running total of the cut to fill balance at each station to the report.

Report Cumulative Cut/Fill: Adds a running total of the cut/fill at each station to the report.

Extend Shorter Sections to Longer: If checked, shorter sections are lengthened to the same left and right offset extents as the corresponding longer sections.

Interpolate Missing Section Stations: If checked, the missing stations are accounted for in the calculations.

Breakout Quantities by Staging: This option breaks out the cut and fill end areas and volumes according to user defined offsets along the road. The "Slope Format" determines how each stage will tie back into the road.

Breakout Quantities by Staging

Slope Format
 Ratio Percent Vertical

Cut Slope: 1.000 Fill Slope: 1.000

	Stage Name	Left Side	Offset	Right Side	Offset
1	Leftside	Left Edge		Offset Right	0
2	Right1	Offset Left	0	Offset Right	10
3	Right2	Offset Right	10	Offset Right	20
4					
5					
6					
7					
8					
9					
10					
11					
12					

Last Stage Name: Right SH

OK Cancel

In the example above, volumes will be reported separately for the left side of the road, the inside right lane (offset 0-10), the outside right lane (offset 10-20), and then the remainder right side of the road will automatically be reported as the "Last Stage".

Topsoil Adjustment File: This optional input file applies topsoil removal and replacement for the calculations. See the Topsoil Removal/Replacement command for more details on the .TOP file.

Mass Diagram Output File: Specify an optional (.MAS) file for mass haul data output for the Draw Mass Diagram and Mass Diagram Report commands.

End-Area Output File: Specify an optional end-area (.EW) file for output that can be used in the Edit-Process End

Area File command.

Prompts

Section File (Existing Ground) to Read *choose existing .SCT file*

Section File (Final Ground) to Read *choose the other existing .SCT file*

Calculate Sections Volume dialog Make selections.

Pulldown Menu Location: Sections

Keyboard Command: calcsct

Prerequisite: Two section (.SCT) files

Calculate End Area

This command allows the user to select two polylines representing an existing grade section and a final grade section, and calculate the end area. Or you can also specify and define cut/fill end areas by picking interior points. The area calculated can be drawn at a user specified point. Optionally, the command writes the stations cut and fill to an earthwork (.EW) file that can be printed/displayed by the *Edit-Process End Area File* command. This command starts with the Calculate End Area dialog.

Horizontal Scale: Specify the horizontal scale of the existing cross section.

Vertical Scale: Specify the vertical scale of the existing cross section.

Station Interval: Only available if *Write Results to EW File* is toggled on. Allows you to specify the station interval that the station prompting will default to as you select the polyline/sections for computation.

Extend Shorter Ends to Longer: Click or leave blank.

Calculate Unsuitable Areas: When checked, the user will be prompted for polyline(s) or interior point(s) that represent unsuitable material. The areas and/or volumes for the unsuitable material is reported out separately from the overall cut and fill areas and/or volumes for overexcavation consideration.

Define end areas by chooses between picking two polylines or picking inside each end area.

Text Scale: Specify the text size scaler, this value is multiplied by the horizontal scale to determine the final text

height.

Decimal Places: Controls the decimals for the cut/fill area labels.

Cut/Fill Prefix/Suffix: Specify prefix and suffix for the cut, fill, and unsuitable labels.

Label Layer: Specify the layer for the cut/fill area labels.

Write Results to EW File: When checked, the results will be written to an earthwork (.EW) file. You may create a new file or choose to append/revise an existing file.

Prompts

Calculate End Area dialog *make choices*

Specify Earthworks File (ew) dialog *specify new or existing file* This box appears if *Write Results to EW File* is clicked.

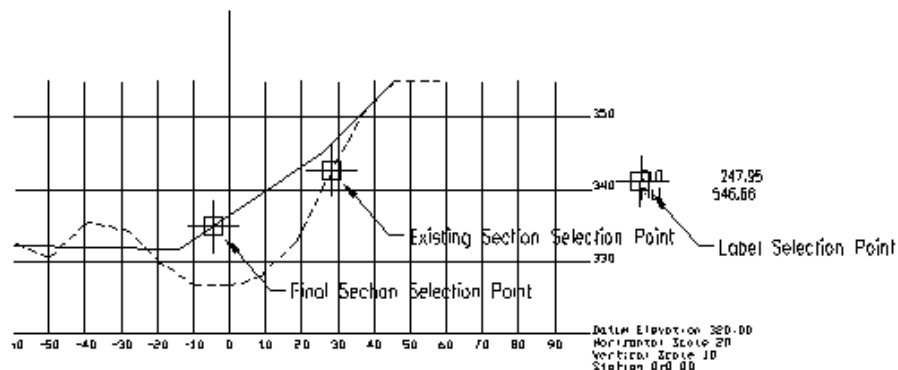
Select existing grade polyline (ENTER to end): *select polyline*

Select final grade polyline: *select polyline*

Calculating End Area...

Cut: 12002.965 Fill: 660.272

Pick Point for Label (Enter for none): *pick point*



Enter the station <0.00> *press Enter* Pressing Enter selects the default station 0+00. If the station does not exist in the file it will be added. If it does it will be revised.

Select existing polyline: *press Enter*

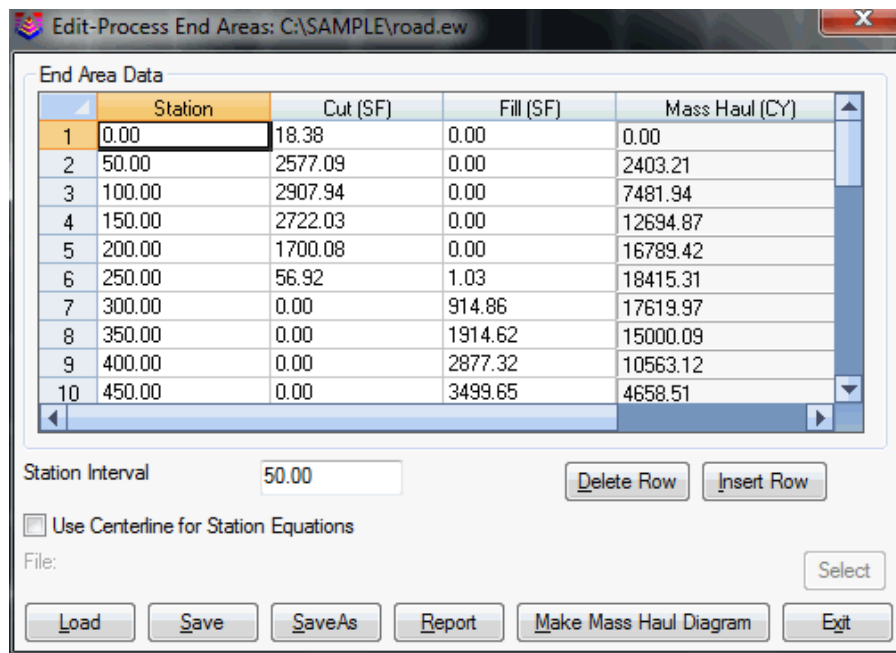
Continue moving along automatically to the next station interval and select polylines. Or enter the station values randomly. The command sorts the .EW file regardless. As a result of this sort feature, the user can select stations in any order and they will be arranged in ascending order for proper volume computation.

Keyboard Command: endarea

Prerequisite: Plot the existing grade and final grade polyline/section

Edit Process End Area File

This command opens an End Area (.EW) file for editing and processing. Data can be entered directly into the spreadsheet. The Calculate Section Volumes command has an option to create an .EW file. The accumulative volume is displayed in the right side column of the spreadsheet. The Report function outputs a report of the stations, intervals, cut and fill. The Make Mass Haul Diagram function makes a .MAS file that can be used by the Draw Mass Diagram and Mass Diagram Report commands. The Use Centerline for Station Equations option applies to end area data with stations numbered using station equations. This option will use the station equations defined in a centerline file to remove the station equations from the end area stations for calculating the true distances for the volume calculations. You will be prompted to select the centerline file.



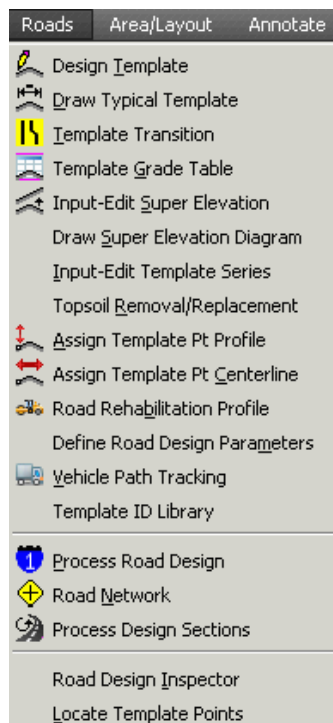
Pulldown Menu Location: Sections

Keyboard Command: ewedit

Prerequisite: None

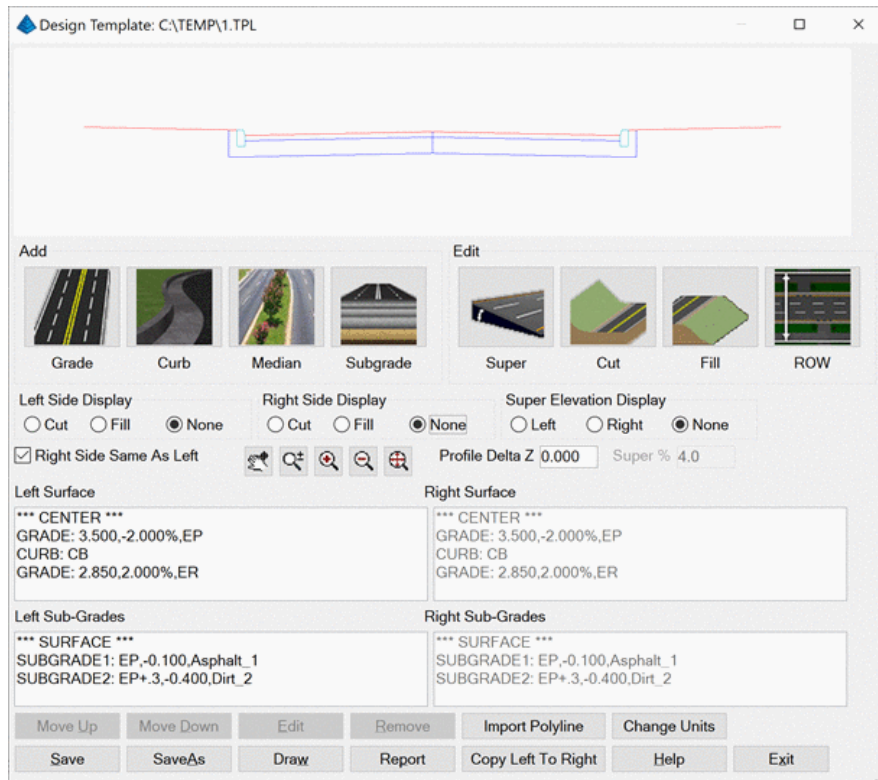
Roads Menu

The Roads menu commands covers template, transitions and processing road designs. The Roads menu is part of the Civil, Construction and Field modules.



Design Template

This command creates a template definition file (.TPL file) used primarily for modeling roadway, railway and other types of corridor designs.



Template files can then be applied through a variety of commands including (but not necessarily limited to):

- Process Road Design
- Road Network
- Draw Typical Template
- Locate Template Points
- Design Pad Template
- Draw Field-to-Finish

In the lower portion of the dialog box are four "list boxes" that list the elements of the template. The Surface elements are listed in order starting from the center of the template moving outward. The Subgrades are listed from top to bottom order. To add a template element, highlight the position in the list above where to insert the element. There is no limit to the number of Surface or Subgrade elements. The following types of objects can be utilized to create a template:

Button



Description

Grades - Typically, linear segments that defined paved surfaces such as roadway pavement, sidewalks and other linear surfaces that are "exposed to daylight."

Curb - Typically, "formed" shapes that help control storm-water run-off.

Median - Typically, "free-form" features such as "roll-over" medians or other types of traffic barricades.



Subgrade - Subsurface material(s) that are used to derive "pavement" volumes (e.g. bituminous asphalt, aggregate base course, etc).



Superelevation (Super) - Parameters that indicate how (and where) the template "grades" should elevate within curves or other areas of superelevation.



Cut Slopes (Cut) - Specialized "grade" slopes that are used to "tie" the template to an existing surface when the elevation of the outermost point of the template is below ground.



Fill Slopes (Fill) - Specialized "grade" slopes that are used to "tie" the template to an existing surface when the elevation of the outermost point of the template is above ground.



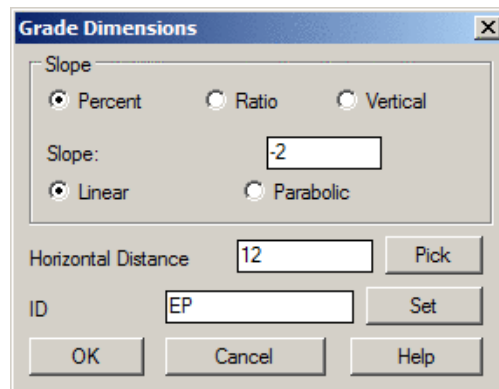
Rights-of-Way (ROW) - Additional constraints utilized by the template in an effort to minimize the "footprint" of the disturbed limits of a cross-section relative to the right-of-way based on a given centerline.

Template Components

Additional categories of functionality include:

- View Controls
- Data Tools

Grades



Percent: Specify this option when the "s" vertical displacement amount of a slope is specified relative to 100 horizontal units of measure.

Ratio: Specify this option when the "s" horizontal amount of a slope is relative to 1 units of measure vertically.

Vertical: Specify this option for an "undefined" (vertical) slope ("s") in which there is no horizontal component.

Slope: Indicate the "s" value relative to the type of slope selected above. Positive slope values indicate an increase in elevation along the grade segment; negative slope values indicate a decrease in elevation along the grade segment.

Linear: Specifies a constant slope along the grade segment.

Parabolic: Specifies a varied slope that gets steeper across the grade until it reaches the full specified slope at the end of the grade.

Horizontal Distance: Specify the horizontal distance component (positive values only) of the grade segment.

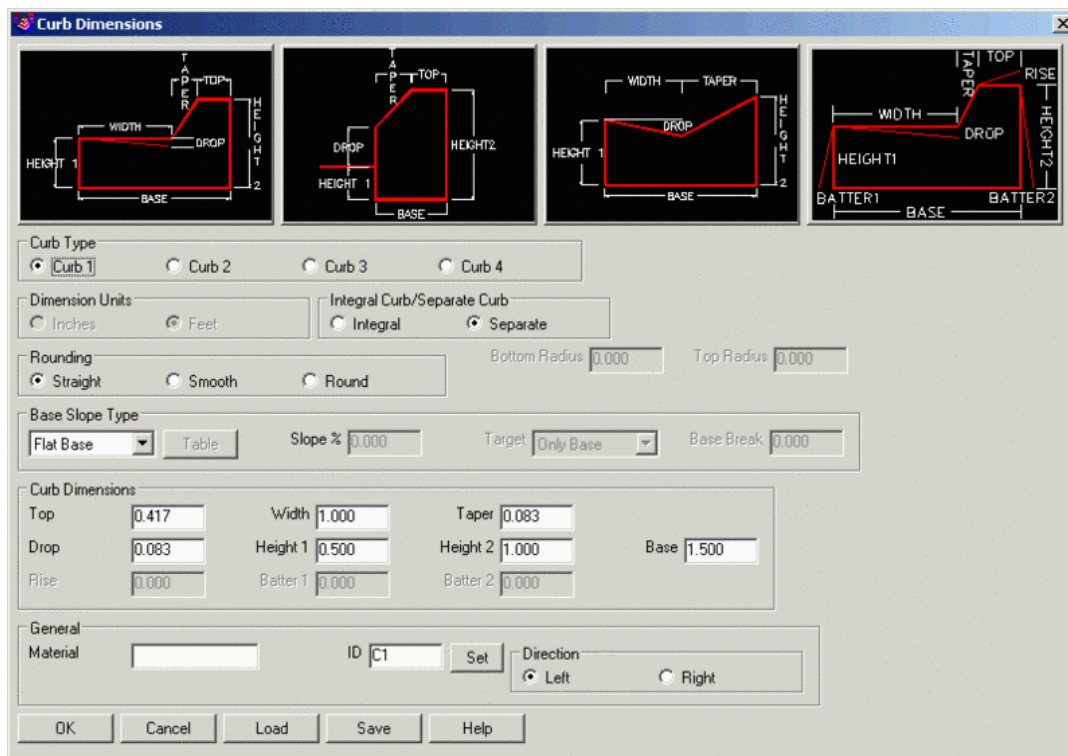
Pick: Prompts to select a linework segment or two points from the drawing to define the grade slope and distance.

ID: The ID (typically a two- or three-character identifier) serves four purposes:

1. The ID will be applied as a description to all final template points generated in the form of a coordinate file.
2. The ID can be used as a parametric design point for subgrade offsets and distances (*e.g.* EP+0.5).
3. Points of common ID may be connected by 3D polylines as an output option of Process Road Design.
4. Quantities can be generated with reference to the ID and material (gravel, concrete, *etc*) entered elsewhere within this command.

Set: Displays a list of grade ID's from the current Template ID Library.

Curb



Curb Type: Select the general shape of the curb "plate" that best matches the curb you wish to define. If your curb does not conform to one of these shapes, you can also utilize the Median component.

Dimension Units: Indicate the desired units for the Curb Dimensions (*e.g.* feet, inches or meters in metric mode).

Integral Curb/Separate Curb: Indicate whether to draw the front line of the curb to separate the curb from the subgrade. For example, fully concrete pavements that contain a curb would be drawn with the "integral" curb option:



Rounding: Indicate the type (or amount) of "smoothness" that should be applied between curb faces.

- Straight - no smoothing occurs between curb faces.

- Smooth - this option will smooth the surface of the curb which only shows when the template is applied in commands such as Process Road Design.
- Round - this option fillet a curve at the bottom and top of the *Taper* segment using the specified Bottom and Top Radius.



Straight

Base Slope Type: The Base Slope Type of the curb can either be flat, set to the slope of the incoming grade or set to a user-specified slope. For the Match Crown method, you can use the Table option to define a lookup table of different curb slopes for different crown grades. For cases with part of the curb at a slope and part flat, you can use the Base Break Offset to set the transition position between sloped and flat. The Target setting for the slope controls which parts of the curb are sloped.

Curb Dimensions: Indicate the various dimensions (in units defined by Dimension Units) of the curb components.

Material: Indicate the name of the material for the curb for reporting purposes from commands such as Process Road Design.

ID: The ID (typically a two- or three-character identifier) serves four purposes as discussed in this ID discussion.

Set: Displays a list of grade ID's from the current Template ID Library.

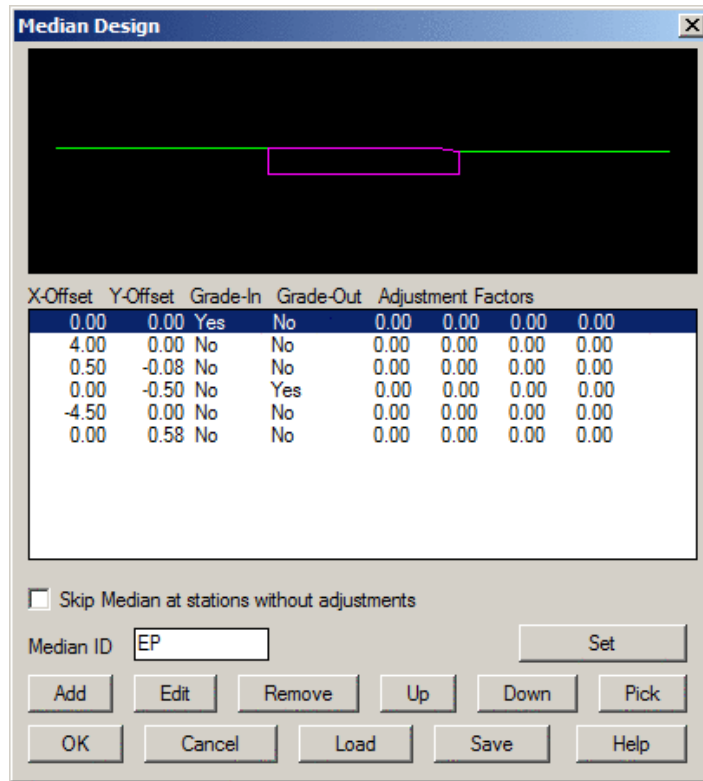
Direction: This option controls which way the curb faces and is needed for divided roadway templates that have curbs facing both ways on either side of the road.

Load: Allows a previously saved curb (*.crb) "plate" to be opened for editing.

Save: Commits the current curb "plate" to a named curb (*.crb) file.

Median

The Median is a flexible, closed figure defined in a clockwise direction. Each median point consists of an X and Y offset. The median must be closed and the program will automatically create the closing segment. The display shows the median in magenta and the grade lines in and out in green. For the display the grade in comes from the left and the grade out goes to the right. The median must define the Grade In point which is the point that ties into the incoming surface grade. Also the Grade Out point must be specified for where the surface grade continues out from the median. These Grade In and Grade Out points emanate from the starting or "from" position in the coordinate dialog where they are specified. Since a single median must be placed on the left or right side (and is typically not used symmetrically with right side same as left), you will need to offset the template centerline one-half the median width within the command Process Road Design in order to center the median. You will also have to move the "C/L" designation, to obtain centering, when using Draw Typical Template.

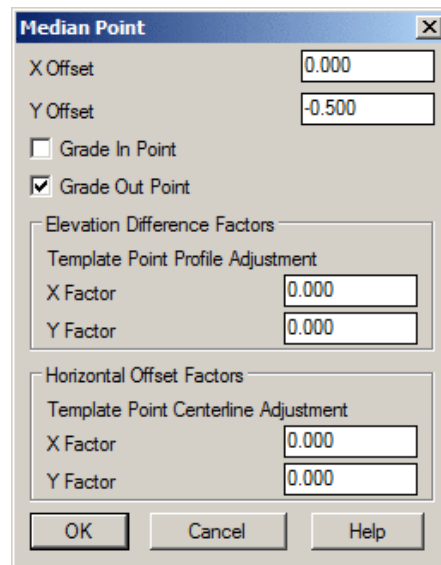


Skip Median at Stations without Adjustments: When enabled, this option creates the median only in the station ranges of a Template Point Profile and/or Template Point Centerline transitions.

Median ID: The ID (typically a two- or three-character identifier) serves four purposes as discussed in this ID discussion.

Set: Displays a list of grade ID's from the current Template ID Library.

Add (Edit): To enter the dimensions of the median, use the Add or Edit button to display a dialog box similar to that below:



X/Y Offset: Provide the X/Y displacement of the median segment relative to the end of the previous median segment.

Grade In/Grade Out Point: Indicate if the median point is the position that accepts the incoming grade and/or generates the location for the out-going grade segment of the template.

Elevation Difference Factors: Controls how to apply Template Point Profiles. For Template Point Profiles, the program figures the amount of vertical adjustment between the transition profile and the normal profile. The amount of this vertical adjustment is multiplied by the adjustment factor and then added to the X/Y Offsets of the median point.

Horizontal Offset Factors: Controls how to apply Template Point Centerlines. For Template Point Centerlines, the program figures the horizontal adjustment between the transition centerline and normal centerline for Template Point Centerlines and applies this adjustment by the factors to the offsets.

NOTE: These adjustment factors allow for dynamic medians. For example, the height of a retaining wall could be controlled using a Template Point Profile and the median points for the vertical sides would have a Y Factor set to 1 to pick up the full vertical adjustment and the median points for the top and bottom edges would have a Y Factor of 0 keep those edges the same.

Remove: Deletes the highlighted X/Y Offset record in the list.

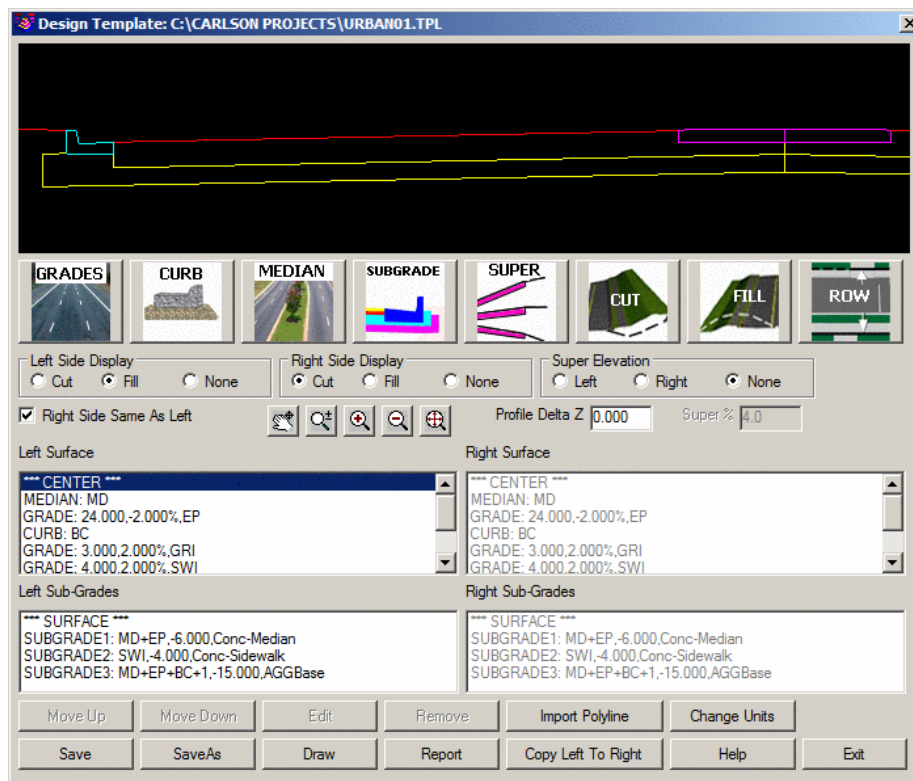
Up (Down): Changes the order of the highlighted X/Y Offset record in the list.

Pick: The Pick button prompts to pick a closed polyline from the drawing to define the median geometry.

Load: Allows a previously saved median (*.mdn) to be opened for editing.

Save: Commits the current median to a named median (*.mdn) file.

You can design a median for "mirroring" to create a centered effect, as illustrated in a paved "rumble strip" section as shown below. The only negative to this method is the appearance of a vertical line in the median plot.



Subgrade

Clicking the Subgrade button displays the dialog box below that permits you to establish "design" materials used to build the design project. Subgrade entries are displayed below the Grade entries and there can be any number of subgrades stacked below one another or side by side.

NOTE: The subgrade values for Horizontal Offset, Distance and Pivot Offset can be specified by *Template ID* points (consider using the Template ID Library to create a consistent set of template ID codes). For example, *EP* could be used in Distance to have the subgrade assume (and be controlled by) the width of the "EP" grade. Additionally, expressions can be used such as *EP+BC* to adopt the distance of the EP segment plus the distance of the BC segment. This is especially useful for template transitions so that if the width of the EP grade varies, the subgrade width will automatically adjust.

Subgrade Type: Chooses where to apply the subgrade. The roadway is the default for subgrades under the road grades. The Cut and Fill Slopes are for subgrades between the pivot (hinge) point at the end of the road grades and the catch (tie) point.

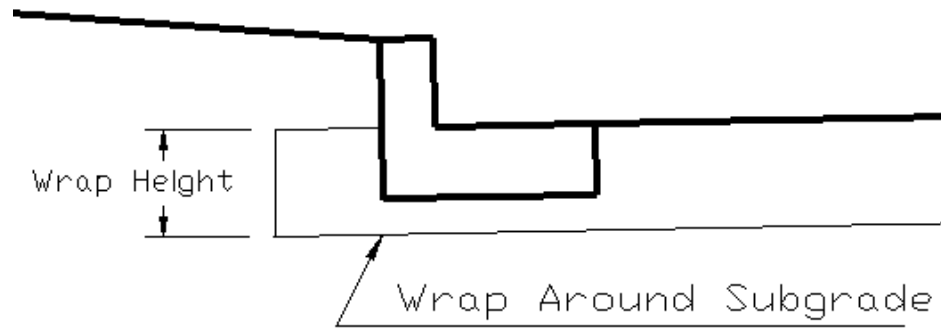
Slope Type: Indicate if the "bottom" of the subgrade surface should match the slope of the grade surface above it or if the subgrade slope should travel at a special slope value. Remember, a positive slope value increases in elevation further from the centerline while a negative slope value decreases in elevation further from the centerline.

Distance Direction: Indicate if the bottom of the subgrade moves away from (Out) or toward (In) the center of the template.

Intersect Surface: Indicate how the subgrade surface should "tie" into another component of the template to form a closed region:

- **Continue Slope:** Specify this option when the subgrade surface should continue along the slope as specified in the Slope Type control. Common uses of this intersect method is for aggregate surfaces used in rural design scenarios and/or railway ballast scenarios. When set to Continue Slope and the subgrade can't find an intersection because the subgrade is too low, then the **Tie Undershoot Vertically At Hinge** option can be used to make the subgrade switch to Straight Up method at the pivot point offset.
- **Straight Up:** Specify this option when the slope is to tie vertically to the surface above it. Common uses of this intersect method are for "rigid" shapes such as concrete roadways or sidewalks and for bituminous asphalt abutting Curb & Gutter.

- **Wrap Around:** Specify this option when the subgrade material is to extend beyond the limit of a previous component (typically, a curb) and then return toward the center of a driving lane to form a closed region as illustrated below:



When enabled, the **Wrap Height** control will become enabled that allows the wrap height (units of feet or meters) to be specified.

- **Tie Slope:** Specify this option when the subgrade is to tie into a previous element with a desired slope value. This slope type is commonly used when a subgrade region is being excavated with a "lay-back" slope. When enabled, the **Tie Slope** control will become enabled that permits the desired tie slope value (*e.g.* a 100% slope = a 1H/1V or 1:1 slope).

Horizontal Offset: Indicate the distance (by numerical or para-metrical equation) from the template centerline where the subgrade should start. The subgrade moves straight down from this location the depth specified in the *Vertical Offset*.

Vertical Offset: Indicate the *cumulative* depth from the elevation of the specified *Horizontal Offset* plus the desired thickness of the subgrade material. For example, if 6" of a pavement subgrade has been defined and 8" of aggregate subgrade is desired upon which the pavement is built, the vertical offset of the aggregate would be specified as 14". The **Match Existing Surface** option adjusts the subgrade depth to match the existing surface which applies for road rehabilitation when the subgrade is overlaying the existing surface.

Units: If applicable, specify the desired units of measure for the *Vertical Offset* control described above.

Distance: Indicate the distance (by numerical or para-metrical equation) of the desired width of the subgrade to the point where the Intersect Slope should start.

Material: Indicate the name of the material for the curb for reporting purposes from commands such as Process Road Design.

Coatings/Notes: This optional description is used in reporting and for labeling in Draw Typical Template.

Superelevation Settings: Indicate if special superelevation pivot points will be used:

Pivot Offset: Indicate the distance (by numerical or para-metrical equation) of the desired superelevation pivot point. This location allows the subgrade slope to break in superelevation independently from where the surface grade breaks. The subgrade will follow the superelevation slope from the centerline to the Pivot Offset. After the Pivot Offset, alternative slope factors govern.

Max Slope After Pivot: Indicate the maximum allowable slope for the subgrade once the Pivot Offset has been reached.

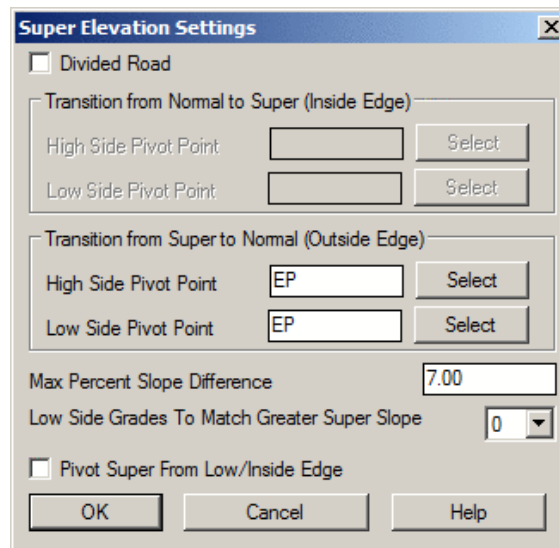
Slope Type After Pivot: Indicate how the slope of the subgrade after the Pivot Offset should be governed:

- **Normal:** This option sets the slope the same as the non-superelevation state.
- **Special:** This option can be used to set the slope to a specific value.
 - **Standard Slope Percent:** Indicate the default slope that should be attempted.

- **Minimum Slope Percent:** Indicate the minimum allowable subgrade slope.

Superelevation (Super)

Clicking on the "Super" (superelevation) button provides an interface of how the template should behave in zones of superelevation via the Superelevation Editor command.



Divided Road: When enabled, the *Transition from Normal to Super (Inside Edge)* controls will be enabled.

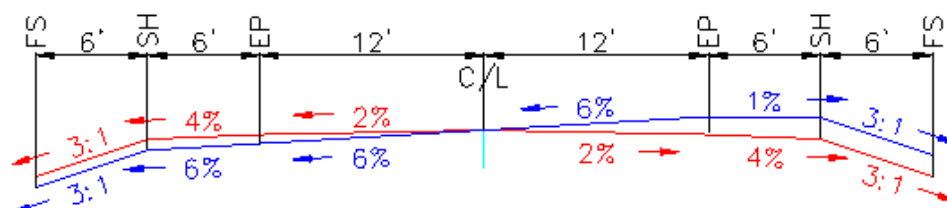
Transition from Normal to Super (Inside Edge): When enabled, identify (by Template ID point or para-metrical equation, *e.g.* EPI+2 or EPI-2) where superelevation pivot points (typically, inside edge of pavement) should occur for the High and/or Low locations of the Pivot Points for a Divided Roadway template.

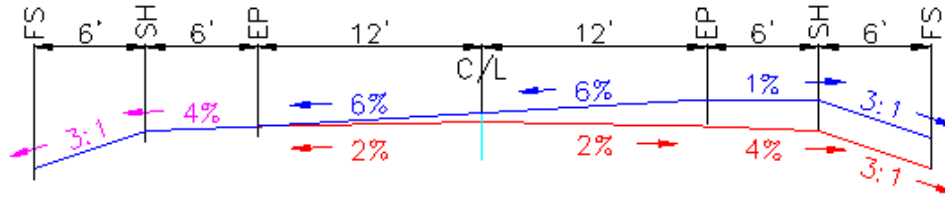
Transition from Super to Normal (Outside Edge): Identify (by Template ID point or para-metrical equation, *e.g.* EP+2 or EP-2) where superelevation pivot points should occur for the High and/or Low locations of the Pivot Point for an Undivided Roadway template.

Max Percent Slope Difference: Indicate the maximum allowable grade difference between the segments on either side of the Pivot Point. Once this maximum allowable slope difference is achieved, template segments outside of the pivot point will elevate as a unit relative to the grade segment inside the Pivot Point.

Low Side Grades to Match Greater Super Slope: This option applies to the template grades that are outside the superelevation pivot point. When the superelevation slope becomes steeper than these outside grades, then these grades are adjusted to match the same super slope. You can adjust up to two additional grades past the superelevation pivot point. For example, consider a template where the superelevation pivot point is the EP grade and the next grade is a shoulder (SH) and you desire this first segment to adjust with the EP grade. Set the "Low Side Grades..." control to "1". If the regular SH slope is -4%, the SH will stay at -4% through the superelevation until the superelevation rate becomes greater than 4% (at this point, the SH crown has been removed). If the superelevation rate of the EP segment continues to 6%, the SH segment will steepen to -6% to keep the SH crown removed.

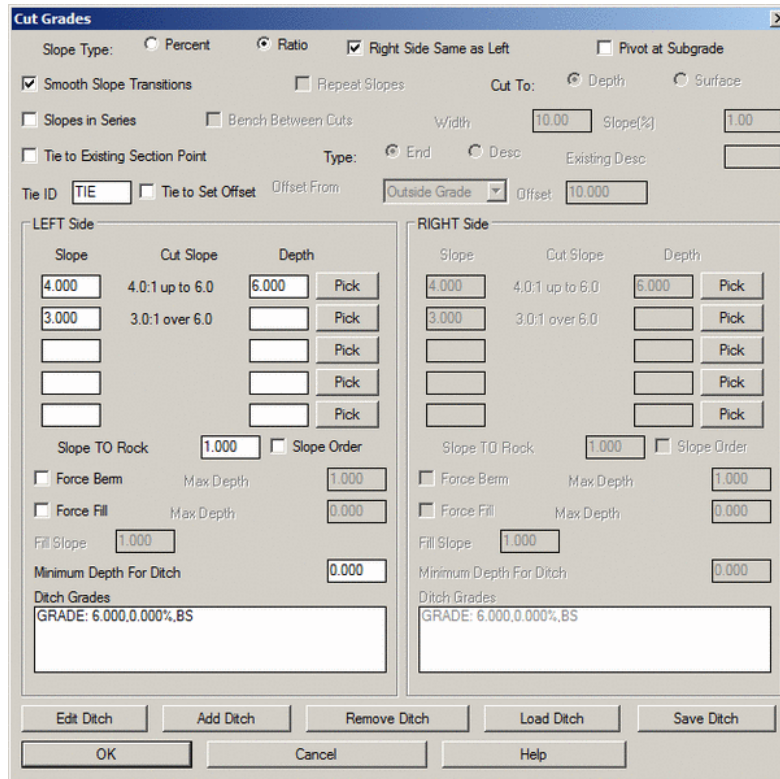
Pivot Super From Low/Inside Edge: Commonly used in Divided Roadway scenarios, when this option is enabled the elevation of the Inside Pivot Point relative to the design profile elevation is held fixed and segments outside of the Low/Inside Pivot Point are raised. There is an edit box to enter in the grade ID for this pivot super point which defaults to the Low Side Pivot Point if this is left blank.





Cut Slopes (Cut)

Use the Cut button to define cut-slope treatments when the outermost point of the template is below existing ground.



Slope Type: Indicate the type of slope unit used for the actual Cut Slope(s)/Fill Slope(s).

Right Side Same as Left: When enabled, the cut-slope treatments of the Left side of the template will also be used for the Right side of the template.

Pivot at Subgrade: This option will position the cut pivot point where the bottom subgrade intersects the template grade. The ditch or up-slope conditions will then occur from this special subgrade "daylight" pivot point, instead of from the outer grade pivot point.

Smooth Slope Transitions: When enabled, this option will gradually transition the slopes from one range to the next. In the Cut Grades dialog box example above, if the depth is 6 feet the slope will be in transition between 4:1 and 3:1.

Slopes in Series: When disabled, a single slope is used based on the depth of the Cut Pivot Point. When enabled, each slope is used to its target depth until an intersection with the ground is reached. Each successive slope in the series starts from where the previous slope ended.

Repeat Slopes: When enabled, the pattern of Cut slopes will repeat until the sequence of slopes "daylight" with existing ground.

Cut To: In addition to bench cut slopes to specific **Depths**, the Cut To option can be used to have each bench cut slope intersect a surface such as a rock section file. With Cut To set to **Surface**, the Process Road Design command will prompt for the cut slope surface for each bench. The bench surface can be either a cross section file, triangulation surface or user-specified fixed elevation. It is important to note that when cutting to a surface, you must specify a Slope and a Depth, even though the depths will not be used. In the below image, a slope of 3:1 is being used up to the first bench, a slope of 2:1 is being used between the first and second bench, and a slope of 1.5:1 is being used between the second bench and the topography.

LEFT Side			
Slope	Cut Slope	Depth	
3	3.0:1 up to 10.0	10.000	Pick
2	2.0:1 up to 20.0	20.000	Pick
1.5	1.5:1 over 20.0		Pick
			Pick

Bench Between Cuts: This option allows you to enter a "bench" pattern consisting of a width at a percent slope to be inserted between each cut slope. This option is commonly used in deep rock cuts as a means to help protect vehicles from loose/falling debris in the rock-cut areas.

Tie to Existing Section Point: When enabled, this option will tie the cut slope from the cut pivot point to either:

- **End:** The outer-most offset-elevation from the Existing Section file.
- **Desc:** The offset-elevation point with a specified description from the Existing Section file. This method is used when survey crews collect sections and designate the specific slope tie points.

Tie ID: Indicate the desired identification where the slope intersects the Existing Surface. This is the same setting as under the Fill Grades dialog.

Tie to Set Offset: When enabled, this option forces the cut slope catch point to a specified fixed offset. This offset can be relative to the:

- **Outside Grade:** The offset is determined from the outer-most Grade of the template.
- **Centerline:** The offset is determined from the Centerline and can be used when you want the slope to always tie into existing ground at a fixed Right-of-Way offset.

Cut Slope(s): There is room to specify up to five cut slopes that can be applied at specific depths. In a simple case of a single cut slope, supply the Slope value and leave the Depth and remaining Slope/Depth boxes blank.

Pick: These buttons prompt to select two points or a linework segment from the drawing to define the cut slope:

Slope to Rock: Indicate the slope that should be used when using a rock section file within Process Road Design.

Slope Order: Enable/Disable this toggle to switch between the two Slope Order modes for rock slopes:

1. **Slope TO Rock:** The cut slope will be the Slope To Rock up to the rock surface. After reaching the top of the rock surface, the regular cut slopes apply.
2. **Slope FROM Rock:** The regular cut slopes apply up to the rock surface. Then the slope from the Slope From Rock applies from the top of the rock surface to the ground surface.

Force Berm: This option will apply the Berm (defined using the Fill icon) in cut up to the **Max Depth** of cut. The **At Top of Cut** controls whether to place the berm at the pivot point from the road, or at the tie point where the cut slope intersects the ground.

Force Fill: When enabled, this option will make the template attempt to find a catch point with a specified **Fill Slope** even when the pivot point is in cut to a **Max Depth** in cut.

Minimum Depth for Ditch: When the Cut Pivot Point is shallower than this value, the ditch grade(s) will not be applied. To always apply the ditch grade(s) when in cut, specify a value of 0.

Ditch Grade(s) can be inserted between the Cut Pivot Point and the Cut Slope(s) as a means to further control and channel storm-water run-off. Ditch Grades are essentially identical to the Grades option used in the regular template. The location of the Cut Pivot Point is based on the outer-most Grade or Median segment. Consider one of the following scenarios:

- Cut Pivot Point is "Shoulder Point" - In this scenario, the last grade segment is taken to be a shoulder point. You could use:
 1. One ditch grade to create a "V-bottom" ditch (the single ditch grade emanates from the shoulder and forms one segment of the ditch with the cut slope comprising the other segment, or,
 2. Two ditch grades to create a "trapezoidal" ditch (the first ditch grade emanates from the shoulder with the second ditch grade forming the trapezoidal ditch bottom with the cut slope comprising the final segment.
- Cut Pivot Point is a "Toe of Foreslope" - In this scenario, the last grade segment is taken to be an additional slope (typically downward) from the shoulder point that is placed regardless of Cut vs. Fill. You could use:
 1. No ditch grades to create a "V-bottom" ditch. The last grade of the template and the cut slope comprise the V-bottom, or,
 2. One ditch grade to create a "trapezoidal" ditch (the single ditch grade forming the trapezoidal ditch bottom).

The option you use is ultimately governed by how/where the Cut Pivot Point should be defined.

Edit Ditch: Highlight a desired ditch grade and use this option (or simply double-click on the ditch segment) to adjust its value(s).

Add Ditch: Use this option to add any number of ditch grades. See the Grades for information pertaining to the various controls.

Remove Ditch: Highlight a desired ditch grade and use this option to remove it from the ditch grade list.

Load Ditch: Recalls a previously saved ditch (*.DIT) file.

Save Ditch: Saves the current ditch geometry to a ditch (*.DIT) file.

Fill Slopes (Fill)

Use the Fill button to define fill-slope treatments when the outermost point of the template is above existing ground. The options for Fill Slopes treatment are similar to those used for Cut and any difference(s) are noted below.

Slope Type: Refer to Cut – Slope Type for information.

Right Side Same as Left: Refer to Cut – Right Side Same as Left for information.

Pivot at Subgrade: Refer to Cut – Pivot at Subgrade for information.

Smooth Slope Transitions: Refer to Cut – Smooth Slope Transitions for information.

Slopes in Series: Refer to Cut – Slopes in Series for information.

Repeat Slopes: Refer to Cut – Repeat Slopes for information.

Fill To: In addition to bench fill slopes to specific **Depths**, the Fill To option can be used to have each bench fill slope intersect a surface. With Fill To set to **Surface**, the Process Road Design command will prompt for the fill slope surface for each bench. The bench surface can be either a cross section file, triangulation surface or user-specified fixed elevation.

Bench Between Fills: Refer to Cut – Bench Between Cuts for information.

Tie to Existing Section Point: Refer to Cut – Tie to Existing Section Point for information.

Tie ID: Refer to Cut – Tie ID for information.

Tie to Set Offset: Refer to Cut – Tie to Set Offset for information.

Fill Slope(s): Refer to Cut – Cut Slope(s) for information.

Pick: Refer to Cut – Pick for information.

Force Cut: When enabled, this option will make the template attempt to find a catch point with a specified slope even when the Pivot Point is in Fill. You can specify the Cut Slope to use and the **Max(imum) Depth** for the slope.

Force Ditch: When enabled, this option has two different methods to apply the Ditch Grades from the Cut definition when the **Max(imum) Depth** of fill has not been exceeded:

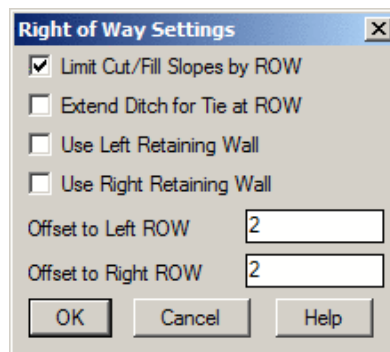
1. **At Base Of Fill: (enabled)** When enabled, the Force Ditch method creates the ditch where the Fill slope intersects existing ground. The **Uphill Only** option creates this base of fill ditch only when the existing ground goes uphill from the tie point.
2. **At Base Of Fill: (disabled)** When disabled, the Force Ditch method applies the ditch grades from the template pivot point.

Use Guardrail: When enabled, this option will extend the last template surface Grade the specified **SHD Extension** distance when the Fill is greater than the Min(imum) Depth.

Use Berm: Berm Grades are the fill equivalent to Ditch Grades. The **Minimum Depth** for Berm Grades will only draw the Berm Grades when the Fill depth is greater than the specified value.

Rights-of-Way (ROW)

The ROW (Right of Way) icon brings up the dialog shown which provides several methods that help govern how/where Cut and Fill slopes intersect existing ground. Right-of-Way data is stored in a Centerline (.CL) file as stations and offsets for the left and right sides of a centerline.



Limit Cut/Fill Slopes by ROW: When enabled, the Cut or Fill slope will become steeper in order to tie into the ground at the Right-of-Way offset. For example, if the Cut slope is 4:1 but this slope ties into the ground past the Right-of-Way, the slope will be modified to something steeper such as 3:1.

Extend Ditch for Tie at ROW: When enabled, this option allows you to tie the catch-slope into the original ground at the Right-of-Way offset by widening the lowest ditch slope along its slope until the Row-of-Way limit is attained. Common practice for the utilization of this option is to obtain additional fill material and/or for the placement of linear fences along the catch-slope (Right-of-Way).

Use Left/Right Retaining Wall: When enabled, the Cut or Fill slope will be applied until the Right-of-Way offset and will then tie into the ground via a vertical slope.

Offset to Left/Right ROW: Specify an additional "buffer" amount (if any) inside the Right-of-Way that is to serve as the desired Right-of-Way catch-point.

Left/Right Side Display: Choose the type of catch-slope treatment (*e.g.* **Cut**, **Fill** or **None**) that should be *displayed* in the dialog box *preview* of the template.

NOTE: The appropriate Cut/Fill slope will be applied to the cross-sections regardless of the Display setting.

Super Elevation: Choose the type of superelevation (*e.g.* **Right**, **Left** or **None**) that should be *displayed* in the dialog box *preview* of the template relative to the specified **Super %** amount.

NOTE: The appropriate superelevation will be applied to the cross-sections regardless of the Display setting.

Right Side Same As Left: When enabled, template geometry specified for the left-side of the template will be mirrored to the right-side of the template to create a symmetrical template. For unique geometry on either side of the template, disable this option and click/highlight in the desired side (Left *vs.* Right, Surface *vs.* Subgrade) to establish the desired geometry.

Profile Delta Z: This option applies a vertical offset to the profile when the template is processed. This vertical offset applies to a template that has an element at the profile position that makes the template different than the

profile grade. For example, when the profile grade is along the centerline at the pavement level and the template has a curb or median that is 0.5 above the pavement, then set the Profile Delta Z to 0.5 so that the template matches the pavement grade after coming down from the 0.5 curb or median.

Button



Action

"Pans" the Preview display of the template (same as a press/hold of the middle-button of a 3-button mouse).

"Dynamic zoom" of the Preview display of the template (same as a "wheel roll" of a wheel-button mouse).

Incremental "zoom in" of the Preview display of the template.

Incremental "zoom out" of the Preview display of the template.

"Zoom extents" of the Preview display of the template that fits/restores the content of the template into the Preview display.

Zoom/Pan Controls

Move Up/Down: Highlight an element whose position you want to change and click the appropriate button to move the template element up or down through the other Surface or Subgrade items.

Edit: Highlight an element whose data you want to modify. Alternatively, double-click on the item to edit it.

Remove: Removes the highlighted element from the list.

Import Polyline: This routine adds Grade element(s) to the template from a selected polyline. The polyline can either be drawn in cross section or in 3D. For the cross section polyline, the delta X of each polyline segment is used for the grade distance and the delta Y is used for the grade slope. For a 3D polyline, the horizontal distance of each polyline segment is used for the grade distance and the polyline elevations are used for the grade slope. After selecting the polyline, the routine prompts for the centerline point along the polyline which is the reference point for the template zero offset position.

Change Units: This option allows you to apply a scale factor to the distances in the template which can be used to convert between English- and Metric-based templates.

Save: Saves the current change(s) to the template (*.TPL) file.

Save As: Description of Control.

Draw: Refer to the Draw Typical Template documentation for information pertaining to this command.

Report: This option provides two different report formats:

1. **Detail Report:** This report provides all of the dimensions and their IDs of the template elements.
2. **IDs Report:** This report provides just the ID's of the template elements.

Copy Left to Right: This action is essentially the same as the Right Side Same As Left in that it clones the current elements on the left side of the template to the right side of the template.

Prompts

Select polyline: *Select a POLYLINE, LWPOLYLINE or LINE entity whose X,Y geometry should be used for the Distance/Elevation change of a particular Grade element.*

Pick centerline position on polyline [end on]: *Pick the location of where the template centerline is relative to the entity selected above.*

Pull-down Menu Location(s): Civil > Roads, Construction > Roads, Takeoff > Roads

Keyboard Command: template

Prerequisite: None

Draw Typical Template

This command draws a template and labels the slopes and distances. The cut and fill treatment can be shown on the left and/or right sides. All the cut/fill slopes are shown for the different depths when multiple slopes are defined. There are options to draw the normal template, super elevation or details of different sections.

You will be prompted to select the template (.TPL) file first, then the Typical Section dialog appears. Specify the parameters and press the Draw button.

Draw Typical Template

Output Options

Superelevation Left Normal Superelevation Right

Detail of Left Super Pavement Detail Detail of Right Super

Shoulder Detail Median Detail Curb Detail

Dimensioning Options

Label Grade IDs Drop Trailing Zeros Text Above Line

Distance Decimals: 0.000 Distance In Inches: Off Suffix:

Slope Decimals: 0.00 Label Slopes: None Surface Only Surface and Subgrades

Draw Title

Draw Subgrade Legend: None Hatch Subgrade

Drawing Scale: 1.000 Vertical Exaggeration: 1.000

Text Scaler: 0.250 Layer: TEMPLATE

Max Super Elevation: 4% 6% 8% 10% Special Super Elevation: 4.000

Left Side Treatment: Fill Cut None

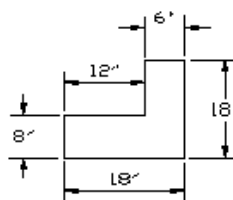
Right Side Treatment: Fill Cut None

Prompts

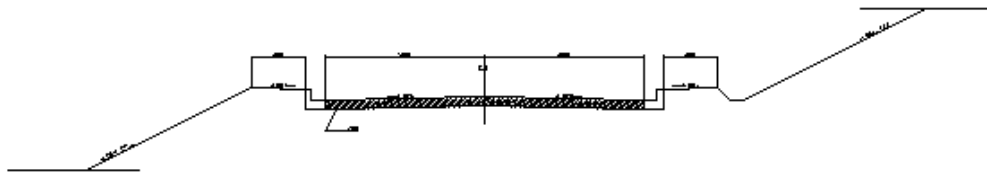
Template File to Read Specify a template file.

Typical Section dialog Set your options then click Draw.

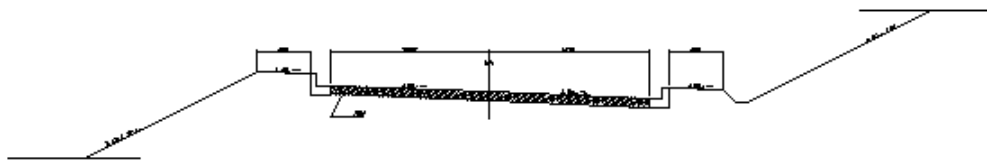
Pick Starting Position: *pick a point*



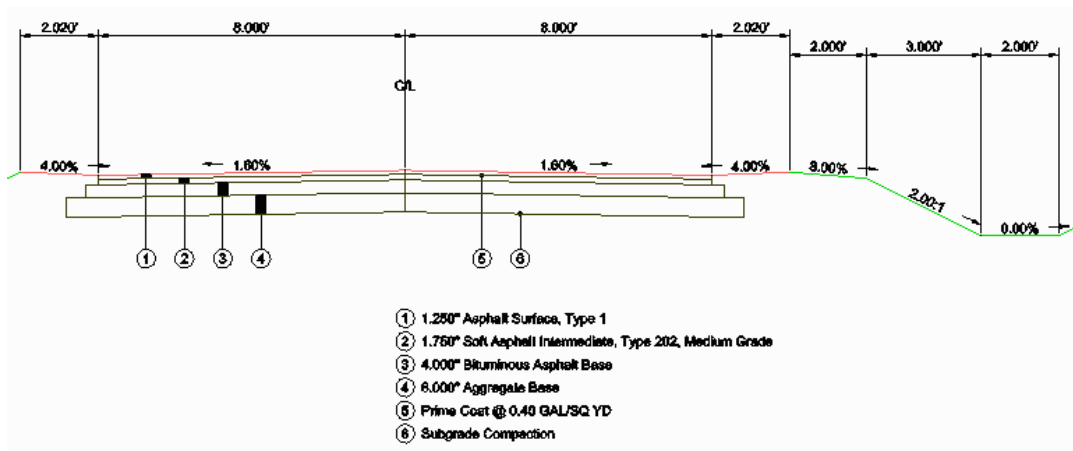
Curb Detail



Normal Typical Template



Typical Template with Left Super Elevation



Typical Template with Subgrade Legend

Pulldown Menu Location: Roads

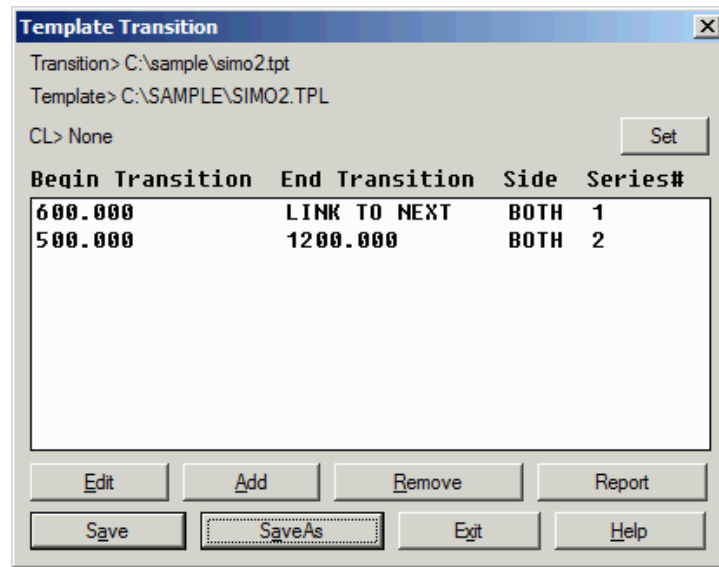
Keyboard Command: tempdraw

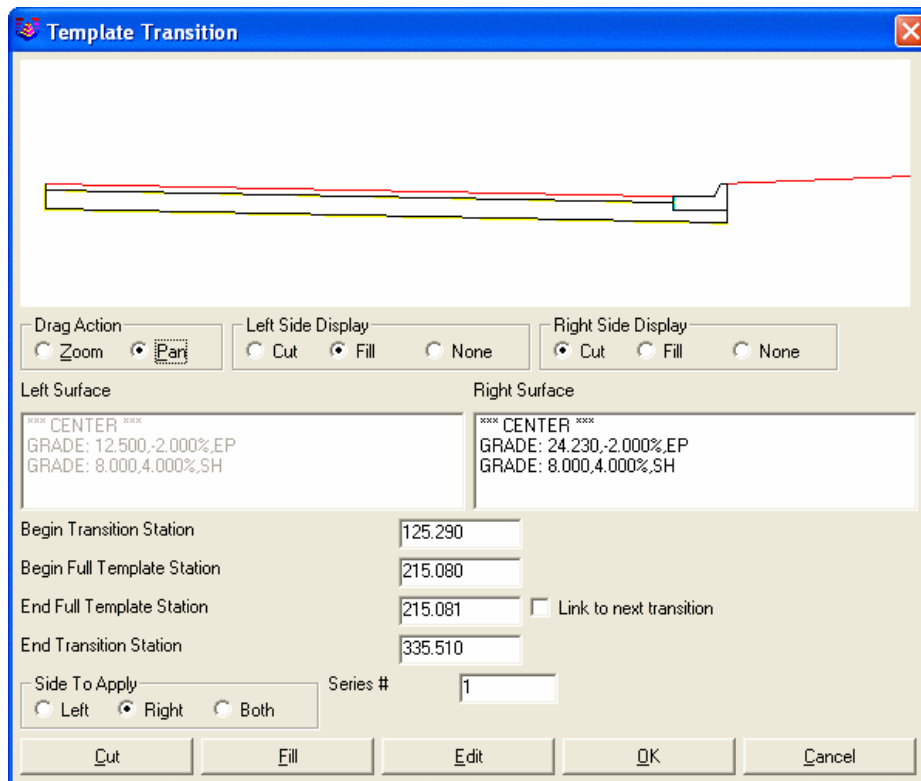
Prerequisite: A template file (.TPL file)

Template Transition

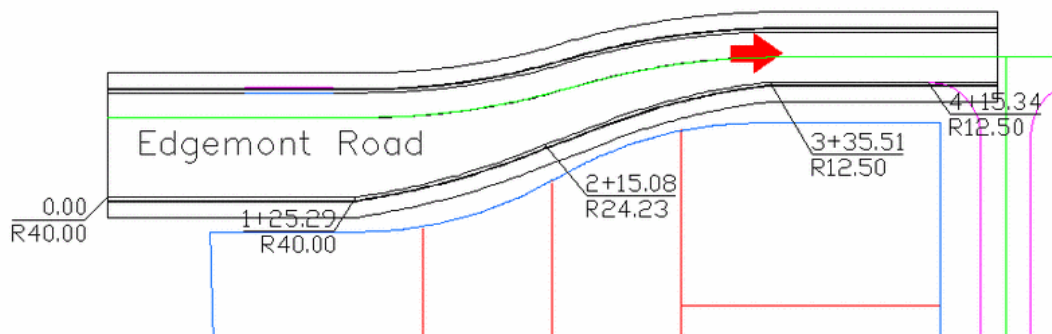
This command creates a template transition file (.TPT file) that can be used for the commands *Locate Template Points* and *Process Road Design*. The template transition is associated with a typical template (.TPL) file. The

template transition file defines changes in grade distances or slopes for a specific template ID through a specified range of stations. Lane widths, for example, can be made to expand and contract. You can only modify existing template grades. Template Transition does not allow curbs, medians, subgrades or cut/fill treatment to be modified. Also new template elements cannot be added and existing elements cannot be removed. For this reason, lanes of road that "emerge" and slope distinctly from standard road lanes would need to be entered as small (0.001 in width) segments in the original template, available for expansion using Template Transition. Template Transition offers one of 3 ways to change template widths and slopes. Another way involves use of Template Point Profile and Template Point Centerline, where a particular template ID can be directed to follow a specific profile and centerline of its own. The third method is template-to-template transitions using Template Series Editor, where distinct templates transition one to another. All three methods require that template IDs "pre-exist" in order to be expanded, or to follow profiles and centerlines, or to transition between template files. So the technique of making very short phantom segments for emerging and disappearing "lanes" or roads with distinct grades is universal. If special slopes are not involved, lanes can expand and contract without creation of phantom segments in the original template. Only clever use of Template Series Editor, where templates with no curbs could "end" and templates with curbs can begin at specified stations, can effectively make "new" features like curbs and medians materialize.





Reviewing the below plan view, when you are given stations and offsets that define a template position like edge-of-pavement (above), you can use Template Transition effectively.



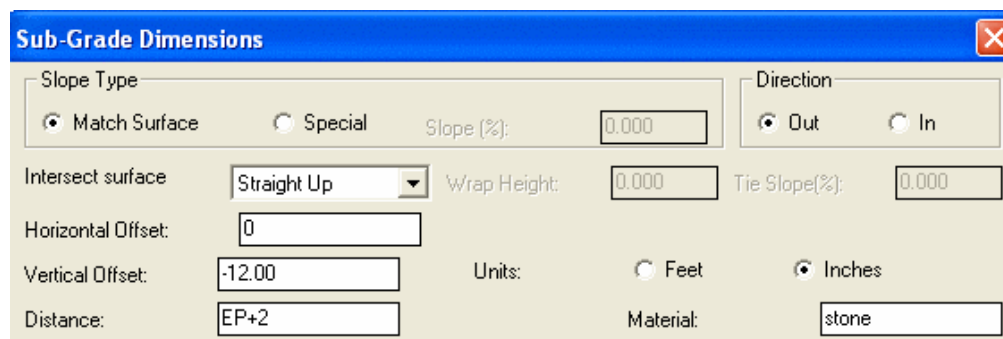
The first Template Transition dialog shows a list of the transitions, covering the above right-lane variable width. To add a transition, click the Add button. This brings up the second Template Transition dialog which shows the transition template for the second segment. The middle sections list the template grades that can be changed. To modify a grade, highlight the grade and click the Edit button. The Report function creates a report of the template transition data.

The Begin Transition Station is where the normal template begins to transition to the modified template. The Begin Full Template Station is where the modified template is used entirely. The End Full Template Station is where the template starts to transition back to normal. The End Transition Station is where the template has returned to normal. This method is designed for elements like passing lanes which expand from normal then contract back to normal. But you can also use this method for roads that start off or end expanded or altered. For example, to start off the road at a 40' edge-of-pavement dimension, it is necessary to transition up from 12.5' (normal dimension). If you need to have 40' at station 0, then enter station -0.01 as the "Begin Transition Station", and enter station 0 as the "Begin Full Template Station". Select the EP grade in the dialog, and change it to 40'. Then click "Link to next transition". The Link to Next Transition option joins the current transition to the next transition without returning to the normal template. This takes you to the second dialog, shown above. You sustain the 40' width from Begin

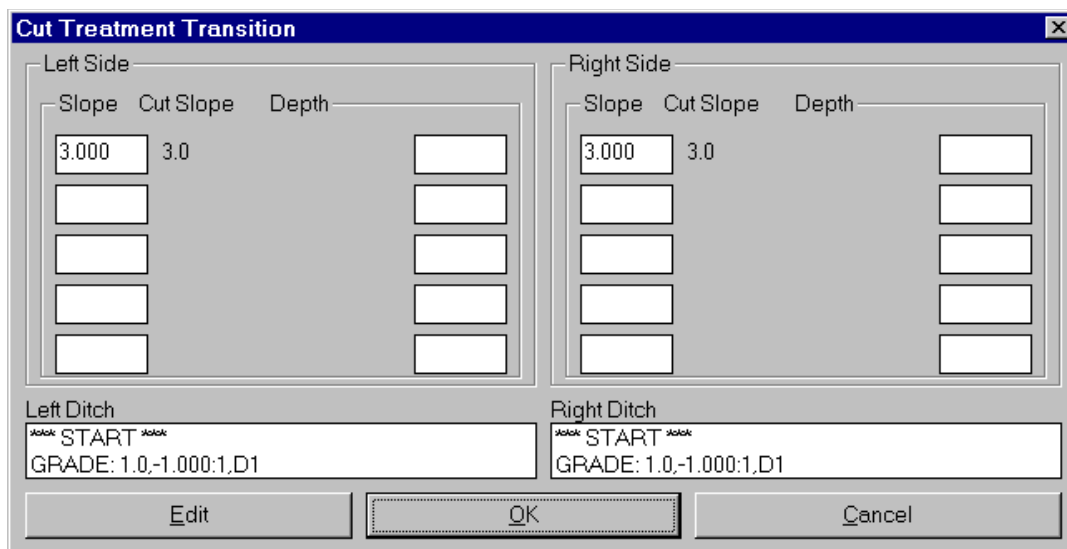
Transition Station 125.29 and transition at station 215.08 to a 24.23' dimension. Then quickly end the transition at station 215.081 for the "End Full Template Station". Finally, transition back to normal 12.5' by entering 335.51 for "End Transition Station".

The Series # setting is for grouping a sequence of transitions separately from other transitions. This Series # allows for independent transitions over the same station range. All transition changes that are part of the same transition should be given the same Series #. For example, when a grade on the right side of the road transitions separately from a grade on the left side of the road, all the transitions for the right side grade could be assigned as Series #1 and all transitions for the left side grade could be assigned as Series #2.

There is another "trick" to using Template Transition with templates that include subgrades. The subgrades will not automatically extend and follow the expanded grade IDs such as EP for "edge-of-pavement", unless the subgrades are defined in terms of the IDs themselves within Design Template. Subgrades that expand "at slope" to intersect a curb, for example, can expand naturally as the curb position moves outward on the right side. But subgrades that go "straight up" at back of curb at offset 14.5' in this example will stay at 14.5', unless defined as shown below by referencing the "EP ID":



Cut and Fill slopes can also be transitioned by picking the Cut and Fill buttons. Ditch and Berm grades can also be modified here.



Transitions can also be applied to the left, right or both sides. This allows you to have separate overlapping transitions for the left and right sides.

Prompts

Template Transition to Edit/Create Choose New to create a transition file or Edit to modify a transition file

Template File to Edit: Specify a transition file

Template Transition dialog

Pulldown Menu Location: Roads

Keyboard Command: tpltrans

Prerequisite: A template .TPT file

Template Grade Table

This command creates a Template Grade Table file (.TGT), which is a lookup file for slopes and distances at stations for grade points within a template. Each side of the template is controlled independently. This user interface provides a simple and easy way to handle complex transitions. A similar result could be produced using a Template Series, or a combination of Template Grade Centerline for horizontal control and Template Grade Profile for vertical control. The advantage of Template Grade Table is that it provides a simpler solution. Besides handling transitions like lane widening, it can also be used to specify superelevation control.

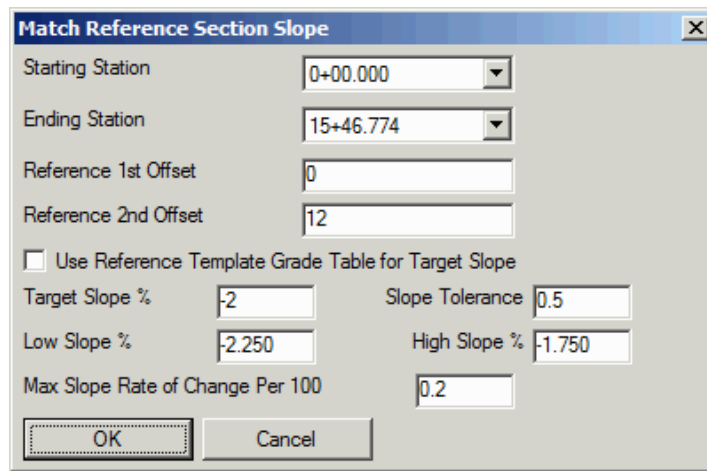
The way that Template Grade Table works is that you select a grade to modify on the left or right side. Then the program has a spreadsheet to define the sequence of transitions for the slope and distance of the selected grade. The grade will use the slope and distance defined in the template file (TPL) until the first station specified. Then the program uses the specified slope and distance going forward. When there is another transition station, the program interpolates the slope and distance between the transitions. For the example dialog below, the grade distance will be 14 at station 50 and will be 16 for all stations after 100. The slope and distance transitions are independent of each other. If one changes and the other doesn't at a station, you only need to enter a value for the one that changes. In the example dialog below, the distance doesn't change after station 100. So only the slope changes are entered after station 100.

A Template Grade Table can be used on a single road with Process Road Design command, or specified for specific roads within a Road Network.

The Template Grade Table is associated with a design template (.TPL) file which is used to set which grades are in the table.

Besides the grades in the template, the table also has the cut and fill slopes. Only the first cut and fill slopes are available. If the template has more complicated cut and fill conditions with multiple cut and fill slopes based on depth, then use the Template Series Files or Template Transition commands to transition these complex cut and fill conditions.

The **Match Slope** function assigns slopes to the grade table using cross slopes from a reference section file. This function can be used to match the template slopes to existing slopes such as for road rehabilitation to match the new road cross slopes to the existing. After selecting the reference section file, there is a dialog to set the range of stations to process and the offsets of the sections to get the cross slope from. The second offset is optional. When only one offset is specified, the program uses the existing slope at the offset. When both offsets are set, the slope is calculated between the two offset points on the existing surface. The Lowest and Highest Slope % settings are optional restrictions on the transition slopes. The Low/High Slope range can also be set by entering the Target Slope and Slope Tolerance. The Use Reference Template Grade Table for Target Slope option is a method to match the slope within the Slope Tolerance to a variable slope. For example, this option applies when matching an existing road that is transitioning through superelevation. When active, this option will make the program prompt for a separate Template Grade Table to use for the slope reference. The Max Slope Rate of Change Per 100 is an optional restriction on how quickly the slopes can change between stations. If you don't want to use a restriction, you can leave the field blank or set a high value.



The **Report** function shows all of the slope and distance changes for all of the template grades.

The **Import** function reads in transition data to the currently highlighted grade in the list. The data can come from either a text file, polyline, superelevation graph or superelevation file. For the text file, the format should have station, slope% and distance separated by a delimiter such as a comma. For the polyline method, the program prompts to select a polyline for the template grade and then prompts for the centerline alignment. There is an option to use another reference alignment for when the template grade is not the first grade from the centerline. Then the grade distances are set using the polyline location. If the reference alignment is selected and is a 3D polyline with elevations, then the import also assigns the grade slopes. For the superelevation graph, the import reads a polyline on a superelevation diagram grid to set the transition slopes. For superelevation file method, the import reads the transition slopes from a .sup file.

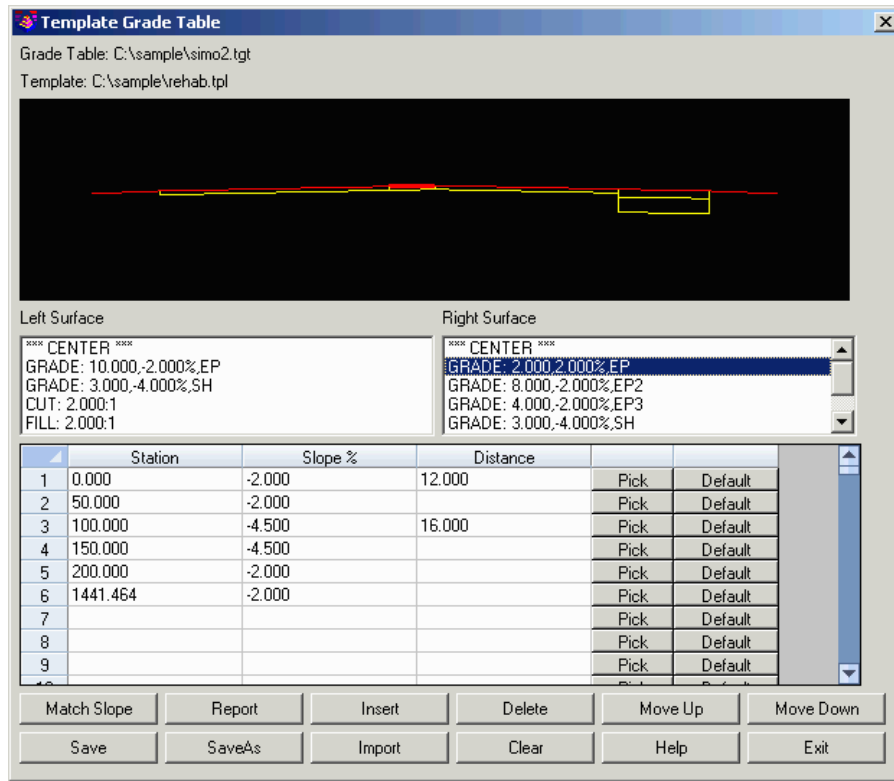
The **Insert**, **Delete**, **Move Up**, **Move Down** and **Clear** functions work on the data in the spreadsheet. The **Pick** button in the spreadsheet lets you screen pick a point to set the station for that row. The **Default** button sets the slope and distance to the values for the grade as defined in the template file (TPL).

Prompts

Template Grade Table to Edit/Create Choose New to create a new Template Grade Table, or Edit to modify an existing one.

Template File to Process: When creating a new Template Grade Table, an existing Template file must be selected to be used with it. When editing an existing .TGT, the Template previously associated with it will be automatically loaded with it.

Template Grade Table dialog



Pulldown Menu Location: Roads

Keyboard Command: tpltbl

Prerequisite: A template .TPL file

Super Elevation Editor

This command is an editor for super elevation stationing. The super elevation data is stored in new or existing super elevation (.SUP) files. When creating a new super elevation file, there is an option to read a centerline file and build the super elevation stationing based on the curves and spirals in the centerline using AASHTO-based stationing or optionally, the Virginia DOT method. The AASHTO calculations are based on the equations in chapter 3 of the 2004 Green Book titled Geometric Design of Highway and Streets. The length of the transition from normal crown to superelevation will be automatically computed by the program using either method based on the design speed and other settings. You can control the amount of this transition that occurs in the tangent leading up to the curve or in the curve itself by either a fixed distance or percentage of the transition. Use the **Transition Part in Tangent By Distance** for the distance method. Otherwise the percentage method is used. The **Include Runout** option sets the transition distance from the transition beginning station to the full super including the crown runoff. Otherwise the transition distance is from the super run-in station (flat outside lane) to the full super. The **Use Transition Curves** option enables fields for the transition curves at each super elevation grade break. For example, if a normal grade is -2% and it starts changing at station 1+00 to reach 4% as station 2+00, then you could have a transition at 1+00 to go from the constant -2% to the rate of change of 6% over 100'. This transition curve will show up in the Draw Superelevation Diagram similar to a vertical curve in Draw Profile.

Superelevation dialog box settings:

- Use Transition Curves:
- Use Reference Centerline:
- Method: AASHTO, Virginia DOT
- Normal Crown Percent Slope: 2.00
- Number of Lanes: 2.0
- Lane Width: 12.0
- Design Speed (mph): 30.0
- Transition Part in Tangent by Distance: Transition Distance in Tangent: 0.0
- Transition Part in Tangent(%): 50.0 Curve(%): 50.0 Include Runout
- Speed Table: AASHTO 4% MAX MSE
-

The main superelevation dialog displays a list of each super elevation transition. These entries should be sequentially entered from lowest to highest stations. To edit the super elevation stationing, highlight the entry line and click **Edit**. The **Add** button creates a new entry below the current highlighted row or at the top of the list if no row is highlighted. The **Delete** button removes the highlighted row from the list. The Report button creates a text report of the super elevation data. The **Calculate** button uses the current Speed Table and Design Speed in the dialog to set the transition stations. The **Save** button saves the super elevation file. To exit the program without saving, click the **Cancel** button.

The **Import** button brings in super elevation data from a delimited text file. Each row of the text file needs to have all the data for one super transitions including the beginning and ending stations. The Import function brings up another dialog for setting the data types for each field in the file. Here are the minimum fields for the import file:

Super Slope, Begin Transition, Begin Full Super, End Full Super, End Transition

For example:

5.0,576.610,806.398,882.794,1112.581

Superelevation dialog box showing a table of transition data:

Begin Transition	Begin Full	End Full	End Transition	Reverse
134+56.299	136+09.633	154+93.813	156+47.146	NO
167+99.492	169+52.825	188+49.357	190+02.690	NO
216+25.447	217+58.780	233+39.638	234+72.971	NO
245+22.847	246+39.514	259+56.086	260+72.753	NO
263+95.966	265+12.633	277+88.099	279+04.766	NO
301+96.686	303+13.353	312+33.060	313+49.727	NO
314+69.343	316+22.676	327+78.579	329+31.913	NO
338+36.688	339+53.354	348+56.227	349+72.894	NO
355+39.324	356+55.991	359+33.854	360+50.521	NO
367+72.865	369+26.199	386+84.897	388+38.231	NO

Speed Table: AASHTO 4% MAX MSE Design Speed: 45.0

The super elevation stationing is entered in the Input/Edit Superelevation dialog. The View Table button shows a table of the super elevation slope for the delta angle and radius at different design speeds. The Calc Super button calculates the slope of full super given the design speed. The station entries are defined as follows:

The screenshot shows the 'Input-Edit Superelevation' dialog box with the following fields and values:

- Station to begin Transition (TS OR PC-1/2 TRANS.):*
- Station to begin super run-in (flat outside lane):
- Station for super at Reverse Crown rate in:
- Normal Grade Slope (%): Design Speed:
- Percent Slope of Full Super (e):*
- Station to begin Full Super (SC OR PC+1/2 TRANS.):*
- Station to end Full Super (CS OR PT-1/2 TRANS.):*
- Compound Curve Percent Slope of 2nd Full Super:
- Station to begin 2nd Full Super:
- Station to end 2nd Full Super:
- Station for super at Reverse Crown rate out:
- Reverse Curve
- Station to end super run-off (flat outside lane):
- Station to end Transition (ST or PT+1/2 Trans.):*

Buttons: View Table, Calc Super, OK, Cancel, Calculate Stations.

Station to begin transition: where normal crown rate begins to transition

Station to begin super run-in: where slope becomes flat

Station for super at normal crown rate in: where slope equals negative of normal crown rate

Station to begin full super: where slope reaches full super slope

Station to end full super: where slopes begins to transition from full super back to normal

Station for super at normal crown rate out: where slope equals negative of normal crown

Station to end super runoff: where slope becomes flat

Station to end transition: where slope returns to normal crown rate

Given these various Station settings, an unequal rate of change can occur between any two stations. However, the program can calculate the stations to set an even rate of transition, as long as it knows the max superelevation, the normal crown slope and the station to start transition, start full super, end full super and end transition. The Calculate Stations button therefore calculates the stations for begin run-in, normal crown rate in, normal crown rate out and end super run-out. To calculate these stations the values with an "*" must be entered.

The Compound Curve option allows you to specify a second superelevation slope for a compound curve. In addition to specifying the second slope, the starting and ending stations for this slope must also be entered. The Reverse Curve option is similar to the Compound Curve option. A typical Reverse Curve is shown below in plan view and as it would appear in the summary dialog:



Begin Transition	Begin Full	End Full	End Transition	Reverse
389+00.000	390+50.000	396+12.000	421+49.000	YES
	399+00.000	417+96.671		NO

Speed Table: AASHTO 4% MAX MSE | Design Speed: 55.0

Buttons: Edit, Add, Delete, Report, Import, Save, SaveAs, Calculate, Cancel, Help

Station 399+00 is the "pivot" where superelevation left flattens and turns into superelevation right.

Prompts

New or Existing Super Elevation File dialog

Superelevation File to Process Specify a superelevation file.

Superelevation Editor dialog

Pulldown Menu Location: Roads

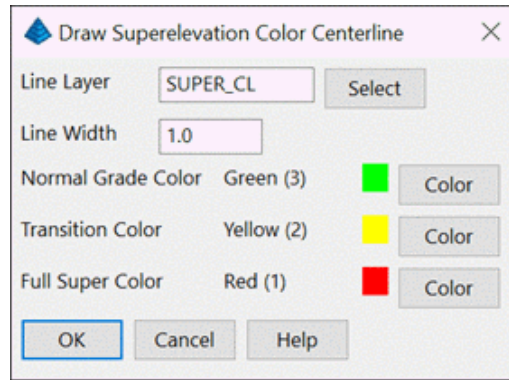
Keyboard Command: super

Prerequisite: None

Draw Super Elevation Color Centerline

This command draws the centerline as a polyline with colors for the super elevation transition zones. The program prompts for a super elevation file (.sup) that defines the transition stations and a centerline file (.cl) that defines the

horizontal alignment. Then there is a dialog to set the colors for the normal grade, transition and full super station ranges. The Line Width sets the width of the polyline.



Pulldown Menu Location: Roads

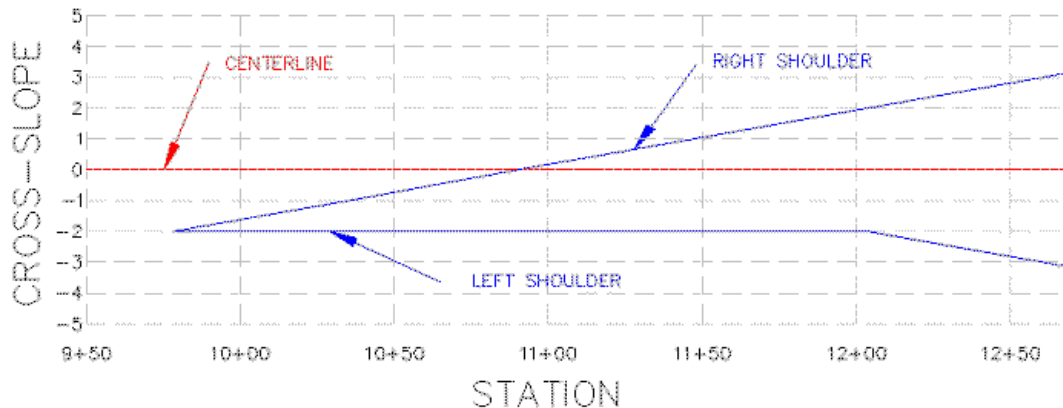
Keyboard Command: superdrawcl

Prerequisite: Super Elevation file (.sup) and Centerline file (.cl)

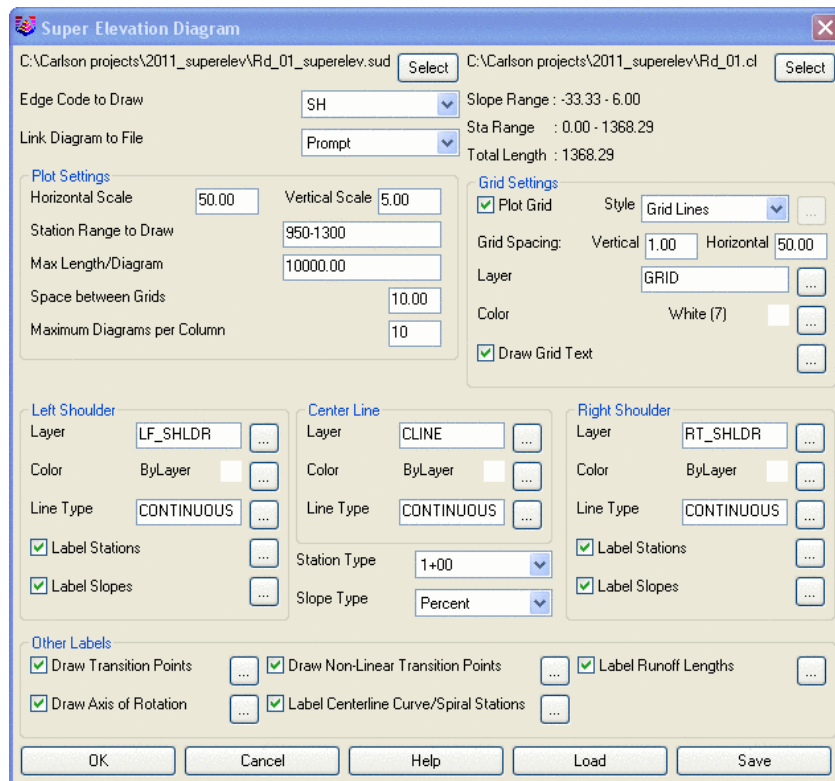
Draw Super Elevation Diagram

This command draws the station and slope diagram for a template grade ID from data in a Template Grade Table (TGT), for the slopes from a super elevation file (SUP), or for Super Elevation Diagram (SUD) file. The Super Elevation Diagram (.SUD) file can be created with the Process Road Design or the Road Network commands.

The Super Elevation Diagram is a graphical representation of the change in cross-slope between the centerline and the left and right shoulders of a roadway. The diagram is typically drawn on a grid, similar to a profile, where the horizontal component represents the stationing of the roadway and the vertical component shows the cross-slope.



Super Elevation Diagram



Draw Super Elevation Diagram Dialog Box

Select: Use this button (at the top-left of the dialog) to browse to and select the Super Elevation Diagram file (.sud) to be drawn.

Edge Code to Draw: Use this dropdown box to select the Template ID to be represented in the Super Elevation Diagram.

Link Diagram to File: This setting has 3 options: *Off*, *Prompt* and *Auto*:

Off: This option will not re-draw the Diagram in the drawing if and when the Super Elevation Diagram (.SUD) file changes.

Prompt: This option will notify the user that the Super Elevation Diagram (.SUD) file has changed and give the option of re-drawing the Diagram in the drawing.

Auto: This option automatically re-draws the Diagram when the Super Elevation Diagram (.SUD) file changes.

Ref CL: Use the **Select** button to associate the Super Elevation Diagram (.SUD) file with a Centerline (.CL) file.

After the (.SUD) file has been selected, various details about the file such as "Slope Range", "Station Range" and "Total Length" are displayed below the **Ref CL:** setting.

Plot Settings

Horizontal Scale: Set as needed.

Vertical Scale: Set as needed.

Station Range to Draw: Enter "ALL" to draw the entire length of roadway or specify a range of stations.

Max Length/Diagram: Specify the maximum length of each diagram. If the "Station Range to Draw" is longer than the "Max Length/Diagram", additional diagrams will be stacked vertically above the first.

Space Between Grids: If the total length of the Diagram requires stacking of multiple Diagrams, this value specifies the distance between Diagram Grids.

Maximum Diagrams per Column: When multiple Diagrams are stacked, this number sets the maximum number of Diagrams per Column. If this number is reached, a second column will be created.

Grid Settings

Plot Grid: Select this option to have centerline and shoulder diagrams drawn on a grid.

Style: Select the Grid Style from several options. *Grid Lines* is the default setting. Other options are:

Ticks Only: This setting draws tick marks for both the station increments along the bottom and the slope increments along the left edge of the diagram.

Ticks and Dots: This setting draws tick marks for both the station increments along the bottom and the slope increments along the left edge of the diagram and draws a series of "dots" in a grid pattern across the rest of the diagram.

Ticks and Checks: This setting draws tick marks for both the station increments along the bottom and the slope increments along the left edge of the diagram and draws a series of "+"-signs in a grid pattern across the rest of the diagram.

Grid Spacing: Set Vertical and Horizontal Grid Spacing as needed.

Layer : Enter the name of the Layer for grid lines, ticks, dots and checks or use the button to select the Layer from a list.

Color: Enter the Color for grid lines, ticks, dots and checks or use the button to select the Color from a list.

Draw Grid Text: Select this option to draw Grid Text for stationing and cross-slopes. Use the button to specify Grid Text settings such as precision, text size scaler, layer, text style, color and vertical/horizontal spacing.

Left Shoulder & Right Shoulder Settings

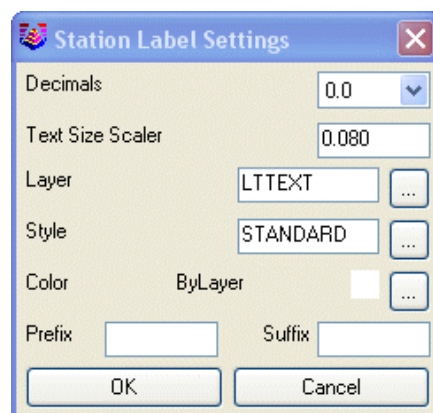
Layer: Enter the name of the Layer for the Left or Right Shoulder or use the button to select the Layer from a list.

Color: Enter the Color for the Left or Right Shoulder or use the button to select the Color from a list.

Linetype: Enter the Linetype for the Left or Right Shoulder or use the button to select the Linetype from a list.

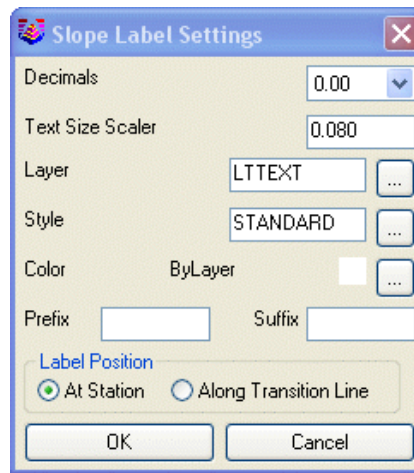
Label Stations: Select this option to label the Station value at each slope change along the Left or Right Shoulder.

Use the button to specify Shoulder Station Label settings such as precision, text size scaler, layer, text style, color and prefix or suffix.



Left & Right Shoulder: Station Label Settings

Label Slopes: Select this option to label the Slopes along the Left or Right Shoulder. Use the button to specify Shoulder Slope Label settings such as precision, text size scaler, layer, text style, color and prefix or suffix. The user also has the option of positioning the Slope label at the station of the Slope change or along the transition line.



Left & Right Shoulder: Slope Label Settings

Center Line Settings

Layer: Enter the name of the Layer for the Centerline or use the [...] button to select the Layer from a list.

Color: Enter the Color for the Centerline or use the [...] button to select the Color from a list.

Linetype: Enter the Linetype for the Centerline or use the [...] button to select the Linetype from a list.

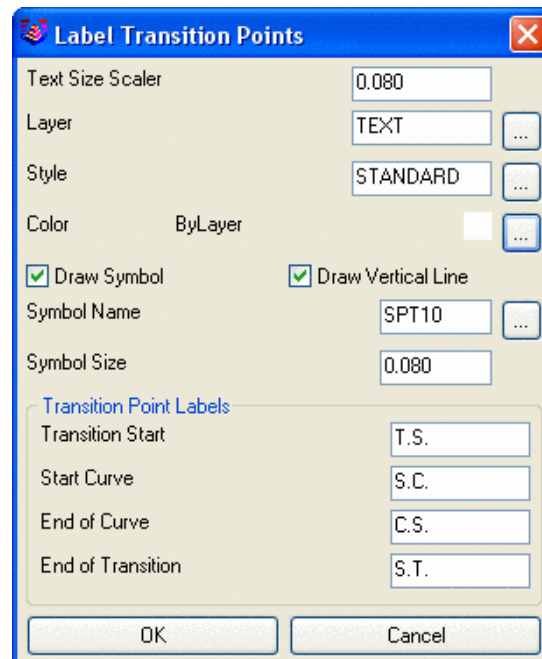
Station and Slope Type Settings

Station Type: Select the desired format for Station. The options are: *Percent* or *Ft/Ft*.


Slope Type: Select the desired format for Slope. The options are: *1+00*, *1+000* or *100..*

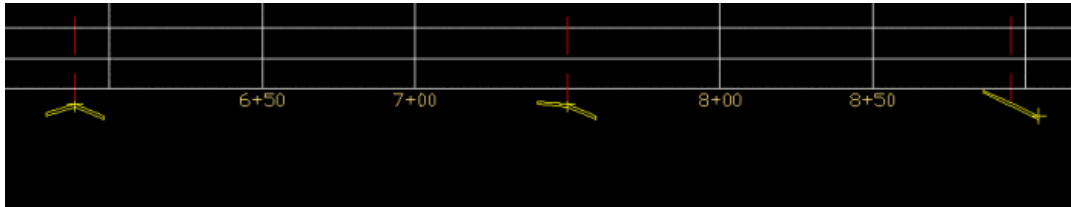
Other Labels

Draw Transition Points: Enable this option and use the [...] button to configure and format the labels for *Transition Start*, *Start Curve*, *End of Curve* and *End of Transition*.

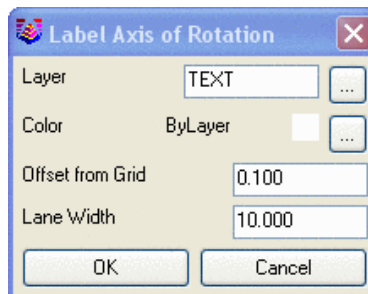


Label Transition Points Dialog Box


Draw Axis of Rotation: Enable this option and use the  button to configure and format the labels for the **Axis of Rotation**. The **Axis of Rotation** is a small icon displayed at the bottom of the grid at critical Super Elevation Points. The icon shows a cross-section view of the pavement slopes and a "+"-sign indicating the point of rotation for the Super Elevation.

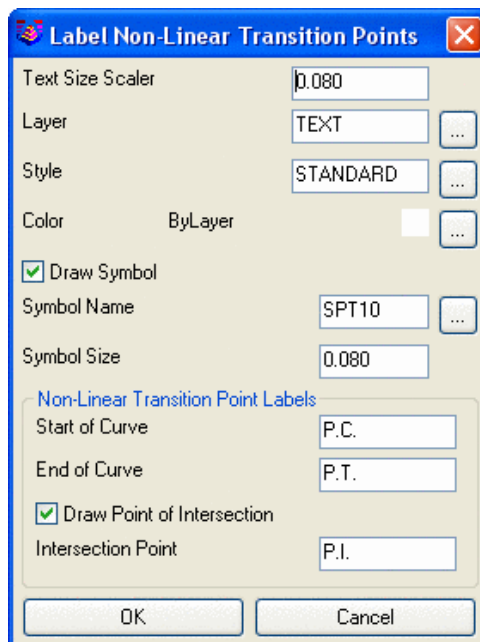


Axis of Rotation




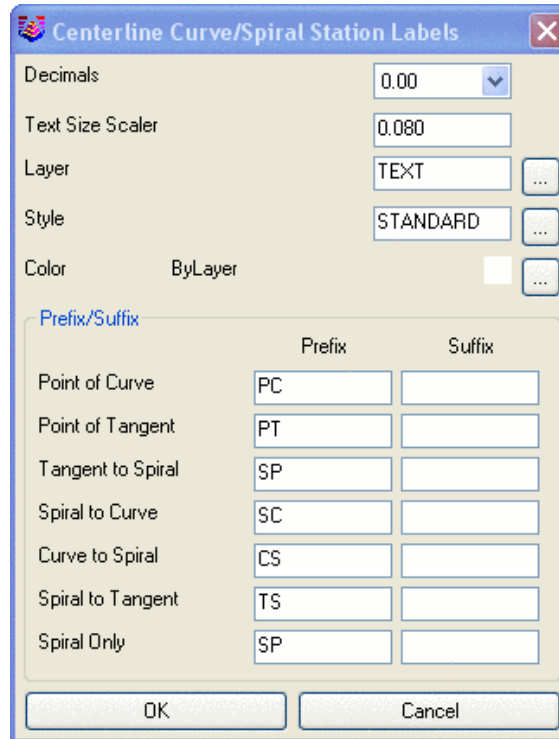
Label Axis of Rotation Dialog Box

Draw Non-Linear Transition Points: Non-linear transition points are drawn when the rate of change of elevation is not constant at the point where the Super Elevation starts or ends. Enable this option and use the  button to configure and format the labels for *Start of Curve*, *End of Curve* and *Point of Intersection*.



Label Non-Linear Transition Points

Label Centerline Curve/Spiral Stations: Enable this option and use the  button to configure and format the labels for *Point of Curve*, *Point of Tangent*, *Tangent to Spiral*, *Spiral to Curve*, *Curve to Spiral*, *Spiral to Tangent* and *Spiral Only*.




The dialog box titled "Centerline Curve/Spiral Station Labels" contains the following settings:

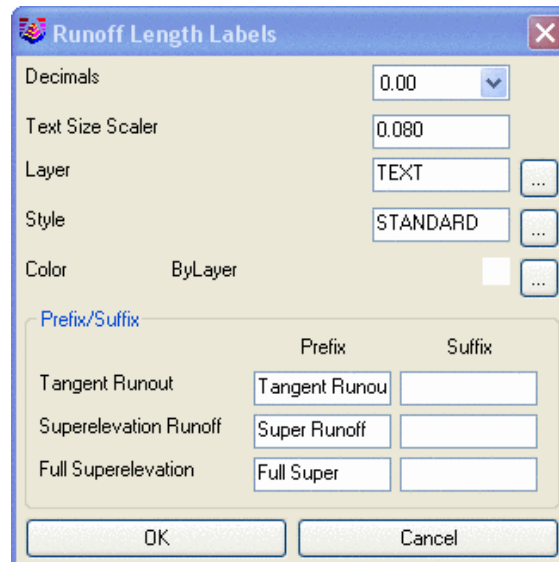
- Decimals: 0.00
- Text Size Scaler: 0.080
- Layer: TEXT
- Style: STANDARD
- Color: ByLayer (unchecked)

	Prefix	Suffix
Point of Curve	PC	
Point of Tangent	PT	
Tangent to Spiral	SP	
Spiral to Curve	SC	
Curve to Spiral	CS	
Spiral to Tangent	TS	
Spiral Only	SP	

Buttons: OK, Cancel

Label Centerline Curve/Spiral Stations

Label Runoff Lengths: Enable this option and use the  button to configure and format the labels for *Tangent Runout*, *Super Elevation Runoff* and *Full Super*. These distances are displayed as linear dimensions above the Diagram Grid.



The dialog box titled "Runoff Length Labels" contains the following settings:

- Decimals: 0.00
- Text Size Scaler: 0.080
- Layer: TEXT
- Style: STANDARD
- Color: ByLayer (unchecked)

	Prefix	Suffix
Tangent Runout	Tangent Runou	
Superelevation Runoff	Super Runoff	
Full Superelevation	Full Super	

Buttons: OK, Cancel

Label Runoff Lengths

Pulldown Menu Location(s): Civil > Roads

Keyboard Command: drawsud

Prerequisite: Super Elevation Diagram file (.sud), Template Grade Table file (.tgt), or Super Elevation file (.sup)

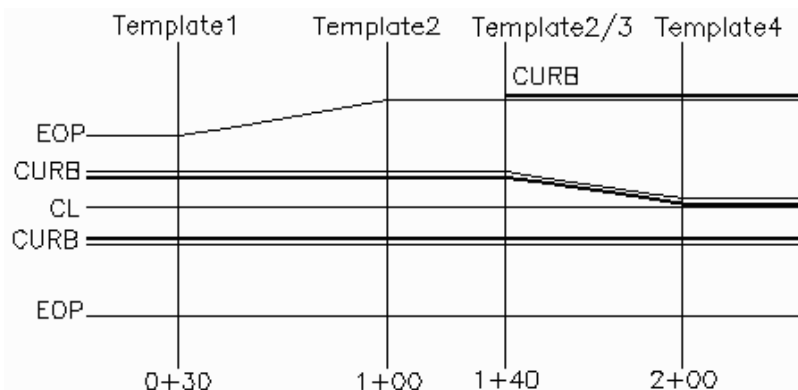
Template Series Editor

Template Series is another method of widening lanes or causing templates to change: direct template-to-template transitioning. Using this command, you specify the station where one template "ends" and the station where another template "begins", and the program auto-transitions between templates.

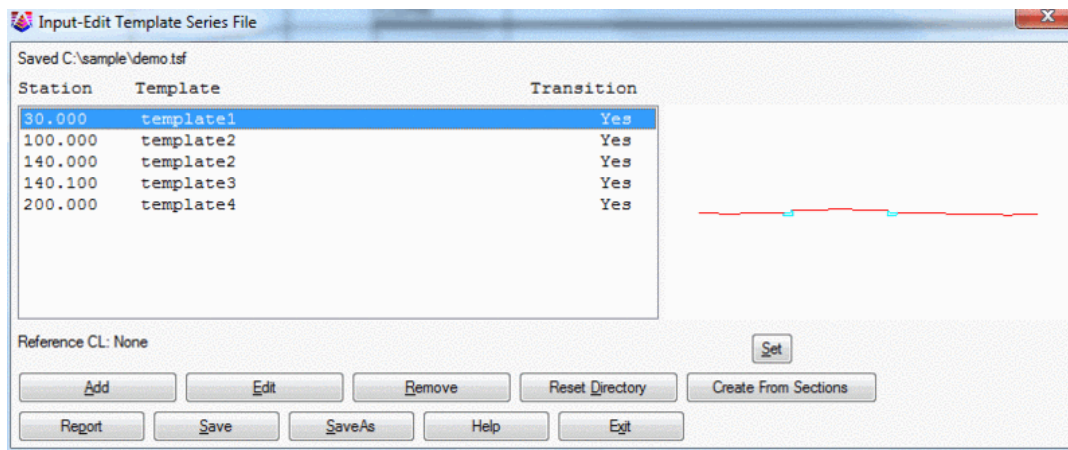
The Template Series is stored in a .TSF file and consists of a sequence of template file names (.TPL) with stationing. The Design Template command is used to create the .TPL files. The Template Series can be used in commands like Process Road Design and Road Network. In these commands, the template selection can be either a regular template (.TPL) or the template series (.TSF).

For the transition to work optimally, the templates should share the same IDs so that the program can connect the template 3D polylines and transition between templates. If the templates are distinct with separate, unrelated IDs, then by ending template1 at station 500 (for example) and starting template2 at station 500.01, a very abrupt transition can be accomplished.

For a design with transitioning templates, the Template Series method is an alternative to the Template Transition method, a third method of Template Grade Table, and to a fourth method of using Template Point Profiles and Template Point Centerlines, where a template ID "follows" a particular centerline and profile. One advantage of the Template Series approach is that it can be used to link different templates together, like non-curb and curb templates, as shown here in plan view:



For the above example, Template 1 applies from station 0+00 to 0+30, then transitions to Template 2 at 1+00 which has a wider EOP distance. This transition occurs between stations 0+30 and 1+00. Then the full Template 2 continues until station 1+40. Then Template 3 starts with a curb replacing a standard EOP/Ditch combination on the left side. So Template 3 would be set to begin at 1+40.1, a short distance past 1+40. This template transitions into Template 4 at station 2+00. Template 4 has a shorter middle grade on the left side. You do not need to enter start and ending templates at station 0+00 or after station 2+00. Therefore, the dialog for this example might look as follows:



Note that you can run Process Road Design to review the design results in plan view, with entry of only the Design Template/Series, the Profile and the Centerline (items 1, 2 and 4 within Process Road Design). You do not need existing cross sections to use Process Road Design. If you process at an interval such as 10 over any desired station range, you can output the Template Polylines and verify the result in plan view. If no sections are found, the program will process from edge of shoulder left to edge of shoulder right, and omit cut and fill slopes. With the correct templates, this would reproduce the plan view shown above.

Template Series Editor is also an effective way to accomplish superelevation, and even simultaneous superelevation and lane widening. Consider the "stages" of pivoting into superelevation of 3%. The first template might be called "Normal Crown" (the lower template). The second template might be called "Reverse Crown" (+2% cross slope). The third template might be called "Full Super" and would be the +3% template. You need the second template because you need to "restrain" the left-hand side of the road from pivoting until the continuous +2% cross slope is reached. If you only used the "Normal Crown" template, say, at station 4+00 and then the "Full Super" template at station 6+00, then at station 5+00, where 1/2 of the transition occurs, the left side cross slope would be -2.5% (transitioning halfway). In reality, the left side should not pivot until station 5+60. If the rate of pivoting is less from normal crown to flat outside lane, and the rate changes after that point, then you would need a fourth template to direct how the road transitions to full superelevation.

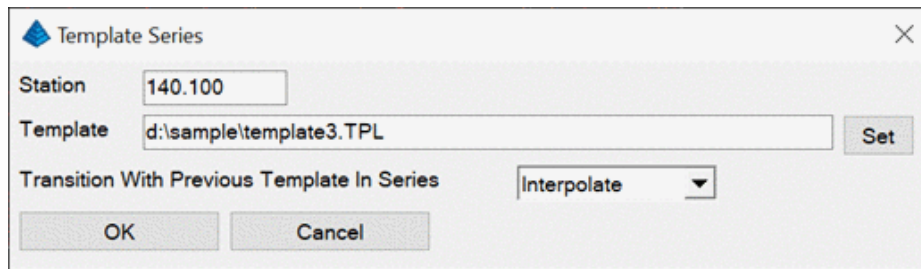
The Reference CL is optional. When it is set, then screen pick is an option for specifying the template transition stations.

The Report function has options for either a summary report of the stations and template, or a detailed report that adds the template dimensions.

The Reset Direction function applies when the folder for the template files (.TPL) has changed and you need to set a new location.

The Create From Sections function reads a section file for a design and creates templates at each change and fills in the template series with these templates. The section file must have descriptions on the section points (ie "EOP").

Here is the dialog for adding and editing templates for the series where you set the template name and station to apply. The Transition With Previous Template In Series as Interpolate will match any common template ID's with the previous template and linear interpolate any changes in distance or slope for the stations between the templates. For the Midway setting, the template dimensions are held unmodified up to the midway station between the templates where the switch occurs. For the None setting, the template switches over at the station in the dialog.



Pulldown Menu Location: Roads

Keyboard Command: tplsseries

Prerequisite: Template Files

Topsoil Removal/Replacement

This command creates a topsoil definition (.TOP) file which defines topsoil removal and replacement zones to be used in the *Process Road Design* command. You can have different topsoil adjustments for different station ranges. These adjustments are applied to the existing ground section in the *Process Road Design* command and will affect the cut and fill volumes. *Process Road Design* will also report the amounts of topsoil removal and replacement.

The command starts by displaying a list of the topsoil stations in the dialog shown below. To add a topsoil adjustment, pick the Add button which brings up a second dialog. You can have different amounts of topsoil removal and replacement for areas in cut and areas in fill. Subsoil is another category of removal that will be combined with any topsoil removal. The Subsoil removal volume is reported separately from topsoil removal by Process Road Design. Subsoil is automatically removed from the site and not used in fill or as a replacement quantity. Therefore, the subsoil element applies only to unsuitable materials that need to be removed. In the example below, we are only removing topsoil in cut (where cutting must take place in any case), and in the cut, we are removing 2' of subsoil which will be hauled off site (since subsoil is not re-used). The removed 0.5' of topsoil in cut will then be replaced in both cut and fill zones of the road within the limits specified by the "Replacement Limit ID". (No topsoil will be replaced on paved surfaces!)

Station	Topsoil Cut	Subsoil Cut	Topsoil Fill	Subsoil Fill	Replace Cut	Replace Fill
0.0-450.0	0.5	2.0	0.0	0.0	0.5	0.5

The Replacement Limit ID is an option to limit the replacement to occur only within the template left offset Limit ID and the right offset Limit ID. If this Limit ID is left blank, then the program will apply the replacement between the left catch point and the right catch point. Topsoil removal is always applied between the catch points. The Limit ID corresponds to a template ID as set in the Design Template routine. Typically, you would use an ID like SH for shoulder and replace topsoil only from the far left and right tie/catch points to the SH or shoulder point. If you use a curb and want to replace topsoil to back of curb, keep in mind that the program takes the basic code "CB" and creates 3 curb points typically, so the back of curb would become CB3 in most L-shaped curbs.

If the Topsoil (".TOP") file is selected within Process Road Design, all quantities of topsoil removal and replacement and subsoil removal are reported, as shown below:

Processing 0+00.00 to 4+42.10

Total Topsoil Removed: 5219.22 C.F., 193.30 C.Y.

Total Subsoil Removed: 20876.89 C.F., 773.22 C.Y.

Total Topsoil Replaced: 5309.57 C.F., 196.65 C.Y.

Hauled-In Topsoil: 90.35 C.F., 3.35 C.Y.

Total Cut : 9106.52 C.F., 337.28 C.Y.

Total Fill: 16402.56 C.F., 607.50 C.Y.

Total SUBGRADE1 - asphalt: 2763.36 C.F., 102.35 C.Y.

Total SUBGRADE2 - stone: 9209.44 C.F., 341.09 C.Y.

Total CURB - concrete: 1078.37 C.F., 39.94 C.Y.

The cut reported in Process Road Design would be the remaining cut after topsoil and subsoil removal, and the fill would be the fill necessary to bring the grade to base of topsoil replacement, on top of which the topsoil is added. The removal of topsoil and subsoil usually creates less cut and more fill, as some of the cut is accomplished by the topsoil/subsoil removal, and in terms of fill, the grade must be brought up to replace the "cavity" created by the topsoil and subsoil removal. Topsoil removal depths and replacement depths can have a dramatic impact on cut and fill quantities, particularly on smaller scale projects like subdivision roads. In this example, every extra 0.1' of topsoil removal produces approximately 100 c.y. of net fill.

Prompts

Topsoil File to Read Specify a topsoil file.

Topsoil dialog Choose your options.

Keyboard Command: topsoil

Prerequisite: None

Assign Template Point Profile

This command assigns profile (.PRO) files to template point ID's like EP (edge of pavement), SH (shoulder) or DL (ditch line), storing this information in a template point profile (.TPP) file which can be used by the *Process Road Design* and *Road Network* commands. The purpose of the profile assignments is to allow separate profiles for template points that are independent of the centerline profile. For example, a ditch grade could have a different profile than the centerline. Multiple template point profiles can be assigned so the amount of control is unlimited. The Template Point Description corresponds to the name set in the *Design Template* command.

If you want the template ID point to follow a special slope or vertical alignment, use Assign Template Point Profile. The combination of using template point centerlines and profiles applied to particular template ID points is a design method sometimes referred to as "strings", where template elements string along special horizontal and vertical

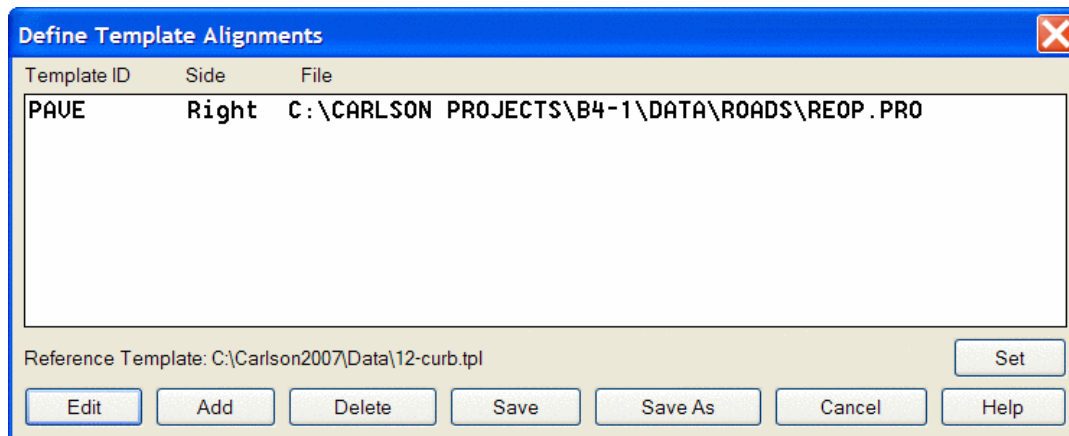
alignments. The rules of the template in terms of distances and slopes to the next point in the template will resume after the template point centerline and profiles are applied.

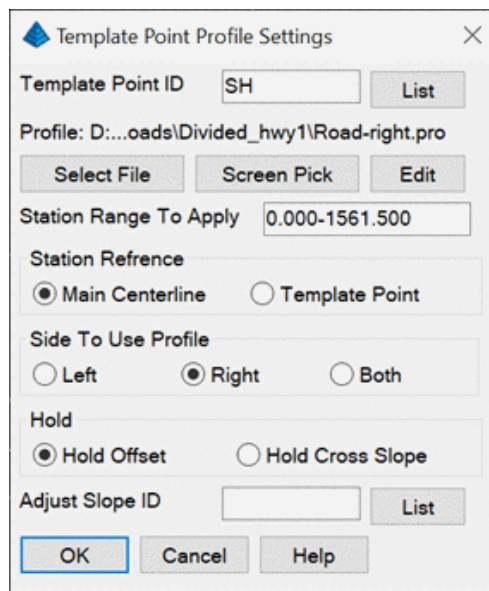
Prompts

First you are prompted to create a new Template Point Profile (.TPP), or edit an existing one.

Next the Define Template Alignments dialog is presented, showing a list of existing Template ID-Profile assignments. To add a new assignment, first pick the Set button to set the Reference Template file (.TPL), then pick the Add button. This brings up the Template Point Profile Settings dialog. First, pick a Template Point Description from the List, which is derived from the components defined in the Template. Next, pick the **Specify File** button, to choose the file (.PRO) to assign to the Template Point ID. Alternatively, instead of picking a profile, you can use the **Screen Pick** button to select a 3D polyline from the drawing which the program will use to generate a profile. Next, enter the Station Range To Apply the assignment. The **Station Reference** sets whether the template profile stationing uses the same stationing as the main CL or uses the template grade stationing. The template grade stationing applies when the template grade is also using a Template Point Centerline so that the template grade has a different length than the main CL. The **Side To Use Profile** specifies if this profile is for the Left, Right, or both sides of the main centerline. The **Hold** setting controls how to apply the profile. Since the template profile can change the relative position of the template ID from the centerline, you have two options for how to fit in the template ID profile: Hold Offset or Hold Slope. Hold Offset will keep the same offset for the template ID and adjust the slope to the template ID. The Hold Cross Slope will keep the same slope to the template ID and adjust the offset to reach the template ID profile elevation. Use Hold Offset when Template Point Profile is used in conjunction with Template Point Centerline, where a single template ID is defined to follow both a special and distinct horizontal alignment (centerline) and vertical alignment (profile). The Adjust Slope ID specifies another grade to apply the change in slope to which can be used for divided highway designs.

Pick OK. Back in the Define Template Alignments dialog, pick Add to add another assignment, Edit to edit an existing assignment, Report to create a report of the template point profile data, Delete to delete a defined assignment, or Save to Exit.





Now Process the road design employing the newly defined Template Point Profile assignment. In the Process Road Design main dialog, pick the Template Point Profile button to select the new file (.TPP). You could also create a new Template Point Profile file directly from this dialog box by picking the Edit button and specifying a new file name.

Pulldown Menu Location: Roads

Keyboard Command: tppset

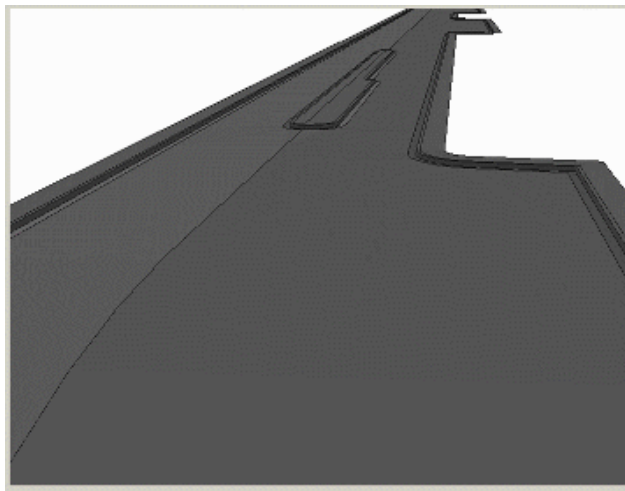
Prerequisite: Profile file (.PRO) or 3D polyline

Assign Template Point Centerline

In roadway design situations involving varying pavement widths, Assign Template Point Centerline is an effective way to control the edge of pavement horizontal alignment as well as any other element in the template. This command assigns centerline (.CL) files to template ID points, independent of the main centerline, thereby controlling the horizontal location. Besides controlling the template grades, you can also control the tie point where the cut/fill slope intersects the existing ground by entering TIE for the Template Point ID.

The assignment of Template ID points to centerline files (.CL) is stored in Template Point Centerline files (.TPC). These files are then used by the *Process Road Design* and *Road Network* commands. The slope to these template points is based on the parameters defined in *Design Template*. Subgrades can be made to follow template IDs if their offset distances are defined not by distance but by reference to the template ID. For example, with a grade ID of EOP and a subgrade that goes one foot past this grade, the subgrade distance can be defined as EP+1 so that the subgrade follows the template point centerline adjustments.

The commands that process Template Point Centerlines such as Road Network will make the template grades follow the template point centerlines including along their curves for the output of the template 3D polylines and road TIN surface. This way Template Point Centerlines can be used to create rounded medians and edge-of-pavements that follow alignments that are independent of the main centerline as shown here.



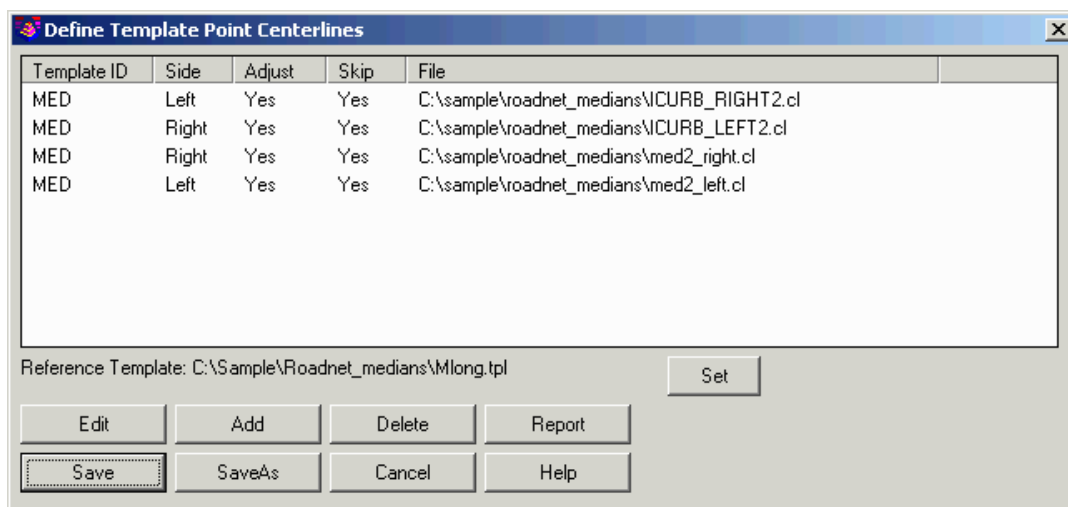
There are a few key settings to a road design with medians as shown here.

- Use Template Point Centerlines to control the alignments of the median and curbs. The Adjust Next Outside Grade Distance option should be ON.
- In the Design Template command, for a template (TPL) with a median crossing the centerline, use the Profile Delta Z set to the height of the median.
- Use the Template Series File command to define the sequence and station ranges of templates (TPL) switching between templates with and without medians. Turn OFF the option to Transition With Previous Template In Series.

Prompts

First you are prompted to create a new Template Point Centerline file (.TPC) or edit an existing one.

Next the Define Template Alignments dialog is presented, showing a list of existing Template ID-Centerline assignments. Use the Add, Edit and Delete buttons to manage the list of template point centerlines. The Report function creates a report of the template point centerlines in the standard report viewer. The Reference Template is optional for setting the road template (TPL) to use for selecting the template point description from a lookup list.

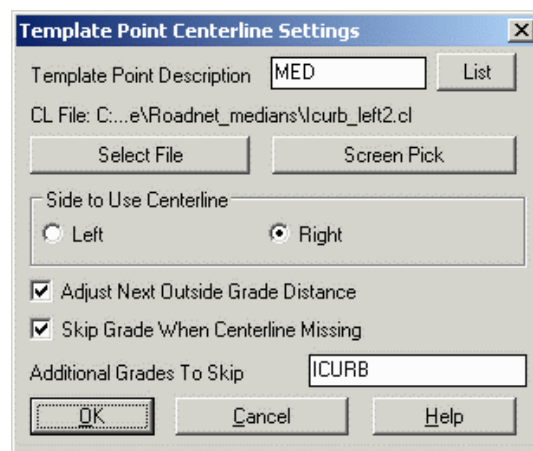


The Add and Edit buttons bring up the Template Point Centerline Settings dialog. First, set the Template Point Description which refers to the Template ID for an element in the Template (TPL) defined by the Design Template command. Next, pick the Specify Centerline File button, to choose the file (.CL) to assign to the Template Point ID. Alternatively, you can use the Screen Pick button to select a polyline from the drawing that the program will use to generate a centerline. Specify if this assignment is for the Left or Right side of the main centerline.

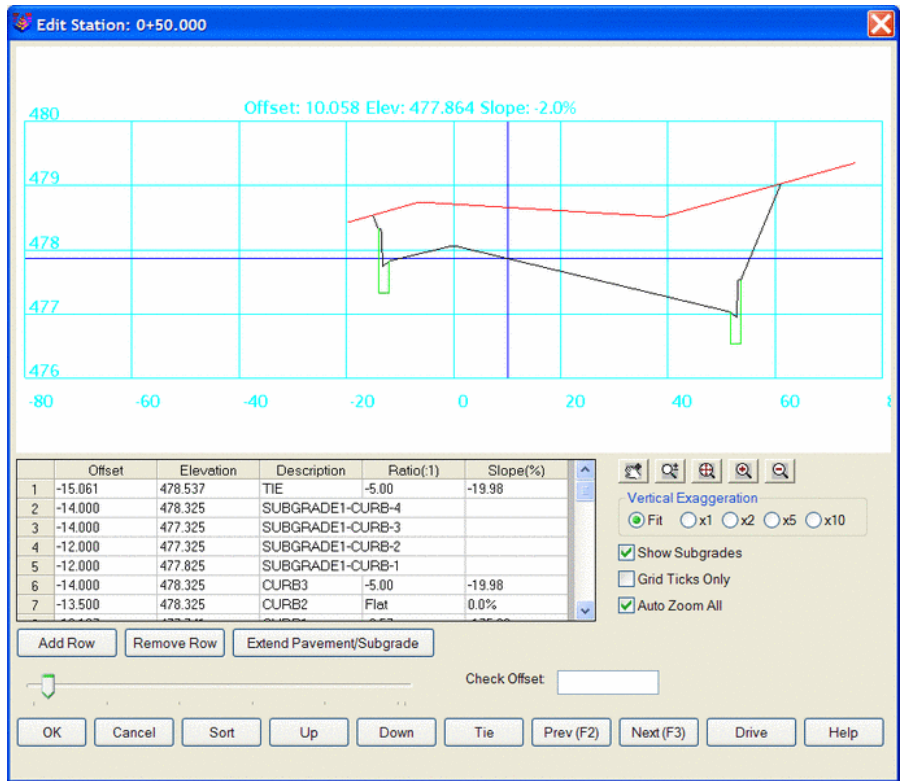
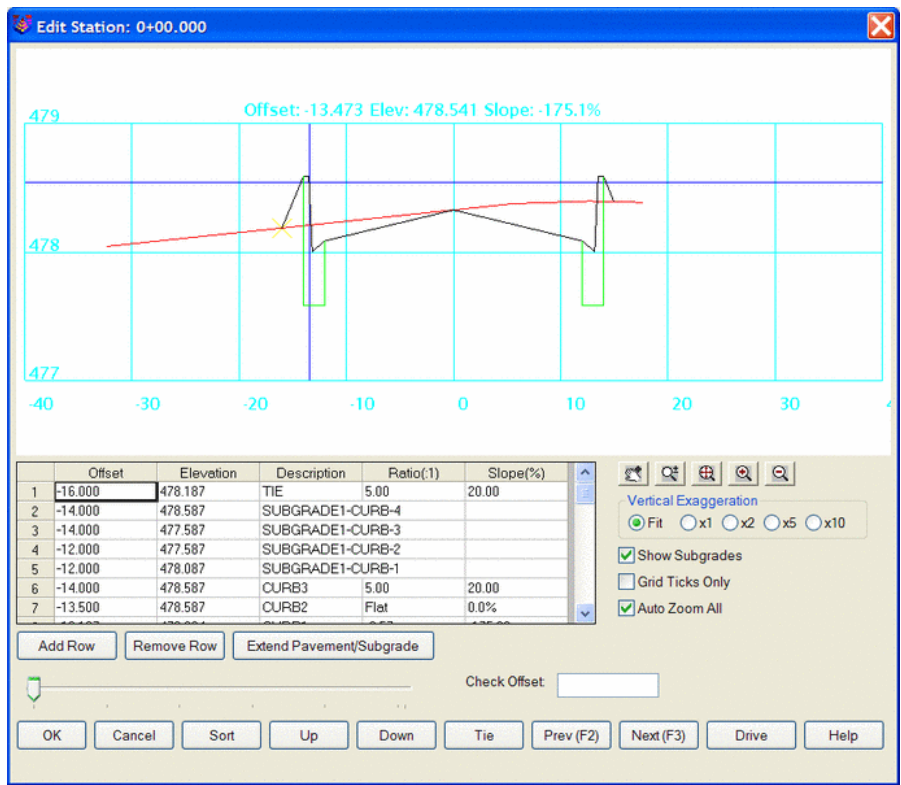
The **Adjust Next Outside Grade Distance** controls how template elements are affected that are outside from the current Template Point Description. This option controls whether to hold the grade distance or grade offset from the main centerline for the outside template elements. When this option is on, the outside grade distances are adjusted by the same amount that the template point centerline adjusts the horizontal alignment. When this option is off, the outside grade distances remain set by their distance defined in the template. For example, consider a template that has an inside median grade (MED) followed by an edge-of-pavement (EP) and the MED has a template point centerline. When this option is on and the template point centerline shifts the MED out 2 feet, then the EP grade is reduced by 2 feet so that the EP offset from the main centerline isn't affected. When this option is off and the MED is shifted out 2 feet, then EP grade distance doesn't change and the EP offset from the main centerline is shifted out 2 feet.

The **Skip Travel Lane for Parallel Grades** applies to a grade for a travel lane that is outside a grade being adjusted by a template point centerline. This option controls whether this outside grade is adjusted to by parallel with the template point centerline or the road main centerline.

The **Skip Grade When Centerline Missing** means that the template will only contain the element at stations that have a template point centerline for it. The Additional Grades To Skip specifies other related template elements to skip in this case. The option applies to template elements like intermittent medians that you only want included in the template for stations where they have a template point centerline defined.



Here are two sections along the roadway, illustrating the varying lane widths on the right side of the main centerline. These sections are created by *Process Road Design* with the TPC set in the input files and these sections are viewed with the *Section File Editor* command.

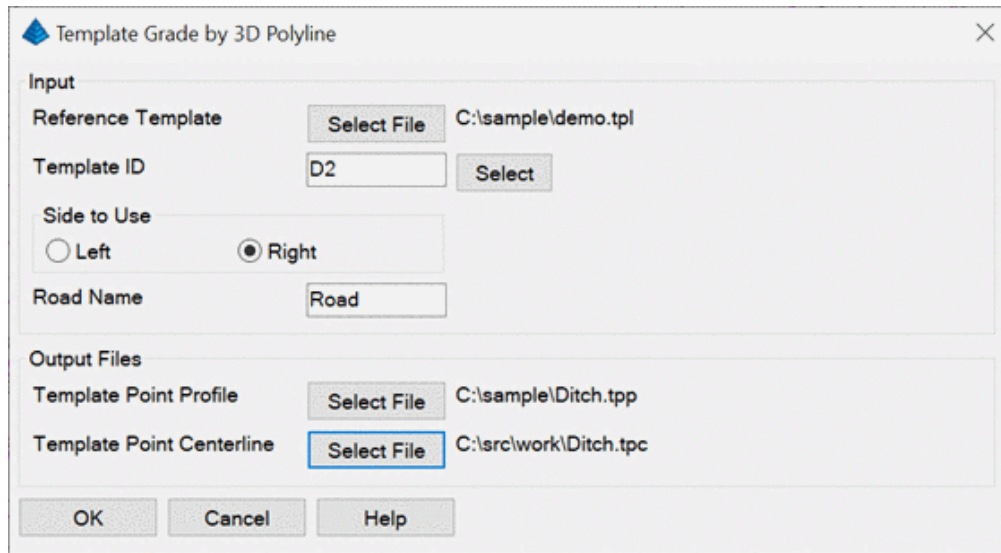


Pulldown Menu Location: Roads
Keyboard Command: tpcset
Prerequisite: Centerline file or polyline

Template Grade By 3D Polyline

This command uses a 3D polyline to define Template Point Profile and Template Point Centerline files which can then be used in Process Road Design and Road Network to have a template grade follow that 3D polyline. For example, the 3D polyline could define the bottom of a ditch and this routine will create the road design files needed to adjust the road template to follow the 3D polyline for the ditch.

After selecting the 3D polyline, there is a dialog with the input and output settings. The Template ID identifies the template element to match to the 3D polyline. The Side To Use tells whether the 3D polyline is for the left or right side of the centerline. The Road Name is used for naming the centerline (CL) and profile (PRO) files which are automatically named and placed in the same folder as the Output Files. The Output Files define the names of the Template Point Profile and Centerline files to create.



Prompts

Select grade 3D polyline: *pick a 3D polyline*
Template Grade by 3D Polyline dialog

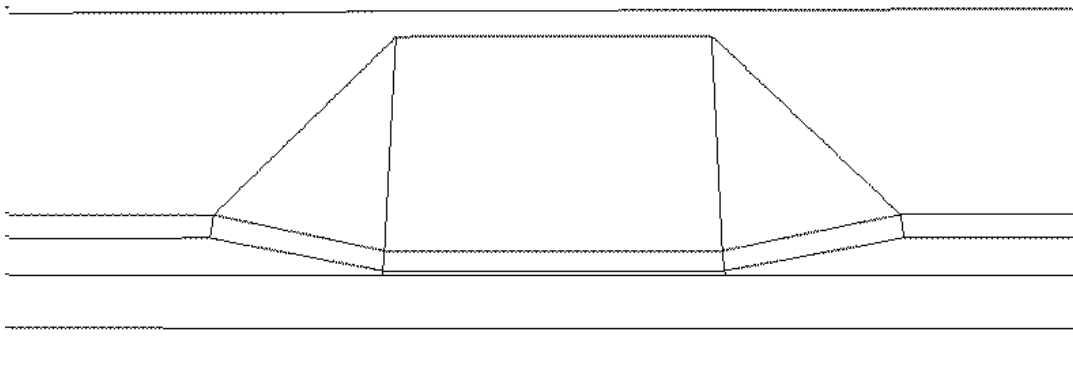
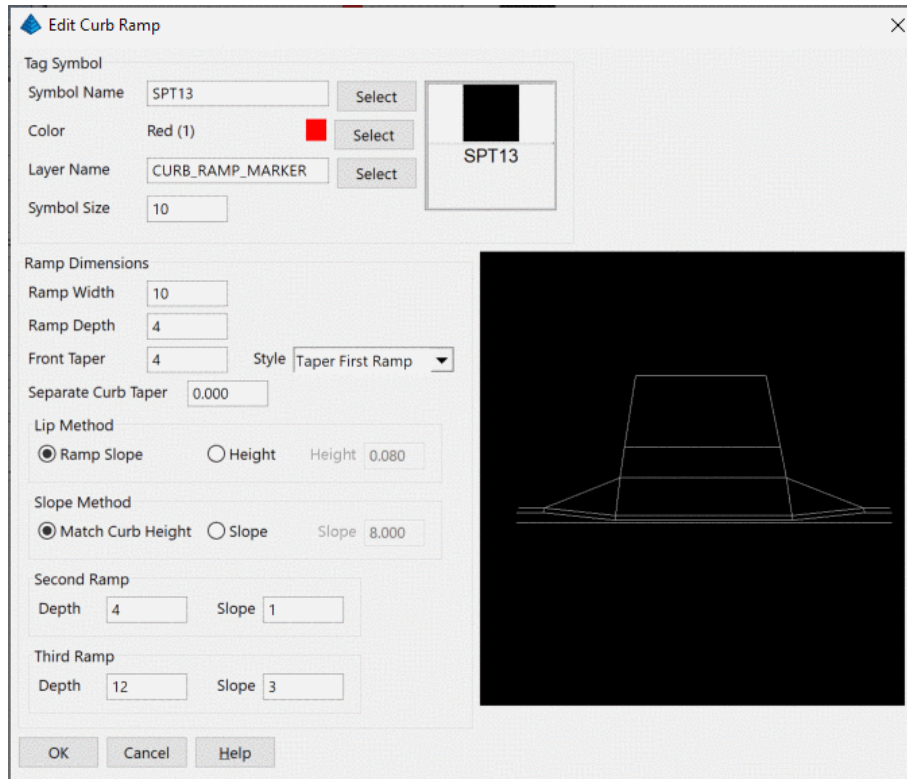
Pulldown Menu Location: Roads
Keyboard Command: prep_tpl_3dp
Prerequisite: 3D polyline

Edit Curb Ramps

This command edits previously drawn symbols for curb ramps.

In the options dialog, select a symbol to use for the curb ramp location markers. The dialog also has the ramp parameters which are described in the Draw Curb Ramp topic in this manual. After the dialog, pick points along the curb polylines for the ramps. The ramp marker should be placed on the curb polyline at the center of the ramp.

To process the curb ramps, run Process Road Design or Road Network and these commands will automatically find the symbol markers and create the curb ramps.



Prompts

Edit Curb Ramps dialog

Select curb ramp: *select existing tagged curb ramp*

Update curb ramp position: **<Exit>** *pick new position or press Enter*

Pulldown Menu Location: Roads

Keyboard Command: curb_ramp_edit

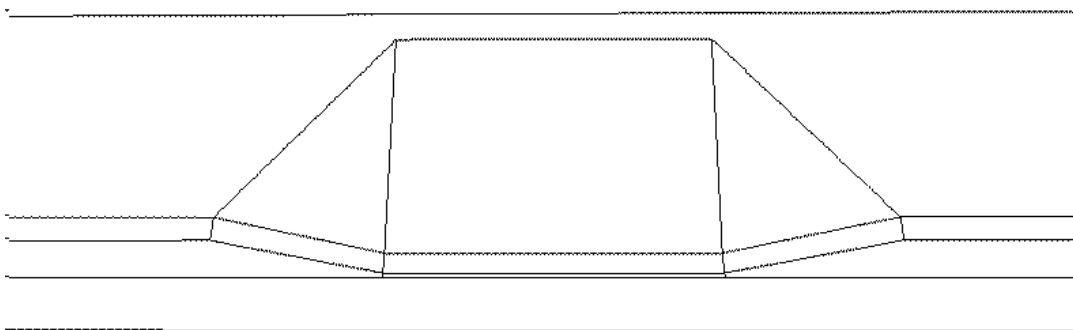
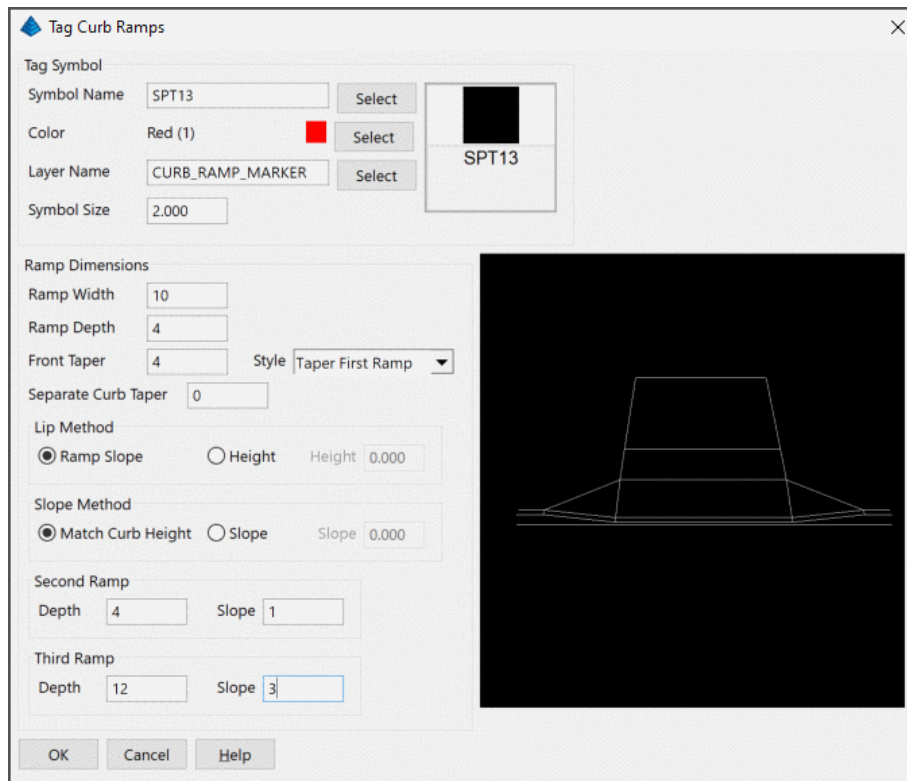
Prerequisite: tagged curb ramp

Tag Crub Ramps

This command draws symbols for where to create curb ramps in Process Road Design and Road Network. The curb ramps are drawn by adjusting the 3D polylines for the top-of-curb and back-of-curb to lower the curb for the ramp. Also, a 3D polyline of the ramp outline is created for each ramp. These curb ramps can be used for sidewalk handicap ramps and for driveways.

In the options dialog, select a symbol to use for the curb ramp location markers. The dialog also has the ramp parameters which are described in the Draw Curb Ramp topic in this manual. After the dialog, pick points along the curb polylines for the ramps. The ramp marker should be placed on the curb polyline at the center of the ramp.

To process the curb ramps, run Process Road Design or Road Network and these commands will automatically find the symbol markers and create the curb ramps.



Prompts

Tag Curb Ramps dialog

Pick curb ramp position: *pick a point along a curb*

Pick curb ramp position (Enter to end): *press Enter*

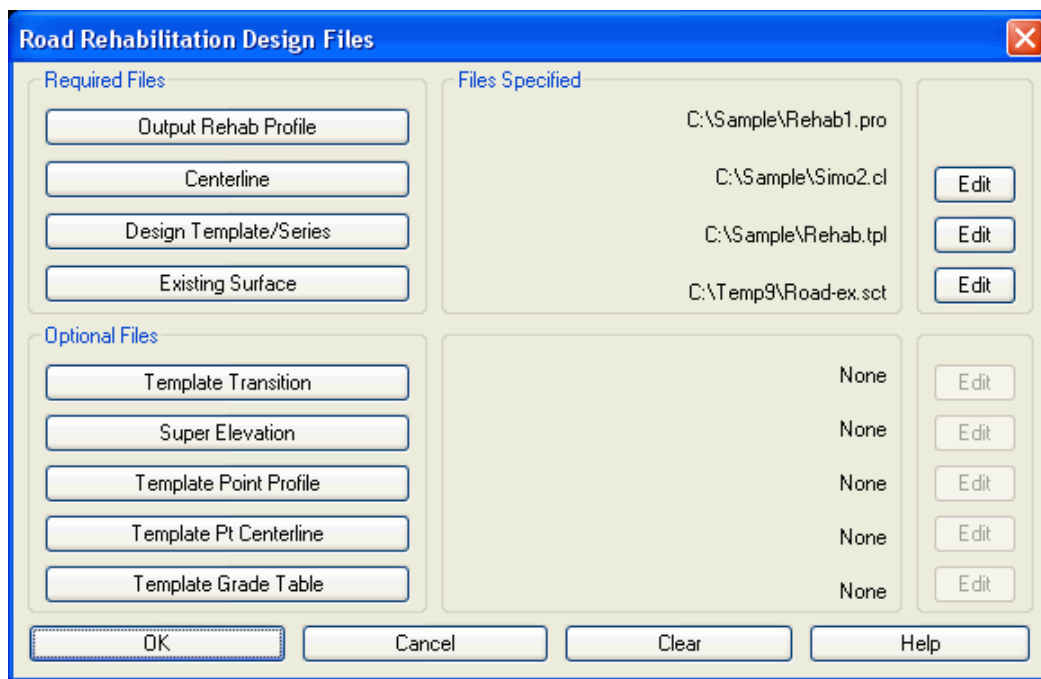
Pulldown Menu Location: Roads

Keyboard Command: tag_curb_ramp

Prerequisite: curb polylines

Road Rehabilitation Profile

This command creates a profile that sets a road design at an elevation to meet the specified overlay thickness along with leveling or milling thickness. The rehabilitation profile created by this command can then be used in Process Road Design to create the rehabilitation design including the rehabilitation surface, sections, quantities and linework.



The first dialog specifies the road design files. All these settings are the same as in Process Road Design except for the Output Rehab Profile. This profile is the output result for this command. The difference with Process Road Design is that the profile is an output instead of an input. Please see the Process Road Design section of the manual for a description of the other input files.

The second dialog has processing options. Again, many of these parameters are the same as Process Road Design. The settings specific to this command are the following:

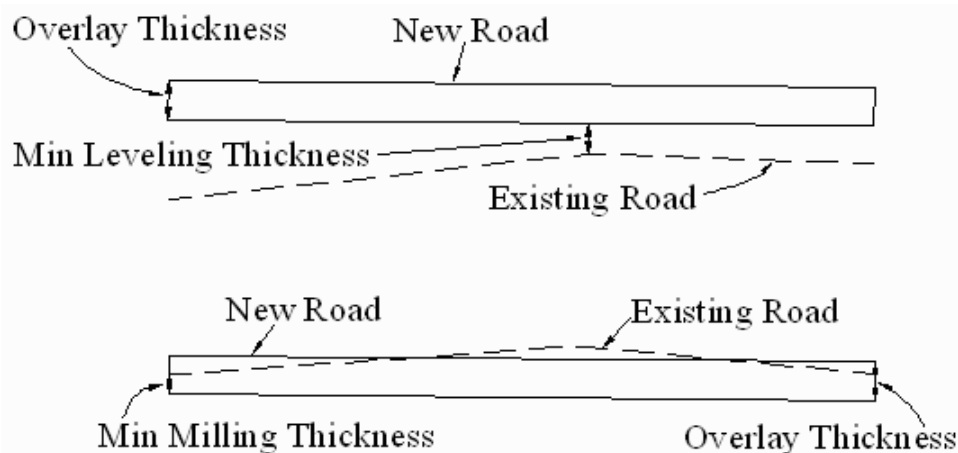
Rehabilitation Method: This chooses between adding a leveling layer to the existing road or stripping the existing road by milling or grinding.

Minimum Leveling Thickness: This is the minimum fill thickness between the existing road and the bottom of the overlay subgrade of the new road.

Minimum Milling Thickness: This is the minimum cut thickness between the existing road and the bottom of the overlay subgrade of the new road.

Overlay Thickness: This is the depth of the overlay subgrade of the new road. This value should match the subgrade thickness defined in the template.

Template IDs to Rehab: These are the grade IDs from the template definition to process for the overlay. The Select button can be used to graphically pick the template IDs. Multiple IDs can be specified by entering the IDs separated by commas. For example, if a road has two lanes with two grades for overlay and the template IDs are LANE1 and EP, then enter "LANE1,EP" in the dialog.



Example 1: Milling

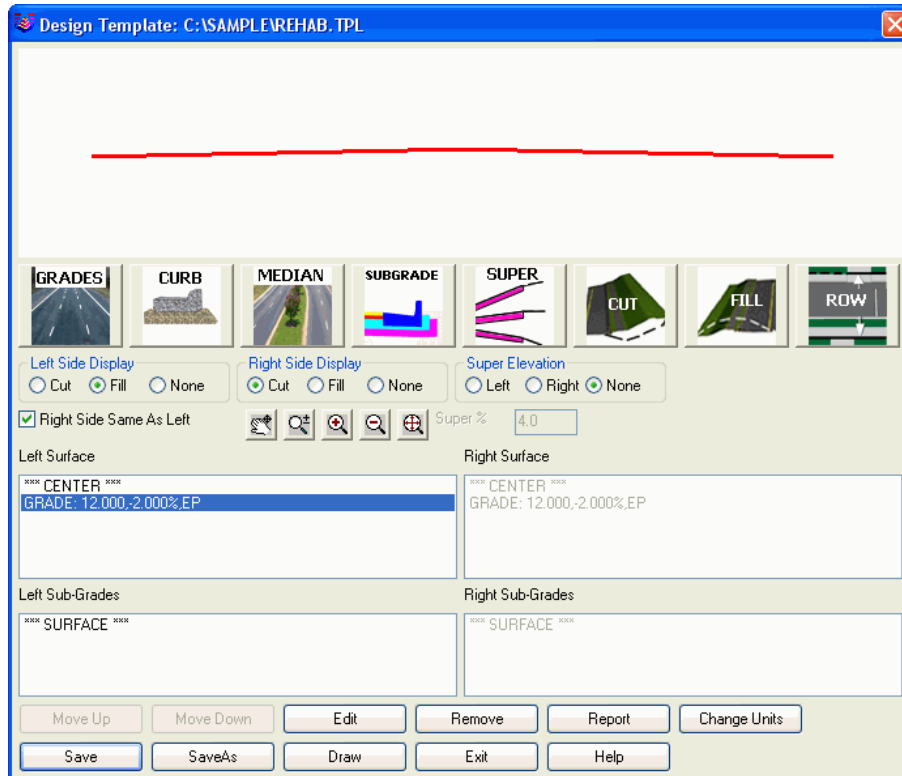
In this case, the existing road will be trimmed by the specified milling thickness.

Step 1: Define Centerline

Use a routine from the Centerline menu to create the .CL file. For example, for entering design plans, use Centerline Editor. For using the geometry of a polyline, use Polyline To Centerline File.

Step 2: Define Template

Run Design Template to create a .TPL file. In this case, the template will be a two lane road with 12' lanes and -2% cross slopes. That's the minimum that needs to be defined for the rehabilitation design. The cut/fill slopes are not required.

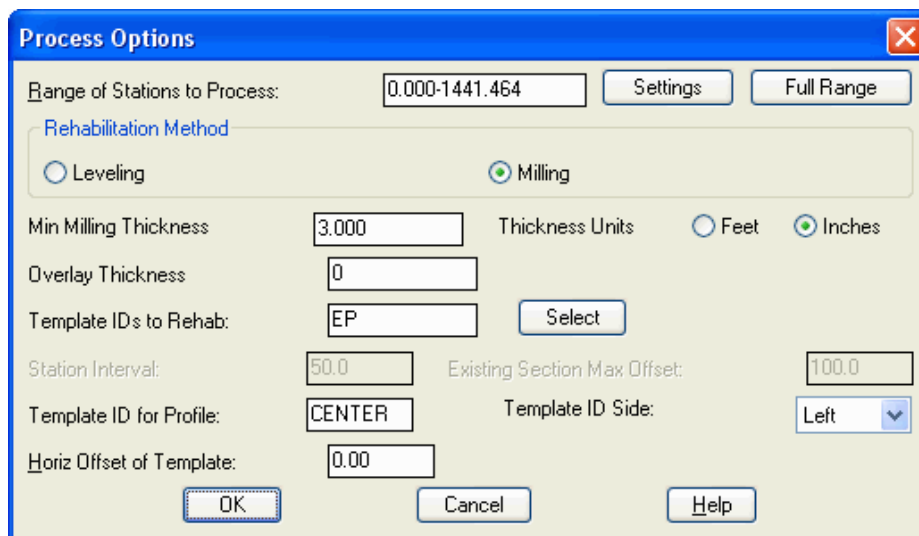


Step 3: Existing Surface

The existing surface can be either a triangulation model or cross sections of the existing road. To create a triangulation surface, you need 3D data for the existing road (points and breaklines) and then run Triangulate & Contour to create a .TIN file. To create cross sections, use the routines in the Sections menu for Section Alignment Editor to set the section intervals and then run one of the Create Section routines to make a .SCT file.

Step 4: Road Rehabilitation Profile

Run this command and specify the 3 files created in steps 1-3. Also set the output profile to create the .PRO file. On the second dialog, choose the Milling method. Set the Milling Thickness to 3 inches. Set the Overlay Thickness to zero since the template doesn't have a subgrade. Set the Template ID to EP to match the grade from the template.



Step 5: Process Road Design

Run this command and specify the 4 files created in steps 1-4. Also set the output Design Section file to create a .SCT file for the rehabilitation design.

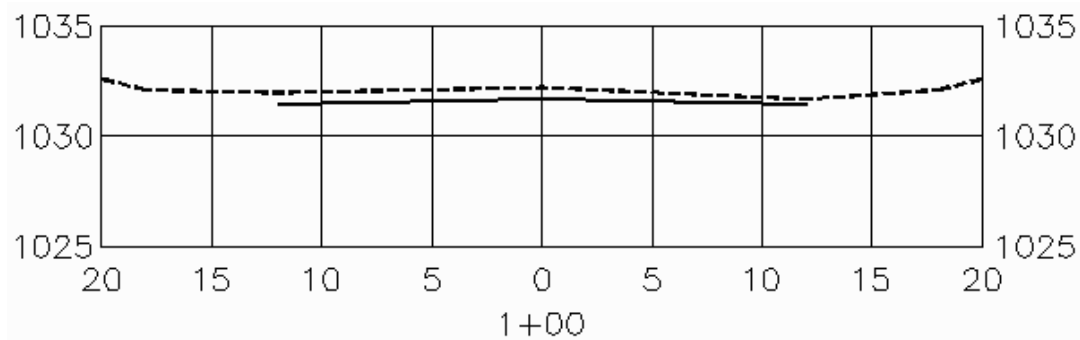
Output:

The report includes the total cut and cuts per station which is the quantity of the milling.

```
Process Road Design
Template File> C:\sample\rehab.tpl
Profile File> C:\sample\rehab1.pro
Existing Surface File> C:\sample\road-ex.sct
Centerline File> C:\sample\simo2.cl
Design Section Output File> C:\sample\rehab1.sct

Processing 0+00.000 to 14+41.464
Total Cut : 9203.469 C.F., 340.869 C.Y.
Total Fill: 0.000 C.F., 0.000 C.Y.
```

Use Section File Editor or Draw Section File to view the design and existing sections.



Example 2: Milling with Overlay

This example adds an overlay thickness to the design from example 1. The steps are the same except for the following:

Repeat Step 2: Design Template

Add a subgrade below the EP grade with a depth of 6 inches.

Sub-Grade Dimensions

Slope Type: Match Surface Special Slope (%): 0.000 Direction: Out In

Intersect surface: Straight Up Wrap Height: 0.000 Tie Slope(%): 0.000

Horizontal Offset: 0.0 Vertical Offset: -6 Units: Feet Inches

Distance: EP Material: Asphalt

Super Elevation Settings

Low Side

Pivot Offset: Max Slope After Pivot (%): Slope Type After Pivot: Normal Special

Standard Slope Percent: 0.000 Minimum Slope Percent: 0.000

High Side

Pivot Offset: Max Slope After Pivot (%): Slope Type After Pivot: Normal Special

Standard Slope Percent: 0.000 Minimum Slope Percent: 0.000

OK Cancel Help

Repeat Step 4: Road Rehabilitation Profile

Use the same settings as example 1 except set the Overlay Thickness to 6 inches.

Repeat Step 5: Process Road Design

Run this command again with the updated Template and Profile.

Output:

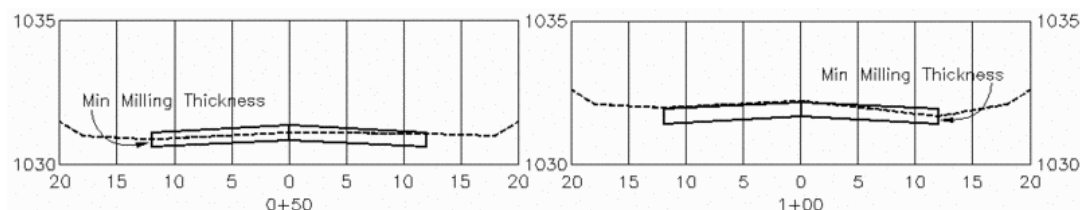
The report includes the updated milling cut quantities along with the overlay subgrade volumes. Since the Milling Thickness stayed at 3 inches, the cut quantities stayed the same as example 1. The Overlay Thickness being thicker at 6 inches leads to more subgrade quantities and raises the new road above the existing in areas.

Process Road Design

```
Template File> C:\sample\rehab.tpl
Profile File> C:\sample\rehab1.pro
Existing Surface File> C:\sample\road-ex.sct
Centerline File> C:\sample\simo2.cl
Design Section Output File> C:\sample\rehab1.sct
```

```
Processing 0+00.000 to 14+41.464
Total Cut : 9203.673 C.F., 340.877 C.Y.
Total Fill: 0.172 C.F., 0.006 C.Y.
```

```
Total Subgrade1 - Asphalt: 17295.031 C.F., 640.557 C.Y., 34592.253 S.F., 3843.584 S.Y.
```

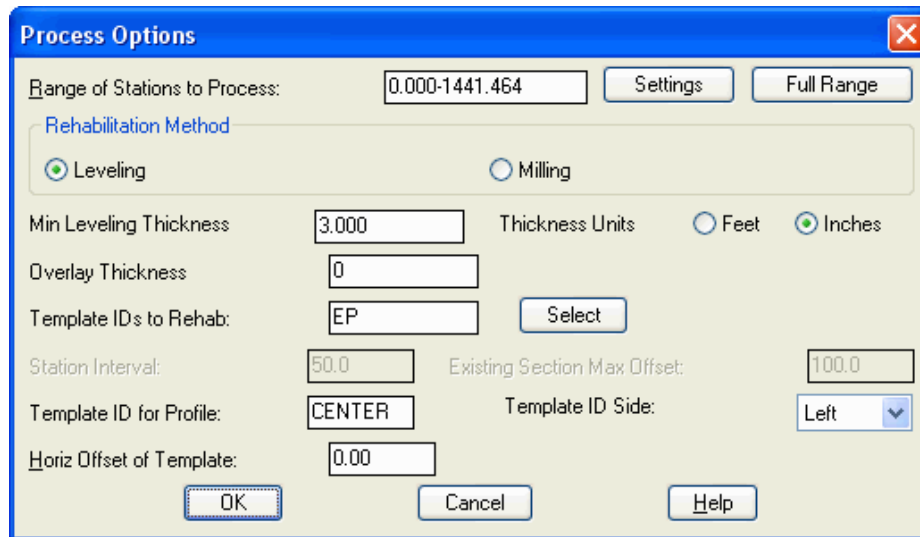


Example 3: Leveling

This example applies leveling to an existing road. The steps are the same as example 1 except for the following:

Repeat Step 4: Road Rehabilitation Profile

On the first dialog, use the same data files as example 1. On the second dialog, choose the Leveling method. Set the Leveling Thickness to 3 inches. Set the Overlay Thickness to zero since the template doesn't have a subgrade. Set the Template ID to EP to match the grade from the template.



Repeat Step 5: Process Road Design

Run this command with the same settings as example 1. The only difference is that the profile is set for leveling instead of milling.

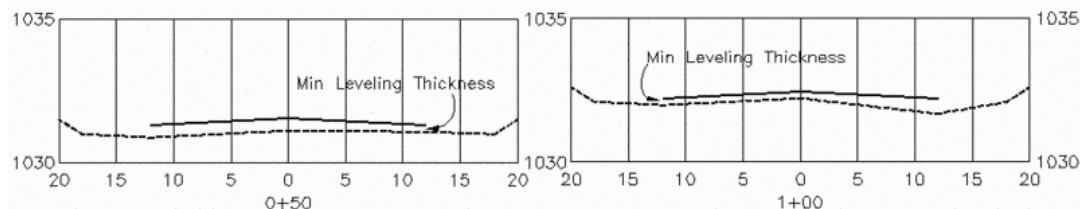
Output:

The report includes the total fill and fills per station which is the quantity of the leveling.

Process Road Design

```
Template File> C:\sample\rehab.tpl
Profile File> C:\sample\rehab1.pro
Existing Surface File> C:\sample\road-ex.sct
Centerline File> C:\sample\simo2.cl
Design Section Output File> C:\sample\rehab1.sct
```

```
Processing 0+00.000 to 14+41.464
Total Cut : 0.000 C.F., 0.000 C.Y.
Total Fill: 8994.509 C.F., 333.130 C.Y.
```



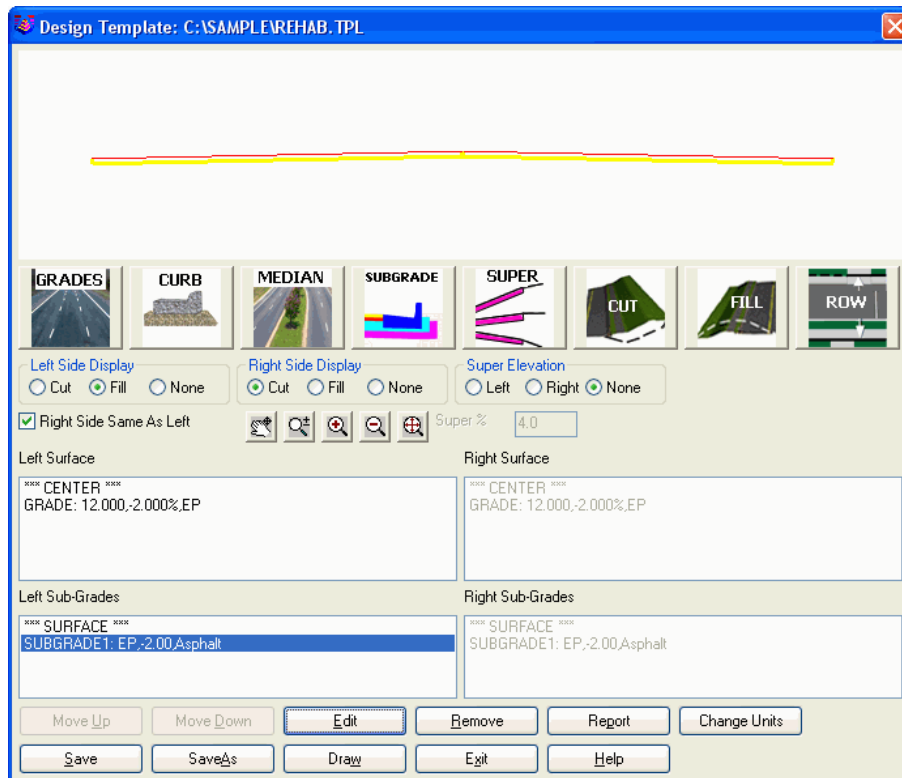
Example 4: Leveling with Overlay

This example adds an overlay thickness with a minimum of 4 inches to the design from example 3. Part of the overlay will be in the subgrade of the template and the rest will be in the Min Leveling Thickness setting. The steps

are the same as example 3 except for the following:

Repeat Step 2: Design Template

Add a subgrade below the EP grade with a depth of 2 inches.



Repeat Step 4: Road Rehabilitation Profile

Use the same settings as example 3 except set the Overlay Thickness to 2 inches and set the Min Leveling Thickness to 2 inches.

Repeat Step 5: Process Road Design

Run this command again with the updated Template and Profile.

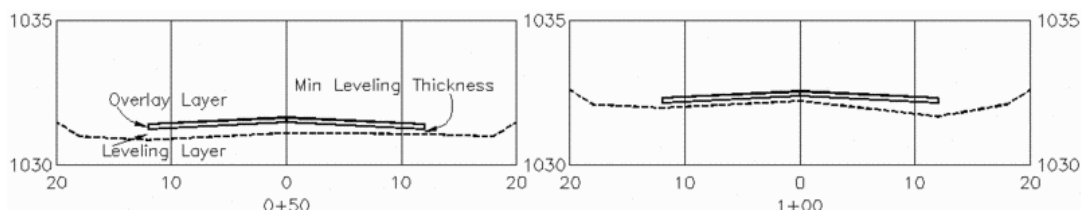
Output:

The report includes the updated leveling fill quantities along with the overlay subgrade volumes.

Process Road Design

```
Template File> C:\sample\rehab.tpl
Profile File> C:\sample\rehab1.pro
Existing Surface File> C:\sample\road-ex.sct
Centerline File> C:\sample\simo2.cl
Design Section Output File> C:\sample\rehab1.sct
```

```
Processing 0+00.000 to 14+41.464
Total Cut : 0.000 C.F., 0.000 C.Y.
Total Fill: 3228.086 C.F., 119.559 C.Y.
Total Subgrade1 - Asphalt: 5765.356 C.F., 213.532 C.Y., 34592.253 S.F., 3843.584 S.Y.
```



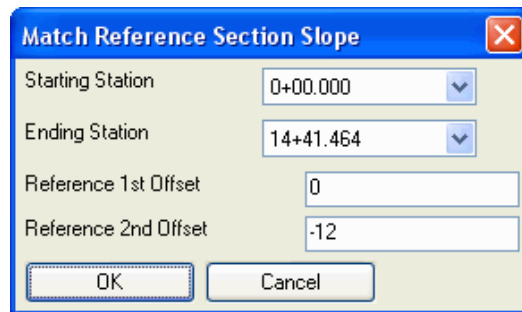
Example 5: Leveling with Overlay and Match Slopes

This example modifies example 4 to have the new road cross slopes match the existing cross slopes instead of being at a fixed design of -2%. The steps are the same as example 4 except for the following:

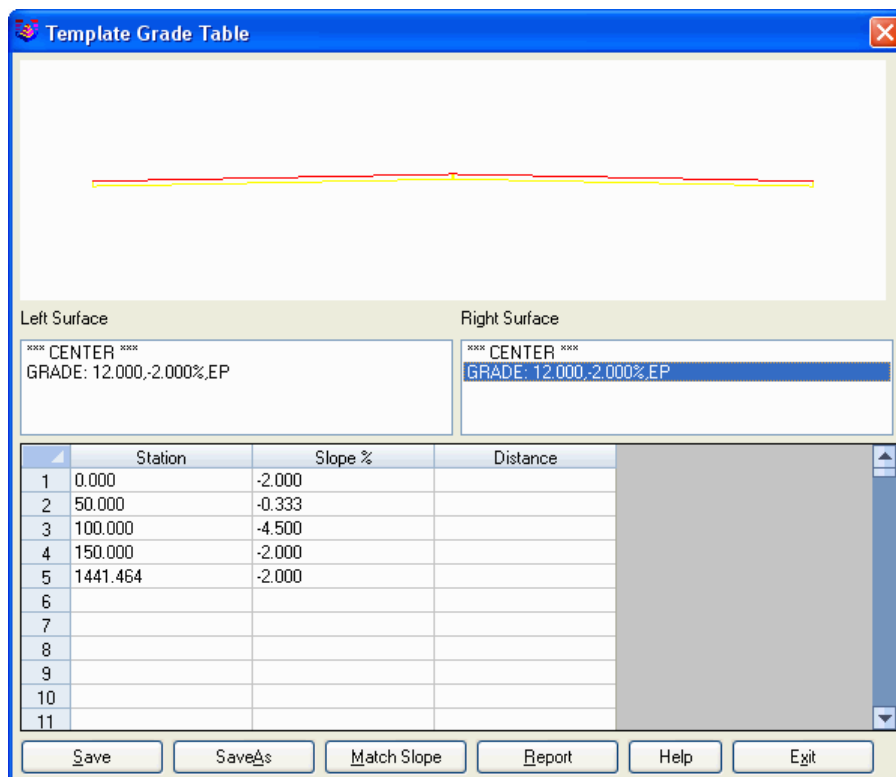
Step 2: Define Template

There are several methods to modify the template for transitions. For this example, the Template Grade Table command is used. This command defines slope and distance transitions for template grades. In this example, we will only use the slope transitions to make the design slopes match the existing road.

Run the Template Grade Table command and create a new .TGT file. Select the rehab template that was used in example 4 as the template to process. In the dialog, highlight the EP grade from the list for the Left Surface. Then pick the Match Slope button. Select the section file for the existing road. Next there is a dialog to set the range of stations to process and the reference offset points which are used to sample the existing surface to get the slope between these offsets. In this example, the full station range is used and the offsets are 0 for the center and -12 for the left EP.



Next, highlight the EP grade from the list for the Right Surface. Then pick Match Slope and select the existing section file. For the offsets, use 0 and 12.



Then back on the main dialog, pick the Save button.

Repeat Step 4: Road Rehabilitation Profile

Use the same settings as example 4 except set the Template Grade Table on the first dialog.

Repeat Step 5: Process Road Design

Run this command again with the updated Template and Profile.

Output:

The report includes the leveling fill quantities along with the overlay subgrade volumes.

Process Road Design

```
Template File> C:\sample\rehab.tpl
Profile File> C:\sample\rehab1.pro
Existing Surface File> C:\sample\road-ex.sct
Centerline File> C:\sample\simo2.cl
Template Grade Table File> C:\sample\simo2.tgt
Design Section Output File> C:\sample\rehab1.sct
```

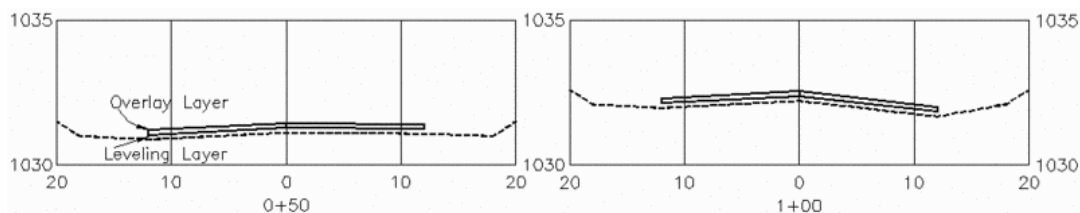
Processing 0+00.000 to 14+41.464

Total Cut : 0.000 C.F., 0.000 C.Y.

Total Fill: 5765.382 C.F., 213.533 C.Y.

Total Left Subgrade1 - Asphalt: 2882.684 C.F., 106.766 C.Y., 17296.127 S.F., 1921.792 S.Y.

Total Right Subgrade1 - Asphalt: 2882.681 C.F., 106.766 C.Y., 17296.127 S.F., 1921.792 S.Y.

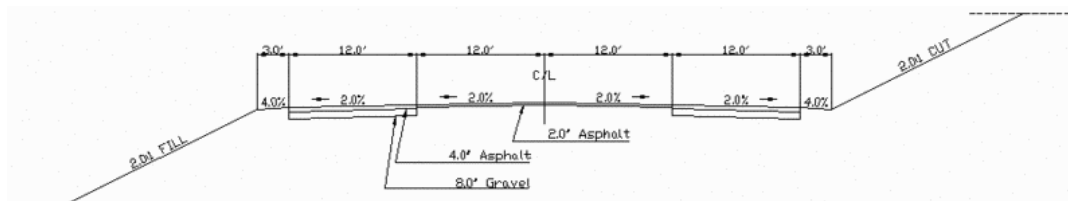


Example 6: Leveling with Overlay and SuperElevation plus Lane Widening

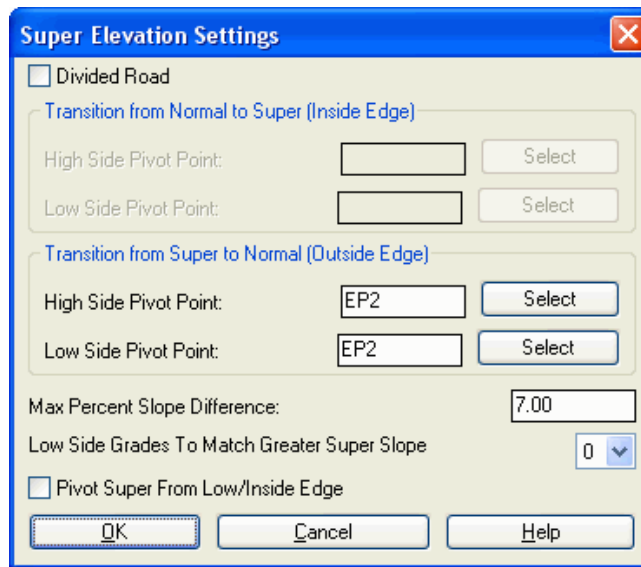
In this variation, the new road has both overlay and new design changes of applying new superelevation and widening a lane. The steps that are different than example 4 are described here.

Step 2: Define Template

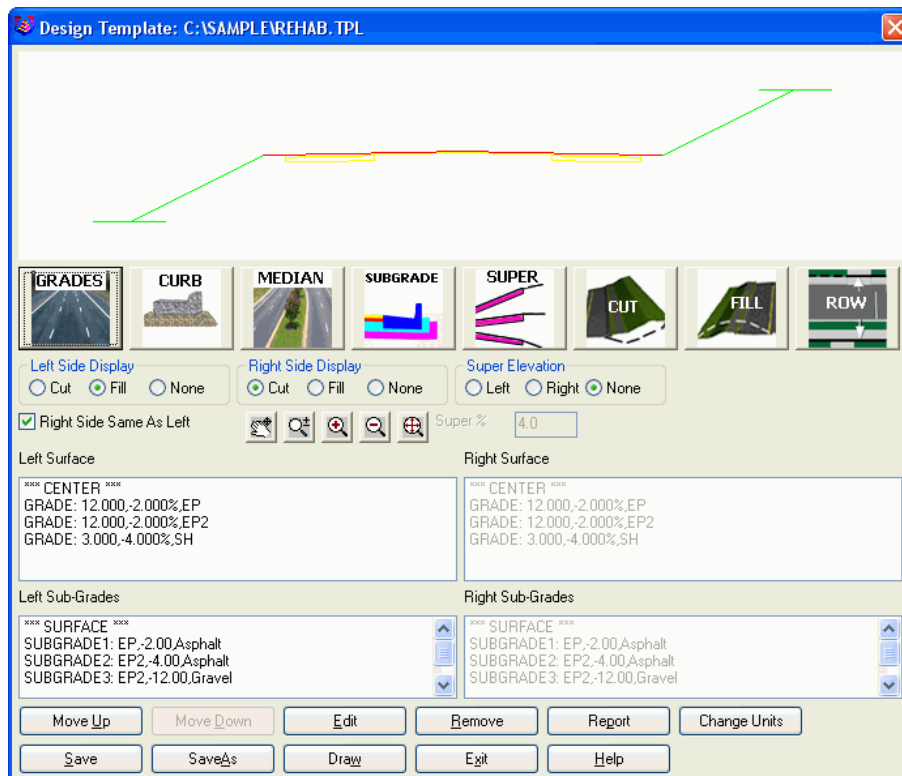
For this example, the template has additional design elements besides the overlay grades. Run the Design Template command to add the new elements.



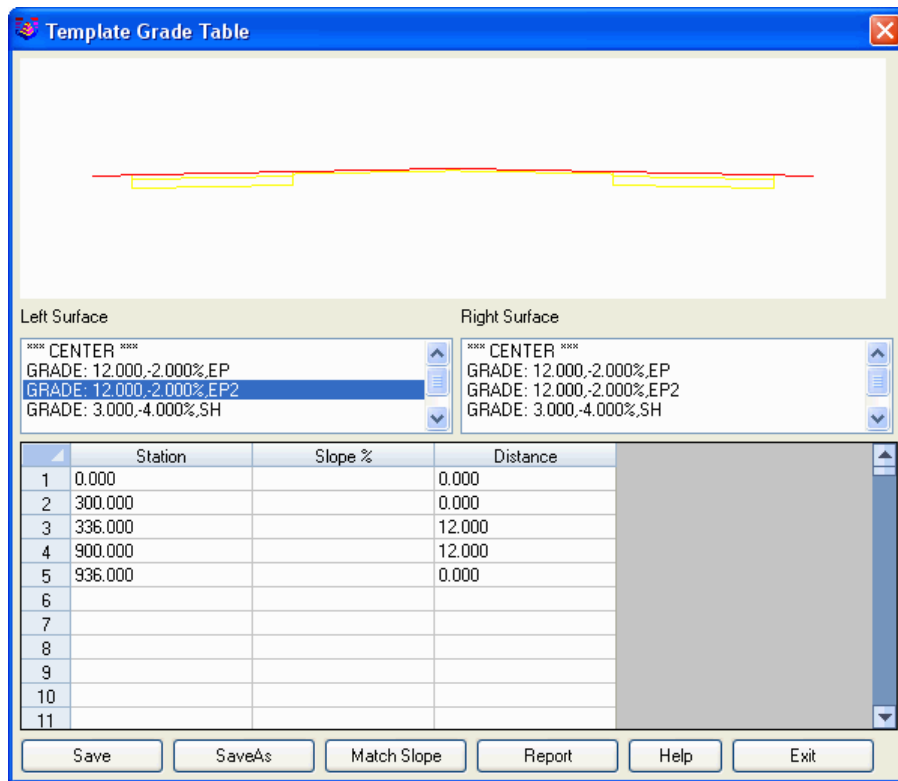
First, pick the Grades button to add a new grade for the new lane using a slope of -2%, distance of 12 and ID of EP2. Then add a new grade for a shoulder with slope of -4%, distance of 3 and ID of SH. Next, subgrades are needed for the new lane since this isn't over the existing road. Add a subgrade of 4 inches of asphalt under the new lane and another subgrade of 8 inches of gravel. Next pick the Cut button and set the cut slope to 2:1 and pick the Fill button and set the fill to 2:1. The cut/fill slopes are needed to tie the new road design elements to the existing surface.



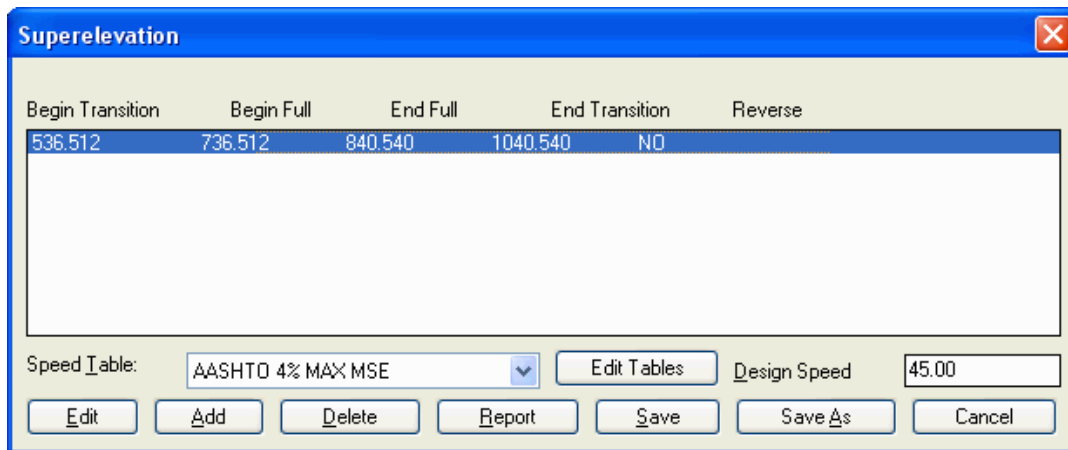
Finally, pick the Super button to set the superelevation transition ID's as EP2 so that the shoulder stays outside the super. Then pick the Save button.



Even though the template has the new lane EP2 defined for both sides, let's actually only apply this new lane for a range of stations on the left side. Run the Template Grade Table command and make a new .TGT file. Select the template file that was just created. Then pick EP2 on the Right Surface list. In the table, fill in the first row with station 0 and a distance of 0. This will eliminate EP2 on the right side. Then pick on EP2 from the Left Surface list. Fill out the table as shown to make the new lane start at station 3+00, reach full size at 3+36, start transitioning back at station 9+00 and return to zero at station 9+36.



Another template transition definition to create is the superelevation. Run the Super Elevation Editor command and create a new .SUP file. Use the option to select a centerline and specify the speed table to have the program set the transition stations. Or use the Add function to manually enter the transition stations and full super slope.



Repeat Step 4: Road Rehabilitation Profile

Use the same settings as example 4 except use the new template (TPL), template grade table (TGT) and superelevation (SUP) created in step 2.

Repeat Step 5: Process Road Design

Run this command again with the new template, profile, template grade table and superelevation.

Output:

The report includes the leveling fill quantities along with the overlay subgrade and the quantities for the new road elements.

```

Process Road Design

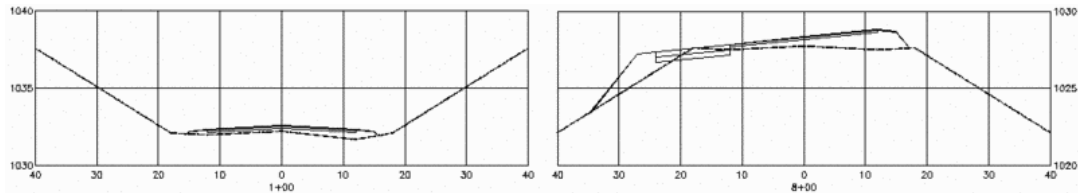
Template File> C:\sample\rehab.tpl
Profile File> C:\sample\rehab1.pro
Existing Surface File> C:\sample\road-ex.sct

```

```
Centerline File> C:\sample\simo2.cl
Template Grade Table File> C:\sample\rehab.tgt
SuperElevation File> C:\sample\rehab.sup
Design Section Output File> C:\sample\rehab1.sct
```

```
Processing 0+00.000 to 14+41.464
Total Cut : 3549.129 C.F., 131.449 C.Y.
Total Fill: 22519.407 C.F., 834.052 C.Y.
Total Left Subgrade1 - Asphalt: 2882.699 C.F., 106.767 C.Y., 17296.127 S.F., 1921.792 S.Y.
Total Left Subgrade2 - Asphalt: 2399.867 C.F., 88.884 C.Y., 7200.000 S.F., 800.000 S.Y.
Total Left Subgrade3 - Gravel: 4799.622 C.F., 177.764 C.Y., 7200.000 S.F., 800.000 S.Y.
Total Right Subgrade1 - Asphalt: 2882.569 C.F., 106.762 C.Y., 17296.127 S.F., 1921.792 S.Y.
```

These sections show the road at a station before the new lane and superelevation and at a station with t



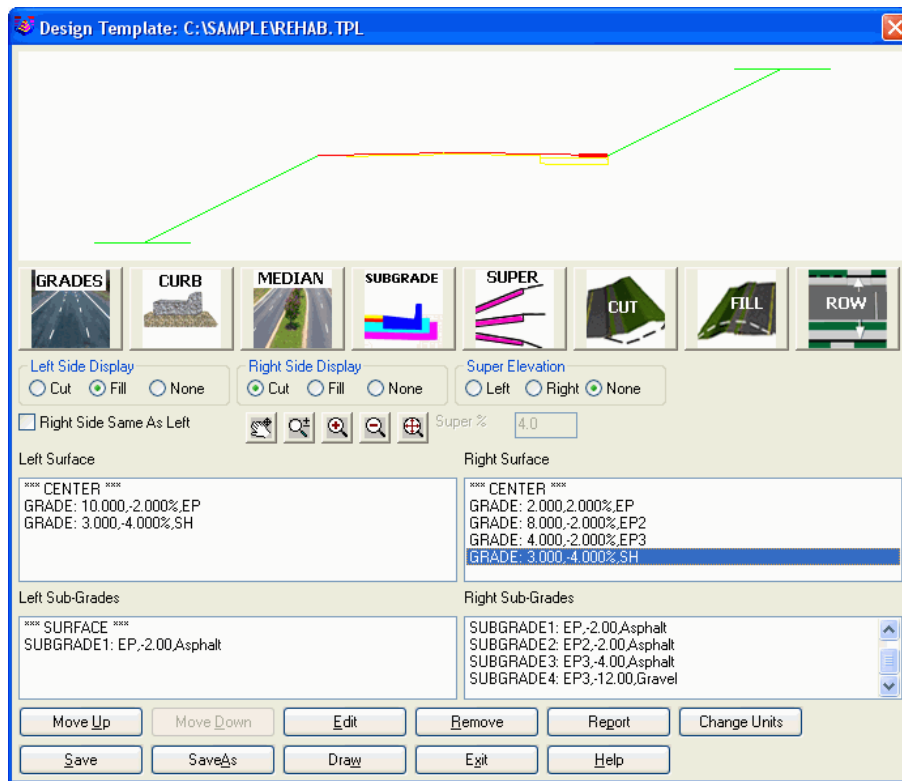
Example 7: Leveling with Overlay with Lane Widening at New Crown position

In this variation, the existing road has two 10' lanes that are being expanded to 12' lanes. The left EP is staying fixed and the extra 4' is added to the right side. So the crown is shifting 2' to the right. The steps are the same as example 4 except for the changes to the template definition.

Step 2: Define Template

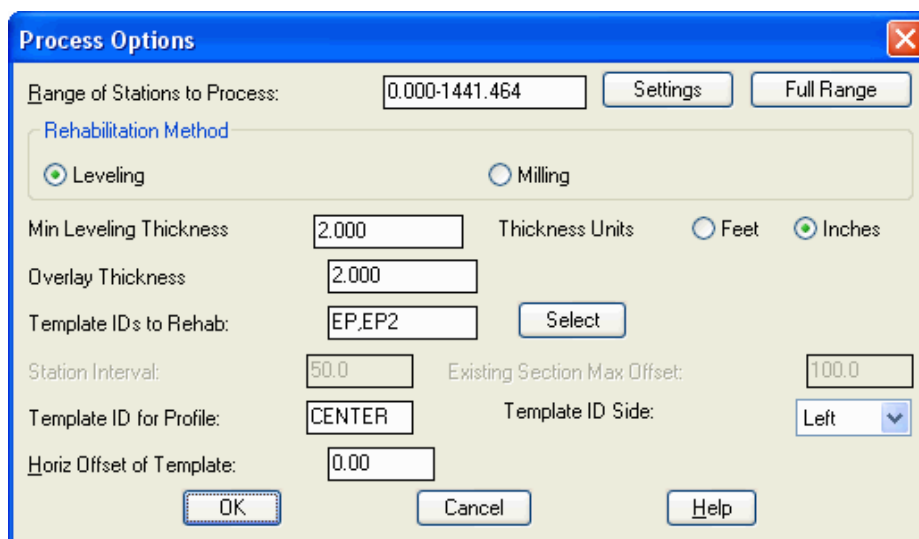
Use the Define Template command to make the template overlay and widening grades. For this example, there is one 10' grade on the left side for the overlay plus a 3' shoulder. On the right side, there are four grades. The first right side grade is 2' for the portion of the overlay that is shifting the crown over. The second grade is 8' for the remainder of the right side overlay. The third grade is 4' for the widening. The fourth grade is a 3' shoulder. The template has subgrades of 2" for the overlay grades and has two subgrades of 4" of asphalt and 8" of gravel for the widening grade. Since this template is asymmetrical, uncheck the toggle for Right Side Same As Left.

Next pick the Cut button and set the cut slope to 2:1 and pick the Fill button and set the fill to 2:1. The cut/fill slopes are needed to tie the new road design elements to the existing surface.



Repeat Step 4: Road Rehabilitation Profile

For the input files, use the template created in step 2, the centerline and existing surface. For the process options, choose the Leveling method and enter 2 inches for the Leveling and Overlay thicknesses. For the Template IDs, specify both EP and EP2 since both of these are overlay grades.



Repeat Step 5: Process Road Design

Run this command with the new template and profile.

Output:

The report includes the leveling fill quantities along with the overlay subgrade and the quantities for the new road elements.

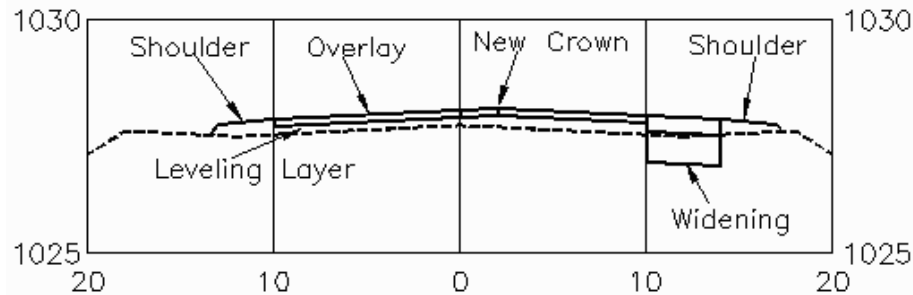
Process Road Design

Template File> C:\sample\rehab.tpl

Profile File> C:\sample\rehab1.pro

```
Existing Surface File> C:\sample\road-ex.sct
Centerline File> C:\sample\simo2.cl
Design Section Output File> C:\sample\rehab1.sct
```

```
Processing 0+00.000 to 14+41.464
Total Cut : 3462.106 C.F., 128.226 C.Y.
Total Fill: 8523.772 C.F., 315.695 C.Y.
Total Left Subgrade1 - Asphalt: 2402.152 C.F., 88.969 C.Y., 14413.199 S.F., 1601.467 S.Y.
Total Right Subgrade1 - Asphalt: 479.988 C.F., 17.777 C.Y., 2881.487 S.F., 320.165 S.Y.
Total Right Subgrade2 - Asphalt: 1921.865 C.F., 71.180 C.Y., 11531.712 S.F., 1281.301 S.Y.
Total Right Subgrade3 - Asphalt: 1921.510 C.F., 71.167 C.Y., 5765.856 S.F., 640.651 S.Y.
Total Right Subgrade4 - Gravel: 3842.962 C.F., 142.332 C.Y., 5765.856 S.F., 640.651 S.Y.
```



Pulldown Menu Location: Roads

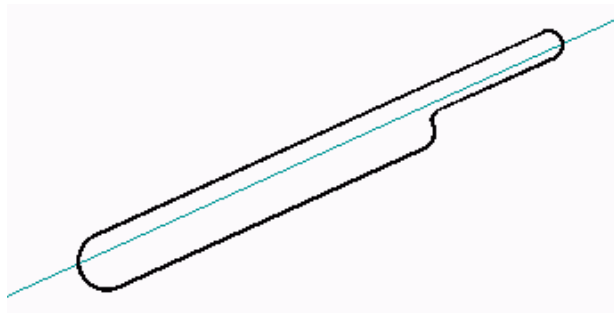
Keyboard Command: rdrehab

Prerequisites: Centerline, template and surface files

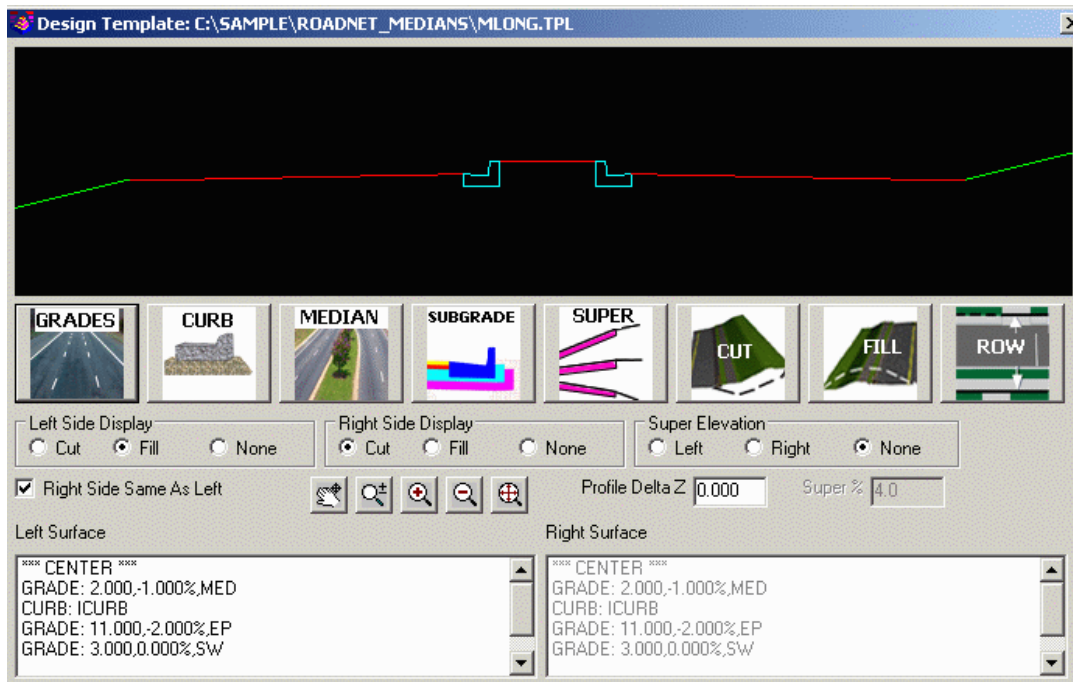
Define Median Adjustments

This command prepares a road design for median transitions by picking a plan view polyline for the median alignment and then creating the road design files for a median for the Template Point Centerline (.tpc) and Template Series File (.tsf).

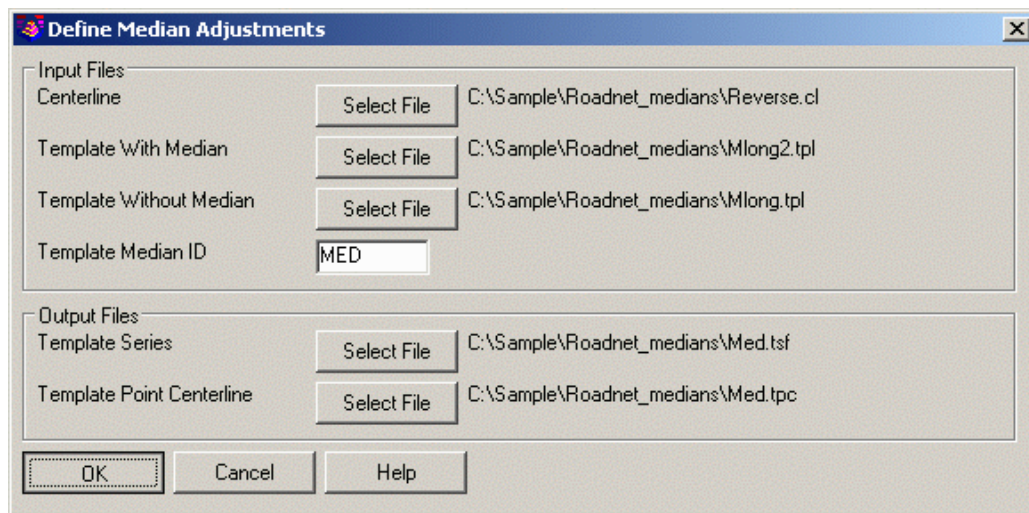
Before running this command, draw a polyline for the median in plan view. The road centerline must cross through the median as shown here.



Also before running this command, use Design Template to make two template files (TPL) where one template has the median and the other is without the median. In the example shown here, this template has the median with the grade of MED and the curb. The template without the median in this example does not have the MED grade and curb.

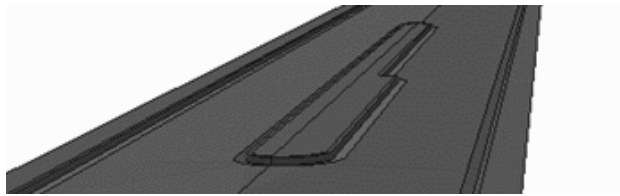


In the dialog for this command, select the input files for the centerline and template with and without the median. Also set the grade ID from the template for the median.



Finally, after the dialog the program prompts to select the median polyline from the drawing. Then the output Template Series File will assign the median and non-median templates based on where the centerline crosses the median polyline. The output Template Point Centerline will assign centerlines for the median using the median polyline.

To apply the median design, run Process Road Design or Road Network and use the Template Series File (TSF) and Template Point Centerline (TPC) for the road design input.



Pulldown Menu Location: Roads

Keyboard Command: prep_median

Prerequisite: Centerline, templates and median polyline

Define Road Design Parameters

This command defines design parameters that can be checked against a road design with the Process Road Design and Road Network commands. Only fill in the parameters to have checked. If you leave a parameter blank, then that parameter is not checked. You can have a different set of parameters for different stations along the road in case road conditions change such as different speed limits. The different sets of parameters are listed by station on the left of the dialog. Use the Add button to add a new parameter set and use the Remove button to remove the set highlighted in the station list. To view a parameter set, pick the station in the Stations list. If you don't need different sets of parameters, then leave the Starting Station as zero and don't add other stations.

Curve Radius	Max Slope%	Range
600.0	6.0	<= 600.0
999.0	5.0	600.0 to 999.0
1499.0	4.0	999.0 to 1499.0
1999.0	3.0	1499.0 to 1999.0
6999.0	2.0	1999.0 to 6999.0
	0.0	> 6999.0

Starting Station: This is the station to begin using the current set of design parameters.

Max Slope: This is the maximum profile slope percent that is allowed.

Max Distance at Max Slope: This is the maximum continuous distance that the profile can be at the max slope.

Min Slope: This is the minimum profile slope percent that is allowed.

Min Curve Radius: This is the minimum horizontal curve radius for the centerline.

Min Curve Radius for Delta Angle Range: This is the minimum horizontal curve radius for the centerline for curves with a delta angle in the specified range. Enter the delta angles in decimals degrees from low to high.

Min Sight Distance: This is the minimum sight distance for the profile.

Min K-Value: This is the minimum k-value for the profile.

Max Slope at Intersection: This is the maximum profile slope at an intersection with another road. The **Setback** is the distance along the profile from the intersection point that this max slope applies. This option only applies to Road Network.

Max Percent of Road over Slope: This is the maximum percent of the road that can be over the specified **Slope**.

Min Vertical Curve Length for Delta Slope over: This is the minimum vertical curve length for PVI's with an algebraic grade difference greater than the specified Delta Slope. The Delta Slope units are in percent slope.

Min Vertical Curve Length for Delta Slope between: This is the minimum vertical curve length for PVI's with an algebraic grade difference between than the specified Delta Slopes. Enter the delta slopes in percent slope format from low to high.

Max SuperElev Rate of Change Per 100: This is the maximum rate of change in the superelevation for the cross slope. The rate units are in percent slope per 100 feet or meters depending on your drawing units.

Check for Tangential Centerline: This option checks the horizontal alignment to make sure all the segments are tangential.

Check for Flat Areas: This option checks the superelevation at the Run-In and Run-Out stations where the outside lane is flat and warns if the profile is also flat at those stations.

Max SuperElevation: This is the maximum superelevation cross slope at full super for different curve radii. A lookup table of curve radius and max slope is used. The curve radii should be entered from low to high.

The **Horizontal** and **Vertical Speed Tables** are for referencing values to fill in for the design parameters.

Pulldown Menu Location: Roads

Keyboard Command: rdparam

Prerequisite: None

Vehicle Path Tracking Profile

This command traces the vertical paths for vehicle dimensions along a profile. The profile is defined by a profile drawing which must be created before running this command. The center of the front axle, the center between the rear axle(s) and if selected the center of the trailer axle(s) follow the profile.



Select profile polyline: *select a profile polyline from a Profile Drawing*

After selecting the profile, the program will display the Vehicle Path Tracking Profile dialog. The Vehicle Path

Tracking Profile dialog has three tabs; Vehicle, Trailer and Profile. Each tab contains data for the corresponding component.

Vehicle

Vehicle Path Tracking Profile

Vehicle | Trailer | Profile

Vehicle Height	12.000
Front to High Point	9.000
Front Overhang	3.500
Wheel Length	15.000
Rear Overhang	5.000
Receiver Offset	6.666
Minimum Clearance	1.000
Frame Height	3.500
Rear Clearance	2.000
Tire Diameter	3.500
Rear Axle Count	2
Rear Axle Offset	4.000

Draw Front
 Draw Rear
 Draw High Point
 Draw Minimum Clearance

Front Layer: PRO_VEHICLE_FRONT_OVERHANG Set
Rear Layer: PRO_VEHICLE_REAR_OVERHANG Set
High Point Layer: PRO_VEHICLE_HIGHPOINT Set
Minimum Clearance Layer: PRO_VEHICLE_MIN_CLEARANCE Set

QK Cancel Load Save Help

Vehicle Height: Height of vehicle at the highest point from the ground.

Front to High Point: Distance between the front of vehicle body to the highest point.

Front Overhang: Distance from front axis to front of vehicle body.

Wheel Length: Distance from front axis to center of rear axis.

Rear Overhang: Distance from rear axis to the back of the vehicle body.

Receiver Offset: Distance from receiver to the back of the vehicle body.

Minimum Clearance: Distance from ground to bottom of the vehicle body.

Frame Height: Distance from ground to the top of the vehicle frame.

Rear Clearance: Distance from ground to the bottom of the vehicle frame.

Tire Diameter: Diameter of vehicle tires.

Rear Axle Count: Number of rear axles.

Rear Axle Offset: Distance between rear axles.

Draw Front: Draws the profile of the front low point of vehicle body.

Draw Rear: Draws the profile of the rear low point of vehicle body.

Draw High Point: Draws the profile of the high point of vehicle body.

Draw Minimum Clearance: Draws the profile of the lowest point of vehicle body between axles.

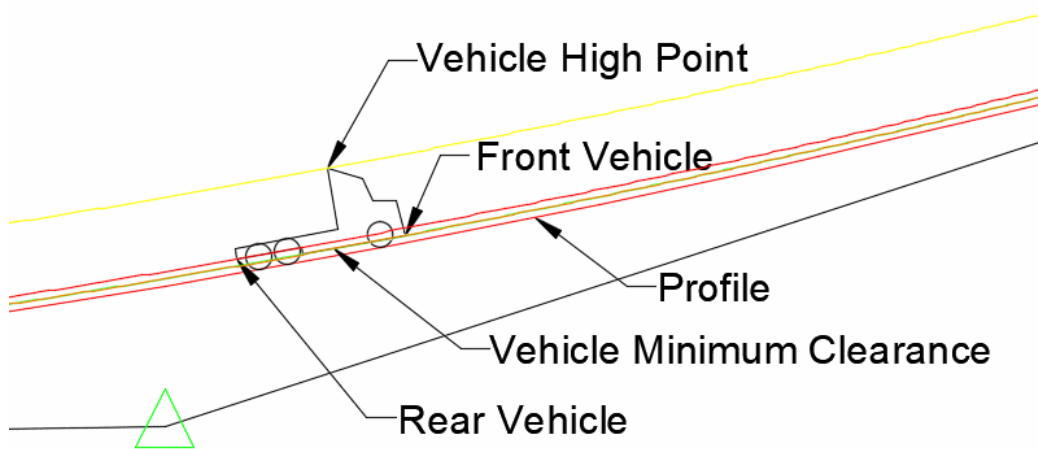
Front Layer: Layer to draw the profile of the front low point of vehicle body on.

Rear Layer: Layer to draw the profile of the rear low point of vehicle body on.

High Point Layer: Layer to draw the profile of the high point of vehicle body on.

Minimum Clearance Layer: Layer to draw the profile of the lowest point of vehicle body between axles on.

Set: Select the corresponding vehicle layer from list.



Trailer

Vehicle Path Tracking Profile

Vehicle Trailer Profile

Use Trailer

Trailer Height	13.000
Trailer Length	53.000
Ride Height	4.000
Front to King Pin	4.250
Front to Landing Gear	13.750
Rear Overhang	8.000
Landing Gear Clearance	1.000
Bumper Clearance	2.000
Tire Diameter	3.500
Axle Count	2
Axle Offset	4.000

Draw Top Front
 Draw Top Rear
 Draw Top
 Draw Landing Gear
 Draw Bumper
 Draw Minimum Clearance

Top Front Layer: PRO_TRAILER_TOP_FRONT [Set]
Top Rear Layer: PRO_TRAILER_TOP_REAR [Set]
Top Layer: PRO_TRAILER_TOP [Set]
Landing Gear Layer: PRO_TRAILER_LANDING_GEAR [Set]
Bumper Layer: PRO_TRAILER BUMPER [Set]
Minimum Clearance Layer: PRO_TRAILER_MIN_CLEARANCE [Set]

OK Cancel Load Save Help

The diagram shows a top-down view of a trailer with various dimensions and components labeled. The 'Trailer Height' is the vertical distance from the ground to the top of the trailer. The 'Trailer Length' is the horizontal distance from the front to the rear. The 'Front to King Pin' is the distance from the front of the trailer to the king pin. The 'Front to Landing Gear' is the distance from the front of the trailer to the landing gear. The 'Rear Overhang' is the distance from the landing gear to the rear of the trailer. The 'Ride Height' is the vertical distance from the ground to the bottom of the trailer. The 'Landing Gear Clearance' is the vertical distance from the ground to the landing gear. The 'Bumper Clearance' is the vertical distance from the ground to the bumper. The 'Front to Landing Gear' and 'Rear Overhang' are also labeled with arrows pointing to the respective dimensions.

Use Trailer: Use this toggle to setup the trailer data and choose profiles to draw.

Trailer Height: Height of trailer from the ground.

Trailer Length: Overall length of trailer.

Ride Height: Height of the bottom of the trailer from the ground.

Front to King Pin: Distance from the front of the trailer to the King Pin (Hitch Point).

Front to Landing Gear: Distance from the front of the trailer to the landing gear.

Rear Overhang: Distance from rear axis to the back of the trailer.

Landing Gear Clearance: Distance from ground to landing gear.

Bumper Clearance: Distance from ground to the bottom of the trailer bumper.

Tire Diameter: Diameter of trailer tires.

Axle Count: Number of trailer axles.

Axle Offset: Distance between trailer axles.

Draw Top Front: Draws the profile of the front high point of the trailer.

Draw Top Rear: Draws the profile of the rear high point of the trailer.

Draw Top: Draws the profile of the highest point of the trailer between the front and rear.

Draw Landing Gear: Draws the profile of the trailer landing gear.

Draw Bumper: Draws the profile of the trailer bumper.

Draw Minimum Clearance: Draws the profile of the lowest point between the landing gear and the axle axis.

Top Front Layer: Layer to draw the profile of the top front point of the trailer on.

Top Rear Layer: Layer to draw the profile of the top rear point of the trailer on.

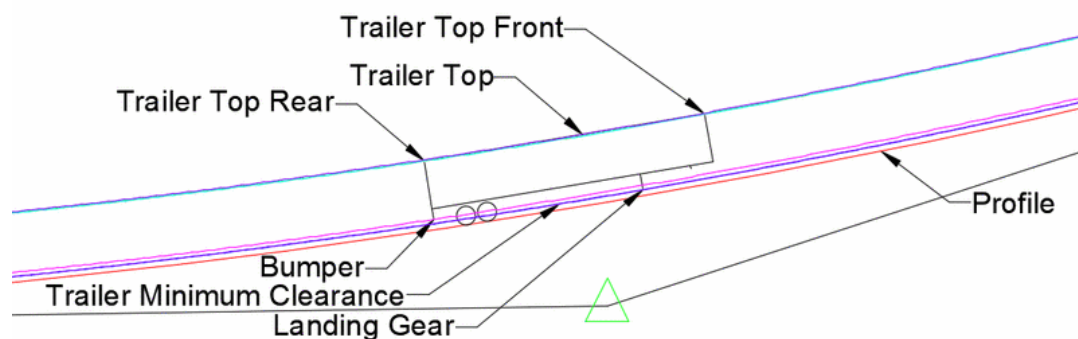
Top Layer: Layer to draw the profile of the top of trailer between front and rear on.

Landing Gear Layer: Layer to draw the profile of the trailer landing gear on.

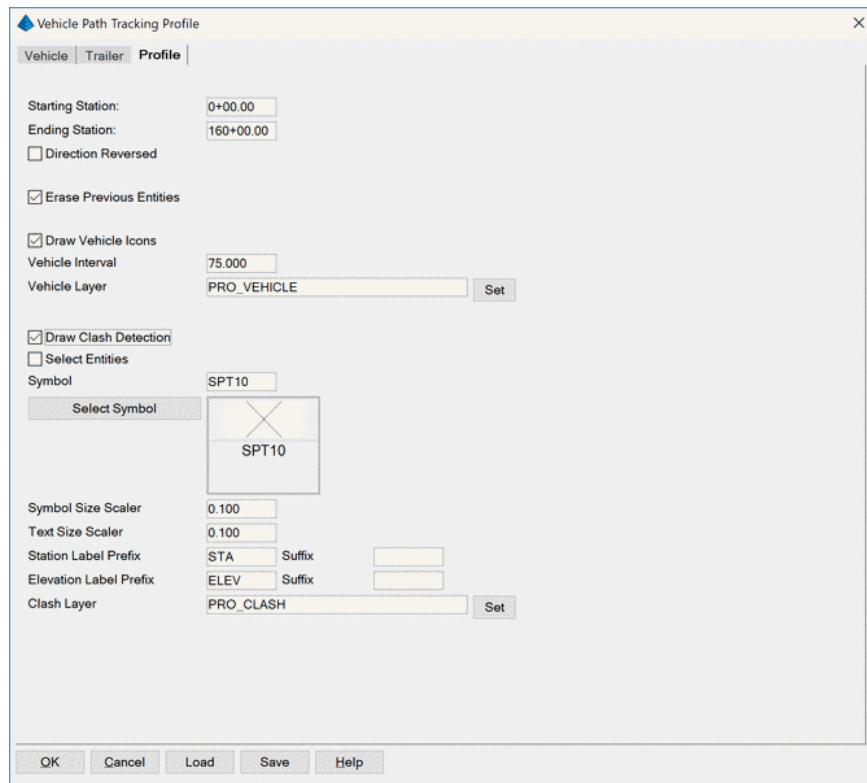
Bumper Layer: Layer to draw the profile of the trailer bumper on.

Minimum Clearance Layer: Layer to draw the profile of the lowest point of the trailer between the landing gear and the axle axis on.

Set: Select the corresponding trailer layer from list.



Profile



Starting Station: Starting station for vehicle and if selected trailer profiles to be calculated.

Ending Station: Ending station for vehicle and if selected trailer profiles to be calculated.

Direction Reversed: Draws the profiles from ending toward starting stations.

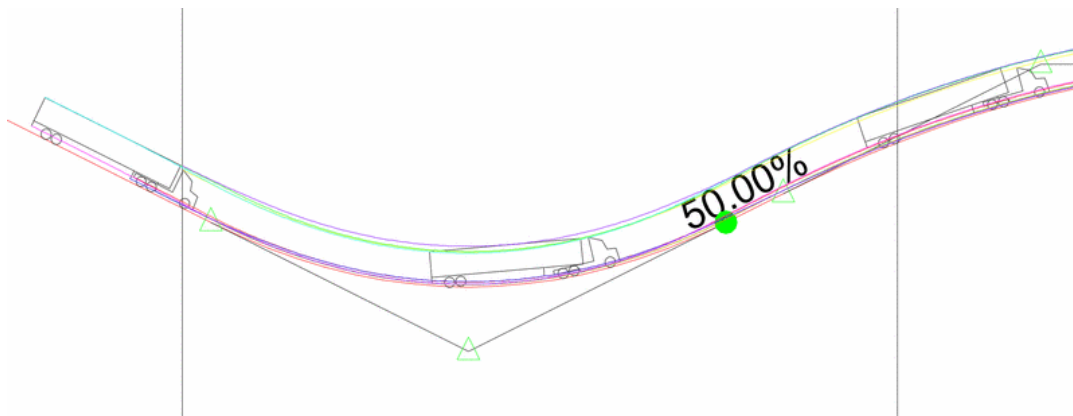
Erase Previous Profile Entities: Removes previously drawn profiles for selected profile.

Draw Vehicle Icons: Draws the vehicle and if selected trailer icons.

Vehicle Interval: Interval distance between icons.

Vehicle Layer: Layer to draw the icons on.

Set: Select the icon layer from list.



Draw Clash Detection: This option will detect locations where the profile paths conflict with the selected Profile.

Select Entities: This option allows for additional CAD objects to be selected and checked for conflicts with profile paths.

Symbol: Symbol to place at the detected clash locations.

Select Symbol: Select symbol from the symbol library.

Symbol Size Scaler: Sets the size for the symbols at the detected clash locations.

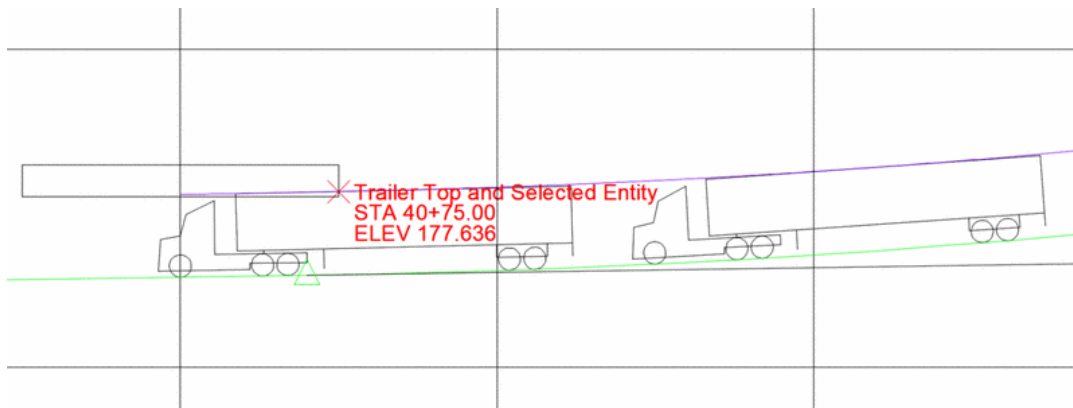
Text Size Scaler: Set the size of the labels for the detected clash locations.

Station Label Prefix / Suffix: Prefix and Suffix for all station labels placed at detected clash locations.

Elevation Label Prefix / Suffix: Prefix and Suffix for all elevation labels placed at detected clash locations.

Clash Layer: Layer to draw the detected clash icons and labels.

Set: Select the clash layer from list.



The **Save** and **Load** buttons save and recall the vehicle dimensions to a .VTV file.

Pulldown Menu Location: Roads > Vehicle Path Tracking

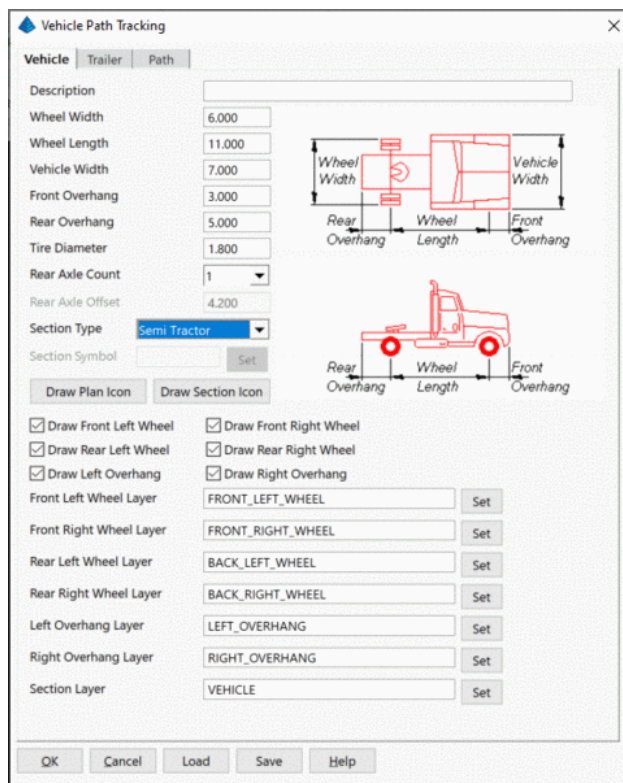
Keyboard Command: auto_track_pro

Prerequisite: Profile Drawing

Vehicle Path Tracking

This command traces the wheel paths for vehicle dimensions along a centerline. The centerline is defined by a polyline which must be created before running this command. The front axle follows this centerline based on the path settings from the Path tab. After specifying the vehicle dimensions and draw options in the command dialog, the program prompts for the centerline polyline and then draws the paths.

For the vehicle settings, pick on the Vehicle Tab. These are the set of dimensions for the vehicle and separate layers for the vehicle points to track.



Description: Description of the Vehicle Path Tracking command settings if saved. (Optional)

Wheel Width: Distance along the wheel axis to the outside of the tires.

Wheel Length: Distance between the front axle and rear axis.

Vehicle Width: Outside width dimension of the vehicle body.

Front Overhang: Distance from front axle to front of vehicle body.

Rear Overhang: Distance from rear axis to the back of the vehicle body.

Tire Diameter: Outside diameter of the vehicle tires.

Rear Axle Count: Number of rear axles.

Rear Axle Offset: Distance between rear axles.

Section Type: Section List for Vehicle types.

Section Symbol: Symbol to represent the vehicle.

Set: Select the corresponding vehicle symbol.

Draw Plan Icon Select location and places plan view detail of Vehicle in CAD.

Draw Section Icon Select location and places section view detail of Vehicle in CAD.

Draw Front Left Wheel: Draws the front left wheel path.

Draw Front Right Wheel: Draws the front right wheel path.

Draw Rear Left Wheel: Draws the rear left wheel path.

Draw Rear Right Wheel: Draws the rear right wheel path.

Draw Left Overhang: Draws the left overhang of the vehicle path.

Draw Right Overhang: Draws the right overhang of the vehicle path.

Front Left Wheel Layer: Layer to draw the front left wheel path on.

Front Right Wheel Layer: Layer to draw the front right wheel path on.

Rear Left Wheel Layer: Layer to draw the rear left wheel path on.

Rear Right Wheel Layer: Layer to draw the rear right wheel path on.

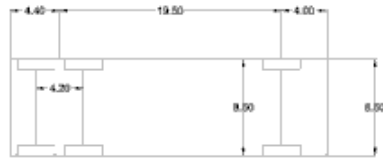
Left Overhang Layer: Layer to draw the left overhang of the vehicle path on.

Right Overhang Layer: Layer to draw the right overhang of the vehicle path on.

Section Layer: Layer to draw the details of the vehicle on.

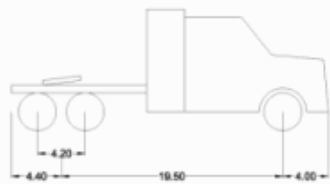
Set: Select the corresponding vehicle layer from list.

Interstate Semi-Trailer [WB-20/65]

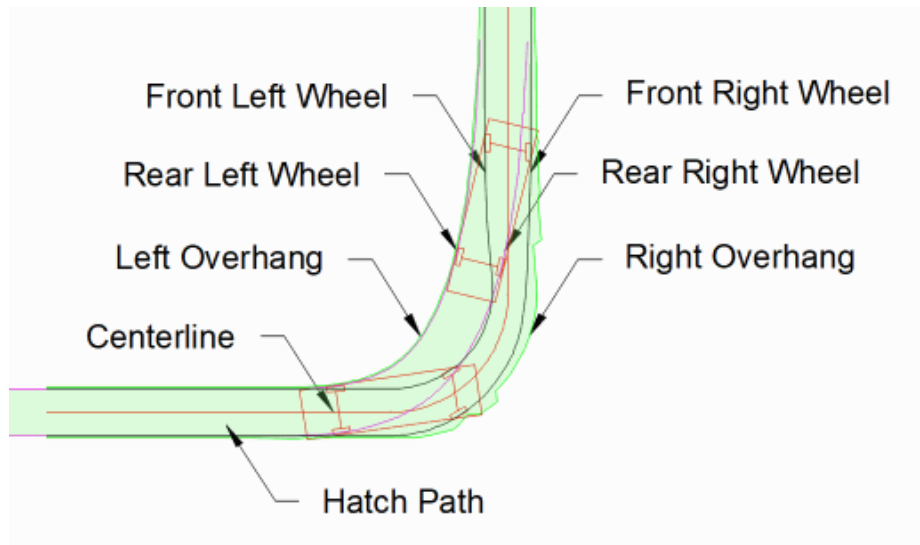


Tire Diameter: FEET
3.33

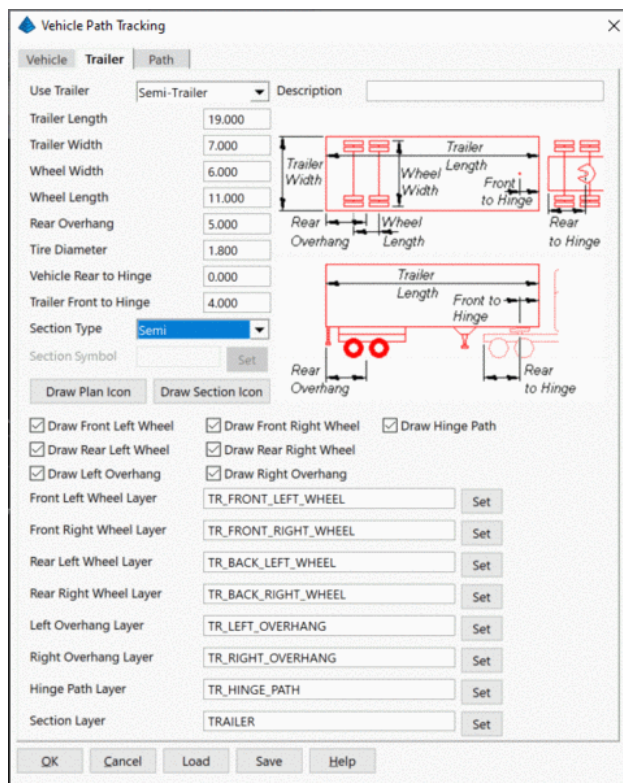
Interstate Semi-Trailer [WB-20/65]



Vehicle Width: FEET
8.5
Wheel Width: 8.5
Tire Diameter: 3.33



For the hinged vehicle settings, pick on the Trailer Tab. These are the set of dimensions for the trailer and separate layers for the trailer points to track.



Use Trailer: Use this list to setup the trailer data and choose paths to draw. Choose None or between Tow and Semi-Trailer to change the values for easier input based on semi-trailers or tow trailers.

Description: Description of the Trailer if saved. (Optional)

Trailer Length: Outside length dimension of the trailer.

Trailer Width: Outside width dimension of the trailer body.

Wheel Width: Distance along the wheel axis to the outside of the tires.

Wheel Length: Distance between the front axle and rear axle.

Rear Overhang: Distance from rear axis to the back of the trailer body.

Tire Diameter: Outside diameter of the trailer tires.

Vehicle Rear to Hinge: Distance from the rear of the vehicle and the hinge location.

Trailer Front to Hinge: Distance from the front of the trailer to the hinge location.

Section Type: Section List for Trailer types.

Section Symbol: Symbol to represent the trailer.

Set: Select the corresponding trailer symbol.

Draw Plan Icon Select location and places plan view detail of Trailer in CAD.

Draw Section Icon Select location and places section view detail of Trailer in CAD.

Draw Hinge Path: Draws the hinge path.

Draw Front Left Wheel: Draws the front left wheel path.

Draw Front Right Wheel: Draws the front right wheel path.

Draw Rear Left Wheel: Draws the rear left wheel path.

Draw Rear Right Wheel: Draws the rear right wheel path.

Draw Left Overhang: Draws the left overhang of the trailer path.

Draw Right Overhang: Draws the right overhang of the trailer path.

Front Left Wheel Layer: Layer to draw the front left wheel path on.

Front Right Wheel Layer: Layer to draw the front right wheel path on.

Rear Left Wheel Layer: Layer to draw the rear left wheel path on.

Rear Right Wheel Layer: Layer to draw the rear right wheel path on.

Left Overhang Layer: Layer to draw the left overhang of the trailer path on.

Right Overhang Layer: Layer to draw the right overhang of the trailer path on.

Hinge Path Layer: Layer to draw the hinge path on.

Section Layer: Layer to draw the details of the trailer on.

Set: Select the corresponding trailer layer from list.

Interstate Semi-Trailer [WB-20/65]

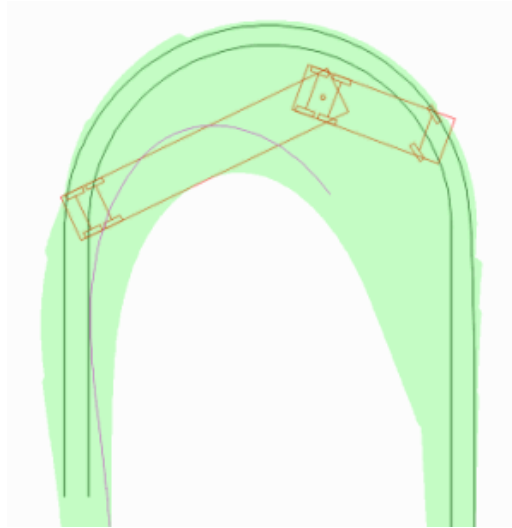
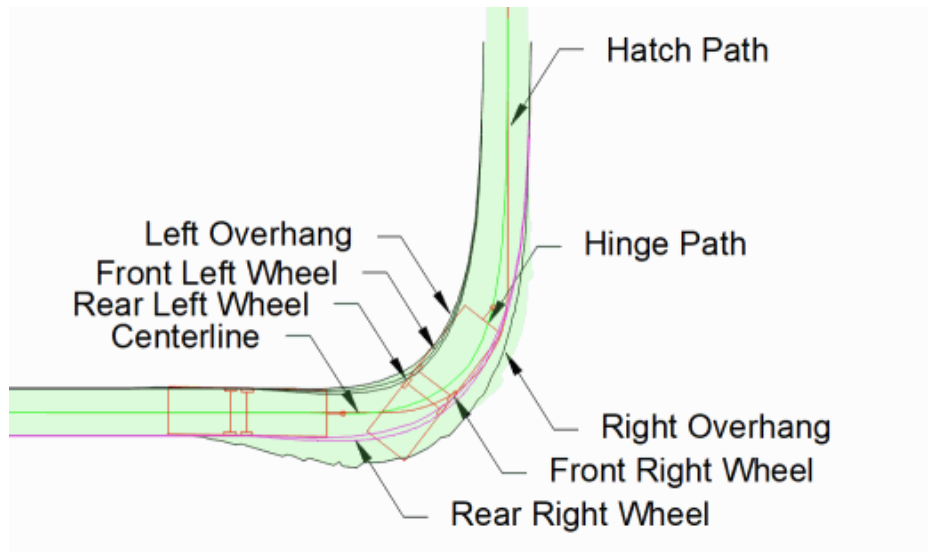


Tire Diameter: FEET
3.33

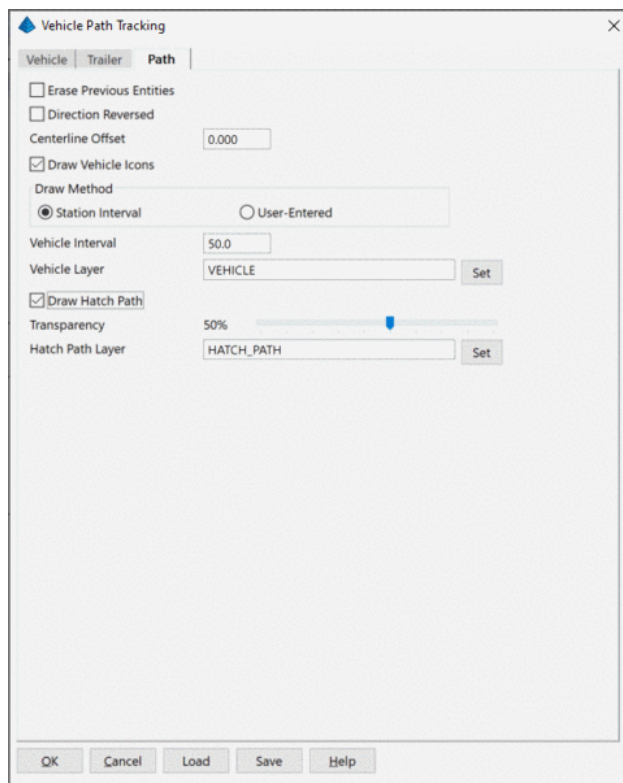
Interstate Semi-Trailer [WB-20/65]



Trailer Width: FEET
8.5
Wheel Width: FEET
8.5
Tire Diameter: FEET
3.33



For the path settings, pick on the Path Tab.



Erase Previous Entities: Removes previously drawn paths for selected centerline.

Direction Reversed: Processes the centerline from the end to the start.

Centerline Offset: Distance from the centerline for the front axle to follow.

Draw Vehicle Icons: Draws the vehicle and trailer symbols with the specified dimensions.

Draw Method Choose between placing vehicle symbols at the specified Station Interval along the centerline or User-Entered for prompting to pick points along the centerline to place the vehicle symbols.

Vehicle Interval: Interval distance between vehicle symbols.

Vehicle Layer: Layer to draw the vehicle and trailer symbols on.

Draw Hatch Path: Hatches the entire path of the vehicle and trailer along the centerline.

Transparency: Transparency applied to the hatch drawn.

Hatch Path Layer: Layer to draw the hatched path on.

Set: Select the corresponding layer from list.

The **Save** and **Load** buttons save and recall the vehicle and trailer dimensions to a .VTP file.

Prompts

Draw Plan Icon/Draw Section Icon

Pick Insertion Point: *pick the point for the detail*

Vehicle Path Tracking dialog

Select centerline polyline: *pick a polyline*

Pulldown Menu Location: Roads - Vehicle Path Tracking

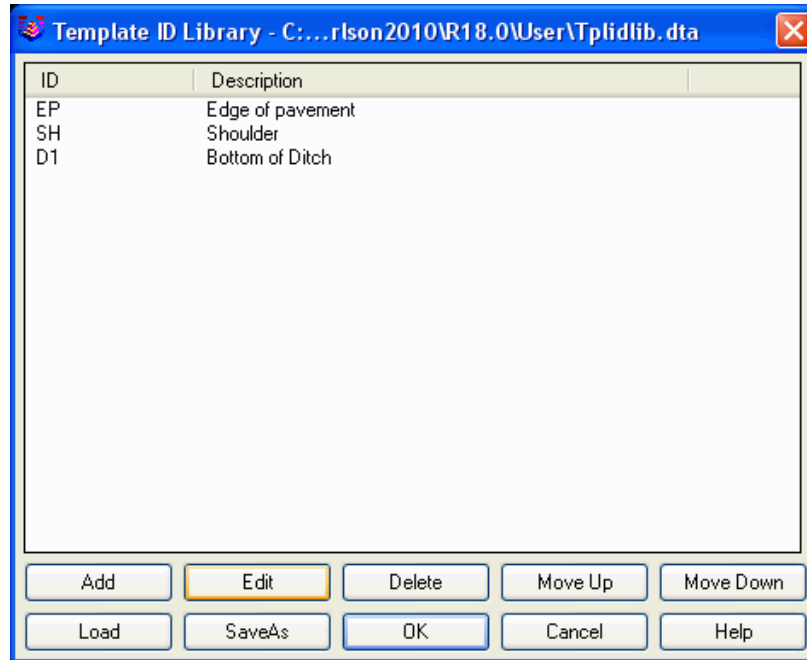
Keyboard Command: auto_track

Prerequisite: Centerline polyline

Template ID Library

This command defines template ID's along with associated descriptions. The ID's can be selected in Design Template when adding grades, curbs or medians. The descriptions are only for identification during the ID selection. In Design Template, there are Set buttons next to the ID edit fields that select from the list of ID's defined in the Template ID Library. The purpose of the Template ID Library is to help with consistent naming of template elements.

The current Template ID Library is stored in the current USER folder under Documents & Settings in a file called tplidlib.dta. The Load and SaveAs functions can be used to store and recall ID settings to a .TID file.



Pulldown Menu Location: Roads

Keyboard Command: tplid.lib

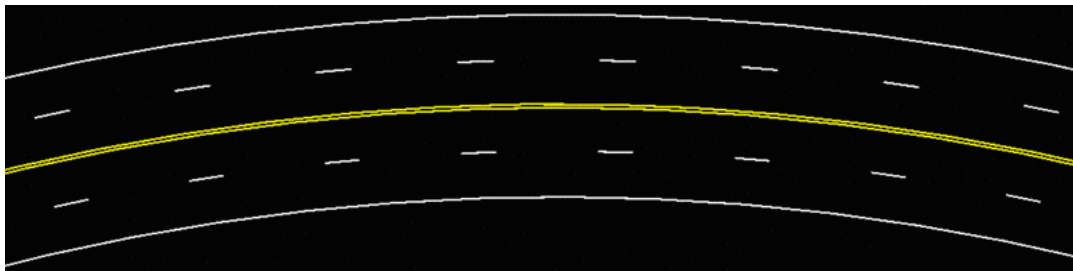
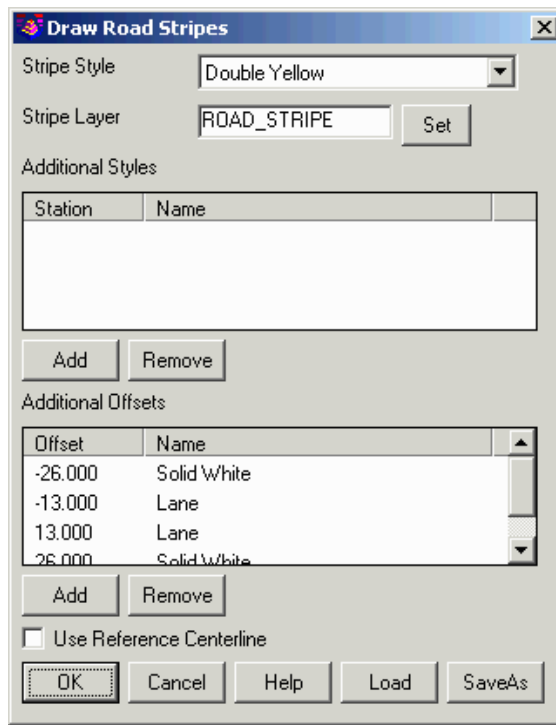
Prerequisite: None

Draw Road Stripes

This command draws polylines for stripe styles defined in Road Stripe Library. After selecting the Stripe Style from the dialog, the program prompts for the horizontal alignment which can be either a centerline file (.CL) or a polyline. Then the polylines for the stripe are drawn.

In the options dialog, the Stripe Style is selected from a list of names defined in the Road Stripe Library. The Stripe Layer is used for all the stripe polylines that are created. The Additional Styles list can be used for using different stripe styles along the alignment such as for a centerline that switches between double yellow to dashed yellow for a passing zone. The Additional Offsets list allows for drawing several road lines at a time based on offsets from the centerline such as drawing the white lane dashes and outside solid white lines along with the center double yellow lines. The Use Reference Centerline applies for dashed stripe styles that you want to control the dash spacing from the main centerline.

The Load and SaveAs buttons can be used to store and recall the stripe settings to a .RST file for archiving or sharing. The .RST file can also be used in Road Network for drawing the road stripes during that command.



Prompts

Draw Road Stripes dialog

CL File/<Select stripe polyline>: *pick polyline*

Pulldown Menu Location: Roads

Keyboard Command: road_stripe

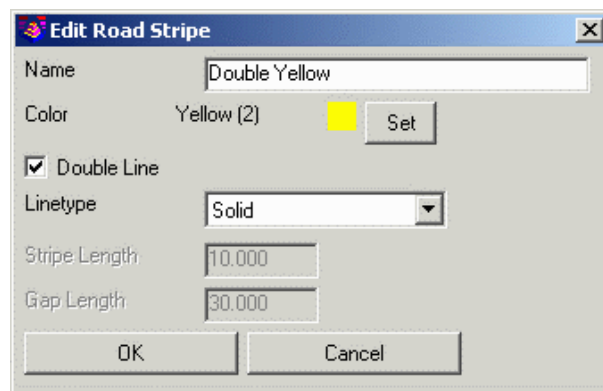
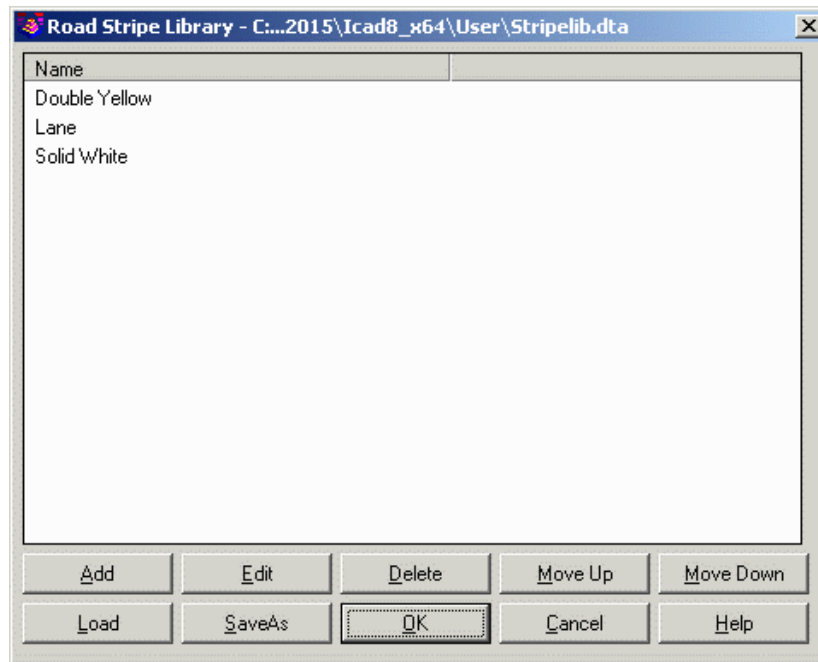
Prerequisite: None

Road Stripe Library

This command defines a collection of line styles. Each line style has a Name for a description, Color and Linetype. The Double Line option is for line styles of two parallel lines. The Linetype can be Solid for continuous line styles or Dash. For Double Lines, the Linetype can also be a solid on one side and dashed on the other such as for passing zones on the center double yellow line. For Dash Linetypes, the Stripe Length and Gap Length control the length of the dash and spacing between the dashes. Use the Add, Edit, Delete and Move Up/Down buttons to manage the list of line styles.

When the line styles are defined in the library, use the Draw Road Stripes command to create the line styles in the drawing.

The current Road Stripe Library is stored in the current USER folder under Documents & Settings in a file called stripelib.dta. The Load and SaveAs functions can be used to store and recall the stripe library to a .RSL file for archiving or sharing.



Pulldown Menu Location: Roads

Keyboard Command: stripe.lib

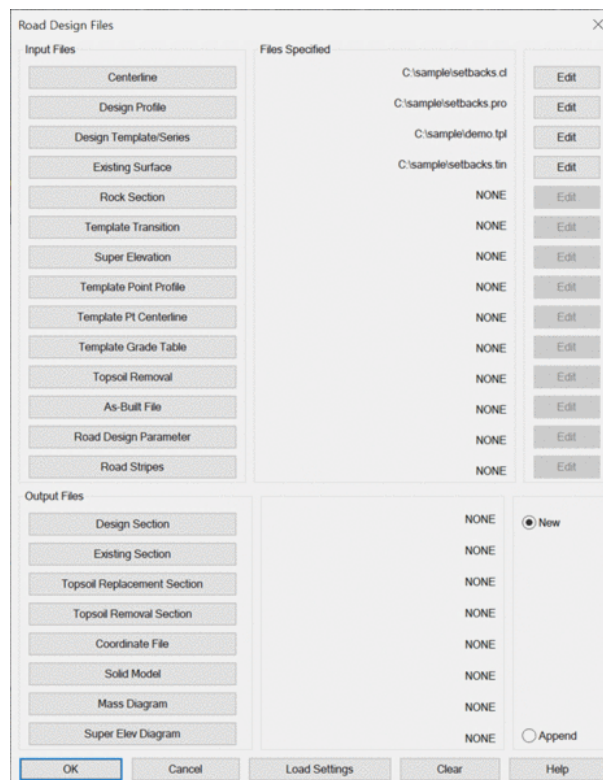
Prerequisite: None

Process Road Design

The primary function of this command is to assemble all of the components for a road design and process them together. While all of the Input Files can be created prior to accessing the Process Road Design command, all

can be edited from the Road Design Files dialog, and many files can actually be created from the Road Design Files dialog itself. The actual processing of the Road Design essentially applies the design template at the design profile elevation along the specified centerline and computing the out slopes and earthworks relative to the existing ground surface. The earthworks report can be shown in the standard report viewer or customized with the Report Formatter option. Secondary functions include creating a final grade section file for plotting with the *Draw Section File* command, creating final grade points in a coordinate file, creating a final surface/contour model, and drawing the road as 3D polylines. You can also output a mass haul diagram profile. The program also has options for applying a superelevation file, template transition file, template point profile, template point centerline, rock section file, an as-built existing section file and a topsoil removal file. *Process Road Design* can be used not just for final road design computations but for levees, channels and any template-based application.

This command begins with the dialog shown below. The top section contains input Files. In a typical implementation of this command, you will have already defined a horizontal centerline for the design to follow, however, you could actually pick the Centerline button, pick the New tab, name the new centerline file (.CL), pick Open, and then back in the main Road Design Files dialog, pick the Edit button and layout the centerline design. The only component that you must have already created before running Process Road Design is an Existing Surface file. As long as there is an Existing Ground Surface, the command will generate the Existing Ground Profile automatically, and the Proposed Finish Grade Profile can be created with the Edit button. Even a Design Template can be created right from here as well. Ultimately, the top 3 Input items (Centerline, Design Profile, and Design Template/Series) are required to Process a Road Design, leading to final sections and full contouring and 3D viewing. The Existing Surface is needed as well to process with earthwork calculations and tie slopes.



The rest of the input files are optional design files. It should be pointed out that Template Point Profile and Template Point Centerline enable template IDs to follow any defined centerline or profile and provide total flexibility of design. Lane widening, matching existing curb lines, special ditches, etc. can be easily accomplished with these two options. The template IDs simply "string along" or follow these pre-defined alignments, and the rules of the template apply to all other template ID points.

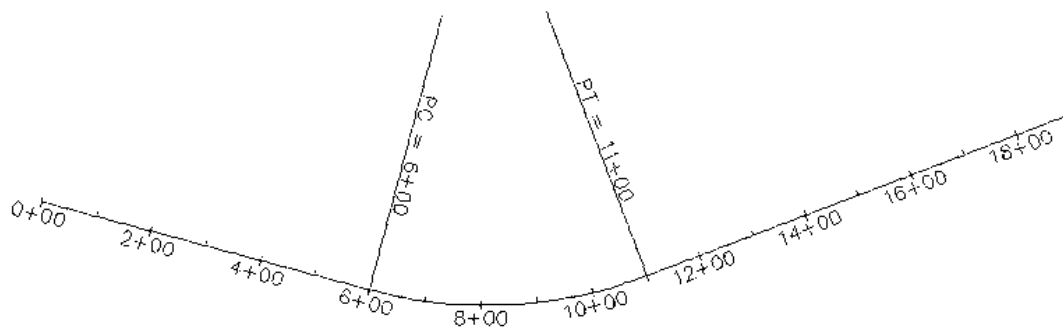
The Output Files section allows you to specify files to store the processing results. The Section File creates a final grade section file that can be drawn with *Draw Section File*. The Topsoil Section File creates the modified existing ground section file if Topsoil Removal is set in the input. This "post-topsoil removal" section file can be used for earthworks calculations to compare any stage of work, using *Calculate Sections Volume* under the

Section pulldown menu. The Coordinate File creates a coordinate file containing every break point in the final grade. The point descriptions include the station, offset and template ID. Whether to include the subgrade points as well as the final surface points is determined by the Include SubGrade Points in Output CRD File option on the next dialog. To the right of the Output Files is the option to create new output files or append to existing output files. If you extend the road, or revise a portion of the project, you can simply "Append" rather than overwrite. The first time that you run this command for stations 0-1000, you would set Output Files to New. Then you could run this command again, possibly with new inputs, for stations 1000-2000 and set Output Files to Append.

On the next dialog, there is a Save Settings button to store all the settings from the first and second dialogs into a specified Road Design File with an (.RDF) file extension. Recorded (.RDF) files can be recalled later using the Load Settings option.

Centerline

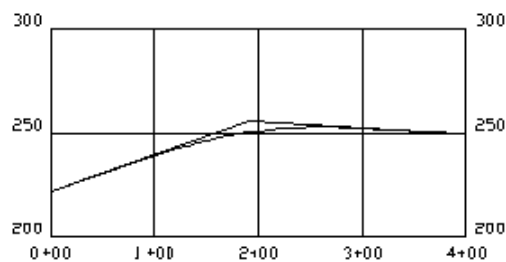
Specify the name of the Centerline file with this option. The (.CL) file contains the horizontal alignment geometry for a project. This parameter file must be specified if you want to have earthworks centroid corrections computed, generate final coordinates, Disturbed Area Polyline, and/or use Triangulate & Contour. The centerline file can be created by the *Design Centerline* or *Polyline to Centerline* commands in the Design pulldown menu.



Example Centerline

Design Profile

Specify the design profile (.PRO) file to derive the centerline elevations when the template is applied. This file defines the vertical alignment and is always required. The profile can be created with any of the profile creation routines in the Profile menu, but typically you would use *Design Road Profile* or *Input Edit Profile*.



Example Design Profile

Design Template/Series

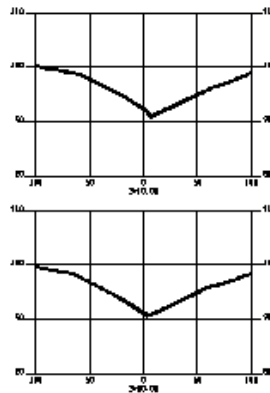
Specify a template definition (.TPL) file or template series (.TSF) file that defines the final grade offsets and elevations and the cut/fill slopes. The template file is created by the *Design Template* command and the template series file (a set of templates ordered by range of stations) is created using *Template Series Editor*. A single template

file or a template series file is required to run *Process Road Design*.

Example Design Template

Existing Surface

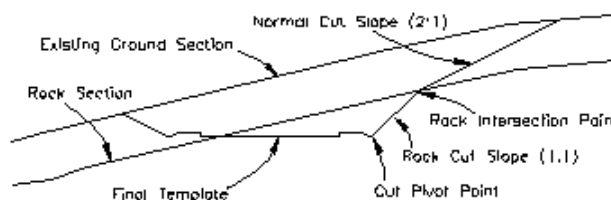
Specify the surface model which will be treated as the existing ground for cut and fill volumes and to calculate the outslope intersections when the template is applied at the profile elevations. This Existing Surface can be defined by either a section file or triangulation. The section file can be created with commands such as *Sections from Surface Entities*, *Input/Edit Section File*, *Sections from Points* or one of the *Digitize Sections* commands on the Section menu. The triangulation file can be created with the *Triangulate & Contour* command.



Example Existing Sections

Rock Section File

This option specifies an optional rock section file that is used as an additional surface. When in cut, a special cut slope is used up to the intersection of the rock surface. After this intersection, the normal cut slopes apply. The special rock cut slope is specified in Design Template under the cut options. If the "pivot point" in cut is below the rock line, then the special rock cut slope will be applied. Note that rock sections can be derived from borings to rock, as modeled, or can be created quickly by using the "translate" command within *Section File Editor* to translate the existing ground sections by a vertical offset (e.g. -6) to an approximate top of rock.



Detail of rock cut slope

Template Transition File

Specify a .TPT file with this option. The Template Transition file allows modified template files to be applied at different ranges of stations on a project. In this way, template IDs can be made to widen (as for passing lanes) and contract. Use the *Template Transition* command under the Design menu to create a template transition file.

Super Elevation File

This option is used to specify a super elevation file (.sup file) that defines the super elevation transition stations on a project. The super elevation file can be created with the *Super Elevation Editor* command.

Template Point Profile

This option lets you have separate profiles for template points that are independent of the centerline profile. This design file is created with the *Assign Template Point Profile* command.

Template Point Centerline

This option lets you have separate centerlines for template points that are independent of the main centerline. This design file is created with the *Assign Template Point Centerline* command.

Template Grade Table

This input file is optional. The Template Grade Table is a method for template transitions that uses a lookup table of distance and slopes at transition stations for each template ID. This design file is created with the *Template Grade Table* command.

Topsoil Removal

This option applies topsoil removal and/or replacement to the existing ground section file. This design file is created with the *Topsoil Removal/Replacement* command.

As-Built File

The As-Built File is a cross section file used to match existing grade and retain as-built portions of a road improvement project. The final cross sections will conform to the as-built cross sections for those template IDs specified in the second dialog. Beyond the specified set of offsets in the as-built cross section file, the design road files will be applied.

Road Design Parameter

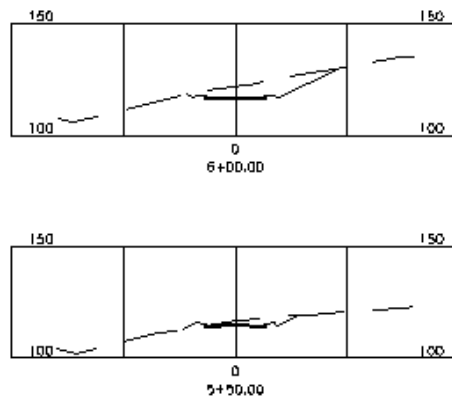
This input file is optional for running checks on the road design for parameters such as min sight distance and max grades. This .RDP file is created with the *Define Road Design Parameters* command.

Road Stripes

This option draws road markings such as double yellow lines along the centerline. The types of markings are defined in the *Road Stripe Library* command. The *Draw Road Stripes* command creates a .RST file for using with Process Road Design. The file sets which markings to draw and sets the centerline offsets.

Output Design Section File

Specify the name of the file to output the final grade sections calculated by applying the template file at profile elevations and calculating the outslope intersection with the existing ground cross sections. This file can then be plotted by using the *Draw Section File* command. After plotting the final sections overlaid on the existing sections, revisions can be made graphically with commands like *PEDIT* and *Polyline by Slope Ratio*. The data output to the file can also be edited and reviewed with the *Section File Editor* command. If the final sections are edited graphically, the revised section data can be updated in the .SCT file with the *Polyline to Section File* command.



Output Section File drawn with Existing Section File by the *Draw Section File* command

Output Existing Section File

This option creates a section file of existing ground. This applies when the existing surface is a triangulation file. The station intervals for the existing section file will match the stations from the design section file.

Output Topsoil Replacement Section File

This option writes out a modified existing ground section adjusted by the topsoil replacement. This option only applies when a Topsoil Removal file is being used.

Output Topsoil Removal Section File

This option writes out a modified existing ground section adjusted by the topsoil removal. This option only applies when a Topsoil Removal file is being used.

Output Coordinate File

This option creates a coordinate file containing every break point in the final grade for the range of processed stations. Using the second dialog, there are additional options to output subgrade and ditch/berm points. The point descriptions include the station, offset and template ID. The station interval is set by the stations in the Existing Section File.

Output Solid Model

This option creates a solid model file (MDL) for all the subgrades.

Output Mass Diagram File

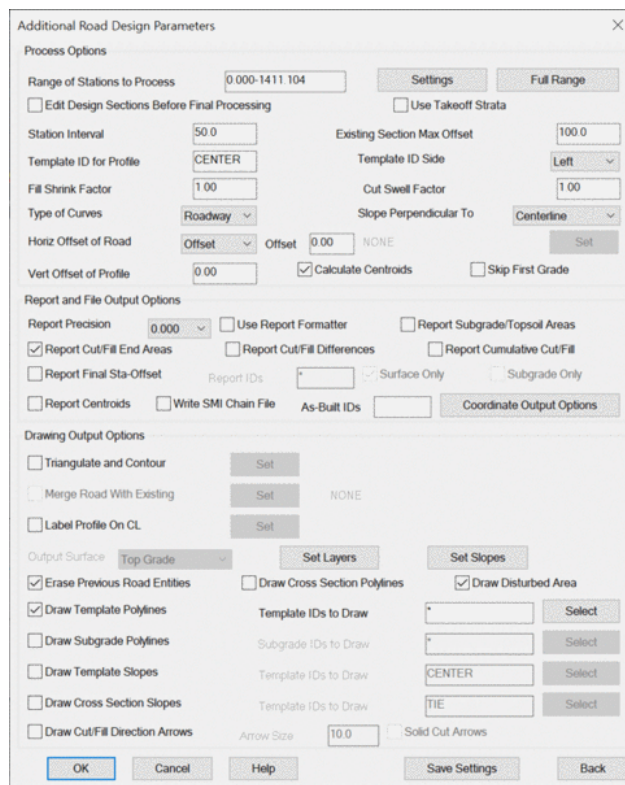
The mass haul diagram can be output as a profile file and shows the cumulative cut and fill along the selected range of stations. Cut and fill is balanced between points on the mass haul profile that cross the Z-axis. Because of the typically large values of cut and fill associated with road and earthwork projects, the vertical scale for the profile may need to be set to 10 times the horizontal scale, or more. The profile preview screen which appears when you select profile for loading will show the elevation range and help suggest an appropriate vertical scale.

Output Super Elevation Diagram File

This option writes out a super elevation transition file (.SUD) that can be used with the Draw Super Elevation Diagram routine. This file contains the template cross slopes and the transition stations.

Running the Road Design Job

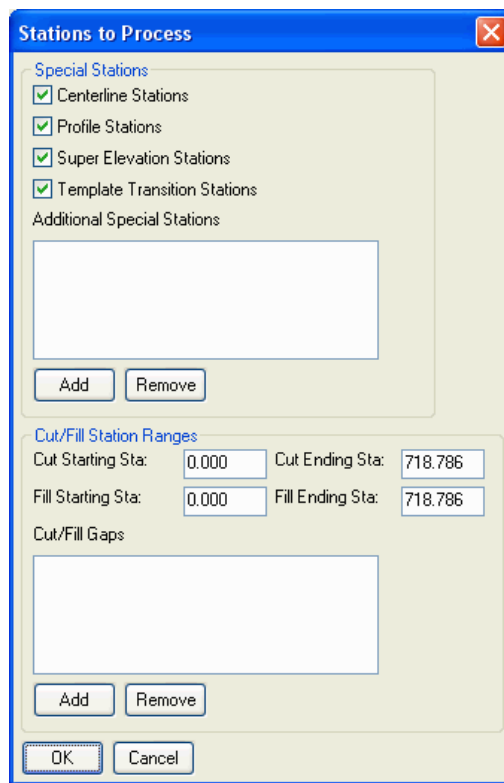
After setting up the files and options in the first dialog click the OK button. The next dialog shown below has processing options.



In the **Process Options** section, the *Range of Stations to Process* field sets the range of station that you want to calculate. Each time you use this command, the existing grade (.SCT) file is scanned and the range in the edit box is set to the minimum and maximum stations in the file. If you change the station range, you can click the *Full Range* button to restore the default full range of stations.

The *Settings* button will interpolate additional existing cross sections (internally) and create final cross sections at special stations like profile high and low points, profile transition stations for PVC and PVT, key centerline points like PC's and PT's, and superelevation and template transition points and any user-defined special stations. These additional station improve volume calculations.

Volumes are calculated using end areas between the range of stations. Also under the *Settings* button, there are controls for the cut/fill starting and ending stations. Instead of cutting off the volumes exactly at this range, the Ending and Starting Stations for Cut and Fill can be used to have the volume taper from zero at the specified Starting Station to the volume at the first station in the range. Likewise the Ending Stations can be used to taper the volume from the last station in the range to zero at the specified Ending Station. You can also specify cut/fill gaps to stop the end area volume calculations over the station range of that gap. This applies in cases like a bridge.



The *Edit Design Sections Before Final Processing* does just that. You can review and edit the final sections in the spreadsheet with graphic view editor similar to the Section File Editor command. For example, you can change the tie slope as selected stations. After making these changes, the modified final sections are used for the rest of the road design process including earthworks and drawing output.

The *Station Interval* and *Existing Section Max Offset* buttons are ghosted if the existing surface is a set of cross sections. If there is no existing surface, or the existing surface is a grid, TIN or FLT file, then you must enter the Station Interval to generate sections along the centerline. Besides the stations at interval, sections can be created at special stations as specified under the Settings button. The *Existing Section Max Offset* controls the max left and right offsets for generating the existing sections when the Existing Surface is defined by a triangulation file. This offset needs to be set far enough for the final sections out slopes to tie into existing. On the other hand, keeping this offset fairly close to the tie point will help make processing run faster.

The *Calculate Centroid* option applies to centerlines containing curves. The centroids of the cuts and fills will be computed, and the radius to these centroids will be calculated. Then the effective interval will be computed between cut and fill centroids. In this way, in a tight curve where fill is concentrated to the outside of the curve and cut is concentrated to the inside of the curve, fill will be increased and cut will be reduced. This also increases the accuracy of volume calculations.

The *Use Takeoff Strata* option uses the strata surfaces created in the Takeoff module to report the strata cut volumes both for the total strata volumes and the strata end areas per station. This method allows for unlimited strata definitions with advanced modeling techniques including Kriging and Inverse Distance to model strata surfaces. In Takeoff, the Drillhole/Strata Settings command is where you define the strata names and modeling methods. Next, the Place Drillhole command creates the drillholes. Once the drillholes are entered, use the Make Strata Surfaces command to build the strata surfaces which are stored as TIN files and associated with the current drawing.

The *Template ID for Profile* allows the profile grade to be applied to another template ID point other than the centerline. This feature might apply, for example, to a 2-lane road that will eventually be part of a 4-lane road being built in stages. The first-stage, 2-lane road would be fully symmetrical and designed around the crown of the road, but the template profile might be one of the edge of pavements. You can specify the template ID (e.g. EP), and

whether the left or right side ID should be used to apply the profile grade.

The *Shrink and Swell Factor* edit boxes allow you to specify a value that the volume calculated will be multiplied by. If you specify any number other than one an additional report showing accumulated adjusted volumes and differences will be produced.

The *Vertical Offset of Profile* edit box will place the template at the profile grade as raised or lowered by the entered offset.

The *Horizontal Offset of Road* will shift the template left or right on the centerline by the specified amount. Use a positive value to offset to the right and use a negative value to offset left. This option is useful, for example, when one side of a divided highway is built before the other side is to be started. In this case, you could define a normal template with a crown in the middle, but would enter a horizontal offset from the crown of the road to the actual centerline of the divided highway. In case the offset is variable instead of a fixed offset, there is an option to specify a centerline file (.CL) for the offset. This method applies for a divided highway where each side of the road follows its own alignment. In this case, the main centerline from the first dialog can be for the centerline along the median between the roads and this main centerline is used for the stationing in the reports and output sections. To process both sides of the divided highway, run Process Road Design twice, once for each side.

The *Slope Perpendicular To* option defines the slope projection method. The centerline method creates the template cut/fill slopes perpendicular to the centerline. The Slope Direction method accounts for the slope of the profile and makes the final surface to match the template cut/fill slope. For example, if the profile is at a 10% slope and the fill slope is at 2:1, then the Centerline method would create fill slopes that are 2:1 perpendicular to the centerline while slightly steeper (1.96:1) for the actual slope that goes in the slope direction with the effect of the profile. For the same case except with the Slope Direction method, the resulting slope perpendicular to the centerline is less steep (2.04:1) while the actual slope in the slope direction is exactly 2:1.

The *Skip First Grade* option skips the first template grade. This option applies when the road design is adding onto an existing road. In this case, the first grade in the template can cover to the edge where the existing road is being kept. Then Process Road Design will skip over the existing road and start the design from the second template grade.

The **Report and File Output Options** include settings for reporting final coordinates (if specified in the previous file output dialog), as well as special features.

To report tons and costs for the subgrade materials, set the material names in Design Template for the .TPL file. Also, in Define Materials Library in the Takeoff menu of the Construction menu, specify the density and cost parameters for the material names. Then the Process Road Design report will include the tons and costs for the subgrade materials.

The *Report Precision* controls the number of decimal places.

The *Use Report Formatter* option allows you to customize the fields to report and their order. It also can output the report to MS Excel or databases.

The *Report Subgrade Areas* option will include an additional line in the report for the end area of each subgrade material.

The *Report Centroids* toggle controls whether the shift in the cut or fill centroid radius shift will be included in the earthworks report.

The *Report Cut/Fill Text* option greatly expands the size of the report by presenting the cut and fill end areas at each station. A sample of the cut/fill text report is shown below. Volumes by end area method are presented between

each line containing station and end areas of cut, fill and optionally rock.

Station	Cut (sf)	Fill (sf)	Rock (sf)	Interval	Cut (cy)	Fill (cy)	Rock (cy)
3+00.00	0.00	101.07	0.00				
				50.00	313.78	93.58	0.00
3+50.00	338.88	0.00	0.00				
				6.09	80.93	0.00	0.00
3+56.09	379.10	0.00	0.03				
				43.91	824.60	0.00	31.84
4+00.00	634.92	0.00	39.12				

The *Report Cut/Fill Differences* option will report the cut/fill ratio and balance at each station.

The *Report Cumulative Cut/Fill Differences* option will report the running totals of cut/fill at each station.

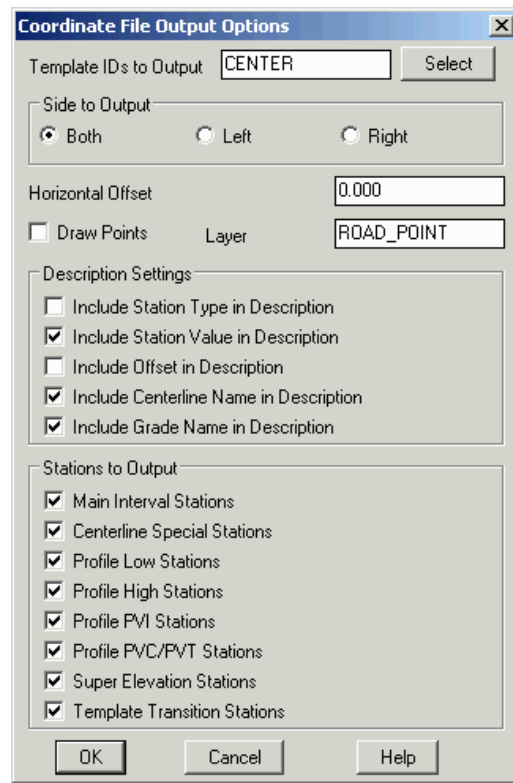
The *Report Final Station-Offset* option will create a report of the final section offset-elevation data in row-column format. The station and profile grade are shown on the left followed by columns of offset and elevation for each data point. There are options to report the surface points only, the subgrade points only or filter the points by ID.

Write SMI Chain File creates a chain (.CH) file that contains the centerline, profile and template data for SMI Construction V.

The *As-Built IDs to Use* option applies only if you have specified an as-built section file as one of the inputs in the previous dialog. Consider a normal road template with 20 feet to edge of pavement (EP) and 10 feet more to shoulder (SH). Going further, assume that when you run this template, it does a fill condition on the right and creates a TIE point. If you wanted to conform the template to match a wider section of road at certain stations, you could edit the output file of a normal run (using Section File Editor) and create new offsets and subgrade points for widening and even force a trapezoidal ditch in cut, as shown in the entries below:

	Offset	Elevation	Description	Ratio(:1)	Slope(%)
12	21.330	1996.950	EP	-50.00	-2.00
13	21.330	1995.920	SUBGRADE1-3		
14	31.500	1996.544	SH	-25.00	-4.00
15	21.330	1996.950	SUBGRADE1-4		
16	38.600	1992.994	BD	-2.00	-50.00
17	40.000	1992.994	BD2	Flat	0.0%
18	46.000	1995.994	TIE	2.00	50.00

Because all the other offsets to the left match by default, this editing will force the template to conform from offsets 21.33 right to the tie at 46 right. As you try different design template or other changes in *Process Road Design*, this as-built information would hold for the specified station. Alternately, you could edit the final cross section directly in *Section File Editor*. Note that you can use distinct, new ID points like BD2 which are not found in the template file, and they will be created if part of the as-built cross section file, and if referenced as *As-Built IDs to Use*. This As-Built method works best when inserting controlled section defined from TIE left to TIE right, which get inserted as completed sections in a run of Process Road Design.



The *Coordinate Output Options* apply when a Output Coordinate File is specified in the first dialog. The *Template IDs to Output* allows you to output any combination of template surface, subgrade, ditch and berm points. The *Side to Output* controls whether to create points on the left, right or both sides of the centerline. The *Horizontal Offset* shifts the points away from the centerline. The *Draw Points* option draws the points in addition to storing the points to the coordinate file. The *Description Settings* control the contents for the point descriptions. The *Stations to Output* control at which stations to create points. Here are example coordinates for station 0+90:

PtNo.	North (y)	East (x)	Elev (z)	Description
122	189497.42	611730.32	90.01	TIE 0+90.00L53.65
123	189461.43	611733.72	108.09	SHD 0+90.00L17.50
124	189457.45	611734.09	107.93	CURB3 0+90.00L13.50
125	189456.95	611734.14	107.93	CURB2 0+90.00L13.00
126	189456.95	611734.14	107.09	CURB1 0+90.00L13.00
127	189455.96	611734.23	107.09	EP 0+90.00L12.00
128	189444.01	611735.36	107.33	CENTER 0+90.00R0.00
129	189432.06	611736.49	107.09	EP 0+90.00R12.00
130	189431.07	611736.58	107.09	CURB1 0+90.00R13.00
131	189431.07	611736.58	107.93	CURB2 0+90.00R13.00
132	189430.57	611736.63	107.93	CURB3 0+90.00R13.50
133	189426.59	611737.00	108.09	SHD 0+90.00R17.50
134	189412.18	611738.36	100.85	TIE 0+90.00R31.97

The **Drawing Output Options** bottom section of the Additional Earthworks Parameters dialog contains output options which are only available when a centerline file is specified.

The *Triangulate & Contour* option will automatically run this command after Process Road Design is done to create the final contours. Triangulate & Contour uses the template 3D polylines to model the final surface, and the disturbed area polyline is used as the inclusion perimeter for the contours. With Triangulate & Contour clicked on, the *Setup* button becomes active. Picking Setup brings up the Triangulate & Contour settings including the contour interval and whether to draw 3D Faces. Also under Setup, there are controls for the colors of the 3D Faces for each template break point. With Triangulate & Contour active, Draw Template Polylines and Draw Disturbed Area Polyline are automatically turned on. The *Merge Road With Existing* option combines the road design triangulation with the existing ground surface and stores the resulting triangulation in the file specified with the Set button. This option is available when the Existing Surface is a triangulation file and the Triangulate & Contour option is active.

The Output Surface chooses between Triangulate & Contour building a surface for the top of the road template or the bottom subgrade.

The *Erase Previous Road Entities* option will erase any entities from the drawing that were created in a previous run of Process Road Design using the same design files. This option allows you to easily re-run Process Road Design and update the drawing entities after changing one of the road design files.

The *Draw Cross Section Polylines* option will create 3D polylines perpendicular to the centerline with each template break point. The interval of these cross section polylines is determined by the station interval of the Existing Sections.

The *Draw Template Polylines* option will create 3D polylines parallel to the centerline by connecting common template point IDs. For example, a template ID could be EP which this option would use to create 3D polylines for EP on the left and right of the centerline. Which template point IDs to connect in set under *Template IDs to Draw*. Setting this to an asterisk (*) will plot all the template break points. The *Select* button shows cross sections of the final templates for graphical selection of the ID's to draw.

Likewise, the *Draw Subgrade Polylines* option will create 3D polylines parallel to the centerline for the specified subgrade breakpoints.

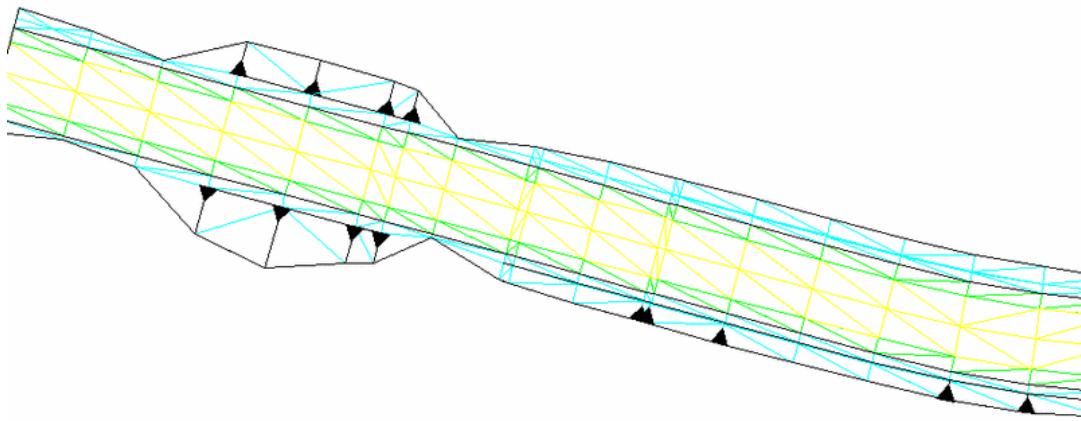
The *Draw Disturbed Area Polyline* option will create a polyline perimeter that represents where the cut/fill slopes tie into the existing ground.

Draw Template Slopes creates slope arrows parallel to the centerline at the specified template ID's. For example, this option can be used to show the slope direction and amounts along the template flowline. The style of the slope arrows is set under the *Set Slopes* button at the bottom of the dialog.

Draw Cross Section Slopes create slope arrows perpendicular to the centerline at the specified template ID's. For example, use this option to show the cross section slope of the pavement lanes. The cross section interval is controlled by the station interval under Process Options. The style of the slope arrows is set under the *Set Slopes* button at the bottom of the dialog.

Label Profile On Centerline creates labels in plan view for the profile stations, elevations and slopes as well as high and low points. This option has the same functionality as the command by the same name in the Profiles menu.

The *Draw Cut/Fill Direction Arrows* option will draw arrow indicators for cut or fill slope direction. The arrows are drawn in plan view and usually are drawn together with the Draw Disturbed Area and Draw Cross Section Plines options. Cut arrows start from the disturbed area limit and point towards the centerline. Fill arrows start from the base of the fill slope and point away from the centerline. The *Solid Cut Arrows* option chooses between solid fill or wire-frame cut arrows. These arrows, especially when drawn as solid cut arrows, help distinguish cut and fill at a glance, when in plan view. In the example below, fill from a berm is shown at the left and cut down to a ditch is shown at the right. The arrows will only draw if there is enough dimension in the cut and fill to fit the entire arrow. So the cut and fill arrows reveal the deeper cut and fill zones.



Prompts

Road Design Files dialog: Choose the design files

Additional Road Design Parameters

Road Design Report dialog

Trim existing contours inside disturbed area (Yes/<No>)? *Y* This prompt appears if Triangulate & Contour is on. This option will trim polylines with elevation that cross the disturbed area perimeter for the road.

Join final contours with existing (<Yes>/No)? *Y* This prompt appears if Triangulate & Contour is on. This option will join the final contours with the existing contours where they join at the disturbed area perimeter.

Portion of Earthworks Report:

```

Template File> C:\DATA\simo2.tpl
Profile File> C:\DATA\rd.pro
Existing Section File> C:\DATA\simo2.sct
Centerline File> C:\DATA\simo2.cl
  
```

```

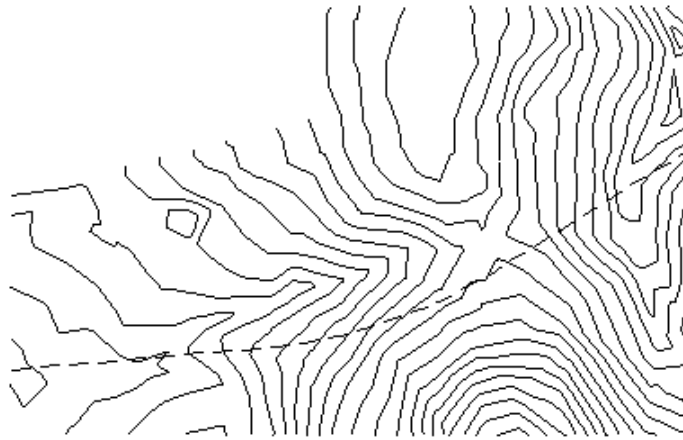
Processing 0+25.000 to 7+51.152
Total Cut : 800563.177 C.F., 29650.488 C.Y.
Total Fill: 1554948.266 C.F., 57590.677 C.Y.
  
```

Station	Cut(sf)	Fill(sf)	Interval	Cut(cy)	Fill(cy)
0+25.000	4407.456	0.000	25.000	4784.266	0.000
0+50.000	5926.559	0.000	25.000	5535.921	0.000
0+75.000	6031.029	0.000	25.000	4840.888	0.000
1+00.000	4425.290	0.000	25.000	3432.528	0.000
1+25.000	2988.971	0.000	25.000	2713.262	3.362
1+50.000	2871.676	7.262			

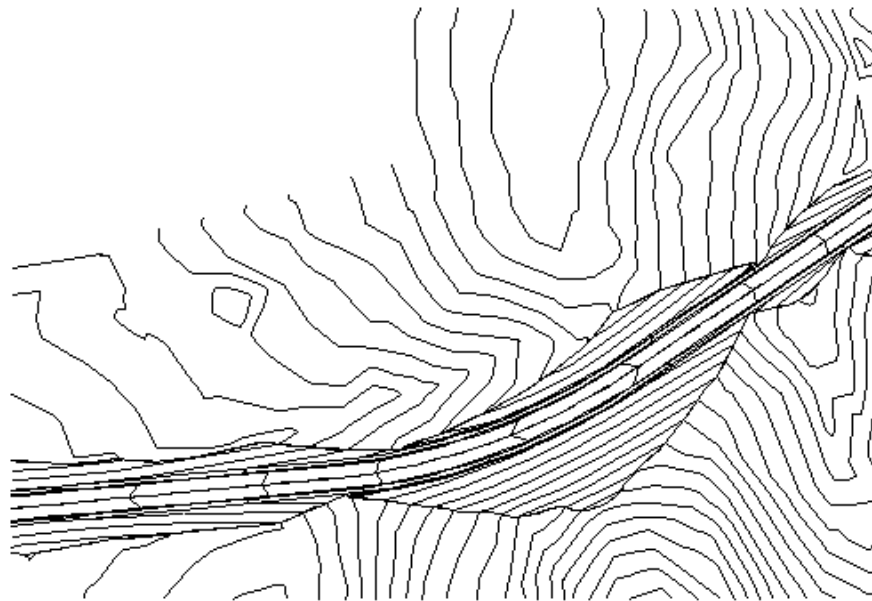
Portion of Final Station-Offset Report:

```

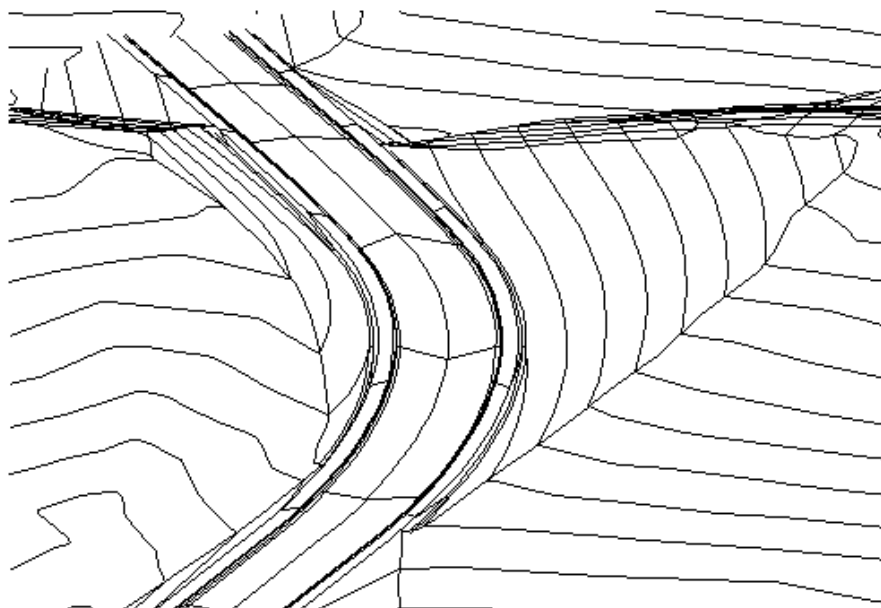
Final Surface Section Report
STATION      P.G.
2+50.000     1013.444 59.619 18.000 12.000 0.000 12.000
              992.634 1013.444 1013.204 1013.444 1013.204
2+75.000     1015.059 65.772 18.000 12.000 0.000 12.000
              991.173 1015.059 1014.819 1015.059 1014.819
3+00.000     1016.499 71.547 18.000 12.000 0.000 12.000
              989.725 1016.499 1016.259 1016.499 1016.259
3+25.000     1017.764 76.733 18.000 12.000 0.000 12.000
  
```



Existing Contours and Centerline



3D template polylines, disturbed area perimeter polyline and final contours



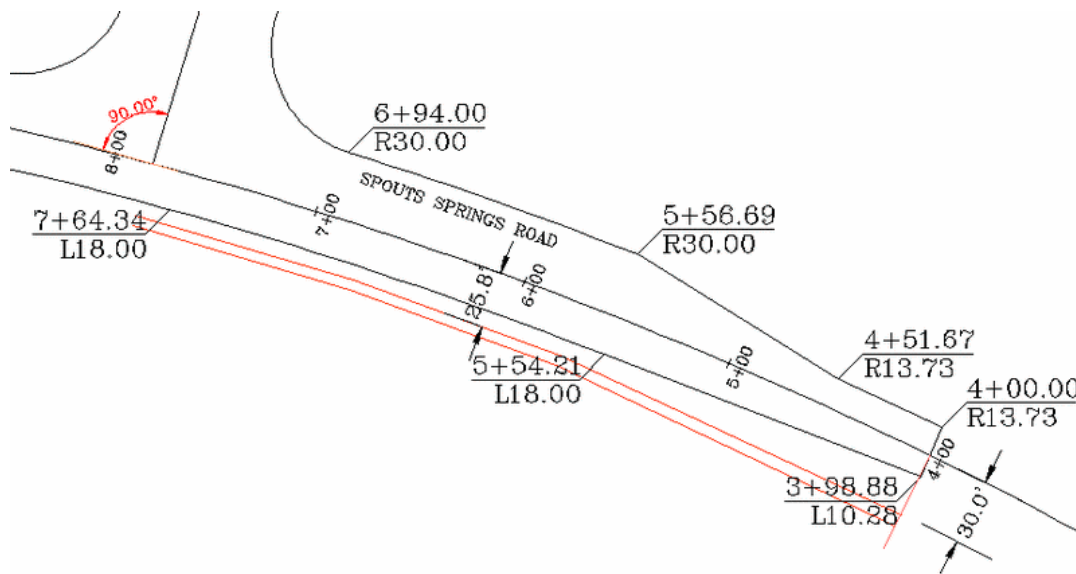
Review of 3 Methods of Transitioning Templates using Process Road Design

The 3 methods of template transitions and super elevation are:

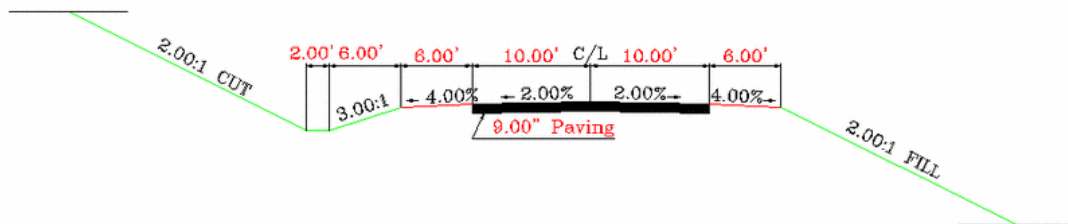
- (1) Template Transition and/or Super Elevation Files
- (2) Template Point Profile and Template Point Centerline files
- (3) Template Series file which transitions between multiple, named templates.

Road widening and lane transitions can be handled by all 3 methods. Special ditches are best handled by method (2), Template Point Profile and Template Point Centerline, especially since Template Transition files only work with lanes or portions of roads defined by the Grade button in Design Template. Template Transition files do not apply to cut and fill segments, unless they are designed as fixed features using the Grade button. Super elevation can often be handled by method (1) or method (3). Bear in mind that new lanes or template elements that emerge and then disappear need to exist as template ID points in all referenced templates, using all 3 methods. These template ID points can be set to 0.001 units from adjacent template points, then "told" to emerge and widen as new lanes with distinct slopes appear. The program will not transition templates that don't share common template ID points.

This deceptively easy looking example below might be approached by a combination of methods 1 and 2. For method 1 to apply (template transition), the slopes of the pavement lanes must be maintained according to the template definition from centerline to outside lane. The ditch portion will be handled by method 2 (template point centerline).

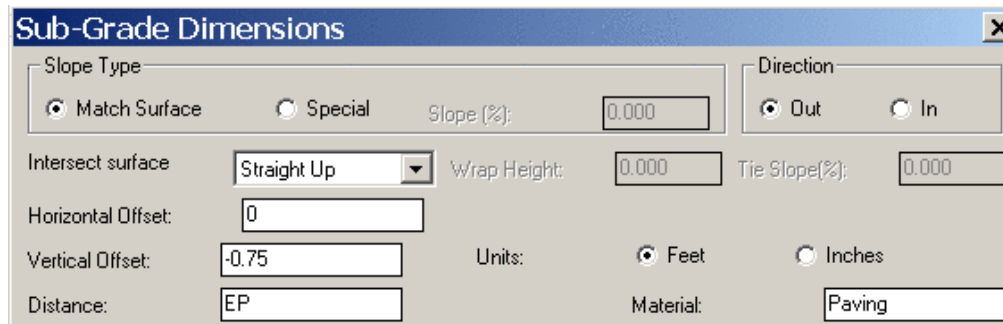


Assume Spouts Springs Road is a hillside road with a ditch cut on the left side and fill on the right side. The trapezoidal ditch is shown. We will design only from station 4+00 to station 6+94 where the intersection begins. The standard template of 10' left lane and 10' right lane might appear as shown below:

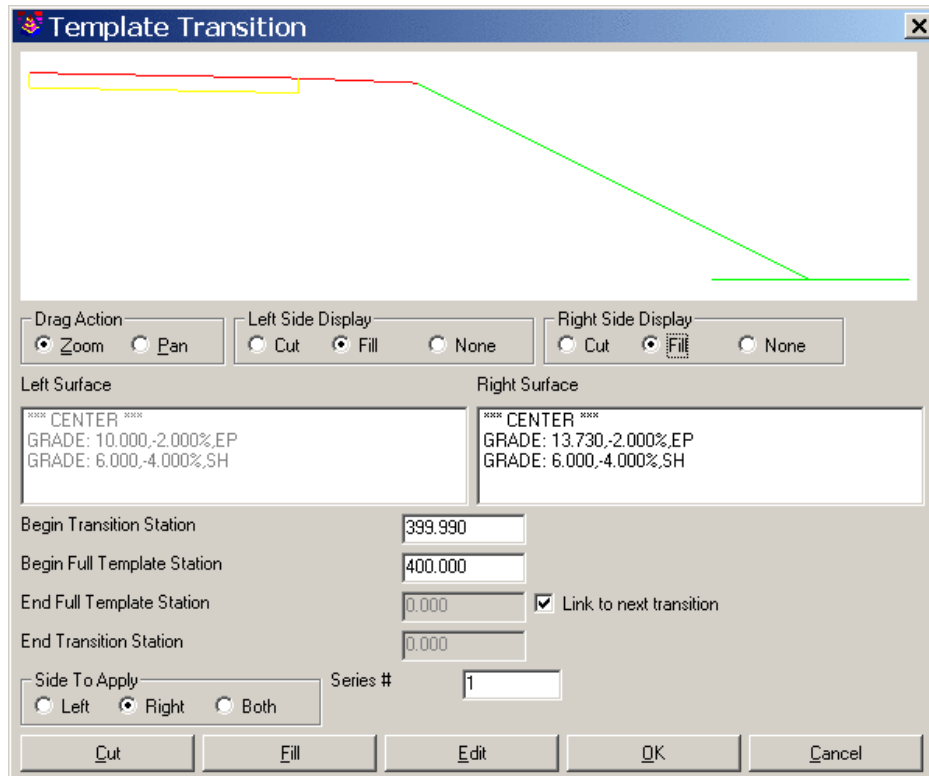


Note that if lanes are designed to expand, its important that the subgrade (9" of paving, shown above) be defined as following the ID, and should not be set to a fixed distance. The "EP" ID is used in the dialog below (top of subgrade

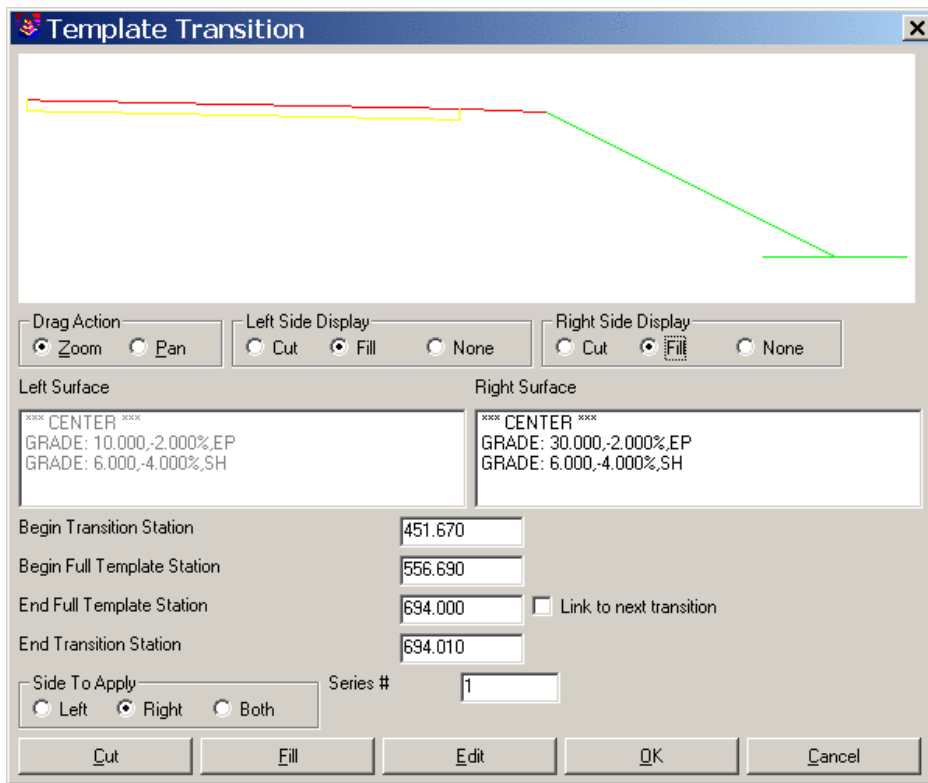
dialog within Design Template) for this example.



The right hand portion of this example would be entered as follows:

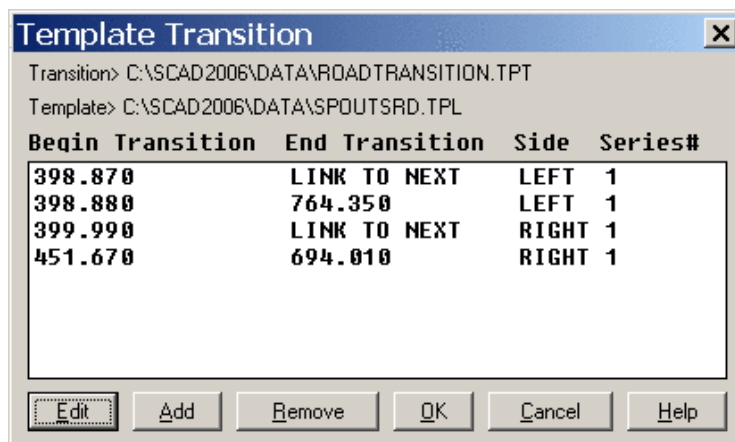


When you click "Add" within the Template Transition main dialog, you are presented with the above screen. Template transitions require that you specify the correct side of the road in the lower left, then click the Grade or lane to alter, which is the first lane on the right, which is set to 13.73 according to the plans. To make sure the lane is fully expanded from the standard 12 to the 13.73 at station 400, it is necessary to set the "Begin Transition Station" to something less than 400, as shown. Then if this "expanded" lane width does not transition back to standard 12 width, but changes again, you must click on "Link to next transition" and leave the "End Full Template" and "End Transition" stations blank. Then you click "Add" again for the final segment, which would be entered as shown:



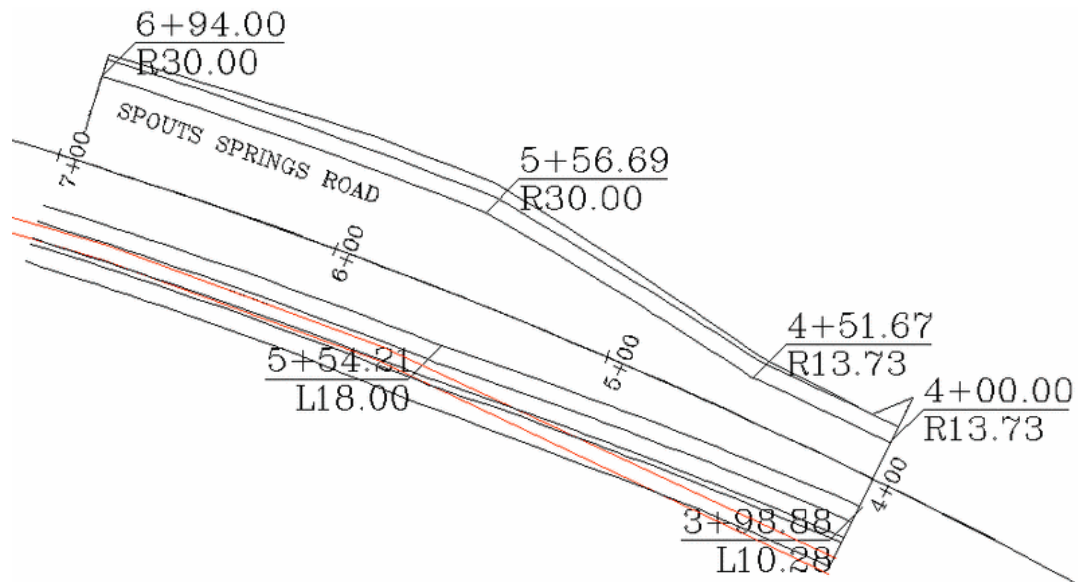
First, you specify "Side to Apply" as "Right", then click the pavement lane and edit it to 30', as shown above. Referencing the plan view drawing for Spouts Road shown above, you transition from station 451.67 to the new 30' road lane width at station 556.69 and hold that to the "End Full Template Station", which is 694.00. Then you can enter an "End Transition Station" just past the end of the key station range, which internally would transition the template back to a standard width of 12' at 694.01 (a moot point as the end of the project is station 694 for this exercise). The key to template transition is that it is designed to transition from normal to expanded or reduced dimension, then transition back to normal. It is ideal for use in passing lanes that appear and then transition back, but requires use of "Link to next transition" to handle a sequence of lane width changes as above. Therefore, where lane widths change often, and don't transition back to the normal template lane width, it is often best to use Template Point Centerline as the method of lane transitioning. We will apply that below to the ditch line.

When the template transition process is repeated for the left driving lane, you obtain a final Template Transition dialog as shown here:

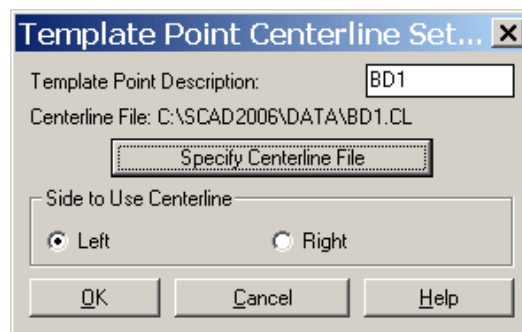


For the left side, the first screen just starts things up by establishing 10.28 as starting left side dimension, then the "Link to next transition" option is used, and the width of 18 is entered, transitioning to 18 at station 554.21 and holding that to an end station of 764.34, transitioning "back" to 12 at the fictitious 764.35, well beyond the 400 to 694 station range of interest. When this template transition file is run in Process Road Design and Triangulate &

Contour is turned on within Process Road Design, the output clearly shows that the lane transitions have followed the lane expansions correctly:

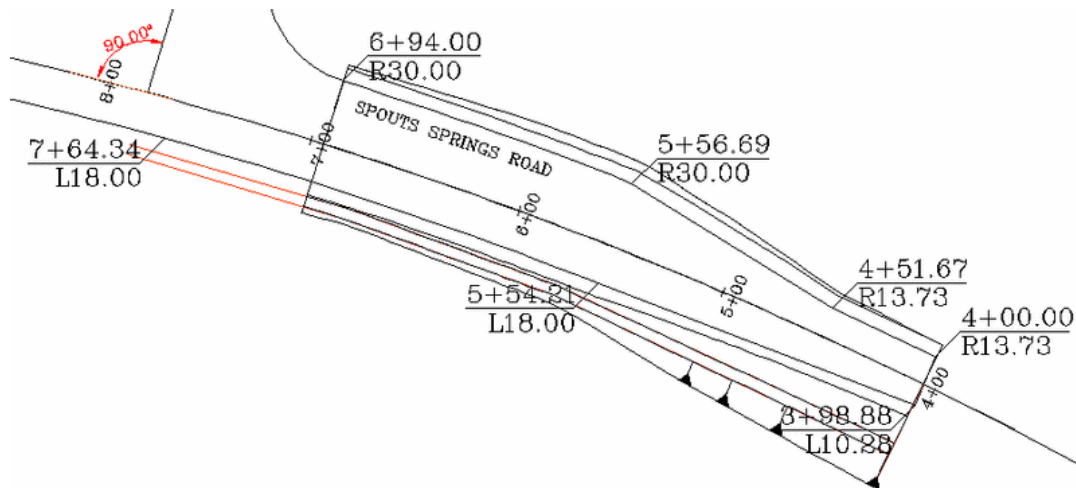


However, it is easy to see that the "design ditch" on the left side of the road, at 2' wide, did not conform to the special ditch which hugs the shoulder at station 7+00 but transitions to further off of the shoulder at 4+00. This special ditch is best handled with Template Point Centerline. To complete the special ditch design, use Polyline to Centerline File on both ditch polylines, calling the inside polyline BD1.CL and the outside polyline BD2.CL, as a reference to the ditch IDs, BD1 and BD2. You can give them a starting station of 0. The stationing of the ditch polyline does not matter, since only the coordinates of the centerline in the command Assign Template Point Centerline are used to determine the template ID position. Within Assign Template Point Centerline, Add each of the ditch sides as shown:

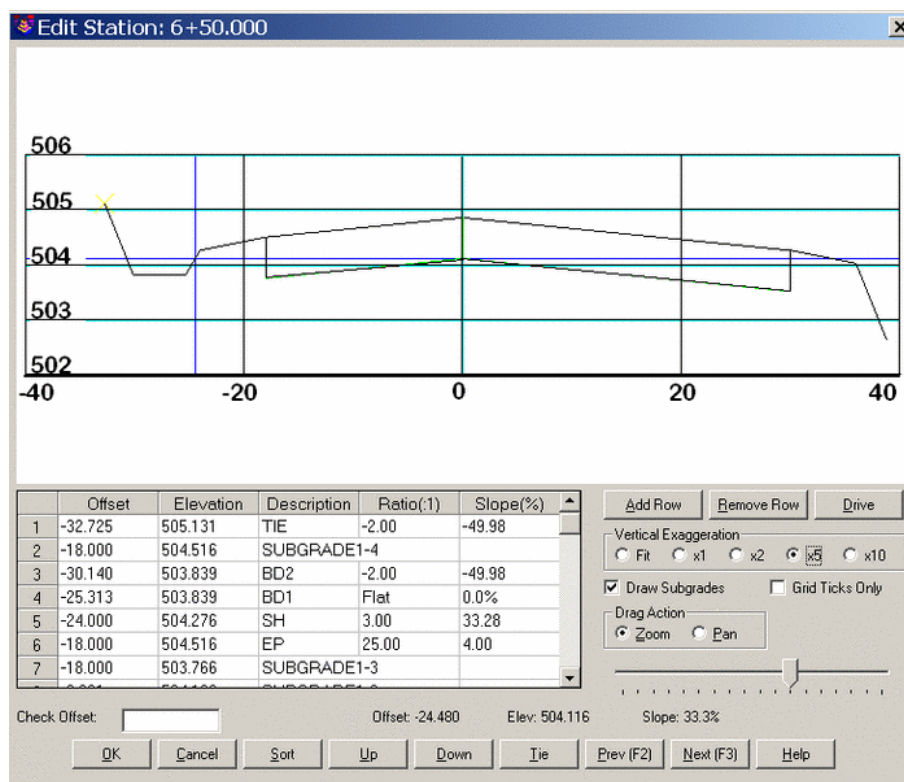


Note that if the ditch always exists on the left side, the ditch grades can be defined using the Grade button in Design Template, rather than using the Ditch feature within the Cut button. For final results, run the Process Road Design command using a combination of the Template Transition File and the Template Point Profile.

The end result is a final drawing that uses the Template Transition file to create the correct edge of pavement and uses the Template Point Centerline file to track along the correct ditch polylines. This is shown below in the final drawing of the 3D polylines generated by Process Road Design:

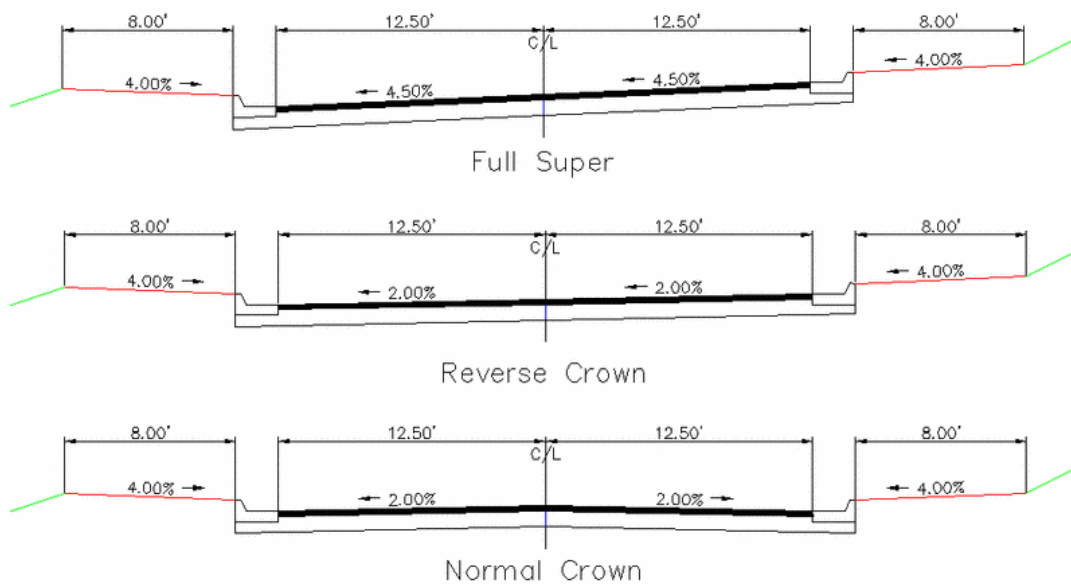


The actual slope to the ditch on the left is held at the design of 3:1, or whatever exists within the template from shoulder (SH) to base of ditch (BD1) in cut. Shown below in the Section File Editor screen editor is station 6+50, where the ditch is designed very close to the shoulder:

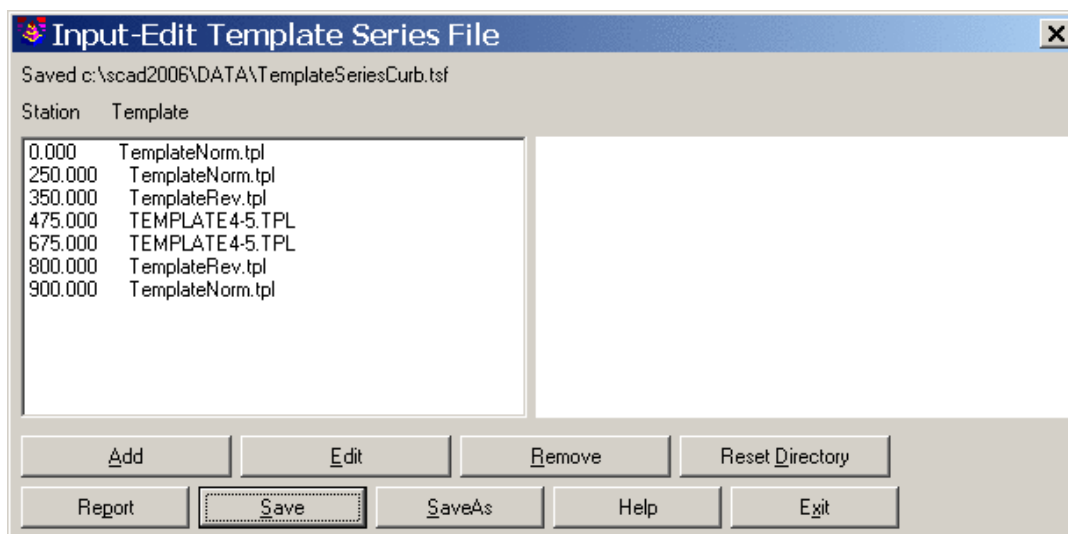


Note that the distance from BD1 to BD2 is irregular, based entirely on the plan view offset of the ditch polylines. Note also that BD1 to SH is 3:1, holding the defined slope. (The cursor position also can be used to verify slope of any portion of the section in "real-time".) Finally, note that the subgrade follows the widening and irregular position of the pavement lane EP for both left and right sides, since the subgrade offset from centerline was defined as EP.

Although superelevation can be handled by use of superelevation files, for most simple applications (2-lane roads in particular), a single curve with superelevation can be handled by a template series file, using only 3 templates: normal crown, reverse crown, full super. This is illustrated below, for a typical 2-lane road template:



The actual Template Series File will consist of 6 entries for one curve: Normal, Reverse, Begin Full Super, End Full Super, Reverse, Normal. You would only need to make one extra template, for simple roads, for every additional curve, for the full super condition, since normal and reverse crown remain the same. Note that the curbs, even on the high side, can be designed to slope downward and catch the shoulder drainage in Design Template by use of "special slope" of -1% in the curb design, or by entering a value for the added "Drop" across the gutter portion. Both methods create a downhill slope to the face of curb. So the above project might be designed as shown below in the Template Series Editor command:



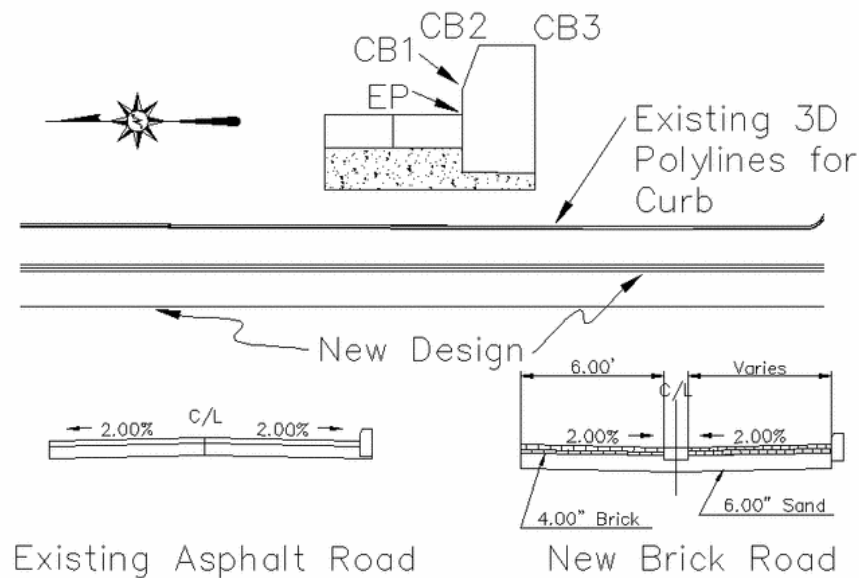
Note that beginning and ending stations are not necessary. If station 0.00 was omitted, Process Road Design would use the normal template in any case from station 0 to 250. Similarly, Process Road Design will use the normal template going forward from station 900 automatically.

Review of 2 Methods of Matching Portions of Existing Roads

There are two main techniques for tying new template designs into existing roads, which may apply to road expansions, urban re-paving, grade improvements and other renovation projects. As more and more roadwork involves road improvement rather than new road development, these techniques become more useful and critical to master. The two techniques are: (1) Use of Template Point Profile and Template Point Centerline files to match existing conditions on portions of roads that do not change, and (2) Use of the "As-Built" cross section feature as one of the input files. An advantage of the As-Built method is that you can insert section points with special IDs for special features, whereas the Template Point Profile and Template Point Centerline methods must follow template IDs that

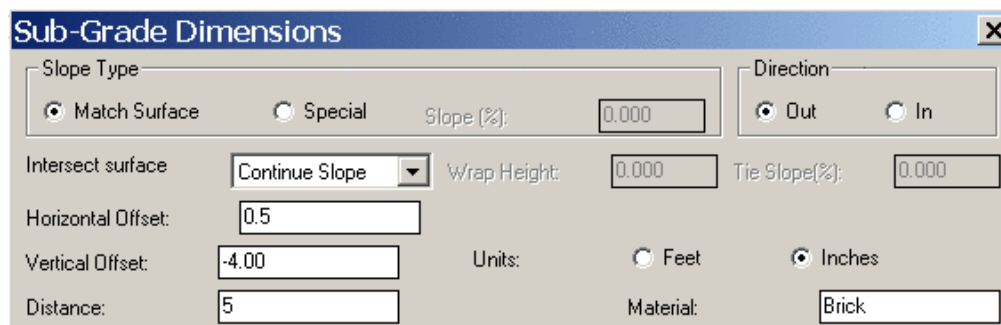
are found in the original, main template design file. But the Template Point or "string" method allows for calculating sections at any interval, while the As-Built section method will revise final sections only at stations found in the As-Built section file.

Consider this alley-way, which consists of a Belgian block style curb (no gutter) that is already in place. The plans are to remove a crowned asphalt alleyway and put in a bricked alleyway on sand, with a central, "depressed" rock drain of 1' width, to avoid water draining against buildings that abut the alley. But the design must match an existing "Belgian block" style curb on the right side of the road, which will not be removed.



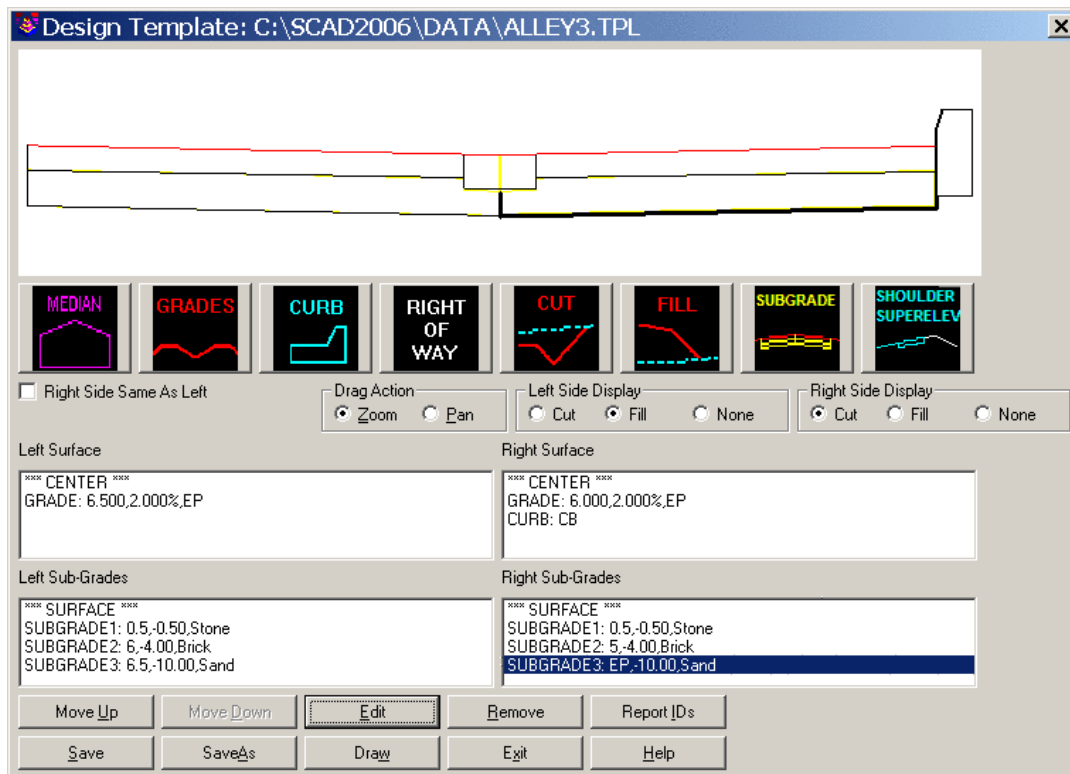
There is a new profile design involved, and a new template. However, the right side of the template will meet the exact grade and offset of the in-place curb, which has been surveyed as back of curb (CB3). Then the command Offset 3D Polyline was used to create the face of curb at EP=CB1, and to create the inside top of curb (CB2). Because of the symmetry and consistency of the curb, only the back of curb needed to be surveyed to hold the existing curb feature in place within Process Road Design. From that survey, the 3D Polyline for the EP is derived, which will be used for Template Point Centerline and Template Point Profile.

Features such as curbs and medians can be designed once within Design Template and then saved as curb or median files, then re-loaded and used in other templates, and applied to the left or right side of the template as desired. The central rock median of 1' total width can be constructed as two subgrades, one on the left side of 0.5' width and one on the right side of 0.5' width. The brick portion can be designed as a 4" thick subgrade as shown below. On the left side, you would need to use the "Straight Up" method of closing the subgrade surface. On the right side, you can use "Continue Slope". When using Continue Slope, it is best to underestimate the length needed to contact the next surface (the right curb), so continue can do an "extend" and find it. If you make the length too long (e.g. 6', which catches the curb which itself tilts back -2%), the program will not trim and will draw the subgrade to the back of the curb. Note that the vertical subgrade depth can be entered as 4 or -4. Both are accepted.

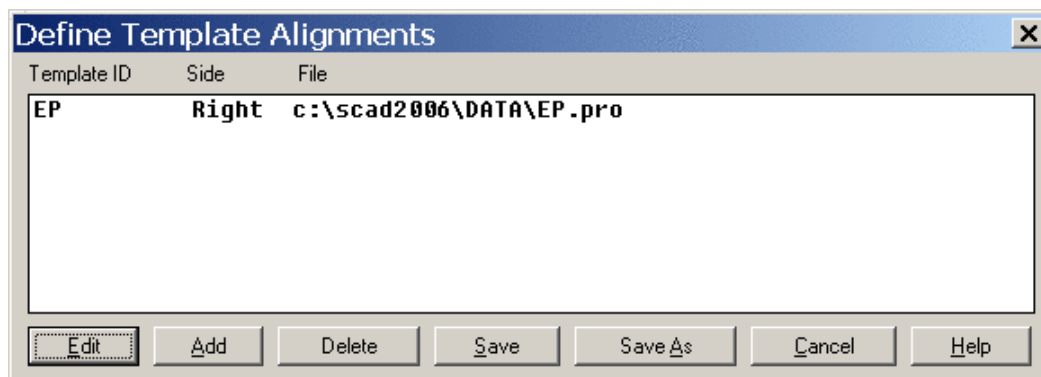


Be sure to define the sand subgrade on the right side (lowest subgrade) to have a distance of EP, a flexible distance

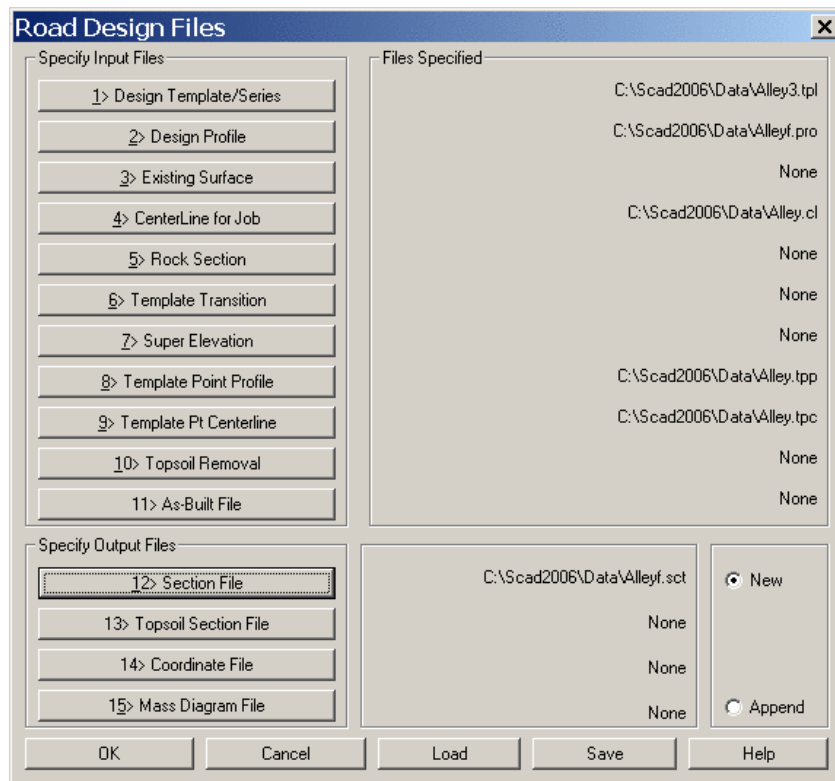
that follows the precise offset of the EP "ID", which will be assigned to follow the face of curb template point profile defined by CB1 above.



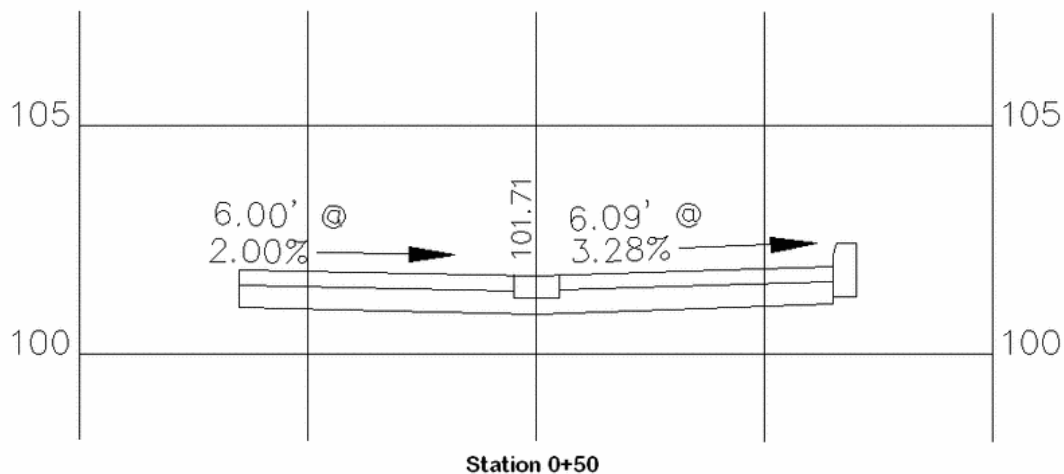
The next step is to set up the face of curb 3D polyline as a template point centerline and template point profile assigned to "EP". First you must do Polyline to Centerline File, pick the inner 3D polyline which is face of curb at proposed road level. Then you must do Profile from 3D Polyline and make a profile for the "EP". Then you assign this centerline and profile to the appropriate ID (EP) to force the curb to contact the correct curb position and elevation. The curb defined in the template matches the pattern of the in-place curb, so by setting EP to the correct template centerline and profile, the curb will "follow" at the correct position. The stationing used for the template point centerline is not critical to the calculation. However, the profile stationing much match and reference the centerline stationing. Therefore, when doing the command Profile from 3D Polyline, answer Yes to the question: "Station by another reference centerline [Yes/<No>]:". Making the Template Point Profile is always best accomplished by this method of Profile from 3D Polyline, referencing the design centerline. The Template Point Profile (and Template Point Centerline) would appear as shown here:



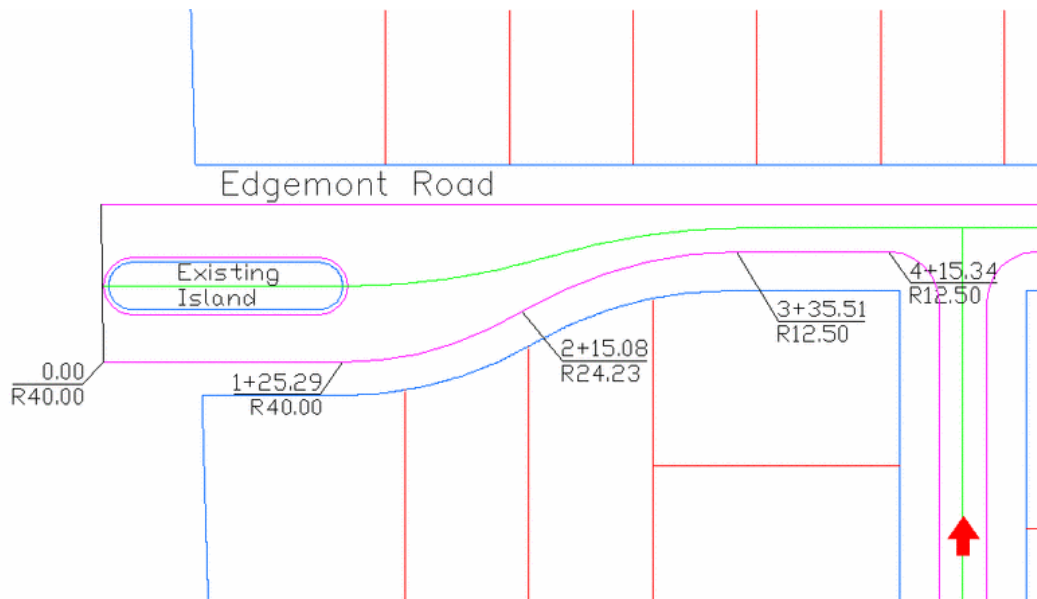
The files in Process Road Design would be set up as follows:



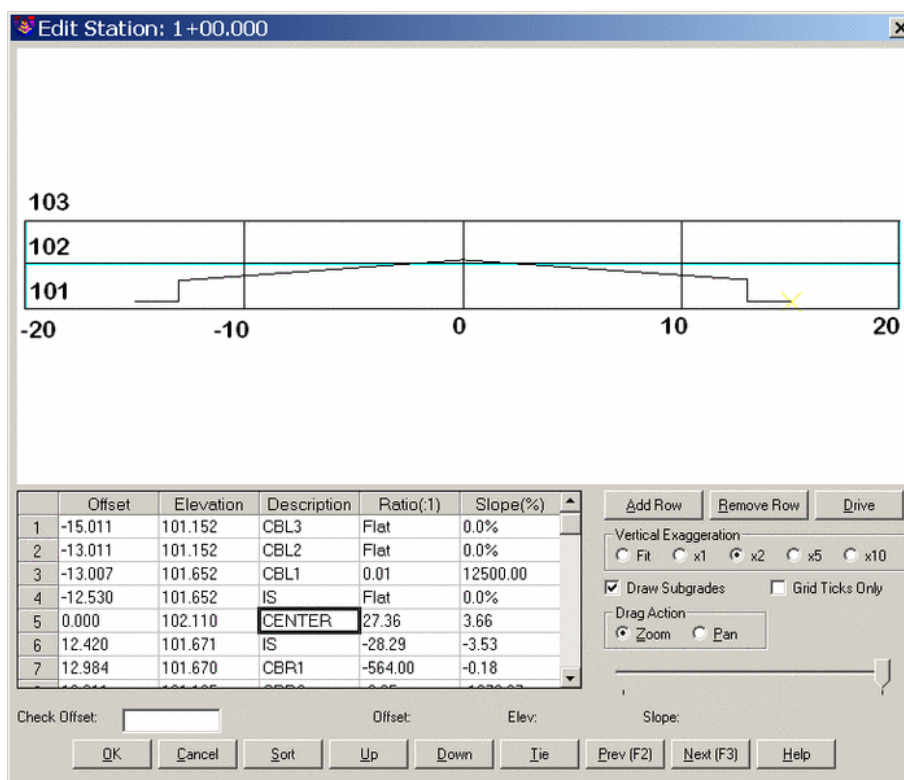
Note that no existing surface file is needed to compute final cross sections from as-built (straight wall on left of alley) to as-built (existing curb on right of alley). A final section is plotted below, showing the unique slope and lane distance determined by the as-built centerline and profile files that control the edge of pavement, and by extension, the curb, which continues with fixed dimensions from the edge of pavement.



A second method of doing as-built road design is to use the as-built cross section method. Whenever as-built cross sections are specified as part of the input files in Process Road Design, and then referenced for use on the Additional Road Design Parameters screen within Process Road, those offset IDs that are referenced will be held. Any matching IDs or new IDs found in the as-built cross sections will be substituted for the designed IDs within the final sections. In the example below, it might be proposed to redesign Edgemont Road from a roadside ditch road to one with a curb and gutter as well as sidewalks. However, the designer might want to keep the existing central median, already curb and gutter with plantings.

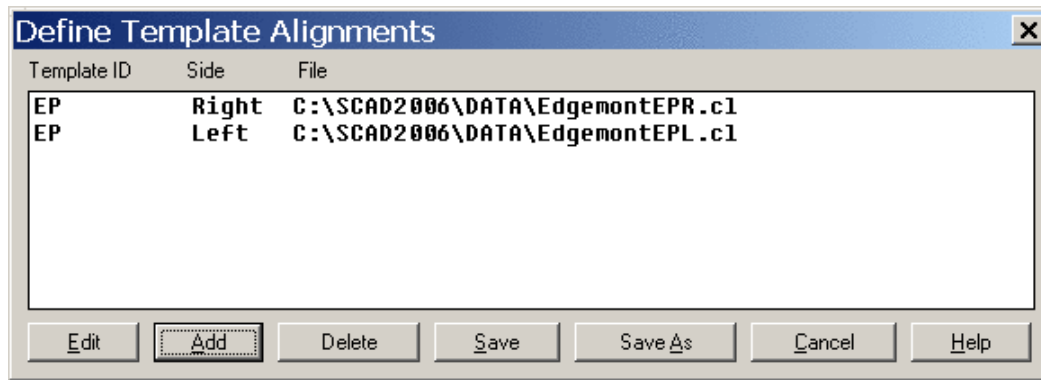


This example raises the challenging issue of inserting special interior points with new IDs into a set of design cross sections, through a length of about 125 feet of road. If a cross section of the island is taken through station 1+00, it might have the following ID points:

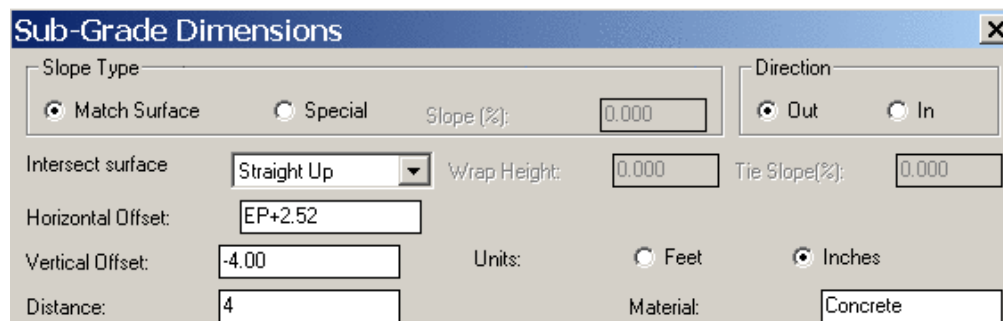


This cross section could then be part of an as-built cross section file (.SCT) which can be recorded at any desired station interval, the smaller the interval, the greater the accuracy. Now if the actual road template is defined as EP for edge of pavement and standard CB for curb, with CENTER for the centerline position, Process Road Design will substitute the As-Built File CENTER ID for the one calculated by the program, and will add in all the unique IDs from the cross section file, from -15.011 left to 15 right. Interestingly enough, this Edgemont Road example would also require a Template Point Centerline for the left and right edge of pavement, to pull the paving edge out to the expanded road dimension, which doesn't taper to normal until station 3+35.51. It would not require a Template Point Profile, so long as the road maintained a consistent design slope from centerline. When using Template Point Centerline, you need to turn the edge of pavement polylines into centerline files. Before doing so, test each polyline

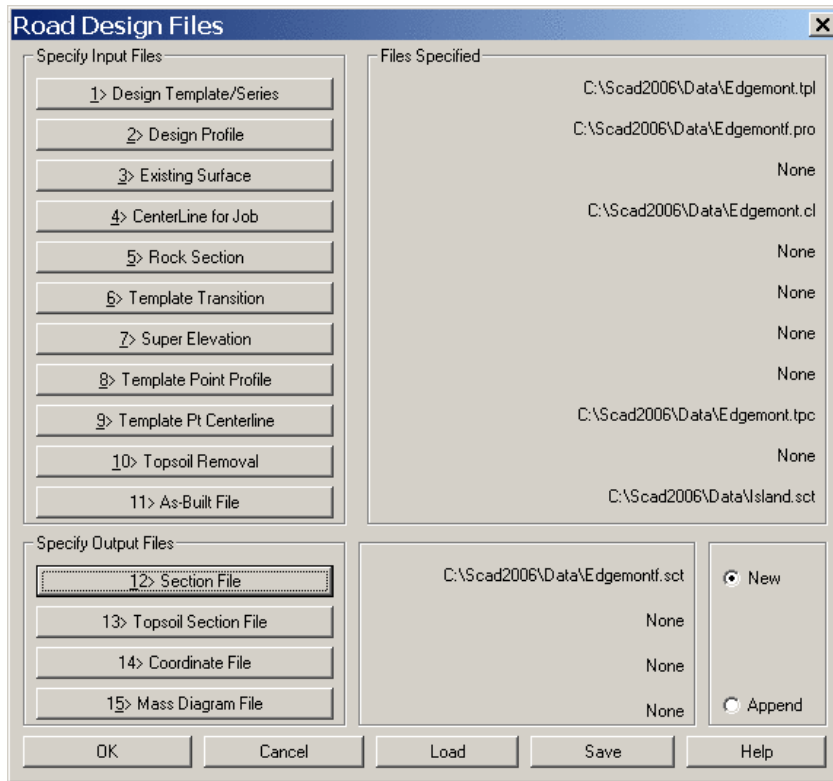
with the command Reverse Polyline (within Polyline Utilities under Edit) to verify that the polyline is drawn in the correct direction, as shown by the phantom arrows. The file Template Point Centerline elements might appear as shown:



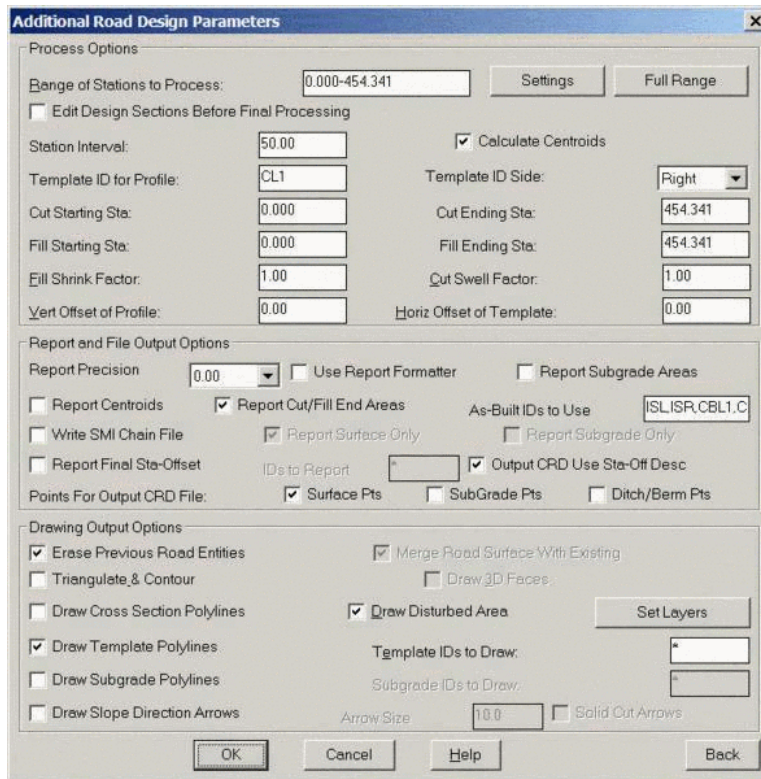
Be aware that a subgrade such as a concrete sidewalk, if it is to be placed behind the curb, must reference the curb or the edge of pavement ID for positioning, whenever the edge of pavement offset is changing based on use of a Template Point Centerline or As-Built cross section file containing duplicated IDs for edge of pavement. You can specify an offset for the sidewalk in the Subgrade option within Design Template, as shown below. The "2.52" offset was used to move past the tilting edge of the back-of-curb, which slightly exceeds 2.50.



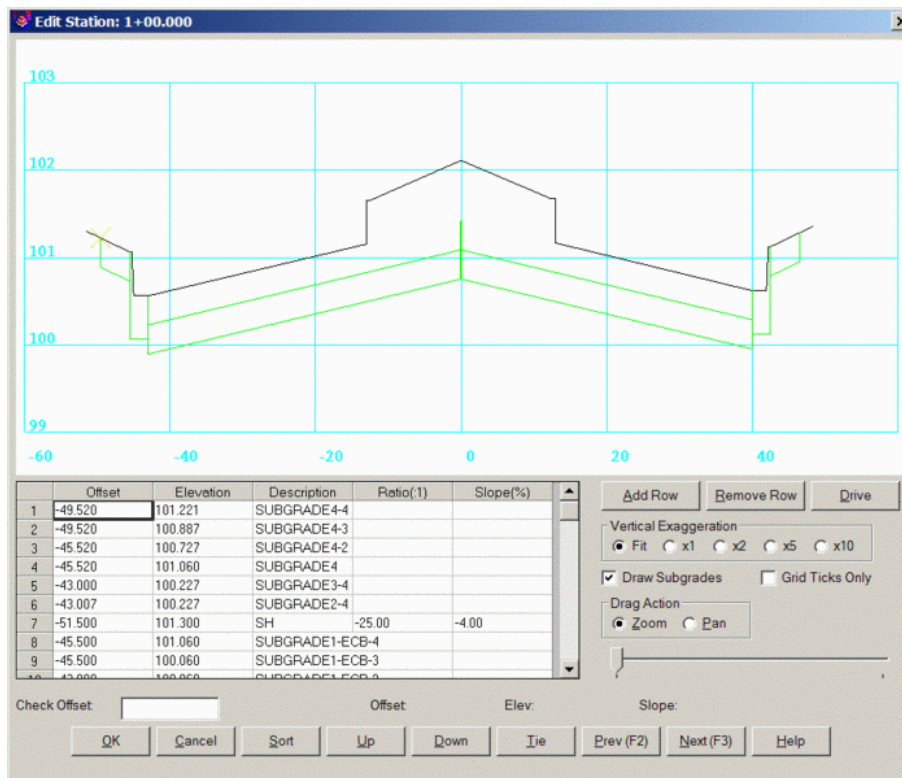
If the Island.sct file is the as-built cross sections, the entire input screen for the Edgemont Road project might appear as follows:



In the next dialog, fill in the descriptions for the section points in the As-Built IDs To Use field.

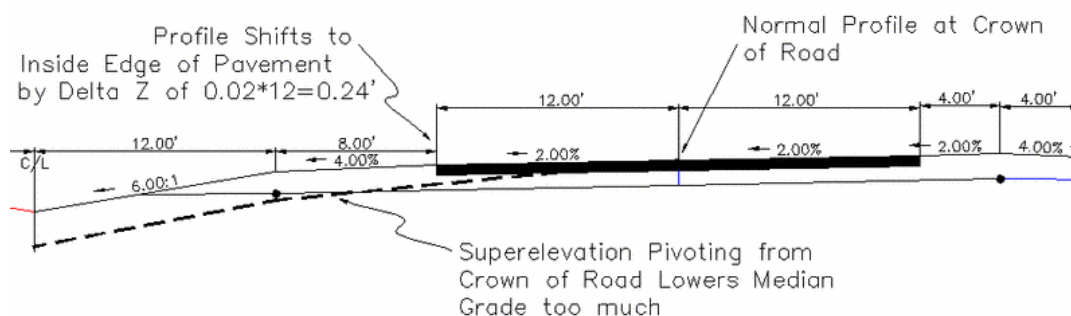


Here is the resulting output section file showing the combination of the design template with the as-built section points.

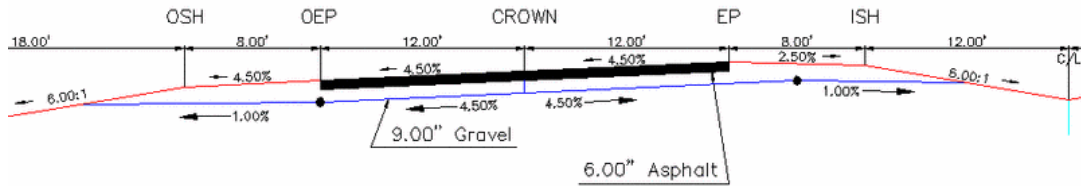


Example Divided Highway with Special Super Elevation Treatment

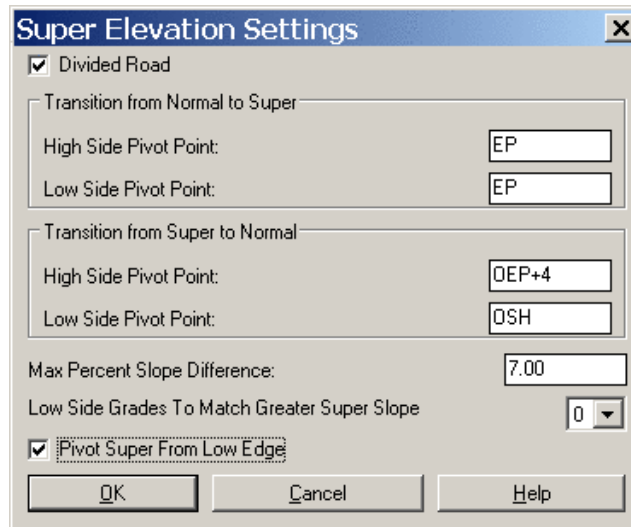
Divided highways such as 4-lane highways with a central depressed, grassy median are among the most challenging roads to define as templates, especially when accurate subgrade elevations and quantities are involved. Rules for superelevation and subgrade pivot points must be applied. And most divided highways do not use the centerline as the profile and require shifting the profile elevation to a specific template ID, like the inside edge of pavement or crown point for each side of the highway. This shifting occurs within Process Road Design. Furthermore, many highway departments have complicated rules for the profile grade. One such rule is that in superelevation, when the pivot lane reaches reverse crown, the profile moves from the crown of the road to the inside edge of pavement. Whatever the delta Z between the crown profile grade and inside edge of pavement profile grade is at reverse crown, this delta Z is subtracted from the profile grade and determines the profile of the inside edge of pavement from reverse crown through full super and back to reverse crown again. This typically improves drainage within the median portion, since a steep superelevation pivoting from the crown of the road can either reduce the median depth, or force the median too low. This is illustrated in the graphic below. Such challenging highways can be designed using special features within Design Template and Process Road Design.



The divided highway template itself can be quite complex. Let's review the requirements of our template below, first left side, then right side, in superelevation of 4.5%.



The main criteria for the design is that the pavement lanes are 12' wide, with 2% slope from the crown point in the middle (except in superelevation). On the interior high side of superelevation shown above, the grade breaks off at the EP or inside edge of pavement, and the maximum algebraic difference is 7%. So at 4.5% superelevation, the normal 4% downhill shoulder slopes instead at 7%-4.5%=2.5%, as shown. This part of the template behavior is controlled by the Superelevation Shoulder button within Design Template, with entries as shown here:



Note that the Super Elevation Settings dialog treats the "interior" of the road in the upper part, and the exterior of the entire road (like a 2-lane road) in the lower part. So the "Low Side Pivot Point" under the lower "Transition from Super to Normal" is where, walking from the middle of the road towards the left, super ends and normal slopes resume. That is set to OSH, or the outside shoulder position, the goal being to slope the full shoulder with the superelevation on the lower outside shoulder lane, then resume normal (non-super) slope at the 6:1 "recovery zone" slope. The entry of OSH as Low Side Pivot Point for Super to Normal controls that. In the upper part of the dialog, the inside "Transition from Normal to Super" sets the Low Side Pivot Point at EP. So at EP, walking from the template center left towards the left side of the road, normal ends at EP and superelevation begins. So the median upslope of 6:1 is normal, as is the shoulder, the super starts at EP. But because the 7% maximum percent slope difference is active, the shoulder can't remain at 4% but goes to 2.5% leading to the 4.5% superelevation. When super subsides to 3% or less, the shoulder would be normal at 4% as specified in the template design in this case.

Referring to the graphic above showing the left side of the divided highway, the gravel for the shoulder is shown running out to "daylight" on the outside recovery zone and on the inside median slope. However, to reduce quantities of stone, the stone runs at a uniform slope of -2% in normal crown, or matches superelevation, but pivots to 1% downhill at the outside OEP and 4' past the inside EP. This is accomplished through the subgrade entry dialog. First, the outside subgrade:

Note that the normal slope of the stone subgrade does not follow the surface but stays at the "special" slope of -2%, matching the surface always only beneath the asphalt portion within the pavement zone. For divided highways, it is always necessary to do at least 2 subgrades for each material: one from the crown or middle of the road "out" to the outslope (as above), and one from the crown or middle of the paved portion in to the interior. Since the crown of the road on each side of the highway is 32 feet left of the center depressed median position, the horizontal offset for the "out" position is 32. Enter the vertical offset as the entire distance from the horizontal offset down to subgrade bottom. In this way, any other thinner subgrades above are deducted from total subgrade quantities of the grade under consideration. If the goal is to "force" a -1% slope in both normal crown and superelevation, then set the Max Slope After Pivot(%) to -1%, and click "Special". Then set both Standard Slope and Minimum Slope Percent to -1%. This ensure that -1% will be used at the pivot offset of OEP, or as specified. Apply this to both subgrades ("in" and "out" from horizontal offset 32). If you simply entered -1% for the Max Slope After Pivot(%) and clicked Normal, slopes on the low side would break over to -1% but slopes on the higher side of each superelevation lane (beneath inside shoulder on the left, outside shoulder on the right) would continue on at the super slope and not break off. You must use the "Special" setting. The low side shoulder for the inside portion of the left side of the road is specified by the "In" subgrade, in this dialog:

Sub-Grade Dimensions

Slope Type: Match Surface Special Slope (%): Direction: Out In

Intersect surface: Wrap Height: Tie Slope(%):

Horizontal Offset: Vertical Offset: Units: Feet Inches

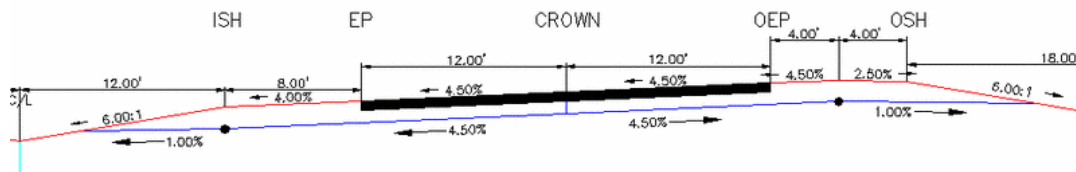
Distance: Material:

Super Elevation Settings

Low Side	High Side
Pivot Offset: <input type="text" value="ISH+4"/>	Pivot Offset: <input type="text" value="ISH"/>
Max Slope After Pivot (%): <input type="text" value="-1.000"/>	Max Slope After Pivot (%): <input type="text" value="-1.000"/>
Slope Type After Pivot: <input type="radio"/> Normal <input checked="" type="radio"/> Special	Slope Type After Pivot: <input type="radio"/> Normal <input checked="" type="radio"/> Special
Standard Slope Percent: <input type="text" value="-1.000"/>	Standard Slope Percent: <input type="text" value="-1.000"/>
Minimum Slope Percent: <input type="text" value="-1.000"/>	Minimum Slope Percent: <input type="text" value="-1.000"/>

OK Cancel Help

The pivot point for the subgrade on the inside left of the template is ISH+4, or 4 feet from inside shoulder to inside edge of pavement, the +4 being the direction walking out from the middle of the template in all cases. The right side of the template is shown next:



On the right side, the high-side subgrade pivot in the "out" direction, walking from the middle of the road outward, is OEP+4. On the right side, the high-side subgrade pivot in the "in" direction is simply ISH, as shown. So the controls exist to specify critical break points on subgrade and surface grades using Design Template. Whether this is the best design can be debated, but the controls are there to create surface and subgrade slope breaks and grade changes.

Referring to the Super Elevation Settings dialog above, the key to setting the superelevation of the divided highway to the inside edge of pavement at reverse crown (minus the 0.24 delta Z from profile grade to inside edge of pavement grade) is to click on the option, "Pivot Super From Low Edge".

Now you must run Process Road Design, using this template, to produce verifiable final cross sections. Set the Process Road "Additional Parameters" dialog such that "Crown" (or whatever ID is used for the center crown point on each side of the road) controls the profile grade.

Additional Road Design Parameters

Process Options

Range of Stations to Process: 0.000-1850.000

Station Interval: 50.00 Calculate Centroids

Template ID for Profile: CROWN Template ID Side: Left

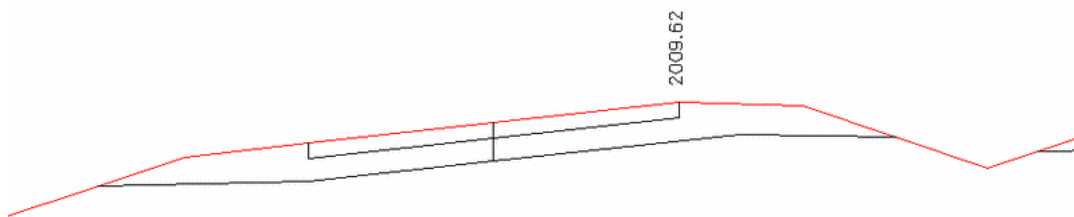
Cut Starting Sta: 0.000 Cut Ending Sta: 1850.000

Fill Starting Sta: 0.000 Fill Ending Sta: 1850.000

Fill Shrink Factor: 1.00 Cut Swell Factor: 1.00

Vert Offset of Profile: 0.00 Horiz Offset of Template: 0.00

The final sections that are produced will shift the profile grade to the inside edge of pavement from reverse crown to reverse crown through superelevation, adjusted -0.24'. A final section is shown plotted below as drawn using Draw Section File:



Pulldown Menu Location: Roads

Keyboard Command: eworks

Prerequisite: Profile file and template file

Road Network

This command synthesizes road network design for subdivisions and commercial and industrial sites by enabling interactive 3D design of all road centerlines, profiles and templates, including cul-de-sacs. A docked dialog on the left of the screen identifying the existing DTM surface and all road files combines with an active CAD screen and command line. You can save drawings and run virtually any standard Autocad command while within the docked dialog. Once the user identifies all centerlines involved, the program detects intersections and end segments suitable for cul-de-sacs, and through user input of design parameters for cul-de-sac dimensions and intersection transitions, the program will process the complete 3D design, with output options including cross sections, 3D faces, TIN files and contours. The many roading files involved in a road network design are all saved to an "RDN" file that can be recalled, modified and re-processed.

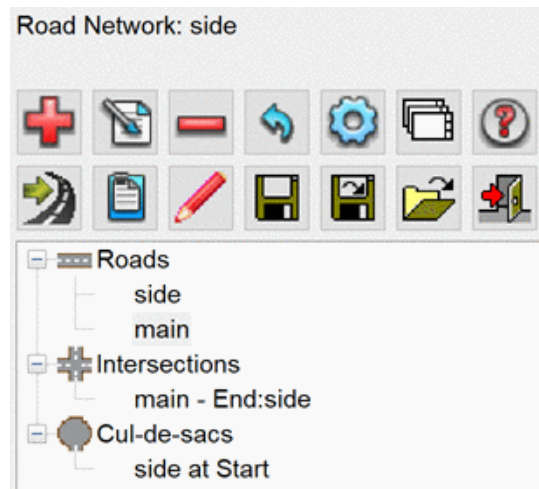
This **Road Network Help** document is divided into 7 parts: *Road Network Task Pane*, *Road Network Settings*, *Adding and Editing Roads*, *Road Network Road Profile Editor*, *Adding and Editing Intersections*, *Adding and Editing Cul de Sacs*, *Road Network Workflow Example #1* and *Road network Workflow Example #2*

When designing roads using Carlson's **Road Network** feature, all work is done through a **Task Pane** that docks along the left side of the drawing screen. Having the **Task Pane** open and active does not prohibit or interfere with normal Command: line or other CAD functionality.

All settings and files associated with a roadway design project are saved in the Road Network (.RDN) file. Upon starting the **Road Network** command, the user is prompted to open an existing or create a new Road Network

(.RDN) file in which to save the project data. Once loaded, the active Road Network filename is displayed at the top of the **Task Pane**.

Once Roads, Intersections and Cul-de-Sacs have been added to the Road Network, selecting any one of them in the project tree highlights the feature and centers it in the drawing screen. Highlighting and centering options may be changed in the **Display Options** tab of the **Road Network Settings** dialog box.



Road Network Task Pane

This area of the project tree lists the Roads defined as part of the Road Network. See **Road Network: Adding and Editing Roads** for additional assistance. The functions are accessed by right-click on the tree or by the icon buttons at the top of the dialog

Add: Pick this button to **Add** a Road to the Network. After adding the Road, the **Edit Road** dialog box is displayed allowing the user to manage and make changes to the **Input Files** and **Output Files** for the selected Road.

Edit: Pick this button to display the **Edit Road** dialog box to manage and make changes to the **Input Files** and **Output Files** for the selected Road.

Remove: Pick this button to delete the selected Road from the Road Network. After Removing the Road from the Network the design files associated with that Road will remain in the project folder.

This area of the project tree lists the Intersections within the Road Network. Intersections are created automatically as intersecting Roads are added to the Network. See **Road Network: Adding and Editing Intersections** for additional assistance.

Edit: Use this button to display the **Edit Intersection** dialog box and make changes to the **Input Data** and **Output Files** for the selected Intersection. Other changes that can be made to the Intersection design are:

- 1) Changing the Primary/Secondary status of the Roads creating the Intersection,
- 2) Making design changes that apply to the entire Intersection,
- 3) Making design changes that apply to one or more Corners of the Intersection.

Reset: Use this button to overwrite all design changes made to the selected Intersection and reset to the original Intersection design.

This area of the project tree lists the Cul-de-Sacs defined as part of the Road Network. See **Road Network: Adding and Editing Cul-de-Sacs** for additional assistance.

Add: Picking this button will display a list of Roads in the Network and prompt the user to "**Select Road**

for **Cul-de-Sac**".... After selecting the Road, the **Edit Cul-de-Sac** dialog box is displayed allowing the user to specify the **Input Data** and **Output Files** for the Cul-de-Sac.

Edit: Use this button to display the **Edit Cul-de-Sac** dialog box and make changes to the **input data** and **output files** for the selected Cul-de-Sac.

Remove: Use this button to **Remove** the selected Cul-de-Sac from the Road.



Process: Use this button to manually trigger the computation process for the Road Network and perform the tasks configured in the **Output Options** tab of the **Road Network Settings** dialog box.



Report: Use this button to Save or Print one of two Reports provided by the **Road Network** feature which are: the **Output Processing** report and the **Input Data Files** report. Default Report settings can be changed in the **Report Options** tab of the **Road Network Settings** dialog box.

The **Output Processing** Report displays the cut/fill and material quantities for each Road, Intersection and Cul-de-Sac of the Road Network.

```
Carlson Edit: C:\Users\Travis Maxwell\AppData\Roaming\Carlson Software\Carlson2023\CAD10_1_X64\USER\scadpr...
File Edit Settings

Process Road Network Mon May 1 17:01:11 2023

Road Network File: D:\Carlson\EXAMPLES\ROADNET.rdn

Total Cut : 1952.5 C.Y.
Total Fill: 708.6 C.Y.
Total Curb: 52.1 C.Y., 1505.9 L.F.
Total Subgrade1 - Asphalt_1: 79.1 C.Y., 12809.4 S.F.
Total Subgrade2 - BINDER: 79.1 C.Y., 12809.4 S.F.
Total Subgrade3 - CABG: 266.7 C.Y., 15061.2 S.F.
Total Subgrade4 - SIDEWALK: 45.5 C.Y., 8445.9 S.F.
Total Intersection Subgrade1 - Asphalt_1: 8.4 C.Y.
Total Intersection Subgrade2 - BINDER: 8.4 C.Y.
Total Intersection Subgrade3 - CABG: 25.2 C.Y.
Cut to Fill Ratio: 2.76

Road CL1
Template File> D:\Carlson\EXAMPLES\CARLSON.TPL
Profile File> D:\Carlson\EXAMPLES\CLI.pro
Existing Surface File> D:\Carlson\EXAMPLES\GOOGLE.tin
Centerline File> D:\Carlson\EXAMPLES\CLI.cl
Road Design Parameter File> C:\Users\Travis Maxwell\Desktop\tmm-ed\WEBINAR\2022-07-07 lotnet\exar

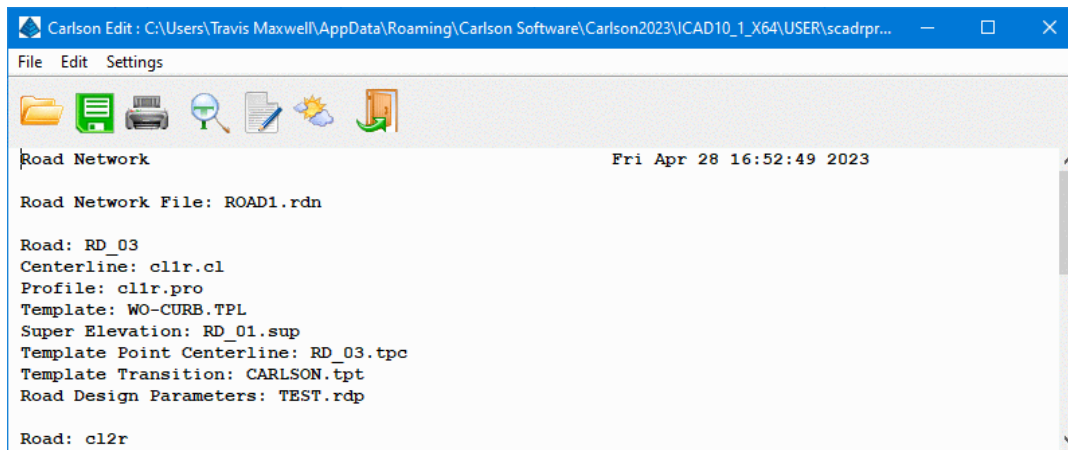
Processing 0+00.000 to 1+34.351
Cut Swell Factor: 1.000
Fill Shrink Factor: 0.900
Total Cut : 188.3 C.F., 6.973 C.Y.
Total Fill: 11,015.5 C.F., 407.980 C.Y.
Cut to Fill Ratio: 0.02
Total Left Curb: 125.5 C.F., 4.648 C.Y., 134.351 L.F.
Total Right Curb: 125.5 C.F., 4.648 C.Y., 134.351 L.F.
Total Left Subgrade1 - Asphalt_1: 201.5 C.F., 7.463 C.Y., 1,209.0 S.F., 134.336 S.Y.
Total Left Subgrade2 - BINDER: 201.5 C.F., 7.462 C.Y., 1,209.0 S.F., 134.336 S.Y.
Total Left Subgrade3 - CABG: 675.0 C.F., 25.001 C.Y., 1,410.6 S.F., 156.728 S.Y.
Total Left Subgrade4 - SIDEWALK: 111.9 C.F., 4.145 C.Y., 671.8 S.F., 74.639 S.Y.
Total Right Subgrade1 - Asphalt_1: 201.5 C.F., 7.463 C.Y., 1,209.0 S.F., 134.336 S.Y.
Total Right Subgrade2 - BINDER: 201.5 C.F., 7.464 C.Y., 1,209.0 S.F., 134.336 S.Y.
Total Right Subgrade3 - CABG: 675.1 C.F., 25.004 C.Y., 1,410.6 S.F., 156.728 S.Y.
Total Right Subgrade4 - SIDEWALK: 111.9 C.F., 4.146 C.Y., 671.8 S.F., 74.639 S.Y.

Processing 2+03.351 to 3+21.551
```

;">

Road Network Output Processing Report

The **Input Data Files** Report displays all of the user-specified design files associated with the Road Network. The user has the option of reporting only the filename or both the path and filename.



Road Network Input Data Files Report



Settings: This button displays the **Road Network Settings** dialog box which is the starting place for all projects designed using the **Road Network** feature. There are 5 tabs in the dialog box: **Process Options**, **Output Options**, **Report Options**, **Display Options** and **Transition Defaults**.



Draw: This function draws the profile or cross section for the selected road. First highlight the road name from the project tree and then pick this button. The program prompts whether to draw the design profile or output section. The draw functions call the Draw Profile and Draw Section File commands.

Save: Pick this button to **Save** the Road Network (.RDN) file.

SaveAs: Pick this button to Save the current Road Network (.RDN) file and give it a new path and/or filename.

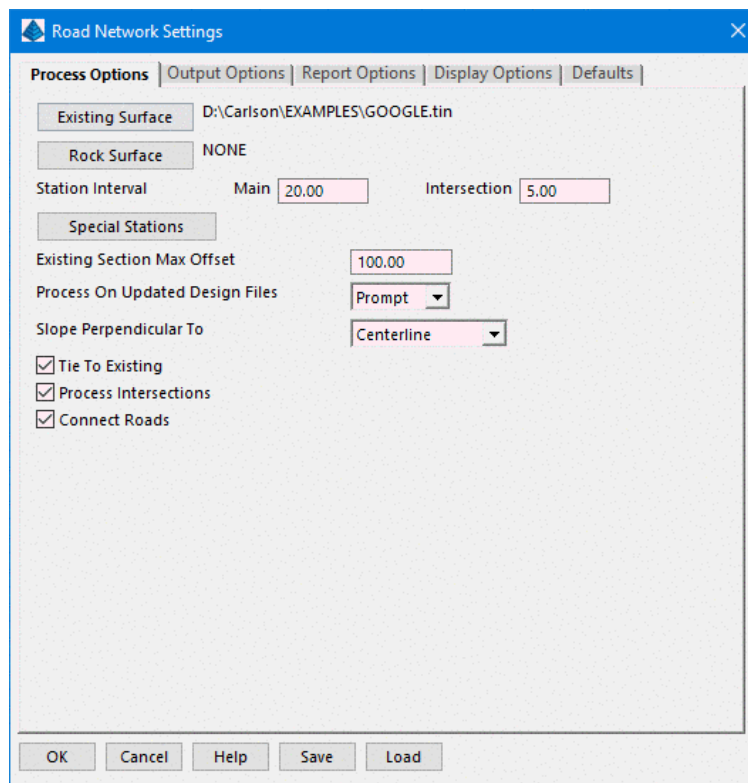
Load: Pick this button to **Load** an existing (.RDN) file.

New: Pick this button to start a **New** Road Network (.RDN) file.

Exit: Pick this button to **Exit** the Road Network command and close the **Task Pane**.

The **Road Network Settings** dialog box is accessible from the **Settings** button on the **Road Network: Task Pane**.

The **Road Network Settings** dialog box is the starting place for all projects designed using the **Road Network** feature. There are 5 tabs in the dialog box: **Process Options**, **Output Options**, **Report Options**, **Display Options** and **Transition Defaults**.

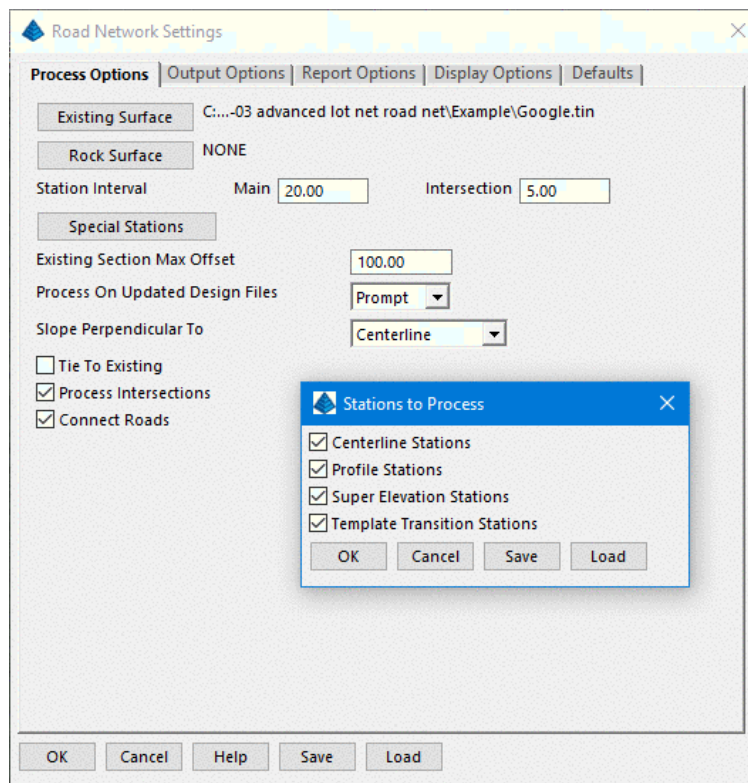


Existing Surface: Use this button to browse to and select the Existing Surface file to be used for the Road Network. Either a TIN or FLT triangulation file are accepted as valid surfaces, both of which can be made within the command Triangulate and Contour. For speed, it is recommended that the binary TIN file format be selected.

Rock Surface: Use this button to set the Rock Surface file to be used for the Road Network. This Rock Surface is optional. When the Rock surface is specified, the program will report rock quantities with the cut. Also, the cut definition in the road template file can have a separate slope to the rock surface.

Station Interval: These settings determine the distance between cross-section samples. The user has the option of specifying one sampling interval for the Intersection and another for the remainder of the Road.

Existing Section Max Offset: Use this setting to specify the furthest distance left and right of the Centerline that cross-sections are to be sampled.



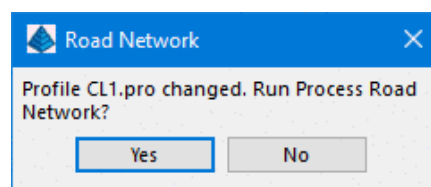
Special Stations: This button displays the **Stations to Process** dialog box (shown above). This box allows the user to decide whether or not cross-sections are to be sampled at critical design points along each Centerline. **Special Stations** include critical points such as the PC & PT for Centerlines and the PVC, PVT, High Point and Low Point for Profiles. "Additional Special Stations" may be added by entering the station number. These settings apply to *all Roads* in the Road Network. To identify **Special Stations** for a particular Road, pick the **Special Stations** button in the **Edit Road** dialog box.

Process On Updated Design Files: This setting has 3 options: *Off*, *Prompt* and *Auto*:

Off: This option allows changes to the design files without triggering an automatic update to the entire Road Network.

Prompt: This option automatically prompts the user, "**Process Road Network?**" when design files are changed.

Auto: This option automatically updates the Road Network any time a design file is changed.



Prompt to Process Updated Road Network Design

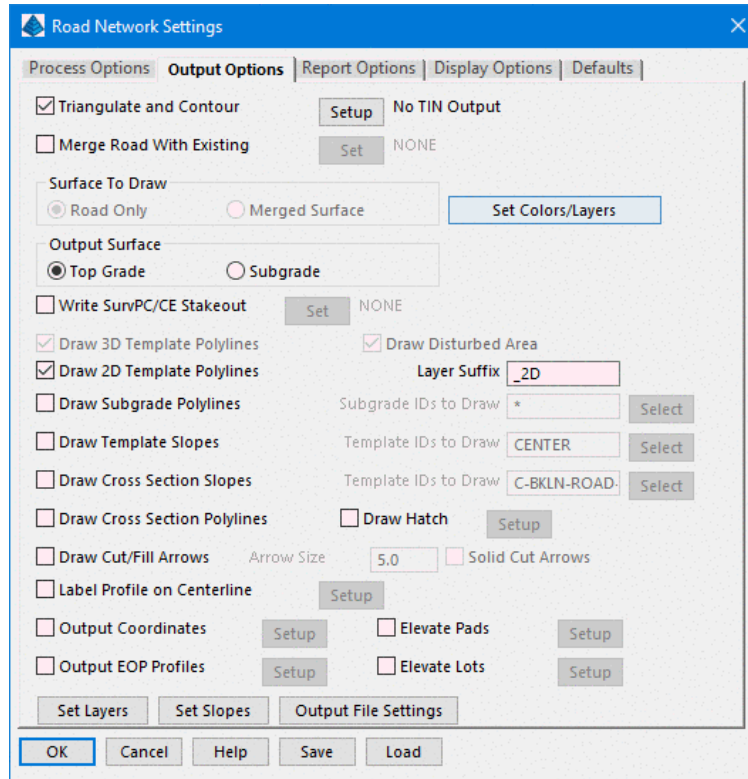
Slope Perpendicular To: This setting allows the user to specify the direction of cut and fill slope projection by selecting one of two options: *Centerline* and *Slope Direction*. The *Centerline* method projects the cut and fill slopes perpendicular to the Centerline of the Road without regard to the Profile of the Road. The *Slope Direction* method considers the Profile of the Road when projecting the specified cut and fill slopes. For example, projecting cut and fill slopes of 2:1, perpendicular to the Centerline, along a length of Road with a Profile slope of 10% would result in a slightly steeper slope (1.96:1) if measured along the top or toe of that slope. If the same conditions exist but the *Slope Direction* method is applied, the resulting slope (when measured perpendicular to the Centerline) is slightly less steep (2.04:1) but when measured along the top or toe of slope will be exactly 2:1.

Tie to Existing: If enabled and cut and fill slopes have been defined in the Template (.TPL) file, this setting will

project the specified slopes to the Existing Ground surface. If not enabled, the Road design will stop at the last **Template ID** preceding the cut and fill slopes.

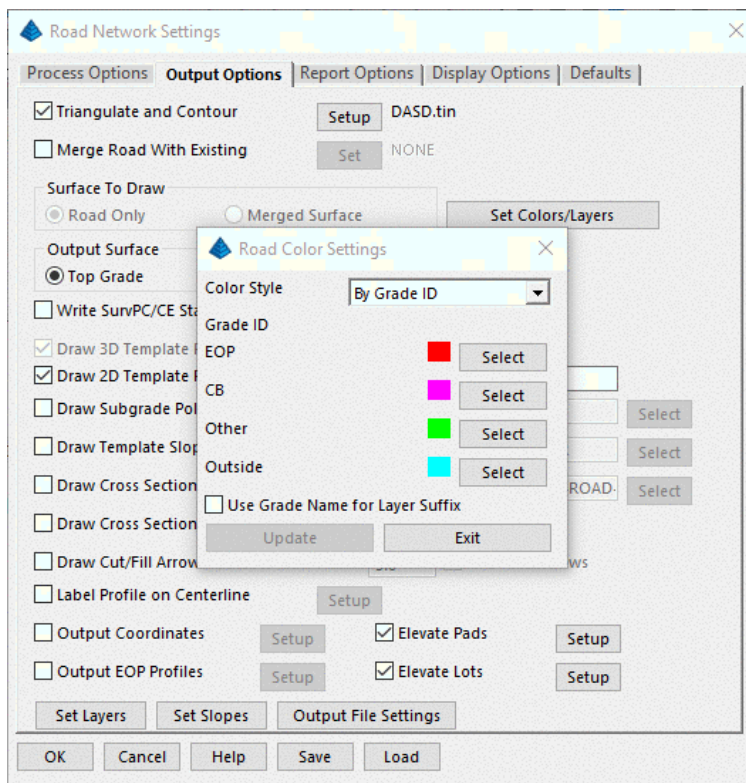
Process Intersections: If enabled this option will calculate all Roads and Intersections. If it is not enabled, each Road will be processed individually.

Connect Roads: This option applies to the 3D polylines/breaklines that are created when Processing the Road Network. If this option is enabled, the 3D polylines for different Roads will be combined around and through Intersections. If it is not enabled, the polylines will be drawn for each Road separately.



Output Options Tab

Set Road Colors: With **Triangulate and Contour** enabled, use the **Set Colors/Layers** button to display the **Road Color Settings** dialog box.



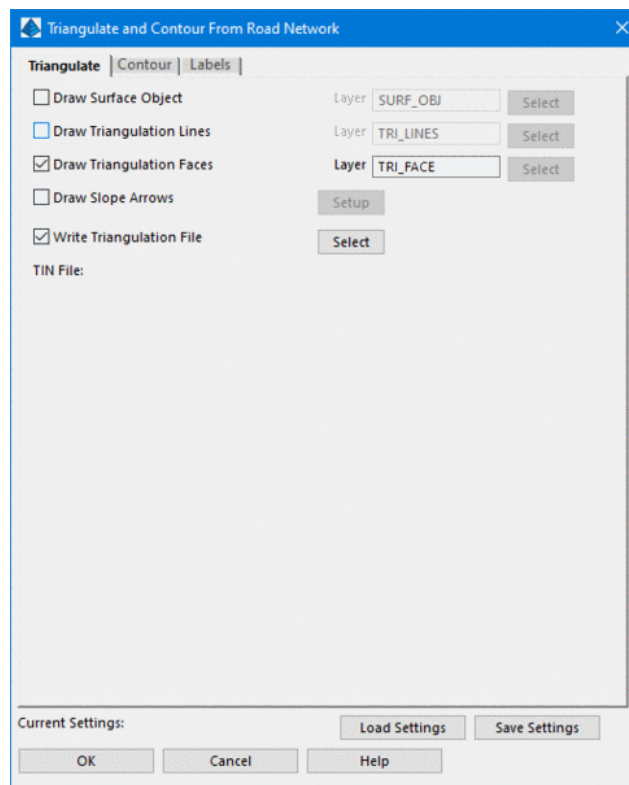
Triangulate and Contour: When enabled, use the **Setup** button to display the **Triangulate and Contour From Road Network** dialog box. Since this command is very similar to the **Surfaces** → **Triangulate and Contour** command, only those Settings and Options directly affecting the Road Network will be discussed here. Please refer to the **Help** files for that command if additional assistance is needed.

In the Triangulate and Contour From Road Network dialog box...

Triangulate tab

Draw Triangulation Faces: The **Road Network** version of this command provides additional controls (beyond those in the standard **Triangulate and Contour** command) for managing the color of the "Triangulation Faces". Once the "Triangulation Faces" option is enabled, the **Set Colors/Layers** button becomes active and, when picked, will display the **Road Color Settings** dialog box (shown below). The color of the faces can be set either by using the **Template IDs** defined in the Template (.TPL) file or using a color range based on the "Cut & Fill Depths" that uses a range of Reds and Blues to show areas and depths of Cut and Fill for the proposed Road Network. After the Road Network has been Processed, these shaded faces can be viewed using the **3D Viewer Window** command. Also within Triangulate & Contour, there is **Draw Slope Arrows** to create arrows in the drawing to show the direction of each triangular "plate" in the Road Network TIN. This can be helpful to visualize where water will be flowing.

Write Triangulation File: Once enabled, use the **Browse** button to specify the path and filename for the roadway design Surface (.TIN) file.



Contour and Labels tabs...

Use these tabs to define the settings for proposed contours and contour labels.

Output Options (Continued)

Merge Road with Existing: When enabled, use the **Set** button to specify the path and filename of a third Surface (.TIN) file to be created by merging the Existing and roadway design Surface (.TIN) files.

Surface To Draw: When creating both the road surface with Triangulate and Contour, and the combined road with existing ground using Merged Road, then this option is used to choose which surface to draw as 3D faces.

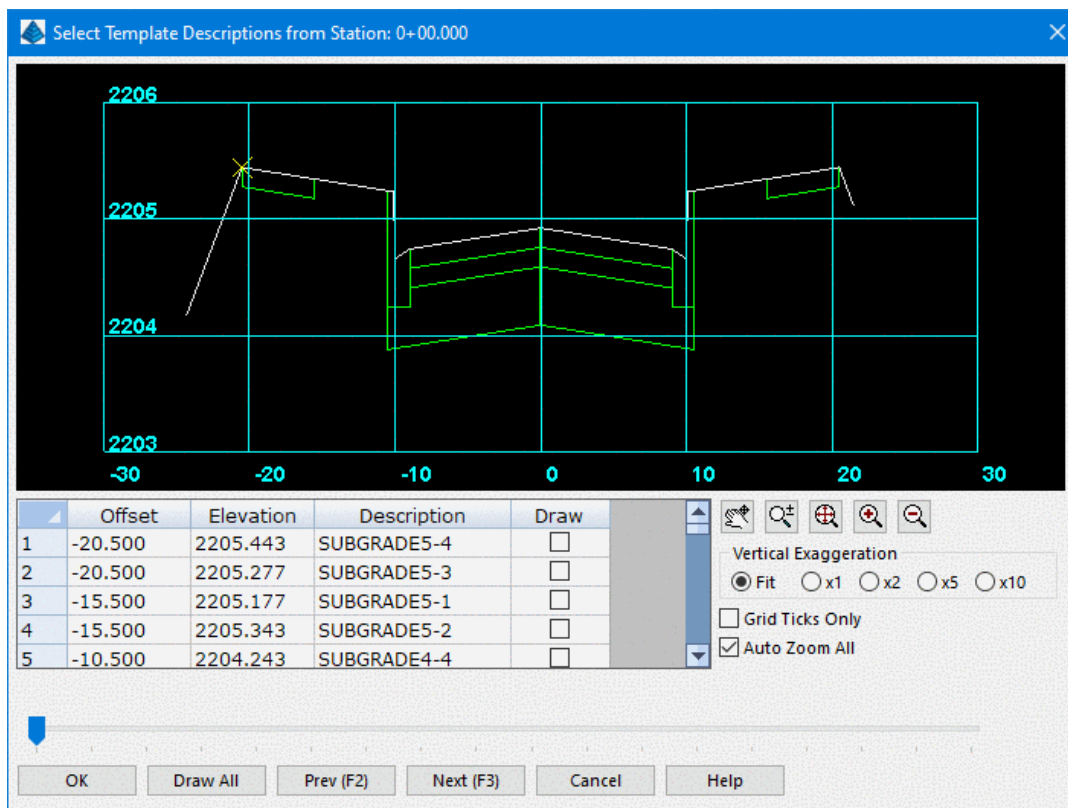
Write SurvCE Stakeout: When enabled, use the **Set** button to specify the path and filename of a SurvCE Stakeout (.RNF) file to be exported. This file can be directly loaded into data collectors using Carlson SurvCE for unlimited field stakeout of the Road Network.

Draw 3D Template Polylines: When enabled, this option will draw all 3D polylines used to generate the roadway design Surface. This option is automatically enabled when the **Triangulate and Contour** option is enabled. The layer for the polylines is set by picking the **Set Layers** button in **Output Options**.

Draw 2D Template Polylines: The 2D Template Polylines are the same as the 3D Template Polylines except that these polylines are created at zero elevation as lwpolyines. The layer names are the same except with the Layer Suffix added.

Draw Disturbed Area: When enabled, this option will draw a closed, zero-elevation polyline around the limits of disturbance of the roadway design Surface. The layer for the polyline is set by picking the **Set Layers** button in **Output Options**.

Draw Subgrade Polylines: When enabled, this option will draw all 3D polylines used to generate the roadway subgrade Surface(s). These polylines can be used to manually generate additional surfaces for modeling, stakeout or machine control purposes. Entering an asterisk (*) in the text box will draw polylines for all **Template IDs**. Once a Road has been added to the Network, the **Select** button will be activated. Picking the **Select** button displays a view of the Template (.TPL) file at the starting station and allows the user to **Draw** polylines for selected Subgrade IDs. If needed, the **Next** and **Previous** buttons at the bottom of the window allow the user to browse through the stations of the road design to find a particular Subgrade ID. The layer for the polylines is set by picking the **Set Layers** button in **Output Options**.



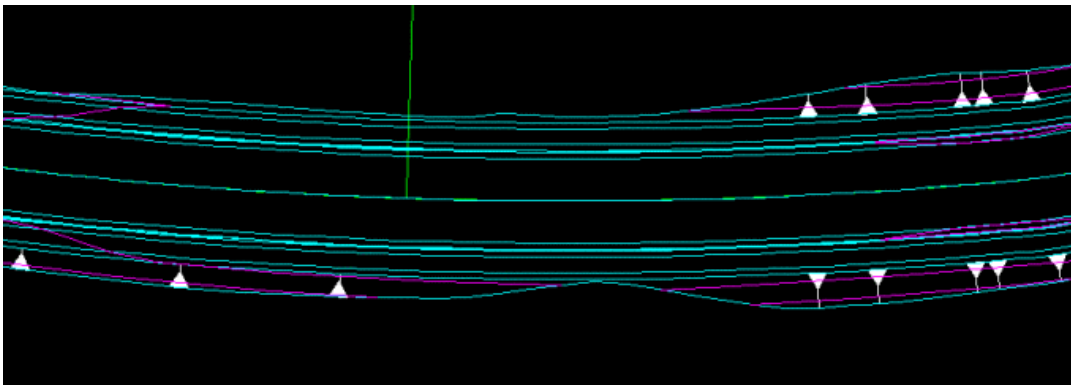
Pick Subgrade Polylines to Be Drawn

Draw Template Slopes: When enabled, this option will draw slope arrows parallel to the Centerline at the selected **Template IDs**. This option may be used to indicate direction and steepness of slope along the flowline of the gutter. Entering an asterisk (*) in the text box will draw slope arrows for all **Template IDs**. Once a Road has been added to the Network, the **Select** button will be activated. Picking the **Select** button displays a view of the Template (.TPL) file (similar to the one shown above) and allows the user to **Draw** polylines for selected **Template IDs**. If needed, the **Next** and **Previous** buttons at the bottom of the window allow the user to browse through the stations of the road design to find a particular **Template ID**. The layer for the slope arrows is set by picking the **Set Layers** button in **Output Options**. Other slope arrow settings are specified by picking the **Set Slopes** button in **Output Options**.

Draw Cross Section Polylines: When enabled, this option will draw a 3D polyline defining the roadway design surface cross-section at each sampled station along the Centerline. These polylines can be used to manually generate additional surfaces for modeling, stakeout or machine control purposes. The layer for the polylines is set by picking the **Set Layers** button in **Output Options**.

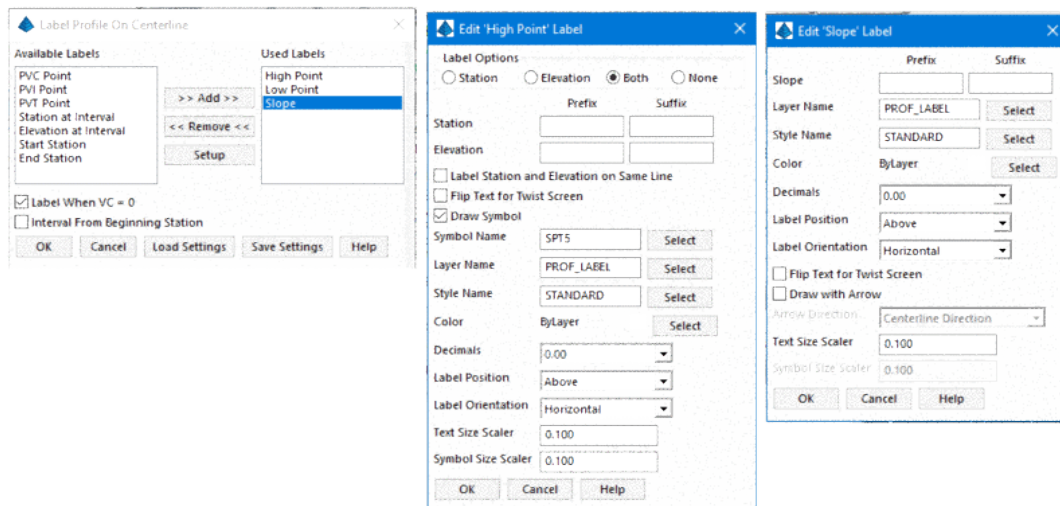
Draw Hatch: This option will hatch the grade areas. Pick the Setup button to define which hatch pattern to use for each Template Grade ID.

Draw Cut/Fill Arrows: When enabled, this option will draw arrows at each sampled cross-section station so that the arrow is pointing down-slope. The example shown below indicates a section of Cut slope transitioning to a section of Fill slope. Once enabled, the user has the ability to adjust the size of the arrows and specify whether or not the Cut/Fill Arrows should be solid.



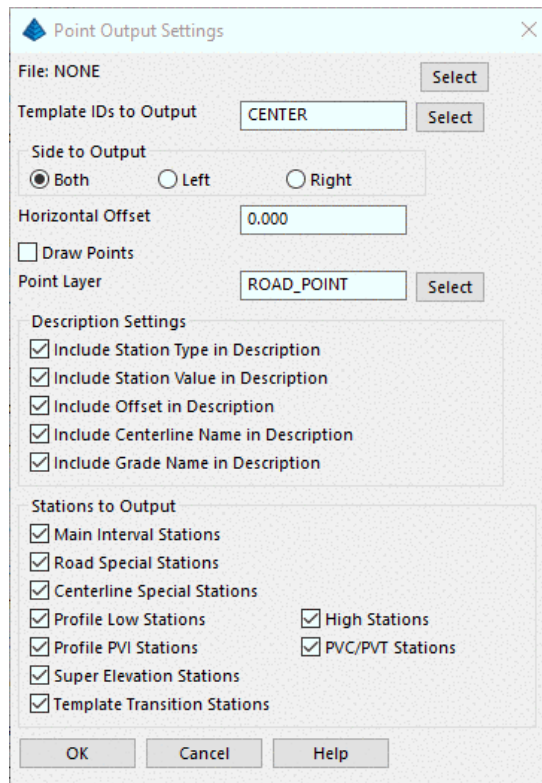
Cut/Fill Arrows On Slopes

Label Profile on Centerline: When enabled, this option labels Profile slopes and critical points such as PVC, PVT, High and Low Points in plan view along the Centerline. Once enabled, use the **Setup** button to open the **Label Profile on Centerline Settings** dialog. Then, from the list of "Available Labels", select the label(s) to be drawn and use the **Add** button to shift them to the list of "Used Labels". Selecting one of the "Used Labels" and then picking the **Setup** button allows the user to configure the label style and settings for each type of label.



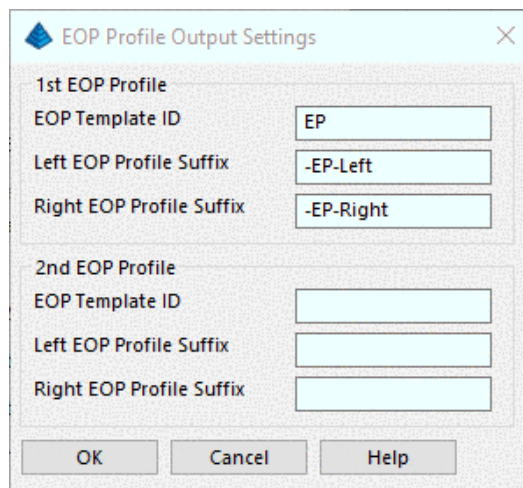
Label Profile on Centerline and Label Setup Dialog Boxes

Output Coordinates: When enabled, this option allows the user to export a Coordinate (.CRD) file containing all of the critical points for the Road Network. Once enabled, pick the **Setup** button to specify the path, file name and other criteria for the point file.



Point Output Settings Dialog Box

Output EOP Profiles: This option creates individual Profile (.PRO) files for the specified template grade such as the edge of pavement. The EOP profiles use the stations of the main centerline and the elevations from the template grade. You can create up to two EOP profiles such as for curb flow line and top-of-curb.



Output EOP Profiles Dialog Box

Elevate Pads: When enabled, this option adjusts the elevation of closed polylines within a specified proximity of the Road Network. Once enabled, use the **Setup** button to open the **Elevate Pad Settings** dialog box and configure the settings.

In the Elevate Pad Settings dialog box...

Reference Template ID: When determining the new pad elevation, all distances and elevation changes are based on the **Template ID** specified here. Type the **Template ID** in the text box or use the **Select** button to choose from a list.

Pad Layer: All original polylines found on this layer (and within the Max Offset of the Reference Template ID) will be elevated.

Max Offset: All original polylines within this distance of the Reference Template ID and on the specified "Pad Layer" will be elevated.

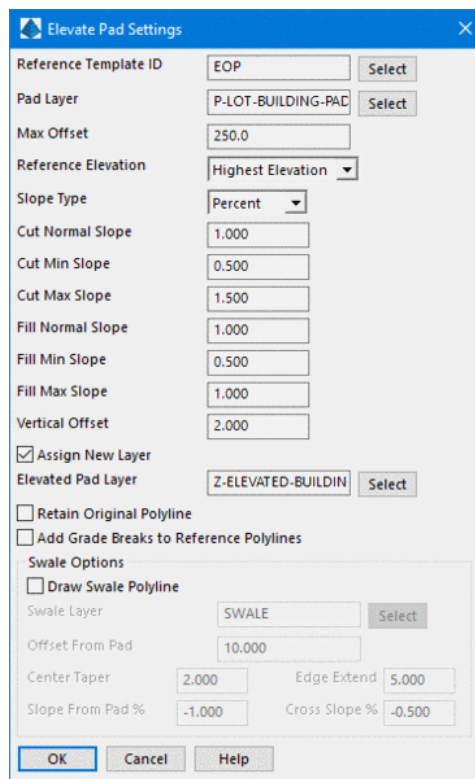
Reference Elevation: This setting has 3 options: "Highest Elevation", "Lowest Elevation" and "Elevation at Middle". Of the elevations found along the **Reference Template ID** that are adjacent to the pad polyline, the command will use either the highest, lowest or middle elevation found to set the new pad elevation.

Slope Type: This setting has 3 options: Percent (%), Ratio (x:y) and Vertical (change in elevation).

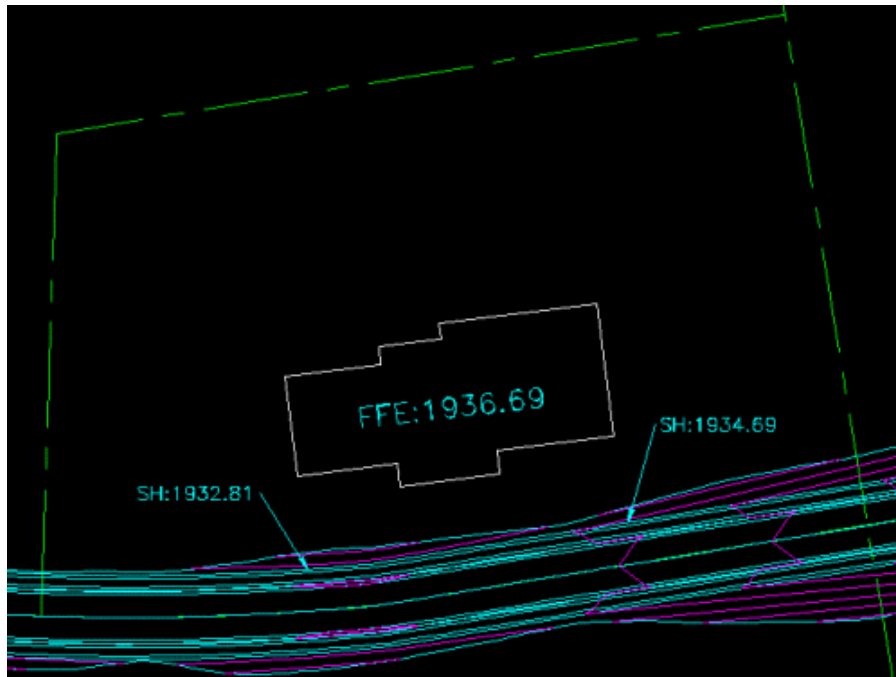
Cut/Fill, Normal/Min/Max Slopes: For future earthwork balancing adjustments, the settings in this dialog are used to specify the range of allowable slopes when in cut or fill conditions.

Assign New Layer: When enabled, this option allows the user to specify a new layer for the new, elevated pad polyline. Once enabled, either type the new layer name in the text box or use the **Select** button to choose the layer from a list.

Retain Original Polyline: This option is only available if the "Assign New Layer" option is enabled and will keep the original, zero-elevation polyline in addition to the new, elevated polyline. If this option is not enabled, the original polyline will be deleted from the drawing.



Elevate Pad Settings Dialog Box In the dialog shown here, all closed polylines on layer PAD that are within 100 feet of the road will have their elevations set based on a 2 percent grade up from the PAVE Template ID point, in either Cut or Fill conditions. In future earthwork balancing adjustments, the polyline can be adjusted a maximum of up to a 10 percent grade or down to a 1 percent grade from the Reference Template ID. The example below shows the results of elevating a pad so that it is 2.0' above (using Vertical option) the highest point along a Reference Template ID of "SH" (Shoulder) on the adjacent Road.



Elevate Pads - Example

Output Options (Continued)

Elevate Lots: When enabled, this option follows a logic similar to that of the **Elevate Pads** routine in that it elevates zero-elevation lot lines relative to a road design and based on a set of grading rules. Once the option is enabled, use the **Setup** button to display the **Elevate Lots Setup** dialog box.

In the Elevate Lots Setup dialog box...

Grading Rules: The necessity of a Grading Rules (.GRR) file is the key difference between elevating pads and elevating lots. If a Grading Rules (.GRR) file has already been prepared, use the **Select** button to browse to and select the file. To create a new file, pick the **Edit** button to open the **Define Grading Rules** dialog box and specify the desired settings. Then, pick the **SaveAs** button to **Save** a Grading Rules (.GRR) file. **Define Grading Rules** is a command that also exists outside of the **Road Network** feature. Please refer to the **Help** files for that command if additional assistance is needed.

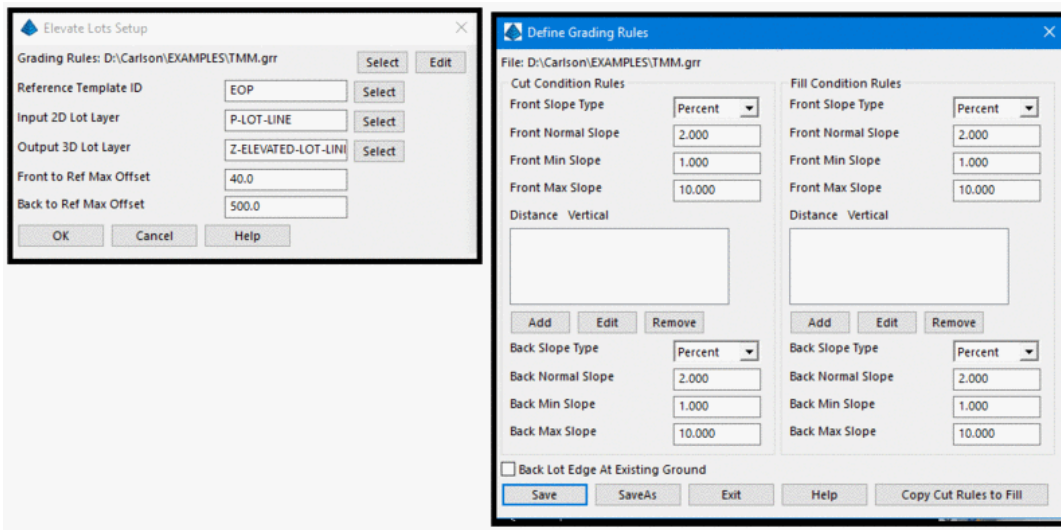
Reference Template ID: When determining the elevations for the new lot line, all distances and elevation changes are based on the **Template ID** specified here. Type the **Template ID** in the text box or use the **Select** button to choose from a list.

Input 2D Lot Layer: All original polylines found on this layer (and within the Max Offset of the Reference Template ID) will be elevated. Either type the layer name in the text box or use the **Select** button to choose the layer name.

Output 3D Lot Layer: This is the layer to which the newly elevated lot lines will be assigned. Either type the layer name in the text box or use the **Select** button to choose the layer name.

Front to Ref Max Offset: Use this setting to specify a distance from the **Reference Template ID** beyond which Lot Frontage polylines will not be elevated.

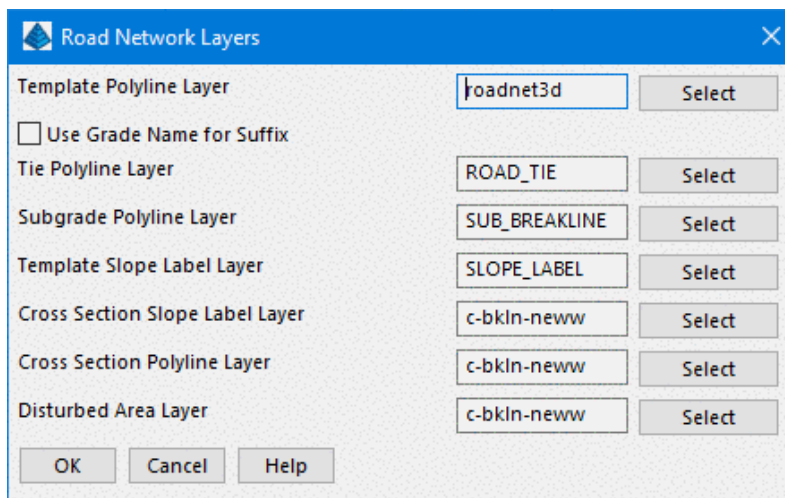
Back to Ref Max Offset: Use this setting to specify a distance from the **Reference Template ID** beyond which Back/Rear Lot polylines will not be elevated.



Elevate Lots and Define Grading Rules Dialog Boxes

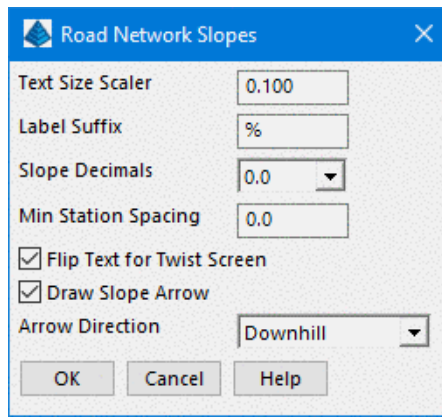
Output Options (Continued)

Set Layers: Pick this button to display the **Road Network Layers** dialog box.



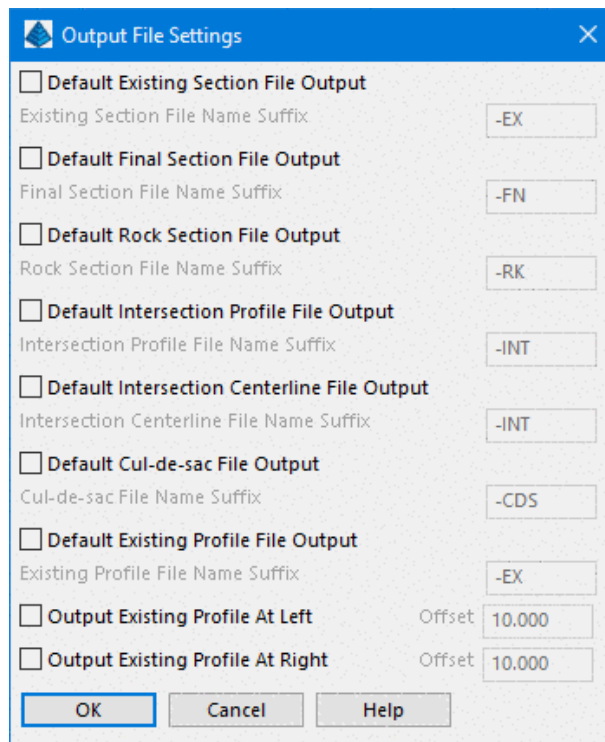
Road Network Layers Dialog Box

Set Slopes: Pick this button to display the **Road Network Slopes** dialog box and configure the settings for drawing slope arrows.



Road Network Slopes Dialog Box

Output File Defaults: Pick this button to specify default output file names for Centerline (.CL), Profile (.PRO) and Section (.SCT) files to be created when Processing the Road Network. Each item in the road design (Road, Intersection, Cul-de-sac) has its own output file names. The default options in this dialog apply to fill out default output file names when the item is first added to the road design. The Output Existing Profile At Left/Right options create offset profiles within the existing ground output profiles.



Output File Defaults Dialog Box

Settings on this tab allow the user to specify defaults for the Road Network **Report** feature. This feature is accessible from the **Report** button of the **Road Network: Task Pane**.

Report Precision: Specify the decimal precision for the report.

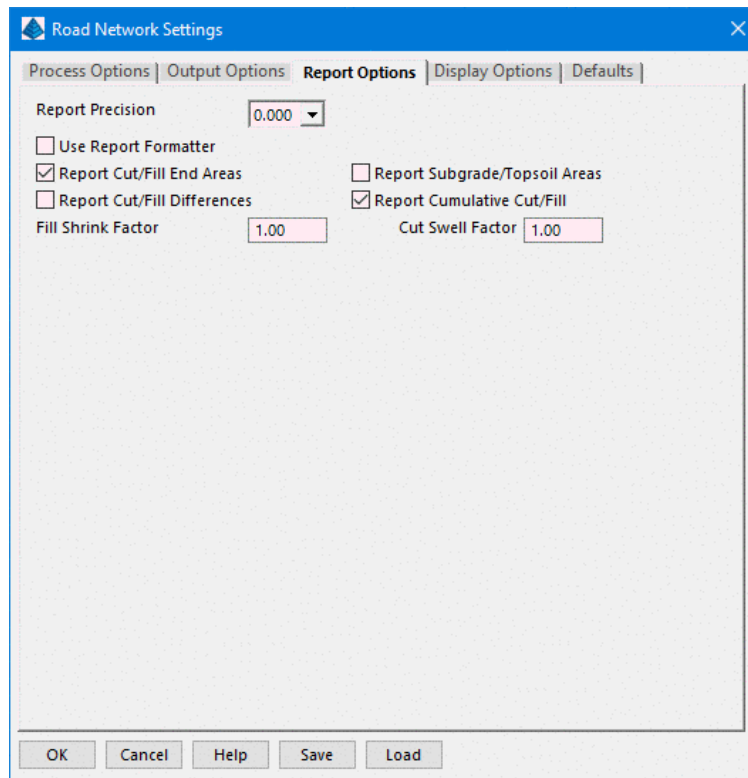
Use Report Formatter: This option allows for customized report layout and contents. Otherwise a standard report is displayed.

Report Cut/Fill End Areas: Specify whether or not to report cut/fill at each station.

Report Cut/Fill Differences: Adds a running total of the cut to fill balance at each station to the report.

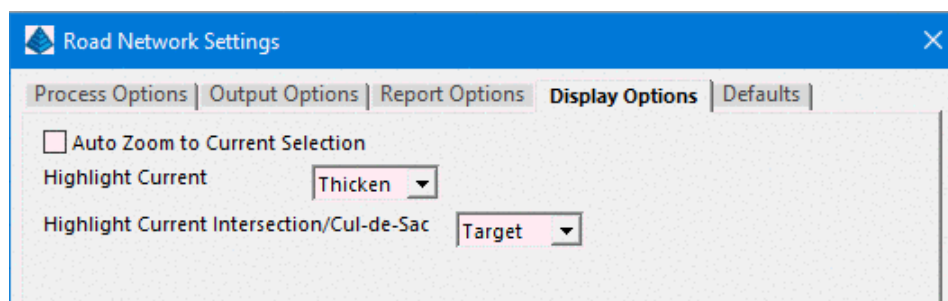
Report Cumulative Cut/Fill: Adds a running total of the cut/fill at each station to the report.

Fill Shrink/Cut Swell Factor: Allows you to specify a value that the volume calculated will be multiplied by.



Report Options Tab

Settings on this tab control how the drawing display updates when items are highlighted in the **Road Network: Task Pane**. The Auto Zoom option zooms the drawing to show the highlighted item. The **Highlight Current Centerline** chooses between no highlight, showing a thick centerline or showing a dashed centerline. The **Highlight Current Intersection/Cul-de-Sac** chooses between no highlight, drawing a pointer arrow or drawing a target circle symbol.



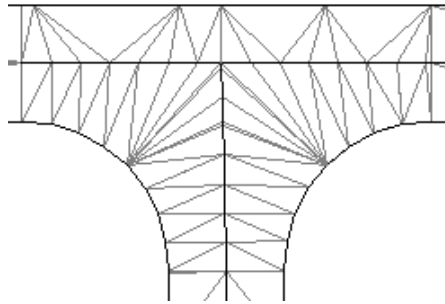
Display Options Tab

Settings in this tab allow the user to specify the default values used for transitioning from Road to Road, from Road to Intersection and from Road to Cul-de-Sac. There are also default dimensions and settings for new cul-de-sacs. You can also specify a default Road Design Parameter file to apply to new roads for checking parameters like max profile slope and min sight distance.

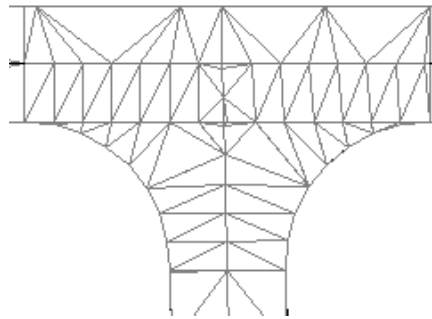
CL Intersections: Use this setting to define the default transition distance and vertical curve length for intersecting Centerlines. See **Road Network: Adding and Editing Intersections** for more.

Side Intersections: Use this setting to define the default vertical curve length for the Profile and the default radius for Corners at Intersections. See **Road Network: Adding and Editing Intersections** for more.

Surface Method: When calculating Intersections, there are two options for handling the cross-sections of the intersecting Roads: "Hold Main Crown", which honors the Primary Road Template through the Intersection, or "Radial from Curb", which grades between the Centerline Profile and the Profile of each Corner of the Intersection. The Profile for the Corner may be defined as the edge of pavement (EP), back of curb (BC) or other point on the cross-section by specifying the **Template ID** in the **Settings** tab of the **Edit Intersection** dialog box.



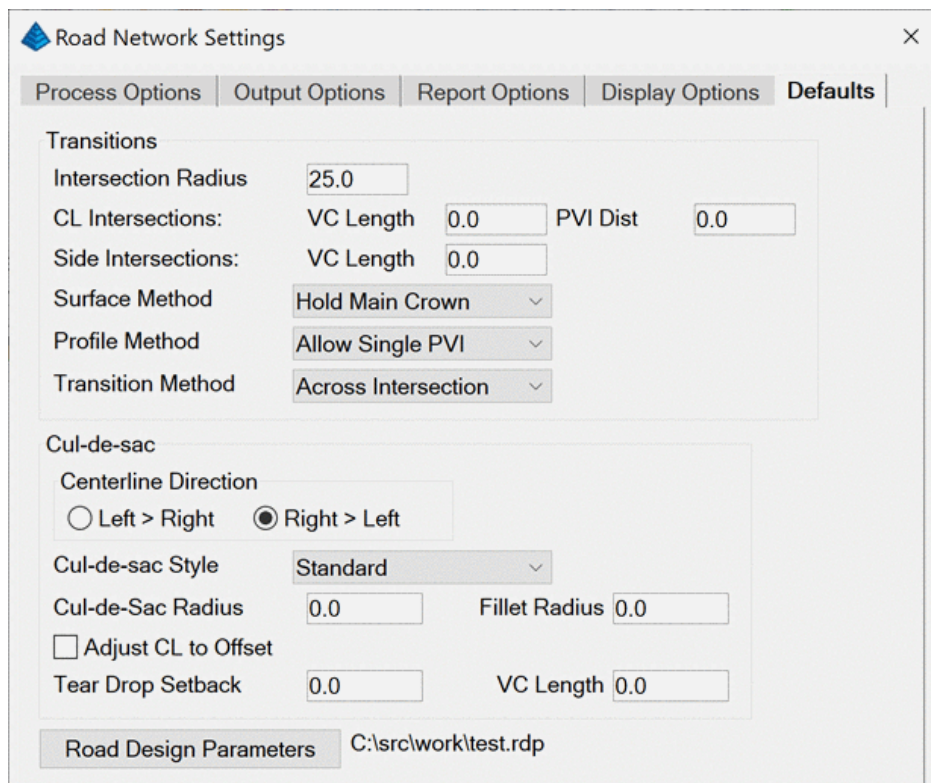
Surface Method: Radial From Curb



Surface Method: Hold Main Crown

Transition Method: This setting applies when a Road has a varying width through an Intersection. The "Across Intersection" option looks at the Primary Road (from start to end of the Intersection) to find the maximum offset distance between the Centerline and edge of pavement, and uses this distance to set the edge of pavement breakline across the Intersection with the Secondary Road. The "Mid Point" option simply finds the pavement width at the Intersection station and uses this distance to set the edge of pavement breakline across the Intersection.

Cul-de-Sac: Use this setting to define the default vertical curve length along the Cul-de-Sac Profile.



Defaults Tab

Roads in a Road Network are managed in the **Road Name** area of the **Road Network: Task Pane**.

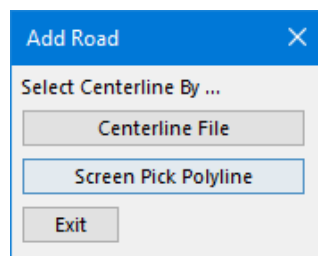
Add: Pick this button to **Add** a Road to the Network. After adding the Road, the **Edit Road** dialog box is displayed allowing the user to manage and make changes to the **Input Files** and **Output Files** for the selected Road.

Edit: Pick this button to display the **Edit Road** dialog box to manage and make changes to the **Input Files** and **Output Files** for the selected Road.

Remove: Pick this button to delete the selected Road from the Road Network. After Removing the Road from the Network the design files associated with that Road will remain in the project folder.

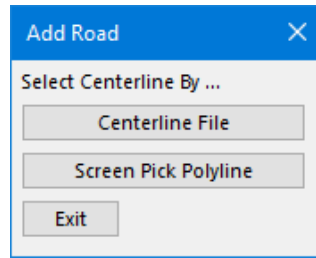
Adding a new Road may be done either by selecting a pre-defined Centerline (.CL) file or by screen-picking a 2D Polyline in the drawing and assigning a new Centerline (.CL) file to it.

Add: Use this button to **Add** a Road to the Road Network or right-click on Roads in the project tree and pick Add Road. After picking the **Add** button, the **Add Road** dialog box gives the user the option to "Select Centerline By..." **Centerline File** or **Screen Pick Polyline**. If the **Centerline File** option is chosen, the user is prompted to browse to and select the Centerline (.CL) file.



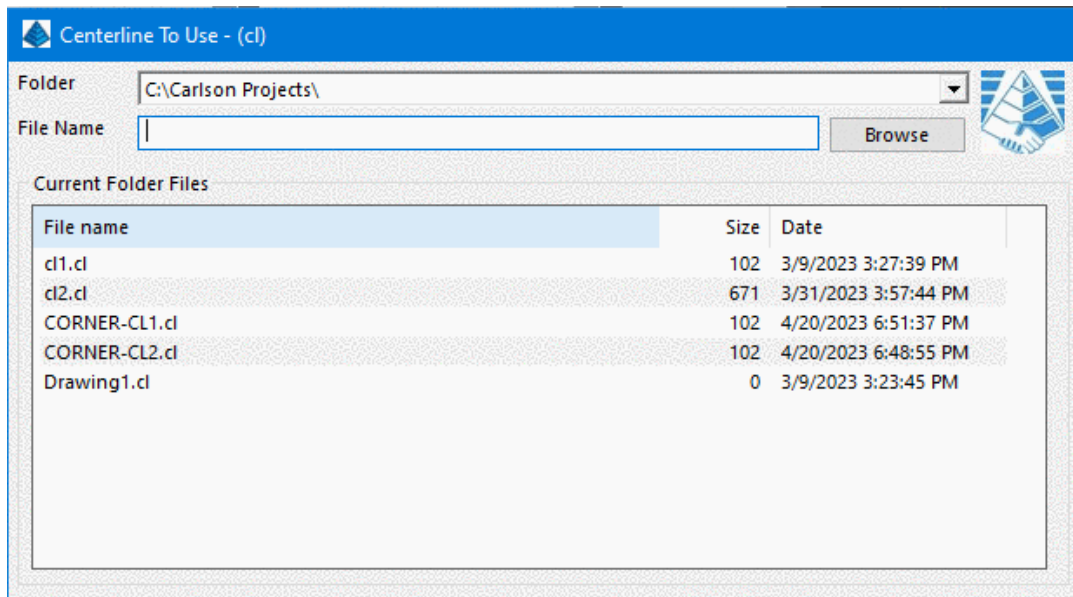
Specify Method to Use to Add Road

If the **Screen Pick Polyline** option is chosen, the user is prompted to select a polyline in the drawing. If an associated Centerline (.CL) file is not found in the project folder, the **Set Centerline** dialog notifies the user that, "No centerline file associated with polyline..." and the user must choose to either select another polyline or to **Assign Centerline File to Polyline**.



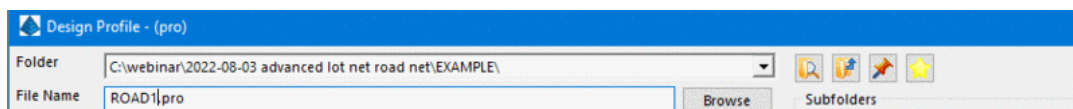
Set Centerline Dialog Box

After picking the **Assign Centerline File to Polyline** button, the **Centerline to Set** file dialog box prompts the user to assign a path and filename for the new Centerline (.CL) file.



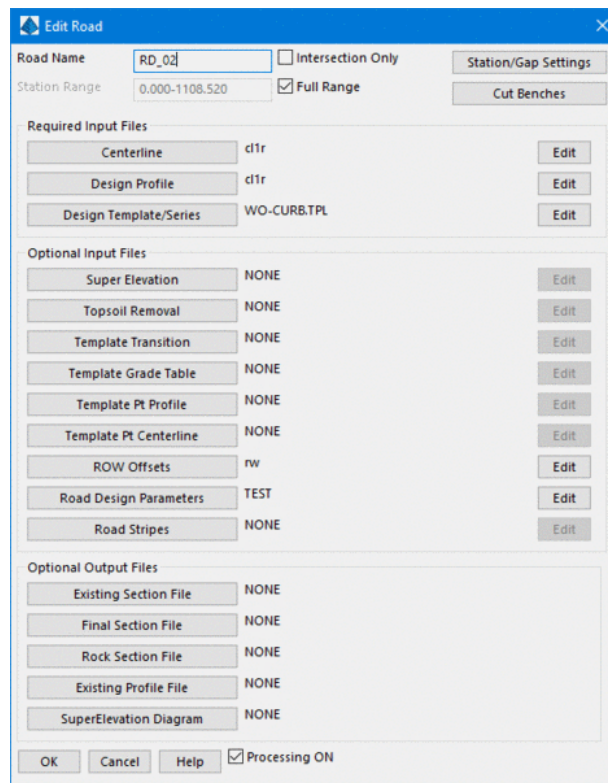
Centerline to Set File Dialog Box

Immediately upon defining the new Road, the **Profile to Use** file dialog box prompts the user to assign a path and filename for the proposed Profile (.PRO) file for the Road. By default, the new Profile (.PRO) file is named the same as the Centerline (.CL) file.



Profile to Use File Dialog Box

After specifying the Centerline (.CL) and Profile (.PRO) files for the Road, the **Edit Road** dialog box is displayed. This dialog serves as the "manager" for all files relating to the specific Road. The **Edit Road** dialog box allows the user to apply settings and associate various files that are specific to the Road - not the entire Road Network. The **Edit** button in the **Road Name** section of the **Road Network: Task Pane** also displays this dialog box.



Edit Road Dialog Box Intersection Only: If this option is enabled, Road Network will only consider the portions of this Road that intersect with other Roads when calculating the design.

Full Range: This option will process the full station range of the road. Otherwise, turn this option off and set the **Station Range** to process a subset of the road.

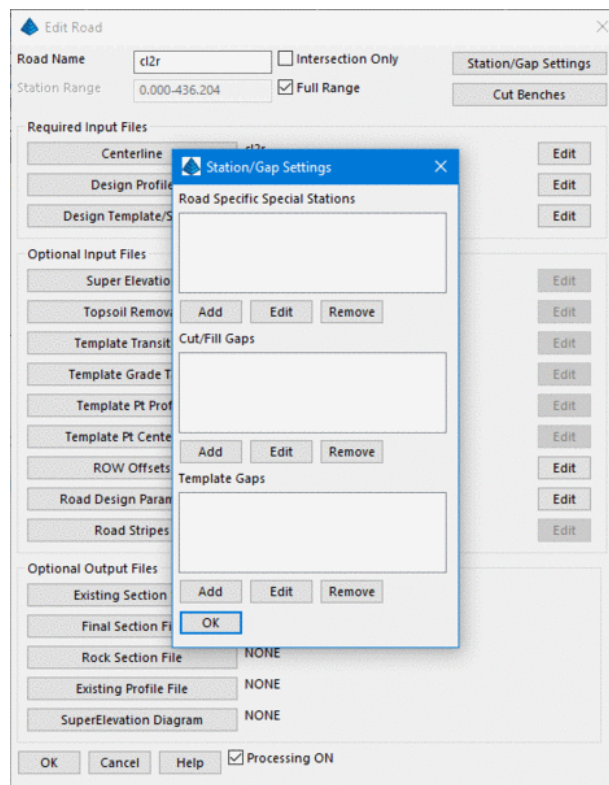
Cut Benches: Pick this button to specify up to 4 triangulation surface files to use when the "Slopes In Series" and "Cut to Surface" options are used in the Template (.TPL) file. In cut conditions, the program will look to intersect with these surfaces before it reaches the final target surface which is the Existing Surface set under Settings. The order of the Cut Bench Surfaces are from lowest to highest cut. So the Bench #1 is closest to the road and Bench #4 is furthest.

Station Settings: Pick this button to display for special stations and cut/fill gaps.

Special Stations: Enter one or more stations at which to sample cross-sections.

Cut/Fill Gaps: Use the Add and Remove buttons to define a series of station ranges for cut/fill gaps where the program will not calculate any volumes or apply the template cut/fill tie slopes. For example, these stations could be used across a bridge.

Template Gaps: These define station ranges to skip over and not create the road.

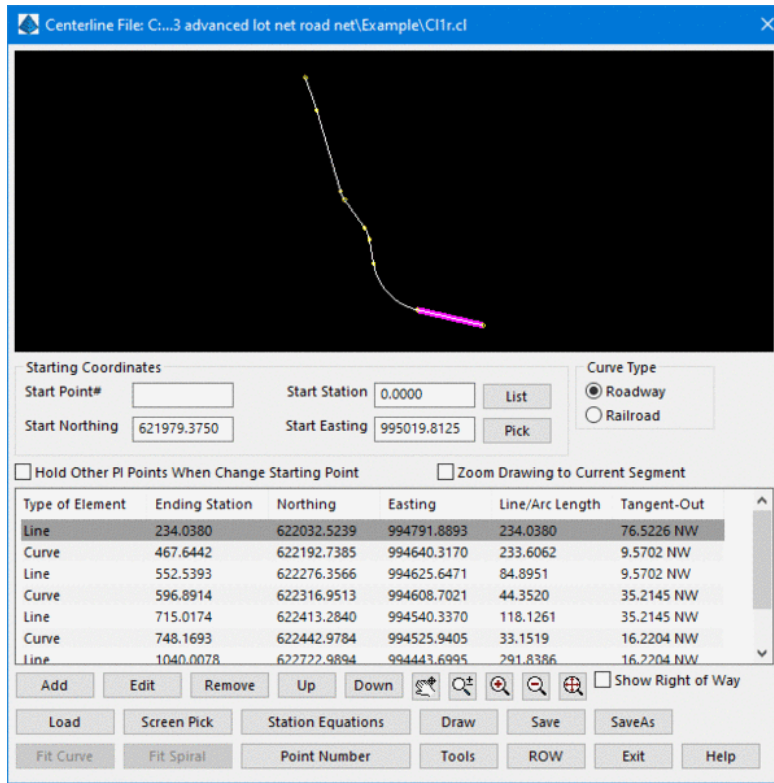


Add Road Specific Special Stations A Centerline (.CL) file, a Profile (.PRO) file and a Template (.TPL) File are **required** in order to process a roadway design using the **Road Network** feature. In addition, the **Road Network** feature accepts several additional files for designing Roads using specific criteria. In the **Edit Road** dialog box, picking the buttons on the left, that are labeled with the file type, will display a file dialog box prompting the user to select an existing or create a new file of that type. The corresponding **Edit** button to the right of each file type will display the editor for that file type.

Required Road Input Files

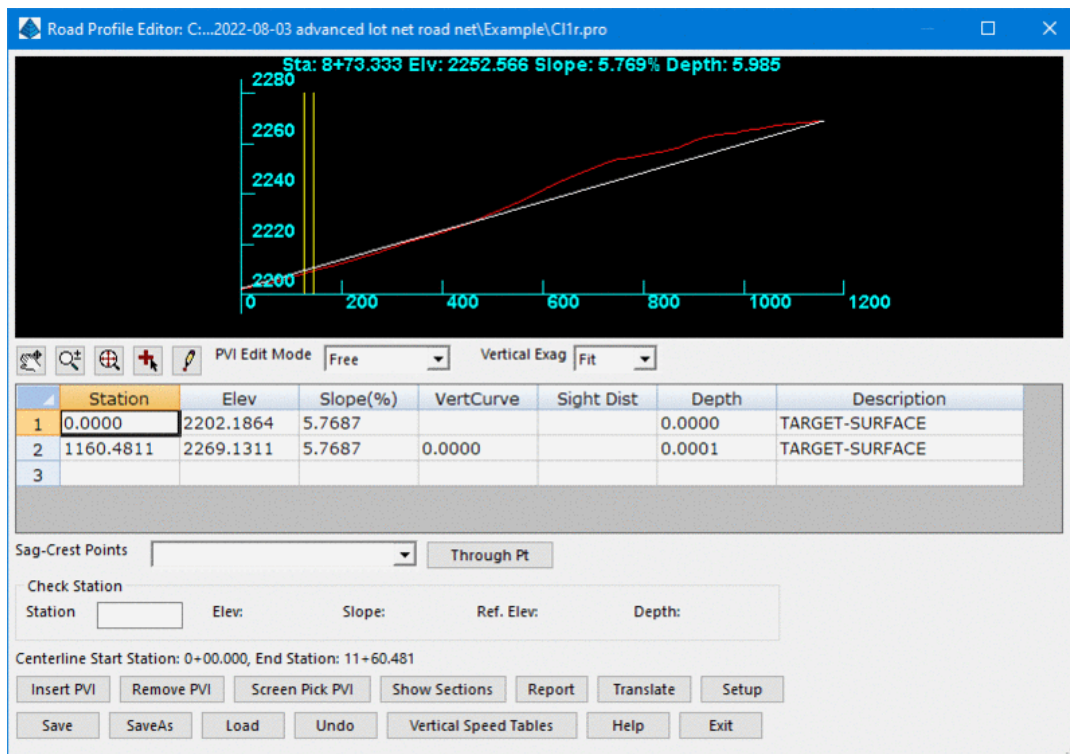
Processing On: This toggle controls whether Road Network processes this road. You may want to temporarily turn off this toggle for a road if Road Network processing isn't fully working and you want to isolate the roads having trouble.

Centerline: Pick this button to select an existing or create a new Centerline (.CL) file from which to define the horizontal alignment of the Road. The **Edit** button opens the **Centerline File** Editor. This Editor is the same as the one used for the **Centerline Editor** command. Please refer to the **Help** files for that command if additional assistance is needed.



Centerline File Editor

Profile: Pick this button to select an existing or create a new design Profile (.PRO) file for the Road. The **Edit** button opens the **Road Profile Editor**. The Editor provides the user with both a "profile-grid-view" and a "table-view" of the Profile (.PRO) file. See **Road Network: Road Profile Editor** for more.



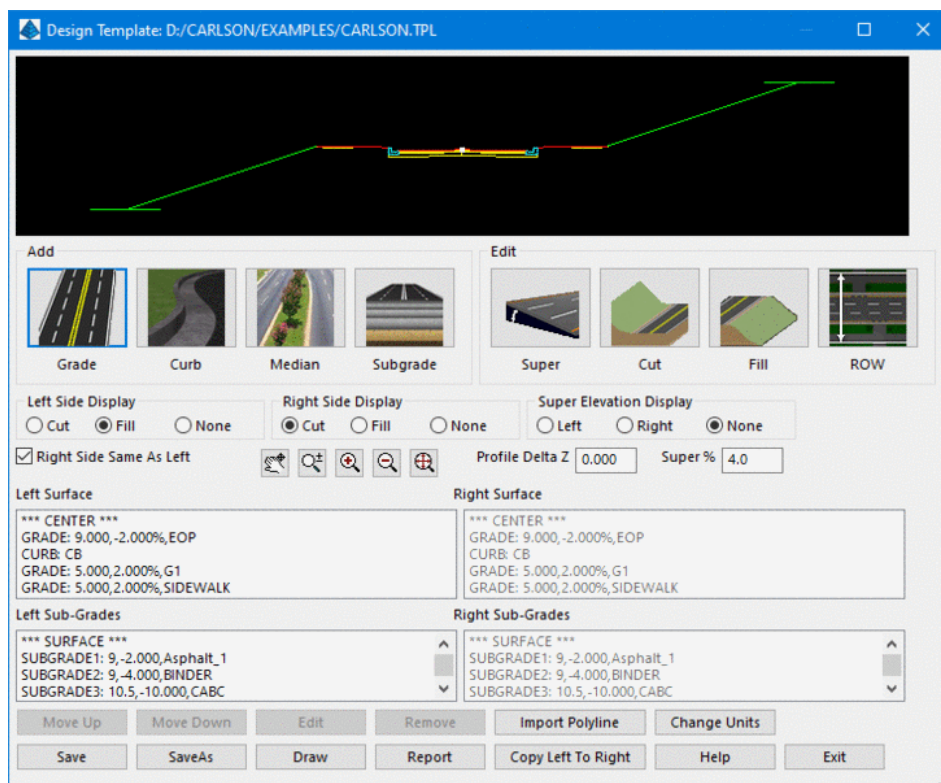
Road Profile Editor

Template: Pick this button to select an existing or create a new Template (.TPL) file or Template Series (.TSF) file for the Road.

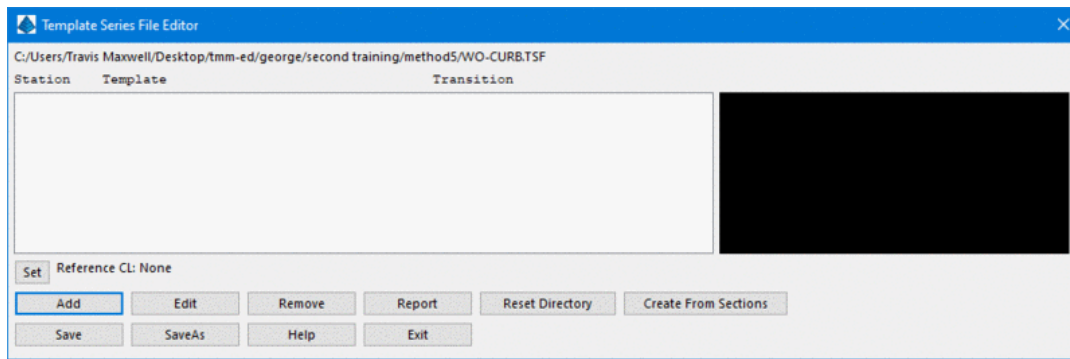
A Template (.TPL) file defines a typical roadway cross-section including pavement, curb, ditches, medians, super-elevations, subgrades, rights-of-way and cut/fill slopes. One of the most critical steps in defining a Road Template for use with the **Road Network** feature is the assigning of a **Template ID** to points on the Template. A **Template ID** is a unique name for each point on the Template and is used to transition from Road to Road, from Road to Intersection and Road to Cul-de-Sac. The **Template ID** serves 4 purposes: (1) the ID will be applied as a description to all final Template points generated in the form of a Coordinate (.CRD) file, (2) the ID can be used as a design point in the Template definition, as in EP+5 indicating 5 feet or meters right of edge of pavement, (3) points of common ID may be connected by 3D polylines in the **Output Options** tab of the **Road Network: Settings** dialog box and (4) Quantities can be generated with reference to the ID and material (gravel, concrete, etc.) also defined in the Template (.TPL) file.

A Template Series (.TSF) file references Template (.TPL) files for Template-to-Template transitioning and is one method used for widening and narrowing of Road sections.

Picking the **Edit** button will open the appropriate **Design Template** or **Template Series File Editor**. These Editors are the same as those used for the **Draw Typical Template** and **Template Transition** commands. Please refer to the **Help** files for those commands if additional assistance is needed.



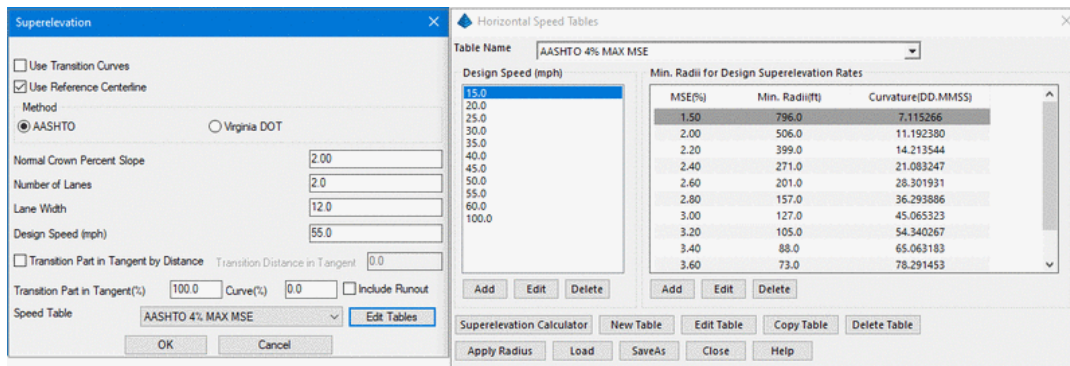
Design Template Editor



Template Series Editor

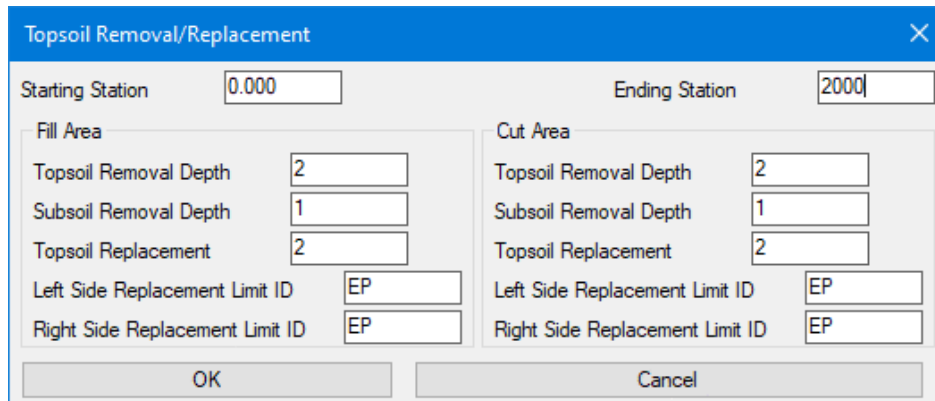
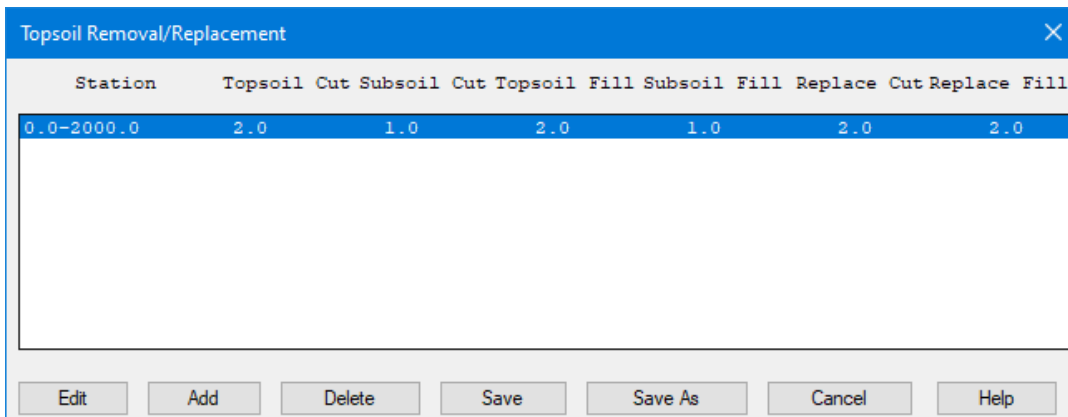
Optional Road Input Files

Super Elevation: Pick this button to select an existing or create a new SuperElevation (.SUP) file for the Road. The **Edit** button opens the **Super Elevation Editor**. This Editor is the same as the one used for the **Super Elevation Editor** command. Please refer to the **Help** files for that command if additional assistance is needed.



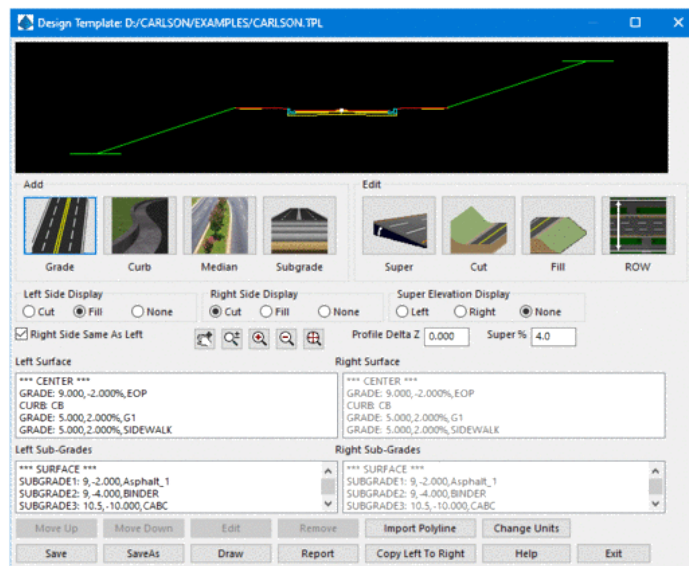
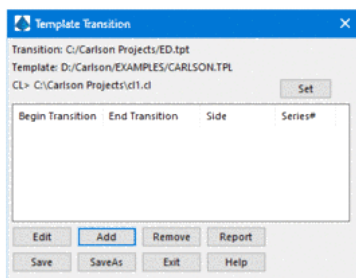
Super Elevation Editor

Topsoil Removal: Pick this button to select an existing or create a new Topsoil Removal (.TOP) file for the Road. This file allows the user to define topsoil removal and replacement zones to be used in the Road design. Different topsoil depths can be used for different station ranges and then are computed as part of the cut and fill volumes. The **Edit** button opens the **Topsoil File Editor**. This Editor is the same as the one used for the **Topsoil Removal/Replacement** command. Please refer to the **Help** files for that command if additional assistance is needed.



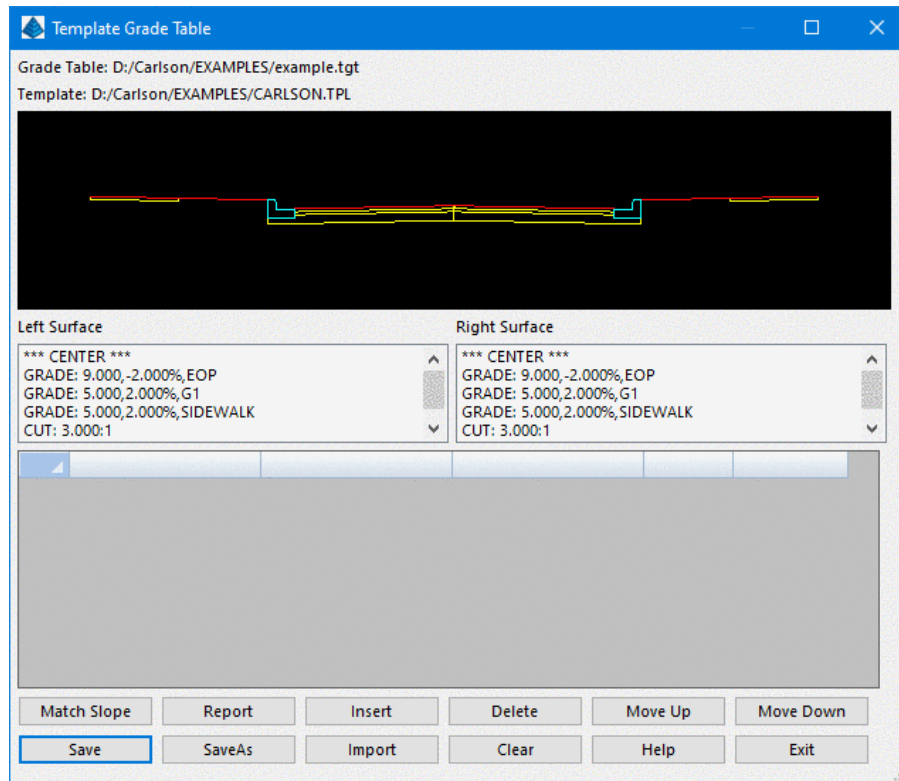
Topsoil Removal/Replacement Editor

Template Transition: Pick this button to select an existing or create a new Template Transition (.TPT) file for the Road. This file allows the user to define changes in grade distances or slopes for a specific Template ID through a range of stations and is another method of widening and narrowing Road sections. The **Edit** button opens the **Template Transition Editor**. This Editor is the same as the one used for the **Template Transition** command. Please refer to the **Help** files for that command if additional assistance is needed.



Template Transition Editor

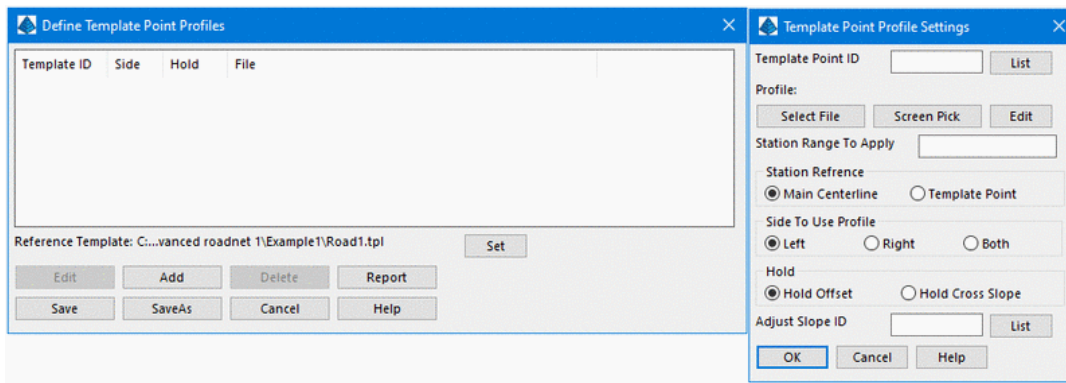
Template Grade Table: Pick this button to select an existing or create a new Template Grade Table (.TGT) file for the Road. This file allows the user to define specific slopes and distances for one or more Template IDs (and for left and right sides independently) that have been assigned in the Template (.TPL) file. The **Edit** button opens the **Template Grade Table Editor**. This Editor is the same as the one used for the **Template Grade Table** command. Please refer to the **Help** files for that command if additional assistance is needed.



Template Grade Table

Template Pt Profile: Pick this button to select an existing or create a new Template Point Profile (.TPP) file for the Road. This file allows the user to assign separate Profile (.PRO) files to specific Template IDs that have been defined in the Template (.TPL) file. This accommodates varying grade changes (for a ditch, for instance) independent of the Profile for the Centerline. The **Edit** button opens **Define Template Alignments** and then picking the **Add** button displays the **Template Point Profile Settings** dialog box. These dialog boxes are the same as the ones used for the **Assign Template Pt Profile** command. Please refer to the **Help** files for that command if additional assistance is needed.

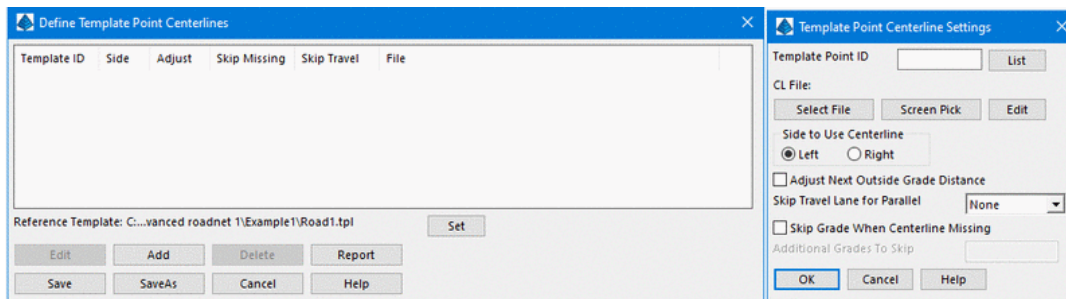
Image error here



Assign Template Pt Profile Dialog Boxes

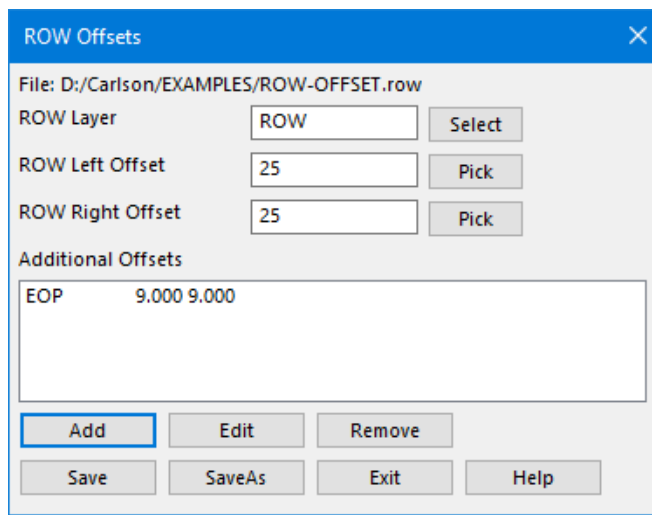
Template Pt Centerline: Pick this button to select an existing or create a new Template Point Centerline (.TPC) file for the Road. This file allows the user to assign separate Centerline (.CL) files to specific Template IDs that have been defined in the Template (.TPL) file. This accommodates varying widths for cross-section surfaces and provides an additional method of managing widening and narrowing of Roads. The **Edit** button opens **Define Template Alignments** and then picking the **Add** button displays the **Template Point Centerline Settings** dialog box. These boxes are the same as the ones used for the **Assign Template Pt Centerline** command. Please refer to the **Help** files for that command if additional assistance is needed.

Image error here

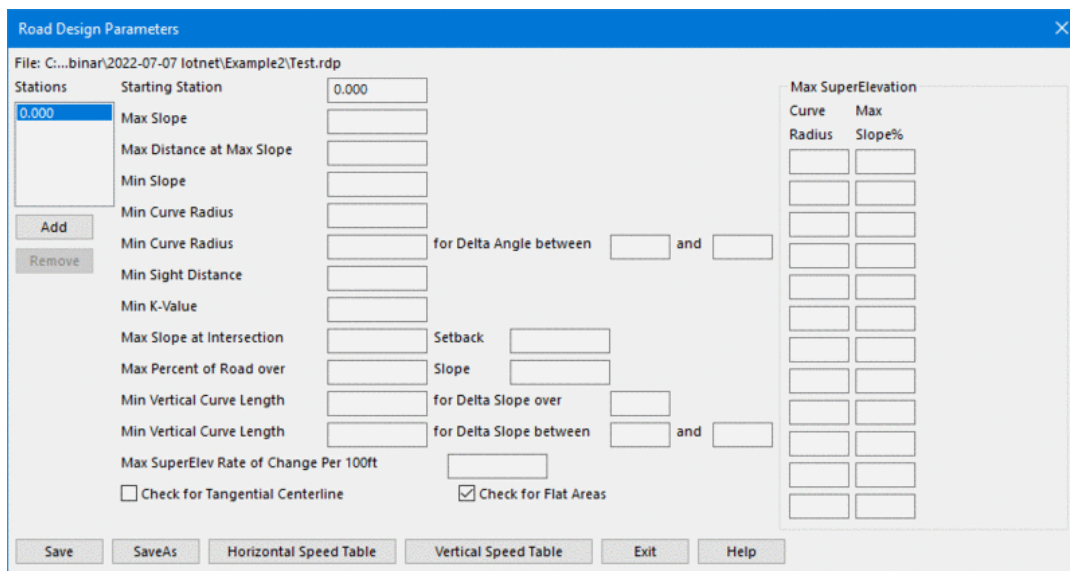


Assign Template Pt Centerline Dialog Boxes

ROW Offsets: The ROW feature draws 2D linework at specified offsets from the centerline. In the dialog, there are settings for the layer for the 2D polyline and the offsets left and right of the centerline. There is also a list of additional offsets to draw. Use the Add, Edit and Remove buttons to setup this list of offsets to draw. The names, offsets and layers for these 2D polylines is stored to a .ROW file.



Road Design Parameters: Pick this button to select an existing or create a new Road Design Parameters (.RDP) file for the Road. This file allows the user to define a set of Road design standards to compare against a roadway design. The Road Network Process function will report a warning when the design is out of compliance with these parameters. The Road Design Parameters can be specific to all stations along a Road or, in the event speed limit or other changes must be applied, a range of stations. The **Edit** button opens the **Road Design Parameters** dialog box. This box is the same as the one used for the **Define Road Design Parameters** command. Please refer to the **Help** files for that command if additional assistance is needed.



Road Design Parameters Dialog Box

Road Stripes: This option draws plan view polylines for road stripes such as double yellow lines along the centerline and dashed white lane lines. See the Draw Road Stripes command for a description of this feature.

Optional Road Output Files

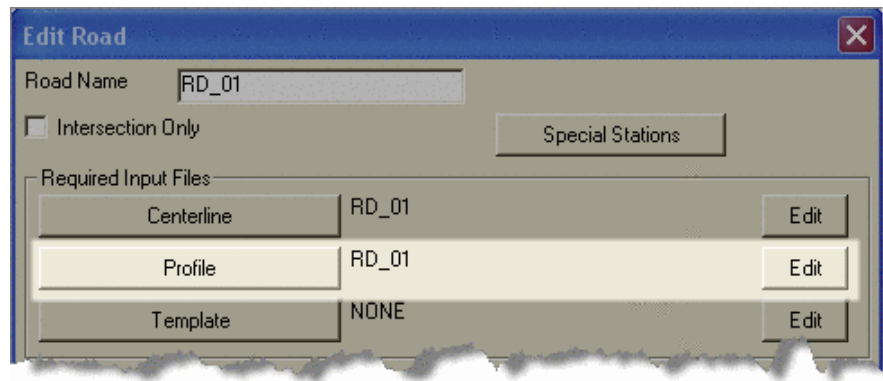
Existing Section File: Pick this button to specify the path and filename for the existing cross-section file to be written. The default filename is set by picking **Output File Defaults** button in the **Output Options** tab of the **Road**

Network Settings dialog box.

Final Section File: Pick this button to specify the path and filename for the final/design cross-section file to be written. The default filename is set by picking **Output File Defaults** button in the **Output Options** tab of the **Road Network Settings** dialog box.

SuperElevation Diagram: Pick this button to specify the path and filename for the SuperElevation Diagram (.SUD) file to be written.

The **Road Profile Editor** is accessible from the **Edit Road** Dialog box.

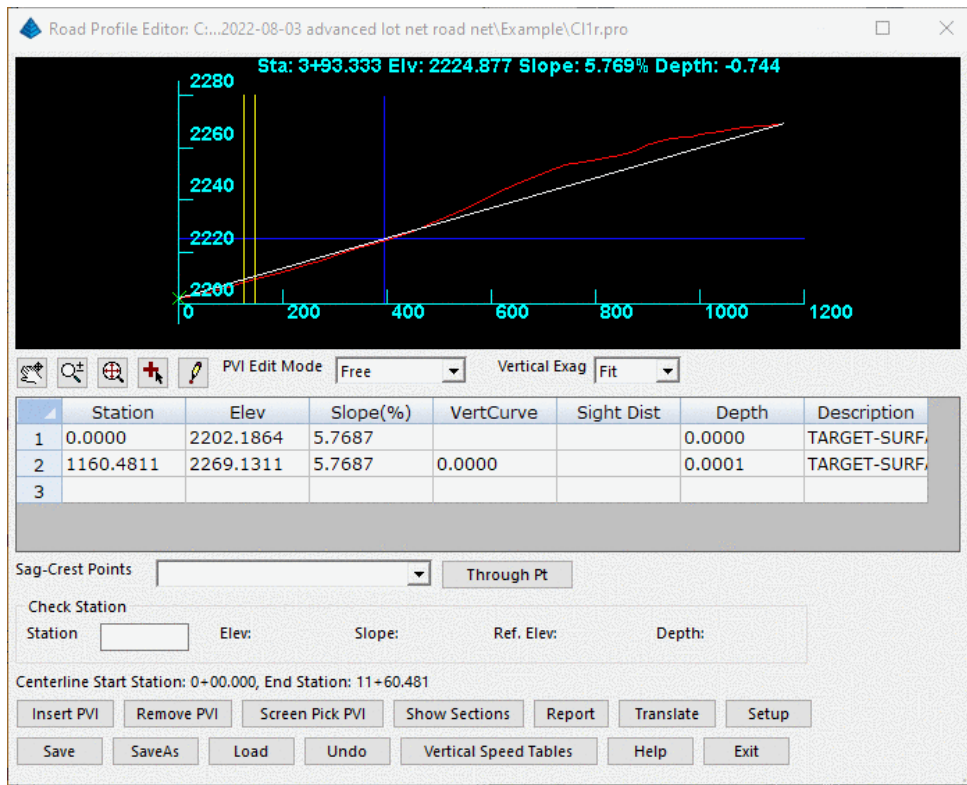


Pick "Edit" to Access the Road Profile Editor

In Carlson's **Road Network** feature, the initial design Profile is automatically generated and has only a starting and ending PVI - with the elevation at both ends tying into existing ground. The crosshairs are locked to the design Profile.

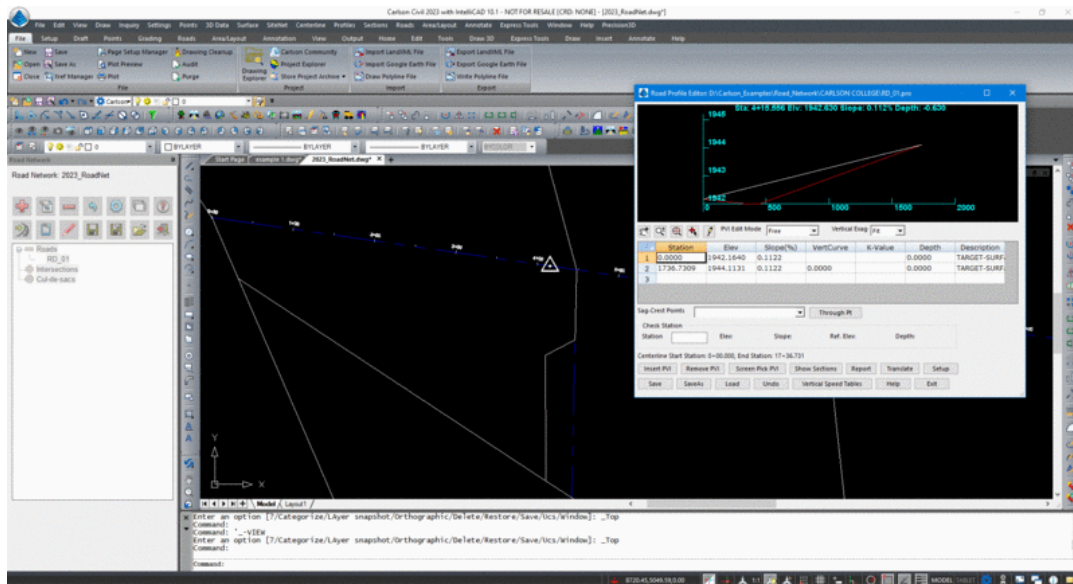
The initial PVIs can be seen in the profile-grid-view where the existing ground Profile is shown in red and the design Profile in white. The initial PVIs are shown in the table-view with the "PVI Description" indicating the PVI elevation is tied to the "TARGET-SURFACE" (existing ground).

The buttons and settings directly below the profile-grid-view allow the user to edit the Profile and adjust the Zoom and Scale factors of the profile-grid-view. The **Insert PVI**, **Remove PVI** and **Screen Pick PVI** buttons at the bottom of the dialog box allow the user to make changes to the Profile using the table-view.



Road Profile Editor

The profile-grid-view provides the user with a dynamic viewer and editor. As the crosshairs move along the design Profile, a "station" symbol on the drawing screen indicates the corresponding position/station along the Centerline. Also, as the crosshairs move along the Profile, the current Station, Elevation, Slope and Depth (between design and existing ground Profiles) are displayed and dynamically updated at the top of the window. The starting and ending stations for the Centerline are displayed above the buttons at the bottom of the window.



Road Profile Editor with Station Indicator in Drawing



Pan, Zoom and Zoom Extents: Use these buttons to change the Zoom factor in the profile-grid-view.

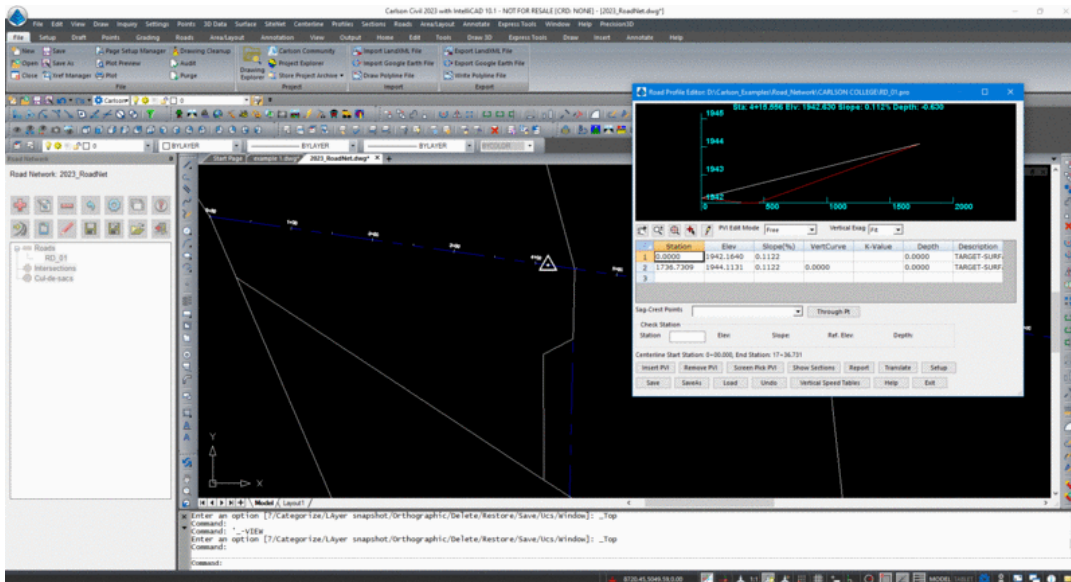


Add PVI: Use this button to "screen pick" the location for a new PVI in the profile-grid-view. After screen picking the new PVI location, the **New PVI** box prompts the user to provide additional design criteria to set the new PVI.

The 'New PVI' dialog box contains the following fields and options:

- Station: 75
- Elevation: 2213.6395
- Slope In (%): 9.5
- Slope Out (%): 100.0
- Vertical Curve: 0.00
- Sight Distance: (empty field)
- Station Resolution Snap: None 1.0 5.0 10.0
- Slope Resolution Snap: None .1 .5 1.0
- Buttons: OK, Cancel, Help

New PVI Dialog Box



Edit PVI and PVI Edit Mode: Use the **Edit PVI** button to change the elevation and station of a PVI in the profile-grid-view by dragging-and-dropping it to a new location. The default **PVI Edit Mode** is "Free" which allows 360-degree motion when dragging-and-dropping the PVI. Other **PVI Edit Mode** options are: **Hold Slope In**, **Hold Slope Out**, **Hold Station** and **Hold Elevation**. The user also can choose to **Hold Vertical Curve Length**, **Hold K-Value** or **Hold Sight Distance** when editing the PVI using drag-and-drop. This setting is controlled in the **Road Profile Settings** dialog box.

Vertical Exag: Use this setting to "Fit" the Profile into the profile-grid-view area of the window or use other pre-defined options such as "x1", "x2", "x5" and "x10" to exaggerate the vertical scale by 1-, 2-, 5- or 10-times.

Sag-Crest Points: After adding one or more vertical curves to the design Profile, a list of the "sag" and "crest" points along the Profile will be listed in the drop-down box.

Through Point: After selecting a PVI in the table-view, pick this button to force a sag or crest point to a specific station and elevation.

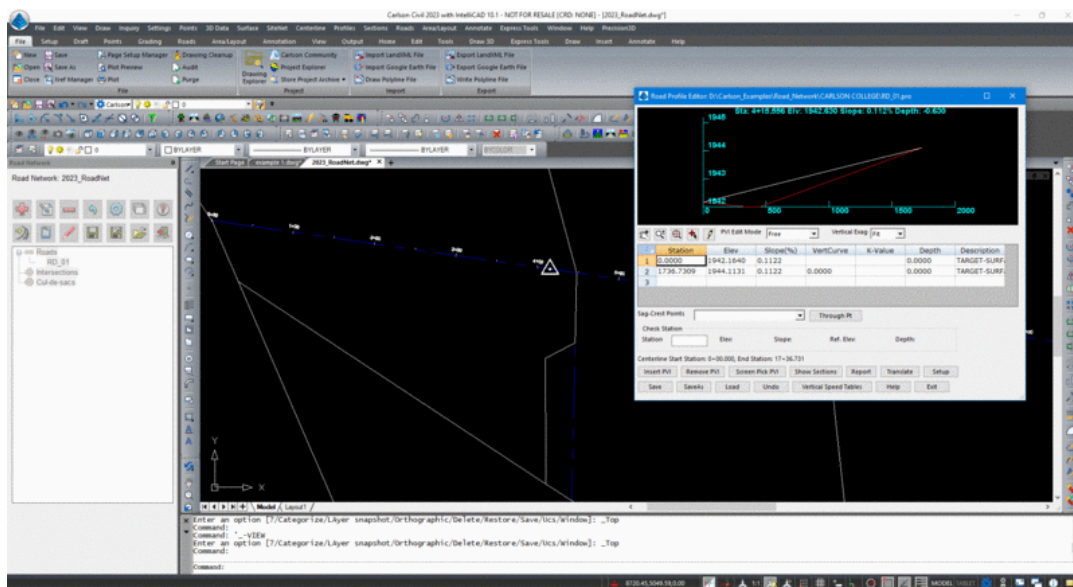
Check Station: To find the precise Elevation, Slope and Reference Elevation (existing ground) for a specific station, enter the station in the text box and *press Enter*.

Insert PVI: Before picking the **Insert PVI** button, the user must use the mouse to select/highlight a cell in the profile table-view. Then, picking the **Insert PVI** button will create a blank row, above the selected row, allowing the user to enter the information for the new PVI.

Remove PVI: Before picking the **Remove PVI** button, the user must use the mouse to select/highlight a cell in the row corresponding to the PVI to be removed. Then, picking the **Remove PVI** button will delete the selected row/PVI from the Profile.

Screen Pick PVI: Picking this button allows the user to change the station of a PVI by screen picking a location in the drawing. Before picking the **Screen Pick PVI** button, the user must use the mouse to select/highlight a cell in the corresponding row of the PVI to be changed. Then, picking the **Screen Pick PVI** button changes the user to the active drawing screen, prompting the user to "Pick PVI Point:" in the drawing area.

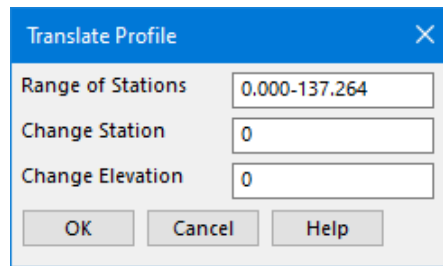
Show Sections: This option is only available if the Template (.TPL) file for the Road has already been specified in the **Edit Roads** dialog box. When picked, the **Show Sections** button will open a "Road Design Section Data" viewer window while keeping the "Road Profile" window open as well. This provides the user a dynamic design environment in which the plan-, profile- and section-views are visible at one time. Additionally, when the "Section" viewer window is open, the notes at the top of the profile-grid-view include the "Cut" and "Fill" end-area at the current station along with the "Cut" and "Fill" volume for the entire Road. These calculations are dynamic and will update if changes are made to the design Profile. Use the Specific Station to check the section at a station. Or move the cursor in the profile preview graphic to change the section station.



style="width: 800px;">

Road Profile View and Section Viewer with Station Indicator in Drawing

Translate: Picking this button will display the **Translate Profile** dialog box and allows the user to change the elevation of the entire Profile or a range of stations along the Profile.



Translate Profile dialog box

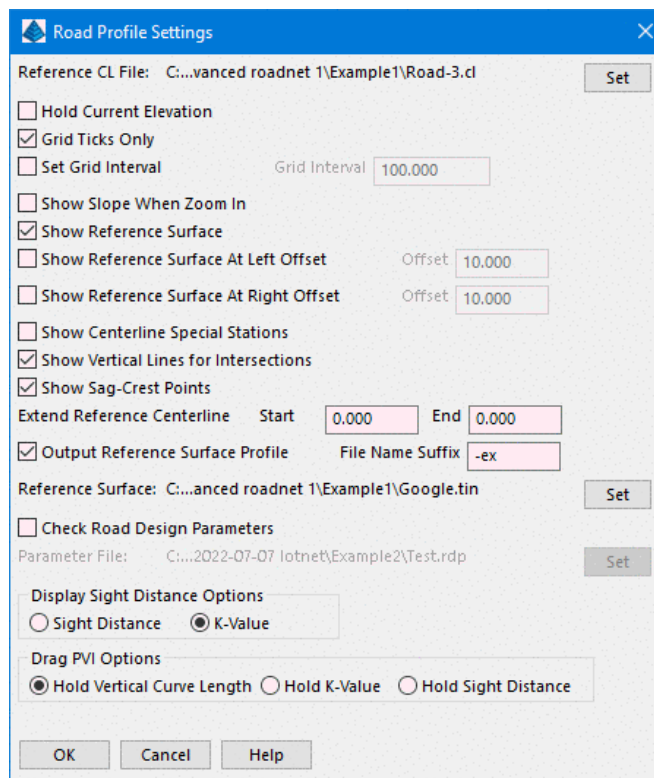
Save: This button saves changes to the Profile (.PRO) file.

Exit: This button exits the **Road Profile Editor** dialog box.

Undo: This button will undo the last change made to the Profile.

Setup: This button opens the **Road Profile Settings** dialog box. See below for more information.

Vertical Speed Tables: Use this button to specify the Vertical Curve Speed Table (.VST) files to use for the design of this Road.



Road Profile Settings Dialog Box

Reference CL File: In the **Road Network** feature, the "Reference CL File" is automatically set to the Centerline (.CL) file associated with the Road.

Hold Current Elevation: When enabled and the station and elevation of a PVI changes, the "Slope Out" of the adjusted PVI will change but the elevation of the next PVI will be left unchanged. Otherwise, if not enabled, the "Slope Out" of the adjusted PVI is held and the elevation of the next PVI is changed.

Grid Ticks Only: When enabled, only grid ticks will be shown in the profile-grid-view. Otherwise grid lines will be used.

Set Grid Interval: If enabled, this option allows the user to manually specify the grid- or grid-tick interval shown in the profile-grid-view.

Show Slope When Zoom In: When enabled, this option allows the user to display the slopes on those vertical

tangents that are long enough to display a slope label when Zoom-ing in closer to the Profile.

Show Reference Surface: When enabled, this option displays the Profile of a "Reference Surface" in addition to the design Profile. The "Reference Surface" is typically the original or existing ground Profile.

Show Reference Surface at Left Offset: When enabled, this option allows the user to see an additional Profile that is offset horizontally from the "Reference Centerline". The offset distance can be specified after the option is enabled.

Show Reference Surface at Right Offset: When enabled, this option allows the user to see an additional Profile that is offset horizontally from the "Reference Centerline". The offset distance can be specified after the option is enabled.

Show Centerline Special Stations: When enabled, critical Centerline stations such as PC, PT, SC, ST, TS and SP are shown in the profile-grid-view.

Show Vertical Lines for Intersections: When enabled, this option will display a vertical line representing the Centerline and Edge of Pavement stations for other Roads in the Road Network.

Show Sag-Crest Points: When enabled, this option displays a marker at the sag and crest points of each vertical curve.

Extend Reference Centerline: When enabled, the user may provide an extended range of stations so as to show Profile data beyond that generated along the associated Centerline (.CL) file. For instance, for a new Road tying into an existing Road (proposed CL file starts at the Intersection of the Centerline of the existing Road) an extended range of stations may be desired in order to see the Profile of the cross-slope, curb, ditch and slope across both sides the existing Road.

Output Reference Surface Profile and Suffix: When enabled, this option will generate an existing ground Profile (.PRO) file and allows the user to specify a suffix for the filename. The defaults for this option are set using the **Output File Defaults** button in the **Output Options** tab of the **Road Network Settings** dialog box.

Reference Surface: The "Reference Surface" is an additional surface Profile shown in the profile-grid-view alongside the design Profile. For the **Road Network** feature, the "Reference Surface" is the surface specified as "Existing Ground" in **Road Network Settings** dialog box.

Check Road Design Parameters: When enabled, this option will compare the current Road design to an established set of design parameters set in a Road Design Parameters (.RDP) file. Please refer to the **Help** files for the **Road Design Parameters** command if additional assistance is needed.

Display Sight Distance Options: Use this radio button to display either a "Sight Distance" or "K-Value" column in the profile-table-view.

Drag PVI Options: Use this radio button to specify the design criteria to "hold" when using the **Edit/Drag PVI** command in the profile-grid-view. The options are to "Hold Vertical Curve Length", "Hold K-Value" or "Hold Sight Distance".

Intersections are created automatically in the **Road Network** feature without any input from the user. Once Intersections are identified, they are listed and managed in the **Intersection** area of the **Road Network: Task Pane**.

Edit: Use this button or right-click on the intersection in the project tree and choose Edit Intersection to display the **Edit Intersection** dialog box and make changes to the **Input Data** and **Output Files** for the selected Intersection. Other changes that can be made to the Intersection design are:

- 1) Changing the Primary/Secondary status of the Roads creating the Intersection,
- 2) Making design changes that apply to the entire Intersection,
- 3) Making design changes that apply to one or more Corners of the Intersection.

Reset: Use this button to overwrite all design changes made to the selected Intersection and reset to the original Intersection design.

As stated above, Intersections are created automatically in the **Road Network** feature without any input from the user. Road Network recognizes and calculates the Intersection using the Centerline (.CL) files associated with the Roads in the Network. If two Roads are added to the Network and they share one or more common point, an Intersection is created and displayed as an Intersection in the **Road Network: Task Pane**.

For all Intersections, one of the two Roads creating the Intersection will be the "Primary" Road and the

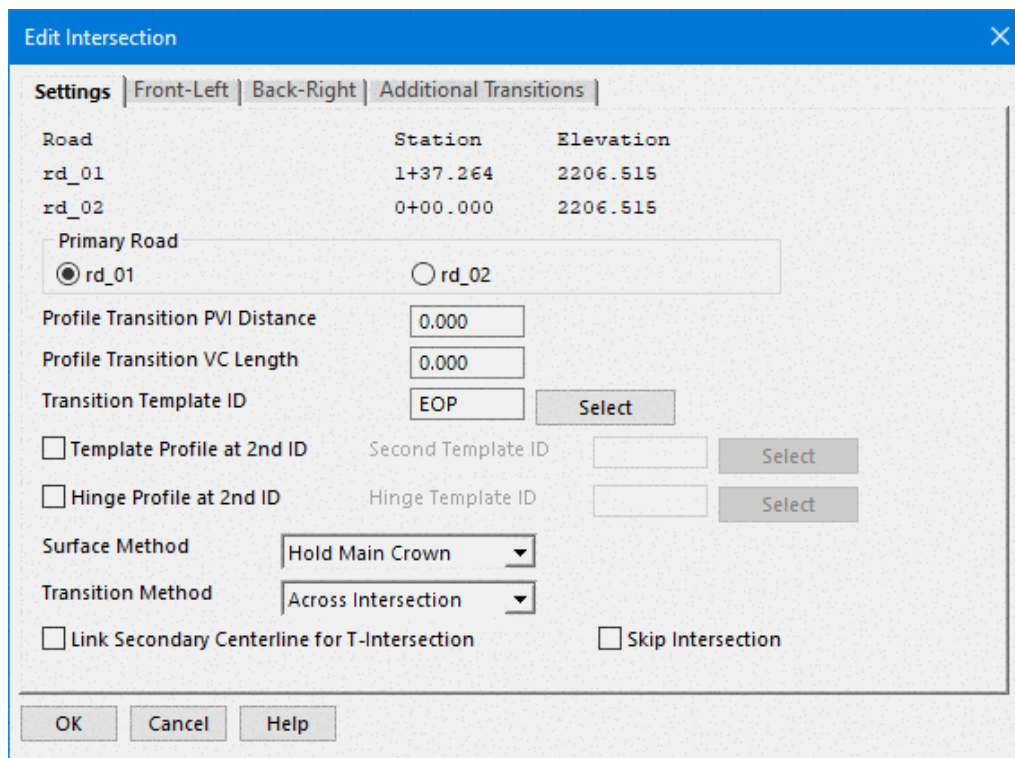
other will be the "Secondary" Road. When setting grade through an Intersection, the Primary Road's Template (.TPL) file takes priority and is used to define the cross-section. The grades of the Secondary Road will adjust to match the Primary Road. Additionally, changes to any of the Primary Road design files - such as the Profile (.PRO) file - will automatically update the affected file(s) of the Secondary Road.

Upon creation of an Intersection, the **Road Network** feature automatically designates one of the Roads as the Primary Road and the other as Secondary. For four-way Intersections, the first Road added to the Road Network will be deemed the Primary Road and the second Road will be Secondary. For T-Intersections, the Road going straight-through the Intersection will be deemed the Primary Road - even if it's added to the Network after the Road that stops at the Intersection. The user can change the Primary Road designation in the **Edit Intersection** dialog box.

Picking the **Edit** button displays the **Edit Intersection** dialog box which has a **Settings** tab and, depending on the type of Intersection, 2 or 4 additional tabs - each representing one **Corner** of the Intersection. The **Corner** tabs are labeled *Front-Right*, *Back-Right*, *Front-Left* or *Back Left*. T-Intersections will have 2 tabs and 4-way Intersections will have 4 tabs.

Intersection Settings

At the top of the **Settings** tab, the station and elevation of the Intersection is shown for all Roads.



The screenshot shows the 'Edit Intersection' dialog box with the 'Settings' tab selected. The dialog has a blue title bar with a close button (X) in the top right corner. Below the title bar, there are four tabs: 'Settings', 'Front-Left', 'Back-Right', and 'Additional Transitions'. The 'Settings' tab is active and contains the following controls:

Road	Station	Elevation
rd_01	1+37.264	2206.515
rd_02	0+00.000	2206.515

Below the table, there are two radio buttons for 'Primary Road': rd_01 and rd_02.

Other settings include:

- Profile Transition PVI Distance:
- Profile Transition VC Length:
- Transition Template ID:
- Template Profile at 2nd ID: Second Template ID
- Hinge Profile at 2nd ID: Hinge Template ID
- Surface Method:
- Transition Method:
- Link Secondary Centerline for T-Intersection
- Skip Intersection

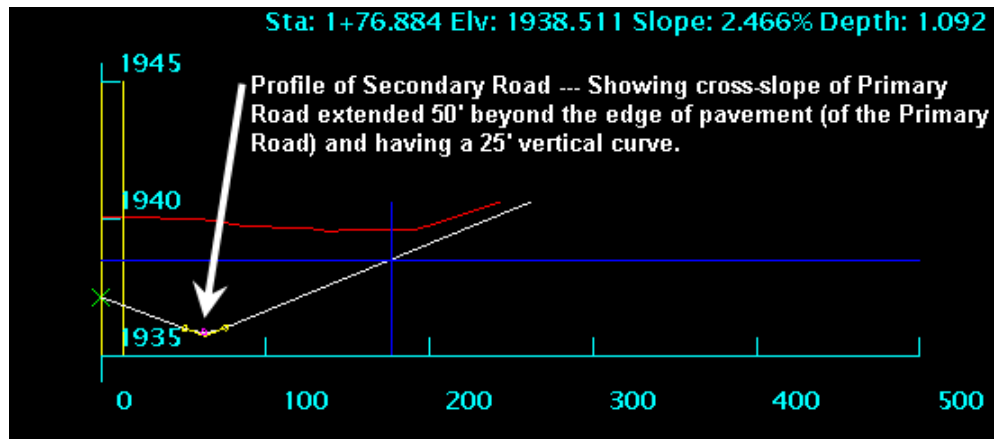
At the bottom of the dialog are three buttons: , , and .

The Settings Tab of the Edit Intersection Dialog Box

Primary Road: Use the radio button to specify the Primary Road of the Intersection.

Profile Transition PVI Distance: This value represents the distance beyond the edge of pavement of the Primary Road (along the Secondary Road Centerline) that the cross-slope of the Primary Road will be extended.

Profile Transition VC Length: This setting allows the user to specify the length of vertical curve to be inserted at the PVI where the extension of the Primary Road's cross-slope and the Centerline of the Secondary Road meet.

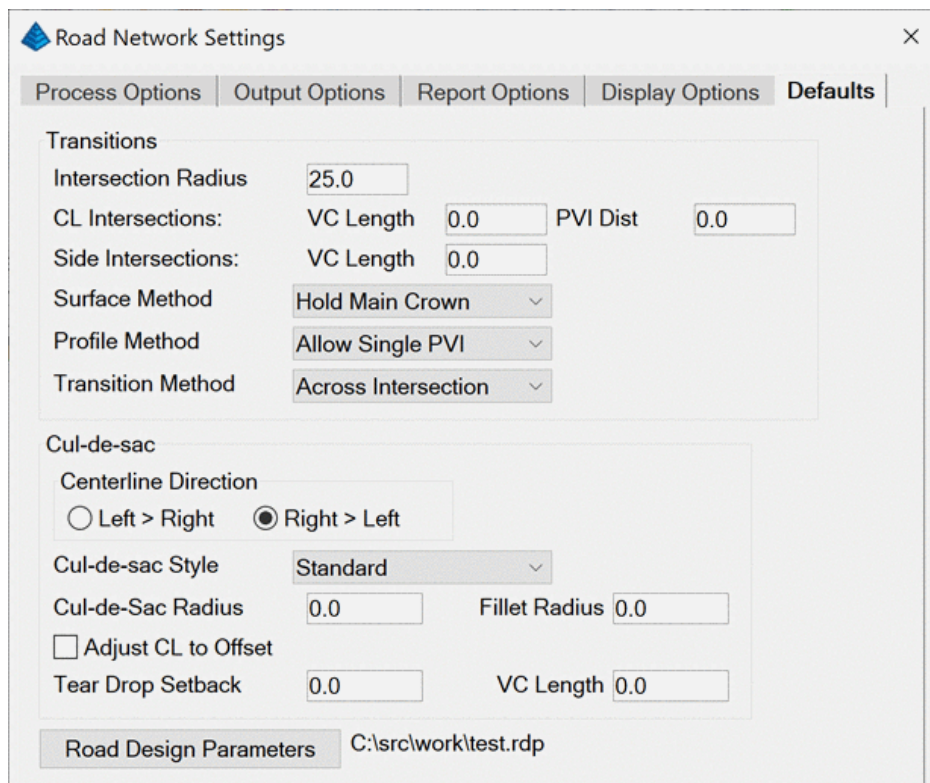


"Profile Transition PVI Distance" and "Profile Transition VC Length"

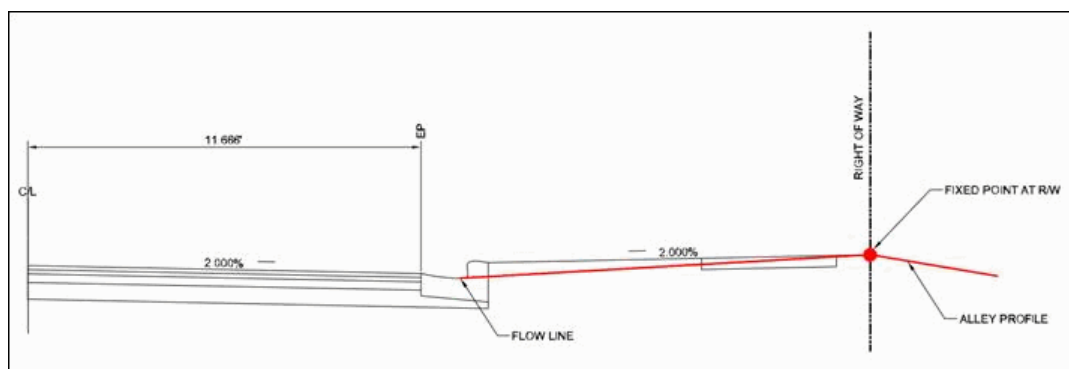
Template ID: This is the point on the cross-section used to define the horizontal (Centerline) and vertical (Profile) alignments around the Corners of the Intersection. Also, the profile for the side road will tie into this Template ID on the main road. The Template ID may be specified as any point on the cross-section - such as edge of pavement (EP) or the back of curb (BC) - as long as it has been defined as a **Template ID** in all of the Template (.TPL) files used to calculate the Intersection. Type the **Template ID** in the text box or use the **Select** button to choose from a list.

Template Profile at 2nd ID: This option sets the template ID for an additional template ID to control in the curb return besides the main ID. When this option is active, each curb return has a button to edit the template point profile for this additional ID. This option applies when the template is transitioning across the intersection on a grade ID besides the main ID.

Hinge Profile and 2nd ID: For the side road profile, this is an optional second point to match from the main road template.



Cross-section of main road showing side (alley) profile tying into single Template ID at flow line



Cross-section of main road showing side (alley) profile tying into Template ID at flow line as well as 2nd Hinge at Right-of-way of main road

Surface Method: See the Transition Defaults section above for details on this setting.

Transition Method: See the Transition Defaults section above for details on this setting.

Link Secondary Centerline for T-Intersection: When this option is enabled, changes to the Centerline (.CL) file of the Primary Road will, if necessary, force the Centerline of the Secondary Road to be extended or trimmed in order to keep the Intersection intact.

Skip Intersection: This option skips creating the curb returns for this intersection. This option applies for crossings like an overpass or railroad.

Note: The default value for several design criteria such as Intersection radius and length of vertical curve can be set in the **Transition Defaults** tab of the **Road Network: Settings** dialog box.

Corner tabs - *Front-Right, Back-Right, Front-Left, Back-Left*

Depending on the type of Intersection ("T" or 4-way), there will be either 2 or 4 additional tabs available in this dialog box. Each of these tabs represent a Corner of the Intersection and allows the user to specify horizontal and vertical **Input Data** and **Output Files** specifically for that Corner.

The screenshot shows the 'Edit Intersection' dialog box with the 'Front-Left' tab selected. The 'Input Data' section includes a 'Radius' field set to 25, a checked 'Tie To Existing' checkbox, an 'Edit Profile' button, a 'Reset' button, a 'VC Length' field set to 0.000, and an 'Edit 2nd Profile' button. Below this is an 'Edit Template Transition' button, a 'Profile Method' dropdown menu set to 'Allow Single PVI', a 'Template Grade Table' button, and the text 'NONE'. An 'Edit' button is located to the right of the 'Template Grade Table' section. The 'Output Files' section contains five rows, each with a button and the text 'NONE': 'Centerline', 'Profile', 'Existing Section File', 'Rock Section File', and 'Final Section File'. At the bottom of the dialog are 'OK', 'Cancel', and 'Help' buttons.

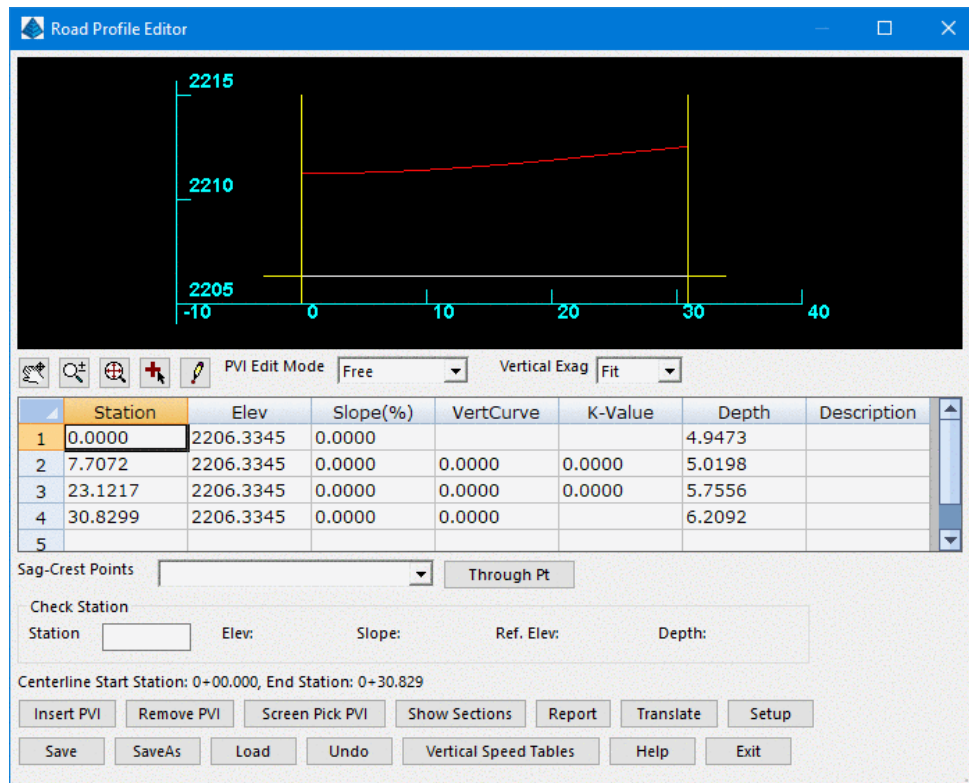
One of the "Corner" Tabs of the Edit Intersection Dialog Box

Intersection Input Data

Radius: Use this value to specify the radius of the curve for this Corner of the Intersection. The **Intersection Template ID** specified in the **Intersection Settings** tab of this dialog box determines the point on the cross-section being affected by this setting.

Tie to Existing: Enable this option to keep cut and fill slopes from projecting to the existing ground through the Intersection. In areas of steep cut or fill, this setting helps avoid overlapping Road and Intersection tie slopes.

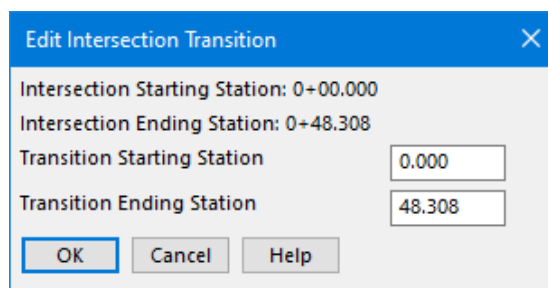
Edit Profile: Pick this button to open the **Road Profile Editor** and make changes to the Profile for this Corner of the Intersection. The **Intersection Template ID** specified in the **Intersection Settings** tab of this dialog box determines the point on the cross-section being represented in the Profile Editor. See **Road Network: Road Profile Editor** for more **Help** with this feature.



Edit Profile for a Corner of an Intersection

Reset: Use this button to overwrite all edits to the Profile of the Corner of the Intersection and reset to the original Profile.

Edit Template Transition: Pick this button to display the **Edit Intersection Transition** dialog box. This allows the user to control the stations for transitioning through the Intersection from a Template on one Road to a different Template on another Road. These Transition stations only apply when the Roads in an Intersection have been assigned different Template (.TPL) files.



Edit Intersection Transition Dialog Box

In the Intersection Transition Dialog Box... The Starting and Ending Stations of the Intersection transition are displayed at the top of the dialog box.

Transition Starting Station: This is the station at which the Primary Road Template ends.

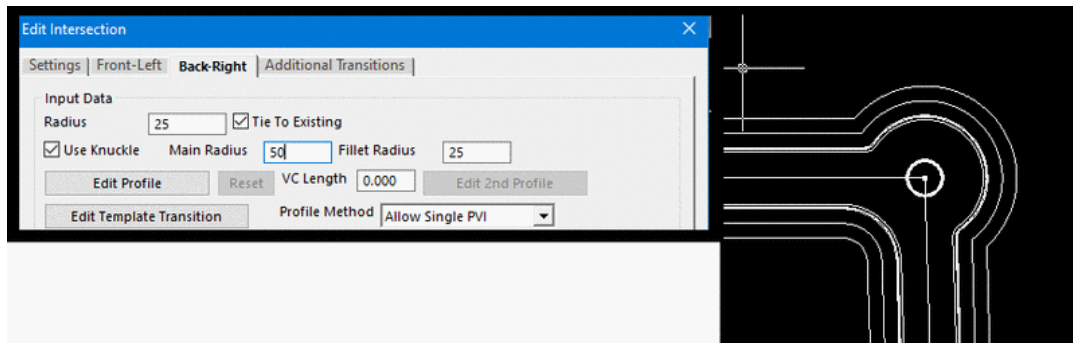
Transition Ending Station: This is the station at which the Secondary Road starts.

Profile Method: When the difference in grade at the Intersection between the Primary Road and the Secondary Road is too severe, two intermediate PVI's must be inserted into the Profile of the Corner of the Intersection in order to properly transition from one Road to another. In some cases, the transition is possible using only one

intermediate PVI in the Corner Profile. **Allow Single VC:** If this option is enabled and if the intersecting grades allow it, only one intermediate PVI will be inserted. **PVIs for Grades In/Out:** If this option is enabled, two intermediate PVIs will be inserted regardless of the intersecting grades. **No Automatic PVI:** This method make the program skip creating any PVI and the default profile will directly connect the starting and ending points of the profile.

Template Grade Table: Pick this button to select an existing or create a new Template Grade Table (.TGT) file defining the grades for the Corner of the Intersection. This file allows the user to define specific slopes and distances for one or more **Template IDs** that have been assigned in the Template (.TPL) file. The **Edit** button opens the **Template Grade Table Editor**. This Editor is the same as the one used for the **Template Grade Table** command. Please refer to the **Help** files for that command if additional assistance is needed.

"L" Intersection with Knuckle: When two centerlines connect at a right angle for an "L" intersection, there is a **Use Knuckle** option for the outside corner that can be used to make a knuckle bulb.



Intersection Output Files

Centerline: Pick this button to output a Centerline (.CL) file representing the horizontal alignment around this Corner of the Intersection. The **Intersection Template ID** specified in the **Intersection Settings** tab determines the point on the cross-section exported to the Centerline (.CL) file.

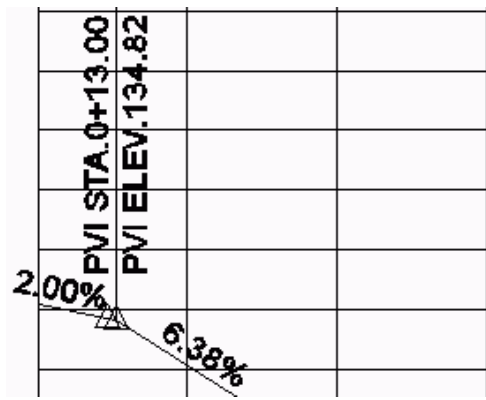
Profile: Pick this button to output a Profile (.PRO) file representing the vertical alignment around this Corner of the Intersection. The **Intersection Template ID** specified in the **Intersection Settings** tab determines the point on the cross-section exported to the Profile (.PRO) file.

Existing Section File: Pick this button to output an Existing Section (.SCT) file for this Corner of the Intersection.

Final Section File: Pick this button to output a Final Section (.SCT) file for this Corner of the Intersection.

Additional Transitions tab

Additional Profile Transition Distance: This option adjusts the transition PVI station on the side profile. The transition station starts as the offset of the Template ID on the main road. The cross slope of the main road is used up to the transition station. For example, if the Template ID is for edge of pavement up to the gutter pan at 11.67 and the side profile needs to match the main crown up the flow line at 13.00, then the Additional Transition Distance should be set to 1.33.



Additional CL Distance (Front Main, Back Main, Left Side, Right Side): These options allow you to extend the station range of the intersection. By default the intersection station range is between the PC points where the intersection arcs begin.

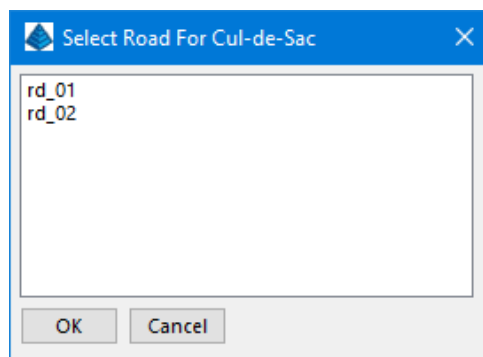
Cul-de-Sacs may be added to any Road in the Network and are managed in the **Cul-de-Sac** area of the **Road Network: Task Pane**.

Add: Pick the Add button or right-click on Cul-de-sacs in the project tree and choose Add to display a list of Roads in the Network and prompt the user to "**Select Road for Cul-de-Sac**".... After selecting the Road, the **Edit Cul-de-Sac** dialog box is displayed allowing the user to specify the **Input Data** and **Output Files** for the Cul-de-Sac.

Edit: Use this button to display the **Edit Cul-de-Sac** dialog box and make changes to the **Input Data** and **Output Files** for the selected Cul-de-Sac.

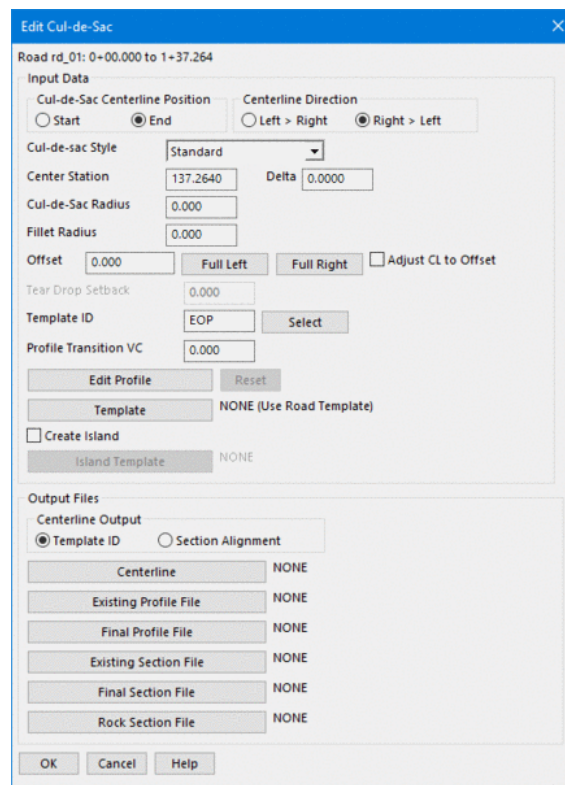
Remove: Use this button to **Remove** the selected Cul-de-Sac from the Road.

Add: Picking this button displays a dialog box listing the Roads in the Network and prompting the user to **Select Road for Cul-de-Sac**.



Select Road for Cul-de-Sac

After choosing the Road and picking the **OK** button, the **Edit Cul-de-Sac** dialog box is displayed.



Edit Cul-de-Sac Dialog Box

Cul de Sac Input Data

Cul-de-Sac Centerline Position: Use this radio button to specify whether the Cul-de-Sac is drawn at the starting or the ending station of the Centerline.

Centerline Direction: This setting applies only if the horizontal alignment of the Cul-de-Sac is to be saved externally as an **Output Centerline (.CL) file**. If so, this setting determines which end of the Cul-de-Sac is the starting and which is the ending station of the new Centerline (.CL) file.

Cul-de-Sac Style: Standard: This style has a rounded bulb and fillet curves to connect with the road. **Tear Drop Mode:** Enabling this option creates a longer transition between the Road and the Cul-de-Sac. When enabled, a value larger than the Cul-de-Sac Radius must be entered as the Setback. An example of a "Tear Drop" Cul-de-Sac having a 45' radius and 75' setback is shown below. **Straight:** This style simply connects the left and right sides of the road which is a way to create curb at the end of the road. This style can be combined with a short end road to make a T-intersection into a hammerhead type cul-de-sac.

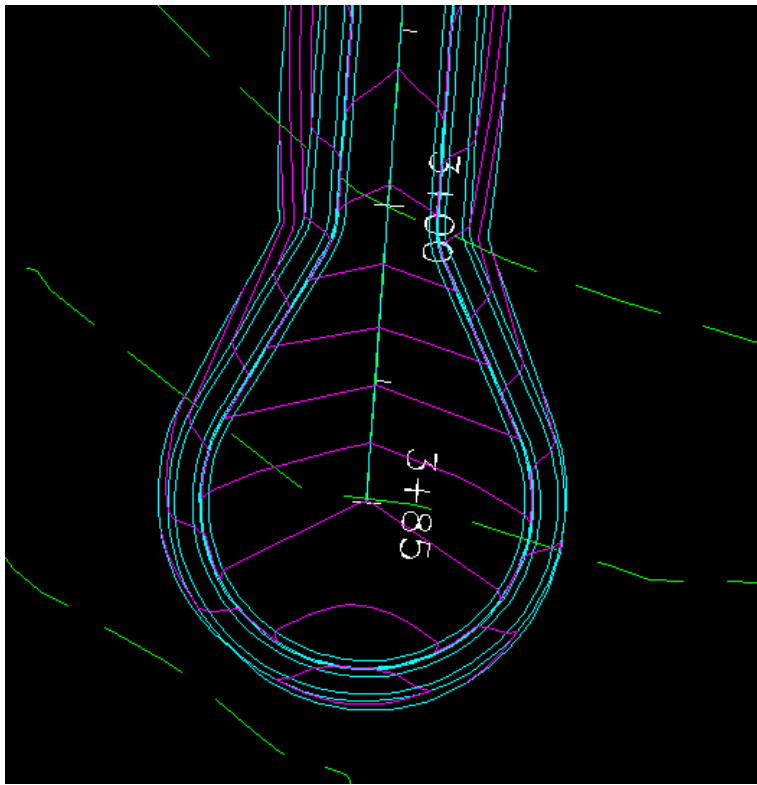
Center Station: Use this setting to precisely locate the center of the Cul-de-Sac along the Road Centerline. By default, the **Center Station** is the starting or ending station of the Centerline depending on whether the user has chosen **Start** or **End** as the desired **Cul-de-Sac Centerline Position**. The station for the center of the Cul-de-Sac may also be entered in the text box or may be specified using a **Delta** value. When using the **Delta** option, the Cul-de-Sac will be shifted the specified distance along the Centerline.

Cul-de-Sac Radius: Use this value to specify the radius of the Cul-de-Sac bulb. The **Cul-de-Sac Template ID** determines the point on the cross-section being affected by this setting.

Fillet Radius: Use this value to specify the radius of the curve that transitions between the Road and the Cul-de-Sac. The **Cul-de-Sac Template ID** determines the point on the cross-section being affected by this setting.

Offset: When set to "0", this setting places the center of the Cul-de-Sac on the Centerline of the Road. Setting this value to a negative(-), greater than "0" value will shift the center of the Cul-de-Sac left of the Centerline by that distance. A positive, greater than "0" value will shift it to the right by that distance.

Adjust CL to Offset: This option adjusts the end of the road centerline to end at the cul-de-sac offset point.



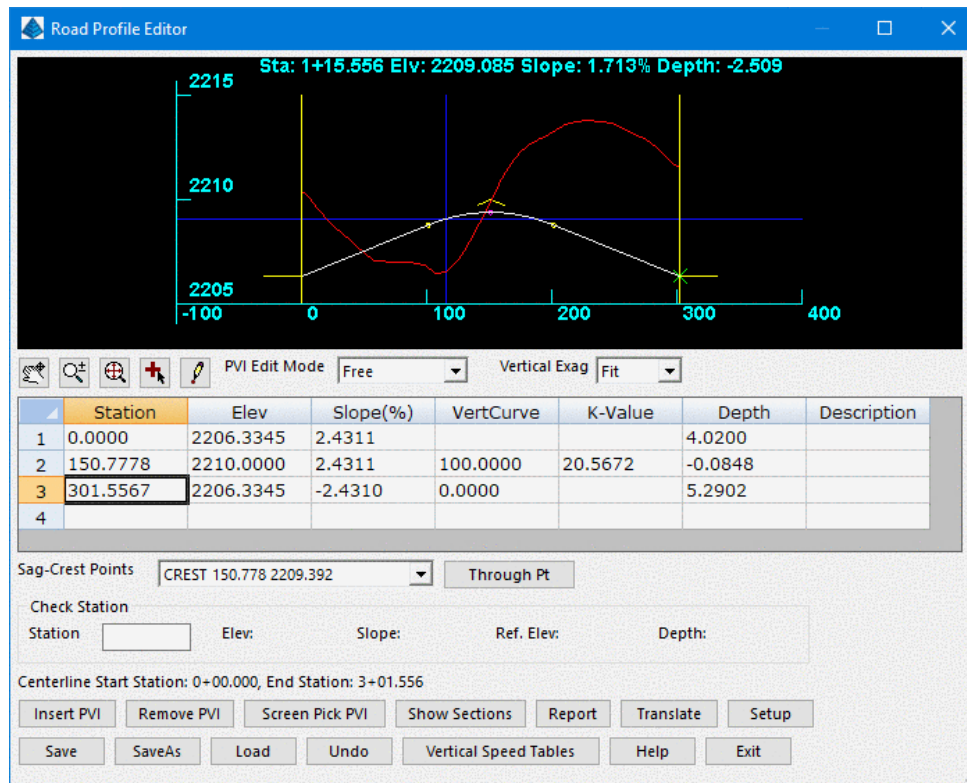
Example of Tear Drop Cul-de-Sac

Template ID: This is the point on the cross-section used to define the horizontal (Centerline) and vertical (Profile) alignments around the bulb of the Cul-de-Sac. The Template ID may be specified as any point on the cross-section - such as edge of pavement (EP) or the back of curb (BC) - as long as it has been defined as a **Template ID** in the Template (.TPL) file used for the Road. Type the **Template ID** in the text box or use the **Select** button to choose from a list.

Profile Transition VC: When adding a Cul-de-Sac to the Road Network, the Profile around the Cul-de-Sac is automatically generated having 3 PVI's - one on each end connecting to the Road and one at the mid-point of the alignment. The **Profile Transition VC** setting is the default length of vertical curve inserted at the middle PVI of the Profile. As shown below, adding a vertical curve at this PVI can have a significant, positive impact on the resulting surface model and contours of the Road Network.

Effect of Adding a Vertical Curve to Cul-de-Sac Profile

Edit Profile: Pick this button to open the **Road Profile Editor** and make changes to the Profile of the Cul-de-Sac. The **Cul-de-Sac Template ID** determines the point on the cross-section being represented in the Profile Editor. See **Road Network: Road Profile Editor** for more **Help** with this feature.



Edit Profile for a Cul-de-Sac

Reset: Use this button to overwrite all edits to the Profile of the Cul-de-Sac and reset to the original Profile.

Template: Use this button to browse to and select an existing Cul-de-Sac Template (.TPL or .TSF) file. Specifying a different Template than the main Road allows the user to define different features for the Cul-de-Sac area such as sidewalk and curb.

Create Island: This option creates a circular island in the middle of the cul-de-sac. The template (.TPL) file uses the grade in the template for the island radius and the template grade slope for the island slope. The island template file does not use the cut/fill slopes.

Cul de Sac Output Files

Centerline: Pick this button to output a Centerline (.CL) file representing the horizontal alignment around the Cul-de-Sac. The **Cul-de-Sac Template ID** determines the point on the cross-section exported to the Centerline (.CL) file.

Existing Profile: Creates a profile of existing ground along the cul-de-sac centerline.

Final Profile: Pick this button to output a Profile (.PRO) file representing the vertical alignment around the Cul-de-Sac. The **Cul-de-Sac Template ID** determines the point on the cross-section exported to the Profile (.PRO) file.

Existing Section File: : Pick this button to output an Existing Section (.SCT) file for the Cul-de-Sac.

Final Section File: Pick this button to output a Final Section (.SCT) file for the Cul-de-Sac.

Rock Section File: Creates a cross section (.SCT) file for the rock surface.

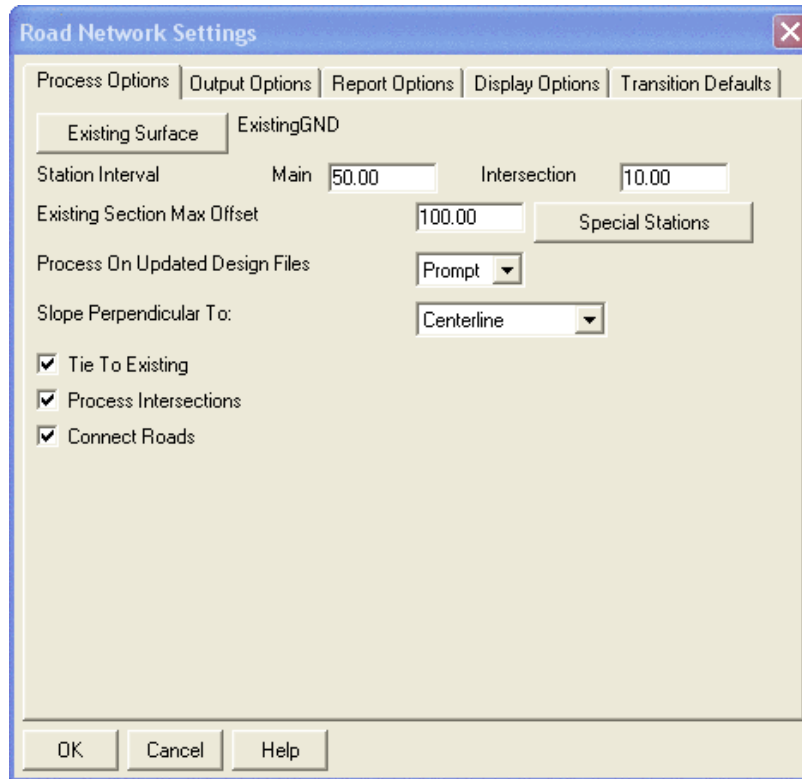
Note: Driveways around a cul-de-sac can be easily added simply by drawing polylines for their centerlines and snapping them to the EOP of the cul-de-sac.

Step 1: Start Road Network and Configure Settings

Open a Drawing (.DWG) file containing the 2D zero-elevation polylines representing Road Centerlines for the project. Start the **Road Network** command and create a **New Road Network** (.RDN) file. After creating the Road

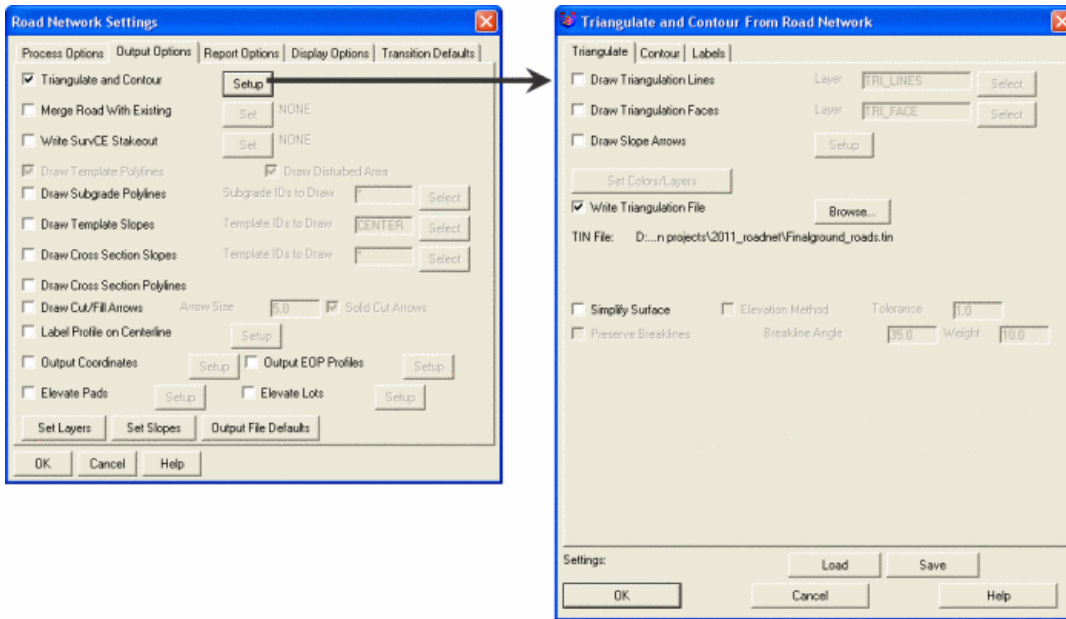
Network file, the **Road Network Task Pane** loads as a docked dialog-box on the left side of the drawing screen.

Configure the Road Network by *picking the **Settings** button* and displaying the **Road Network Settings** dialog box. In the **Process Options** tab, *pick the **Existing Surface** button* and browse to and select the Existing Ground Surface (.TIN or .FLT) file to be used for the project.



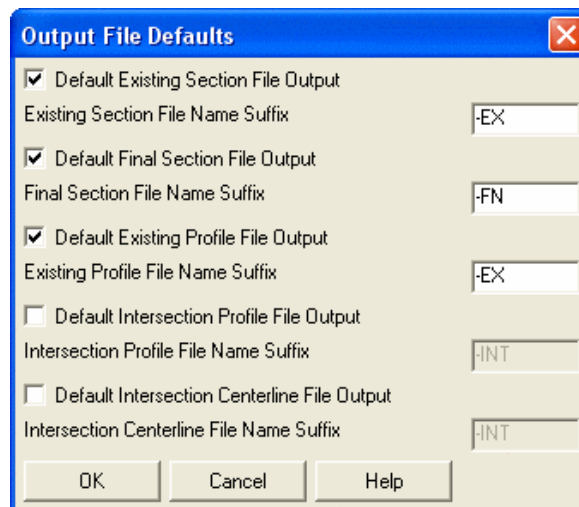
Process Options Tab

Next, *switch to the **Output Options** tab* and *pick the **Setup** button* next to **Triangulate and Contour**. *Select the **Write Triangulation File** option* and then *pick the **Browse** button* to set the path and filename for the design Surface (.TIN) file for the Roads.



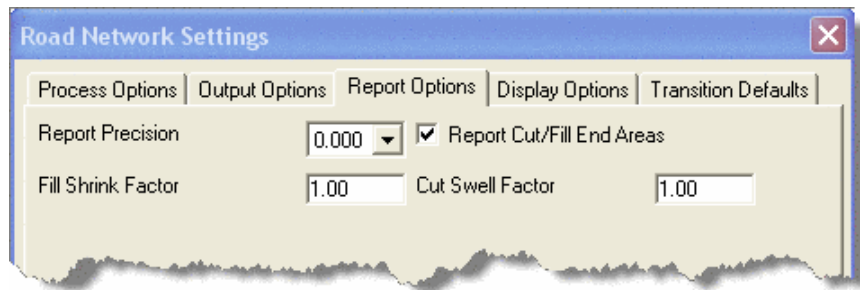
Output Options Tab

Also in the **Output Options** tab, pick the **Output File Defaults** button to display the **Output File Defaults** dialog box. Pick the **Output File Defaults** button to specify additional Centerline (.CL), Profile (.PRO) and Section (.SCT) files to be saved when Processing the Road Network.

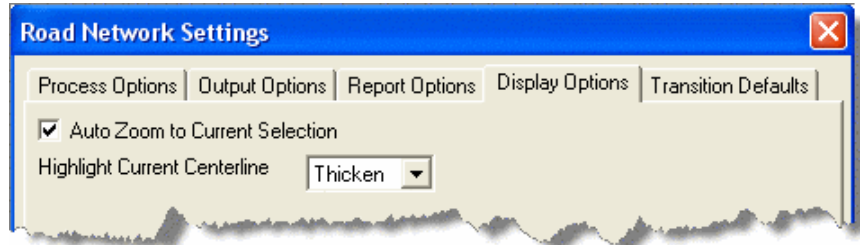


Output File Defaults Dialog Box

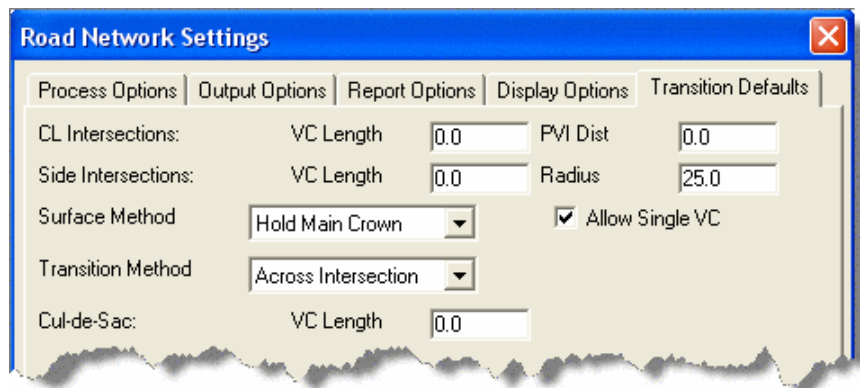
Next, review the **Report Options**, **Display Options** and **Transition Defaults** tabs of the **Road Network Settings** dialog box and make any necessary changes.



Report Options Tab



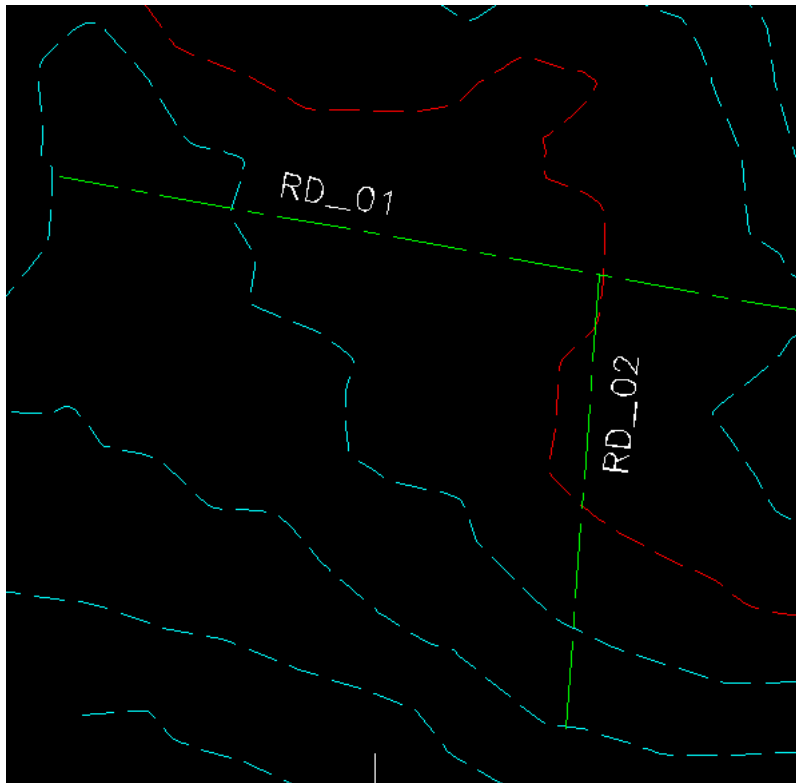
Display Options Tab



Transition Defaults Tab

Pick the **OK** button to close the **Road Network Settings** dialog box and then pick the **Save** button on the **Task Pane** to save the settings to the Road Network (.RDN) file.

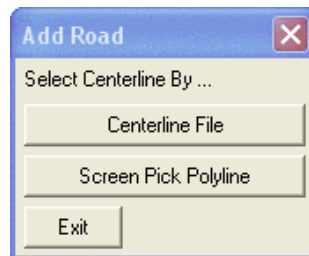
Step 2: Add Roads to the Network



RD.01 and RD.02

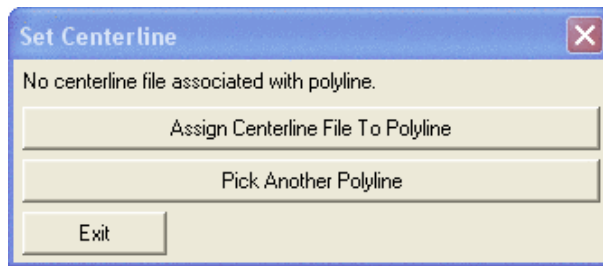
In the project tree, highlight Roads and right-click and choose Add Road.

After picking the **Add** button, the **Add Road** dialog box provides two methods for adding a Road to the Network. *Pick the **Screen Pick Polyline** button.*



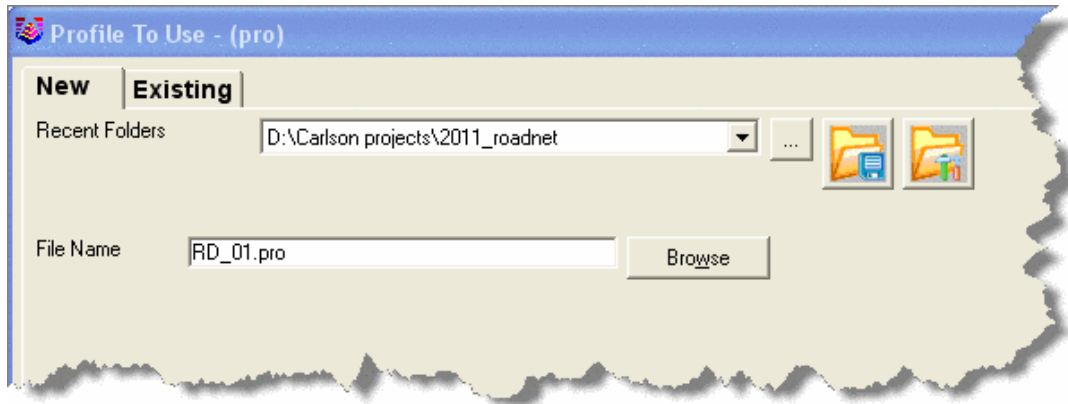
Specify Method to Use to Add Road

The prompts then switch to the Command: line where you are prompted to *Select Centerline Polyline* in the drawing. At the next prompt, *pick the **Assign Centerline File to Polyline** button* and set the path and filename for the new Centerline (.CL) file.



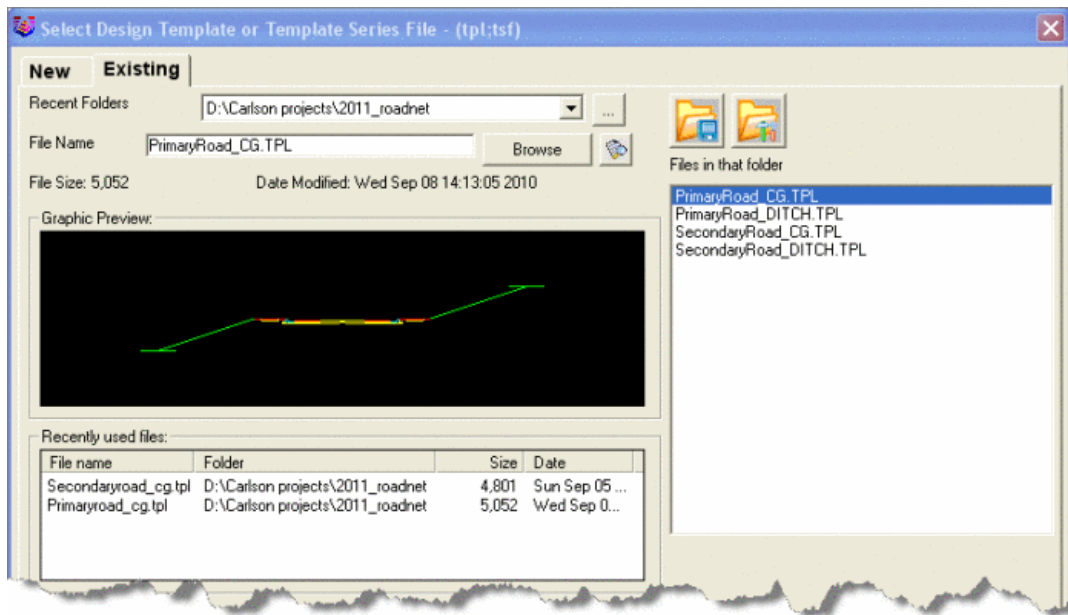
Set Centerline Dialog Box

Immediately after creating the new Centerline file, the **Profile to Use** file dialog box is displayed. In this box, you must set the path and filename for the proposed Profile (.PRO) file for the Road. By default, the new Profile (.PRO) file is named the same as the Centerline (.CL) file.



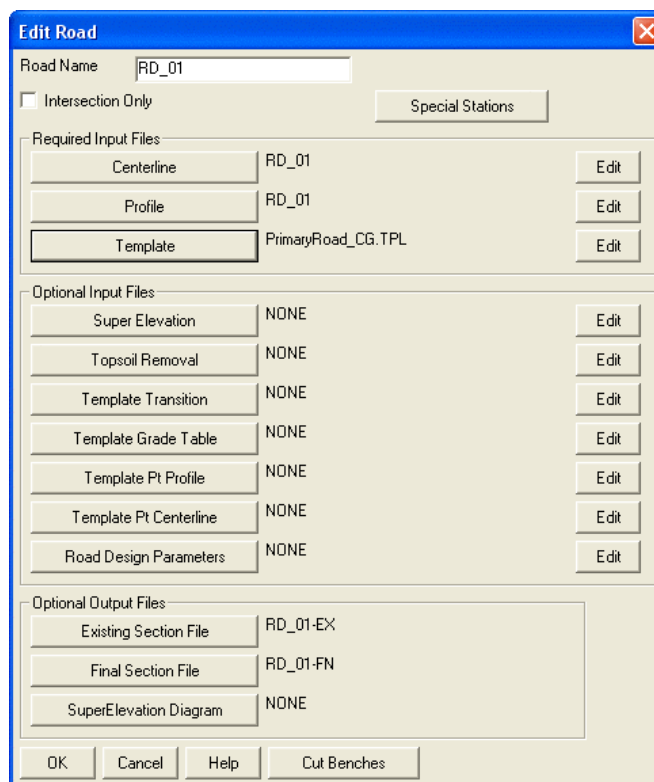
Profile to Use File Dialog Box

After specifying the Centerline (.CL) and Profile (.PRO) files for the Road, the **Edit Road** dialog box is displayed. The only other **Required Input File** is a Template (.TPL) file. *Pick the **Template** button* to browse to and select the desired Template file.



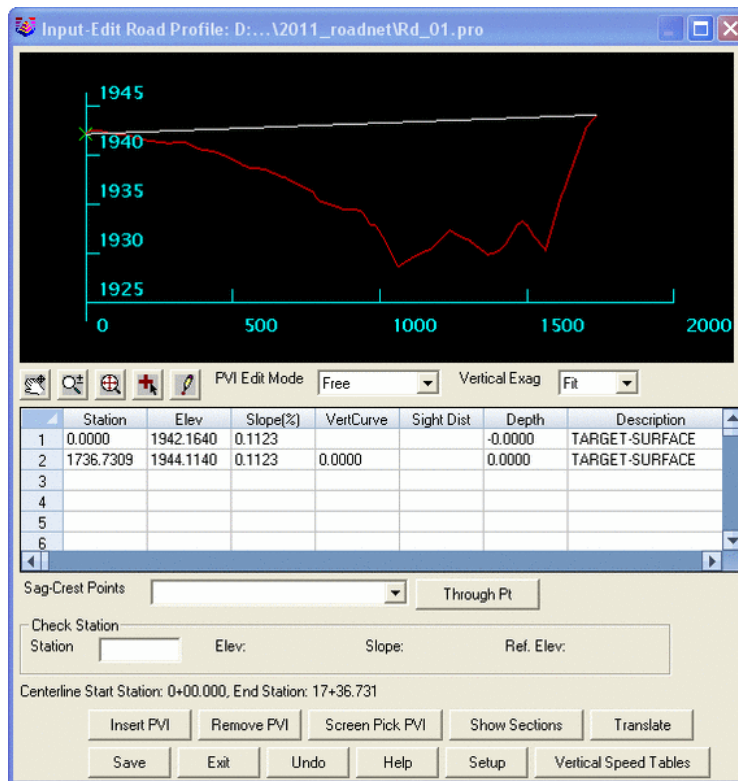
Select Template (.TPL) file Dialog Box

The **Edit Road** dialog box serves as the "manager" for all files relating to the specific Road. The **Edit** button in the **Road Name** area of the **Road Network: Task Pane** also displays the **Edit Road** dialog box.




Edit Road Dialog Box

Pick the **Edit** button to the right of the **Profile** button to open the **Road Profile Editor**.



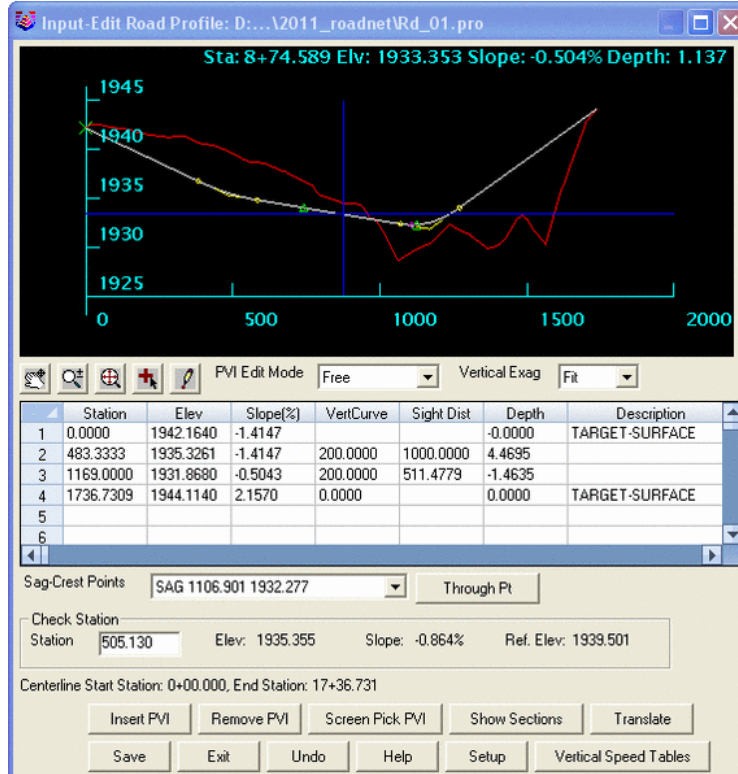
Road Profile Editor

In Carlson's **Road Network** feature, the initial design Profile is automatically generated and has only a starting and ending PVI - with the elevation at both ends tying into existing ground. The movement of the crosshairs is locked to the design Profile. The initial PVIs can be seen in the profile-grid-view where the existing ground Profile is shown in red and the design Profile in white. The initial PVIs are shown in the table-view with the "PVI Description" indicating the PVI elevation is tied to the "TARGET-SURFACE" (existing ground).

Pick the  **Add PVI** button to create a new PVI by screen-picking a point in the profile-grid-view at the top. After picking the **Add PVI** button, the **New PVI** dialog box is displayed.

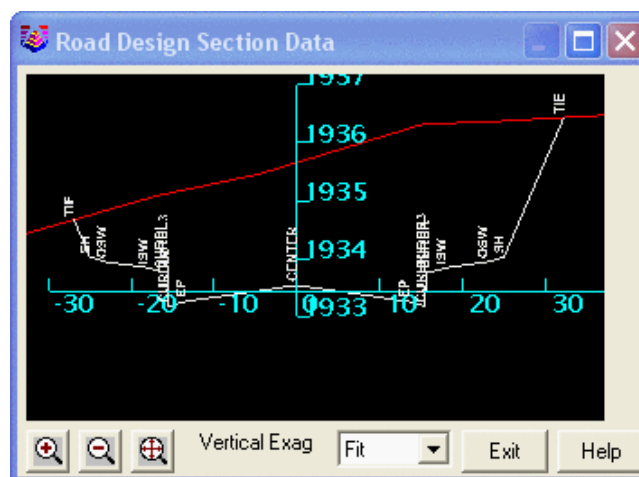
New PVI Dialog Box

Enter a length for a vertical curve or change other settings as desired and then pick the **OK** button. Repeat as needed for additional PVIs and vertical curves.



Road Profile Editor

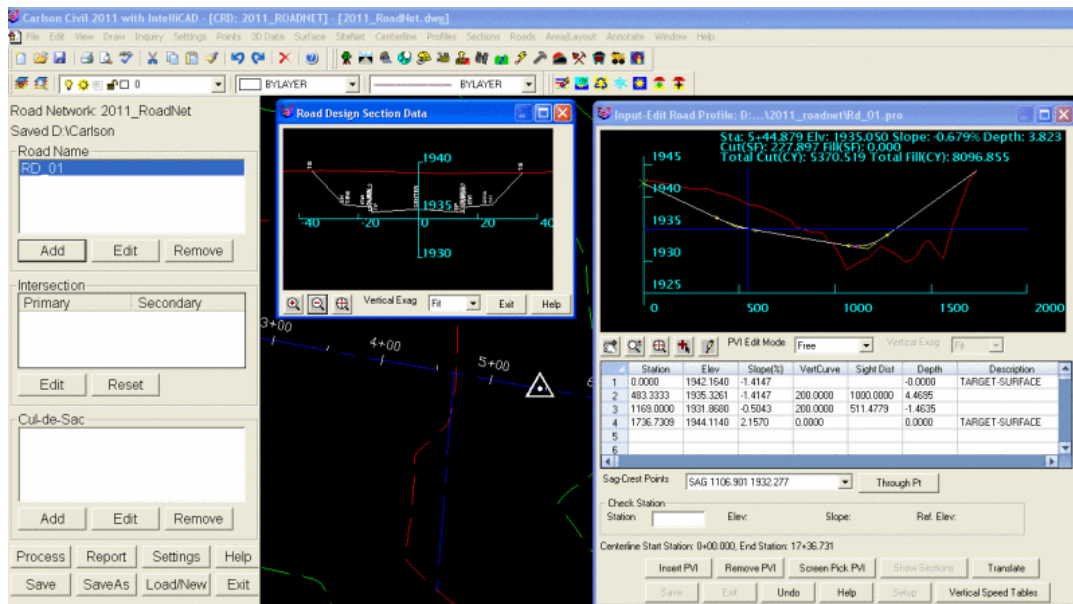
Pick the **Show Sections** button at the bottom of the **Road Profile Editor** to display a **Section View** of the Road. Moving your crosshairs along the design Profile dynamically updates the **Section View**.



Section Viewer

When the **Section View** window is open and active, the **Road Profile Editor** also remains open and active. If you

position the **Road Profile Editor** and the **Section View** window so that the drawing view of the Road is unobscured, you can move your crosshairs along the design Profile and have a dynamic design environment allowing you to see the plan-, profile- and section-views at one time. Additionally, when the **Section View** window is open, the notes at the top of the profile-grid-view include the "Cut" and "Fill" end-area at the current station along with the "Cut" and "Fill" volume for the entire Road. These calculations are dynamic and will update if changes are made to the design Profile.



Road Profile View and Section Viewer with Station Indicator in Drawing

Pick the **Exit** button to close the **Section Viewer** and then pick the **Save** button in the **Road Profile Editor** to save changes to the Profile (.PRO) file. Pick the **Exit** button to close the **Road Profile Editor**.

Repeat the steps above to define additional Roads in the Network.

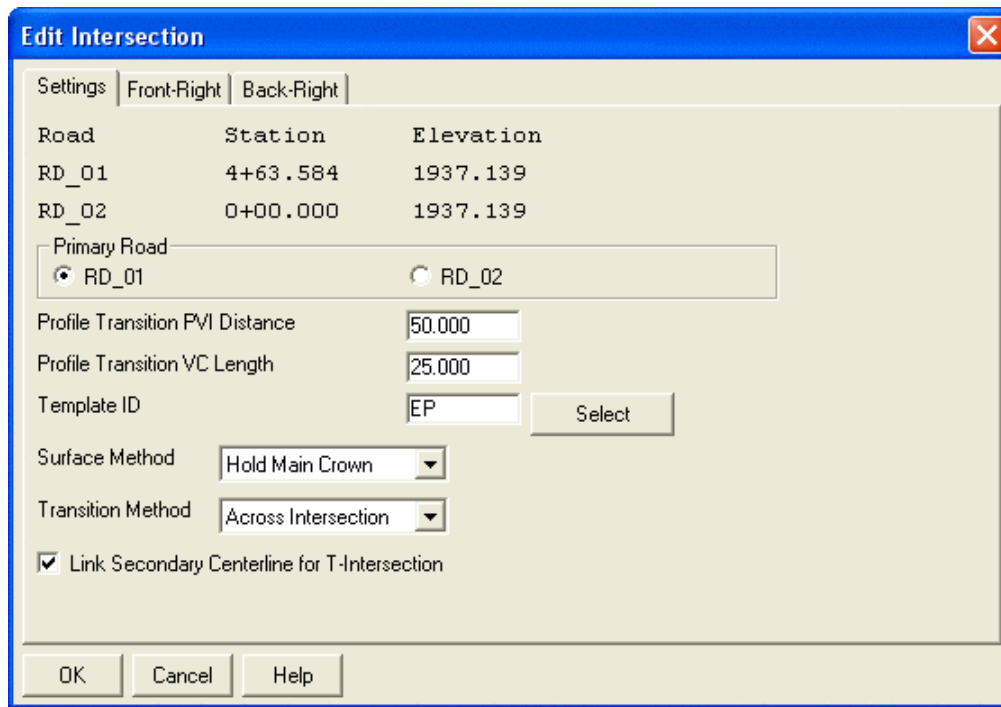
See **Road Network: Adding and Editing Roads** if you need additional assistance.

Step 3: Adding and Editing Intersections

After Adding the next Road, the **Road Network** command recognizes the creation of an Intersection and the Primary and Secondary Roads are displayed in the **Intersection** area of the project tree.

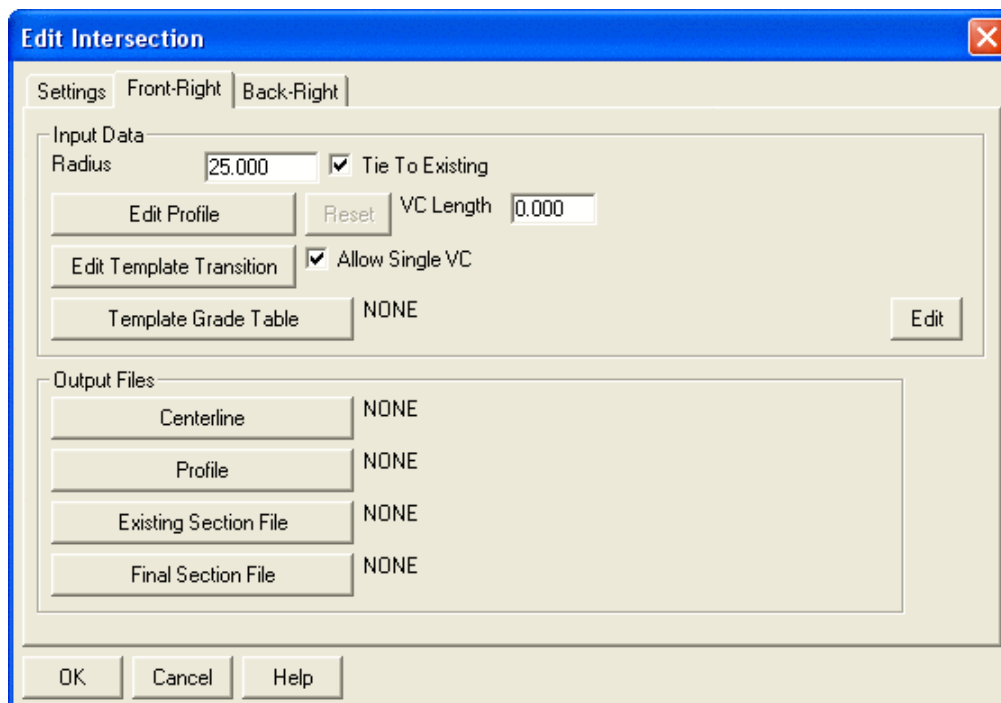
Select the Intersection, right-click and pick the **Edit Intersection** function to display the **Settings** tab of the **Edit Intersection** dialog box. Make changes as needed.

Note: Changes made here apply to all Corners of the Intersection.



The Settings Tab of the Edit Intersection Dialog Box

Or, you can switch to one of the **Corner** tabs - *Front-Right*, *Back-Right*, *Front-Left*, *Back-Left* to make changes to only one Corner of the Intersection.



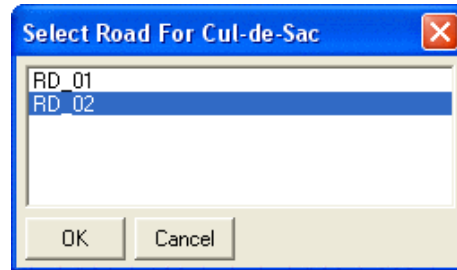
One of the "Corner" Tabs of the Edit Intersection Dialog Box

Pick the **OK** button to close the **Edit Intersection** box and save changes.

See **Road Network: Adding and Editing Intersections** if you need additional assistance.

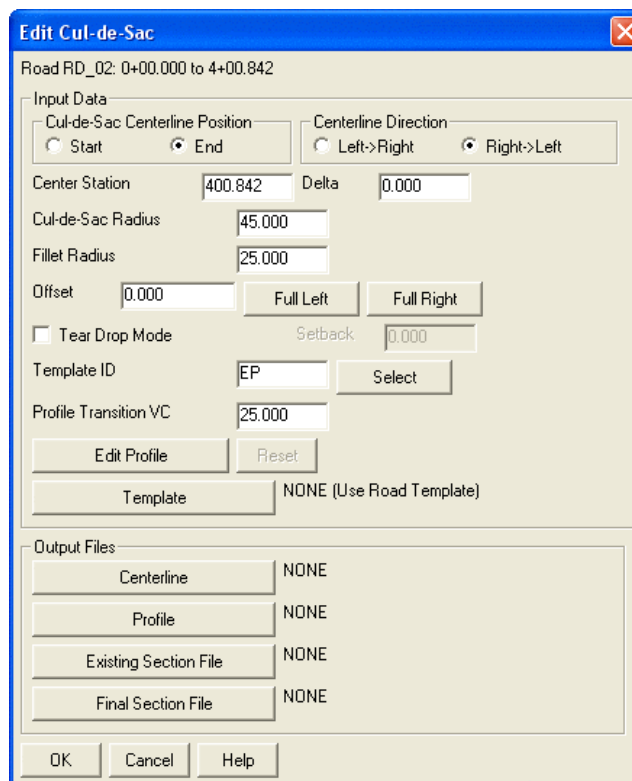
Step 4: Adding and Editing Cul-de-Sacs

Pick the **Add** button in the **Cul-de-Sac** area of the **Road Network Task Pane** to display a dialog box listing the Roads in the Network and prompting you to **Select Road for Cul-de-Sac**.



Select Road for Cul-de-Sac

After choosing the Road and picking the **OK** button, the **Edit Cul-de-Sac** dialog box is displayed. At a minimum, you must enter a **Cul-de-Sac Radius** and **Fillet Radius** to define the Cul-de-Sac.



Edit Cul-de-Sac Dialog Box

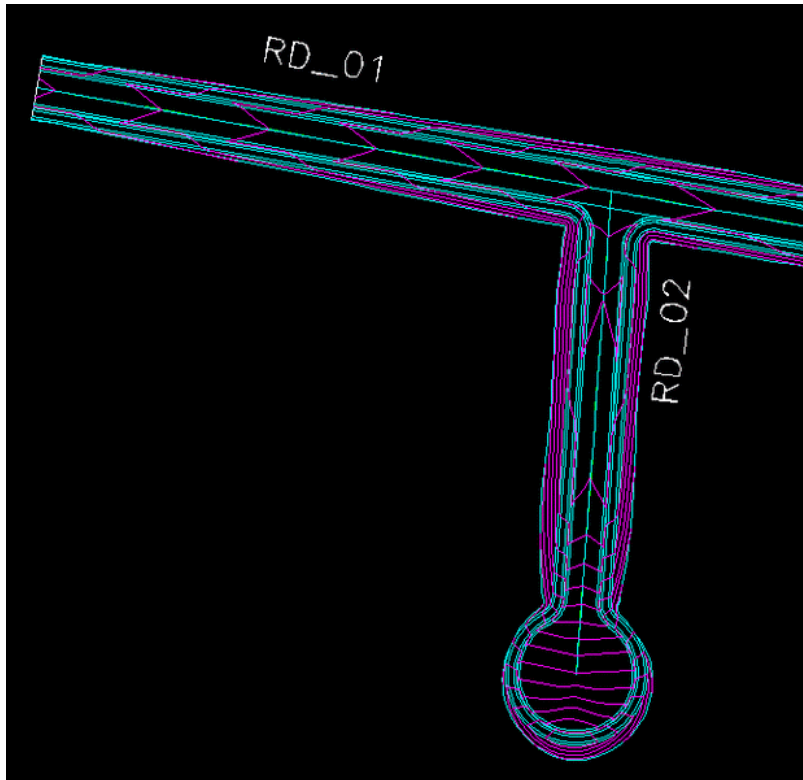
Pick the **OK** button to close the **Edit Cul-de-Sac** box and save changes.

See **Road Network: Adding and Editing Cul-de-Sacs** if you need additional assistance.

Step 5: Save, Process and View the Road Network

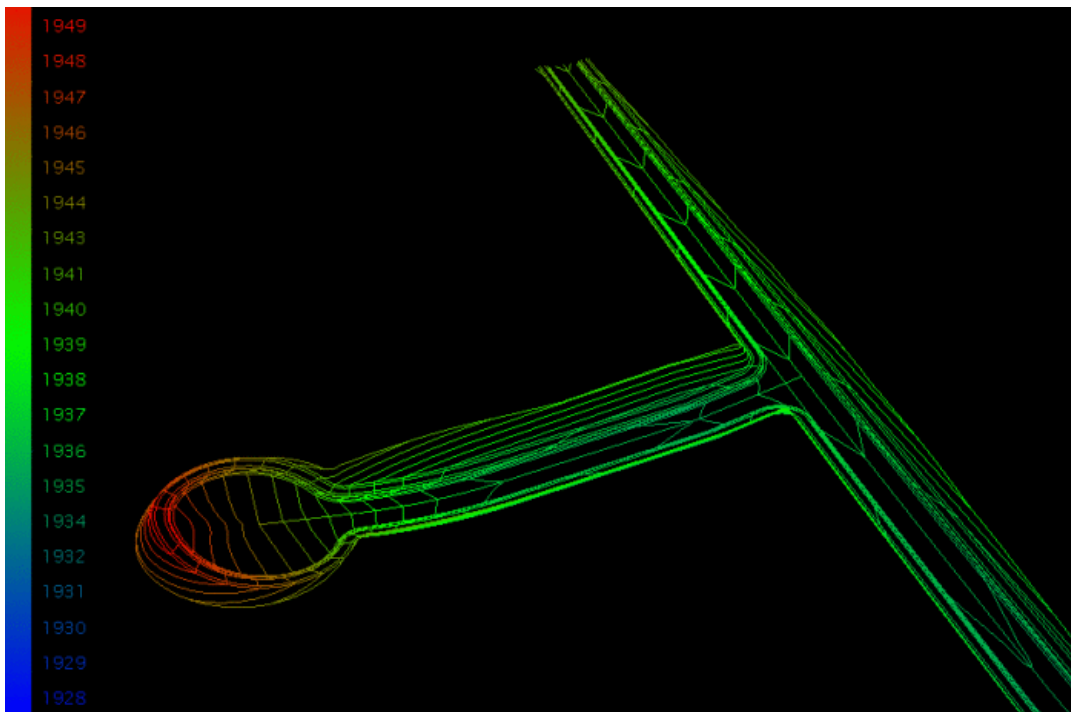
Pick the *Save* button on the **Road Network Task Pane** to **Save** the Road Network (.RDN) file.

Then, pick the *Process* button on the **Road Network Task Pane** to calculate the road design and perform the functions specified in **Road Network Output Options**. The resulting contours and breaklines are shown below.



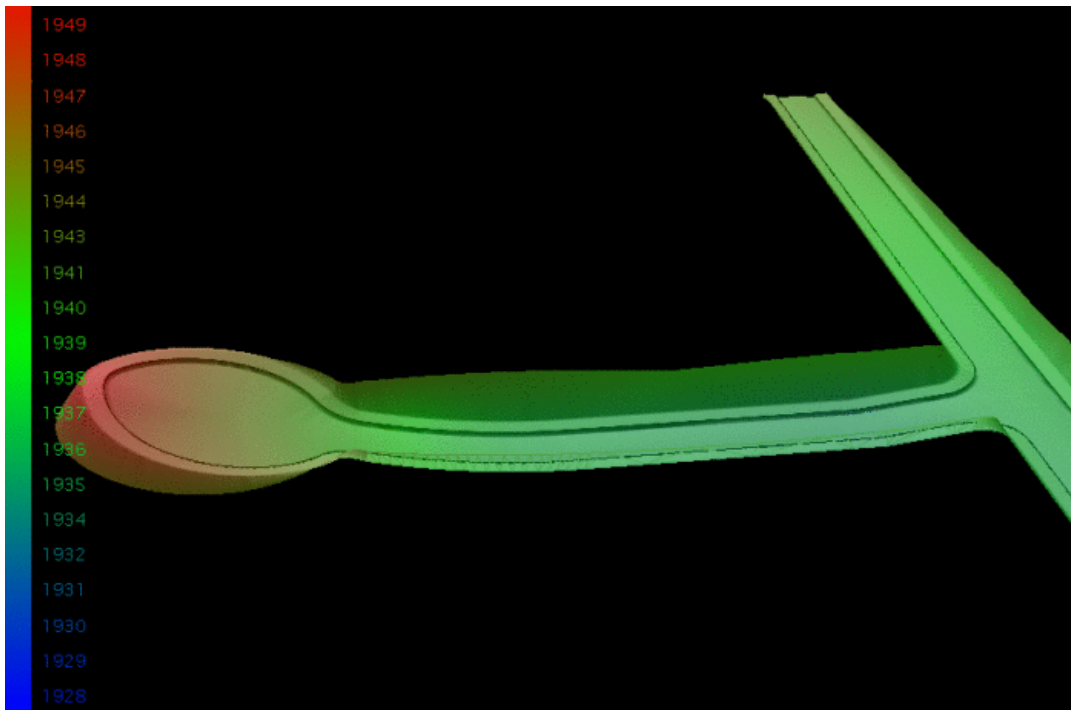
Contours and Breaklines After Processing Road Network

The elevated breaklines and contours can now be viewed using the **3D Viewer Window** command as shown below.



Breaklines and Contours as Seen in the 3D Viewer Window

Or, use the **Surface 3D Viewer** command to view the Surface (.TIN) file as shown below.



Surface (.TIN) File as Seen in the Surface 3D Viewer

Or, use the **Surface 3D Flyover** command to drive the Surface (.TIN) file as shown below.



Surface (.TIN) File in the Surface 3D Flyover Command

Step 6: Reports

Pick the **Report** button on the **Road Network Task Pane**. Then, pick the **Output Processing** button to display the report. This report displays the cut/fill and material quantities for each Road, Intersection and Cul-de-Sac of the Road Network.

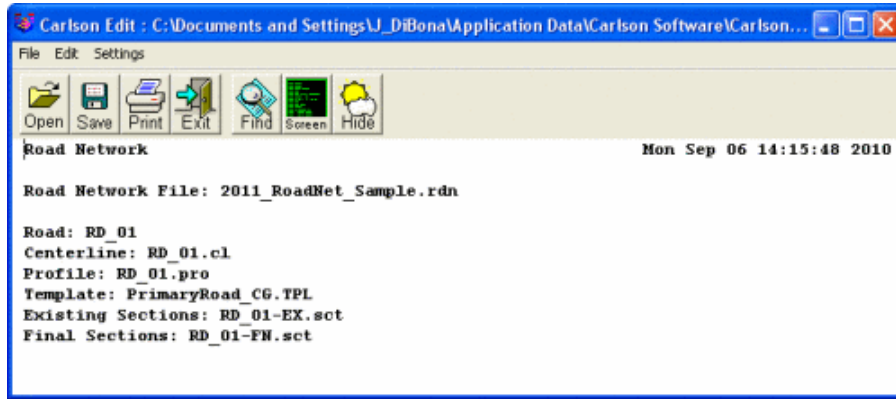
```

Carlson Edit : C:\Documents and Settings\J_DIBona\Application Data\Carlson Software\Carlson 2011\VCADU...
File Edit Settings
Open Save Print Exit Find Screen Hide
Process Road Network Mon Sep 06 14:12:55 2010
Road Network File: D:\Road_Network\2011_RoadNetwork\2011_RoadNet_Sample.rdn
Total Cut : 5791.6 C.Y.
Total Fill: 3726.0 C.Y.
Total Left Subgrade1 - ASPHALT: 144.5 C.Y.
Total Left Subgrade2 - STONE BASE: 463.8 C.Y.
Total Left Subgrade3 - CONC: 106.7 C.Y.
Total Right Subgrade1 - ASPHALT: 144.9 C.Y.
Total Right Subgrade2 - STONE BASE: 466.8 C.Y.
Total Right Subgrade3 - CONC: 107.6 C.Y.
Total Left Curb - CONC: 89.8 C.Y.
Total Right Curb - CONC: 89.3 C.Y.
Road RD 01
Template File> D:\Road_Network\2011_RoadNetwork\PrimaryRoad_CG.TPL
Profile File> D:\Road_Network\2011_RoadNetwork\RD_01.pro
Existing Surface File> D:\Road_Network\2011_RoadNetwork\ExistingGND.tin
Centerline File> D:\Road_Network\2011_RoadNetwork\RD_01.cl
Design Section Output File> D:\Road_Network\2011_RoadNetwork\RD_01-FH.sct
Existing Section Output File> D:\Road_Network\2011_RoadNetwork\RD_01-EX.sct
Processing 0+00.000 to 17+36.731
Total Cut : 156372.176 C.F., 5791.562 C.Y.
Total Fill: 100602.889 C.F., 3726.033 C.Y.
Cut to Fill Ratio: 1.55
Total Left Subgrade1 - ASPHALT: 3902.208 C.F., 144.526 C.Y., 23413.053 S.F., 2601.450 S.Y.
Total Left Subgrade2 - STONE BASE: 12521.554 C.F., 463.761 C.Y., 1726.232 S.F., 191.804 S.Y.
Total Left Subgrade3 - CONC: 2882.121 C.F., 106.745 C.Y., 8648.698 S.F., 960.966 S.Y.
Total Right Subgrade1 - ASPHALT: 3912.567 C.F., 144.910 C.Y., 23475.208 S.F., 2608.356 S.Y.
Total Right Subgrade2 - STONE BASE: 12602.966 C.F., 466.777 C.Y., 1737.455 S.F., 193.051 S.Y.
Total Right Subgrade3 - CONC: 2905.506 C.F., 107.611 C.Y., 8718.611 S.F., 968.735 S.Y.
Total Left Curb - CONC: 2423.981 C.F., 89.777 C.Y., 1740.909 L.F.
Total Right Curb - CONC: 2412.347 C.F., 89.346 C.Y., 1732.553 L.F.
Station Cut(sf) Fill(sf) Interval Cut(cy) Fill(cy)
0+00.000 28.451 3.523
50.000 92.006 3.262
0+50.000 70.915 0.000
50.000 149.837 0.000
1+00.000 90.909 0.000

```

Road Network Output Processing Report

Repeat this step but, this time, pick the **Input Data Files** button to display the report. This report displays all of the user-specified design files associated with the Road Network. For this report, you are given the option of reporting only the filename or both the path and filename.



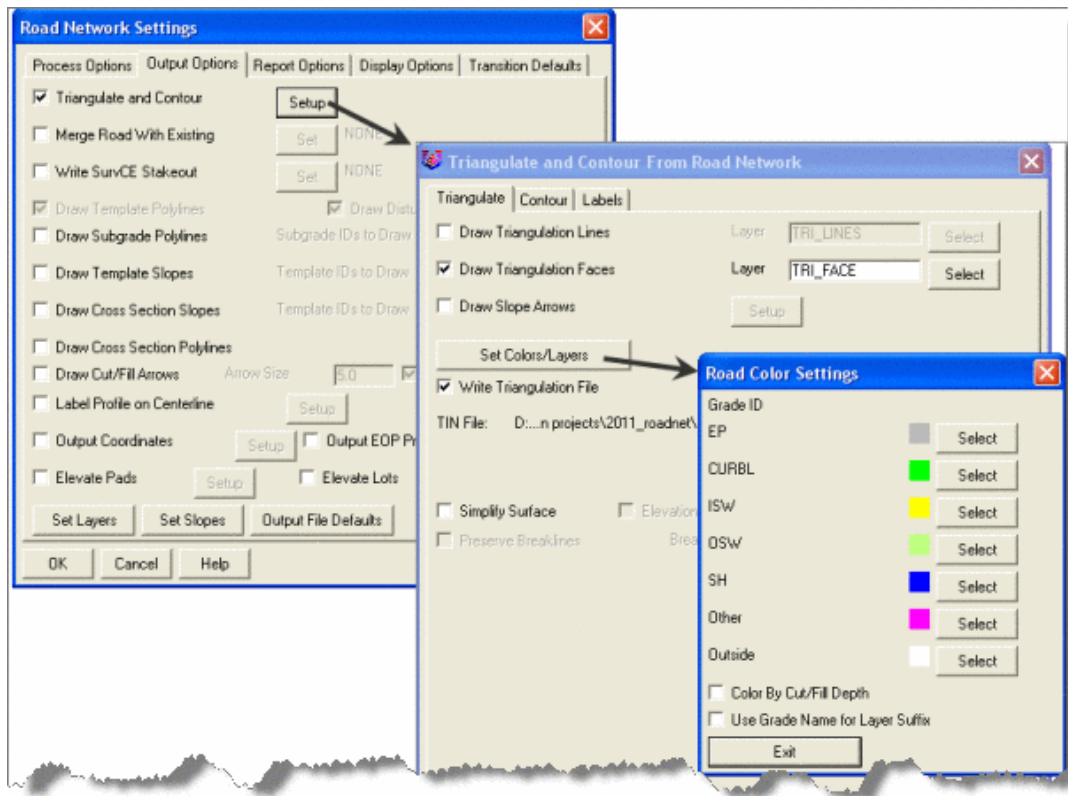
Road Network *Input Data Files* Report

Step 7: Additional Settings and Tools in the Road Network

Draw Triangulation Faces with Color and View in 3D Viewer Window

Pick the **Settings** button on the **Road Network Task Pane** and then pick the **Output Options** tab. Now, pick the **Setup** button next to **Triangulate and Contour** to open the **Triangulate and Contour from Road Network** dialog box.

Select the **Draw Triangulation Faces** option and then pick the **Set Colors/Layers** buttons to display the **Road Color Settings** dialog box (shown below). The color of the faces can be set either by using the **Template IDs** defined in the Template (.TPL) file or using a color range based on the "Cut & Fill Depths".

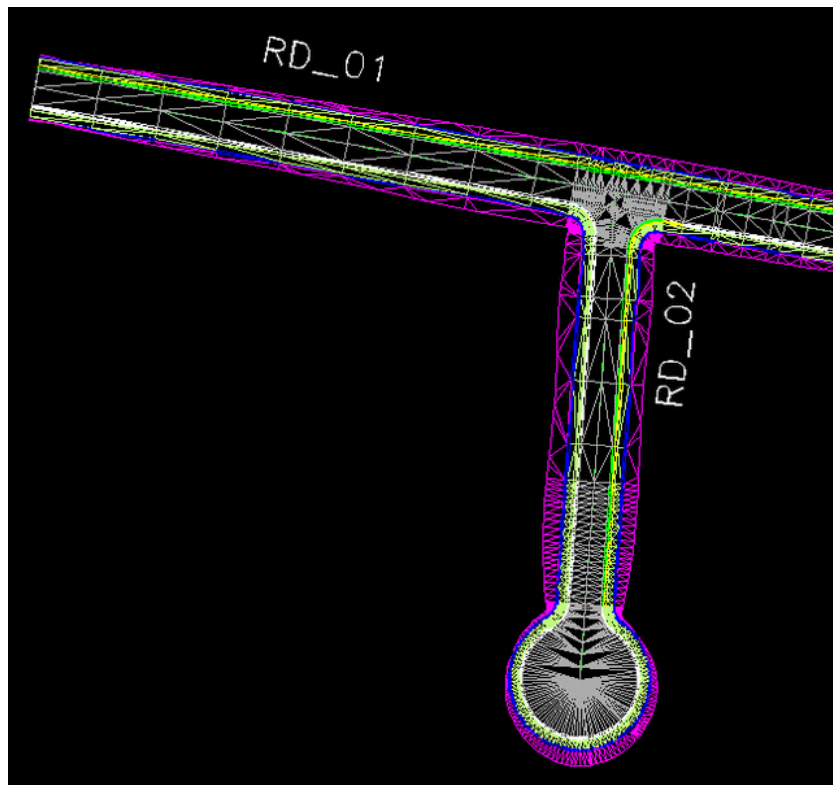


Set Road Colors In the "Triangulate and Contour From Road Network" Dialog Box

*Pick the **Exit** button to close the **Road Color Settings** box and then *pick the **OK** button twice to exit both the **Triangulate and Contour** and **Road Network Settings** dialog boxes.**

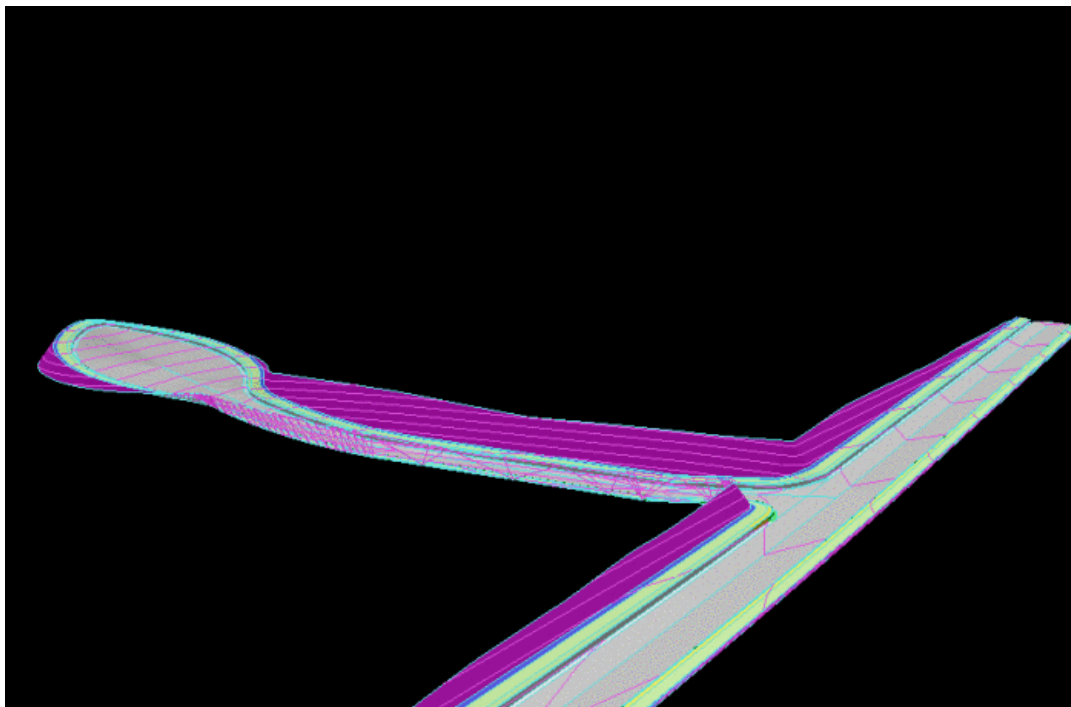
*Pick the **Save** button on the **Road Network Task Pane** to **Save** the Road Network (.RDN) file.*

Then, *pick the **Process** button on the **Road Network Task Pane** to calculate the road design and perform the functions specified in **Road Network Output Options** The image below shows only the Triangulation Faces after Processing.*



Triangulation Faces with Color After Processing Road Network

The elevated Triangulated Faces can now be viewed using the **3D Viewer Window** command as shown below.



Contours and Triangulation Faces with Color in the 3D Viewer Window

Merge Road with Existing

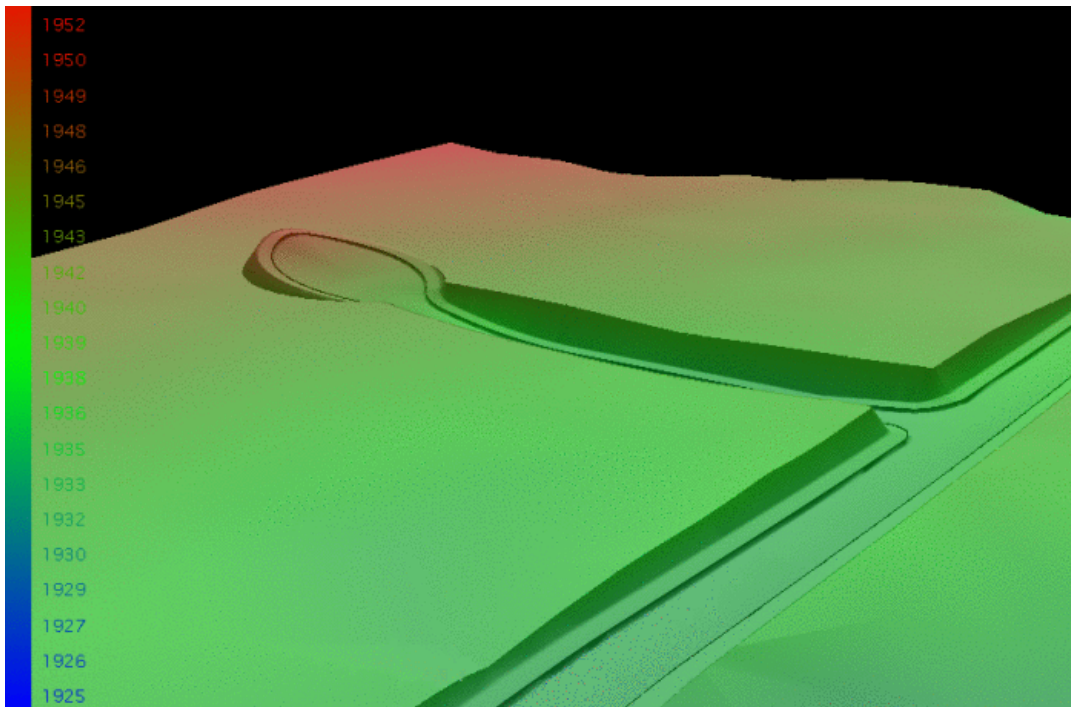
Pick the **Settings** button on the **Road Network Task Pane** and then pick the **Output Options** tab. Select the **Merge Road with Existing** option and then pick the **Set** button to set the path and filename of a 3rd Surface (.TIN) file to be created by combining the design Surface file and the Existing Ground Surface file.

Pick the **OK** button to close **Road Network Settings**.

Pick the **Save** button on the **Road Network Task Pane** to **Save** the Road Network (.RDN) file.

Then, pick the **Process** button on the **Road Network Task Pane** to calculate the road design and perform the functions specified in **Road Network Output Options**

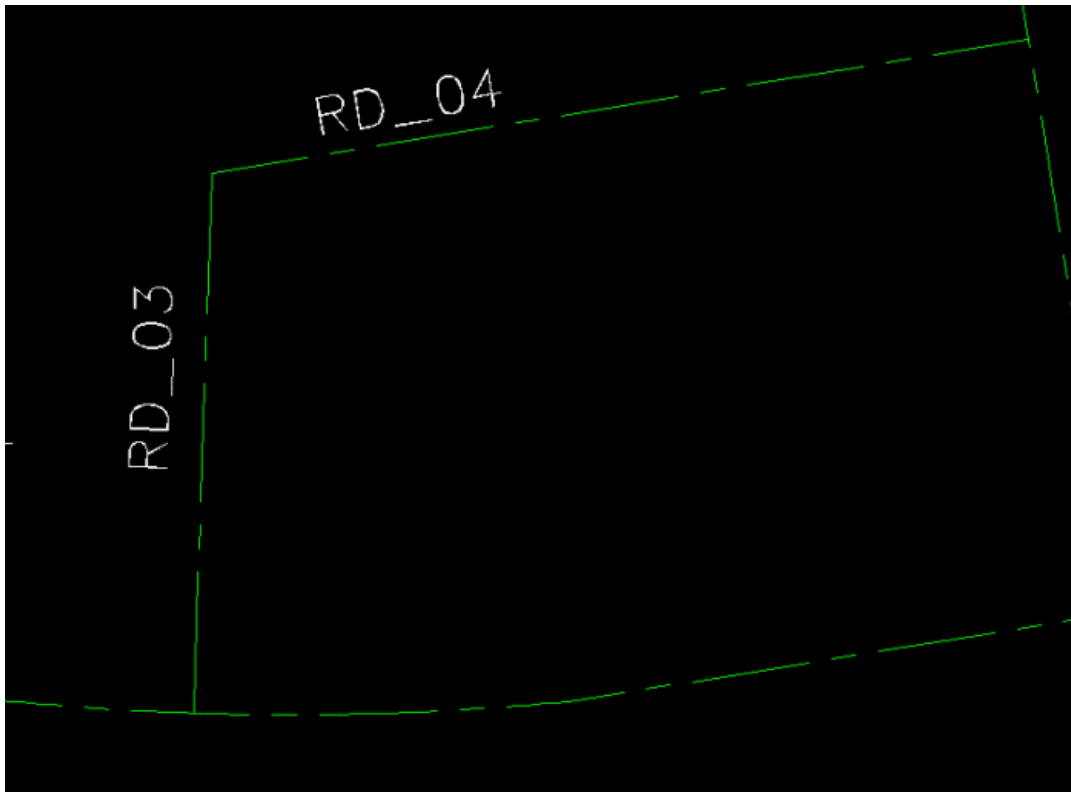
The combined Surface (.TIN) file can now be viewed using the **Surface 3D Viewer** command as shown below.



Merged Existing Ground and Road Surfaces in 3D Surface Viewer Window

Add Knuckle Intersection

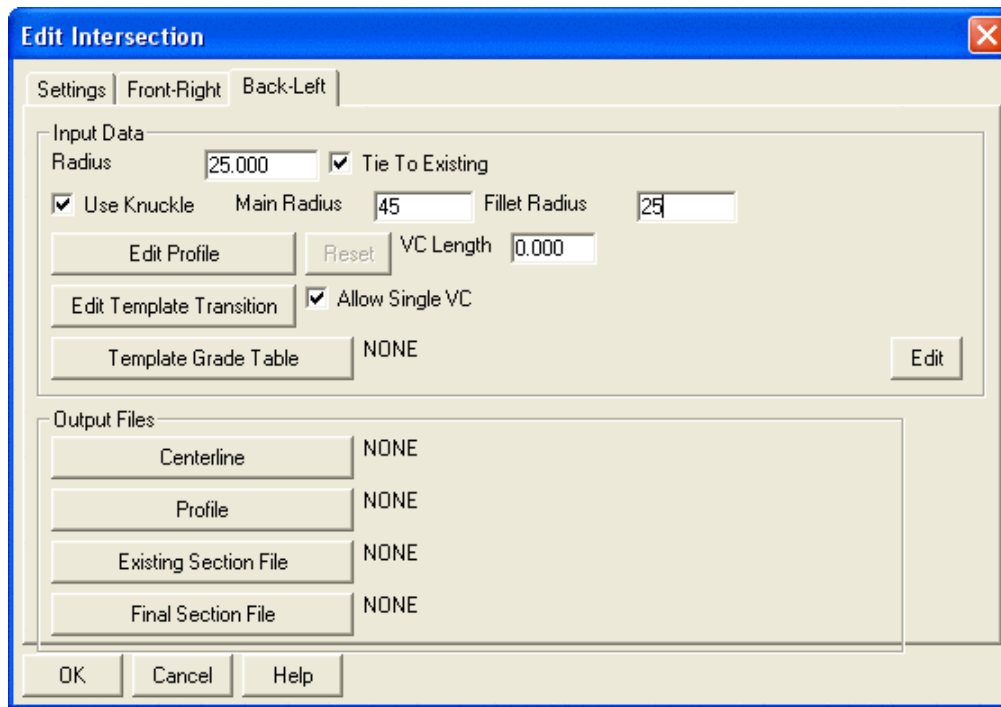
Using the steps outlined in **2 Add Roads to the Network** above, Add two more Roads to the Network.



RD_03 and RD_04

Upon adding the Roads, the new Intersections are automatically added to the **Intersection** area of the **Road Network Task Pane**.

To create a "Knuckle" style Intersection between RD_03 and RD_04, *select the **RD_03 (Primary)/ End:RD_04 (Secondary)** Intersection* in the project tree and then right-click and *pick the **Edit Intersection** function* to display the **Settings** tab of the **Edit Intersection** dialog box. *Pick one of the "Corner" tabs* of the **Edit Intersection** dialog box. *Select the **Use Knuckle** option* and *enter a **Main Radius** and **Fillet Radius** value* for the Knuckle Intersection.

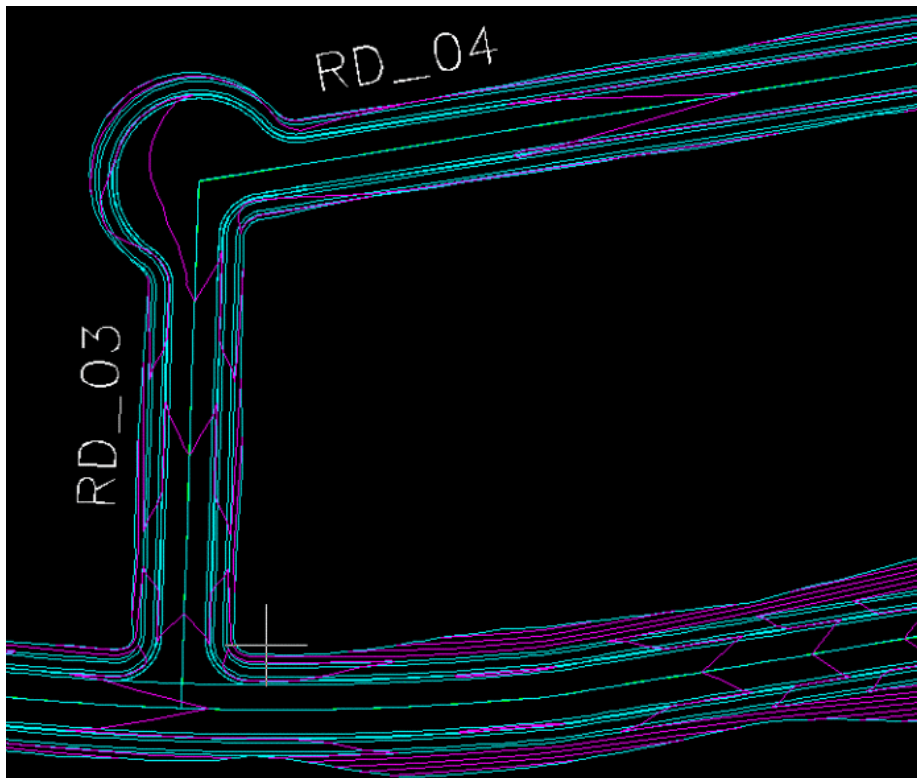


A "Corner" Tab of the Edit Intersection Dialog Box

Pick the **OK** button to close **Edit Intersection**.

Pick the **Save** button on the **Road Network Task Pane** to **Save** the Road Network (.RDN) file.

Then, pick the **Process** button on the **Road Network Task Pane** to calculate the road design and perform the functions specified in **Road Network Output Options**. The resulting contours and breaklines are shown below.

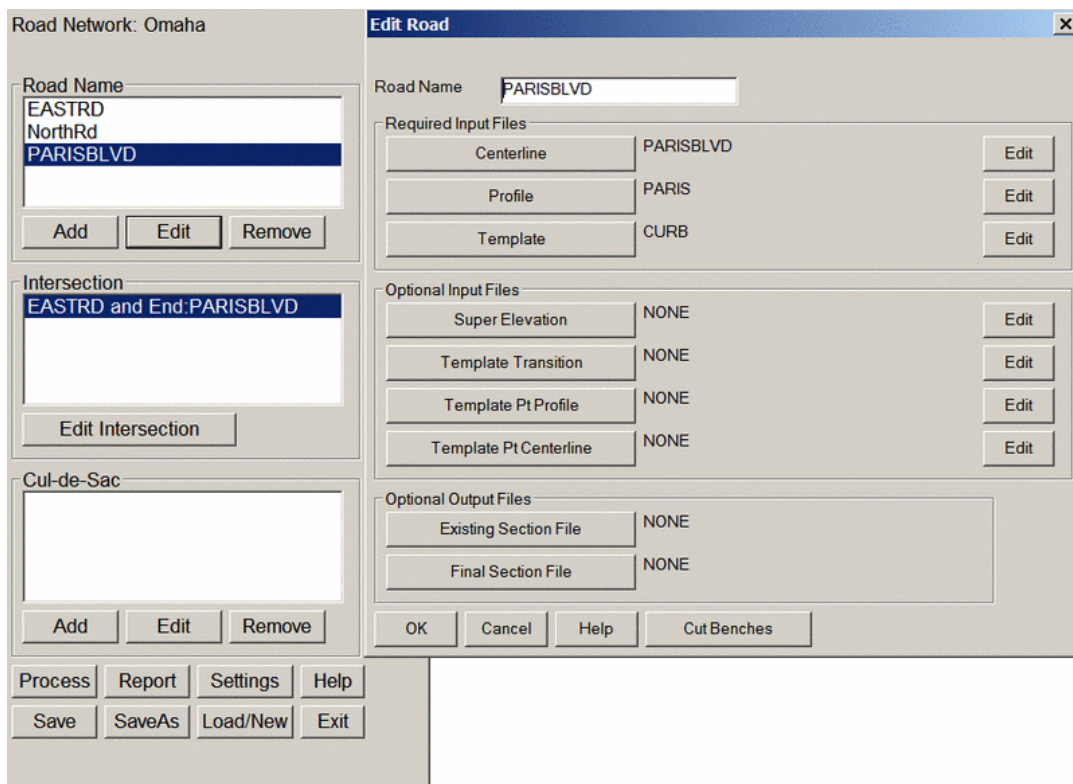


Step 1: Start New Road Network

Start the Road network command. If you have previously run Road Network with the current drawing, the Road Network docked dialog will open with the last Road Network (.RDN) file you worked with. If this happens, but you prefer to create a new Road Network (.RDN) file, click the Load/New button at the bottom of the Road Network docked dialog.

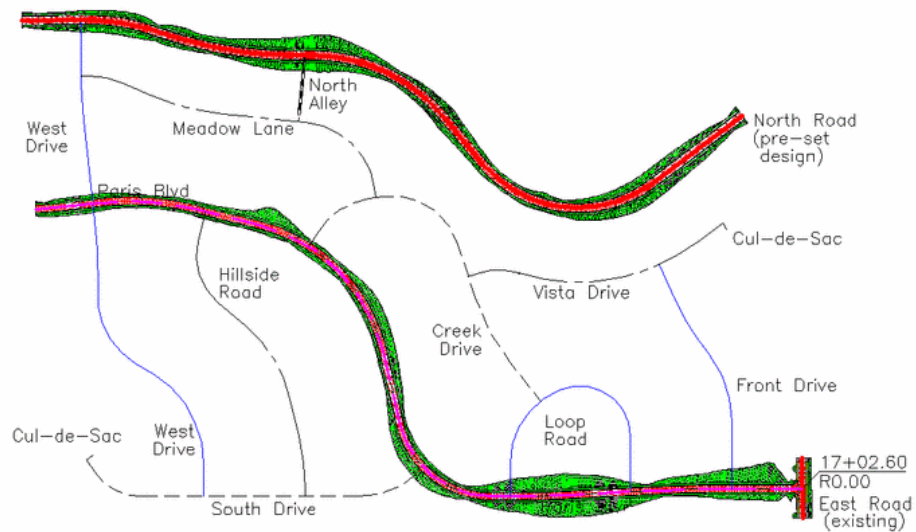
Step 2: Add Roads

Back in the main dialog, click "Add" in the upper left "Road Name" portion, and identify all of the main road and secondary (intersecting) road centerlines. For this example, we will start by identifying North Road and East Road as the main roads and Paris Boulevard as the first secondary road. Note that centerlines may be picked as polylines or loaded as centerline files. All centerlines (horizontal alignments) must have, at minimum, an associated profile (vertical alignment) and an associated template. In the Road name dialog portion, select a road and click Edit to review the files. Note that by selecting Paris Boulevard and East Road, the program automatically detects the first intersect. As you follow the design below, you will see that we follow the hierarchy of the road precedence as outlined in the graphics. At every intersection, there needs to be a primary controlling road (template cross slopes are held) and secondary adjusting road (centerline profile adjusts to template of primary road at some transition distance).

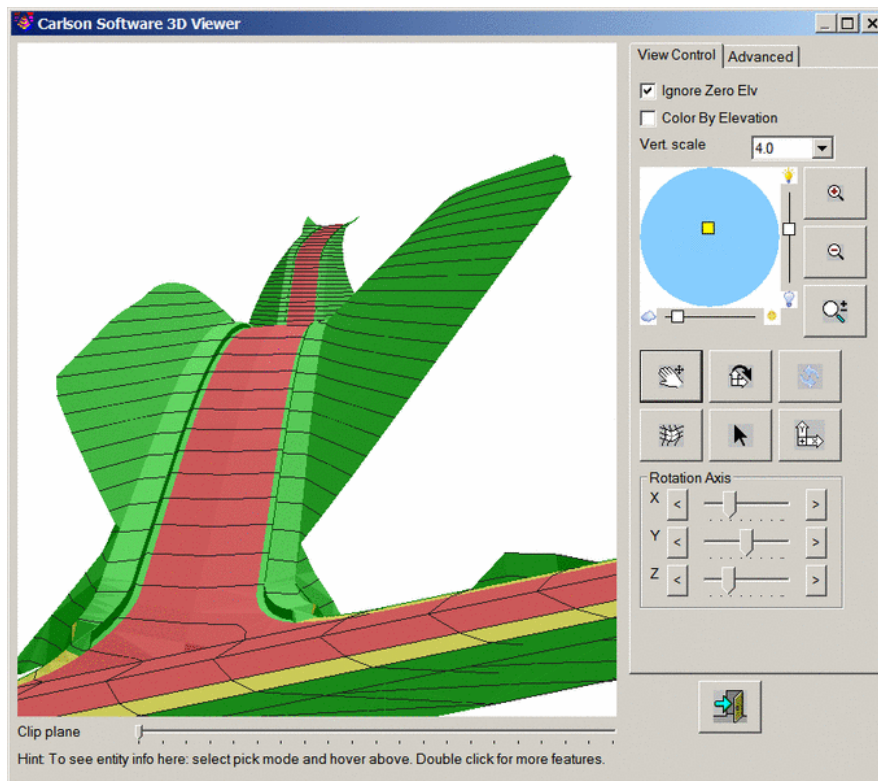


Step 3: Process, Review and Add more

Click Process to compute the design. With the Triangulation option enabled under the "Settings" dialog, the program will Triangulate and Contour and create the drawing shown below. If you edit any road feature or dialog entry and click Process again, the program automatically clears the last Triangulate and Contour drawing and creates a new final design drawing. In this way, you can trial-and-error your design for all roads, or build the design in stages.

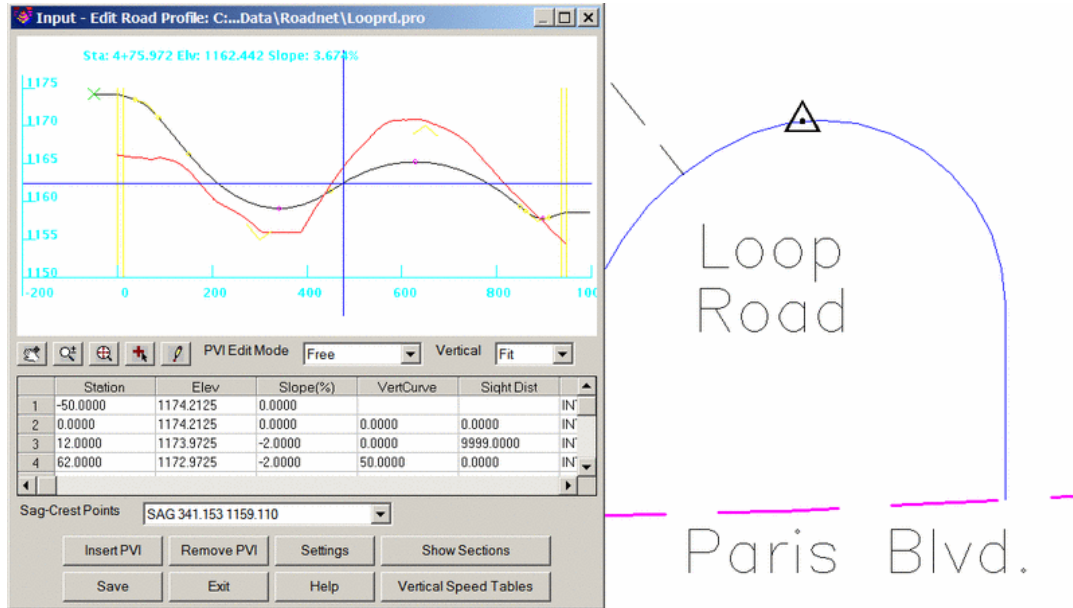


Viewing the file in the 3D Viewer Window command with a 4.0 vertical exaggeration, you can even see how the curb-and-gutter Paris Boulevard ends abruptly as it transitions to the roadside ditch template of East Road.

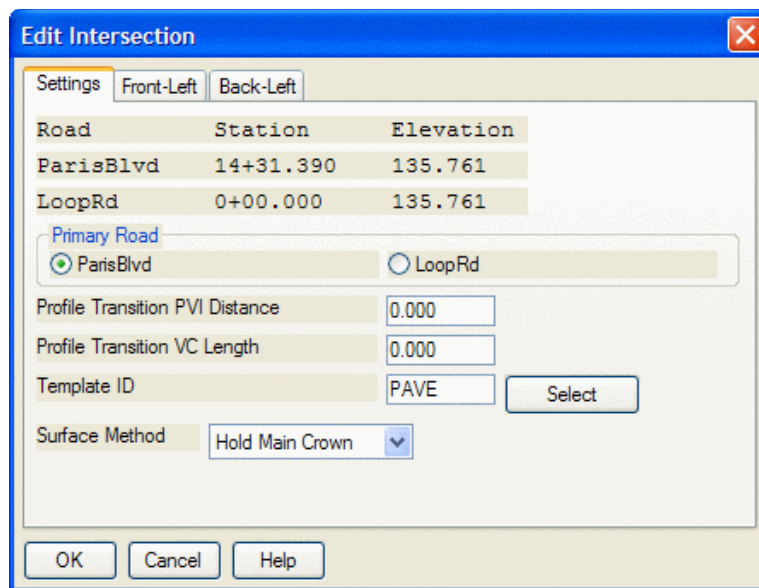


Next we can review the effect of adding Front Drive, Loop Road and West Drive into the equation. If you click Edit after adding Loop Road as above, you have the option to change any aspect of the centerline, profile or template file, and you can add optional files such as road width change files and superelevation files. For example, if you choose to edit the profile, the program derives the existing grade from the existing surface triangulation file specified

in Settings, and you are able to design graphically and interactively as shown:

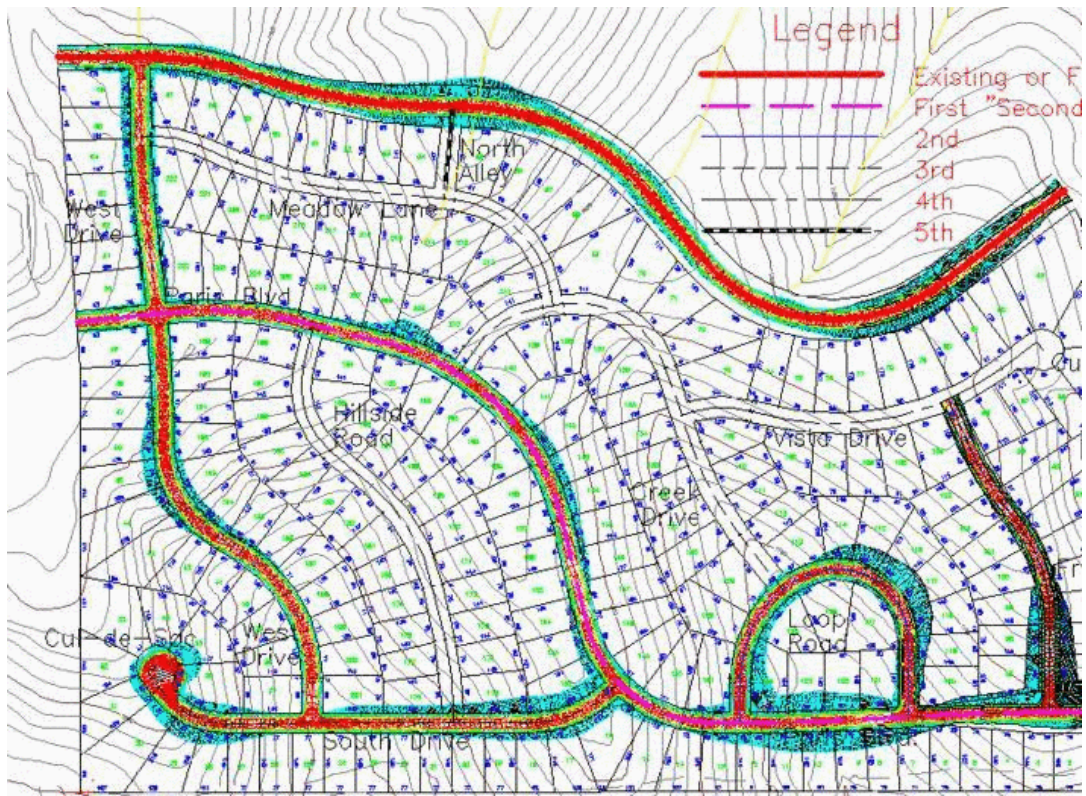


You can also more closely analyze the intersections of any road. If you select the intersection at ParisBlvd and Start:LoopRd, you obtain the multi-tab dialog:

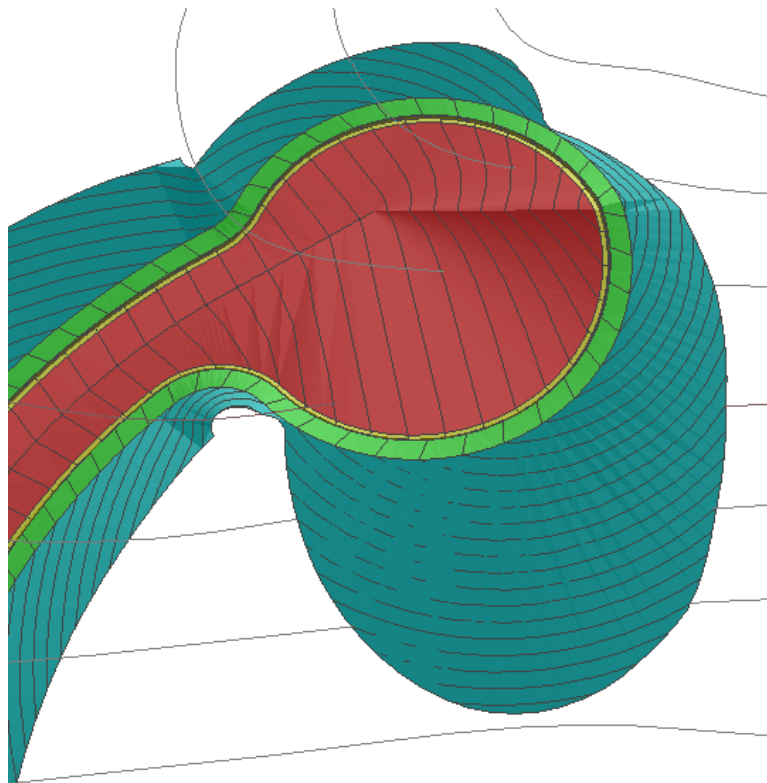


Since we do not have a crossing intersection, we only obtain a "Front-Left" tab and a "Back-Left" tab, left being the left side of the primary road (Paris) and front being the first "curve return" treatment on the outside of the loop and back being the second "curve return" treatment on the inside of the loop. If this was a crossing intersection, you would have 2 more tabs in the dialog: "Front-Right" and "Back-Right". Completing West Drive, Front Drive and South Drive leads to the following plan view and 3D view. Clicking Add within the Cul-de-Sac portion of the docked dialog enables you to specify at cul-de-sac at the end of South Drive.

Clicking Process now produces the following:

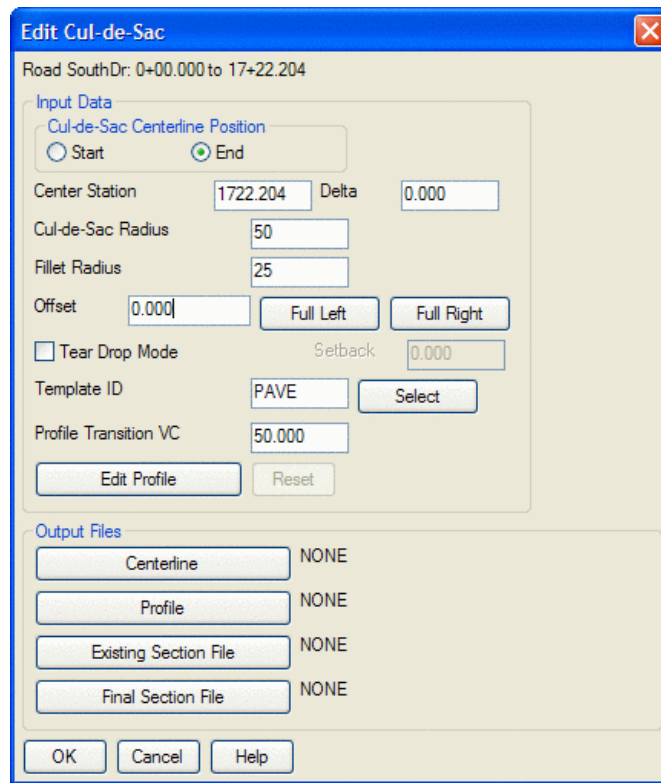


A close-up view of the cul-de-sac, in 3D, reveals the detail of the design, showing a raised "fold" due to no vertical curve transition at the projected high point at the back of the cul-de-sac:

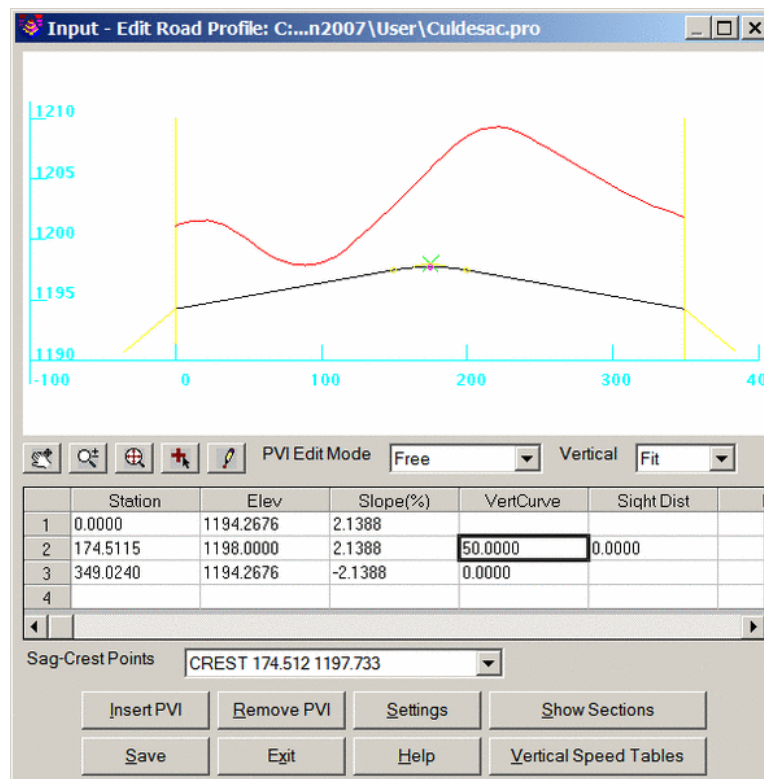


This dimple effect can easily be eliminated by lowering the elevation of the "PVI" at the projected intersect point in the back of the cul-de-sac, and by adding a vertical curve transition of, say 50'. This is done by highlighting the South Drive Cul-de-Sac and clicking Edit.

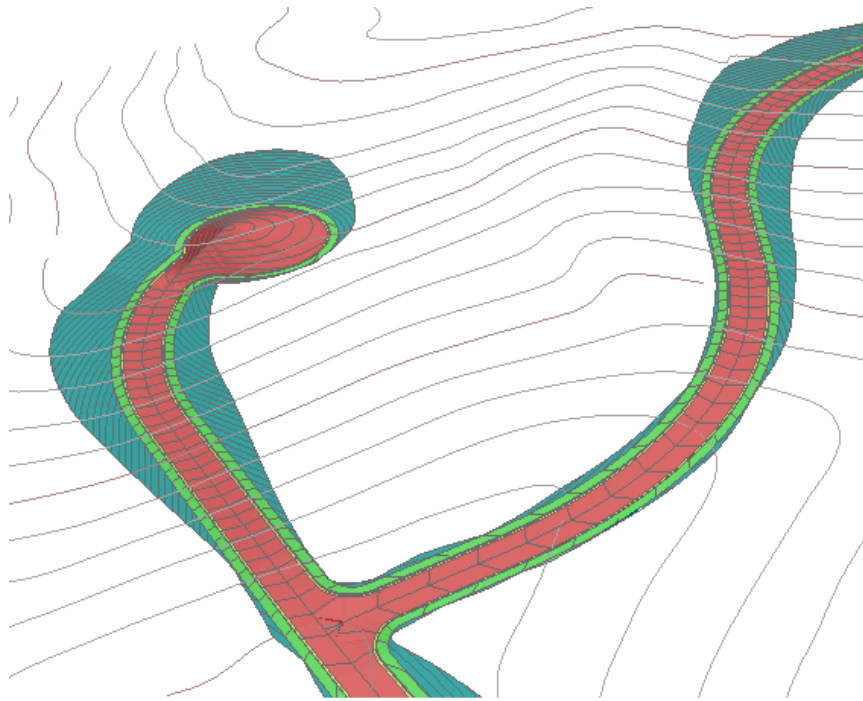
Clicking Edit on the selected SouthDr at End cul-de-sac leads to this dialog:



The first thing we do is change the Profile Transition VC from 0.0 to 50.0, as shown. Then we need to click Edit Profile to lower the profile at the back of the cul-de-sac. This profile refers to the edge-of-pavement grade.



Now, after clicking Process, the cul-de-sac has a better design:



Pulldown Menu Location: Roads → Road Network

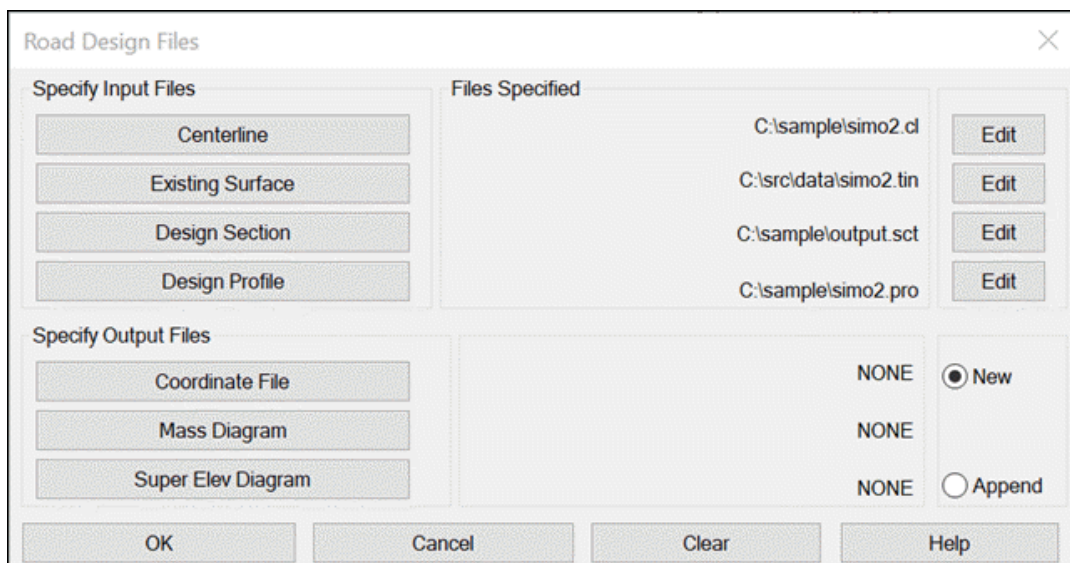
Keyboard Command: roadnet

Prerequisite: Existing Ground Surface (.TIN) file, Template (.TPL) file

Process Design Sections

This command will process design sections to calculate volumes and output surfaces. The main purpose is to process design sections that have been manually adjusted. One workflow is to use Process Road Design to generate design sections from a centerline, profile and template design. Next these design sections can be adjusted with Section File Editor for special stations as needed. Then use Process Design Sections to run these modified sections.

The first dialog specifies the required input files of the centerline, existing surface, design sections and design profile. The optional output files for coordinates, mass diagram and superelevation diagram are set here.



The second dialog has the processing and output settings which are a subset of the settings from Process Road Design. The only setting specific to Process Design Sections is the Interpolate Stations which allows for processing a smaller station interval than the input Design Sections. These interpolated stations interpolate from the Design Sections and use the Design Profile to account for any non-linear variations in the profile between stations.

Please see the Process Road Design section of this manual for a description of rest of these settings.

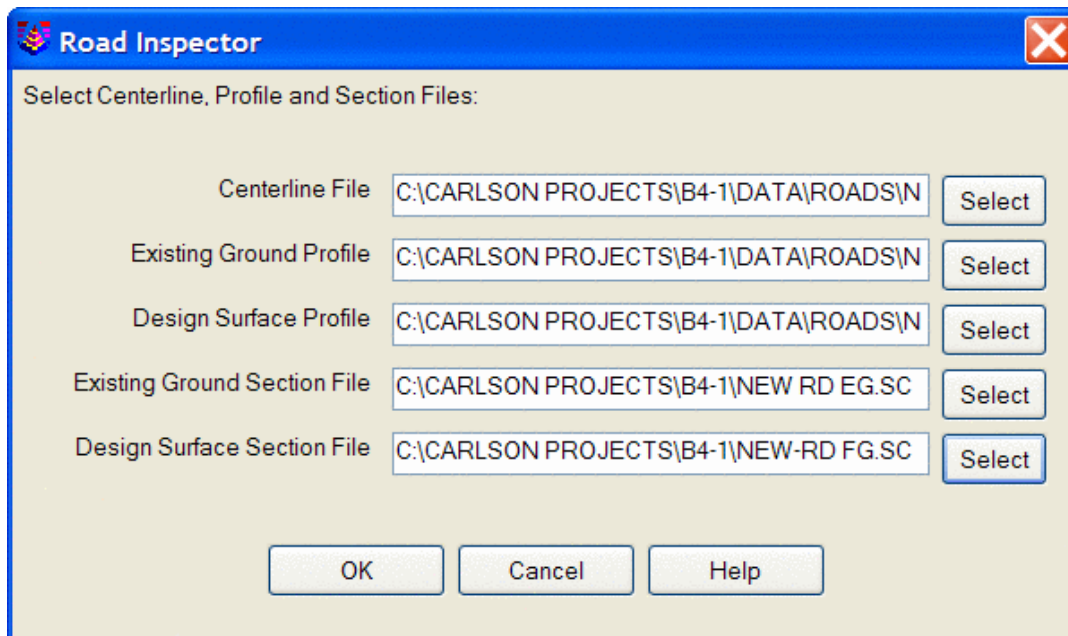
Pulldown Menu Location: Roads

Keyboard Command: reworks

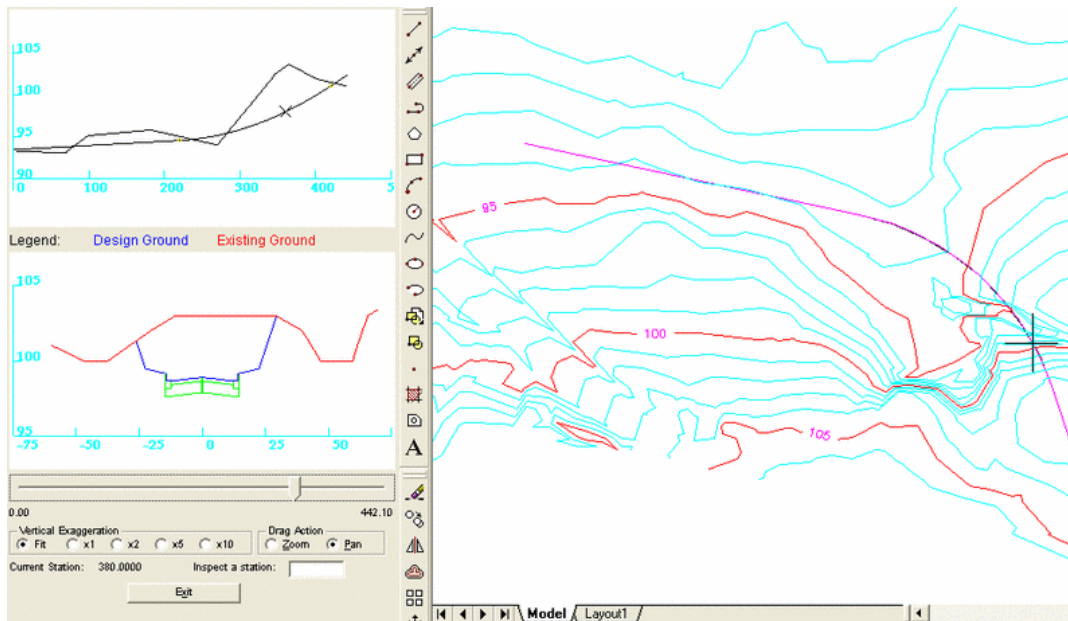
Prerequisite: Centerline, Profile, Existing and Final Cross Sections

Road Design Inspector

This routine takes a full suite of road files (existing and final profiles, existing and final cross sections and the centerline) and presents three graphical windows showing the road in plan view, profile view and section view, with a slide bar that let's you "drive" the road or project from start to end. In addition to applying to roads, the command applies to any set of existing and final sections that follow a horizontal alignment (centerline), such as for channels and embankments. As you move the slide bar left and right, your position is shown in profile view, in plan view and in cross section view. The cross section graphics can be scaled to fit the allotted screen space, or can be set to a scale such as 1H:1V or set to exaggerated scales (2H:1V up to 10H:1V).



The current station is displayed as you move the slide bar. You can zoom and pan the cross section view, and you can also enter a specific station to study. Stations that do not exist in the cross section files will be interpolated.



Pulldown Menu Location: Roads

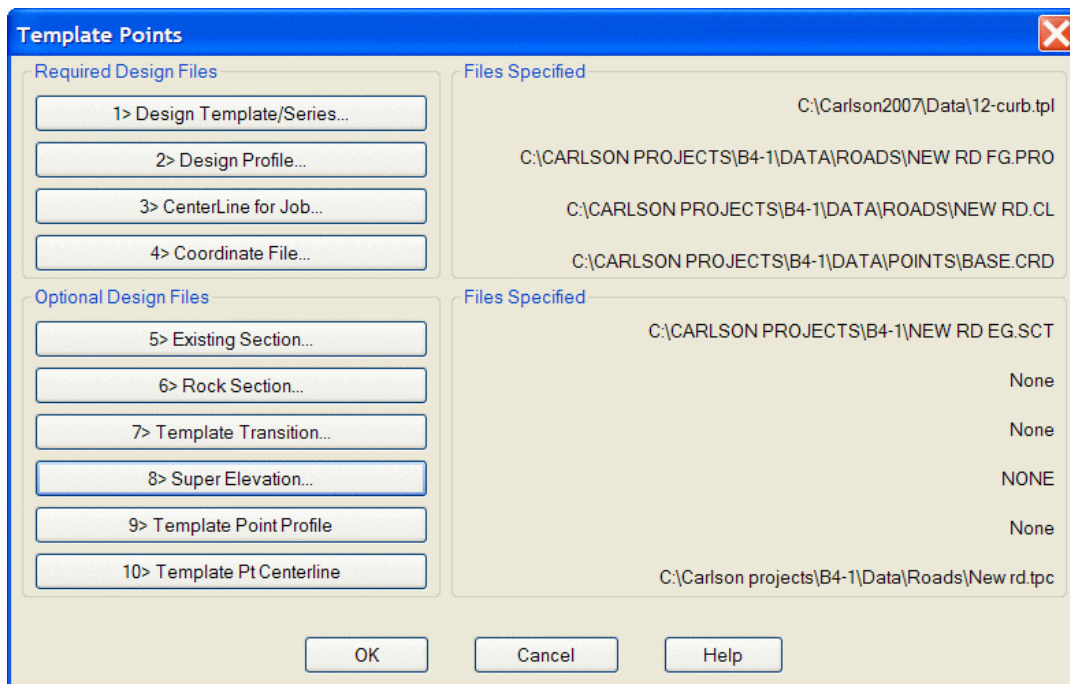
Keyboard Command: rdcheck

Prerequisite: Existing and final Profiles (vertical alignment), Centerline (horizontal alignment), Existing and Final Cross Sections

Locate Template Points

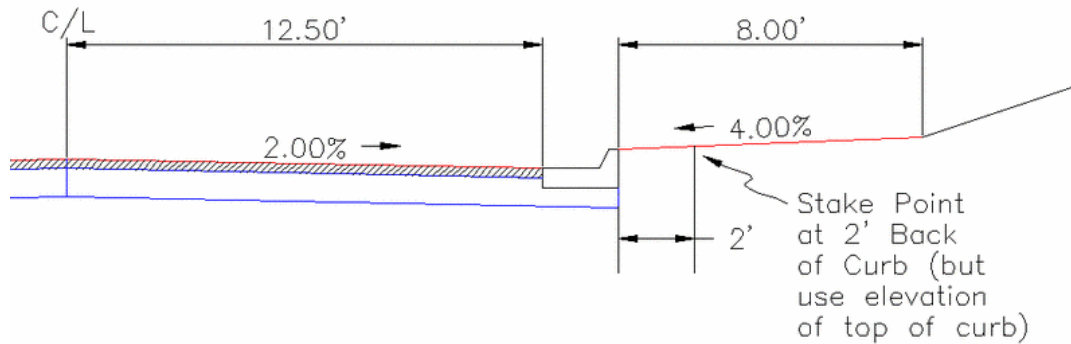
This command creates Carlson points along a centerline either at picked points, point numbers, entered individual station and offset or at station interval with offset, in all cases using the elevations calculated from the template design files. The first offset prompt is for the location of the point. The second offset prompt is for what elevation to use. For staking template points (e.g. edge of pavement) you usually enter the same offset for the position and for the elevation. But if you are staking back of curb, which might be at offset 14.5, you might enter 16.5 for the

position (to stake 2' back of curb) and 14.5 for the vertical elevation (to use the elevation of back of curb itself). The points are stored in a coordinate (.CRD) file. The station and offset of the point is stored in the point descriptions. If the points method is used and existing Carlson points are selected by number, range or "point group", then new points at the same position are created with interpolated elevations and new descriptions. The command starts with the dialog shown below. The required design files include the template file, the profile which defines the vertical alignment, the centerline file which contains the horizontal alignment and the coordinate file for storing the resulting points. All these design files must be created before running this command. To specify a design file, pick on the type of file button. The optional files include an existing section file for calculating the cut and fill slopes, a rock section file for special cut slopes in rock, a template transition file and a super elevation file. For example, if an existing section is specified, template points can be calculated further from the centerline, all the way from the shoulder out to the "catch" or tie point in cut and fill.



If you choose the Station/Offset method, you can specify whether to create points at a station interval. Otherwise the program prompts for each station at which to create points. If the Station/Offset method is used, you will be prompted whether to calculate points on the left, right or both sides of the centerline and whether to offset the calculated elevation by a delta Z amount. If you choose the Points method, you can pick points on the screen (using snaps on entities if desired) or you can specify point numbers individually, by selection set, by range or by point group. You can also select whether to calculate elevations from the template surface or from a subgrade and you can add a description prefix to all descriptions.

A classic application of this routine would be for road staking such as setting back of curb points. Many survey companies prefer to stakeout roads by pre-calculated point numbers rather than calculating from road design files in the field to stakeout road offsets. So if the goal was, for example, to stake 2 feet behind the back of curb, but use the elevation of the top of curb, and the shoulder rose at 4% behind the curb, then the vertical difference to the top of curb would be $-2 \times 0.04 = -0.08$. The program will calculate this automatically by the method of asking for the distinct offset to use for the elevation. Obviously, if you want to stake to the exact surface elevation at the offset specified, then enter the same offset for both position and elevation. The prompting for this back of curb example is shown below.



Prompts

Template Points dialog: Specify the required files and optional files.

Additional Options dialog: Choose Station/Offset or Points method, as shown below:

Additional Template Points Options

Template ID for Profile: CENTER ID Side: Left

Profile Vert Offset: 0.000 Template Horiz Offset: 0.000

Type of Curves: Roadway

Location Method: Station-Offset Points

Stations to Create Points: Interval Centerline Stations Profile Stations User-Entered

Station Interval: 100.0 Starting Station: 0.000 Ending Station: 1500.000

Plot Points Layer: ROAD_PT Select

Include Station-Offset In Description Include Elevation In Description

Description Prefix: Description Prefix

Grade To Process: Top Surface

OK Cancel Help

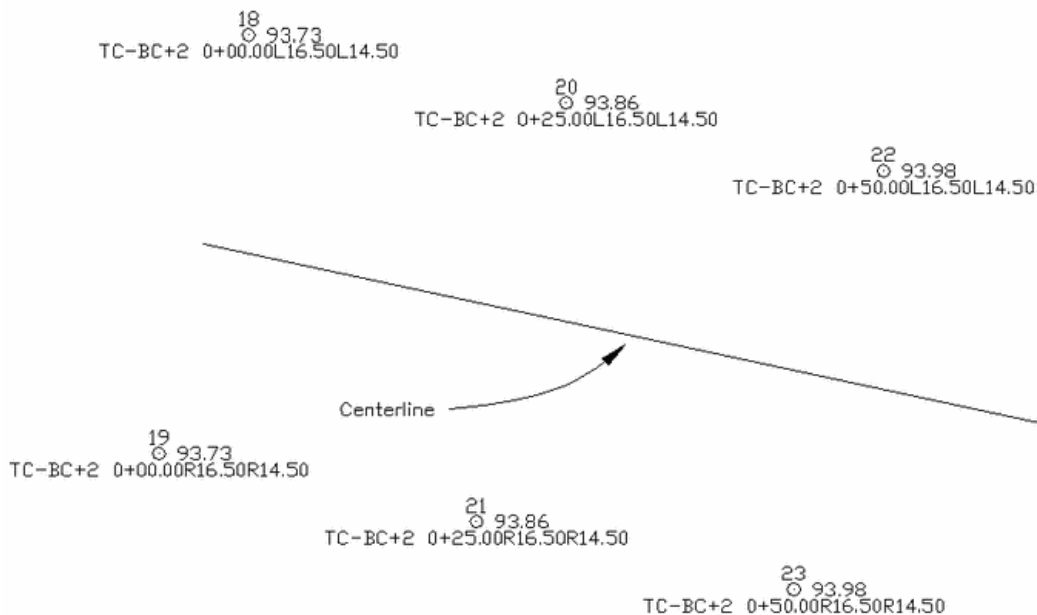
Offset for X,Y position: 16.5 In this case, this is the pavement width (12.5) plus curb width (2) plus back-of-curb offset (2). The northing and easting for the points will be calculated with this offset.

Offset for elevation <16.5>: 14.5 The elevations for the points will be calculated at this offset (back of curb in this example).

Apply offset to left, right or both sides (Left/Right/<Both>)? press Enter Note that if you want to have the points number sequentially on the left side and sequentially on the right, then do L for left first and R for right

second. If you answer "Both" then the numbering will go sequentially left to right on each station (see below).

Offset to process (Enter to End): *press Enter* Or, enter an offset to calculate another X,Y position, or the same X,Y position but on a different side of the road if doing L and R distinctly.



Pulldown Menu Location: Roads

Keyboard Command: tplpts

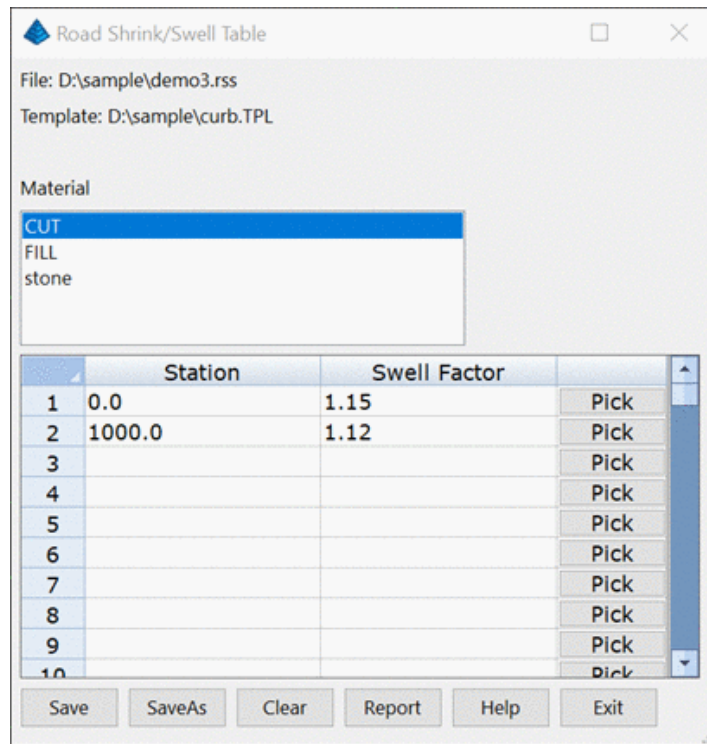
Prerequisite: A template file, profile file and centerline file

Shrink/Swell Table

This command defines different shrink/swell factors by station range for cut/fill quantities and subgrade materials. These factors are stored in a .RSS file which can be used in the Process Road Design command as an optional input file. Without the shrink/swell table, Process Road Design uses a single factor for shrink and a single factor for swell for the whole road. These single factors are set in the second dialog of Process Road Design.

This command prompts for a Design Template file (.tpl) and reads the material names from the subgrades in the template.

The stations and shrink/swell factors are entered in the spreadsheet in the dialog shown here. In the dialog, first highlight the material in the list and then the table of stations and factors for that material are shown in the spreadsheet. The cut material uses a swell factor, and the fill and subgrade materials use a shrink factor. Use the Pick button to screen pick a station along the centerline in the drawing.



Pulldown Menu Location: Roads
Keyboard Command: shrinktable
Prerequisite: Road Template (.tpl)

Hydrology Module

6

Surface Menu

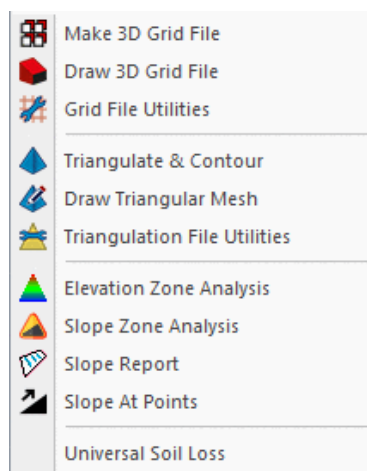
Surface Commands

The Hydrology Module contains many routines that perform Hydraulic and Hydrological calculations for Civil Engineering Design. Each command is designed to work with other commands to complete a design task. For example, the automated pond grading software creates a Stage-Storage Table that can be dynamically linked into Detention Basin Routing Calculations.

Carlson Hydrology was developed using industry standard calculation methods and models. The following represents the standards followed. Stormwater Collection Calculations follows HEC-22 methods and guidelines for surface drainage collection and the Energy Equation for Pipe Networks. Calculations from HEC-22 include Inlet Spread/Capture calculations and Pipe Network Capacity using the Energy Equation per HEC-22. The Rational and SCS TR-55 methods are used to calculate peak design discharge along with development of runoff Hydrographs. In addition to calculations, Carlson also provides a wide range of tools that can be used to obtain public information needed to model a Site. NOAA Precipitation totals and NRCS Soil Data can be queried based on the site location using webservices. In addition to stormwater permitting and design calculations for Site Development, Carlson also provides tools to prepare input files for HEC-RAS which use Carlson DTM and channel alignments to create the input file to HEC-RAS. Pressure Pipe Networks can setup a Utility Network such as a Public Water System where pressure/flow analysis are performed. Many other tools are provided which are commonly used in stormwater design such as a Culvert and Lined Swale designs.

Surface Commands

The pull-down menu for the Surface commands of the Hydrology module is shown here. These commands are identical to the same commands used in the Civil and Survey Modules and are provided here so that Carlson Hydrology can be a stand along product. Much more detailed instruction can be found in the Help Menu for Surfaces in the Civil and Survey Help Topics. The following describe the use of these tools for Hydraulic and Hydrological Design.



Make 3D Grid File

Almost all Civil Engineering Stormwater Design require a surface model. Surface models are used to determine drainage areas and provide a reference elevation model for Stormwater Systems. In many cases, surface models created by Triangulation Methods need to be simplified in order for operations such as automated watershed delineation. Grid files are much simpler and provide enough detail to delineate watersheds and are recommended when delineating watersheds.

Draw 3D Grid File

The main purpose of drawing a 3D grid file is to check the dataset used to create the grid file. They can also be used in processes that evaluate and map each grid for a particular property such as slope. These include the creation of Slope Maps and calculation of Sediment Runoff using the Universal Soil Loss Equation.

Grid File Utilities

Grid File Utilities is included when changes need to be made to a grid surface.

Triangulate & Contour

In addition to grid surface files, Triangulate Surfaces can also be created with the Hydrology Module. Whereas grid files can be used to determine watershed boundaries, tin surfaces are used when precision is needed. Tin surfaces produce much more detailed surface models that should be used for determining Inlet and pipe invert elevation. They provide a more exact surface model and do not average any of the input data.

Draw Triangular Mesh

Triangular meshes should always be drawn to determine if any errors exist in the model.

Elevation Zone Analysis

Elevation Zone Analysis produces maps where elevation ranges are delineated and group together. They are very helpful in Pressure Pipe Design to determine service areas for water systems such as locating storage tanks and the design of booster pumps.

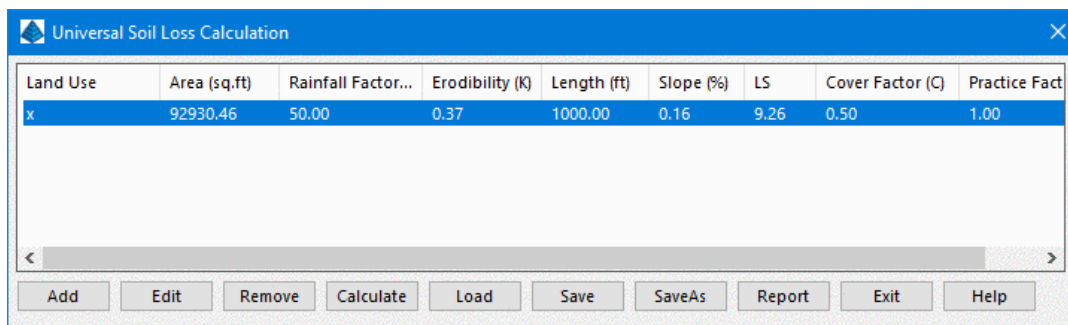
Slope Zone Analysis

Slope Analyses creates a map where areas with similar slopes are grouped together. The main purpose for stormwater and erosion control design is to delineate steep areas that must be avoided with collection systems and area that will require additional erosion control provisions. A common use of slope analysis for erosion control design is to quickly delineate slope banks that will require matting and additional erosion control provisions. **Slope Report** Slope reports provide the same information as slope maps in a non graphical format. **Slope at Points** Slope at Points provide graphed numerical values for the slope which can be used in a wide range of Hydraulic Design such as Swale Design. It is recommended that Slope Points be provided along with Slope Zone Analysis.

Universal Soil Loss

This command calculates the volume of sediment that can be expected from a watershed by soil erosion due to precipitation. It allows the user to specify multiple watershed areas, each with its own set of geometric and hydrological parameters. The Universal Soil Loss Equation (USLE) is used in calculating the soil loss. For each area, the area, slope and length can be manually entered by the user or it can be calculated by the program directly. For direct calculation of the geometric properties of the area, the user must have a grid file that models the surface. This can be created using the *Make 3D Grid File* command. In addition, the area must be defined by closed polylines for inclusion perimeter. Exclusion perimeters are optional for excluding areas from calculations.

The program starts with the dialog below, where the user can add as many areas as needed to include in the USL calculation. Each area added is shown in the list box with all its parameters listed. To add a new area, click the "Add" button. To edit the parameters of an existing area, highlight that item and click the "Edit" button. To remove an existing area, highlight it and click "Remove".



The "Edit" or "Add" button brings up the dialog box shown here, where the various parameters of the area can be specified or edited. The "Landuse" is just an identifier for the area and has no further significance. Soil Erodibility, K (tons/acre) is a property of the soil, which determines the amount of sediment resulting from a precipitation event in an area. The rainfall factor, R, is a dimensionless factor that accounts for the relationship between erosive forces of falling rain and runoff. The Cover factor, C, is a dimensionless factor that relates the effectiveness of vegetal cover in reducing erosion. The Topographical factor, Ls, is a dimensionless length slope factor that accounts for variations in length and slope in the area. The (Conservation) Practice factor, P, is a dimensionless factor to determine how landuse effects its erodibility.

If the area of the watershed is known and is entered manually, then the length and slope of the area have to be entered manually as well and the Ls factor will be calculated from these geometric properties. The area can also be calculated directly if the boundary is defined as a closed polyline and the grid file that models the surface is also made. The user clicks the button "Select area" and the program asks the user to select the grid file as well as the closed polyline representing the area. Then, the Ls factor and the slope are calculated by the program and displayed (the "length" is not needed in this case). After filling in all values, click on "Calculate USL" to calculate the soil loss rate per unit area for the area selected. The user can change the parameters corresponding to this area and recalculate, if needed. Click "OK" to return to the main dialog box. The area should now appear in this dialog box if the parameters as specified.

After all required areas are input, the sediment volume can be calculated by clicking the "Calculate" button on the main dialog. This brings up the USLE Calculation dialog box as shown here. Specify the Delivery ratio, which determines what portion of the gross erosion is actually left for deposition at the final destination, accounting for losses during sediment transport. Also, specify the Time period for which deposition has occurred. Specify the Density of the sediment, so as to be able to determine the volume of the deposit from its mass in tons. Also, specify the amount of Rainfall (inches or cm) for which runoff volume has to be calculated. The program then calculates the Runoff volume based on the total area and the amount of rainfall. It also calculates the sediment volume, using the Universal Soil Loss Equation (USLE) and adds it to the sediment volume and reports it as the total pond volume. A report of the form shown below is generated. This report also gives a detailed account of the calculations performed. For further information about the estimation of the various parameters used in this program or about the USLE, please refer to "Applied Hydrology and Sedimentology for Disturbed Areas" (1981), Barfield, B.J., Warner, R.C. and Haan, C.T., Oklahoma Technical Press.

Soil Data

Land Use: Development

Rainfall Factor (R): 50.000

Soil Erodability (K): 0.370

Cover Factor (C): 0.500

Support Practice Factor (P): 1.000

Geometric Properties

Area Units: Sq. Foot Acres Sq. Meter

Area: 1.000 Select Area

Slope Length (< 1000): 1000.000 ft

Slope: 1.000 %

Topographic Factor (LS): 0.257

Gross Erosion for the Area: 2.375 tons/acre/year

Calculate USL OK Cancel Help

Hydrological Data

Delivery Ratio: 0.37

Time Period: 1.00 Year

Sediment Density: 62.40 lb/cu.ft.

Runoff: 1.00 in

OK Cancel Help

Carlson Edit : C:\Users\Travis Maxwell\AppData\Roaming\Carlson Software\Carlson2023\CAD10_1_X64.US...

File Edit Settings

Pond Volume Calculation based on Universal Soil Loss Equation Sun Jan 29 15:36:33 2023

Input Data

Landuse	Area (ft^2)	Rainfall Factor	Erod-ability	Length (ft)	Slope (%%)	Ls	Cf	Pf
Development	92913.5	50.0	0.37	1000.0	16.2	9.26	0.50	1.00

Runoff = 1.000 inch.
Sediment Density = 62.400
Time = 1.0 years
Drainage Ratio = 0.37

Total Area = 2.133 acres
Gross Erosion = 182.748 Tons/yr.
Total Yield = 67.617 Tons/yr.
Runoff Volume = 0.178 acre-ft
Sediment Volume = 0.050 acre-ft
Total Volume = 0.228 acre-ft

Universal Soil Loss Equation : Application in this Program

Required User Input :

Note : The following values are to be entered by the user for each area in the watershed. The method for obtaining these parameters is left to the user.

Rainfall Factor (R) : Default = 50;
Erodability Factor (K) : Default = 0.37
C Factor (C) : Default = 0.5
P Factor (P) : Default = 1.0

Geometric Information :

Area (sq units) : Can be entered by the user or the program calculates the area of a user-selected closed polyline. The user also has the option to enter the area in different units.

Slope, S (%) : Can be entered by the user or the program calculates the slope when the user selects an area from the screen.

Length (L) : Can be entered by the user. If the user selects an area from the screen, the length is not calculated since the Gross Erosion is directly calculated.

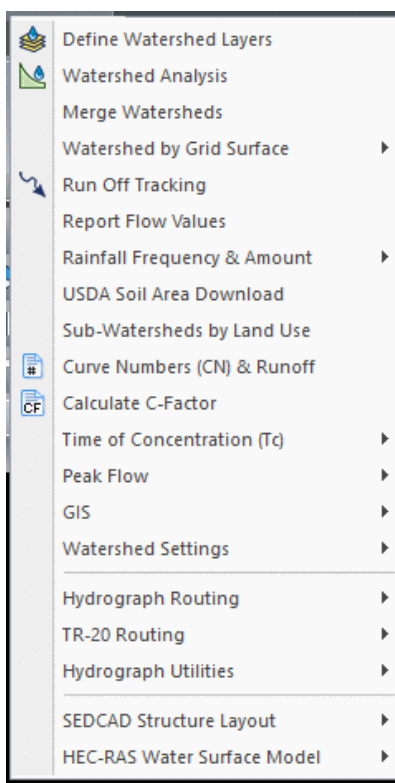
Pulldown Menu Location: DTM in Hydrology

Keyboard Command: soilloss

Prerequisite: Use *Make 3D Grid File* to create a grid file that models the surface

Watershed Menu

The Watershed menu is shown below. The first section of commands are for watershed analysis and are primarily based on TR-55. These commands are arranged in the order that they would be applied. The first commands calculate the watershed boundary. Using the watershed area and land use types, the curve number can be calculated, which leads to time of concentration and hydrographs. Then the peak flow can be calculated. The second section of commands are for hydrograph routing using TR-20. The bottom section has commands for linking to SEDCAD, HEC-RAS and HEC-2.



Define Watershed Layers

This dialog box is the interface to assign specific Ground Covers to closed polylines in a drawing, based on their drawing layers. There are 3 modes of operation; Rational Method, SCS Method, and HydroCAD.

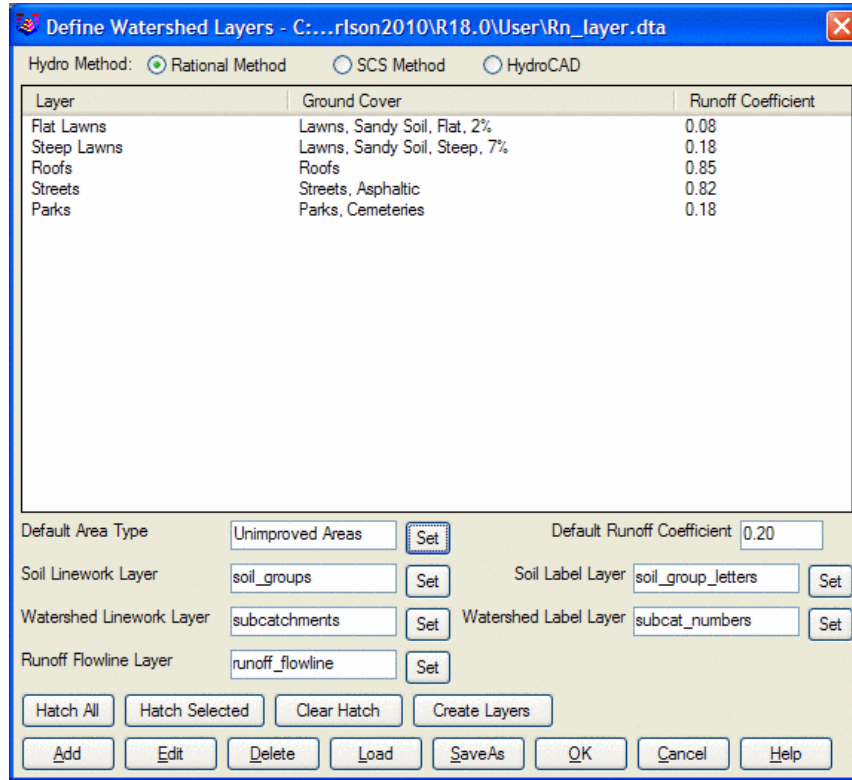
With the dialog set to Rational Method, the runoff coefficients are the C-Factors in the Rational Equation $Q = C \cdot I \cdot A$. Q is flow, I is rainfall intensity and A is area. The Rational Method is often used for urban and residential flow analysis. For example, building layers can be assigned a high runoff coefficient (C factor) such as 0.85 and wooded areas would be assigned a low runoff coefficient such as 0.20.

With the dialog set to SCS Method, the coefficients are set to values from 0-100, so Roofs might be 85 and Woods 20. A Soil Type must also be specified for each Ground Cover.

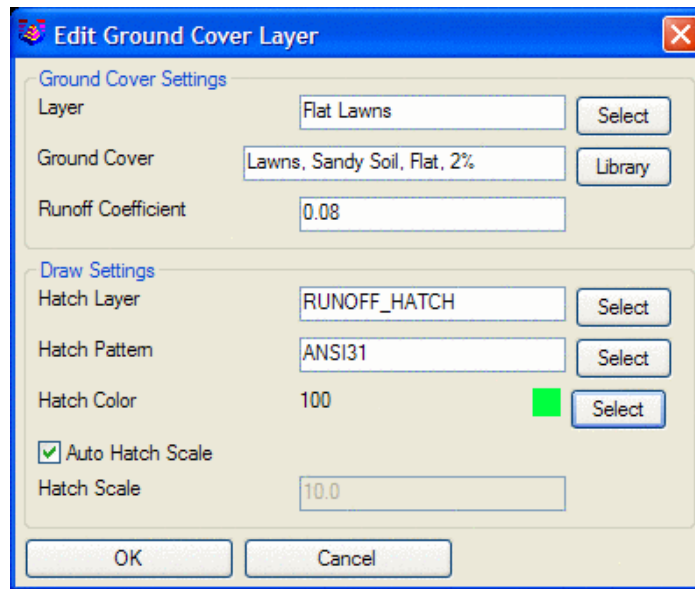
With the dialog set to HydroCAD, the runoff coefficients are not set at all, but are added when the data is exported to HydroCAD.

With the Rational or SCS Methods, the runoff coefficient area polylines are used to determine the weighted runoff coefficients for drainage areas in conjunction with other Carlson Hydrology commands such as Watershed Analysis and Edit Sewer Structure. The runoff coefficient polylines are automatically clipped by the drainage perimeter polyline to find the coefficient sub-areas within the drainage perimeter. Therefore, it is important to close all polylines, use distinct layers for features that have distinct runoff values, and to assign a runoff coefficient to the unassigned, "remainder" areas. It is also important to enclose areas beyond the site with closed polylines and assign runoff coefficients to those layers to account for the off-site water entering the site.

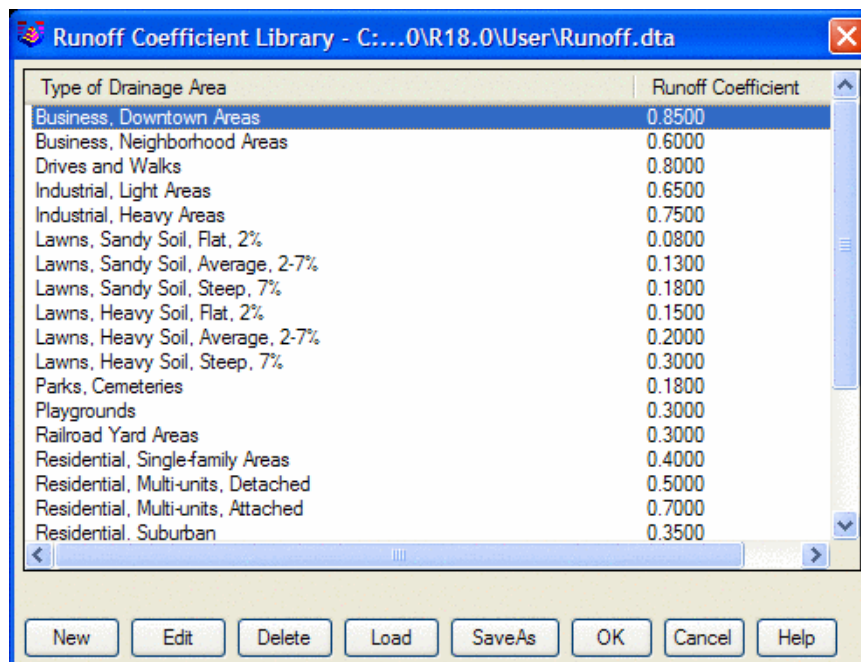
Looking at Rational Method first, the initial dialog box would look something like this:



Adding or editing a layer brings up the next dialog:

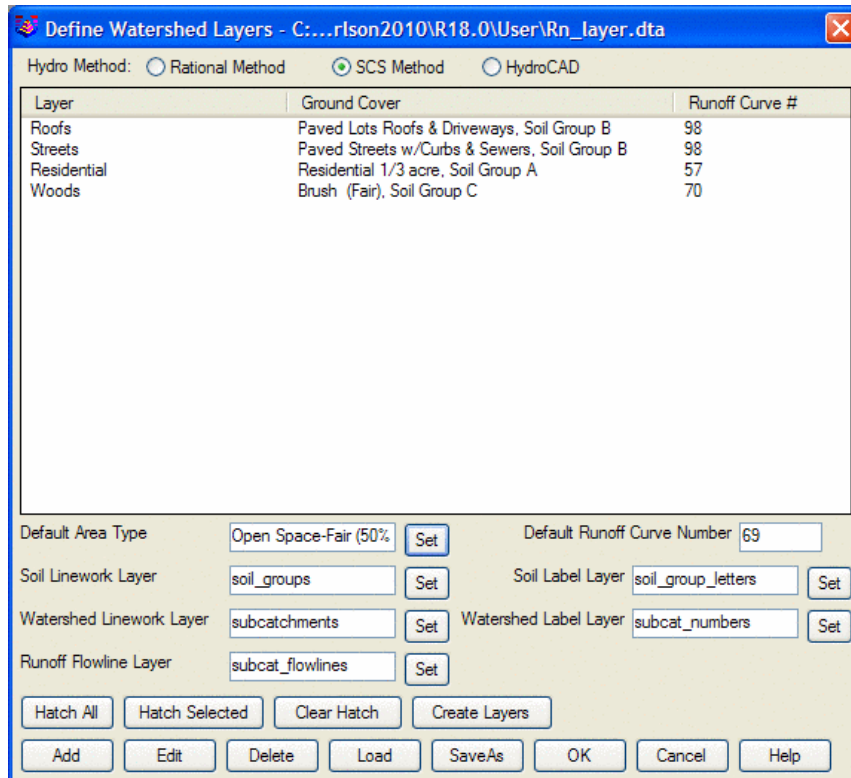


The Library button brings up the next dialog, where new Ground Covers can be added from the Rational Method library.

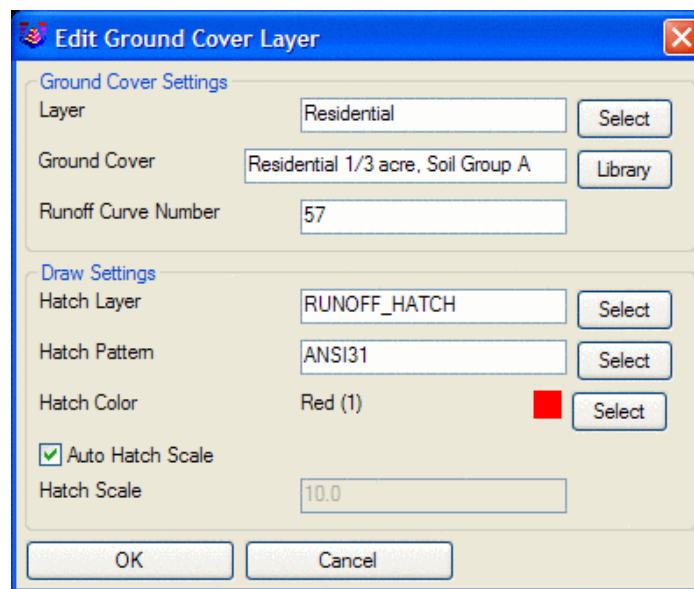


For each layer, an area name and runoff coefficient are assigned and can be selected from the library. This library itself is defined under the Network pulldown menu, option Drainage Runoff Library within the Sewer Network Libraries "flyout". Each layer also has hatch settings for drawing the runoff areas. The hatch settings include the layer, color, pattern and scale. The Auto Hatch Scale option will size the hatch scale to fit the runoff area. The Hatch All button will hatch all the runoff areas in the drawing as closed polylines and defined in the list. The Hatch Selected will hatch the area of the currently selected layer from the list. The purpose of the hatch functions are for visual checks that the layers and closed polylines are set right.

With the SCS Method in use, the primary dialog would look something like this:



Adding or editing a layer brings up the next dialog:



The Library button brings up the next dialog, where new Ground Covers can be added from the SCS Method library. Select a Cover Description line, and then pick a Soil Group button, A, B, C or D.

Runoff Curve Number Library - C:\...R18.0\User\Runoffcn.dta

Cover Description	A	B	C	D
Open Space-Poor (<50% grass)	68	79	86	89
Open Space-Fair (50% to 75% grass)	49	69	79	84
Open Space-Good (>75% grass)	39	61	74	80
Paved Lots Roofs & Driveways	98	98	98	98
Paved Streets w/Curbs & Sewers	98	98	98	98
Paved Streets w/ Open Ditches	83	89	92	93
Gravel Streets (including ROW)	76	85	89	91
Dirt Streets (including ROW)	72	82	87	89
Commercial/Business Districts	89	92	94	95
Industrial Urban Districts	81	88	91	93
Residential 1/8 acre or less	77	85	90	92
Residential 1/4 acre	61	75	83	87
Residential 1/3 acre	57	72	81	86
Residential 1/2 acre	54	70	80	85
Residential 1 acre	51	68	79	84
Residential 2 acres	46	65	77	82
Newly Graded Areas	77	86	91	94
Fallow - Bare Soil	77	86	91	94
Fallow - CR (Poor)	76	85	90	93

Buttons: A, B, C, D, New, Edit, Delete, Load, SaveAs, OK, Cancel, Help

Using the HydroCAD method, the initial dialog might look something like this:

Define Watershed Layers - C:\...rison2010\R18.0\User\Rn_layer.dta

Hydro Method: Rational Method SCS Method HydroCAD

Layer	Ground Cover
HC_grass_poor	<50% Grass cover, Poor
HC_grass_fair	50-75% Grass cover, Fair
HC_grass_good	>75% Grass cover, Good
HC_water_surface	Water Surface
HC_water_surface_0_imp	Water Surface, 0% imp
HC_paved_parking	Paved parking
HC_unconnected_pavement	Unconnected pavement
HC_roofs	Roofs
HC_unconnected_roofs	Unconnected roofs
HC_paved_roads_w_curb_and_s...	Paved roads w/curbs & sewers
HC_paved_roads_w_open_ditches	Paved roads w/open ditches, 50% imp
HC_gravel_roads	Gravel roads
HC_dirt_roads	Dirt roads
HC_urban_commercial	Urban commercial, 85% imp
HC_urban_industrial	Urban industrial, 72% imp
HC_one_eighth_acre_lots	1/8 acre lots, 65% imp
HC_one_quarter_acre_lots	1/4 acre lots, 38% imp
HC_one_third_acre_lots	1/3 acre lots, 30% imp
HC_one_half_acre_lots	1/2 acre lots, 25% imp

Default Area Type: Woods, Good [Set] Default Runoff Curve Number: 0

Soil Linework Layer: soil_groups [Set] Soil Label Layer: soil_group_letters [Set]

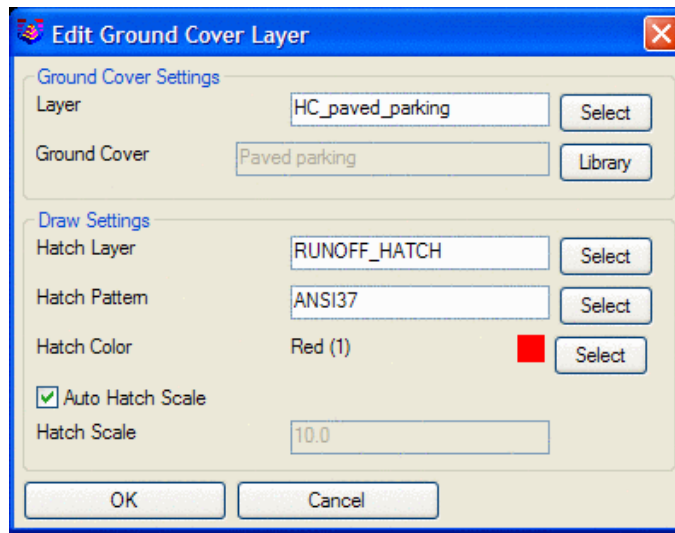
Watershed Linework Layer: subcatchments [Set] Watershed Label Layer: subcat_numbers [Set]

Runoff Flowline Layer: subcat_flowlines [Set]

Buttons: Hatch All, Hatch Selected, Clear Hatch, Create Layers, Add, Edit, Delete, Load, SaveAs, OK, Cancel, Help

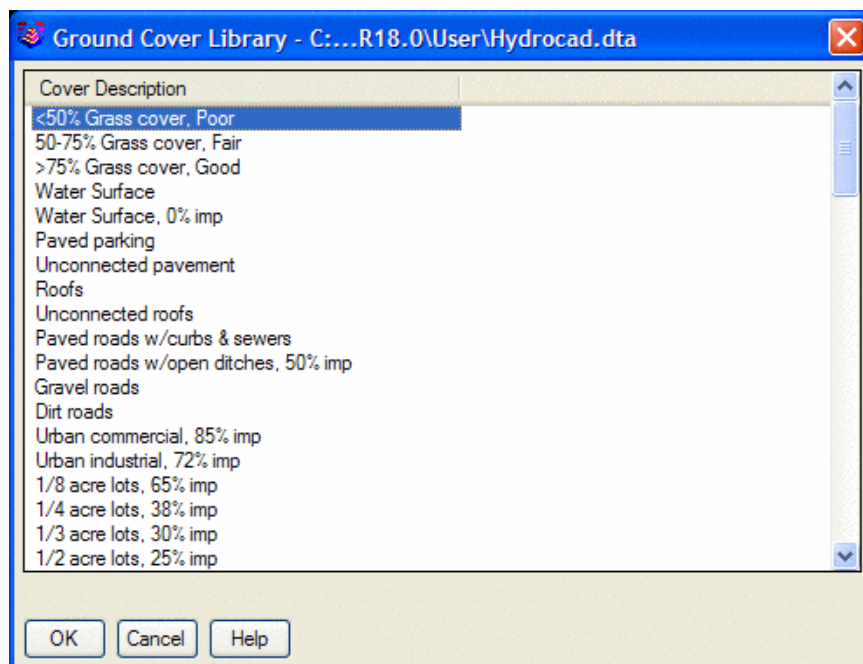
Note there are no Curve Numbers listed, and the Default Runoff Curve Number field is grayed out.

Editing a layer brings a dialog like this:



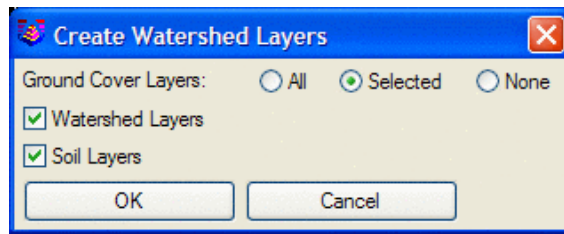
Note that you can change the Layer name, but not the description of the Ground Cover. This is because the specific descriptions provided for HydroCAD Ground Covers must exactly match the description specified within HydroCAD itself.

Picking the Library button accesses the HydroCAD Ground Cover library.



Back in the primary Define Watershed Layers dialog box, the list of layers and other settings can be saved as .RCL files. It is useful to save and recall the configurations in .RCL files using the SaveAs and Load options. The currently loaded assignment is applied within the command Watershed Analysis.

The Create Layers button creates the layers from the list in the current drawing. All layers can be created, or certain layers can be selected before picking the Create Layers button and you can specify to create only the selected layers.



The Color TIN button applies the colors defined for the ground cover areas to colorize a triangulation file which is helpful for visualizing the surface in 3D viewers.

There are settings for the default area name and default coefficient that are used for any part of the drainage area that is not covered by one of the runoff layer polylines.

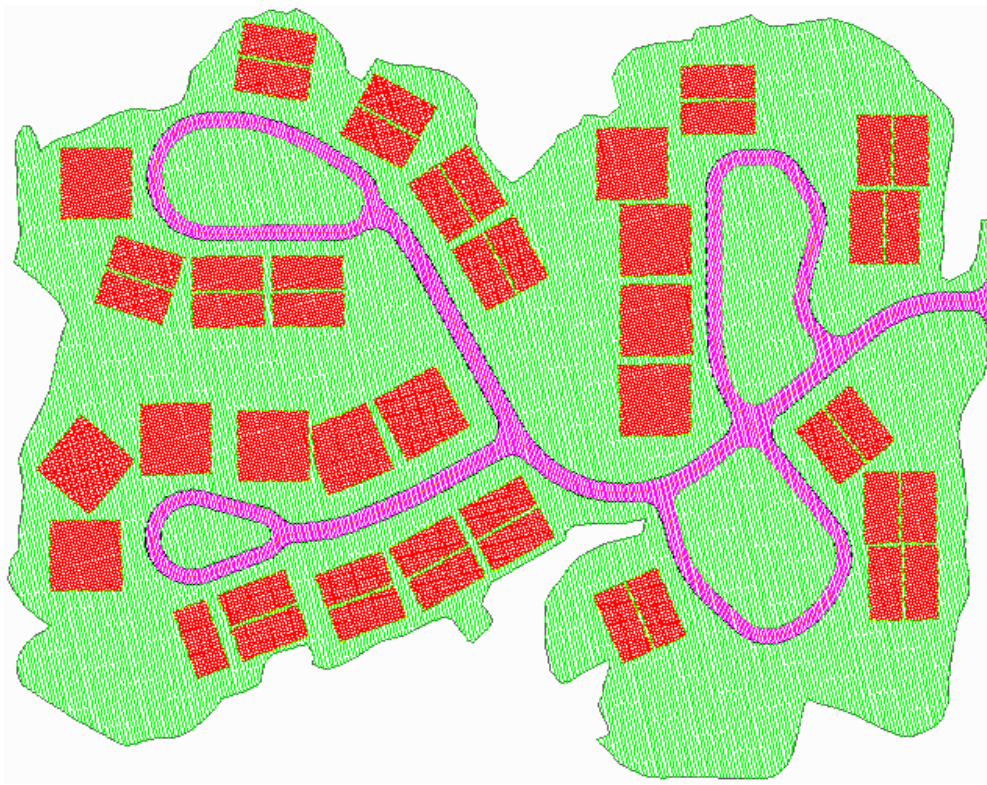
The Soil layers are optional for finding the soil sub-areas within each runoff sub-area. These Soil layers are used in commands that calculate the runoff sub-areas within a watershed boundary such as the Select Watershed function within the Curve Numbers & Runoff command. When the Soil layers are assigned, the program will get all the linework on the Soil Linework layer to build a topology of the soil areas. The linework does not need to be made of closed polylines but the linework collected together should enclose the soil areas. Then the program takes the text entities in the drawing that are on the Soil Label layer. The text is used to identify the soil group. The first character in the text should be A, B, C or D for the four soil groups. The program looks for the text to be within the soil area to assign the soil group to that area.

The Watershed Linework layer is used in commands for selecting the watershed area by picking a point within the area. For instance, the Select Watershed routine in Curve Numbers & Runoff will prompt whether to select by perimeter (closed polyline) or interior point (watershed layer method), when the Watershed layer is defined. Similar to the Soil Linework layer, the program will get all the linework in the drawing that is on the Watershed layer and build a topology of the watershed areas. The linework does not need to be made of closed polylines but should make closed watershed areas when taken together. When you pick the interior point, the program finds the watershed linework that encloses the point to get the watershed perimeter.

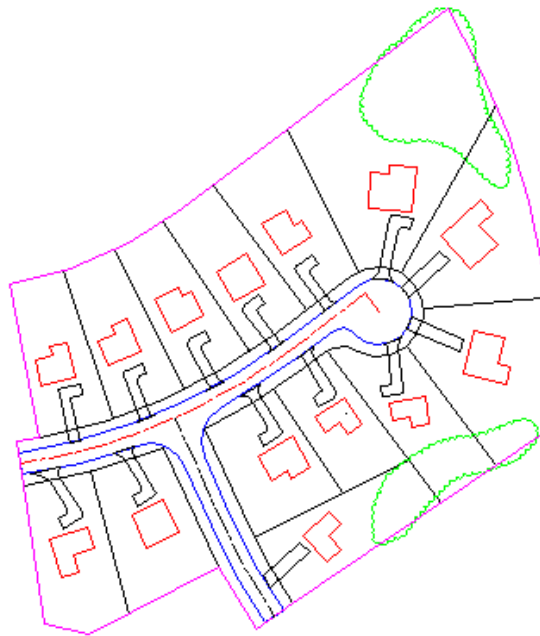
The Watershed Label layer is used by the Hydro Network commands to match the drawing watershed area with the subcatchment node in the network. The match is between the value of a text entity on the watershed layer with the name of the subcatchment node.

The runoff polyline areas use region logic where a polyline inside another on the same layer is used as an exclusion. A limitation is that polylines on the same layer must not intersect each other. For polylines on different layers, there can be polylines within other polylines and for any given point, the smallest enclosing polyline is used to determine the runoff coefficient.

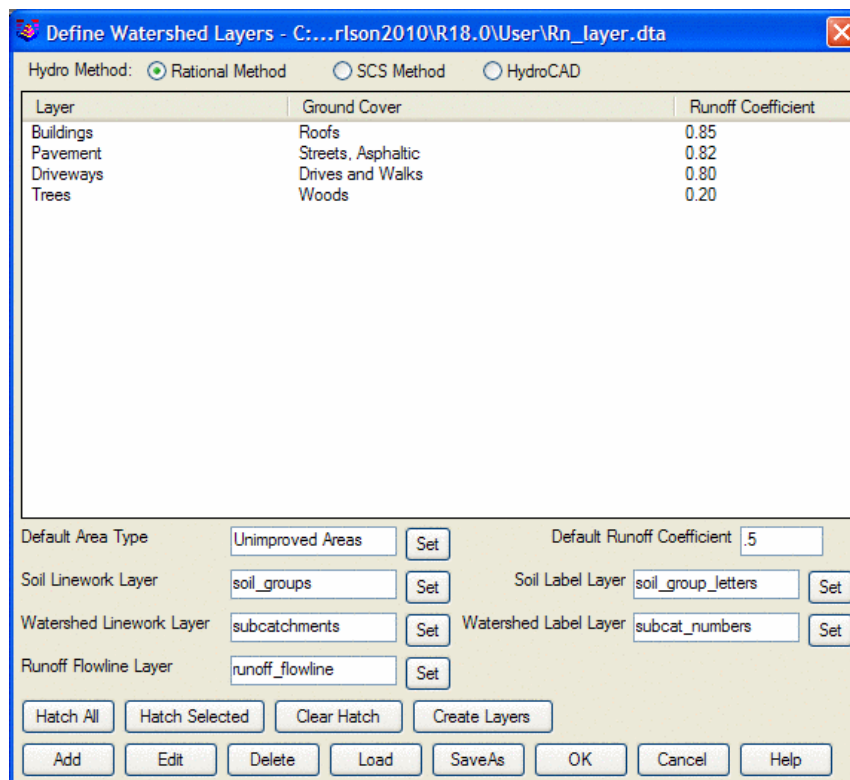
Example 1: In the example below, the site perimeter polyline is on the Regions layer, the building pads are on the Pads layer and the edge of pavement polylines are on the Roads layer. All these polylines are closed polylines. The areas within the buildings are inside both the Region and Pads polylines and the Pads govern because they are the smaller area. Likewise the road areas are governed by the Roads layer and road interior islands are not counted for Roads because the interior Roads polyline acts as an exclusion perimeter. The rest of the area is set to the Regions layer.



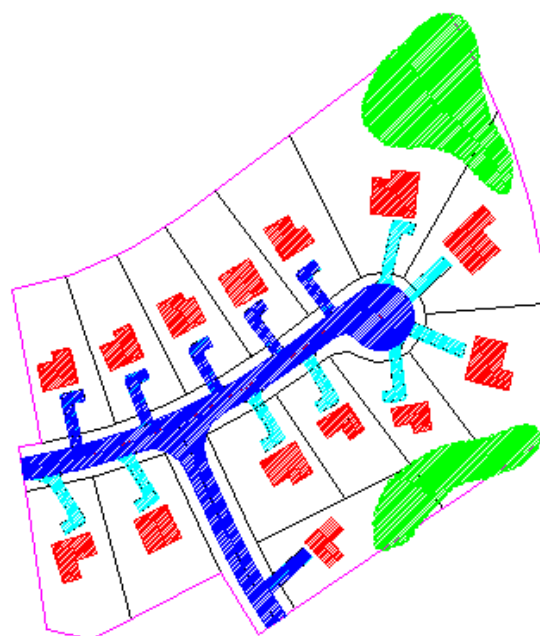
Example 2: Consider the subdivision shown below.



Buildings, roads, driveways, lot lines and wooded areas are in distinct layers. As soon as the command is selected the dialog below appears. The applicable layers can then be organized as follows within the command. Note that the lot lines do not have any hydrology impact and are not included in the layer-runoff coefficient assignment.



Example 1 used the built-in logic to remove closed polylines from outer enclosing closed polylines. So in the example 2 case, the overall property boundary had a runoff coefficient of 0.2 that was assigned its runoff coefficient by layer, and all other assigned closed polylines found within it (roads, buildings, driveways) will be calculated distinctly. For example 2, the entire "remainder" area that is not assigned and is given a default runoff coefficient, such as 0.5 shown above. Therefore, within any site perimeter, both the "unassigned" method for remainder areas or the assigned, outer boundary layer method for the remainder areas can be used. When the "Hatch All" button is clicked, the drawing will hatch in the defined colors and layers, as shown below:



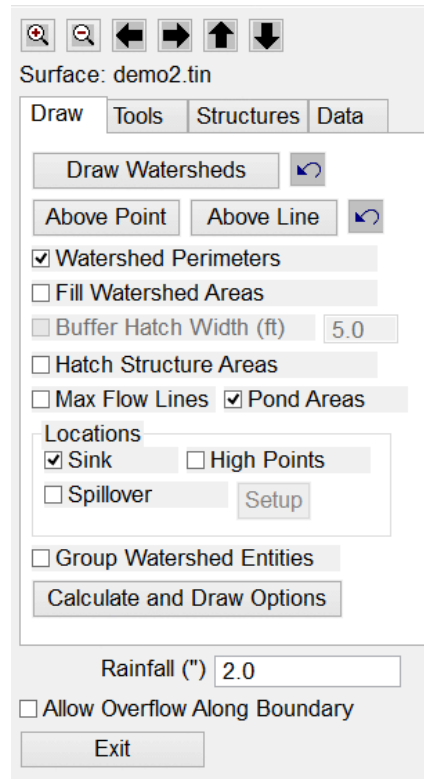
Pulldown Menu Location: Watershed

Keyboard Command: define_runoff_layers

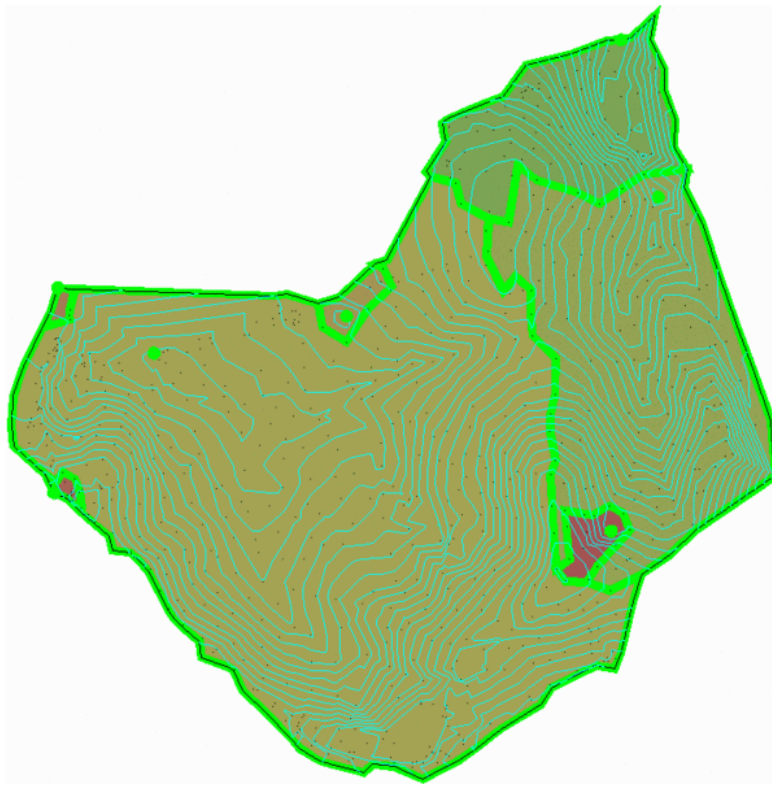
Prerequisite: Closed polylines on different layers for the different areas

Watershed Analysis

This command has a collection of tools to analyze the runoff of a surface defined by a triangulation or grid surface file. After selecting the surface file of the surface, the program docks a dialog on the left side of the drawing window. While the Watershed Analysis dialog is running, other AutoCAD and Carlson commands are not available. To zoom or pan the drawing view, use the buttons at the top of the dialog, or use the middle button of a wheel-mouse.



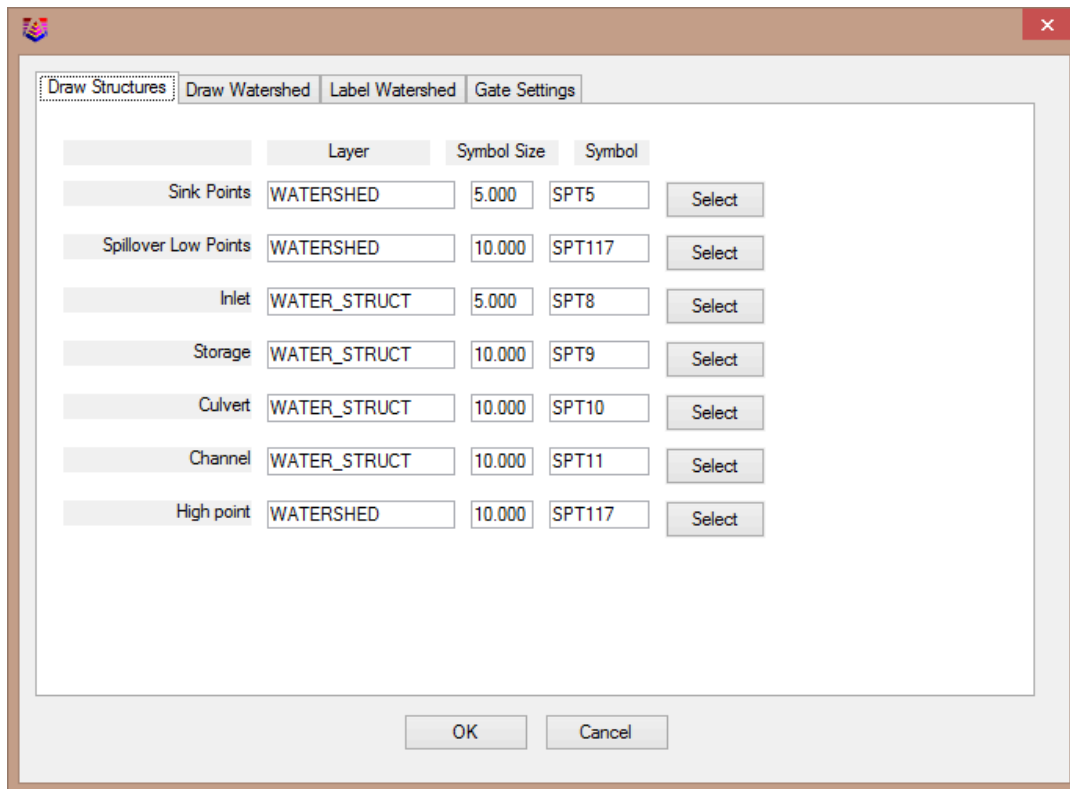
Watershed Analysis calculates the flow connections between the triangles and along the edges of the triangulation. The **Rainfall** amount is used in the processing for figuring the runoff volume to determine when the volume is enough to spillover a local depression in the surface. Besides the Rainfall amount, the runoff coefficients as defined in Define Runoff Layers are also used to calculate the runoff volumes. When the local depression is small enough the runoff will continue through. Otherwise this spot is called a sink for where the runoff stops. The **Allow Overflow Along Boundary** option applies to watersheds that have runoff that hits the surface border. This option will check whether this border runoff can spillover and merge with the neighboring watersheds along the border.



The **Draw Watersheds** function draws the watershed areas using the settings under the Draw tab. The back arrow next to the Draw Watersheds button will erase any previous Draw Watershed entities. The **Watershed Perimeters** option will draw closed polyline perimeters for each watershed area. The **Fill Watershed Areas** option will solid fill hatch each area using different colors. The **Buffer Hatch** option will hatch the perimeters of the watershed areas with the specified width instead of hatching in the entire watershed area. The **Hatch Structure Areas** option will hatch the drainage areas covered by structure inlets defined in the Structures tab. The **Sink Locations** setting draws a symbol at the low point for each drainage area. The **High Point Locations** option draws a triangle symbol at the highest point within each watershed. Typically, this high point will be along the watershed boundary polylines that follow the high points along the ridges between the watersheds. The **Pond Areas** option draws a solid fill hatch in blue for the area covered by the runoff volume of low points. In the example shown, the Fill Watershed Areas and Sink Locations options are active. The **Max Flow Lines** option draws polylines for the longest flow line within each watershed. These longest flow polylines can be used to calculate the time of concentration. The **Group Watershed Entities** option will make AutoCAD groups for the set of entities drawn for each watershed.

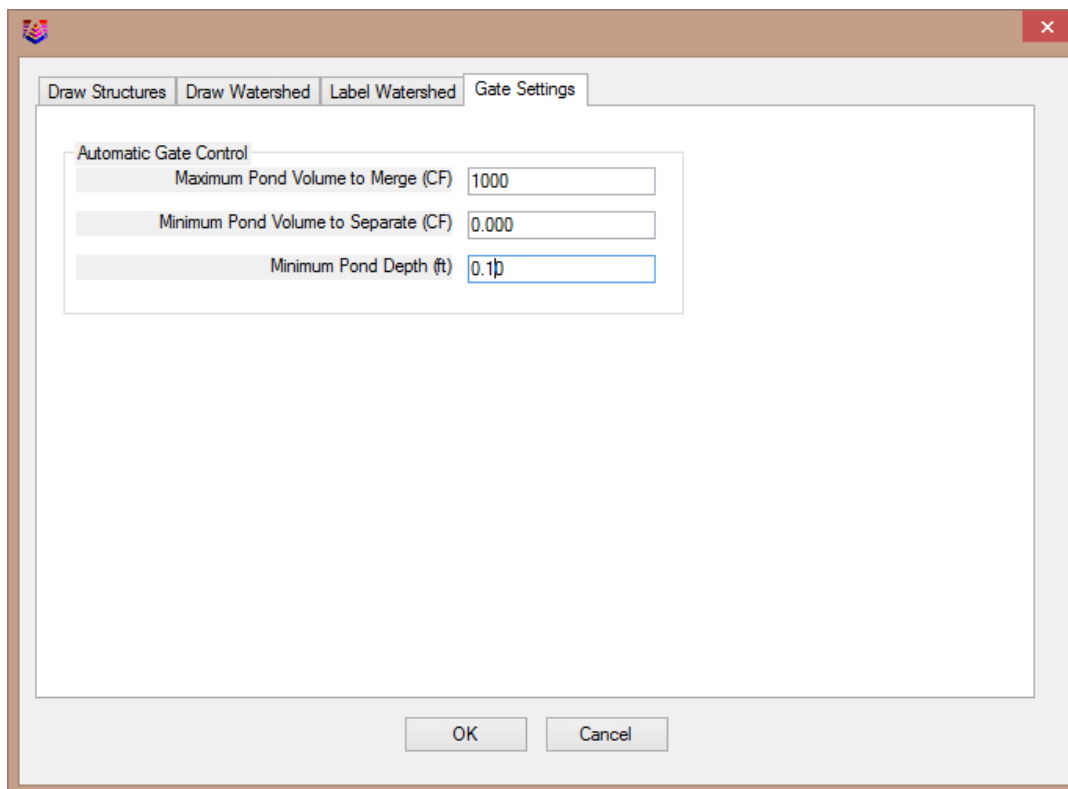
The **Spillover Location** option draws symbols at low points within the watershed area that fill up with runoff and spillover on the way to the lowest (sink) location of the watershed. The Setup button allows you to specify criteria for identifying spillover points. These settings include the minimum drainage area, storage volume, drainage volume.

These settings allow you to filter out small spillover points (ie a pothole) and only draw the significant ones. The **Calculate and Draw Options** buttons allow you to control what is going to be drawn how, it contains settings for the symbol, size and layers to use for the entities created by Watershed Analysis and a number of labeling options.



Gate Control

The watershed calculation can be controlled through use of "gates". Gate is placed at the points of likely overflow from one watershed to another. The gate in its natural state is simply an indication that potentially the connection can occur at the point, joining two watershed areas into one if there is too much runoff volume in at least one of the watershed areas to be contained within. The gate can also be forced closed, which indicates that two areas will not be joined regardless the runoff amount. The example of such case is when there are two large ponded areas you want to treat separately in calculations or further design. On flip side, there are instances when you may want two areas to be joined regardless the overflow actually occurring, for example one area is very small. This is accomplished with a gate forced open. There two ways to control gate state: automatically and manually. Here are the settings for automatic control.



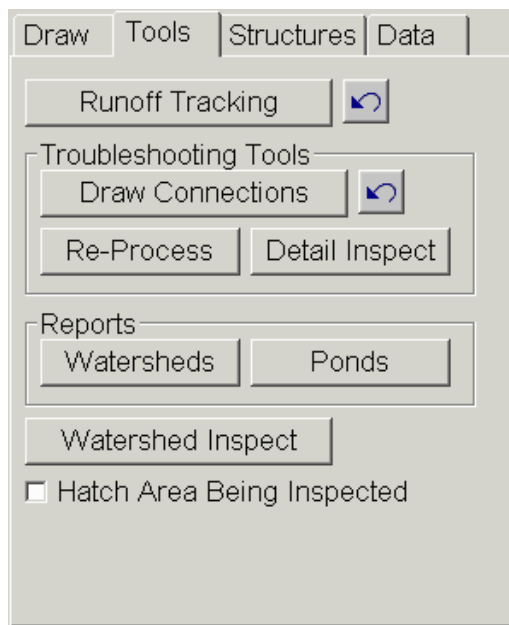
If amount of ponded runoff in an area exceeds the maximum pond volume to merge, the downstream gate will be closed automatically. If area is too small or too shallow, its downstream gate will be forced open, joining it with watershed downstream.

To control gates manually, please first draw gate labels, exit Watershed Analysis, then double-click on gate labels in the drawing to control their state. Once done editing the gate state, just re-run the calculation to draw and report the new adjusted watersheds.

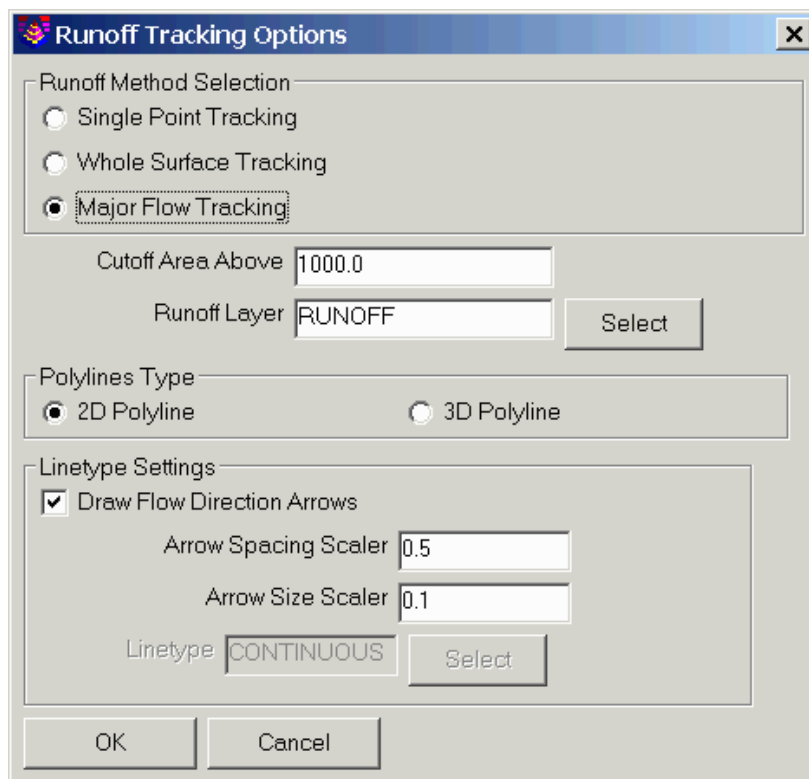
Runoff Tracking

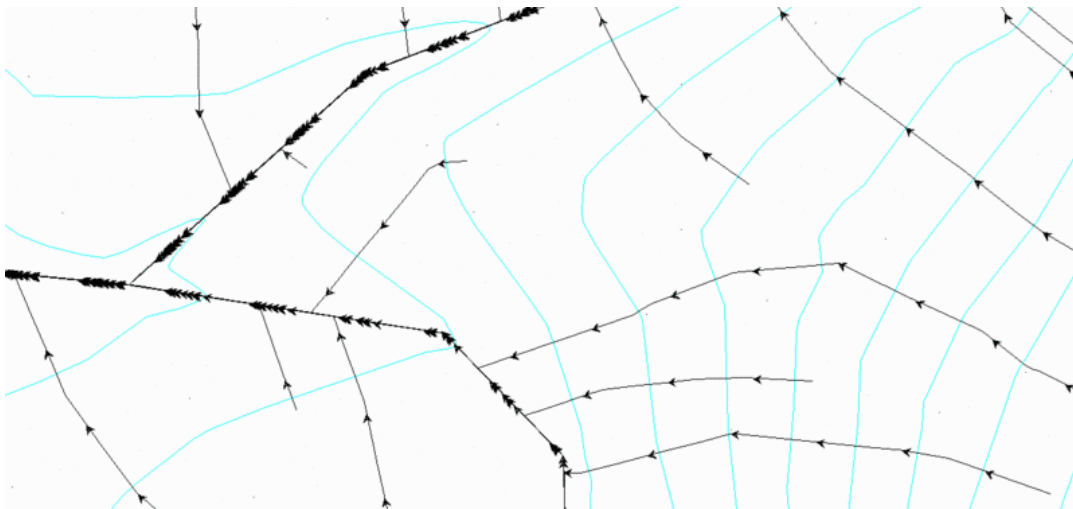
The **Above Point** function reports the watershed data of the current pointer position in real-time as the pointer is moved around. The watershed data is shown in a tooltip next to the pointer position. This data has values for the overall watershed that the position is in including the sink elevation, sink name, drainage area and average slope percent. This data also has values for the watershed above the current point including the drainage area and runoff volume. Plus this data shows the elevation and runoff coefficient at the current point. If the position is picked with the mouse, then the program draws a polyline perimeter for the drainage area above the current point.

The **Above Line** function is similar to Above Point except that you pick two points and the program draws the watershed for all flow that crosses the line between these two points. For example, you can pick points at the left and right banks of a stream to get the drainage area for that stream above these points.

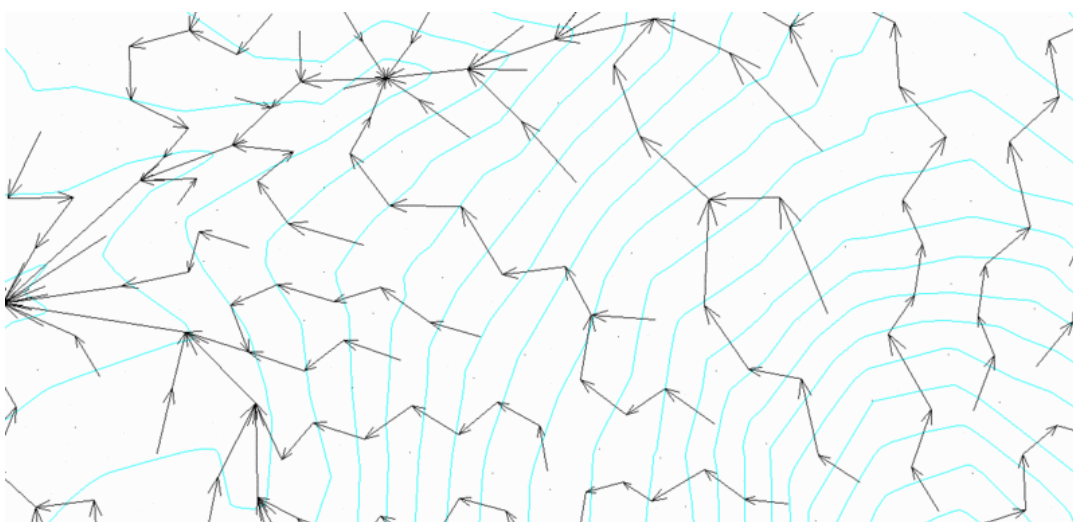


Under the Tools tab there are several analysis routines. The **Runoff Tracking** function draws flow lines that follow the surface. The **Single Point Tracking** method draws the flow lines starting from the picked high points. The **Whole Surface Tracking** method draws a flow line starting from the middle of each triangle in the triangulation. The **Major Flow Tracking** method draws starting in triangles where the drainage area coming into triangle exceeds the specified **Cutoff Area Above** value. The flow lines can be drawn as either 2D or 3D polylines. For 2D polylines, the linetype can be specified or the special linetype with flow direction arrows can be used. This special flow linetype has controls for the size and frequency of the flow arrows.





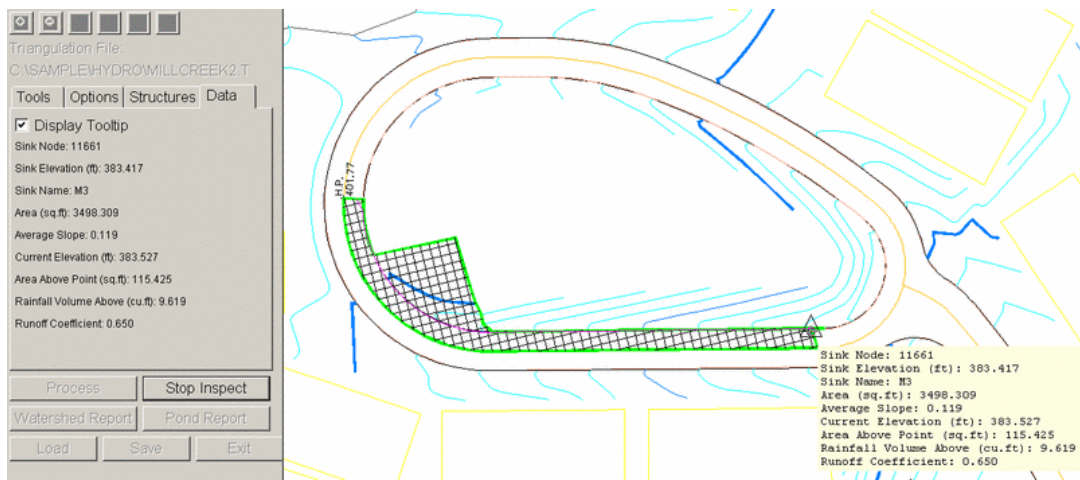
The **Draw Connections** function draws lines with arrows between the triangles for how the program has determined their flow connections.



When a triangulation file is processed by Watershed Analysis, some of the flow connection data is stored into the triangulation file to speed up reprocessing. The **Re-Process** function resets this flow connection data to start the flow calculations from scratch.

The **Detail Inspect** function reports flow connection data at the pointer position in real-time as the pointer is moved. This data includes the current position triangle number, connecting flow triangle number, sink node number, watershed name, border elevation, ridge elevation, low elevation, downstream sink number, number of source triangles, number of source nodes, current elevation and spillover elevation.

The **Watershed Inspect** function reports runoff flow data at the pointer position in real-time as the pointer is moved. The runoff data is shown in a tooltip next to the pointer and in the **Data** tab. This data has values for the overall watershed that the position is in including the sink elevation, sink name, drainage area and average slope percent. This data also has values for the watershed above the current point including the drainage area and runoff volume. Plus this data shows the elevation and runoff coefficient at the current point. When the Hatch Area Being Inspected option is active, the watershed area for the current position is hatched during inspection.



The **Watersheds Report** function runs the report formatter to choose which of the watershed parameters to report. The **Ponds Report** function reports the position and depth of each ponding area.

Here are some of the values contained in watershed report:

Rain volume - total volume of the runoff for the area

Holding volume - the maximum volume the watershed can contain near the sink

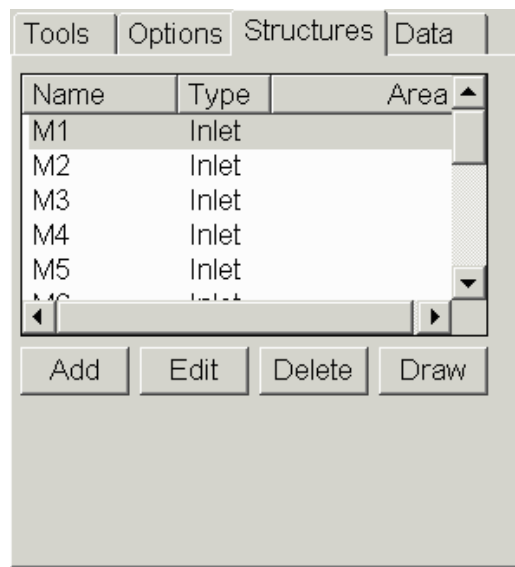
Ponding volume - the volume of all the ponds within the watershed

Uncontained volume - the difference between amount of runoff and the volume of runoff trapped on the slope or at the sink

Additionally, the properties of the pond at the sink are reported: surface elevation, max depth, volume and area.

Structures

Besides calculating the runoff of the triangulation surface, Watershed Analysis can also process the runoff effects from structures for inlets, storage ponds, culverts and channels. The structures in Watershed Analysis are simply for placement and watershed delineation. These structures do not have design considerations for parameters like pipe size. In the **Structure** tab, there is a list of the structures to apply with the current surface. The list shows the name, type and drainage area for each structure. The Draw function will draw symbols for each structure. The Inlet structures act as sinks in the watershed and capture all the flow that comes to the inlet point. Each inlet is defined by a single point and a name. The Storage Tank structures also act as sinks and are defined by a single point and name. The Culvert structures route the flow from the culvert inlet to the outlet. The culverts are defined by two points for the inlet and outlet and by a name. The Channel structure is the same as the Culvert except that it can have more than two points to define the flow path. The structure data can be stored to a Watershed Structure File (wst) using the **Save** button. The **Load** button can read the structure data from either a wst file or from a sewer network file (.sew).



Pulldown Menu Location: Watershed

Keyboard Command: watershed

Prerequisite: Triangulation File

Merge Watersheds

This command combines the polyline perimeters of two neighboring watersheds into a single polyline. The original two polylines are erased. The original polylines must have a common side and must be closed polylines on the layer of Watershed Linework Layer defined in the Define Watershed Layers command. To select the watershed polylines to merge, the program prompts to pick a point inside the watershed area.

Prompts

Pick inside 1st watershed: *pick a point*

Pick inside 2nd watershed: *pick a point*

Pulldown Menu Location: Watershed

Keyboard Command: wshed_merge

Prerequisite: closed polylines on watershed layer

Run Off Tracking

This command draws 3D polylines starting at user picked points downhill until they reach a local minimum or the end of the grid or TIN. In effect it simulates the path of a rain drop. The surface is modeled by a grid file as created by Make 3D Grid File or a triangulation file created by Triangulate & Contour. The program also reports the horizontal and slope distances, average slope, maximum slope, and vertical drop. These values can be used for time of concentration calculations. Runoff tracking is a convenient way to identify distinct watershed areas and is an alternative to the automated Watershed Analysis command.

Prompts

Enter the run off path layer <RUNOFF>: *press Enter*

Select Surface Model dialog box

Choose the grid file or triangulation file that models the surface. If a grid is selected, it will prompt:

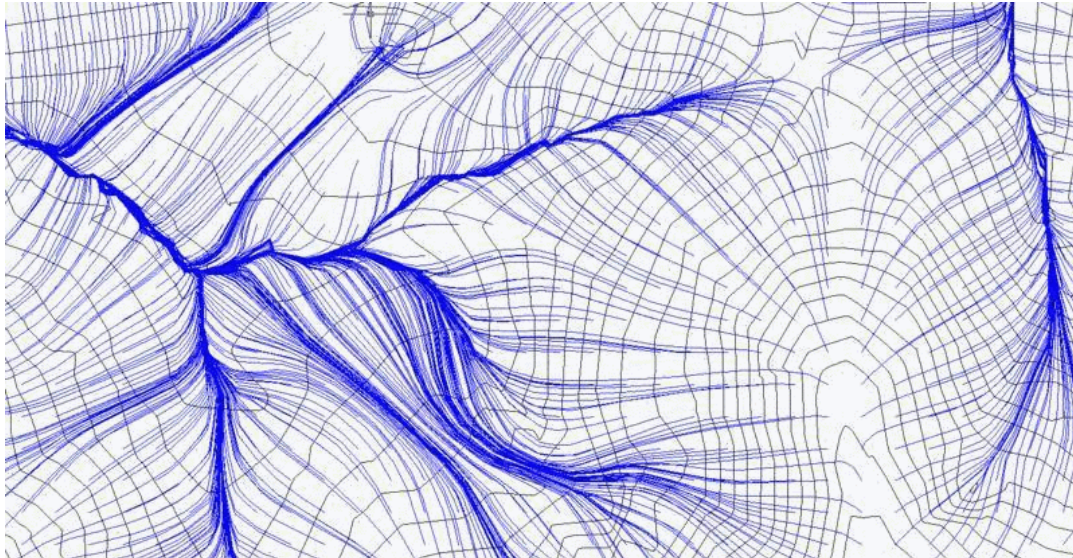
Extrapolate grid to full grid size (Yes/<No>)? *Yes* If the limits of the surface data doesn't cover the entire grid area, then the values for the grid cells beyond the data limit must be extrapolated in order to compute slopes in that area. This prompt only appears if there are grid cells without values.

Local pond spillover depth <4.80>: *press Enter* This allows the runoff line to continue past flat or low points in the grid or TIN, by allowing these area to fill up with water, in essence, up to the specified depth, thus letting the runoff polyline continue on.

Draw tracking for all grid cells or pick individuals [All/<Pick>]: *press Enter* Pressing Enter leads to individual picking of runoff tracking lines, while A for All would fill draw runoff polylines starting from each grid cell or each triangulation triangle.

Pick origin of rain drop: *pick a point at the top of the run off polyline*

Pick origin of rain drop (Enter to end): *press Enter*



Pulldown Menu Location: Watershed

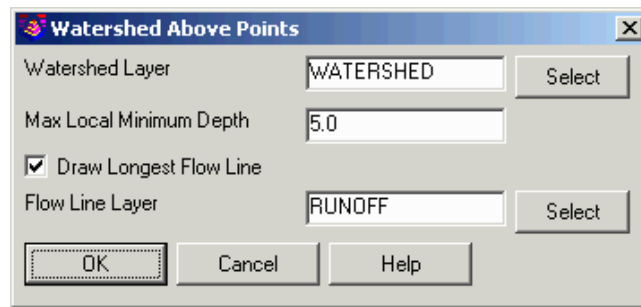
Keyboard Command: runoff

Prerequisite: A .grd file created by Make 3D Grid File or a .flt (TIN) file created by Triangulate & Contour.

Watershed Above Points

This command creates a closed 3D watershed polyline of the area that flows through a line connecting two points. The command also creates a watershed report of the area, slopes and longest flow line. There is also an option to draw a 3D polyline for the longest flow line. In the options dialog, the Max Local Minimum Depth is the most the program will allow the flow to jump over.

This command uses a surface model defined by a grid file. The Watershed Analysis method is another command to generate watersheds that uses triangulation surfaces. A triangulation surface model has the advantage of finding sharp breaks that a grid might skip over such as for narrow ditches or curbs. For regular surfaces such as a surface defined by existing contour polylines, the grid surface model has the advantage of efficiently modeling the surface which makes it faster. Also when using a grid surface model, it is sometimes better to decrease the grid resolution (have fewer grid cells) because this helps avoid local minimums.

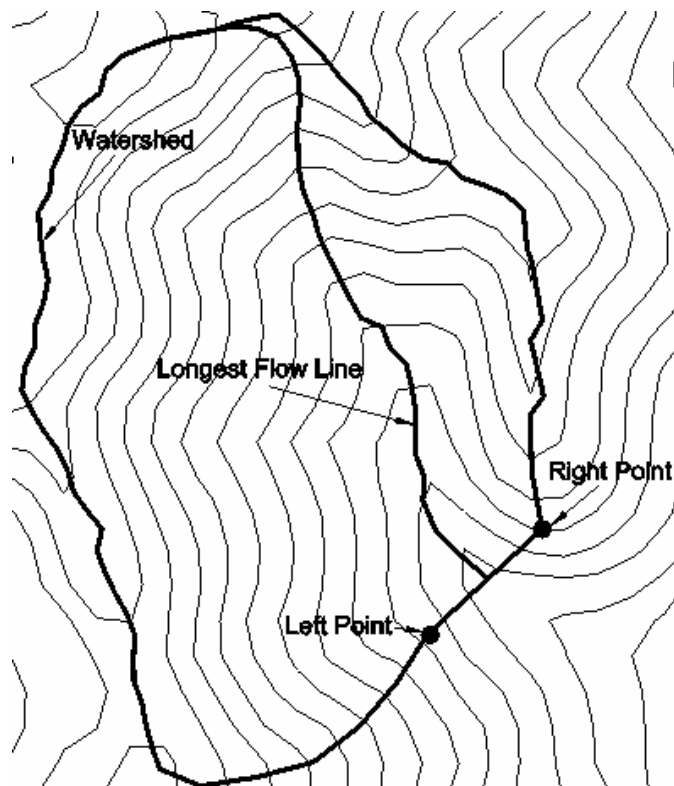


Watershed Report

Surface: C:\sample\Quads3b.grd
Lower left grid corner : 940321.80,365189.63
Upper right grid corner: 967093.86,381003.90
X grid cell size: 92.00, Y grid cell size: 98.23

Horizontal Area: 2156534.4 sq ft, 49.507 acres
Slope Area: 3196442.4 sq ft, 73.380 acres
Average Elevation: 1990.46
Average Slope: 52.30%

Longest horizontal distance: 2062.0, Longest slope distance: 2199.2
Vertical drop: 710.0, Avg slope: 34.4%, Max slope: 71.0%



Prompts

Select Surface Model *Choose .grd file*

Watershed options dialog

Pick bottom LEFT bank of watershed: *pick a point*

Pick bottom RIGHT bank of watershed: *pick a point*

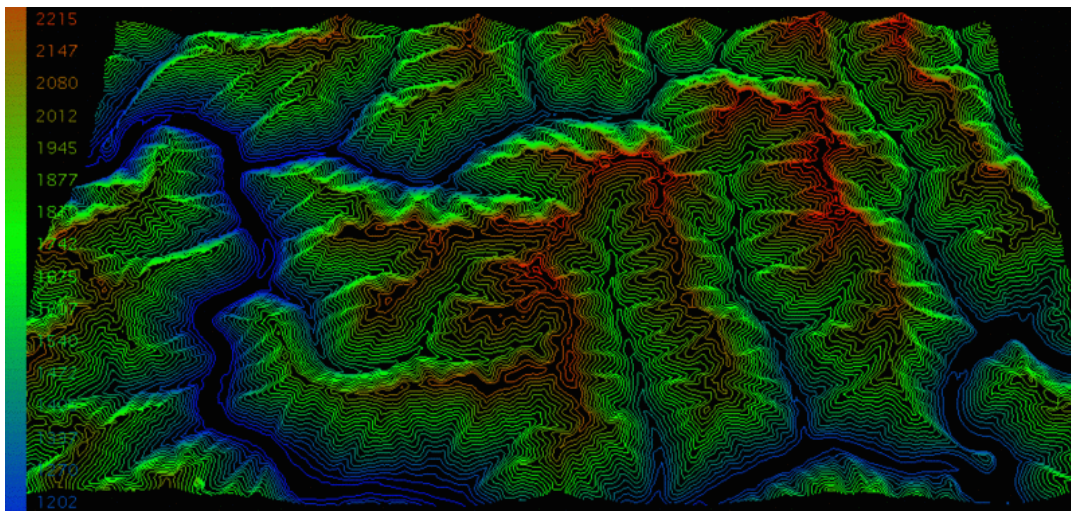
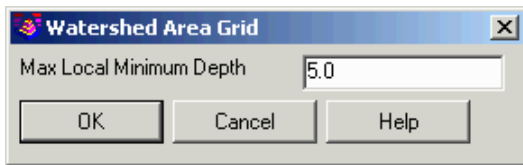
Pulldown Menu Location: Watershed > Watershed By Grid Surface

Keyboard Command: waterabove

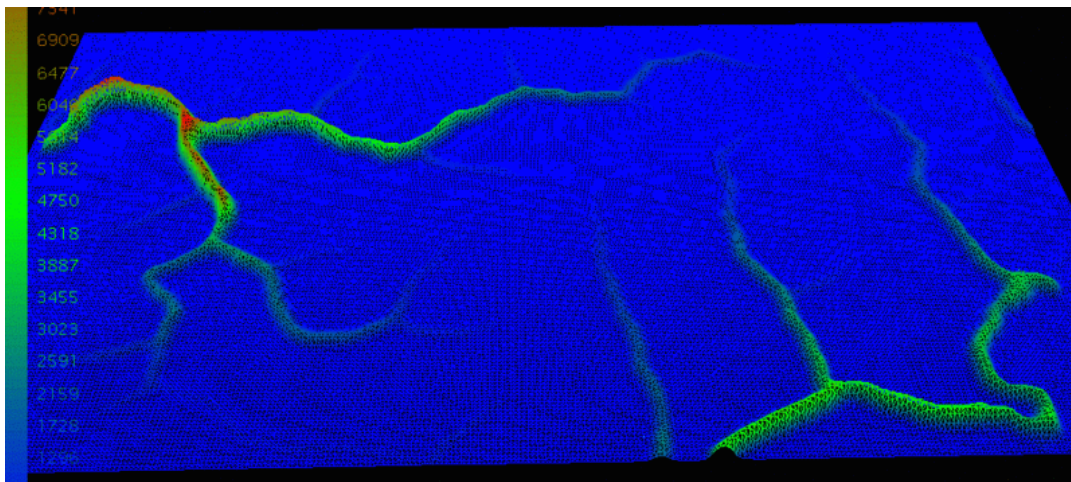
Prerequisite: surface .GRD file

Watershed Area Grid

This command creates a grid model of the drainage area above each node. A grid of the surface is used for input. For each grid node, the program calculates the drainage area above this node. The Max Local Minimum Depth is used to merge drainage areas that have a depth less than this specified value. The units of the area grid are square units of the horizontal dimension of the surface grid. When the surface grid is in feet, the area grid values are in square feet. Likewise a surface grid in meters will make an area grid in square meters. The values of the area grid can get very large and difficult to view in 3D. To handle these large values, you can use Grid File Utilities math functions to scale down the values.



3D view of contours from original surface grid



3D view of watershed area grid

Pulldown Menu Location: Watershed > Watershed By Grid Surface

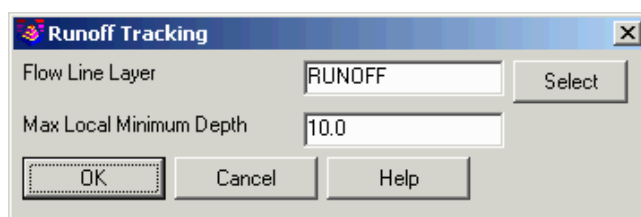
Keyboard Command: watergrd

Prerequisite: surface GRD file

Run Off Tracking - Grid Surface

This command draws a 3D polyline starting at a picked point and going downhill until reaching a local minimum or end of the surface. In effect this flow line simulates the path of a rain drop. The program also reports the distance and slope of the flow line. These values can be used in time of concentration calculations. This command uses a grid model for the surface which works well for existing ground surfaces. The Run Off Tracking command that is in the main Watershed menu works with triangulation surfaces which works well for design surfaces.

In the options dialog, the Max Local Minimum Depth is the most the program will allow the flow to jump over.



Prompts

Select Surface Model *Choose .grd file*

Watershed options dialog

Pick origin point of rain drop: *pick a point*

Horizontal distance: 10315.58, Slope distance: 10392.95

Vertical drop: 982.08, Avg slope: 9.52%, Max slope: 52.34%

Pick origin point of rain drop (Enter to end): *press Enter*

Pulldown Menu Location: Watershed > Watershed By Grid Surface

Keyboard Command: runoff_grd

Prerequisite: surface .GRD file

3D Polyline Flow Values

This command simply reports the horizontal and slope distances, vertical drop, maximum slope, and average slope of a 3D polylines or Carlson Profile File(.pro). 3D polylines may be created by the Watershed Analysis or Run Off Tracking commands. The profile can be any vertical alignment saved as a .pro file. When using the profile file option, The profile file can represent the vertical alignment of a swale or channel. The reported values could be applied to the Time of Concentration routine.

Prompts

Type of flow line [<3DPolyline>/Profile]? 3DP

Select 3D polyline flow line or Enter to end:

Horiz dist: 198.92, Slope dist: 198.95, Vertical drop: 3.37 Average slope: 1.69%, Maximum slope: 1.69%

Select 3D polyline flow line or Enter to end: Enter

Command:flowvals

Type of flow line [<3DPolyline>/Profile]? profile

Select profile file from file dialog

Horiz dist: 198.92, Slope dist: 198.95, Vertical drop: 3.37 Average slope: 1.69%, Maximum slope: 1.69%

Pulldown Menu Location: Watershed

Keyboard Command: flowvals

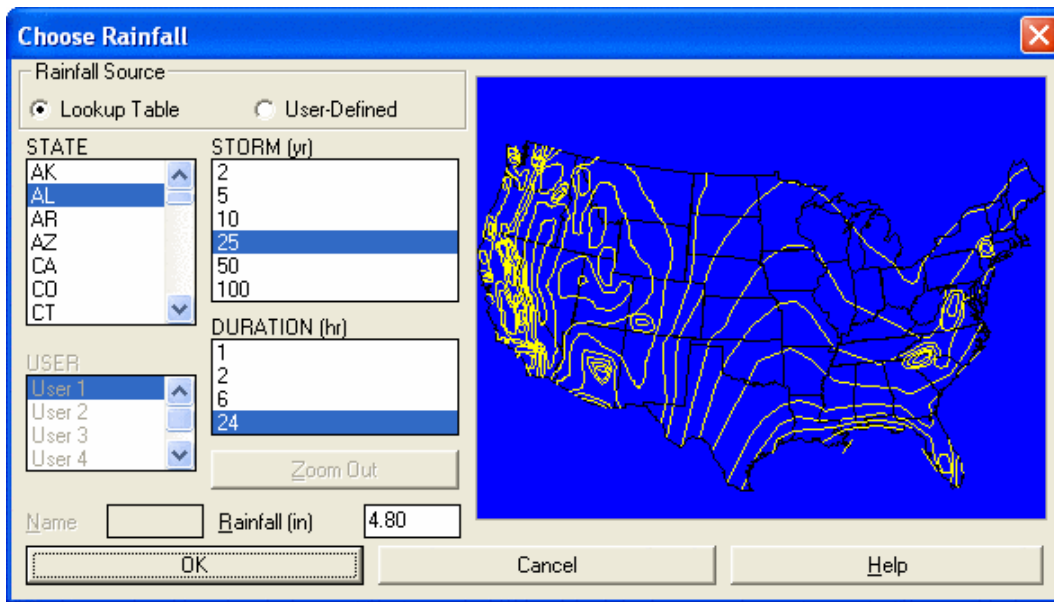
Prerequisite: 3D polyline or Vertical Profile file(.pro)

Rainfall Frequency and Amount

This command allows you to view rainfall maps while entering the rainfall amount to be used by other hydrology commands. First choose a storm and duration from the list. Then choose your location from the State list or pick your location on the map. You can enter the rainfall amount in the box in the lower left or pick your location on the map.

Reference maps based on TP-40 and TP-47 are provided for all fifty states for the different storm intervals. You can also setup user-defined lookup tables for up to five areas. For each area, you can specify a name and rainfall amounts for each storm interval. The first time the you select a user-defined storm interval, the rainfall amount will be blank. Enter in the rainfall amount and the next time that interval is selected, your entered value will be there. All rainfall amounts are in inches. The user-defined values are stored in a file called rainmap.ini in the Carlson USER directory.

TP-40 and TP-47 are older rain tables. The NOAA Atlas 14 is a more recent rain table.



Pulldown Menu Location: Watershed

Keyboard Command: rainmap

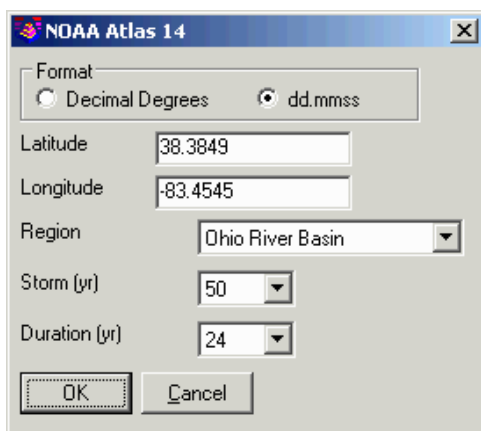
Prerequisite: None

Rainfall Frequency and Amount - NOAA Atlas 14

This command looks up the rainfall depth from NOAA Atlas 14 for the specified rain event and location. In the dialog, enter the lat/lon and NOAA region of the location and choose the storm event. The first time that you look up a storm event for a region, then program will download the rainfall model from NOAA. The rainfall depth is reported at the command line.

Currently, data is available for all US States except the northwest: WA, OR, ID, MT and WY.

For information on NOAA Atlas 14, go to the NOAA website at: <http://hdsc.nws.noaa.gov/hdsc/pfds/>



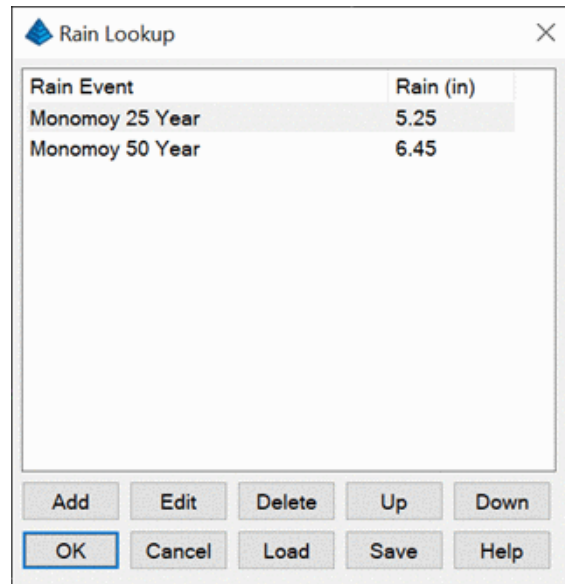
Pulldown Menu Location: Watershed

Keyboard Command: noaa14

Prerequisite: None

Rainfall Frequency and Amount - Lookup Table

This command stores a list of rain event names and their rainfall depths. To set the current rainfall depth for the Watershed routines, double-click on a rain event from the list or highlight the rain event on the list and pick OK. Use the Add, Edit, Delete, Up and Down buttons to setup the rain events. The Load and Save buttons store and recall the rain events to a .rain file.



Pulldown Menu Location: Watershed

Keyboard Command: rain_lookup

Prerequisite: None

USDA Soil Area Download

This command stores a list of rain event names and their rainfall depths. To set the current rainfall depth for the Watershed routines, double-click on a rain event from the list or highlight the rain event on the list and pick OK. Use the Add, Edit, Delete, Up and Down buttons to setup the rain events. The Load and Save buttons store and recall the rain events to a .rain file.

Pulldown Menu Location: Watershed

Keyboard Command: rain_lookup

Prerequisite: None

Sub-Watersheds By Land Use

This command divides land-use polylines into closed polylines within a watershed polyline. The closed land-use polylines inside the watershed can then be used to determine the area of each land-use for the watershed. The Curve Numbers & Runoff command has an option to select closed polylines for determining the weighted average curve number from the polyline areas.

Prompts

Select closed polyline of watershed: *pick the polyline*

Select land-use closed polylines.

Select objects: *pick the polylines*

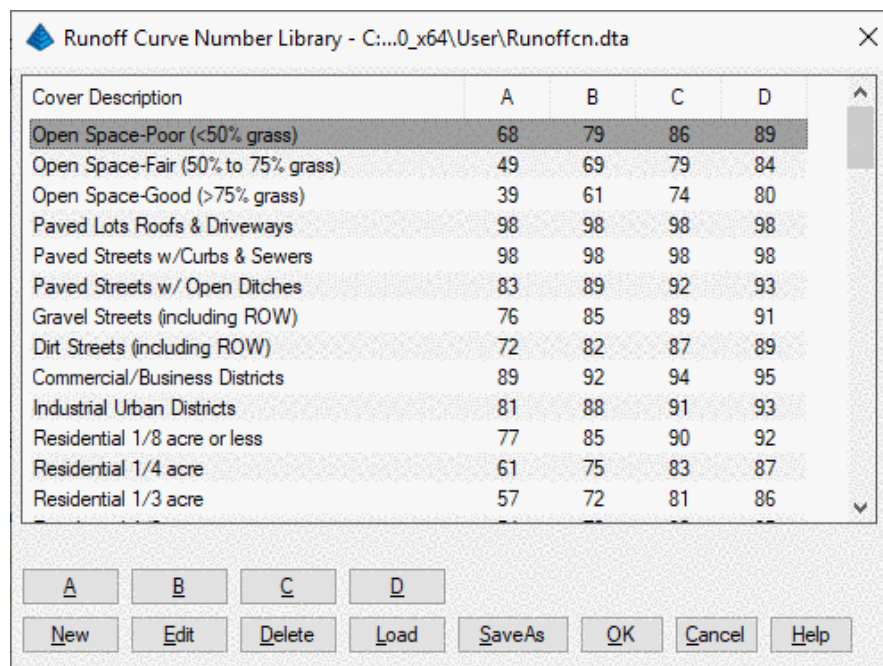
Pulldown Menu Location: Watershed

Keyboard Command: landarea

Prerequisite: Closed polylines for the watershed and land-use areas.

Curve Numbers & Runoff

This command calculates the weighted curve number (CN) as used by the SCS Method of runoff calculation. It will also calculate total, potential runoff from an area. The curve number is used by routines based on the TR-55 program. The weighted curve number is a weighted average of the curve numbers for each subarea of the watershed. The weights are based on the areas. The Description and Soil Type fields are used in the report. Shown here is the Curve Number Library from which to select curve numbers. The initial Curve Number Library is taken from the SCS TR-55, Urban Hydrology for Small Watersheds. You can modify the library using the New and Edit buttons and you can use the Load and SaveAs buttons to store the library to separate .rcn files.



Cover Description	A	B	C	D
Open Space-Poor (<50% grass)	68	79	86	89
Open Space-Fair (50% to 75% grass)	49	69	79	84
Open Space-Good (>75% grass)	39	61	74	80
Paved Lots Roofs & Driveways	98	98	98	98
Paved Streets w/Curbs & Sewers	98	98	98	98
Paved Streets w/ Open Ditches	83	89	92	93
Gravel Streets (including ROW)	76	85	89	91
Dirt Streets (including ROW)	72	82	87	89
Commercial/Business Districts	89	92	94	95
Industrial Urban Districts	81	88	91	93
Residential 1/8 acre or less	77	85	90	92
Residential 1/4 acre	61	75	83	87
Residential 1/3 acre	57	72	81	86

First highlight a row on the spreadsheet, and then select the curve number from the library using the Select CN button and click on the Select Subarea button to select all the subarea closed polylines. These polylines can be generated by the Sub-Watershed by Land Use command. The program will sum the polylines that are selected for a total area. The Subarea By Interior Point button allows you to select subarea closed polylines by picking inside the polylines. If you have the runoff layers defined for the watershed beforehand, you can simplify the process by selecting the closed polyline of the whole watershed, the subareas and their curve numbers, soil types, cover descriptions as well as area values will be filled into the spreadsheet automatically. More information regarding the soil types is given below.

When all the land-use curve numbers and areas are entered, enter the rainfall for the storm in question and then click on the Calc button to calculate the weighted curve number and the runoff given the weighted curve number. This

curve number can then be used in the Time of Concentration and Peak Flow commands. The Runoff Volume equals the Runoff Q times the total area. You can also save the table entries to a curve number (.cn) file and reload these values later.

Curve Number - Runoff Calculation

Area Units: Sq. Feet Sq. Mile Acres Area Precision: 0.000

	CN	Soil Group	Description	Area	% Area
1	98		Paved Lots Roofs & Driveways	202074.840	24.53
2	98		Paved Streets w/Curbs & Sewers	69826.680	8.48
3	69		Open Space-Fair (50% to 75% grass)	551774.520	66.99
4					

Select CN Select Subarea Subarea By Interior Pt Select Watershed Remove Row

Rainfall: 0.00 in

Calculated Curve Number: 78.6 Total Area: 823676.040

Calculated Runoff: 0.000 in Volume: 0.000 acre-ft

Export CSV Load CN Save CN Report Clear Exit Help

A typical Report is shown here:

Carlson Edit: C:\Users\shanlu\AppData\Roaming\Carlson Software\Carlson2021\ICAD10_X64\USER\scadrprt...

File Edit Settings

Runoff Curve Number and Runoff Tue May 12 23:42:18 2020

Project: West End Estate By: RBC Date: 05/12/20
 Location: Omaha, NB Checked: Date:
 Present

1. Runoff Curve Number (CN)

Cover description	CN	Soil Group	Area(Sq.Ft.)
Paved Lots Roofs & Driveways	98		202074.840
Paved Streets w/Curbs & Sewers	98		69826.680
Open Space-Fair (50% to 75% grass)	69		551774.520

CN (weighted): 78.6
 Total Area: 823676.040 Sq.Ft.

2. Runoff

Return Period: 50 YEAR
 Rainfall, P: 2.00 in
 Runoff, Q: 0.5060 in
 Runoff Volume: 0.7973 Acre-Ft

The following information regarding soil types comes from Part 630 of the National Engineering Handbook. This can be found at the following link: <https://directives.sc.egov.usda.gov/OpenNonWebContent.aspx?content=17757.wba>

The four hydrologic soil groups (HSGs) are described as:

Group A-Soils in this group have low runoff potential when thoroughly wet. Water is transmitted freely through the soil. Group A soils typically have less than 10 percent clay and more than 90 percent sand or gravel and have gravel or sand textures. Some soils having loamy sand, sandy loam, loam or silt loam textures may be placed in this group if they are well aggregated, of low bulk density, or contain greater than 35 percent rock fragments. The limits on the diagnostic physical characteristics of group A are as follows. The saturated hydraulic conductivity of all soil layers exceeds 40.0 micrometers per second (5.67 inches per hour). The depth to any water impermeable layer is greater than 50 centimeters [20 inches]. The depth to the water table is greater than 60 centimeters [24 inches]. Soils that are deeper than 100 centimeters [40 inches] to a water impermeable layer are in group A if the saturated hydraulic conductivity of all soil layers within 100 centimeters [40 inches] of the surface exceeds 10 micrometers per second (1.42 inches per hour).

Group B-Soils in this group have moderately low runoff potential when thoroughly wet. Water transmission through the soil is unimpeded. Group B soils typically have between 10 percent and 20 percent clay and 50 percent to 90 percent sand and have loamy sand or sandy loam textures. Some soils having loam, silt loam, silt, or sandy clay loam textures may be placed in this group if they are well aggregated, of low bulk density, or contain greater than 35 percent rock fragments. The limits on the diagnostic physical characteristics of group B are as follows. The saturated hydraulic conductivity in the least transmissive layer between the surface and 50 centimeters [20 inches] ranges from 10.0 micrometers per second (1.42 inches per hour) to 40.0 micrometers per second (5.67 inches per hour). The depth to any water impermeable layer is greater than 50 centimeters [20 inches]. The depth to the water table is greater than 60 centimeters [24 inches]. Soils that are deeper than 100 centimeters [40 inches] to a water impermeable layer or water table are in group B if the saturated hydraulic conductivity of all soil layers within 100 centimeters [40 inches] of the surface exceeds 4.0 micrometers per second (0.57 inches per hour) but is less than 10.0 micrometers per second (1.42 inches per hour).

Group C-Soils in this group have moderately high runoff potential when thoroughly wet. Water transmission through the soil is somewhat restricted. Group C soils typically have between 20 percent and 40 percent clay and less than 50 percent sand and have loam, silt loam, sandy clay loam, clay loam, and silty clay loam textures. Some soils having clay, silty clay, or sandy clay textures may be placed in this group if they are well aggregated, of low bulk density, or contain greater than 35 percent rock fragments. The limits on the diagnostic physical characteristics of group C are as follows. The saturated hydraulic conductivity in the least transmissive layer between the surface and 50 centimeters [20 inches] is between 1.0 micrometers per second (0.14 inches per hour) and 10.0 micrometers per second (1.42 inches per hour). The depth to any water impermeable layer is greater than 50 centimeters [20 inches]. The depth to the water table is greater than 60 centimeters [24 inches]. Soils that are deeper than 100 centimeters [40 inches] to a restriction or water table are in group C if the saturated hydraulic conductivity of all soil layers within 100 centimeters [40 inches] of the surface exceeds 0.40 micrometers per second (0.06 inches per hour) but is less than 4.0 micrometers per second (0.57 inches per hour).

Group D-Soils in this group have high runoff potential when thoroughly wet. Water movement through the soil is restricted or very restricted. Group D soils typically have greater than 40 percent clay, less than 50 percent sand, and have clayey textures. In some areas, they also have high shrink-swell potential. All soils with a depth to a water impermeable layer less than 50 centimeters [20 inches] and all soils with a water table (210-VI-NEH, May 2007) 7-3 Part 630 National Engineering Handbook Chapter 7 Hydrologic Soil Groups within 60 centimeters [24 inches] of the surface are in this group, although some may have a dual classification, as described in the next section, if they can be adequately drained. The limits on the physical diagnostic characteristics of group D are as follows. For soils with a water impermeable layer at a depth between 50 centimeters and 100 centimeters [20 and 40 inches], the saturated hydraulic conductivity in the least transmissive soil layer is less than or equal to 1.0 micrometers per second (0.14 inches per hour). For soils that are deeper than 100 centimeters [40 inches] to a restriction or water table, the saturated hydraulic conductivity of all soil layers within 100 centimeters [40 inches] of the surface is less than or equal to 0.40 micrometers per second (0.06 inches per hour).

Dual hydrologic soil groups-Certain wet soils are placed in group D based solely on the presence of a water table within 60 centimeters [24 inches] of the surface even though the saturated hydraulic conductivity may be

favorable for water transmission. If these soils can be adequately drained, then they are assigned to dual hydrologic soil groups (A/D, B/D, and C/D) based on their saturated hydraulic conductivity and the water table depth when drained. The first letter applies to the drained condition and the second to the undrained condition. For the purpose of hydrologic soil group, adequately drained means that the seasonal high water table is kept at least 60 centimeters [24 inches] below the surface in a soil where it would be higher in a natural state.

Matrix of hydrologic soil group assignment criteria-The decision matrix in tables 7-1 and 7-2 can be used to determine a soil's hydrologic soil group. Check both tables before making a final decision. If saturated hydraulic conductivity data are available and deemed to be reliable, then these data, along with water table depth information, should be used to place the soil into the appropriate hydrologic soil group. If these data are not available, the hydrologic soil group is determined by observing the properties of the soil in the field. Factors such as texture, compaction (bulk density), strength of soil structure, clay mineralogy, and organic matter are considered in estimating the hydraulic conductivity of each layer in the soil profile. The depth and hydraulic conductivity of any water impermeable layer and the depth to any high water table are used to determine correct hydrologic soil group for the soil. The property that is most limiting to water movement generally determines the soil's hydrologic group. In anomalous situations, when adjustments to hydrologic soil group become necessary, they shall be made by the NRCS state soil scientist in consultation with the state conservation engineer.

Pulldown Menu Location: Watershed

Keyboard Command: curveno

Prerequisite: None

Calculate C-Factor

One of the most commonly used equations for the calculation of peak stormwater discharge from small watersheds is the Rational Formula. The C-Factor in this equation is a dimensionless runoff coefficient that represents the fraction of rainfall that becomes stormwater runoff. Although there are many factors that affect this ratio, the type of surface coverage is the one most often implemented. For watersheds with mixed surface coverage, a weighted C-Factor can be calculated using a weighted average method.

Average C-factor is calculated by dividing the product of Coverage Area and C-factor which is divided by the total watershed area.

C-Factor Calculation

Area Units: Sq. Feet Sq. Mile Acre

Area Precision: 0.00

	C-Factor	Description	Area	% Area
1	0.250	WOODS	144348.33	39.18
2	0.950	DRIVEWAY	11526.10	3.13
3	0.950	PAVEMENT	18400.07	4.99
4	0.950	HOUSE	20400.00	5.54
5	0.300	GRASS	173752.99	47.16
6				

Select C-Factor Select Subarea Subarea By Interior Pt Select Watershed Remove Row

Calculated C-Factor: 0.369 Total Area: 368427.49

Export CSV Report Clear Exit Help

C-Factor Calculation dialog

This tool can be used like a spreadsheet by manually selecting a C-Factor along with the type and total surface coverage. However, the C-Factor can also be calculated using closed polylines within your design drawing. To do so, draw the outside boundary of the total watershed and create closed polylines for each Sub Area. Please note that there will always be one area that is calculated as the total drainage area minus the sum of all the sub-areas. This is typically a pervious layer such as lawns. Be sure and have the overall watershed boundary and each sub-area boundary created. The following procedure describes this pMethod.

Within the C-factor Calculation window, set your units as you want them to be reported along with area precision. Please note that you do not have to use a pre-defined C-factor definition.

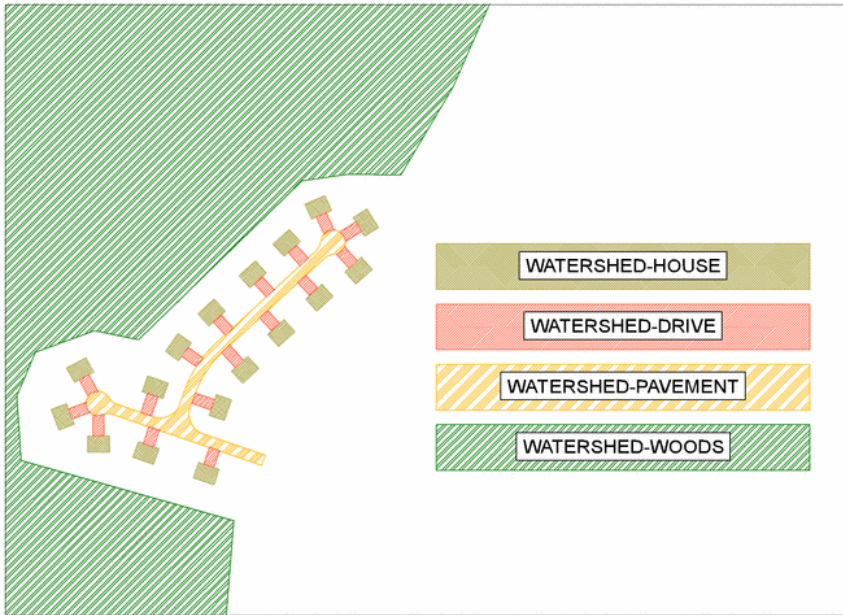
Select the first row, first cell of the dialogue and click the Select C-Factor command button.

Use the Select Subarea command to select a sub-area represented by a closed polyline within the overall watershed boundary.

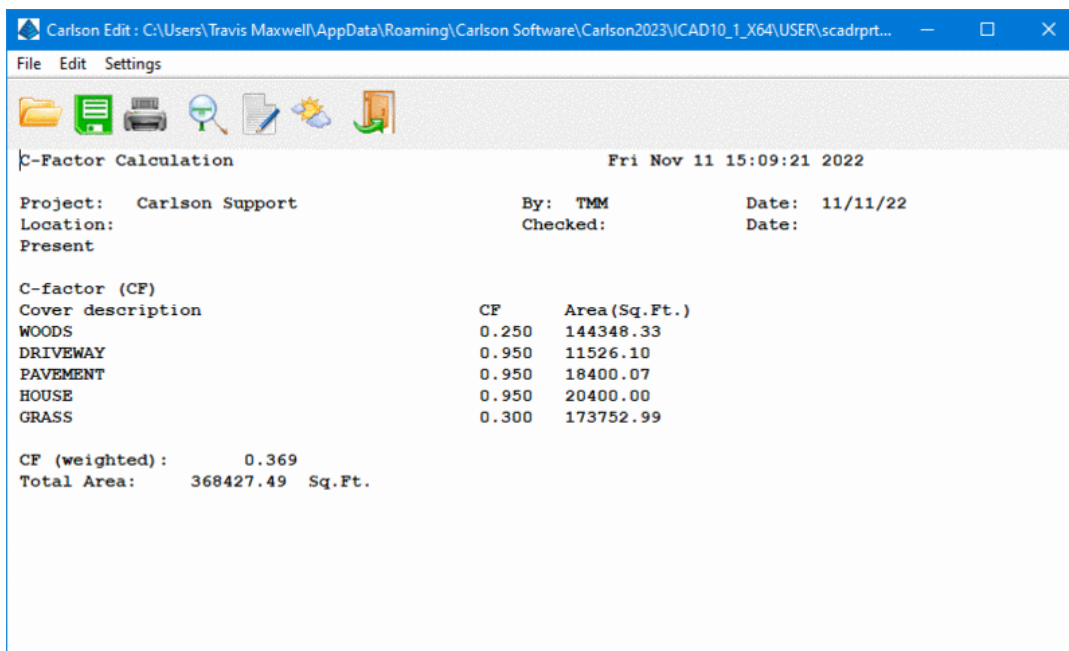
After all subareas have been selected for each C-Factor, use the Subarea By Interior Pt command to calculate the remaining area to complete the C-Factor calculation.

Select the Report Button to report the calculation.

If a Watershed Layer Definition file has been setup and the design drawing has sub-areas delineated by layer accordingly, C-Factor will be calculated automatically when the overall watershed is selected. To make sure each sub-areas are assigned correctly, use the Watershed Layer Definition tool to hatch them as shown below.



C-Factor Report



Pulldown Menu Location: Watershed > Calculate C-Factor

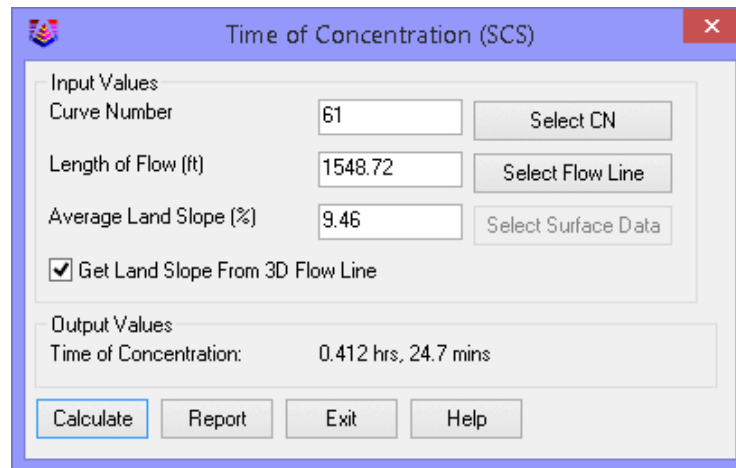
Keyboard Command: calc_cfactor

Prerequisite: None

Time of Concentration - SCS Method

This command calculates the time of concentration (T_c) by the SCS method from *A Method of Estimating Volume and Rate of Runoff in Small Watersheds*. The T_c is the time required for water to flow from the most distant point in the watershed to the measurement point.

The SCS method calculation based on the curve number, length of flow, and average slope. The curve number defaults to the weighted curve number from the Curve Numbers & Runoff routine. When the three inputs are entered, click on Calculate to compute the Tc. Select CN from Table button opens the CN dialog and allows you to select the CN based on the drainage area type and the soil type. Click on Select Flow Line from Screen button to use a 3D polyline in the drawing. This sets the length of flow and average land slope. A 3D polyline that models the flow can be created with the Watershed Above Point or Run Off Tracking commands. While reading in the 3D polyline, the Tc is calculated by adding the Tc's for each segment of the polyline. This yields a different and more accurate Tc than using the average slope with the Calculate button.



Time of Concentration Dialog

Time of Concentration (SCS)		Tue Jun 07 09:25:17 2016	
Project:	Hampton Creet	By:	M Long
Location:	North Carolina	Checked:	Date: 06/07/16
Present			Date: 06/07/16
Curve Number	: 61		
Length of Flow	: 1548.72 ft		
Average Land Slope	: 9.46 %		
Time of Concentration	: 0.412 hrs, 24.7 mins		

Pulldown Menu Location: Watershed -> Time of Concentration > SCS Method

Keyboard Command: flowtc2

Prerequisite: None

Time of Concentration - TR-55 Method

This command calculates the time of concentration (Tc) by the TR-55 method. Time of concentration is the time required for water to flow from the most distant point in the watershed to the reference point.

The TR-55 method for calculating Time of Concentration(Tc) is a velocity method for determining Tc. Stormwater runoff is assumed to begin as Sheet Flow. Once the depth of Sheet Flow increases, stormwater runoff concentrates into shallow concentrated flow. Channel Flow occurs when multiple channel flows join together and travel along a channel. In the case of un-urbanized existing conditions, Sheet Flow is assumed to be at least 100'. For developed conditions, there is no limitation on the length of sheet flow. Total Time of Concentration is equal to the sum of each flow segment. If grade or surface changes occur along a flow segment, that flow segment can be sub-divided where slope changes. Use the Add button to add flows. Sheet, Shallow and Channel flows can be added individually or all together.

When adding a flow, the Select Flow Line From Screen function allow you to pick a 3D polyline and split it into segments of sheet, shallow and channel flow precisely. The Manning's n for the sheet and channel flow can be chosen from a table by clicking the Select button. Clear button will clear all fields in the dialog for entering new data.

Tc (min.)	Length (ft)	Slope (%)	Method
2.911	100.00	1.330	Combo/Sheet Flow
3.995	100.00	2.750	Combo/Shallow Flow
0.926	500.00	3.680	Combo/Channel Flow

Total Tc: 0.131 hr, 7.8 min

Buttons: Add, Edit, Delete, Report, PDF Report, Load, Save, SaveAs, Exit, Help

Type of Tc Method

- Sheet Flow
- Shallow Flow
- Channel Flow
- Combo Flow (Sheet + Shallow + Channel)

Buttons: OK, Cancel, Help

Select Flow Line From Screen

Sheet Flow | Shallow Concentrated Flow | Channel Flow

Segment ID: A

Manning's n: 0.020 [Select]

Two-yr 24-hr Rainfall: 2.000 in

Flow Length (100 ft max): 100.000 ft

Land Slope: 1.330 %

Time of Concentration: 0.049 hr, 2.911 min

Split Shallow and Channel Flow by Slope Zone

Slope Tolerance (%): 0.000

Minimum Distance (ft): 10.00

Total Time of: 0.131 hrs, 7.8 mins

Buttons: Calculate, Report, PDF Report, Clear, OK, Cancel, Help

Time of Concentration (TR-55)

Select Flow Line From Screen

Sheet Flow | **Shallow Concentrated Flow** | Channel Flow

Segment ID: shallow flow

Velocity Factor: 2.516 ft/s Select

Flow Length: 100.000 ft

Land Slope: 2.750 %

Time of Concentration: 0.067 hr
3.995 min

Split Shallow and Channel Flow by Slope Zone

Slope Tolerance (%): 0.000

Minimum Distance (ft): 10.00

Total Time of: 0.131 hrs, 7.8 mins

Calculate Report PDF Report

Clear OK Cancel Help

Time of Concentration (TR-55)

Select Flow Line From Screen

Sheet Flow | Shallow Concentrated Flow | **Channel Flow**

Segment ID: Channel Flow

Manning's n: 0.020 Select

Cross Section Flow Area: 10.000 s.f

Wetted Perimeter: 20.000 ft

Flow Length: 500.000 ft

Land Slope: 3.680 %

Time of Concentration: 0.015 hr
0.926 min

Split Shallow and Channel Flow by Slope Zone

Slope Tolerance (%): 0.000

Minimum Distance (ft): 10.00

Total Time of: 0.131 hrs, 7.8 mins

Calculate Report PDF Report

Clear OK Cancel Help

TR-55 Method Report

Project: Hampton Creek
 Location: NC
 Present

By: M Long
 Checked:
 Date: 06/08/16
 Date:

Sheet Flow	Segment ID: Sheet Flow
Manning's Roughness Coeff. (n):	0.020
Flow Length, L (total L <= 100 ft):	100.00 ft
Two-yr 24-hr Rainfall, P:	2.00 in
Land Slope, s:	1.33 %
Tc:	0.049 hr (2.9 min)
Shallow Concentrated Flow	Segment ID: Shallow Flow
Land Cover / Flow Regime	Forest with heavy ground litter and hay meadows
Shallow Flow Velocity Factor, Kv:	2.516
Flow Length, L:	2.50 ft
Watercourse Slope, s:	2.75 %
Average Velocity, V:	0.42 ft/s
Tc:	0.002 hr (0.1 min)
Channel Flow	Segment ID: Channel Flow
Cross Sectional Flow Area, a:	10.00 SqFt
Wetted Perimeter, Pw:	20.00 ft
Hydraulic Radius, r:	0.50 ft
Channel Slope, s:	3.68 %
Manning's Roughness Coeff. (n):	0.020
Velocity, V:	9.00 ft/s
Flow Length, L:	500.00 ft
Tc:	0.015 hr (0.9 min)
Total Tc:	0.066 hr (3.9 min)

Pulldown Menu Location: Watershed -> Time of Concentration > TR-55 Method

Keyboard Command: flowtc

Prerequisite: None

Time of Concentration - Rational Method

This command calculates the time of concentration (Tc) by the Rational method. The Tc is the time required for water to flow from the most distant point in the watershed to the measurement point.

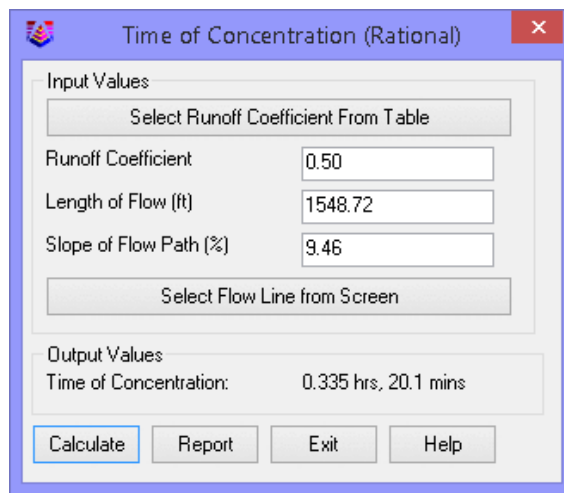
The Rational method calculation based on the runoff coefficient, length of flow and average slope. These values are set in the dialog shown. The formula is:

$$Tc = (1.8(1.1 - c)L^{1/2}) / S^{0.33}$$

Where: c = Runoff coefficient

L = flow length (ft or m)

S = average land slope (%)



Time of Concentration Dialog

```

Time of Concentration (Rational)                               Thu Jun 09 09:06:27 2016

Project: Hampton Creek                                         By: M Long           Date: 06/09/16
Location: Winston Salem, NC                                  Checked:             Date: 06/09/16
Present

Runoff Coefficient      : 0.50
Length of Flow          : 1548.72 ft
Slope of Flow Patch     : 9.46 %

Time of Concentration   : 0.335 hours, 20.1 minutes

```

Pulldown Menu Location: Watershed -> Time of Concentration > Rational Method

Keyboard Command: flowtc3

Prerequisite: None

Time of Concentration - Kinematic Wave Method

Kinematic Wave Equation is often used to calculate the time of concentration for overland flow. This equation has the rainfall intensity as a dependent variable, which is a function of the time of concentration. Therefore it's an iterative process to determine the time of concentration.

Where,

tc = time of concentration (minutes)

i = rainfall intensity (in/hr)

L = overland flow length (ft)

n = manning's roughness coefficient

S = slope of the surface (ft/ft or m/m)

Time of Concentration (Kinematic Wave)

Wed Jun 08 15:32:24 2016

Project: Hampton Creek
Location: North Carolina
Present

By: M Long
Checked:

Date: 06/08/16
Date: 06/08/16

Surface Manning's n : 0.4000
Length of Flow : 1548.72 ft
Average Land Slope : 9.46 %
Rainfall ID : NCDENR - Greensboro Area
Rainfall Return Period : 100 YEAR

Time of Concentration : 0.984 hrs, 59.0 mins

Pulldown Menu Location: Watershed -> Time of Concentration > Kinematic Wave Method

Keyboard Command: flowtc5

Prerequisite: None

Time of Concentration - Regression Equation, LA County

The regression method uses an iterative process to calculate the time of concentration. The study of regression equation relied on Tc computations from a large number of subareas. The subareas were taken from diverse hydrology studies that used the kinematic wave theory equations to calculate Tc. The following equations are from Hydrology Manual of Los Angeles County Department of Public Works.

$$T_c = \frac{0.31 * L^{0.483}}{(C_d * I_t)^{0.519} * S^{0.135}}$$

$$I_t = I_{1440} * \left(\frac{1440}{t}\right)^{0.47}$$

$$C_d = (0.9 * IMP) + (1.0 - IMP) * C_u$$

where: T_c = Time of concentration, in minutes

L = Longest flow path length from watershed boundary to outlet, in feet

C_d = Developed runoff coefficient, ratio of runoff rate to rainfall intensity, in in/in

I_t = Intensity at time t , in in/hr

S = Slope of longest flow path, in ft/ft

IMP = Percent Impervious, percent expressed as 0.0 to 1.0

C_u = Undeveloped runoff coefficient, ratio of runoff rate to rainfall intensity, in in/in

Select Regression Equation Method from Watershed > Time of Concentration menu. In the dialog, specify the Soil Type, Proportion Impervious Data, Flow length and Slope, and the Rational rainfall data. The flow path data can be obtained from picking a 3D polyline and the program calculates the length and slope of the flow automatically. Click on Calculate button to see the result.

Time of Concentration (Regression Equation - LA County)

Input Values

Soil Type No. 2, ALTAMONT CLAY LOAM Library

Impervious (%) 55 Select

Length of Flow (ft) 1000.00 Select Flow Line

Average Land Slope (%) 1.2 Select Surface Data

Get Land Slope From 3D Flow Line

Rational Rainfall ID Alexandria 4 W Library

Return Period 25 Year

Output Values

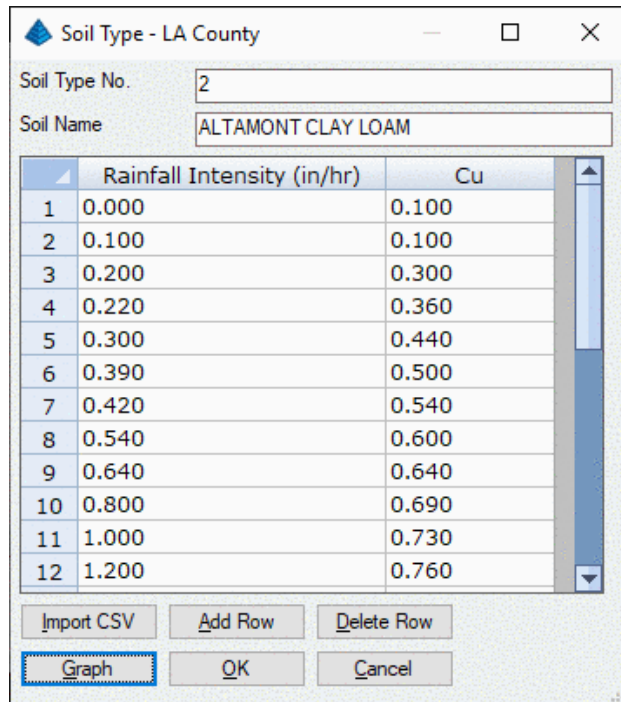
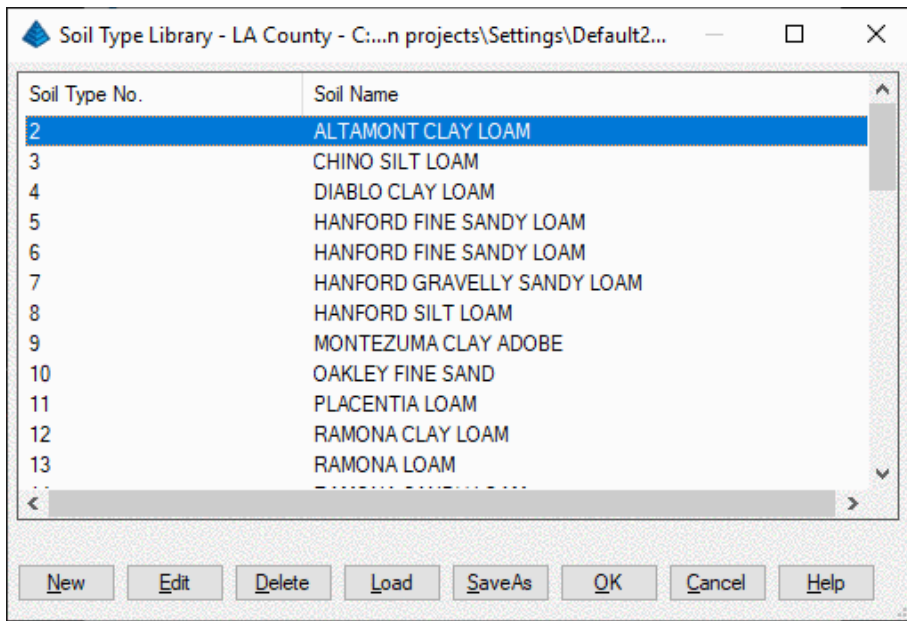
Undeveloped Runoff Coefficient (C_u): 0.821

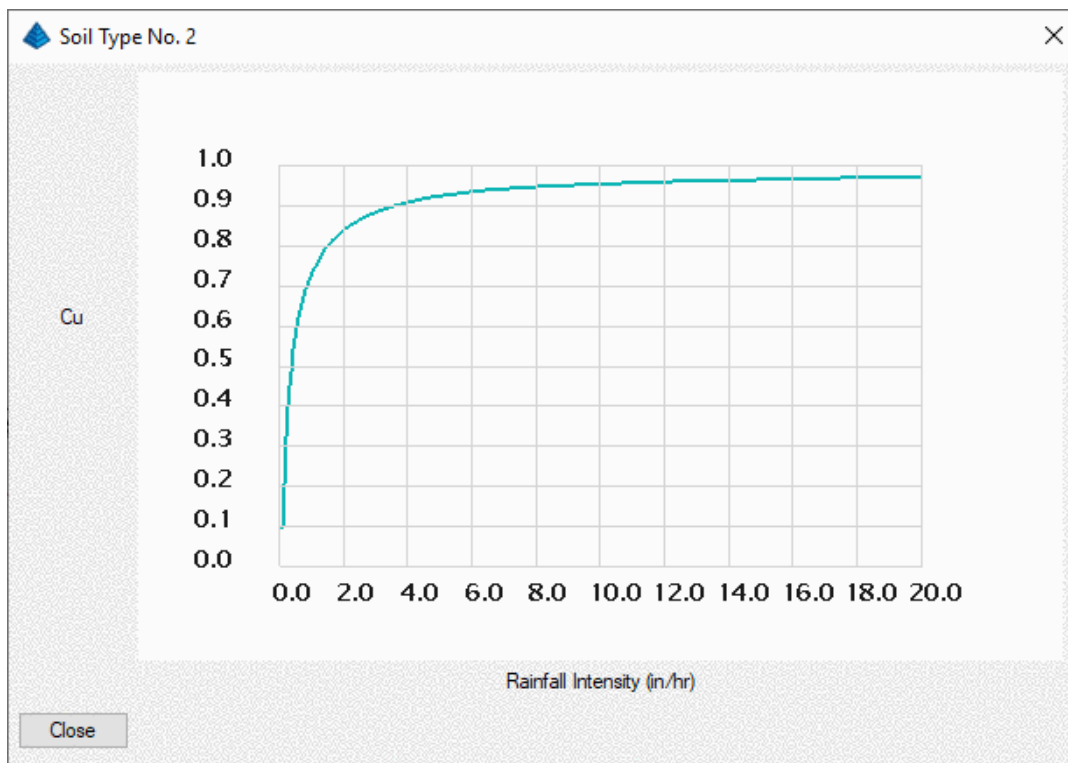
Developed Runoff Coefficient (C_d): 0.864

Time of Concentration: 0.213 hrs, 12.8 mins

Calculate Report Exit Help

The Soil Type library has all the tables from the Appendix C in the Hydrology Manual. New Soil Type and Runoff Coefficient Data can be added to the library. The Import CSV function can easily import existing spreadsheet to the library. Graph dialog shows the C_u /Intensity curve.

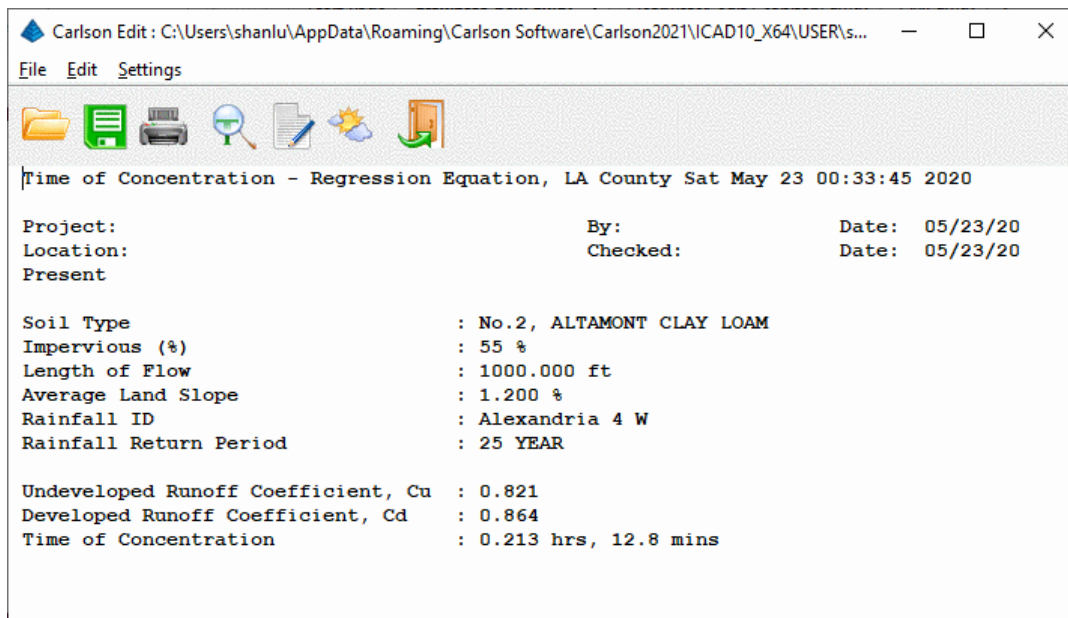




The Proportion Impervious Data library is from the Appendix D in the Hydrology Manual. New data is allowed to add to the library.

LA County Code	Land Use Description	% Impervious
1111	High-Density Single Family Residential	42
1112	Low-Density Single Family Residential	21
1121	Mixed Multi-Family Residential	74
1122	Duplexes, Triplexes and 2-or 3-Unit Condominiums and Townhouses	55
1123	Low-Rise Apartments, Condominiums, and Townhouses	86
1124	Medium-Rise Apartments and Condominiums	86
1125	High-Rise Apartments and Condominiums	90
1131	Trailer Parks and Mobile Home Courts, High-Density	91
1132	Mobile Home Courts and Subdivisions, Low-Density	42
1140	Mixed Residential	59
1151	Rural Residential, High-Density	15
1152	Rural Residential, Low-Density	10
1211	Low- and Medium-Rise Major Office Use	91
1212	High-Rise Major Office Use	91
1213	Skyscrapers	91
1221	Regional Shopping Center	95

The Tc report is in Carlson report format.



Pulldown Menu Location: Watershed -> Time of Concentration > **Regression Equation (LA County)**

Keyboard Command: flowtc7

Prerequisite: None

Time of Concentration - Maricopa County, AZ

This command calculates the time of concentration (Tc) by the following equation developed by Papadakis and Kazan in Maricopa County, AZ. This is due to the extent of imperviousness associated with urban watersheds and to the fact that the time of concentration is usually very short.

$$T_c = 11.4L^{0.5}K_b^{0.52}S^{-0.31}i^{-0.38}$$

where: Tc = time of concentration, in hours

L = length of the longest flow path, in miles

S = watercourse slope, in feet/mile

Kb = watershed resistance coefficient, see table taken from design manual

i = rainfall intensity, in inches/hour

Table 3.1
EQUATION FOR ESTIMATING K_B IN THE T_C EQUATION

$K_B = m \log_{10} A + b$				
Where A is drainage area, in acres				
Type	Description	Typical Applications	Equation Parameters	
			m	b
A	Minimal roughness: Land surfaces that are relatively smooth and/or well graded. Surface runoff is sheet flow.	Commercial/industrial areas Residential areas Parks and golf courses	-0.00625	0.04
B	Moderately low roughness: Land surfaces have irregularly spaced roughness elements that protrude from the surface but are still relatively uniform. Surface runoff is predominately sheet flow around the roughness elements.	Agricultural fields Pastures Desert rangelands Undeveloped urban lands	-0.01375	0.08
C	Moderately high roughness: Land surfaces that have significant large to medium-sized roughness elements and/or poorly graded land surfaces that cause the flow to be diverted around the roughness elements. Surface runoff is sheet flow for short distances draining into meandering drainage paths.	Hillslopes Brushy alluvial fans Hilly rangelands Disturbed lands, mining, etc. Forests with underbrush	-0.025	0.15
D	Maximum roughness: Rough land surfaces with torturous flow paths. Surface runoff is concentrated in numerous short flow paths that are often oblique to the main flow direction.	Mountains Some wetlands	-0.030	0.20

Note: A is the area of the entire subbasin, not the area of the surface type A, B, C or D within the subbasin.

Select Maricopa County Method from Watershed > Time of Concentration menu. In the dialog, specify the unit of area, flow length and slope, and the basin surface type. The flow path data can be obtained from picking a 3D polyline and the program calculates the length and slope of the flow automatically. Click on Calculate button to see the result.

Time of Concentration (Maricopa County - AZ)

Input Values

Area Units: Acre

Length Units: Mile

Slope Units: %

Surface Type: Type A: minimal roughness, residential

Drainage Area: 9.96 [Select]

Length of Flow: 0.4837

Slope of Flow Path: 0.10

[Select Flow Line from Screen]

Rational Rainfall ID: Rain1 [Library]

Return Period: 25 Year

Output Values

Time of Concentration: 0.543 hrs, 32.6 mins

[Calculate] [Report] [Exit] [Help]

Pulldown Menu Location: Watershed -> Time of Concentration > Maricopa County Method

Keyboard Command: flowtc6

Prerequisite: None

Time of Concentration - Kirpich Method

This command calculates the time of concentration (T_c) by the Kirpich method, which is developed for small drainage basins in Tennessee and Pennsylvania, with basin areas from 1 to 112 acres. The formula is:

$$T_c = KL^{0.77} / S^n$$

where: L = basin length (ft)

S = basin slope (ft/ft)

K = 0.0078 for Tennessee, 0.0013 for Pennsylvania

n = 0.385 for Tennessee, 0.5 for Pennsylvania

The estimated T_c is multiplied by 0.4 if the overland flow path is concrete or asphalt, or by 0.2 if the channel is concrete-lined.

Select Kirpich Method from Watershed > Time of Concentration menu. In the dialog, choose a location of either Pennsylvania or Tennessee, and specify the basin conditions. The flow path data can be obtained from picking a 3D polyline and the program calculates the length and slope of the flow automatically. Click on Calculate button to see the result.

Time of Concentration (Kirpich)

Input Values

Location: Pennsylvania Tennessee

Special Basin Conditions

None

Concrete/Asphalt Overland

Concrete-lined Channel

Length of Flow Path: 1548.721 ft

Average Basin Slope: 9.461 %

Select Flow Line From Screen

Output Values

Time of Concentration: 0.020 hrs, 1.2 mins

Calculate Report Exit Help

Project: Hampton Creek
 Location: Winston Salem, NC
 Present

By: M Long Date: 06/09/16
 Checked: Date: 06/09/16

Location: Pennsylvania
 Basin Condition: Regular
 Length of Flow Path: 1548.721 ft
 Average Basin Slope: 9.461 %

Time of Concentration: 0.020 hrs, 1.209 mins

Peak Flow Dialog

Pulldown Menu Location: Watershed -> Time of Concentration > Kirpich Method

Keyboard Command: flowtc4

Prerequisite: None

Time of Concentration - Kerby-Kirpich Method

The Kerby-Kirpich method offers an alternative means of calculating the time of concentration for a watershed. With this method, the overall time of concentration is divided into two components: overland flow and channel flow. The Kerby equation is used to calculate overland flow, and the Kirpich equation is used to calculate channel flow.

Kerby equation (overland flow)

$T_{ov} = K (LN)^{0.467} / S^{-0.235}$ where: T_{ov} = Overland Flow Time of Concentration (min)

K = Unit Conversion Coefficient, in which K = 0.828 for traditional units and K = 1.44 for SI units

L = Overland Flow Length (in feet or meters as dictated by K) (ft/ft)

N = Retardance Coefficient - The Select button will show Common Kerby Coefficient

S = Slope of Terrain conveying the overland flow (ft/ft)

Kirpich equation (channel flow)

$T_{ch} = KL^{0.0770} S^{-0.385}$ where: T_{ch} = Channel Flow Time of Concentration (min)

K = Unit Conversion Coefficient, in which K = 0.0078 for traditional units and K = 0.0195 for SI units

L = Channel Flow Length (in feet or meters as dictated by K) (ft/ft)

S = Slope of Terrain conveying the main channel flow (ft/ft)

Select Kerby-Kirpich Method from Watershed > Time of Concentration menu.

In the dialog for the Overland Flow, input the Kerby - Overland Flow Length (can be user entered), or Select Flow Line. The flow data can be obtained from picking a 3D polyline and the program calculates the length and slope of the flow automatically. User can also Select Surface Data and choose a surface model to get the flow slope. Enter or Select Retardance Coefficient. For the Channel Flow, input the Kirpich - Channel Flow Length (can be user entered), or Select Flow Line. User can also Select Surface Data and choose a surface model to get the flow slope. Calculate button to see the result.

Time of Concentration - Kerby-Kirpich Method

Kerby - Overland Flow

Flow Length: 300 ft [Select Flow Line]

Flow Slope: 0.031 ft/ft [Select Surface Data]

Get Land Slope From 3D Flow Line

Retardance Coefficient: 0.020 [Select]

Time of Concentration: 0.072 hrs, 4.3 mins

Kirpich - Channel Flow

Flow Length: 1200.000 ft [Select Flow Line]

Flow Slope: 0.045 ft/ft [Select Surface Data]

Get Land Slope From 3D Flow Line

Time of Concentration: 0.101 hrs, 6.0 mins

Total Tc: 0.173 hrs, 10.4 mins

[Calculate] [Exit]

Pulldown Menu Location: Watershed -> Time of Concentration > Kerby - Kirpich Method

Keyboard Command: flowtc8

Prerequisite: None

Peak Flow - Graphical Method

This command calculates peak flow using the graphical method from the TR-55 program. The program is run through the dialog shown below. The inputs in the top section default to the values from the Curve Numbers & Runoff and Time of Concentration routines. When all the inputs are entered, click on the Calculate button to obtain the peak flow at the bottom line. The peak flow value can then be used for Detention Pond Sizing or Channel Design.

```

Graphical Peak Discharge
Project: Parking          By: TW
Date: 11/13/95
Location: West           Checked:      Date:
Developed
1. Data:
   Drainage area:.....A = 27.1500 Acres
   Runoff Curve Number:.....CN = 70
   Time of Concentration:.....Tc = 0.75
2. Frequency.....yr = 100
3. Rainfall,P(24-hour).....in = 6.00
4. Initial abstraction, Ia..... = 0.8571
5. Compute Ia/P..... = 0.1429
6. Unit peak discharge, qu.....csm/in = 410.22
7. Runoff,Q.....in = 2.8052
8. Pond & swap adjustment factor,...Fp = 1.00
9. Peak Discharge,qp.....cfs = 48.8172

```

More information regarding the various storm types can be found at the below links:
<https://www.lmnoeng.com/RainfallMaps/RainfallMaps.htm>
<https://www.wcc.nrcs.usda.gov/ftpref/wntsc/H&H/TRsTPs/TP149.pdf>

The below map, provided by the National Resource Conservation Service, marks where the various storm types occur in the USA. This map can also be found at the below link.
https://www.nrcs.usda.gov/Internet/FSE_DOCUMENTS/stelprdb1044171.pdf

Fig. B-2. Approximate geographic boundaries for SCS rainfall distributions (SCS, 1986):

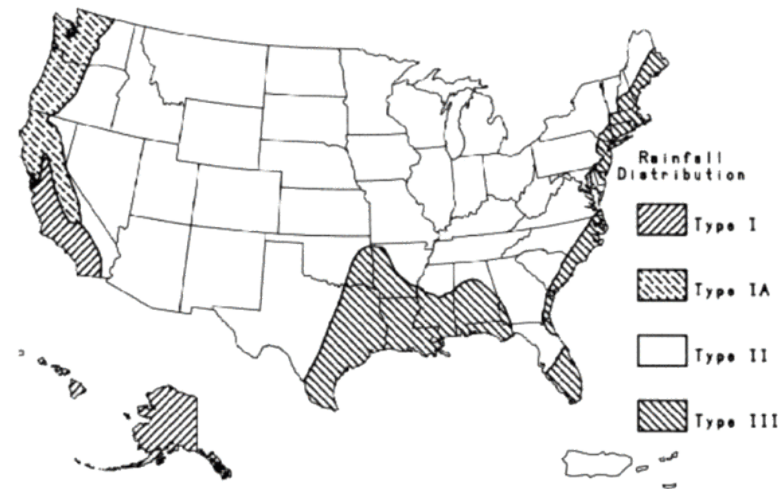


Figure B-2--Approximate geographic boundaries for SCS rainfall distributions.

Pulldown Menu Location: Watershed
Keyboard Command: peakflow
Prerequisite: None

Peak Flow - Rational Method (General)

This command calculates peak flow using the Rational Method, $Q=CIA$. The inputs needed are the size of drainage area, C-Factor, Time of Concentration, Return Period and a Rainfall file which contains the Point Precipitation Intensity table from the Rainfall Library. The peak flow value can then be used for Detention Pond Sizing or Channel Design.

Peak Flow Rational Method Report:

Pulldown Menu Location: Watershed
Keyboard Command: peakflw3
Prerequisite: None

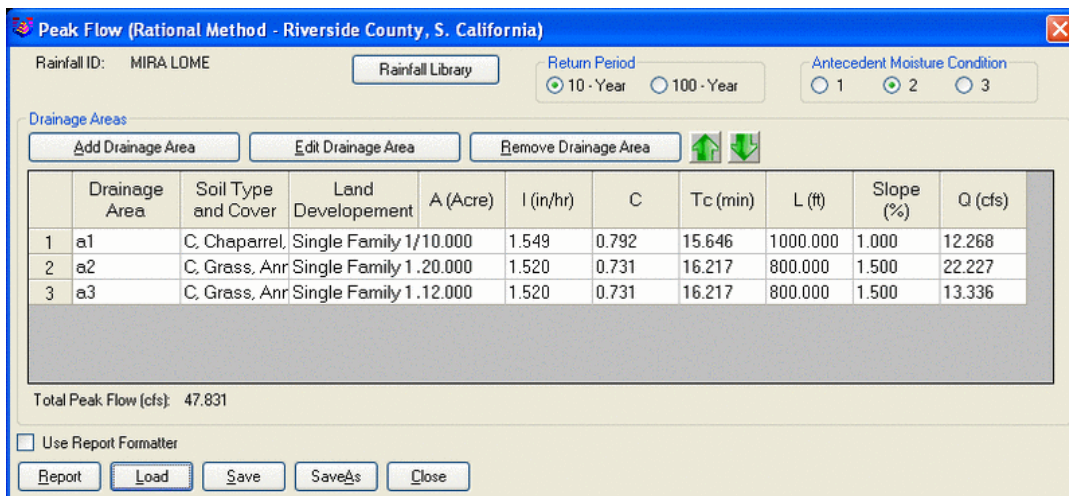
Peak Flow - Rational Method(Riverside S. California)

This command is design to compute peak flow in the Riverside County, southern California. In Riverside county as well as Southern California, peak flow is calculated based on the Rational method, $Q = CIA$, but it also has its own computing criteria because of its extremely varied topography, soil types and land uses, and its storm events that are different than other regions.

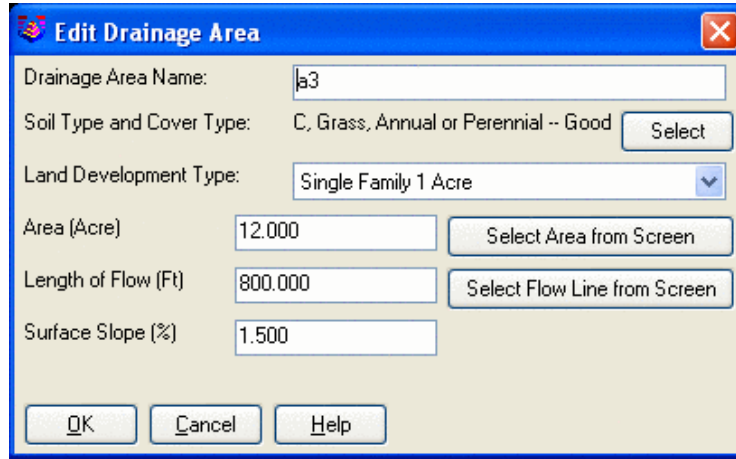
First, Riverside County usually requires a peak flow calculation on 10-year and 100 year storm events. So rainfall data of riverside county should be defined in the rainfall library. We provide two methods, Rainfall Accumulation and Rainfall Intensity, which allows any combination of return periods of 2-year, 5-year, 10-year, 25-year, 50-year and 100-year rainfall definition. IDF curves would be calculated. Second, Riverside County defines a list of land development types, which affect the subarea runoff index and percentage of the impervious cover, and therefore lead to the calculation of the time of concentration, rainfall intensity, runoff coefficient and then peak flow. The types of land development are: Commercial, Apartment, Mobil home park, Condominium, Single family(1/4 Acre lot, 1/2 Acre lot, 1 Acre lot), Undeveloped with poor, fair or good cover. Please refer to the Riverside County Hydrology Manual for details.

Select Rational Method (Riverside County) from Watershed > Peak Flow menu. In the peak flow dialog, choose a rainfall from the rainfall library, and specify the return period and the antecedent moisture condition. The spreadsheet displays the list of drainage area data in design. Add Drainage Area button allows you to add a new drainage entry, Edit Drainage Area button allows you to edit the highlighted entry and Remove Drainage Area button removes the highlighted entry. When you add or edit a drainage area, the Add/Edit Drainage Area dialog opens. Enter the drainage area name, specify the soil type and cover type from a look up table, and choose the land development type. The area can be entered either manually or automatically by selecting a closed polyline from screen. The flow line data can also be obtained from picking a 3D polyline and the program calculates the length and slope of the flow automatically. Click on OK button to commit the drainage area entry.

The design data can be saved to a .riv file, and loaded back to the dialog. Report button reports the design data and hydrology calculation results. The Use Report Formatter option allows you to report data to a spreadsheet instead.



Peak Flow Dialog



Add/Edit Drainage Area

Prompts

Peak flow dialog: Fill in values

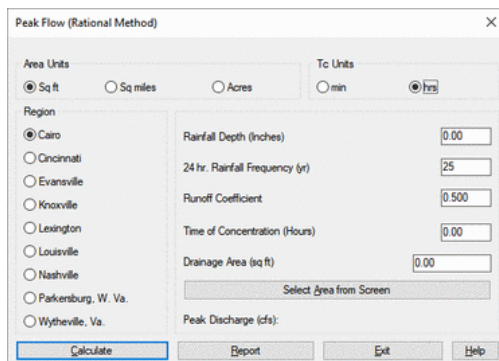
Pulldown Menu Location: Watershed -> Peak Flow > Rational Method (Riverside County, S. California)

Keyboard Command: peakflw5

Prerequisite:

Peak Flow - Rational Method (KYDOT)

This command calculates peak flow using the rational method, $Q=CIA$, with rainfall intensity coefficients specific to regions of Kentucky. The program is run through the dialog shown below. The weighted Runoff Coefficient or C-factor can be calculated by the Curve Number & Runoff routine. The peak flow value can then be used for Detention Pond Sizing or Channel Design.



Pulldown Menu Location: Watershed

Keyboard Command: peakflw2

Prerequisite: None

Peak Flow - Tabular Hydrograph Method

This command calculates peak flow using the tabular hydrograph method from the TR-55 program. The program is run through the dialog shown below. The Curve Numbers & Runoff and Time of Concentration routines can be used to calculate the subarea input values. When all the inputs are entered, click on the Calculate button. The input values can be saved to a file by clicking the Save button. Then the Load button can be used later to recall these entered values. The peak flow report lists the flow for each subarea at different time. The peak flow value is listed at the end of the report. This value can then be used for Detention Pond Sizing or Channel Design.

See the TR-55 manual for more details on this routine. One difference between Carlson and the TR-55 example is that Carlson interpolates the flow for the subarea Ia/P between the two nearest table Ia/P values whereas TR-55 uses the one closest Ia/P table entry. Consider a subarea with an Ia/P value of 0.14 and table entries of 100 cfs at 0.1 Ia/P and 75 cfs at 0.3 Ia/P. TR-55 would use 100 cfs from the nearest 0.1 Ia/P entry. Carlson would interpolate between 100 and 75 cfs resulting in 95 cfs.

Name	Area	Tc	Tt	Downstream Names	CN
1	.3	1.5	0.0	3,5,7	65
2	.2	1.25	0.0	3,5,7	70
3	.1	.5	.5	5,7	75
4	.25	.7	0.0	5,7	70
5	.2	1.5	1.25	7	75
6	.4	1.5	0.0	7	70
7	.2	1.25	.75		75

Peak Flow Tabular Hydrograph Method

Subarea name (sq. mi.)	Drainage area	Time of concentration	Travel time for subarea	Downstream subarea names	Travel time summation	Rainfall	Curve	Runoff number
1	0.3000	1.50	0.00	3,5,7	2.50	6.00	65	2.35
2	0.2000	1.25	0.00	3,5,7	2.50	6.00	70	2.81
3	0.1000	0.50	0.50	5,7	2.00	6.00	75	3.28
4	0.2500	0.75	0.00	5,7	2.00	6.00	70	2.81
5	0.2000	1.50	1.25	7	0.75	6.00	75	3.28
6	0.4000	1.50	0.00	7	0.75	6.00	70	2.81
7	0.2000	1.25	0.75		0.00	6.00	75	3.28

Time	11.0	11.3	11.6	11.9	12.0	12.1	12.2	12.3
Subarea Discharge (cfs)								
1	0	0	1	1	1	1	1	2
2	0	1	1	1	2	2	2	2

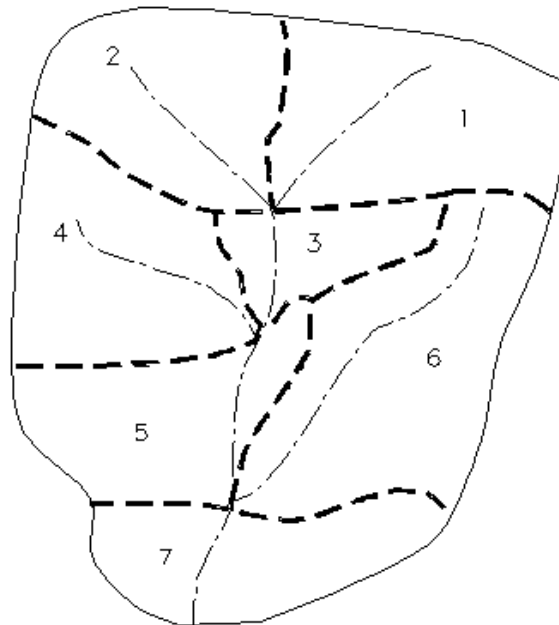
3	1	1	2	2	2	2	3	3
4	2	2	3	3	4	4	4	5
5	3	4	5	7	7	8	9	10
6	4	6	7	10	11	11	12	14
7	6	8	11	15	18	24	34	51
Total	17	23	30	40	45	53	65	87

Time	12.4	12.4	12.6	12.7	12.8	13.0	13.2	13.4
Subarea	Discharge (cfs)							
1	2	2	2	2	3	3	3	4
2	2	2	3	3	3	4	4	5
3	3	3	4	4	5	6	7	11
4	5	6	6	7	7	9	11	16
5	11	13	16	20	26	48	80	115
6	16	19	23	29	39	74	127	186
7	75	104	137	165	184	202	173	139
Total	114	149	189	230	267	345	406	475

Time	13.6	13.8	14.0	14.3	14.6	15.0	15.5	16.0
Subarea	Discharge (cfs)							
1	5	6	8	17	38	85	134	130
2	6	8	13	27	56	101	122	92
3	20	39	65	96	91	59	29	17
4	28	54	96	161	185	147	82	48
5	143	156	152	126	97	67	44	32
6	234	257	253	213	166	116	78	57
7	107	85	69	52	41	31	25	21
Total	544	606	654	692	674	608	516	397

Time	16.5	17.0	17.5	18.0	19.0	20.0	22.0	26.0
Subarea	Discharge (cfs)							
1	94	64	46	35	25	19	15	10
2	59	39	28	22	16	13	10	7
3	13	11	10	9	7	6	5	3
4	33	27	23	21	17	15	11	8
5	25	21	18	16	13	12	9	3
6	46	39	33	30	25	22	17	6
7	18	16	15	13	12	11	8	1
Total	289	217	173	146	115	99	74	39

Peak Discharge: 692 cfs



Pulldown Menu Location: Watershed

Keyboard Command: peakflow

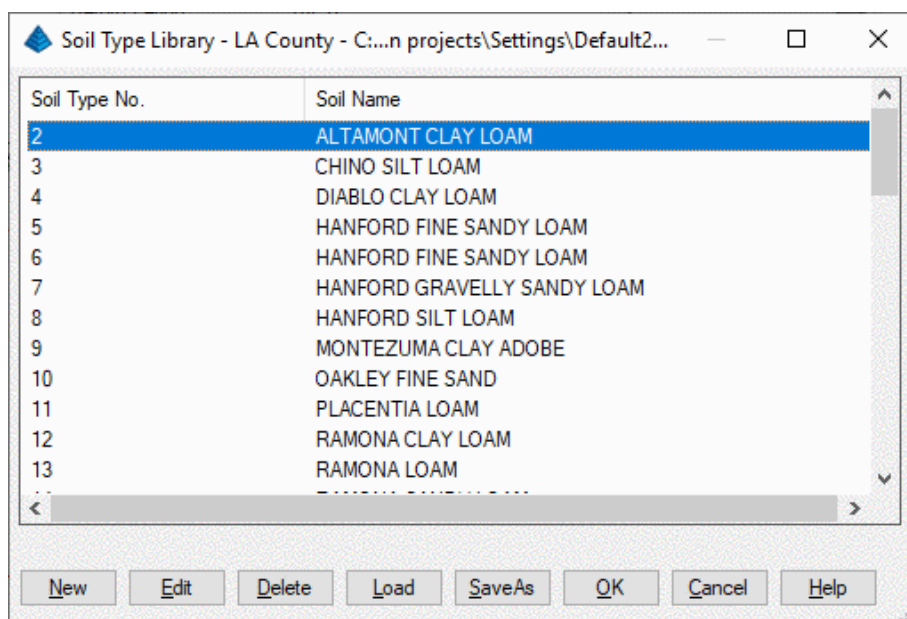
Prerequisite: None

Peak Flow - Modified Rational Method (LA County)

The Peak Flow - Modified Rational Method is an extension of the rational method used to create runoff hydrographs from a watershed of any size over a specific time period. The Rational method is limited to considering storms with a duration equal to the time of concentration and provides only a peak flow. The Modified Rational Method can consider single event storms with changing intensities and longer durations. It was developed as a means to produce hydrographs for storage design based on the rational method. Please refer to Chapter 12, Hydrology Manual of Los Angeles County Department of Public Works for hydrograph calculation.

Select Modified Rational Method (LA County) from Watershed > Peak Flow menu. In the dialog, specify the Drainage Area, Soil Type, Proportion Impervious Data, Time of Concentration, Storm duration and the Rational rainfall data. The drainage area can be obtained from picking a closed polyline and the program calculates the area automatically. Click on Calculate button to see the result.

The Soil Type library has all the tables from the Appendix C in the Hydrology Manual. New Soil Type and Runoff Coefficient Data can be added to the library.



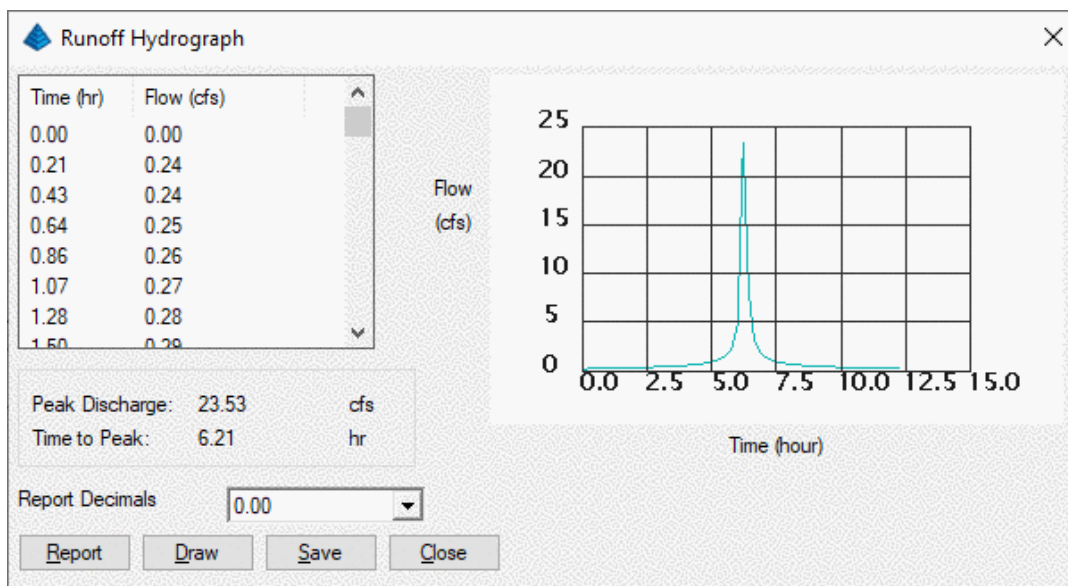
The Proportion Impervious Data library is from the Appendix D in the Hydrology Manual. New data is allowed to add to the library.

Proportion Impervious Data - C:\...ad10_x64\User\Imperviousstable.dta

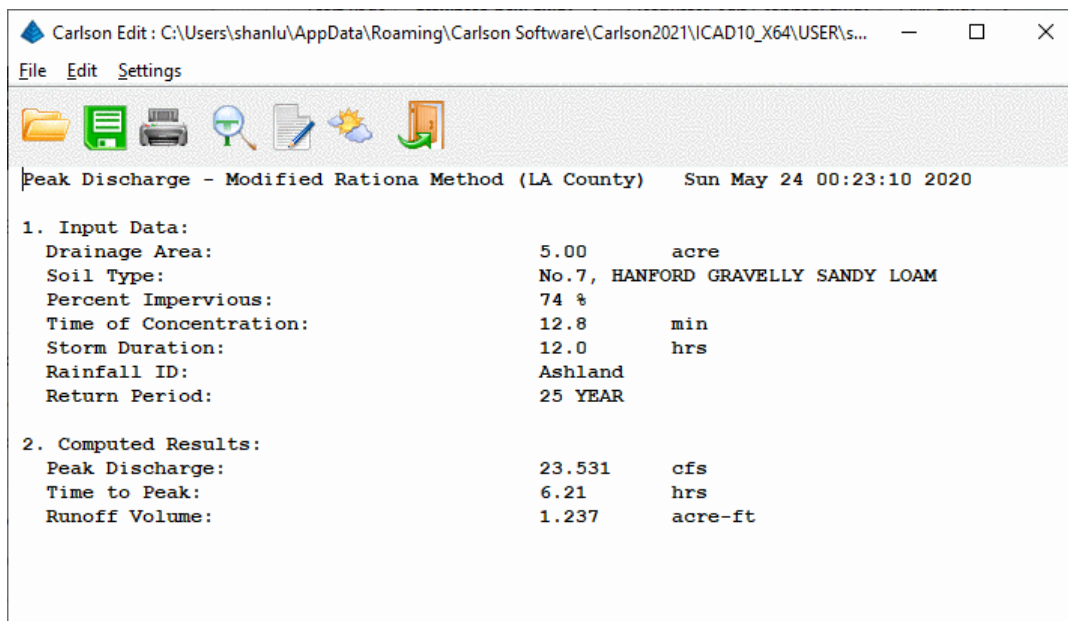
LA County Code	Land Use Description	% Impervious
1111	High-Density Single Family Residential	42
1112	Low-Density Single Family Residential	21
1121	Mixed Multi-Family Residential	74
1122	Duplexes, Triplexes and 2-or 3-Unit Condominiums and Townhouses	55
1123	Low-Rise Apartments, Condominiums, and Townhouses	86
1124	Medium-Rise Apartments and Condominiums	86
1125	High-Rise Apartments and Condominiums	90
1131	Trailer Parks and Mobile Home Courts, High-Density	91
1132	Mobile Home Courts and Subdivisions, Low-Density	42
1140	Mixed Residential	59
1151	Rural Residential, High-Density	15
1152	Rural Residential, Low-Density	10
1211	Low- and Medium-Rise Major Office Use	91
1212	High-Rise Major Office Use	91
1213	Skyscrapers	91

New Edit Delete Load SaveAs OK Cancel Help

Peak Flow Hydrograph:



Peak Flow Report:



Pulldown Menu Location: Watershed > Peak Flow

Keyboard Command: peakflw_modrat

Prerequisite: None

Watershed Settings (Save and Load)

These commands save and load watershed parameters to a data file with a .HYD file name extension. The watershed values include settings from the commands in the top portion on the Watershed menu such as rainfall, storm type, weighted curve number. These commands allow you to recall these values after reloading the drawing at a later time.

Pulldown Menu Location: Watershed

Keyboard Command: saveshed, loadshed

Prerequisite: none

Pipe Routing Hydrograph

Pipe routing transforms a sub-basin runoff hydrograph from the point of discharge in the pipe to a downstream location. As the hydrograph flows to the downstream area it undergoes changes in shape and distribution due to translation and reach storage effects. This program implements the Storage-Indication methodology, which accounts for storage effects and is derived from the continuity equation:

$$\Delta S / \Delta t = I - O$$

Where: ΔS = Change in storage (cubic ft or cubic meter)

Δt = Routing time step (hours)

I = Inflow discharge (cfs or cms)

O = Outflow discharge (cfs or cms)

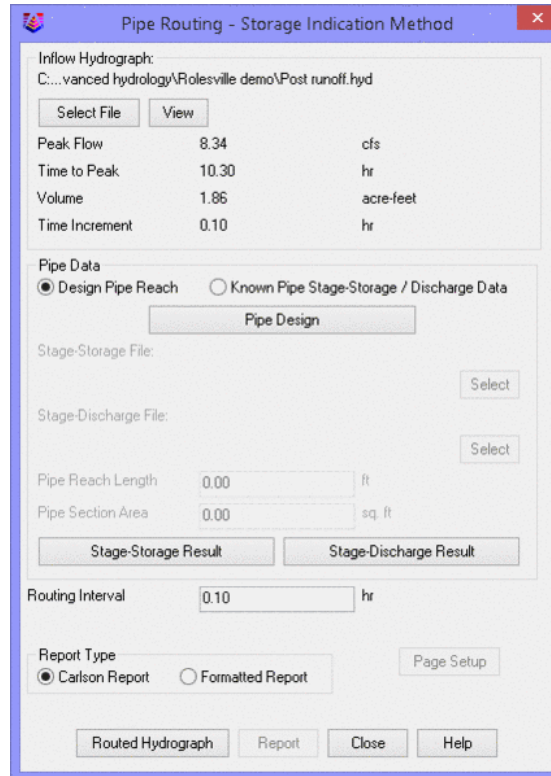
Please refer to the National Engineering Handbook Section 4 for the detail of the Storage-Indication method.

Select Pipe Routing from Watershed > Hydrograph Routing menu. This command opens a dialog for hydrograph routing. First, select the inflow hydrograph file. Second, either design a pipe or enter the stage-storage and

stage-discharge data of a known pipe. When design a pipe, enter pipe parameters in the pipe design dialog, the headwater and tailwater elevations are also need to be specified for stage-discharge calculation. If the pipe is known, select its stage-storage and stage-discharge files, and then specify the pipe length and its section area for calculating flow travel time.

The numerical accuracy and stability of pipe routing is sensitive to the routing interval Δt . The average reach travel time is computed by dividing the flow velocity corresponding to the average discharge for the inflow hydrograph into the pipe. If the travel time is less than the time interval, then negative outflow could occur and the calculation is not accurate and the program generates a warning message asking for a smaller routing interval.

After entering all information, click Routed Hydrograph button to generate the routing hydrograph. A dialog is opened with the tabular and graphic hydrograph data, from there you can draw the hydrograph on screen and save it to a file.



Routing Hydrograph - Pipe Routing

Pipe Design

Shape: Circular

Diameter: 36.00 in

Width: 0.00 in

Manning's n: 0.0130 [Select]

Number of Barrels: 1

Inlet Loss Coefficient: 0.00

Inlet Invert Elev: 363.00 ft

Outlet Invert Elev: 333.00 ft

Length: 3000.00 ft

Slope: 0.0100 ft/ft

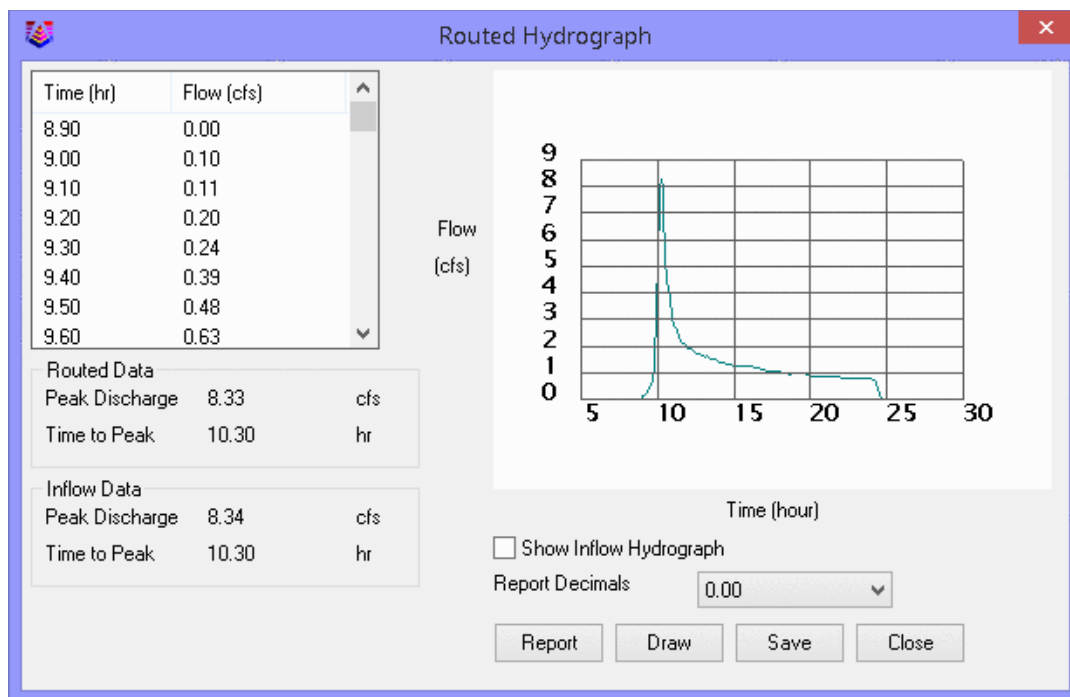
Max. Headwater Elev: 370.00 ft

Tailwater Elev: 0.00 ft

Stage Increment: 0.10 ft

OK Cancel Help

Pipe Routing - Pipe Design Dialog



Routed Hydrograph Dialog

Pulldown Menu Location: Watershed > Hydrograph Routing > Pipe Routing

Keyboard Command: piperout

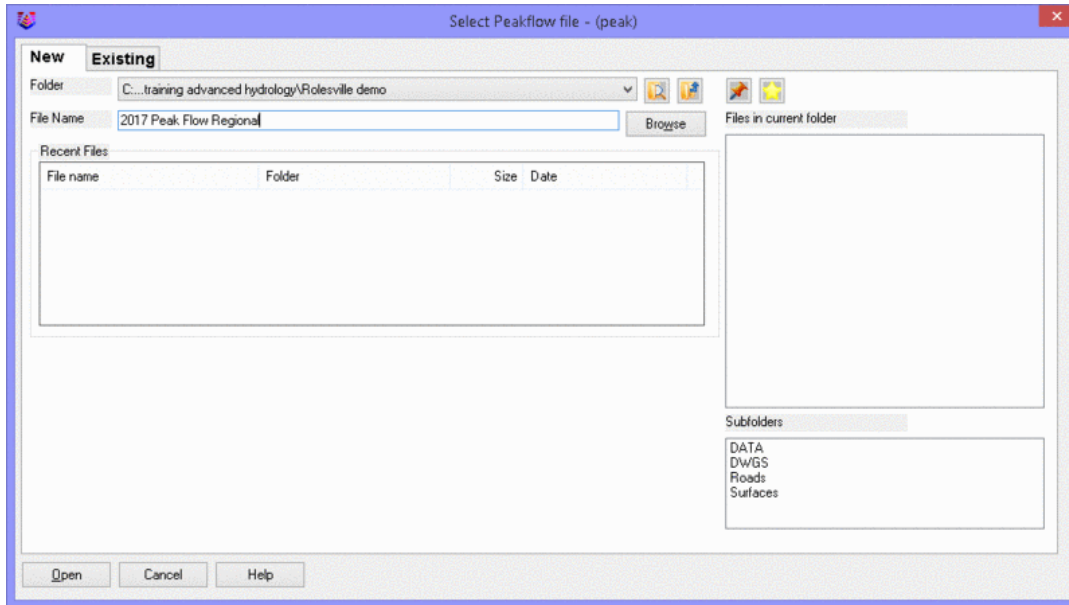
Prerequisite: None

Design Detention Pond

This command calculates peak flow using the regional method, as stated in the DR-401 manual. Regression Equation Methodology has been adopted by the Cabinet to calculate peak flows. The method is described in USGS Water Resources Investigations Report 03-4180 (2003) titled "Estimating the Magnitude of Peak Flows for Streams in Kentucky for Selected Recurrence Intervals". The method divides the state into seven regions. Regression

equations have been developed for each region based on stream gage station data and other parameters that are applicable to the watersheds that were analyzed.

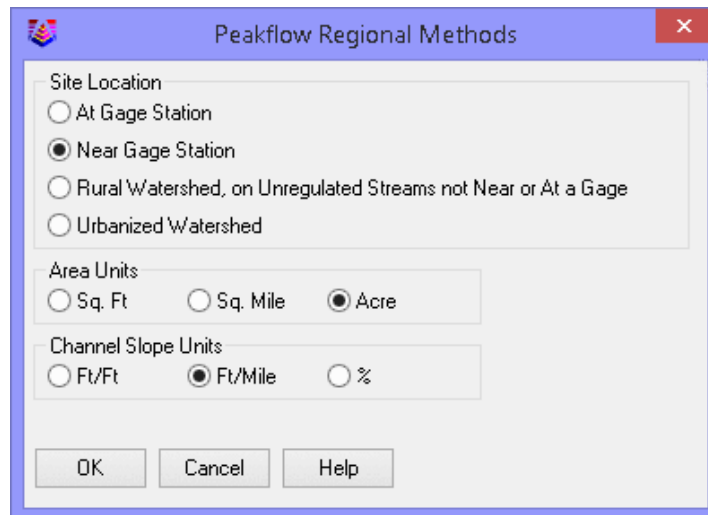
When you first run the command, the Select Peak Flow File - (peak) dialog box opens. You can start a completely new pond project or open an existing pond project.



Select or Create *.peak File

Peak Flow Regional Methods:

When you start a new Peak Flow Regional project, the Peak Flow Regional Methods settings dialog will open first to let you select the Site Location, Area Units, and Channel Slope Units. Select OK



Peak Flow Regional Methods

Rural Regional Methods:

Set Project Name, Select Region, Return Period, Input manually or select a closed polyline to calculate Drainage

Area. Input manually or select a Flow Line from drawing file to define Channel Slope. Press Calculate to see the Peak Flow (cfs).

Rural Regional Method

Project Name: 2017 Peak Flow Regional

Region: Region 1, North

Return Period: 2 Year

Drainage Area: 15.60 Acre

Channel Slope: 7.0000 Percent

Peakflow (cfs): 25.62

Buttons: Calculate, Report, Load, Save, Save As, Exit

Peak Flow - Rural Regional Method

Mon Jun 06 15:01:23 2016

Project:	2017 Peak Flow Regional
Region:	Region 1, North
Return Period:	2 Year
Drainage Area	A = 15.60 Acre
Main Channel Slope	S = 7.00 %
Peak Flow:	Qp = 25.62 cfs

Prompts

Peak Flow Regional: Fill in values.

Pulldown Menu Location: Watershed > Peak Flow > Regional Method

Keyboard Command: regional_peakflow

Prerequisite: none

Hydrograph Editor

This command can edit an existing Hydrograph file (.hyd) or be used to create a custom Hydrograph file by manually entering time and flow rate in a spreadsheet style table.

If a design requires a special hydrograph that is either developed by a method outside of Carlson or is used to simulate a special runoff series created by a non standard outlet structure such as a storm system with a pump, a Hydrograph file can be created with this command by manually entering the time series.

This hydrograph file can then be used as the input runoff time series when routing a storm water detention model.

In the Graph function, the dialog shows a graph of the data and has functions to make a report and to draw the graph.

Pulldown Menu Location: Watershed > Hydrograph Routing

Keyboard Command: spreadhyd

Prerequisite: None

Runoff Hydrograph - SCS Method

This function applies the SCS hydrograph procedure and the same methodology as used in TR-20 to generate runoff hydrograph by specifying either a triangular hydrograph shape or a curvilinear one. There are five unit hydrograph options: Delmarva, SCS Curvilinear, SCS Triangular, Clark and Snyder unit hydrograph method. The SCS method is best suited for large watersheds.

Runoff Hydrograph - SCS Method

Unit Hydrograph: SCS Triangular [Set]

SubBasin

Area Units: Sq Ft Acres Sq Miles

Drainage Area: 50.0000000 [Select]

Runoff Curve Number: 75.00 [Set]

Clark UH Storage Coefficient, R: 2.00

Tc Units: Minute Hour

Time of Concentration: 60.0 min [Set]

Base Flow: 0.0000 cfs

Antecedent Moisture Condition: 1 2 3

Report Type: Standard Report Formatted Report [Report Setup]

Calculate HEC-HMS Unit Hydrograph Runoff Hydrograph Report Close Help

Rainfall

Storm Type: TYPE II 24-HOUR [Library]

Rational Rainfall ID: Rain2 [Library]

Return Period: 25 Year

Rainfall Depth: 7.93 in

Time Increment: 0.10 hr

Peak Attenuation Factor: 484

Calculation

Peak Discharge: 126.919 cfs

Time to Peak: 12.70 hr

Volume: 20.70 acre-ft

SCS Curvilinear Unit Hydrograph uses the dimensionless unit hydrograph developed by Victor Mockus. This dimensionless hydrograph has its ordinate values expressed in a dimensionless ratio Q/Q_p (discharge at time t to total discharge) or Q_a/Q (accumulated volume at time t to total volume) and its abscissa values as T/T_p (time t to peak time). When calculating the unit hydrograph for certain watershed, Delta, the internal calculation increment, is $0.1333T_c$ for each given subarea. Slightly adjust Delta so that T_p on the unit hydrograph will fall exactly on one of the ordinates spaced at Delta. This method creates the unit hydrograph for one inch of direct runoff using the 51 evenly spaced Q/Q_p ratios used internally by TR-20.

SCS Triangular Unit Hydrograph uses an equivalent triangular hydrograph having the same units of time and discharge to represent the dimensionless curvilinear unit hydrograph. Please refer to National Engineering Handbook Section 4 for the equations used in generating both SCS Curvilinear and SCS Triangular unit hydrograph.

Delmarva Unit Hydrograph is an alternative to the 484 dimensionless unit hydrograph developed for the Delmarva region (Delaware, Maryland, Virginia peninsula) based on data from four gaged watersheds. Peak rate factor of 284 is used in this method.

Clark Unit Hydrograph is a synthetic unit hydrograph method. There's no need to develop a unit hydrograph through the analysis of past observed hydrograph. Instead, a time vs. area curve built into the program is used

to develop the translation hydrograph resulting from a burst of precipitation. The resulting translation hydrograph is routed through a linear reservoir to account for storage attenuation effects across the subbasin. The time of concentration defines the maximum travel time in the subbasin; the storage coefficient is used in the linear reservoir that accounts for the storage effects.

Snyder Unit Hydrograph is based on a study of watersheds in the Appalachian Highlands. In basins ranging from 10 to 10,000 square miles.

Snyder relations are: $t_p = C_t(LL_c)^{0.3}$

Where:

t_p = basin lag (hr)

L = length of the main stream from the outlet to the divide (mi)

L_c = length along the main stream to a point nearest the watershed centroid (mi)

C_t = Coefficient usually ranging from 1.8 to 2.2

$Q_p = 640C_pA/t_p$

Where:

Q_p = peak discharge of the unit hydrograph (cfs)

A = drainage area (mi²)

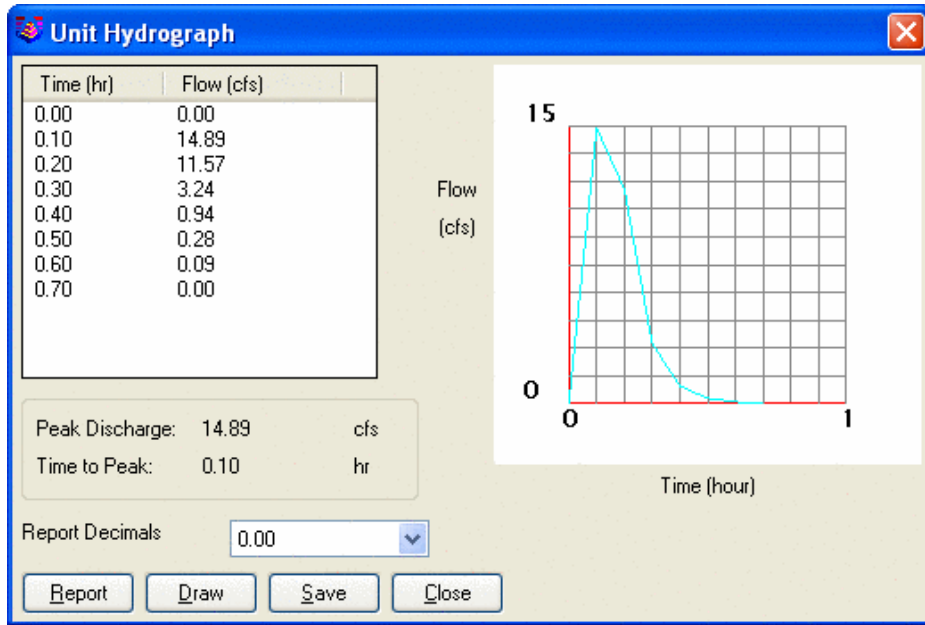
C_p = storage coefficient ranging from 0.4 to 0.8, where larger values of C_p are associated with smaller values of C_t

Select SCS Method from Watershed > Hydrograph Routing > Watershed Hydrograph menu. This command opens a dialog for hydrograph calculation. First choose which method to use, SCS Curvilinear or SCS Triangular. Then enter the sub-basin and rainfall data. When creating an SCS Triangular unit hydrograph, you need to enter the unit hydrograph peak attenuation factor, which controls the area under the unit hydrograph before the time to peak. Usually a factor of 484 is used. This factor varies from about 600 in steep terrain to 300 in very flat swampy areas. When creating an SCS Curvilinear unit hydrograph, the peak attenuation factor is default to 484.

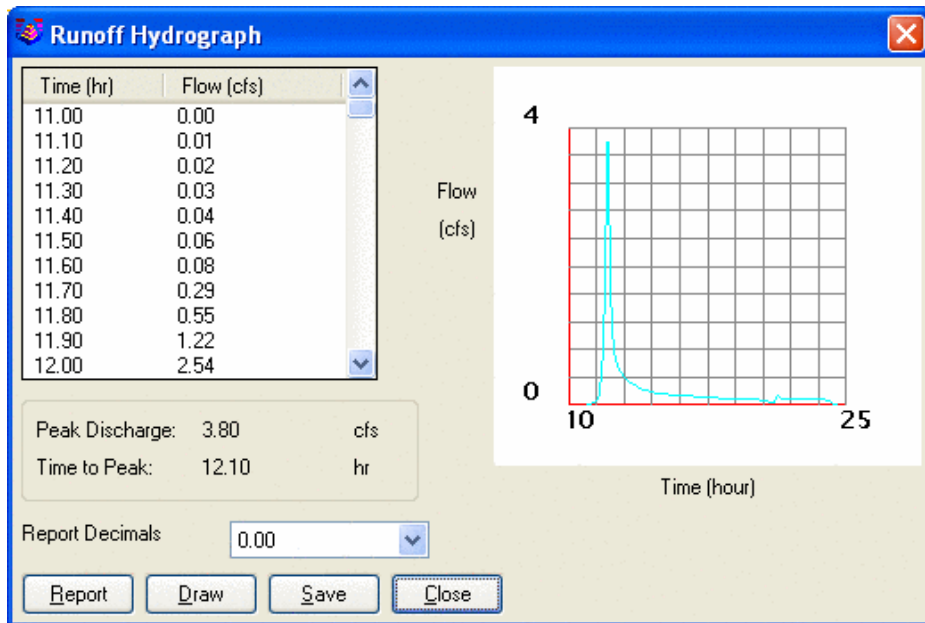
The program can get Rainfall Depth automatically based on user specified Rational rainfall ID, return period and time of concentration.

After entering all information, click Calculation button to generate the unit and runoff hydrograph. Unit Hydrograph button and Runoff Hydrograph button open a dialog with the tabular and graphic data of the unit and runoff hydrograph correspondingly, from there you can draw the hydrograph on screen, generate the hydrograph data in the report format and save the hydrograph to a file.

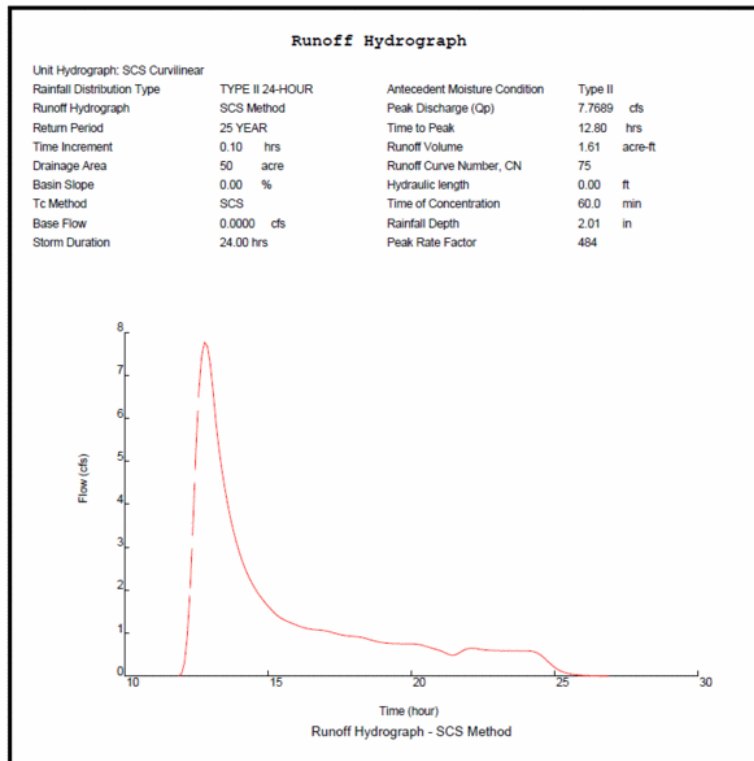
The calculation result can be reported in PDF format with design data and hydrograph curve.



Unit Hydrograph Dialog



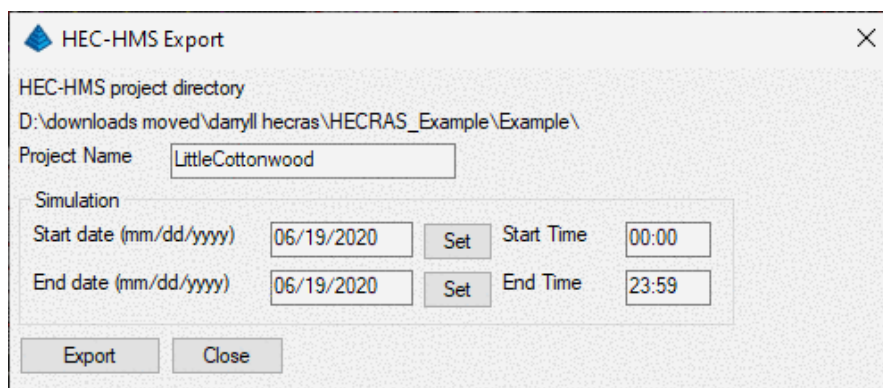
Runoff Hydrograph Dialog



Data can also be exported to HEC-HMS. Clicking the HEC-HMS button reveals the HEC-HMS Support dialog. Use the Set button to assign a job directory. Input an existing HEC-HMS project name for import or create a project name for export. Then click the desired function - Import, Export or Close at the bottom to interface with HEC-HMS. You may learn more about HEC-HMS, which stands for Hydrologic Engineering Center - Hydrologic Modeling System, a very robust system for modeling both large and small areas with tributary flows, as well as download the software here:

<https://www.hec.usace.army.mil/software/hec-hms/>

Input the Project Name, choose Start Date, End Date, Start Time, End Time then click Export.



Pulldown Menu Location: Watershed > Hydrograph Routing > Watershed Hydrograph > SCS Method

Keyboard Command: runoffhyd

Prerequisite: None

Runoff Hydrograph - TR-55 Tabular Method

The TR-55 Tabular Method approximates TR-20, which is a more detailed hydrograph procedure. This method can develop partial composite runoff hydrographs at any point in a watershed by dividing the watershed into homogeneous subareas. The tabular discharge values for the various rainfall distribution, presented in *TR-55, Urban Hydrology for Small Watersheds* (U.S. Soil Conservation Service), are used in this method. Tabular discharges expressed in csm/in are given for a range of subarea Tc's from 0.1 to 2 hours and reach Tt's from 0 to 3 hours.

Select TR-55 Tabular Method from Watershed > Hydrograph Routing > Watershed Hydrograph menu. This command opens a dialog for hydrograph calculation. Choose the area unit and select the appropriate storm type for the project location. See the storms maps in the TR-55 manual to determine the storm type. Enter the rainfall depth for the watershed. Select the Ia/P Interpolation option to interpolate the Ia/P ratio, otherwise the calculated Ia/P value would be rounded to the constants 0.10, 0.30 or 0.50. Next, enter the watershed data in the spreadsheet. In the Subarea Name column, enter a short name for each subarea, using numbers or letters. In the Area column, type the area value. You can also click the Select Area button on the bottom of the spreadsheet to calculate the drainage area by selecting a closed polyline.

In the Tc (time of concentration) and Tt (Travel Time) columns, enter the corresponding values. Tt specifically refers to the total time for water to flow from a subarea's outfall to composite outfall point. When the Tc value for a particular subarea doesn't exist in the tabular hydrograph for the given distribution, it will be rounded off in order to determine which hydrograph table to read from. Please refer to Tr-55 for the method used for rounding off Tc and Tt values.

In the Downstream Subarea Name column, type the list of the names of the downstream subareas. Separate each name with a comma. In the CN (Runoff Curve Number) column, enter the CN value of the subarea. Insert Row and Delete Row buttons allow you to do the spreadsheet editing. You can also load a runoff tabular file that you previously saved by clicking on Load button, and save current calculation by Save and SaveAs buttons. The data is stored in a .tab file.

After entering all information, click on Calculation button to generate the runoff hydrograph. Runoff Hydrograph button opens a dialog with the tabular and graphic hydrograph data, from there you can draw the hydrograph on screen and save it to a file.

Area Units: Sq. Ft Sq. Mile Acres

Storm Type: I IA II III

Rainfall Depth: 3.00 in Base Flow: 0.00 cfs Ia / P interpolation

	Subarea Name	Area	Tc (hrs)	Tt (hrs)	Downstream Subarea Name	CN
1	1	2.0000	0.50	0.40	2,3	85
2	2	3.0000	0.70	0.60	3	92
3	3	3.0000	0.90	0.60		75
4						

Calculation:
 Peak Discharge: 6.01 cfs
 Time to Peak: 13.20 hrs
 Volume: 0.42 acre-ft

Buttons: Calculate, Hydrograph, Report, Load, Save, Save As, Close, Insert Row, Delete Row, Select Area

Runoff Hydrograph - TR-55 Tabular Method

Pulldown Menu Location: Watershed > Hydrograph Routing > Watershed Hydrograph > TR-55 Tabular Method

Keyboard Command: runoffhyd2

Prerequisite: None

Runoff Hydrograph - Rational Method

The Rational Method is best known for estimating peak stormwater discharge which is used to evaluate collection system capacity. However, the Rational Method cannot be used on its own to determine capacity of Stormwater Systems that involve storage and are non-steady state. It can be combined with one of the following methods to produce a hydrograph capable of modeling non-steady state stormwater systems such as a detention basin.

Modified Rational

Dekalb Rational

Universal Rational

User Defined Rational.

Each method requires a predefined IDF curve and the time of concentration to determine the peak flow using the Rational formula. This value is the peak discharge value in each Hydrograph.

The **Modified Rational Method** calculates the peak discharge rate using the standard $Q=CIA$ formula where

Q=Peak Stormwater Discharge

C=The Rational Runoff Coefficient

I=Stormwater Runoff Intensity (in/hr)

A=Watershed Drainage Area in Acres

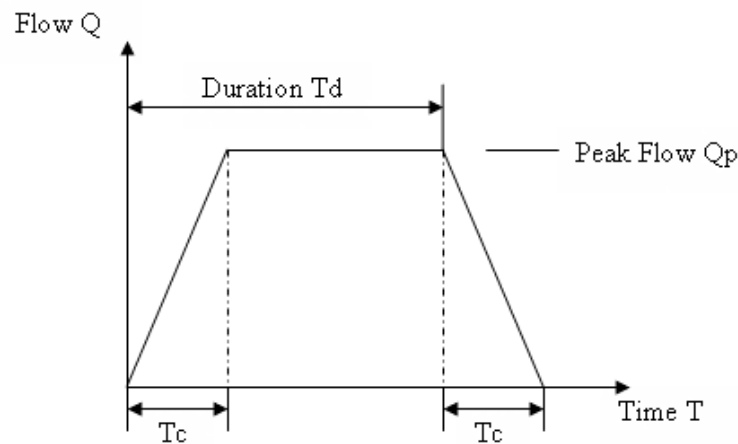
Note that the Intensity value "I" is the **Runoff Intensity** which is equal to the rainfall intensity at the Time of Concentration for a given design storm event. However, this value does not give us any indication of the total volume of stormwater runoff which is needed for modeling Stormwater Detention Systems. If the Storm Event duration is exactly equal to the time of concentration, the Runoff Hydrograph is represented by the following.

This represents a rare storm event. Most storm events last much longer than the time of concentration and will produce greater runoff Volume than what is represented in the previous Hydrograph. To determine storage volume requirements, the length of the storm event must be included in the Hydrograph as shown below:

The stormwater duration is either provided in Regulatory Requirements or can be estimated related to a storm frequency by total Volume of Runoff. For collection system design, only the peak rate is needed to size a collection system to a specific storm event. However, systems with detention volume must be able to handle both peak discharge and total Runoff Volume in order to handle all the variations of the 10-year storm. A 10-year storm event with a high intensity will have a short duration compared to a 10-year event with a low intensity. As shown on the table below, each rainfall event represented a 10-year storm.

By using the 24-hour storm event, we can assume that the detention basin will have enough storage for the storm events shown in the following table.

Using the 24-hour volume is not written in stone. It is common to design detention basins using the 6-hour storm event. . The **Rising Limb Factor** is used to multiply the time of concentration to get the peak time when it's not at the time of concentration; the **Receding Limb Factor** changes the slope of the receding limb. Both factors are default to 1. Peak flow Q_p is calculated by CIA, where the rainfall intensity I is at rainfall duration T_d . The **Flow Multiplier** is used to multiply the Q_p for the hydrograph peak flow. The **Time Steps Before Peak** determines the hydrograph time increment.



Modified Rational Hydrograph

The Delta Rational method uses the Modified Rational method as its foundation. Peak discharge is computed using the Rational formula, then the time to peak and the time of base are adjusted to five and ten times the time of concentration. Ten discharge values at increments of time of concentration are computed by scaling the peak discharge appropriately according to the following template.

T/T_c	Q/Q_p if $T_c < 20.0$ min	Q/Q_p if $T_c \geq 20.0$ min
0	0	0
1	0.16	0.04
2	0.19	0.08
3	0.27	0.16
4	0.34	0.32
5	1.00	1.00
6	0.45	0.30
7	0.27	0.11

8	0.19	0.05
9	0.12	0.03
10	0.0	0.0

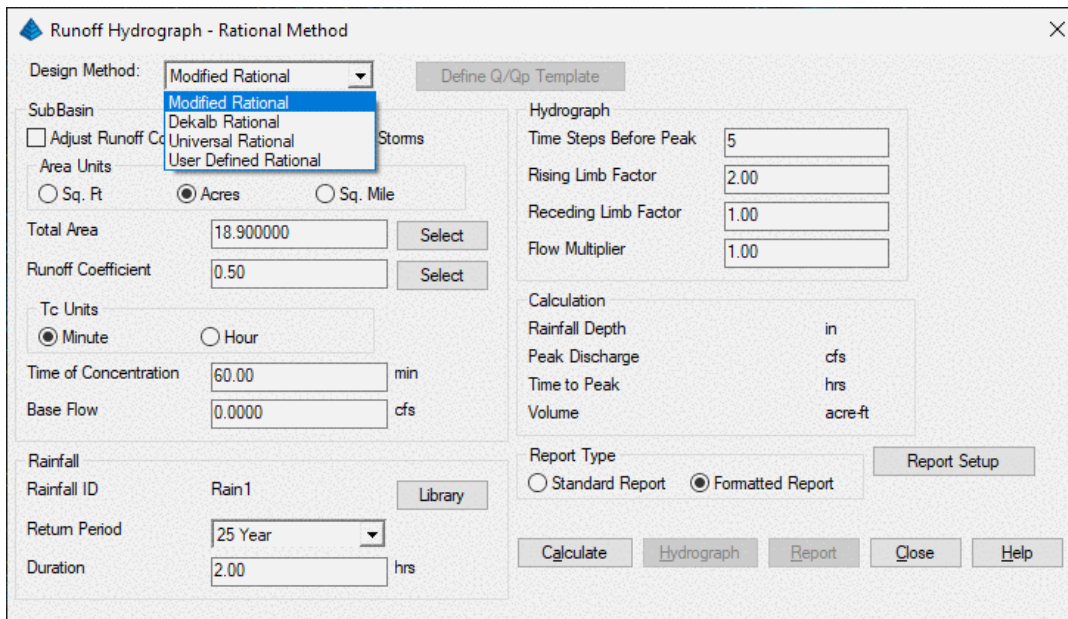
The Universal Rational method uses multiples of predefined coefficients (see the following table) and the peak discharge to compute ordinates at different times. Peak discharge is assumed to be reached at a time equal to three times the time of concentration. The time of base of the hydrograph is computed as eleven times the time of concentration.

Time (min)	Q
0	0.0
Tc	0.21Qp
2Tc	0.30Qp
3Tc	Qp
4Tc	0.54Qp
5Tc	0.39Qp
6Tc	0.25Qp
7Tc	0.18Qp
8Tc	0.15Qp
9Tc	0.14Qp
10Tc	0.13Qp
11Tc	0.0

The User Defined Rational method is similar to the Dekalb Rational method and the Universal Ration method, but with different templates provide by users. This method computes a hydrograph by using a dimensionless hydrograph template that has values for T/Tc versus Q/Qp.

Select Rational Method from Watershed > Hydrograph Routing > Watershed Hydrograph menu. This command opens a dialog for hydrograph calculation. First choose what method to use, then enter the sub-basin and rainfall data. The IDF curves will be computed after you choose a rainfall from the rainfall library. When use the User Defined Ration method, you will need to define the Q/Qp template.

After entering all information, click Calculation button to generate the runoff hydrograph. Runoff Hydrograph button opens a dialog with the tabular and graphic data, from there you can draw the hydrograph on screen and save the hydrograph data to a file.



Runoff Hydrograph - Rational Method

Pulldown Menu Location: Watershed > Hydrograph Routing > Watershed Hydrograph > Rational Method

Keyboard Command: runoffhyd3

Prerequisite: None

Pipe Routing Hydrograph

Pipe routing transforms a sub-basin runoff hydrograph from the point of discharge in the pipe to a downstream location. As the hydrograph flows to the downstream area it undergoes changes in shape and distribution due to translation and reach storage effects. This program implements the Storage-Indication methodology, which accounts for storage effects and is derived from the continuity equation:

$$\Delta S / \Delta t = I - O$$

Where: ΔS = Change in storage (cubic ft or cubic meter)

Δt = Routing time step (hours)

I = Inflow discharge (cfs or cms)

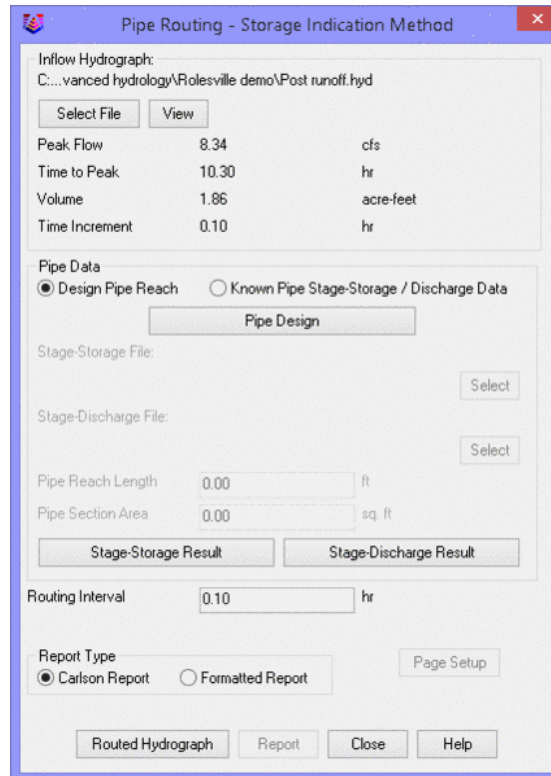
O = Outflow discharge (cfs or cms)

Please refer to the National Engineering Handbook Section 4 for the detail of the Storage-Indication method.

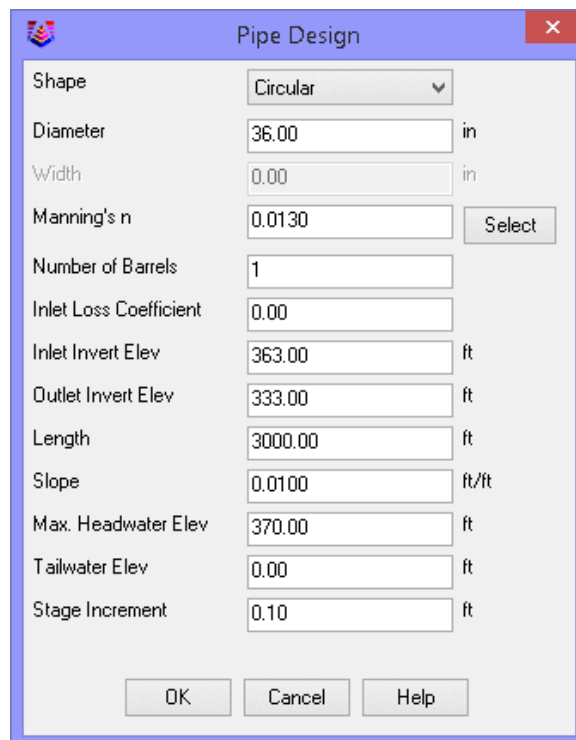
Select Pipe Routing from Watershed > Hydrograph Routing menu. This command opens a dialog for hydrograph routing. First, select the inflow hydrograph file. Second, either design a pipe or enter the stage-storage and stage-discharge data of a known pipe. When design a pipe, enter pipe parameters in the pipe design dialog, the headwater and tailwater elevations are also need to be specified for stage-discharge calculation. If the pipe is known, select its stage-storage and stage-discharge files, and then specify the pipe length and its section area for calculating flow travel time.

The numerical accuracy and stability of pipe routing is sensitive to the routing interval Δt . The average reach travel time is computed by dividing the flow velocity corresponding to the average discharge for the inflow hydrograph into the pipe. If the travel time is less than the time interval, then negative outflow could occur and the calculation is not accurate and the program generates a warning message asking for a smaller routing interval.

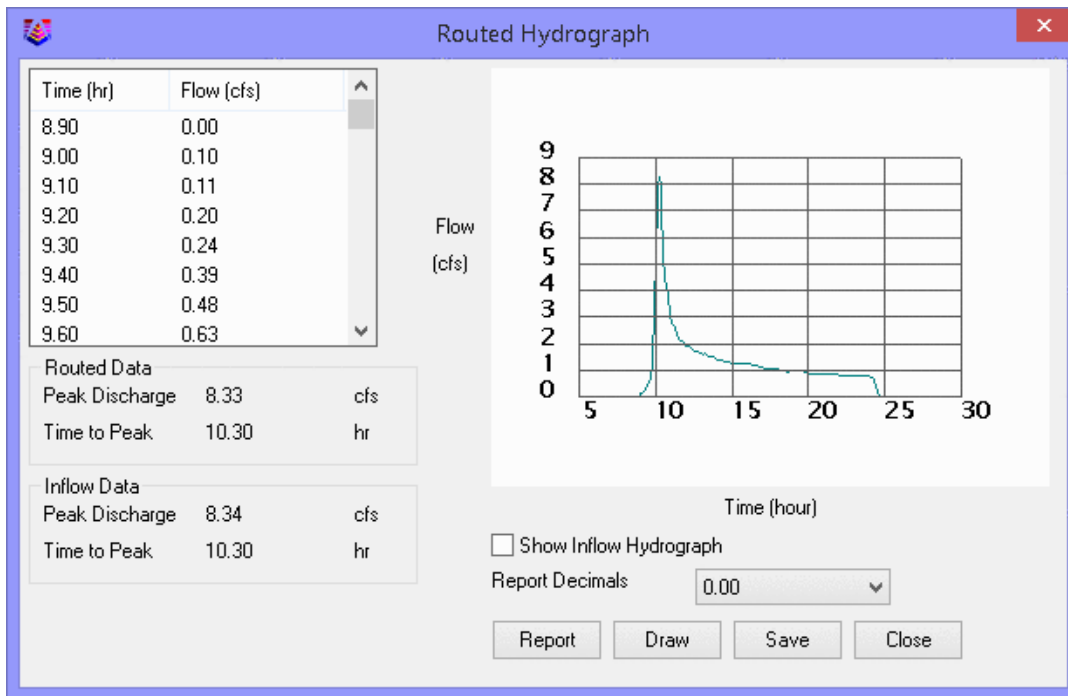
After entering all information, click Routed Hydrograph button to generate the routing hydrograph. A dialog is opened with the tabular and graphic hydrograph data, from there you can draw the hydrograph on screen and save it to a file.



Routing Hydrograph - Pipe Routing



Pipe Routing - Pipe Design Dialog



Routed Hydrograph Dialog

Pulldown Menu Location: Watershed > Hydrograph Routing > Pipe Routing

Keyboard Command: piperout

Prerequisite: None

Reservoir Routing Hydrograph

Reservoir routing is used to determine the peak flow attenuation that a hydrograph undergoes as it enters a reservoir. A reservoir is defined by a stage-storage curve and a stage-discharge curve, which can be entered or computed using commands in the Structure pull-down menu. A runoff hydrograph travels through the reservoir to an outlet. The outflow hydrograph is calculated by Storage Indication method, which is developed from the continuity equation:

$$\Delta S / \Delta t = I - O$$

Where: ΔS = Change in storage (cubic ft or cubic meter)

Δt = Routing time step (hours)

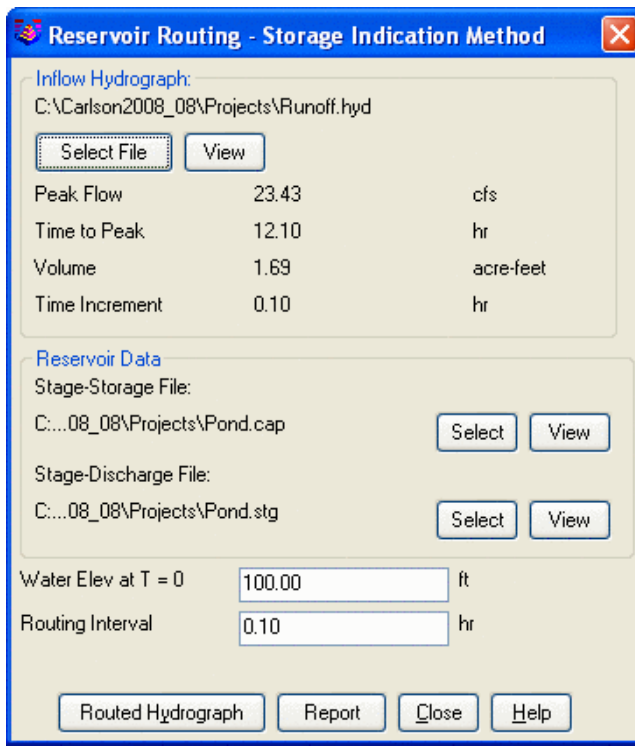
I = Inflow discharge (cfs or cms)

O = Outflow discharge (cfs or cms)

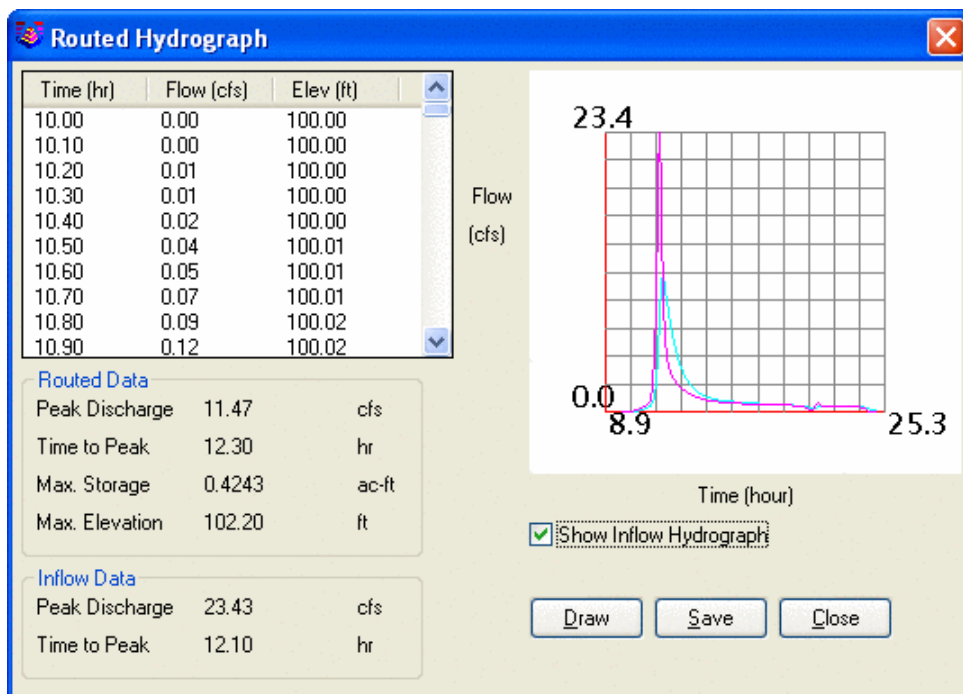
Please refer to the National Engineering Handbook Section 4 for the detail of the Storage-Indication method.

Select Reservoir Routing from Watershed > Hydrograph Routing menu. This command opens a dialog for hydrograph routing. First, select the runoff hydrograph file that you want to do the reservoir routing. Second, select the stage-storage and stage-discharge curve files of a known reservoir. Then specify the initial water elevation at the time when the routing starts and the routing interval. When the routing interval is too large, negative outflows can occur and the program would prompt a error message asking for a smaller interval.

After entering all information, click Routed Hydrograph button to generate the routing hydrograph. A dialog is opened with the tabular and graphic hydrograph data, from there you can draw the hydrograph on screen and save it to a file.



Routing Hydrograph - Reservoir Routing



Routed Hydrograph Dialog

Pulldown Menu Location: Watershed > Hydrograph Routing > Reservoir Routing

Keyboard Command: pondrout

Prerequisite: None

Channel Routing Hydrograph - Convex Method

Channel routing is used to model the attenuation of a hydrograph's peak flow rate as it travels along a channel. Attenuation occurs due to the inherent storage capacity of a channel when the capacity of the channel is greater than the flow capacity.

The Convex method is derived from inflow-outflow hydrograph relationships. The following equation is used:

$$O_{i+1} = CI_i + (1 - C)O_i$$

where: O_{i+1} = Outflow discharge at (i+1)th time step (cfs or cms)

C = Routing Coefficient

I_i = Inflow discharge at (i)th time step (cfs or cms)

O_i = Outflow at (i)th time step (cfs or cms)

The routing coefficient C can be computed using an empirical equation by SCS.

$C = V / (V + 1.7)$ for English units

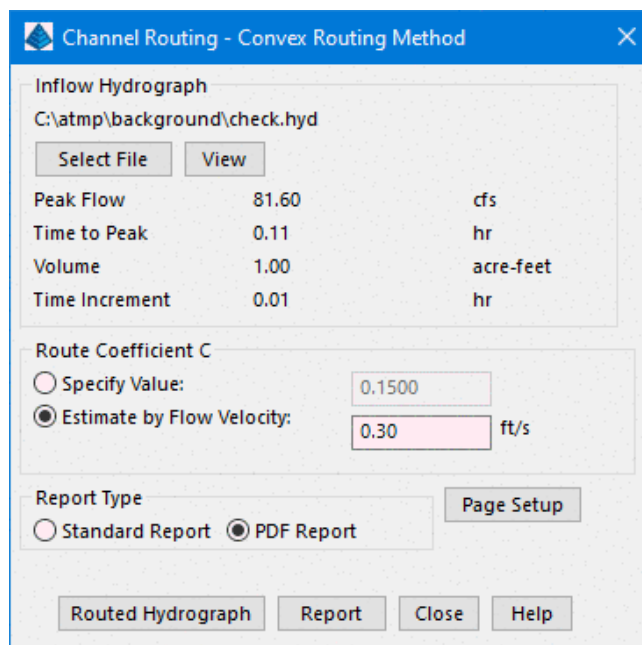
$C = V / (V + 0.518)$ for Metric units

where: V = Flow velocity (ft/s or m/s)

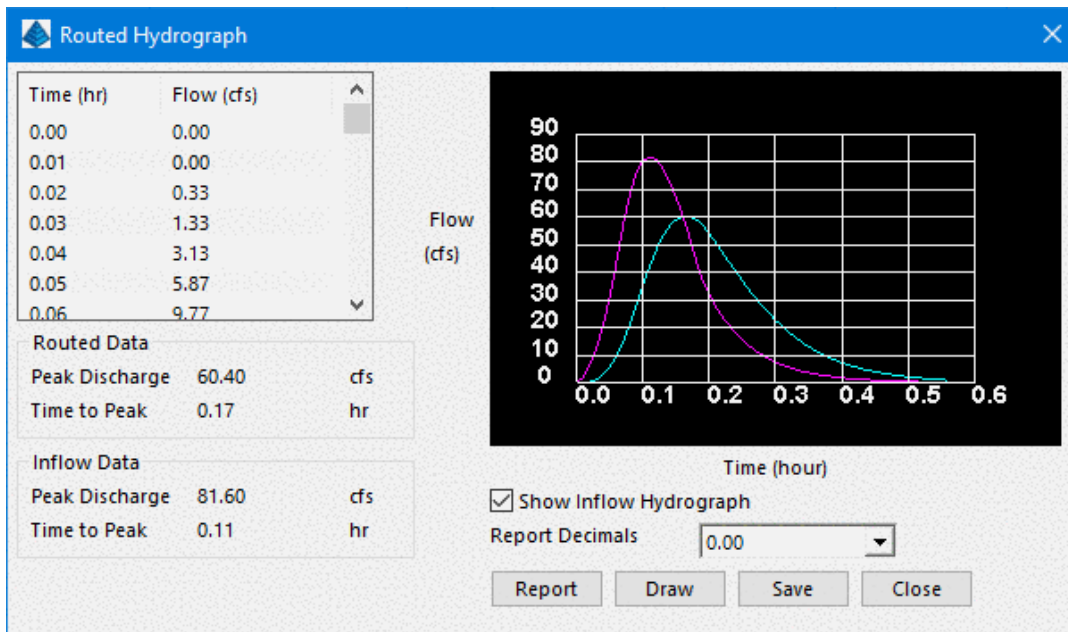
Please refer to the National Engineering Handbook Section 4 for the detail of the Convex method.

Select Convex Method from Watershed > Hydrograph Routing > Channel Routing menu. Specify the inflow hydrograph and enter the routing coefficient C or channel flow velocity to let the program to calculate the coefficient. The routing interval is the same as the time increment of the inflow hydrograph.

After entering all information, click on Routed Hydrograph button to generate the routing hydrograph. A dialog is opened with the tabular and graphic hydrograph data, from there you can draw the hydrograph on screen and save it to a file.



Channel Routing Hydrograph - Convex Method



Routed Hydrograph Dialog

Pulldown Menu Location: Watershed > Hydrograph Routing> Channel Routing > Convex Method

Keyboard Command: chanrout

Prerequisite: None

Channel Routing Hydrograph - Modified Att-Kin Method

Channel routing is used to model the attenuation of a hydrograph's peak flow rate as it travels along a channel. Attenuation occurs due to the inherit storage capacity of a channel when the capacity of the channel is greater than the flow capacity. .

The Modified Att-Kin (attenuation-kinematic) method is used in the TR-20 and TR55 methodologies. It is a combination of the storage indication method and the kinematic wave method to reflect the reservoir and translation effects on natural flood waves. The hydrograph is first routed through a reservoir assuming that the outflow is proportional to the storage. The resulting outflow hydrograph is then translated using kinematic wave principles. The storage routing portion allows for attenuation and translation. Kinematic routing translates the outflow hydrograph but doesn't attenuate the peak.

The continuity equation is

$$I_i - (O_{i+1} - O_i) / 2 = (S_{i+1} - S_i) / \Delta t$$

where: I_i = Inflow hydrograph discharge at (i)th time step

O_i = Outflow hydrograph discharge at (i)th time step

O_{i+1} = Outflow hydrograph discharge at (i+1)th time step

S_i = Storage at (i)th time step

S_{i+1} = Storage at (i+1)th time step

Δt = Routing time step

Substitution of $S = KO$ leads to

$$O_{i+1} = C_m I_i + (1 - C_m) O_i$$

where: $C_m = 2\Delta t / (2K + \Delta t)$

$K = L / (60mV)$

$V = Q / A$

$A = (Q / x)^{1/m}$

where: L = Channel length (ft or meter)

V = Average velocity in the channel (ft/s or m/s)

Q = Average channel flow, default value is the peak discharge of the inflow hydrograph (cfs or cms)

A = Wet cross section area of the channel (sq. ft or sq. meter)

x = Flow equation multiplier, > 0.0

m = Flow equation exponent, default value is 5/3, usually is $1.0 \leq m \leq 2.8$

The coefficient x can be estimated using Manning's Formula:

$$x = S^{1/2} / nP^{2/3}$$

where: S = Average slope of the channel

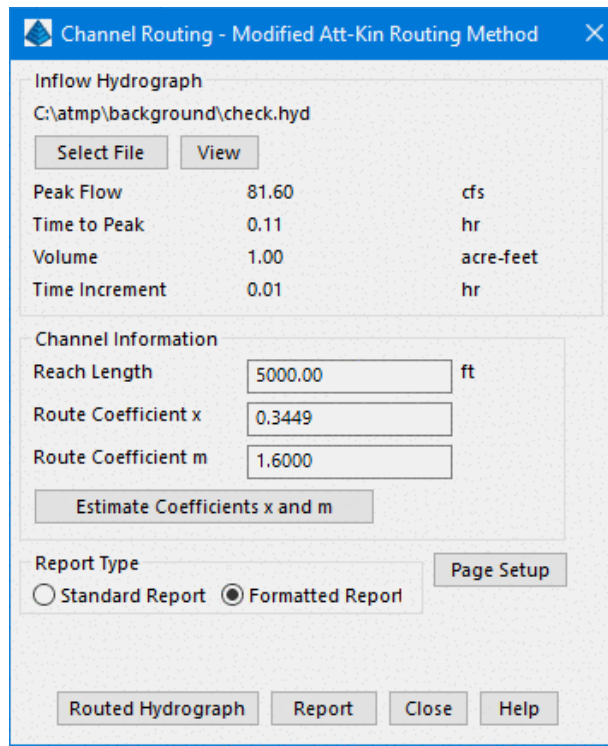
n = Manning's n value

P = Wetted perimeter of the flow (ft or m)

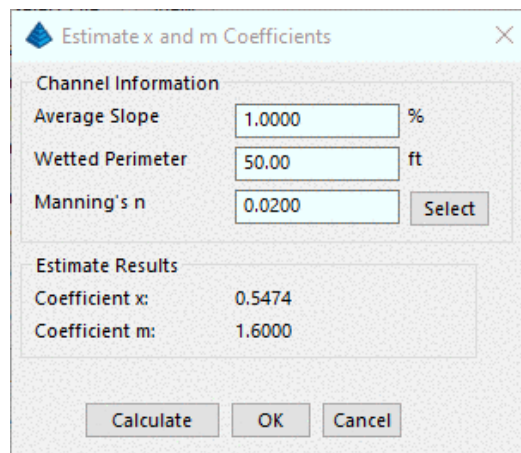
Please refer to the TR-66 for the detail of the Modified Att-Kin method.

Select Modified Att-Kin Method from Watershed > Hydrograph Routing > Channel Routing menu. Specify the inflow hydrograph and enter the channel length, the routing coefficient x and the time coefficient m. The routing interval is as same as the time increment of the inflow hydrograph. Button Estimate Coefficients x and m allows you to enter the parameters of the Channel to determine the x and m coefficients. Once the parameters are entered, click on the Calculate button to compute the coefficients. OK button accepts the values and proceeds to the previous dialog.

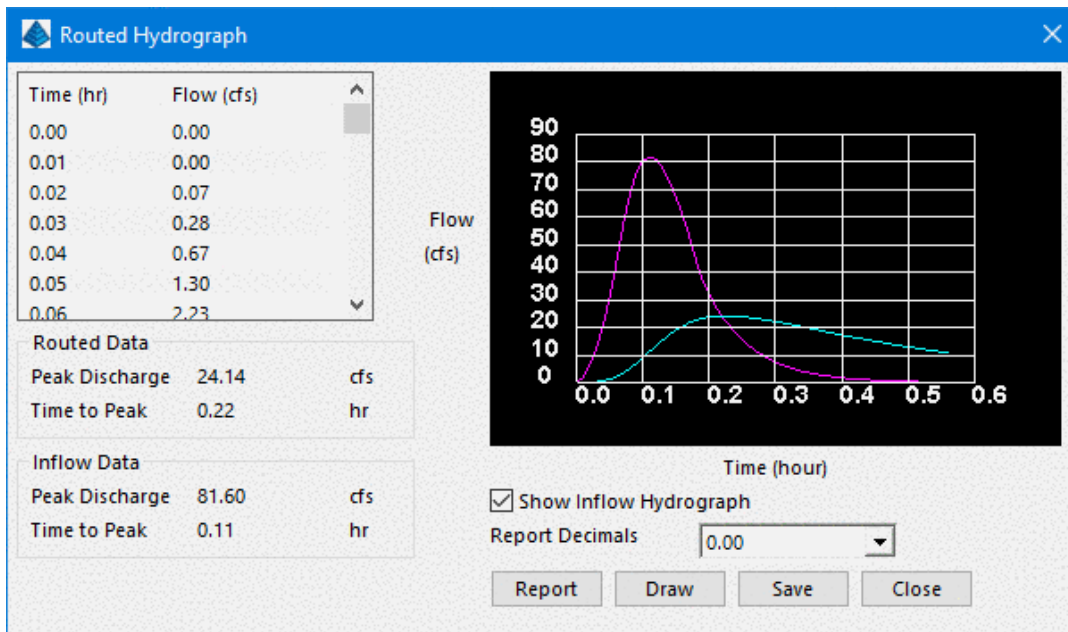
After entering all information, click Routed Hydrograph button to generate the routing hydrograph. A dialog is opened with the tabular and graphic hydrograph data, from there you can draw the hydrograph on screen and save it to a file.



Channel Routing Hydrograph - Modified Att-Kin Method



Channel Routing Hydrograph - Estimate Coefficients Dialog



Routed Hydrograph Dialog

Pulldown Menu Location: Watershed > Hydrograph Routing > Channel Routing > Modified Att-Kin Method

Keyboard Command: chanrout2

Prerequisite: None

Draw Flow Polylines TR20

Locate Structures TR20

This command is no longer being supported. Running the command will produce the message "Program is removed as obsolete. Please use NRCS WinTR20."

Locate Reach

This command has been removed

Pulldown Menu Location: Watershed > TR-20 Routing

Keyboard Command: ttreach

Edit Layout Element

This command is no longer in use.

Hydrograph Development

This command is no longer being supported. Running the command will produce the message "Program is removed as obsolete. Please use NRCS WinTR20."

Single Runoff Hydrograph

Program is removed as obsolete. Please use NRCS WinTR20

Add Hydrographs

This function takes two hydrograph files, .hyd and or .h1 format, and produces a new hydrograph file that is a combination of the two. Please note if the peak rates in the hydrographs do not occur at the same time step, the

resulting peak will be less than if the two peak rates were added together.

This is useful when a combined hydrograph is needed when two existing hydrographs already exist. They can either be Carlson format time,flow rate or TR-20 format which has a file extension of .h1. The program adds the two sets of data as they would combine when routed together in a storm water system.

Please note that the time interval used in the output hydrograph is the smaller of the two hydrographs time steps if they are not equal. However, the units for time and discharge rate of each input hydrograph must match. The command is completed by selecting a name and location for the combined hydrograph using the provided file dialogue.

Pulldown Menu Location: Watershed > Hydrograph Utilities

Keyboard Command: mergehyd

Prerequisite: hydrograph files

Subtract Hydrographs

This function subtracts two hydrographs and writes the new hydrograph to a file.

Pulldown Menu Location: Watershed > Hydrograph Utilities

Keyboard Command: subtracthyd

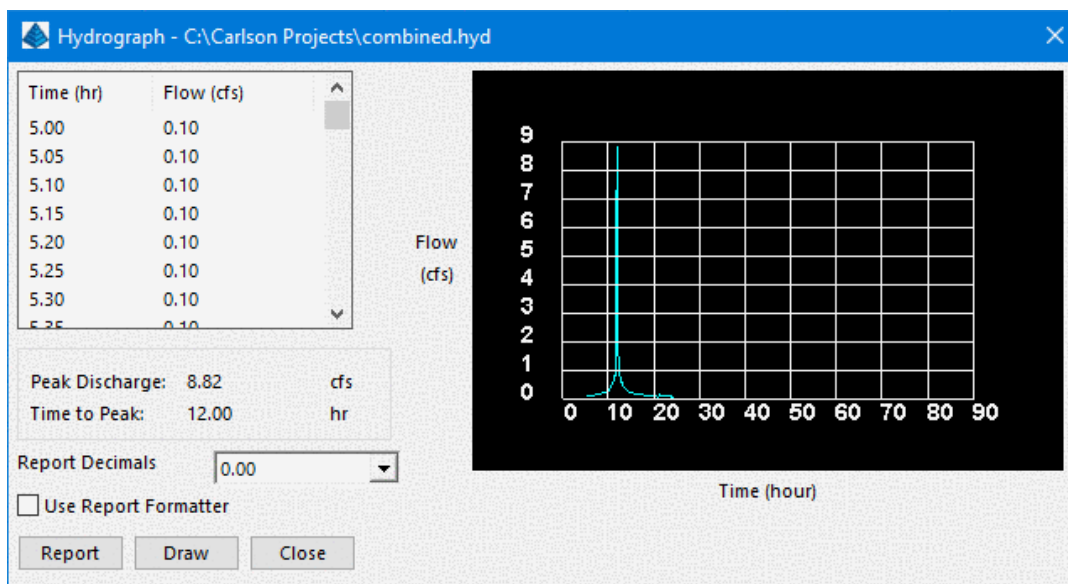
Prerequisite: hydrograph files

Report Hydrograph

This command reports a hydrograph from a hydrograph file with the extension of:

- .h1 - *.h1 files are created by SEDCAD, Hydrograph Development, or the Single Runoff Hydrograph commands.
- .hyd - *.hyd files are created by all commands under Watershed > Hydrograph Routing menu and sewer network hydrograph routing programs.

After specifying the desired hydrograph file, the following dialog box is presented:



Hydrograph Values and Graphical Representation: The Time vs. Flow values for the hydrograph are displayed in a list and a graphical representation of the hydrograph are provided.

Peak Flow and Time: The Time to Peak (T_p) and Peak Discharge (Q_p) are provided.

Report Decimals: Indicate the desired level of Precision to be displayed in the *Text Report*.

Use Report Formatter: When enabled, this control displays the *Report Formatter Options* dialog box that gives you greater control over the content and type of report you would like to produce.

Report: Displays a hydrograph report showing Time vs. Flow values.

Draw: When clicked, a graphical representation of the hydrograph will be placed into the drawing through the functionality found in the Draw Hydrograph command.

Pulldown Menu Location: Watershed

Keyboard Command: viewhyd

Prerequisite: A hydrograph file

Draw Hydrograph

SEDCAD

Civil Software Design is the author of SEDCAD, which is sold separately from Carlson. SEDCAD is a comprehensive hydrology and sedimentology package, useful for all varieties of runoff and sediment control design calculations. SEDCAD can be run directly from the Carlson Hydrology menu. The directory where SEDCAD is installed must be defined in the Carlson Configure command.

Pulldown Menu Location: Watershed > SEDCAD Structure Layout

Keyboard Command: sedcad

Prerequisite: Installed and licensed version of SEDCAD

SEDCAD Draw Flow Polylines

This command draws polylines in the SEDCAD layer that represent flow lines. When drawing a network of flow lines, first draw the main branch. Then begin drawing the other flow lines from the top of flow and use the Join option to connect onto the main branch. Draw Flow Polylines is the first command in a series that produce the Junction, Branch, and Structure labels for SEDCAD.

Prompts

End/Pick point: *pick a point*

Undo/End/Join/Pick point: *pick a point*

Undo/End/Join/Pick point: *pick a point*

Undo/End/Join/Pick point: *press Enter*

Draw another flow polyline (<Yes>/No)? *press Enter*

End/Pick point: *pick a point*

Undo/End/Join/Pick point: *pick a point*

Undo/End/Join/Pick point: *Join*

Select flow polyline at place to join: *pick the main branch at the junction*

Draw another flow polyline (<Yes>/No)? *No*

Pulldown Menu Location: Watershed > SEDCAD Structure Layout

Keyboard Command: sedcad1

Prerequisite: None

SEDCAD Locate Structures

This command is the second step for creating the SEDCAD layout. Locate Structures places triangle symbols on flow polylines that represents structures for SEDCAD.

Prompts

Symbol size <4.0>: *press Enter*

Pick location on flow polyline for structure: *pick a point on a polyline*

Pick location on flow polyline for structure: *pick a point on a polyline*

Pulldown Menu Location: Watershed > SEDCAD Structure Layout

Keyboard Command: sedcad2

Prerequisite: flow polylines

SEDCAD Label Structure Layout

This command is the third and final step for creating the SEDCAD layout. Label Structure Layout draws text labels for the junctions, branches, and structures in the network. A junction, branch, and structure report is also generated. Flow polylines and structure symbols must be drawn before running this routine. This command uses the labeling rules as described in the SEDCAD manual.

Prompts

Symbol size <4.0>: *press Enter*

Junction offset tolerance <10.0>: *press Enter* Flow lines that meet the main branch within this distance of each other are considered the same junction.

Select flow polylines and structure symbols.

Select objects: *pick the polylines and symbols*

J5,B1,S1

J4,B2,S1

J4,B1,S1

J3,B2,S1

J3,B1,S2,S1

J2,B2,S1

J2,B1,S2,S1

J2,B3,S1

J2,B1

J1,B2,S1

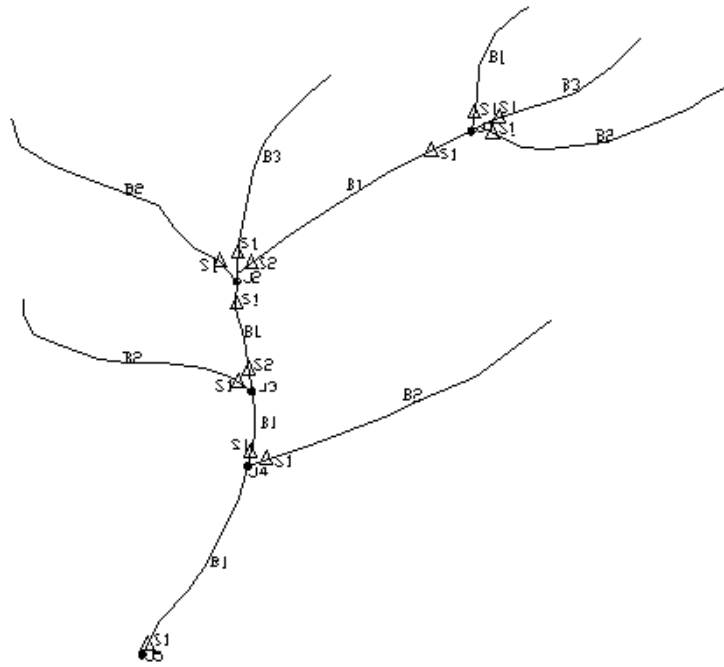
J1,B3,S1

J1,B4,S1

J1,B1

Write report to file (Yes/<No>)? *press Enter*

Write report to printer (Yes/<No>)? *press Enter*



Example of labeled SEDCAD structure layout

Pulldown Menu Location: Watershed > SEDCAD Structure Layout

Keyboard Command: sedcad3

Prerequisite: flow polylines and structure symbols

HEC-RAS Manager

Hydrologic Engineering Center's (CEIWR-HEC) River Analysis System, also known as HEC-RAS is a computer program developed by the US Army Corps of Engineers to perform one-dimensional steady flow, one and two-dimensional unsteady flow calculations, sediment transport/mobile bed computations, and water temperature/water quality modeling for major rivers and streams. Although the software has many features for modeling natural and man made channels, it does not have a CAD platform that enables automated channel geometry input. HEC-RAS Manager works with HEC-RAS to provide that additional functionality.

HEC-RAS Manager interfaces directly with HEC-RAS to provide needed surface, reach and cross-section alignments needed by HEC-RAS to perform hydraulic calculations. The advantage of this method is final results are calculated entirely by HEC-RAS so that there are no compatibility issues when submitting flood studies as part of FEMA's Flood Insurance Program. Although HEC-RAS is capable of performing unsteady state, 2 dimensional analysis, the Manager was designed for Steady State, one dimensional studies which is the most common application of the software. The latest version of HEC-RAS software can be obtained from Hydrologic Engineering Center <http://www.hec.usace.army.mil/software/hec-ras/>.

However, the Manager cannot support the most current version of HEC-RAS. It is recommended that version 6.0 be obtained from the Corps Github page.

<https://github.com/HydrologicEngineeringCenter/hec-downloads/releases>

HEC-RAS projects require a surface (tin), a Reach Alignment file (mxs) and a Reach Section file (xsct) in order to produce cross section geometry used by HEC-RAS. Additionally, left and right bank location at each cross section can be determined from one of the following options.

1. Left and Right Bank are calculated by specifying the percentage of flow that occurs in the center of the channel.
2. Left and Right Bank are specified within the section file as point descriptions.
3. Left and Right Bank are determined by polylines that represent left and right bank.

to create or edit the .mxs Reach Alignment file.

To create cross sections for HEC-RAS, the following items are needed.

1. A tin file representing terrain within the study area.
2. A polyline representing the center of channel/stream, drawn from downstream to upstream.
3. If applicable, two polylines representing right and left bank of the channel/stream.

Using these items, create a Reach alignment file(.mxs) with the stream centerline and create a Reach Section file to establish cross section width, spacing and station spacing. The stream centerline file is used to create the section alignment using the command **editmxs** and cross sections are created using the command **setgrid**. For additional information on creating these files, see the chapter on creating Road Sections in the Help files for the CIVIL module.

Select the "Create/Update Project" and specify the .MXS and .XSCT created in the previous step. Specify Steady State Flow Rates, set left, channel and right bank Mannings values and name the River and Reach Name.

Next specify the Upstream and Downstream Boundary Condition type. These values are used to estimate flow condition both upstream and downstream of the Study Area. The following are typical boundary condition methods and their use.

1. **None.** Note that None is only a place holder and will not result in a functioning model.
2. **Known Water Surface.** This is the most common Upstream/Downstream method when performing analysis on an existing Studied area. A good strategy in this situation is to start and end your study area using an Existing FEMA Cross Section and known water surface elevation.
3. **Normal Depth.** Normal Depth should only be used as a downstream boundary condition for an open ended Reach. Depending on conditions, either provide the slope of the channel or friction slope just downstream of the study area.
4. **Critical Depth.** Critical Depth should be used when there is instability in the model and can be used as an upstream or downstream condition.
5. **Rating Curve.** Rating Curves represent stage/discharge table values and are not often used for Steady State studies. Use this method when upstream or downstream stage/discharge is non linear.

In all cases, locate the starting and ending cross section far enough away from the area of study to reduce error in the model.

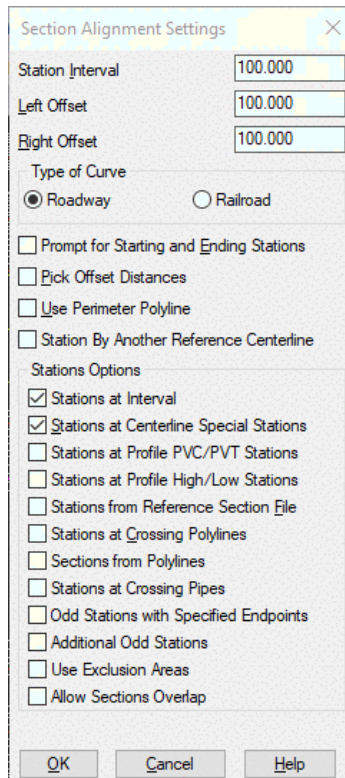
Example HEC-RAS Flood Study

The following must be obtained prior to performing HEC-RAS model calculations.

1. Tin File Digital Terrain Model representing elevation data within the area of study.
2. Centerline polyline representing the centerline of the stream/channel.
3. Reach Alignment file, .mxr, representing stream centerline and station locations. Use the command "EDITMXS" to specify the center line alignment and Cross Section spacing.
4. Reach Section File created using the command "SCTGRID". Use the alignment file created in step 3.
5. Determine left, center and right Mannings Runoff Coefficient for channel. If an existing flood study is available, use the n values used in that study if conditions match.
6. Polylines representing Left and Right Bank, drawn in the same direction as the Reach.
7. Upstream/Downstream Boundary Condition Method and Values.
8. Channel Discharge Rate(s) by Station. For FEMA Limited and Detailed Studies, flow rate will be located within the County's FIS Report. For Existing Non-FEMA Flood Models, use discharge rates within the model under Steady State Conditions. For Unstudied Areas with large drainage areas, USGS Regression Equations can be used once drainage area is determined. For Small Drainage Areas, calculate design flow using the Rational or Curve Number Methods.

Channel Geometry Modeling

With the Section Alignment File, issue the command **sectgrid** and specify the correct .tin and .mxs files to create the .xsct Reach Section file. Also specify Station Interval, Left and Right Offset Distance and select OK. Use the command **scto3dp** to draw the Reach Section file. Remove any cross sections that do not represent the entire channel cross section.



Input-Edit Section Alignment

New File C:\Users\scummings\Downloads\darryll

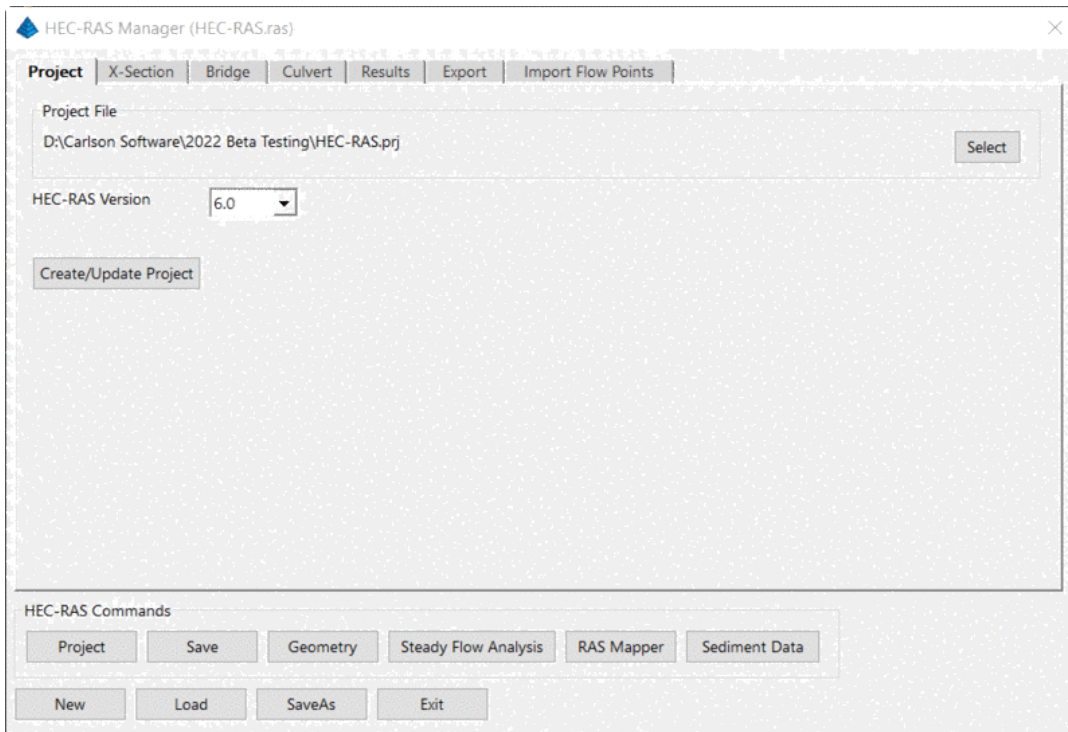
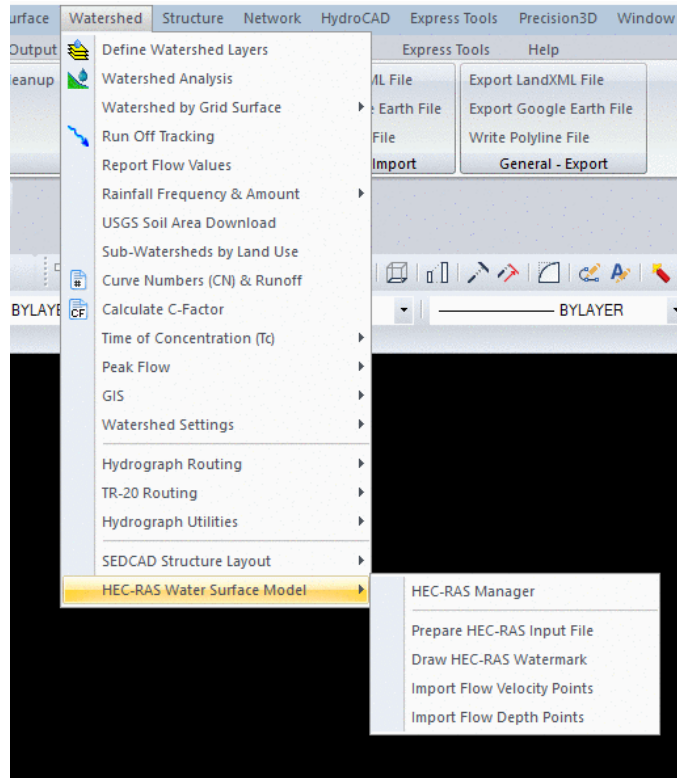
Station	Left Offset	Right Offset	Interval
0.000-2400.000	100.000	100.000	100.000
24.197	100.000	100.000	--
48.202	100.000	100.000	--
66.114	100.000	100.000	--
77.077	100.000	100.000	--
105.337	100.000	100.000	--
240.039	100.000	100.000	--
271.904	100.000	100.000	--
275.502	100.000	100.000	--
304.344	100.000	100.000	--
311.320	100.000	100.000	--
314.522	100.000	100.000	--
344.542	100.000	100.000	--
359.809	100.000	100.000	--
366.286	100.000	100.000	--
506.707	100.000	100.000	--
519.207	100.000	100.000	--
538.669	100.000	100.000	--
563.185	100.000	100.000	--

Edit Add Delete Save SaveAs Exit Help

HEC-RAS Model Creation

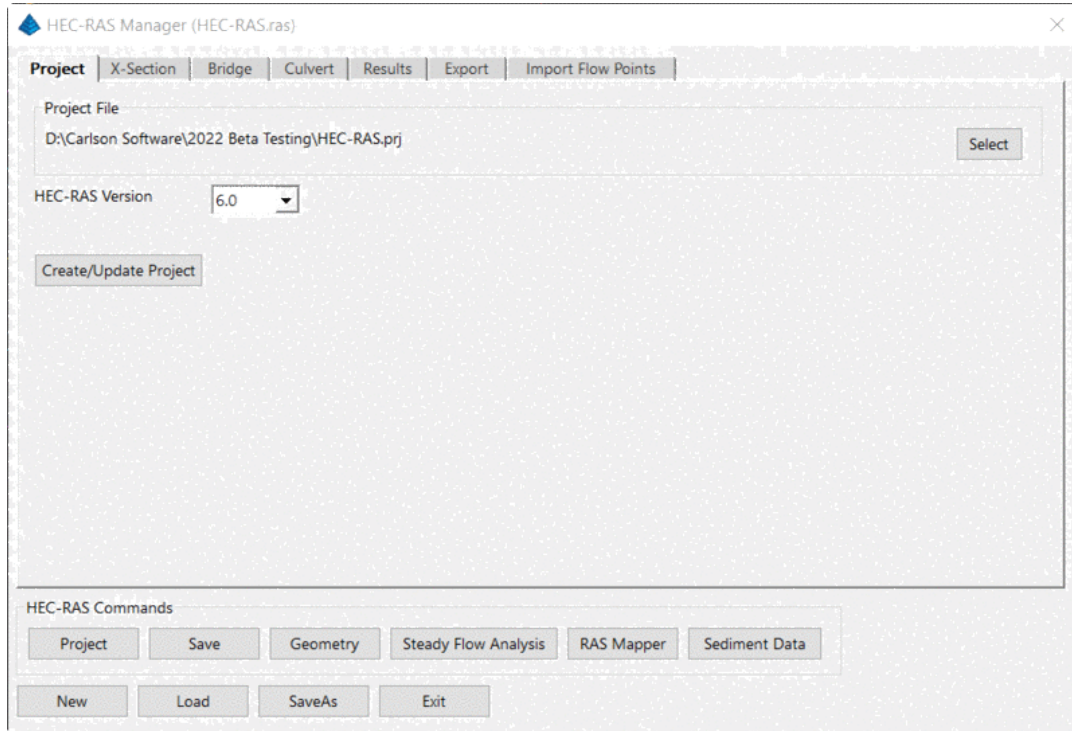
To start the HEC-RAS Manager, run the command **hecrcas** which may be typed in the Command Ribbon or found

in the Hydrology module, Watershed dropdown, HEC-RAS Surface Model, HEC-RAS Manager.



Specify a Project File Name. Please note that HEC-RAS project files use the same file extension as projection files for shape files.(.prj)

Specify the version of HEC-RAS installed on your system. Please note that you may be required to uninstall the latest version of HEC-RAS if it is a later version than 6.0. Currently, version 6.0 is the newest version of HEC-RAS that will work with this tool. This version is available at <https://github.com/HydrologicEngineeringCenter/hecdownloads/releases>



Select the **Create/Update Project** button to update an existing project or to create a new one. When clicked the Create HEC-RAS Project dialog appears. Use the Set buttons to assign the Reach Alignment file, the Reach Section file and the HEC-RAS project name. Fill in the River Name and the Reach Name. Use the Set buttons to assign the Manning's friction coefficient (n) to the banks and bottom of the channel. Choose the Bank Position Method, which determines where the bank is located. Choices are Percentage of Width, Section Description or Bank Polylines. If one or more Steady State Flows need to be included, click the Add button below the box and add Station and Flow. Once added, you may edit or delete these by highlighting the flow and clicking the appropriate button below.

Assign Boundary Conditions

Select the Upstream and Downstream Boundary Condition method and assign values. The following are the available Boundary Conditions and their typical application.

None represents a place holder for one of the following Boundary Conditions. Using "None" as a boundary condition will cause an error. .

Known Depth is the best boundary condition to use when trying to duplicate an existing FEMA Flood Study or when depth of flow is known by other means.

Critical Depth occurs when flow in a channel has a minimum specific energy and a Froude number of 1. It represents the inflection point between super and sub-critical flow. Critical Depth is a good boundary condition for the following.

1. The supercritical flow regime has been specified.
2. The calculation of critical depth has been requested by the user.
3. This is an external boundary cross section and critical depth must be determined to ensure the user entered boundary condition is in the correct flow regime.
4. The Froude number check for a subcritical profile indicates that critical depth needs to be determined to verify the flow regime associated with the balanced elevation.
5. The program could not balance the energy equation within the specified tolerance before reaching the maximum number of iterations.

For additional information on using Critical Depth for Boundary Conditions, see HEC-RAS documentation.

Normal Depth is the depth of flow in a channel when the slope of the water surface and channel bottom is the same and the water depth remains constant along the Reach. Normal depth occurs when gravitational force of the water is equal to the friction drag along the channel and there is no acceleration of flow. Normal depth is only used as a downstream boundary condition for an open ended reach. To use the Normal Depth Option, the slope of the

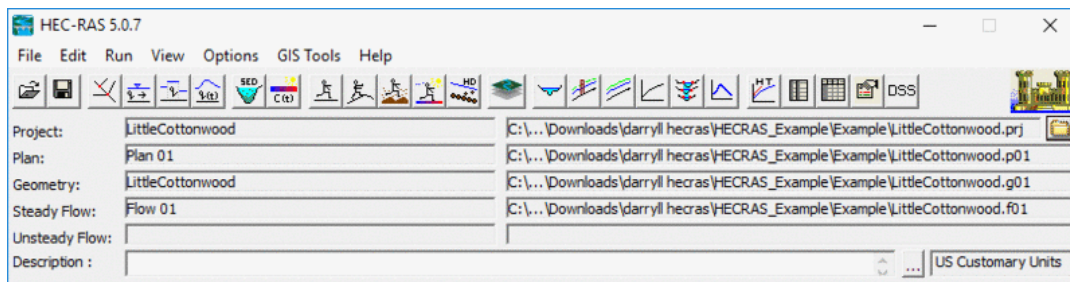
energy grade line must be entered and can be assumed to be the slope of the water surface near the end of the Reach. Another good value to use is the slope of the channel near the downstream boundary. It is recommended that if this option is used the most downstream section of the reach should be beyond the area of study. A good rule of thumb is to have at least two cross sections below the study area to minimize any error in the estimated slope.

Rating Curve option assigns a stage-discharge relationship either at the beginning or end of the study area. An example of when a downstream Rating Curve can be used is when the outfall of the system is a detention pond or other structure where stage-discharge cannot be determined by channel geometry. For upstream conditions, a common rating curve represents calculated depths of upstream flow for each storm event such as the 50 and 100 year event storm events. By using a rating curve, upstream boundary flow condition will adjust according to discharge rate.

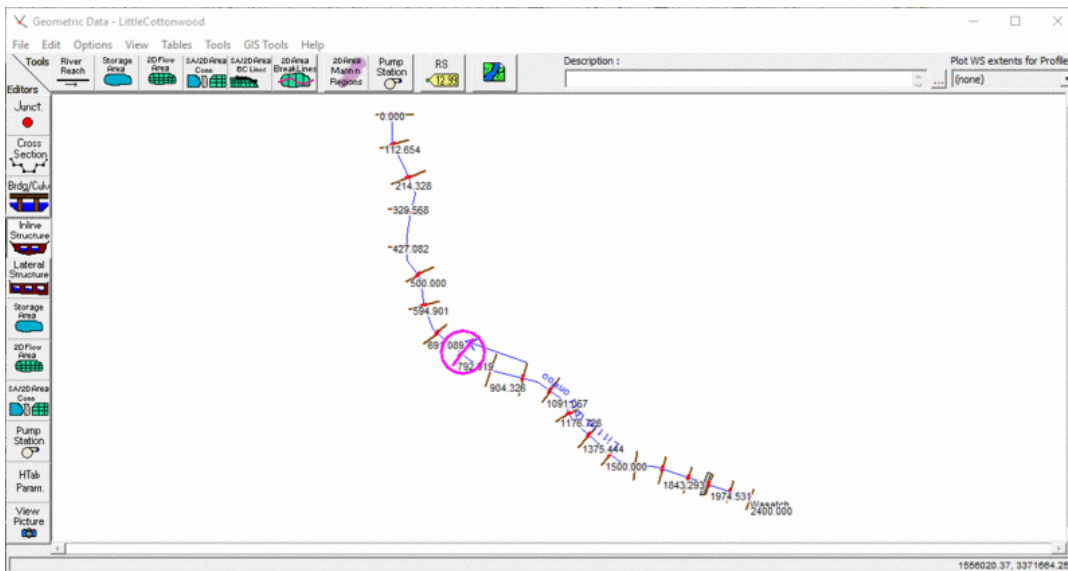
Export HEC-RAS Model

Select the Export Button which will create the minimum inputs needed to create a HEC-RAS project. The HEC-RAS project file .prj, plan file .p01, geometry file .g01 and a steady flow .f01 files are created. These are the minimum files needed for a HEC-RAS project. The user is returned to the HEC-RAS Project Manager dialog after the Export.

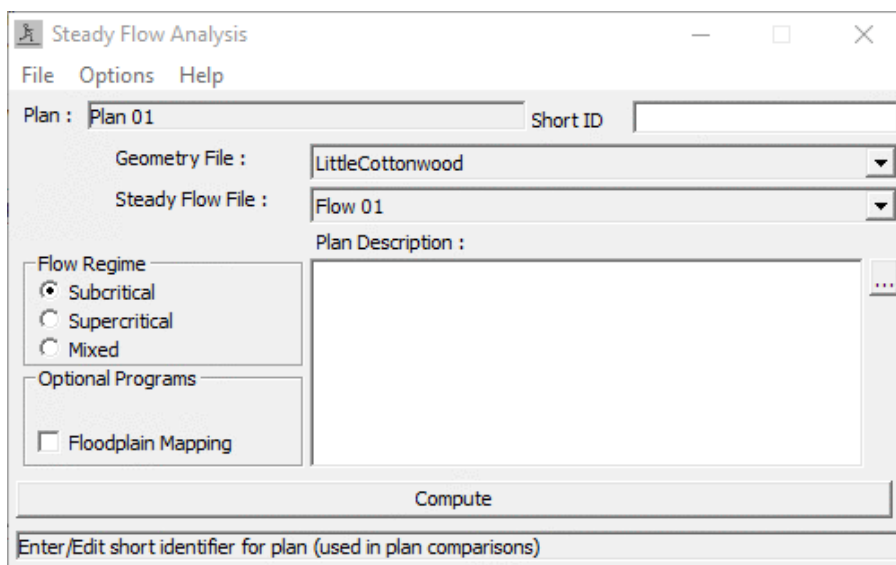
Selecting the **Project** button calls the main HEC-RAS project screen, allowing for viewing, editing and calculation of the Model.



Selecting the **Geometry** button calls the HEC-RAS Geometric Data dialog, with tools and editors to add complexity to the reach.

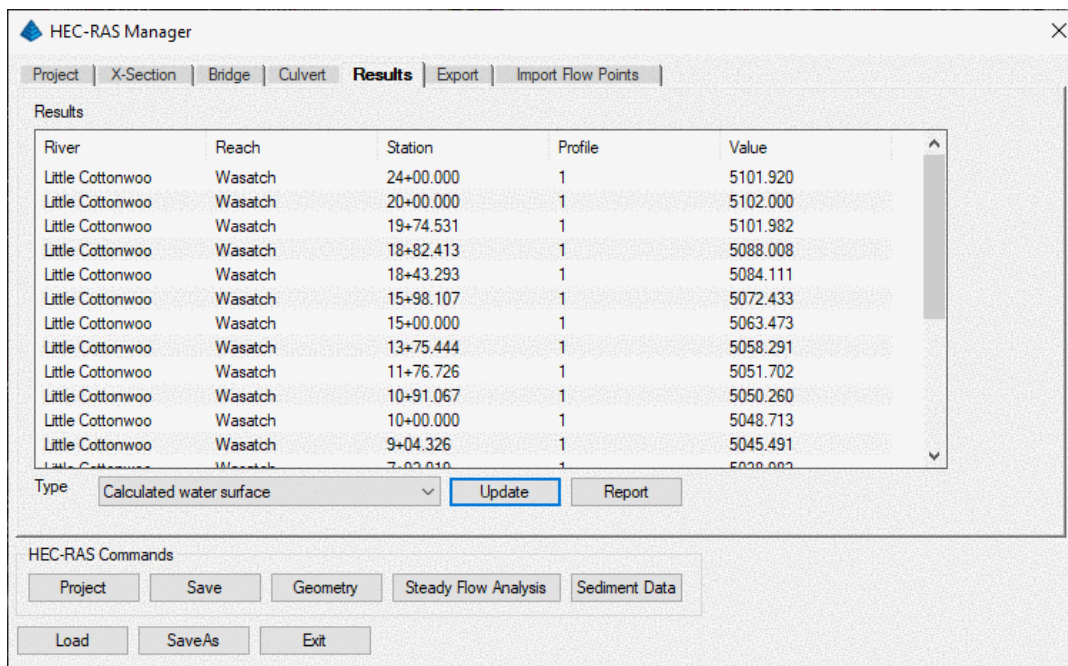


Selecting the **Steady Flow Analysis** button calls the HEC-RAS Steady Flow Analysis dialog. Select the type of Flow Regime to model and check the Floodplain Mapping option if needed. Click the **Compute** button to analyze the model.



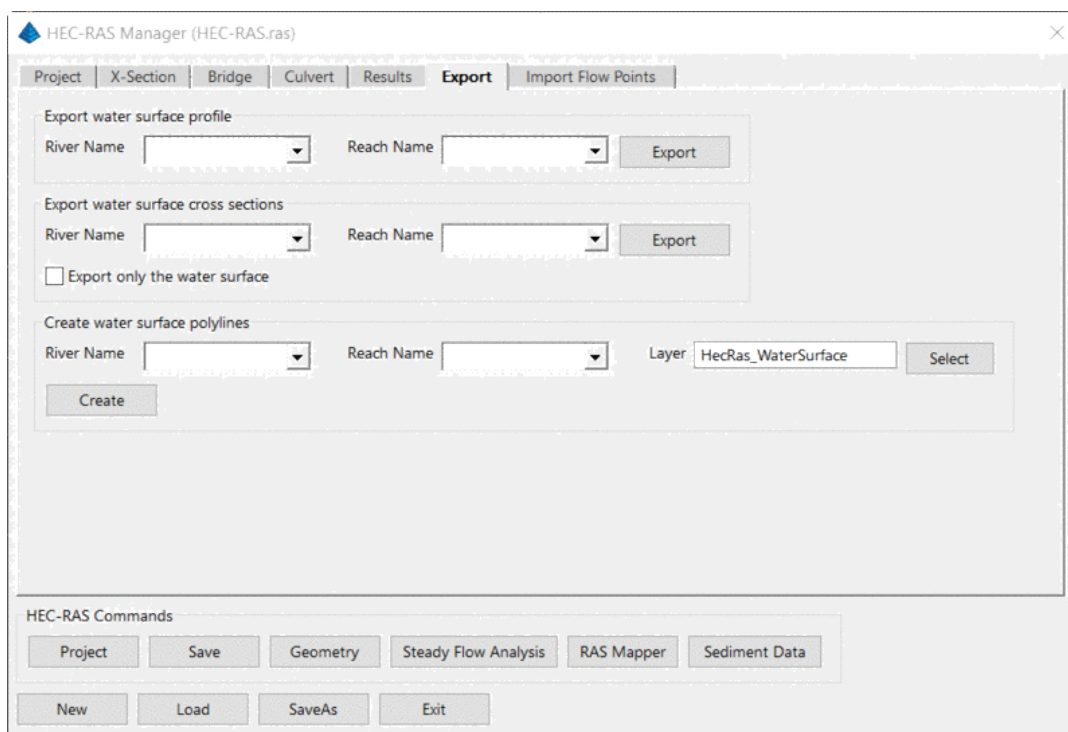
Once the computations are completed, the HEC-RAS Finished Computations dialog is displayed or errors are listed.

Viewing Results In addition to the reports available from HEC-RAS, the HEC-RAS Manager can produce tables representing the following results.



To view Results such as the Calculated water surface for each cross section, select this option for the type of results and select the Update button to show water surface elevations for each cross section. To create a report, select the Report Button. Make sure that the current model has been calculated before creating Reports.

Export Options



Export water surface profile.

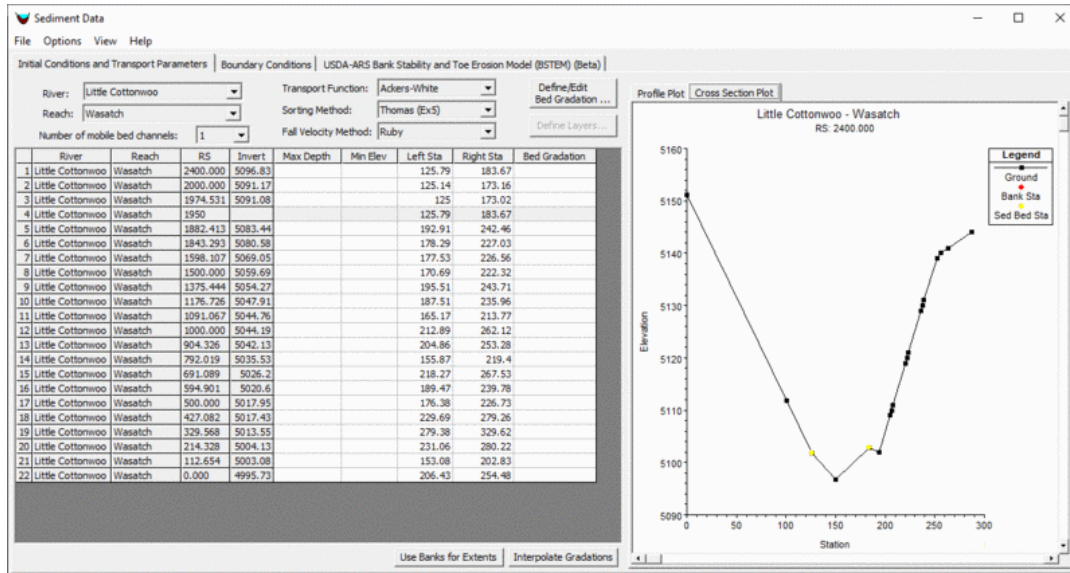
This option will export the water surface profile for the selected River/Reach. A profile file(.pro) will be created

which can be used to plot Station/WSELV using Carlson's Draw Profile Copmmand. **Export water surface cross sections.**

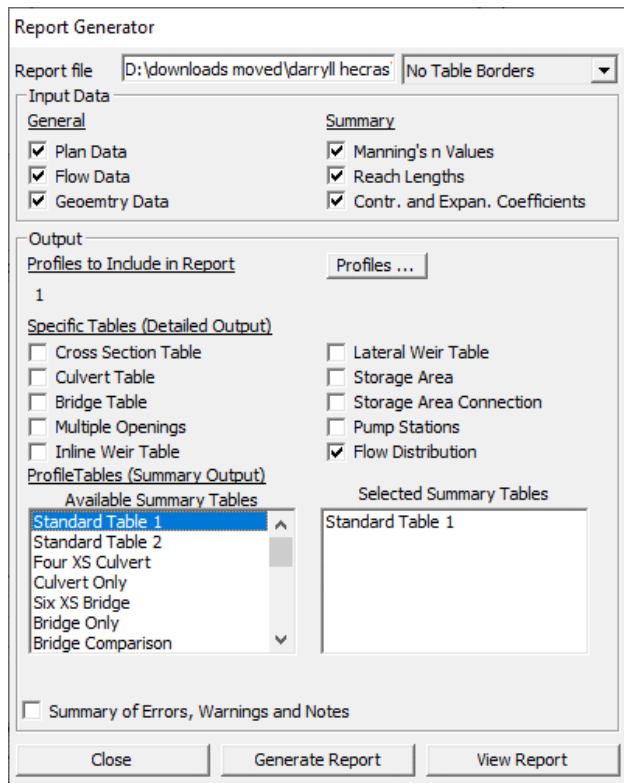
This option will export the water surface in cross section view for the selected River/Reach. A cross section file (.xsct) will be created which can be drawn using the command **drawxsct**. **Create water surface polylines.**

This option will create polylines representing the left and right edge of flood inudation. These polylines will be drawn in the current cad file.

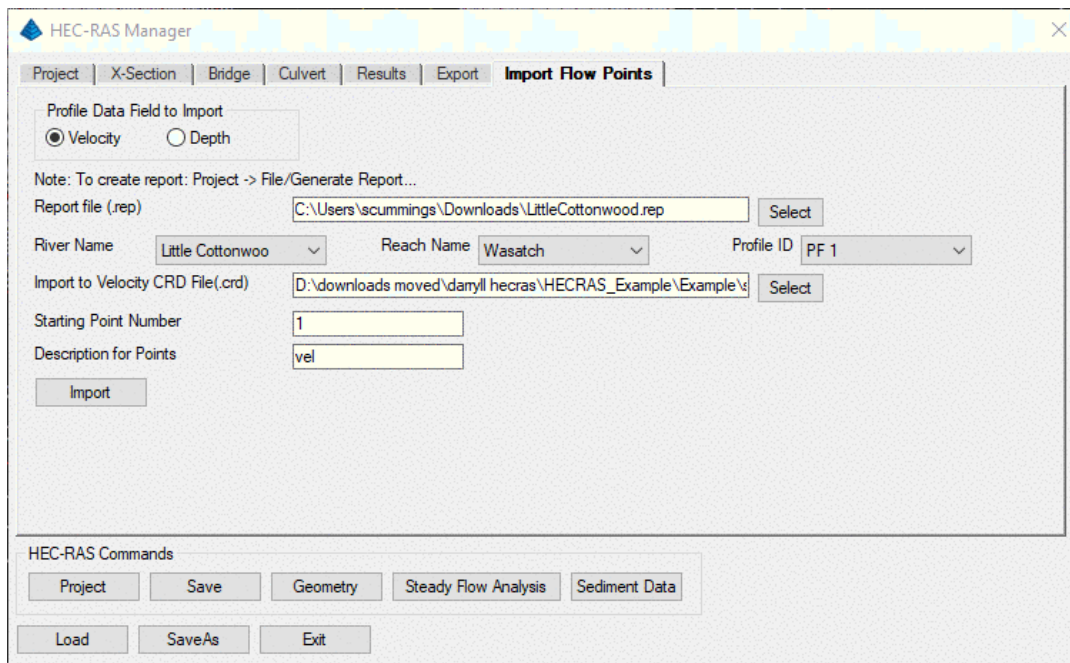
Advanced Options In addition to Hydraulic Modeling, HEC-RAS is capable of performing sediment transport operations as shown below. Please see HEC-RAS documentation for additional details.

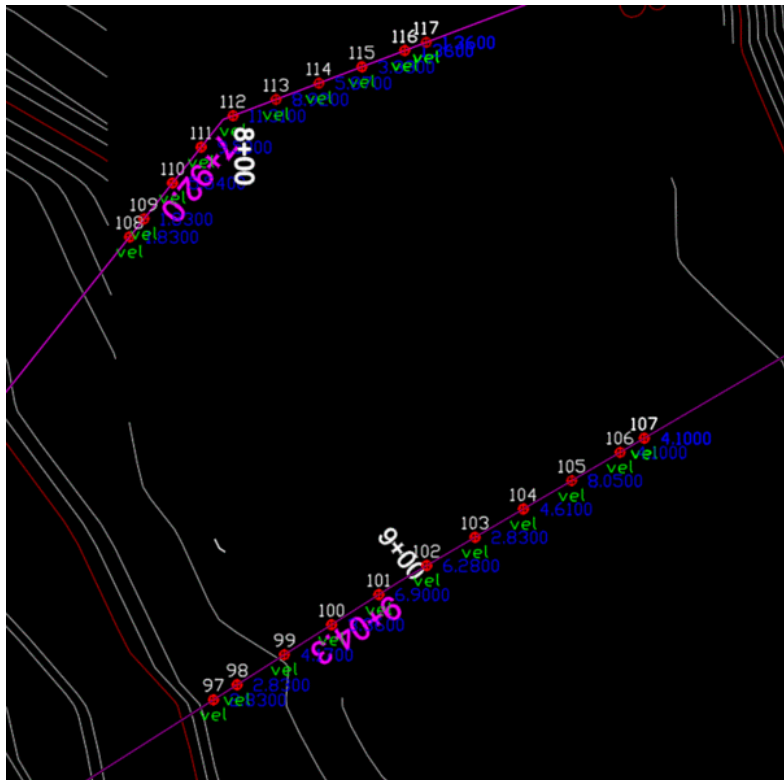


The HEC-RAS **Export Flow Points** Command is used to Export flow or velocity points from the HEC-RAS geometric model to a Carlson CRD file for use in the drawing. To prepare, select the Project tab, then select "Generate Report" found under the HEC_RAS project File dropdown. Set the report up as shown below, naming the file, choosing Profiles and clicking the "Generate Report" button.



Importing Flow Points to HEC-RAS Manager Select the **Import Flow Points** Tab and specify import Velocity or Depth. Select the previously generated Report file just generated. The River Name, Reach Name and Profile ID should appear in the dropdowns. Pick the River, Reach and Profile you want to extract. Select the filename and location of the CRD file that will contain the flow points. Starting Point Number and a Description for Points. Click the "Import" button to write the points into the drawing's CRD file. **Draw-Locate Points.**





Pulldown Menu Location: Watershed
Keyboard Command: HECRAS
Prerequisite: HEC-RAS v. 6.0.0 or newer

Prepare HEC-RAS Input File

HEC-RAS is designed to perform one-dimensional hydraulic calculations for a full network of natural and constructed channels. It could be considered to be an advanced Windows-based version of the HEC-2 program. This program makes it easier for CADD and GIS systems to import their data directly for river network analysis. It is also very convenient because the output from the program can be exported directly to CADD programs where this data can be used to create water surface models for inundation mapping. The software is free and can be downloaded from <http://www.hec.usace.army.mil/software/hec-ras/>.

In the program of Prepare HEC-RAS Input File, every channel is defined by a cross-section file (with or without multiple skews) and a MXS file (please see the material on Sections in Chapter 6 of this manual). This program reads cross-section files and the corresponding MXS files, and then creates a HEC-RAS geometry data file.

Data Format

HEC-RAS input files consist of three data sections:

- * A header, containing data relevant to all sections of the data in the file.
- * A description of the stream network, containing reach locations and connectivity.
- * A description of the model cross-sections, containing cross-section location and geometric data as well as additional HEC-RAS modeling information.

The header information is mainly for the purpose of identifying the project and is mostly not used by the program. The only important information needed by the program is the "Units" section and the value must be "ENGLISH" or "METRIC".

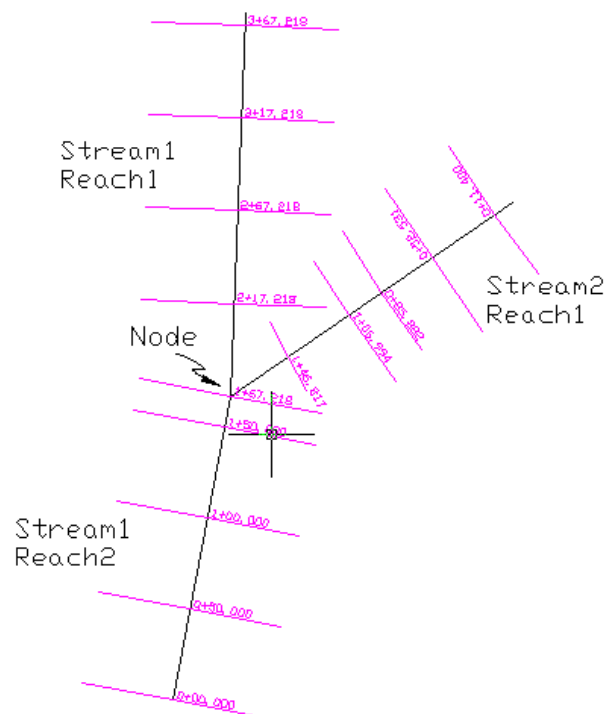
The network is modeled as a set of interconnected streams. Each stream is a set of interconnected reaches. Each reach, hence, **MUST** have a unique Stream ID and Reach ID.

The Stream Network section contains a series of Point Numbers and the corresponding coordinates. In addition, this section has information pertaining to each Reach. For each Reach, the following information is provided:

- * Stream ID and Reach ID. These are 16 character alphanumeric strings. Together these two items uniquely identify a Reach.
- * Starting (FROM or upstream) point and ending (TO or downstream) point of the Reach. The FROM point and TO point here are given by their Point Numbers, as identified above.
- * The coordinates on the Centerline of the Reach, starting with the FROM point coordinates and ending with the TO point coordinates.

The Cross-Sections portion of the input file contains data describing the geometric properties at each cross section in the network. The following information is provided at each Cross-Section:

- * Stream and Reach ID, to identify which Reach the Cross-Section is on.
- * Station, position of the Cross-Section, relative to the Stream. The Station is taken as the distance from the current station to the end of the stream. For this purpose, the stream **MUST** be drawn Downstream to Upstream. **THIS IS THE MOST FUNDAMENTAL REQUIREMENT OF THE PROGRAM.** If the Stream is drawn in the other direction, then, it must be reversed using the command *Reverse Polyline* under *Edit>Polyline Utilities*
- * Cut Line: Series of point coordinates, identifying the surface line of the Cross-Section. HEC-RAS identifies the cross-sections as going from left to right as seen from upstream to downstream. The user only needs to make sure that the stream network is drawn in the right direction (downstream to upstream); all other conventions are taken care of by the program.



Modeling Guidelines

Some additional guidelines in drawing the river network in the CAD so as to model correctly for HEC-RAS:

* All the Reaches in the Stream Network must be connected at common End Points; disjointed Stream Networks are not allowed; Reaches must also NOT cross each other.

* Streams cannot contain parallel flow lines. If three reaches connect at a node or End Point, at the most TWO of them can have a common Stream ID. (Please note that a Reach is uniquely identified by a Reach ID and a Stream ID.)

* Cross-Section lines can cross a Reach line only once and cannot cross other cross section lines.

Program Execution

Before starting the "Prepare HEC-RAS Input File" command, all the SCTfiles and their corresponding MXS files should have been created for every Reach. Points where two streams meet would form a node in the stream network. Sections of a stream between such nodes should be modeled as a Reach. and drawn as a separate polyline. Now, change to the Civil Design Menu. The MXS file for each Reach is created using the command *Input Edit Section Alignment* under the *Sections* pulldown menu. Based on any of the methods for creating section files (described in chapter 6 of this manual), the Section file for the Reach is created. The user must manage the .MXS file and the .SCT file corresponding to each Reach. At this point, a Stream ID and Reach ID may be assigned to every Reach, based on a convenient naming convention, which is entirely up to the user. These IDs would be needed when creating the HEC-RAS input file.

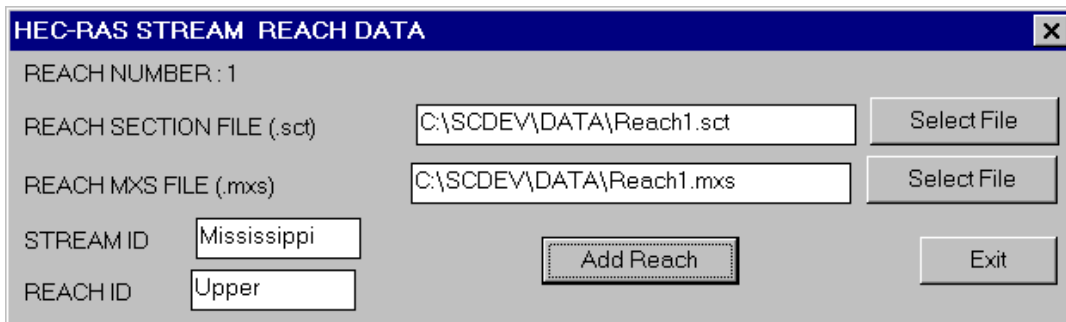
The program starts by asking the user for the Header information. The user can input as much information in this dialog box as possible. The "Units" can be "Metric" or "English".

The screenshot shows the "HEC-RAS Header Information" dialog box. It features a title bar with a close button (X). The main area contains several configuration options:

- Units:** Two radio buttons are present. "English" is selected, and "Metric" is unselected.
- DTM Type (TIN, Raster etc.):** A dropdown menu is set to "Raster".
- Stream Layer:** A dropdown menu is set to "Stream".
- Cross-Section Layer:** A dropdown menu is set to "Xsect".
- Map Projection:** A dropdown menu is set to "StatePlan".
- Datum:** A dropdown menu is set to "NAD27".
- DTM Name:** A text input field contains the text "Mississip".
- Number of Reaches:** A text input field contains the number "3".
- Number of Cross-Sections:** A text input field contains the number "20".
- Projection Zone:** A text input field contains the number "5101".
- Vertical Datum:** A dropdown menu is set to "NAD27".

At the bottom of the dialog, there are two buttons: "OK" and "Cancel".

Next, the user will be prompted to enter the .MXS and .SCT file names, the Stream ID and Reach ID for each Reach that you wish to add to your model. The user can enter data (IDs and file names) for as many Reaches as wished. That is, the user can create input files for each Reach individually and import them individually into HEC-RAS or create a combined input file for all the Reaches in the Stream Network. This makes it very convenient to add more Reaches to the HEC-RAS model at a later stage or do the analysis for various sections separately. After entering as many Reaches as needed, the user presses "Exit" to stop entering any further Reaches and to continue with the program execution.

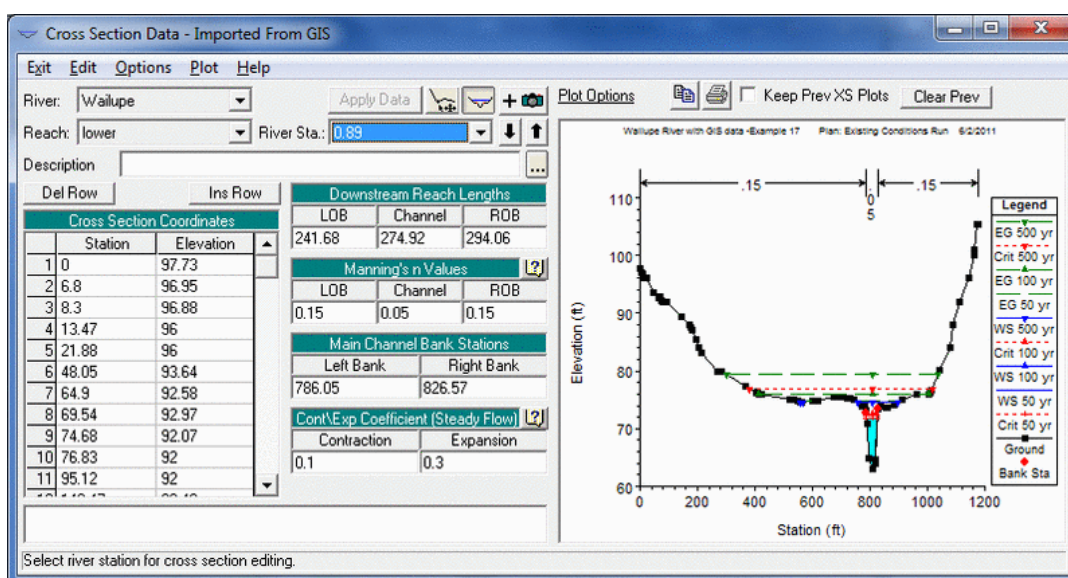


On pressing "Exit", the user is prompted for the Input HEC-RAS file to be created. HEC-RAS input files have a .GEO extension. When the file is chosen at the prompt, the program creates the input file for HEC-RAS. This file can be used to import geometric data into HEC-RAS, as described below. You must have HEC-RAS version 2.0 or higher installed on your computer.

HEC-RAS

After starting HEC-RAS, select "Geometric Data" from under the "Edit" pulldown menu to open the Geometric data editor, which has a CAD screen and various options. From the "File" pulldown menu on the Geometric data editor, run "Import Geometric Data -> GIS Format" command and select the .GEO file just created. This loads the geometric data into HEC-RAS and displays it in the form of a Stream network, with EndPoint, Stream ID and Reach IDs, Cross Sections stationing information, along with directions of in each Reach.

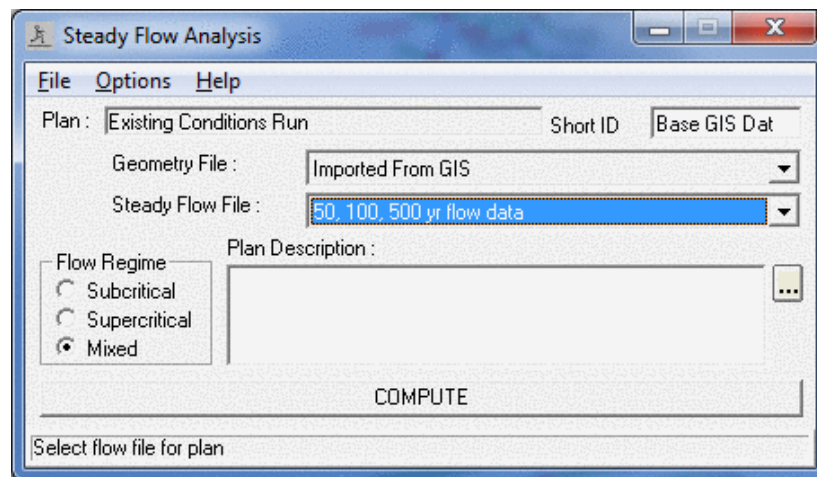
Since Carlson only provides default values, the cross section data should be checked next. For example the manning's n, left over bank and right over bank position, and the lengths of the left and right over banks. By default, the left bank is given to be at 0.45 times the cross-section length and the right bank is given to be at 0.55 times the cross-section length. In order to correctly model the channel geometry, the location of the banks must be accurately defined for each cross-section. The left and right over bank lengths are defaulted to the centerline length (which may not be equal in the case of a sharp bend in the stream).



Geometric data can be stored by running "Save Geometric Data" from the "Geometric Data Editor". The file extension assigned for Geometric data files is *.g*, which means that successive geometric data files will be given file extensions in a numeric sequence, beginning with *.g01.

Flow data can be entered in the "Steady Flow Data Editor", which can be brought up by selecting "Steady Flow Data" from the Edit pulldown menu of the main HEC-RAS window. The data that can be selected here are the number of profiles that need to be run, flow in each reach for each profile simulation and the Hydraulic boundary conditions at each Reach for each Profile simulation. This information is stored in a file with the extension *.f01 and so on for successive files.

Once all the geometric data and flow data is entered, run command "Run->Steady Flow Analysis". After selecting the type of flow condition (sub-critical, super-critical or mixed), the user selects the "Compute" button to complete the analysis. If there are errors or serious warnings, the program reports them in a text editor. Otherwise, the program shells out to a DOS screen and completes all the necessary calculations. Several options are available for viewing and editing output from the HEC-RAS program, which are best explained in its manual.



Pulldown Menu Location: Watershed->HEC-Ras Water Surface Model

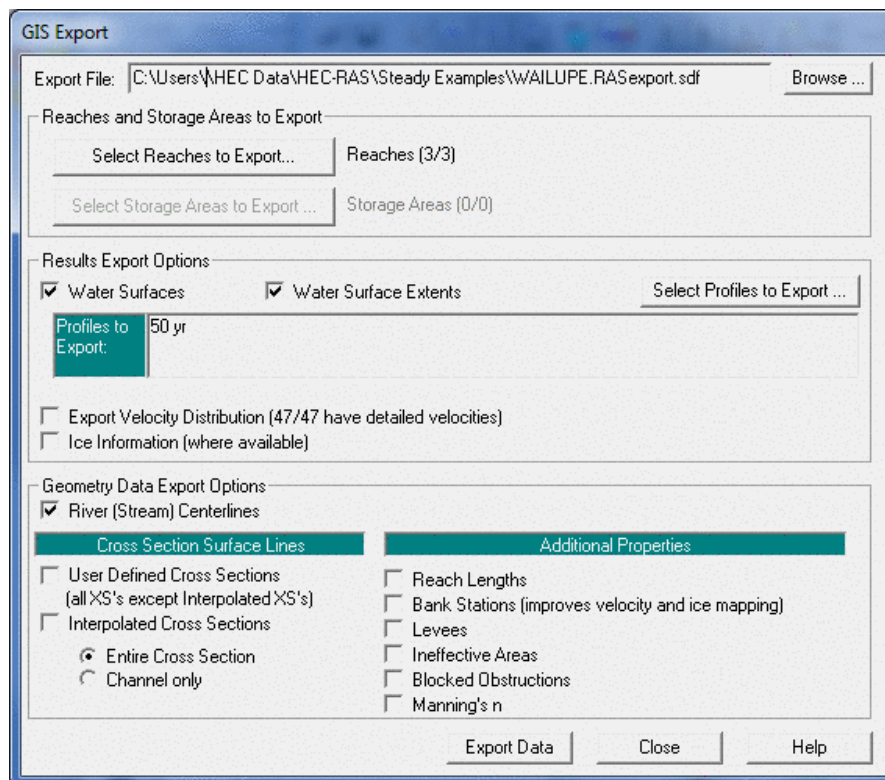
Keyboard Command: sct2ras

Prerequisite: Section data (.sct)

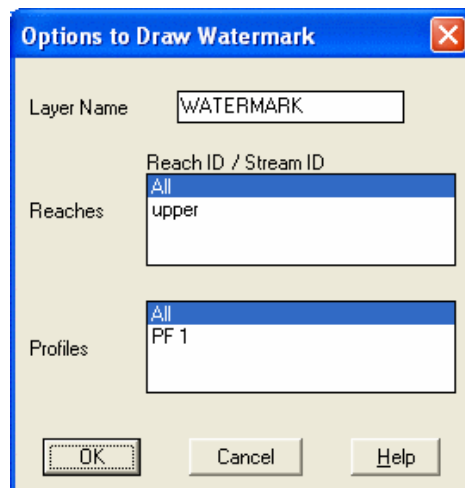
Draw Hec-Ras Watermark

HEC-RAS Watermark represents the boundary around a stream or channel model that is the extents of flooding for a given Storm Frequency Event. This boundary is more commonly referred to as the Flood Hazard Area in FEMA documentation. This command uses an existing HEC-RAS "SDF" file to transfer data from the flood model to a format that can be imported into CAD.

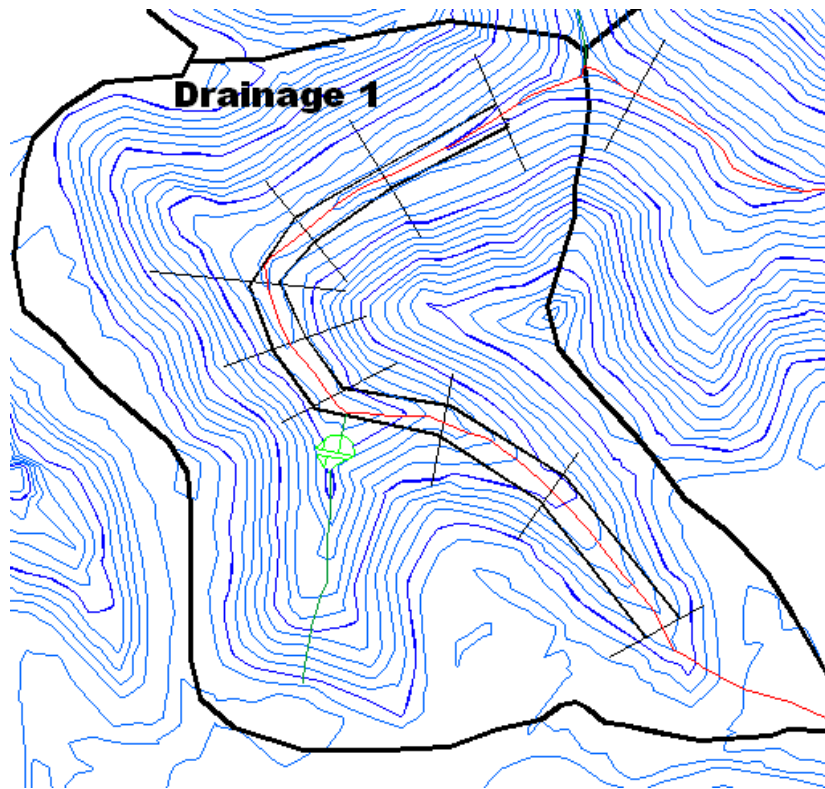
Please note that the flood hazard area can also be created using Carlson's HEC-RAS Manager, but if the Flood Study is completed or has been provided by an outside source, this command will map the Flood Hazard Area from an existing Flood Study without having to perform additional calculations. To export the information needed to map a model's watermark, an SDF file is needed which may come with the model package or can be created by opening the model in HEC-RAS and exporting an SDF file. If an SDF file has not been created by HEC-RAS for a completed study, it can be created using HEC-RAS. To create a SDF file in HEC-RAS, run command File->Export GIS Data to save the data needed to map the Flood Hazard Area. Below is the GIS Export dialog. Make sure to select the option "Water Surface Extents", otherwise the watermark data is not saved.



After loading the .SDF file, a dialog window prompts for the reaches and profiles that you would like to draw the watermark.



Shown next is an example HEC-RAS watermark plot based on a run of HEC-RAS using the file Hydrolesson.dwg, and using an input flow rate of 20,000 cfs, a Manning's n of 0.013 for the left and right bank conditions, and with the boundary condition set to critical depth:



Please note that this method only uses geometry at each cross section to determine watermark location. In between Stations, the location of the watermark boundary is interpolated. If additional precision is needed, additional cross sections must be added to the model.

Existing HEC-RAS models that do not have a SDF file included in their distribution must be reprocessed by HEC-RAS to create this file. This is common when obtaining existing flood studies that were performed prior to HEC-RAS having the ability to export an SDF file. When available, please compare the resulting Watermark delineation to existing published maps such as those provided by FEMA.

Pulldown Menu Location: Watershed->HEC-RAS Water Surface Model

Keyboard Command: drawras

Prerequisite: Prepare HEC-RAS Input File, and the program, HEC-RAS or programs that duplicate the output of HEC-RAS

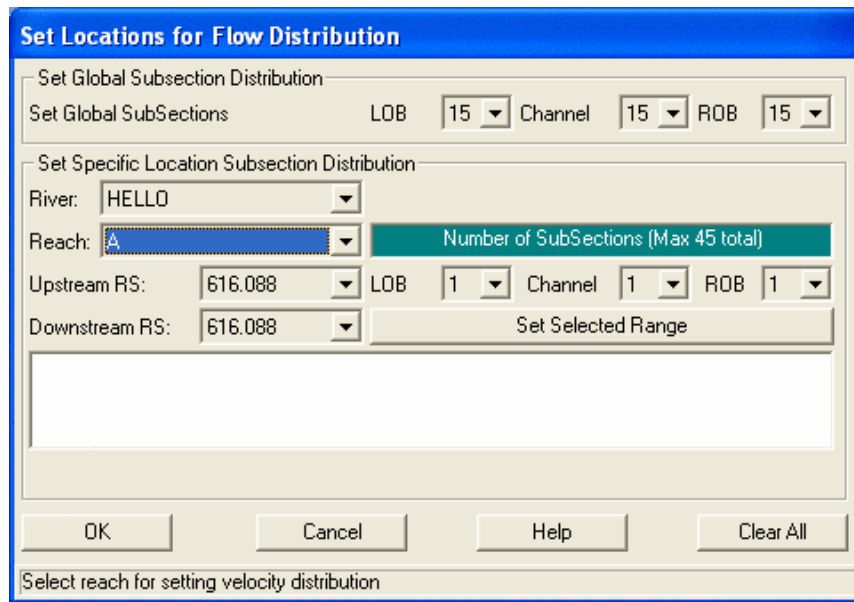
Import Flow Velocity Points

This function extracts the flow velocity distribution from the HEC-RAS output report file (.REP). The velocity points are extracted at every cross section along the river channel. All points are imported to a Carlson coordinate file (.CRD) and can be plotted in a TIN.

Running HEC-RAS

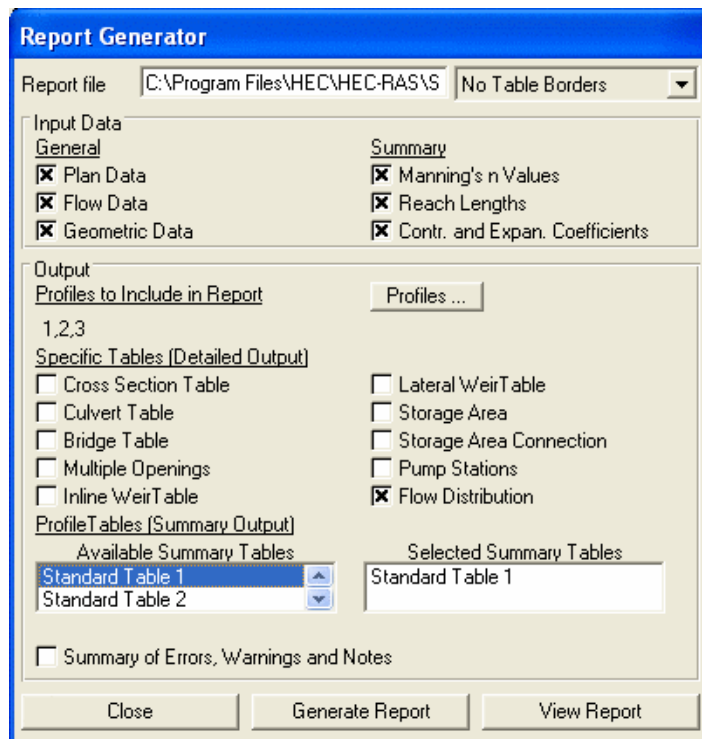
In order to get the flow velocity at all cross sections, some guide lines in running HEC-RAS are provided as below.

1. From the Watershed > HEC-RAS Water Surface Model menu in the Hydrology Module, choose Prepare HEC-RAS Input File command to make a HEC-RAS geometry file (.GEO), which contains the cross section data of one or more reaches. Then in HEC-RAS, in Geometric Data dialog, select Import Geometry Data of GIS format from File menu and load the .GEO file.
2. When running Steady/Unsteady Flow Analysis, in the Steady/Unsteady Analysis dialog, choose Flow Distribution Locations command from the Options menu. This command allows you to subdivide the left bank, channel and right bank. Specify as many subsections as needed. You can define up to 45 subsections.



HEC-RAS: Flow Distribution Dialog

3. After finishing the flow analysis, select Generate Report command from File menu to display the Report Generator dialog. In the Output field, make sure to check the Flow Distribution check box and set the Summary Tables to Standard Table 1.

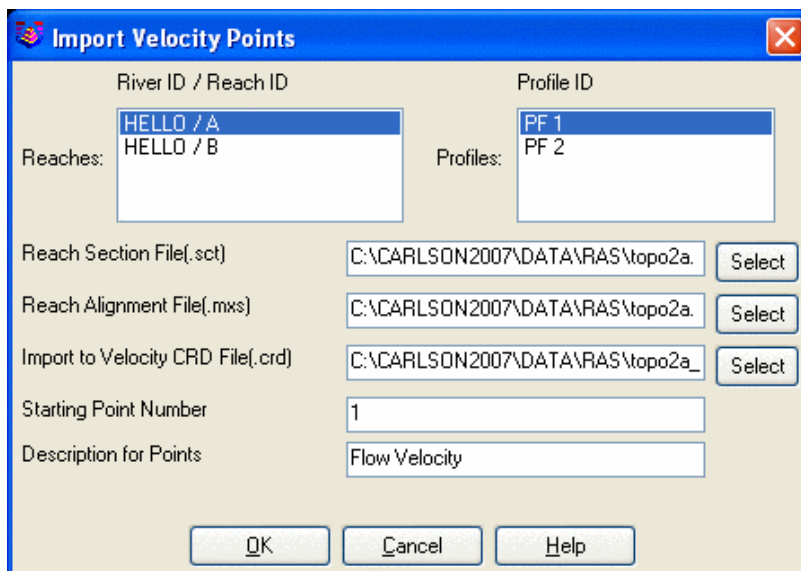


HEC-RAS: Report Generator Dialog

Importing Velocity Points

Select Import Flow Velocity Points from Watershed > HEC-RAS Water Surface Model menu. This command takes the HEC-RAS output file (.REP) and displays the Reaches list and Profiles list in the Import Hydraulic Depth Points dialog. In the Reach Section and Alignment File boxes, type or select a section file(.SCT) and the corresponding section alignment file (.MXS) that have been used to generate HEC-RAS input file (.GEO). In the Import to Flow Velocity CRD File box, type or select a CRD file. In the Starting Point Number box, enter the starting point number, the default number is 1. In the Reach and Profile lists, choose the reach and profile that you want to output, and then

click OK button to extract the flow velocity distributions and write data to the .CRD file.



Import Hydraulic Velocity Points Dialog

Prompts

Import Flow Velocity Points dialog: Fill in values.

Pulldown Menu Location: Watershed > HEC - RAS Water Surface Model > Import Flow Velocity Points

Keyboard Command: crdrasvt

Prerequisite: HEC-RAS output report file (.REP) and the corresponding section file (.SCT) and section alignment file (.MXS)

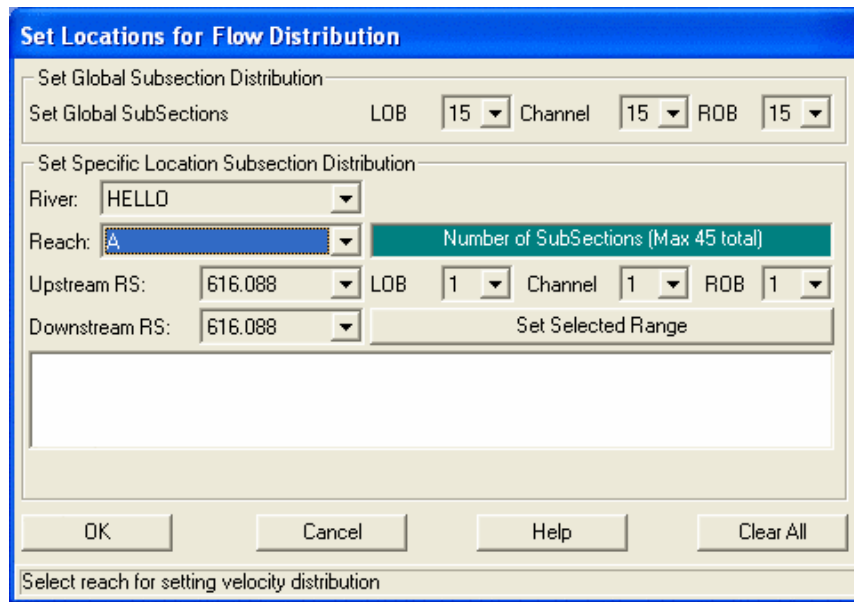
Import Flow Depth Points

This function extracts the flow depth distribution from the HEC-RAS output report file (.REP). The depth points are extracted at every cross section along the river channel. All points are imported to a Carlson coordinate file (.CRD) and can be plotted in a TIN.

Running HEC-RAS

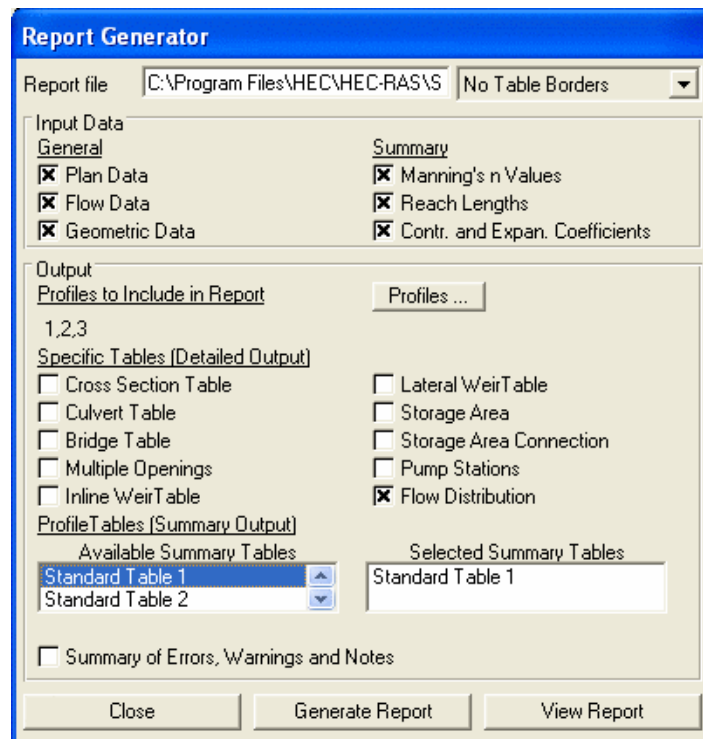
In order to get the flow depth at all cross sections, some guide lines in running HEC-RAS are provided as below.

1. From the Watershed > HEC-RAS Water Surface Model menu in the Hydrology Module, choose Prepare HEC-RAS Input File command to make a HEC-RAS geometry file (.GEO), which contains the cross section data of one or more reaches. Then in HEC-RAS, in Geometric Data dialog, select Import Geometry Data of GIS format from File menu and load the .GEO file.
2. When running Steady/Unsteady Flow Analysis, in the Steady/Unsteady Analysis dialog, choose Flow Distribution Locations command from the Options menu. This command allows you to subdivide the left bank, channel and right bank. Specify as many subsections as needed. You can define up to 45 subsections.



HEC-RAS: Flow Distribution Dialog

3. After finishing the flow analysis, select Generate Report command from File menu to display the Report Generator dialog. In the Output field, make sure to check the Flow Distribution check box and set the Summary Tables to Standard Table 1.

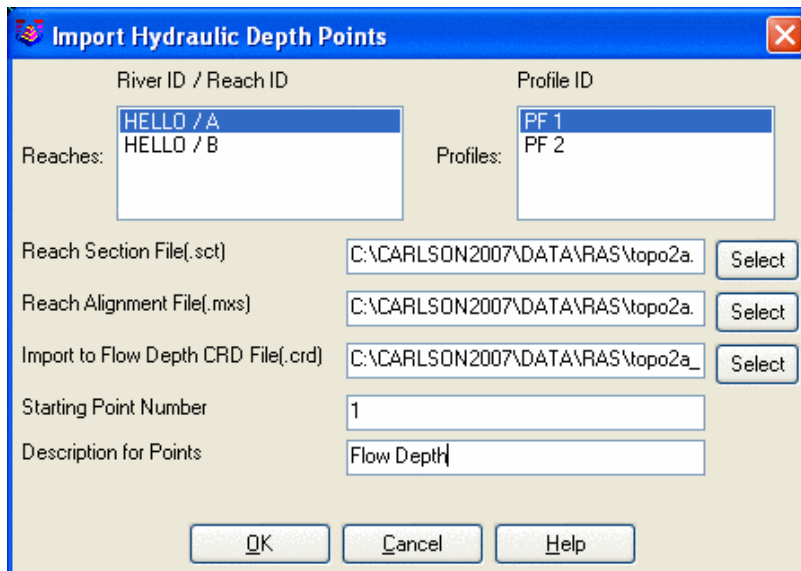


HEC-RAS: Report Generator Dialog

Importing Depth Points

Select Import Flow Depth Points from Watershed > HEC-RAS Water Surface Model menu. This command takes the HEC-RAS output file (.REP) and display the Reaches list and Profiles list in the Import Hydraulic Depth Points dialog. In the Reach Section and Alignment File boxes, type or select a section file(.SCT) and the corresponding section alignment file (.MXS) that have been used to generate HEC-RAS input file (.GEO). In the Import to Flow Depth CRD File box, type or select a CRD file. In the Starting Point Number box, enter the starting point number, the default number is 1. In the Reach and Profile lists, choose the reach and profile that you want to output, and then

click OK button to extract the flow depth distributions and write data to the .CRD file.



Import Hydraulic Depth Points Dialog

Prompts

Import Flow Depth Points dialog: Fill in values.

Pulldown Menu Location: Watershed > HEC - RAS Water Surface Model > Import Flow Depth Points

Keyboard Command: crdrasdt

Prerequisite: HEC-RAS output report file (.REP) and the corresponding section file (.SCT) and section alignment file (.MXS)

Channel Routing Hydrograph - Modified Att-Kin Method

This function extracts the flow depth distribution from the HEC-RAS output report file (.REP). The depth points are extracted at every cross section along the river channel. All points are imported to a Carlson coordinate file (.CRD) and can be plotted in a TIN.

Running HEC-RAS

In order to get the flow depth at all cross sections, some guide lines in running HEC-RAS are provided as below.

1. From the Watershed > HEC-RAS Water Surface Model menu in the Hydrology Module, choose Prepare HEC-RAS Input File command to make a HEC-RAS geometry file (.GEO), which contains the cross section data of one or more reaches. Then in HEC-RAS, in Geometric Data dialog, select Import Geometry Data of GIS format from File menu and load the .GEO file.
2. When running Steady/Unsteady Flow Analysis, in the Steady/Unsteady Analysis dialog, choose Flow Distribution Locations command from the Options menu. This command allows you to subdivide the left bank, channel and right bank. Specify as many subsections as needed. You can define up to 45 subsections.

HEC-RAS: Flow Distribution Dialog

3. After finishing the flow analysis, select Generate Report command from File menu to display the Report Generator dialog. In the Output field, make sure to check the Flow Distribution check box and set the Summary Tables to Standard Table 1.

Importing Depth Points

Select Import Flow Depth Points from Watershed > HEC-RAS Water Surface Model menu. This command takes the HEC-RAS output file (.REP) and display the Reaches list and Profiles list in the Import Hydraulic Depth Points dialog. In the Reach Section and Alignment File boxes, type or select a section file(.SCT) and the corresponding section alignment file (.MXS) that have been used to generate HEC-RAS input file (.GEO). In the Import to Flow Depth CRD File box, type or select a CRD file. In the Starting Point Number box, enter the starting point number, the default number is 1. In the Reach and Profile lists, choose the reach and profile that you want to output, and then click OK button to extract the flow depth distributions and write data to the .CRD file.

Import Hydraulic Depth Points Dialog

Prompts

Import Flow Depth Points dialog: Fill in values.

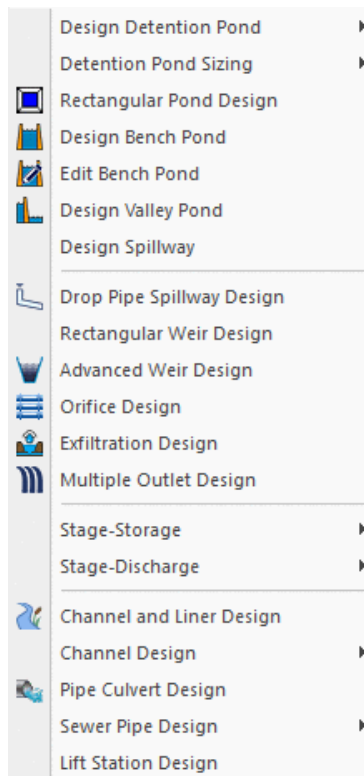
Pulldown Menu Location: Watershed > HEC - RAS Water Surface Model > Import Flow Depth Points

Keyboard Command: crdrasdt

Prerequisite: HEC-RAS output report file (.REP) and the corresponding section file (.SCT) and section alignment file (.MXS)

Structure Menu

Shown here is the Structure pulldown menu that contains commands for hydraulic structures including ponds, channels, pipes and outlets. The Design Bench Pond and Design Valley Pond commands are described in the Civil Design manual.



Design Detention Pond

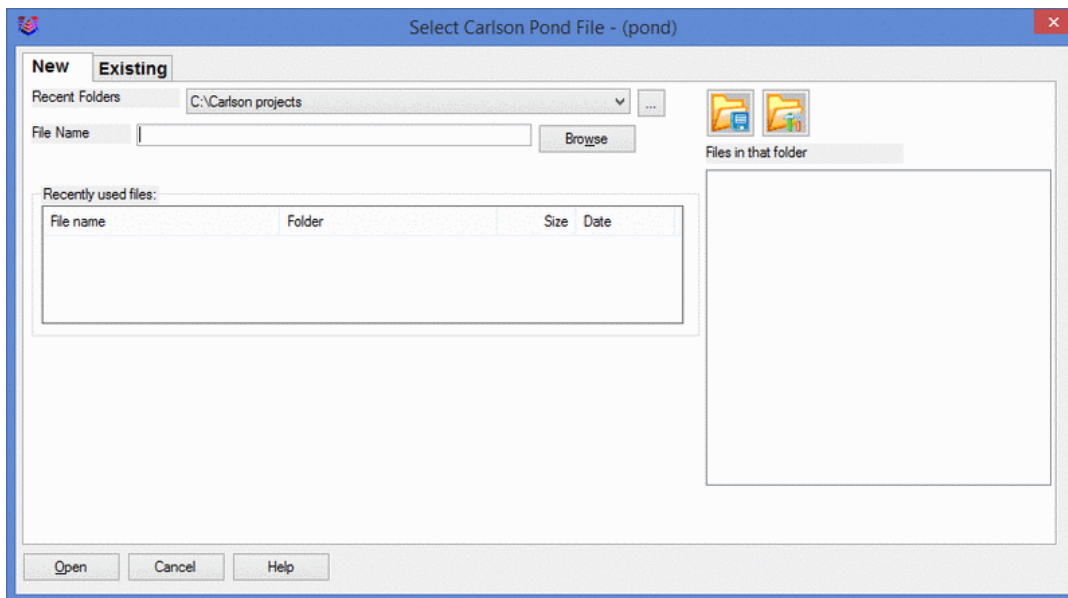
Design Detention Pond is a powerful stormwater model program that helps engineers analyze and design simple and complex watershed. This program lets you:

- Analyze pre- and post-developed watershed conditions and pond sizes;
- Analyze one watershed condition and pond sizes;

- Use any rainfall duration or distribution to compute hydrographs;
- Compute hydrographs for multiple events;

- Automate almost every aspect of drainage design computations;
- Build customized reports;

When you first run the command, the Select Carlson Pond File dialog box opens. You can start a completely new pond project or open an existing pond project.

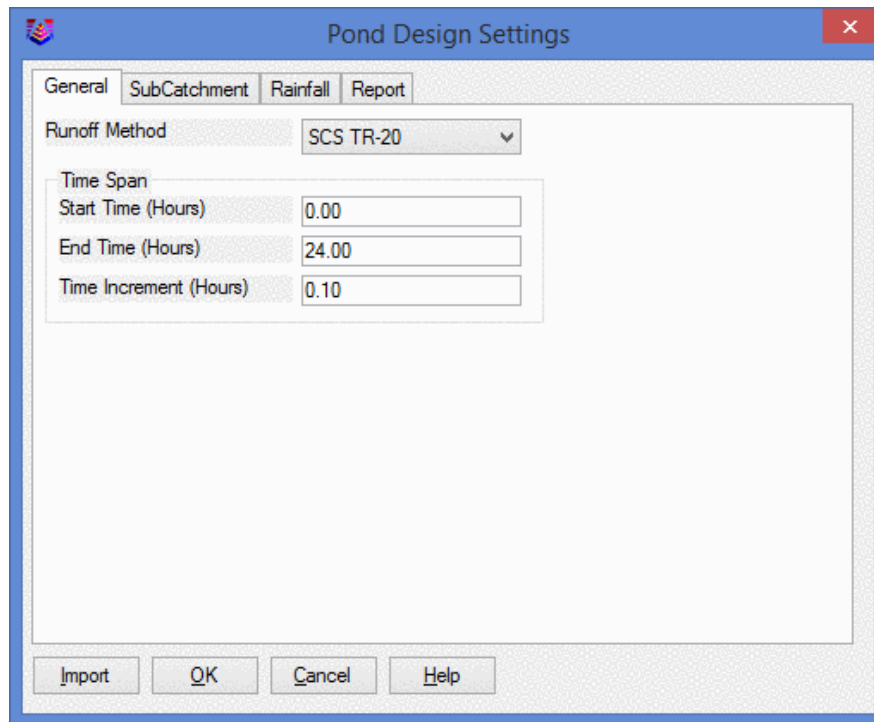


Select Carlson Pond File

Pond Design Settings

When you start a new pond project, the pond design settings dialog will open first to let you set up the project watershed information, rainfall data, calculation method and report format. The **Import** button allows you to import the project settings from an existing pond project. The pre- and postdevelopment watershed surface files and layers files are imported too.

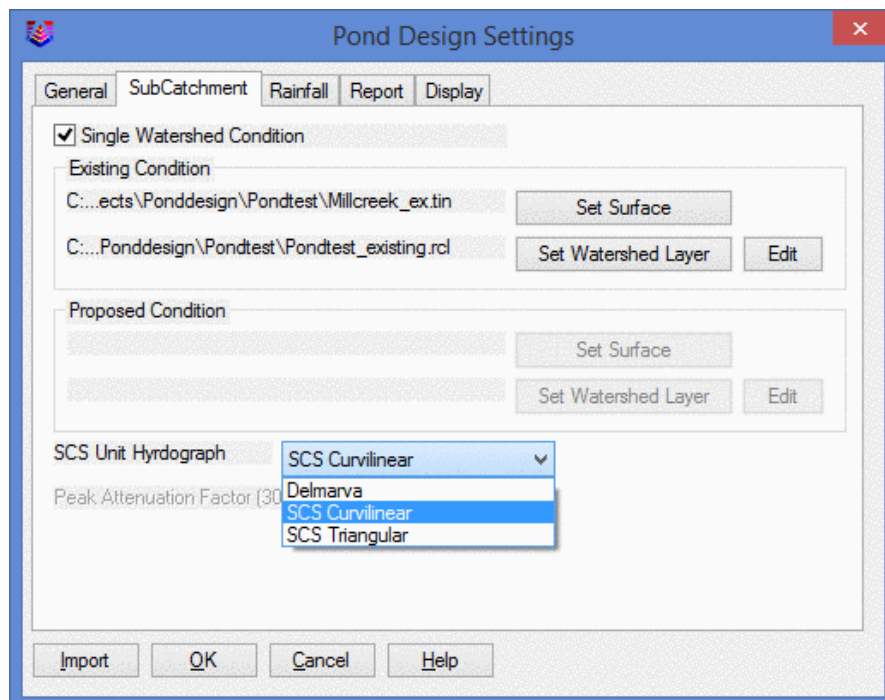
The general tab allows selection of the runoff hydrograph method, and setup of the hydrograph time span. There are SCS TR-20 method and Rational method for producing a runoff hydrograph for a given subcatchment.



Pond Design Settings

Subcatchment:

Subcatchment tab allow you to set up the watershed conditions for existing and proposed conditions. Check **Single Watershed Condition** box when designing a pond when there's only one condition, either existing or proposed condition. The watershed layer file is defined in the program of Define Watershed Layers. The Edit button can lead you to the program to make any changes. Please refer the Define Watershed Layers documentation to set up the watershed layers. Select SCS Unit Hydrograph, supported options Delmarva, SCS Curvilinear, and SCS Triangular. If SCS Triangular is selected the Peak Attenuation Factor may be set.



Rainfall:

The rainfall tab defines rainfall data for the SCS and Rational runoff methods. The storm type determines what the rainfall distribution is selected. The rain library button allows you to choose a storm type from the available rainfall distributions, and you can also create new rainfall data.

The AMC (Accident Moisture Condition) specifies the moisture level in the ground immediately prior to the storm. Use a value of 2 for normal conditions.

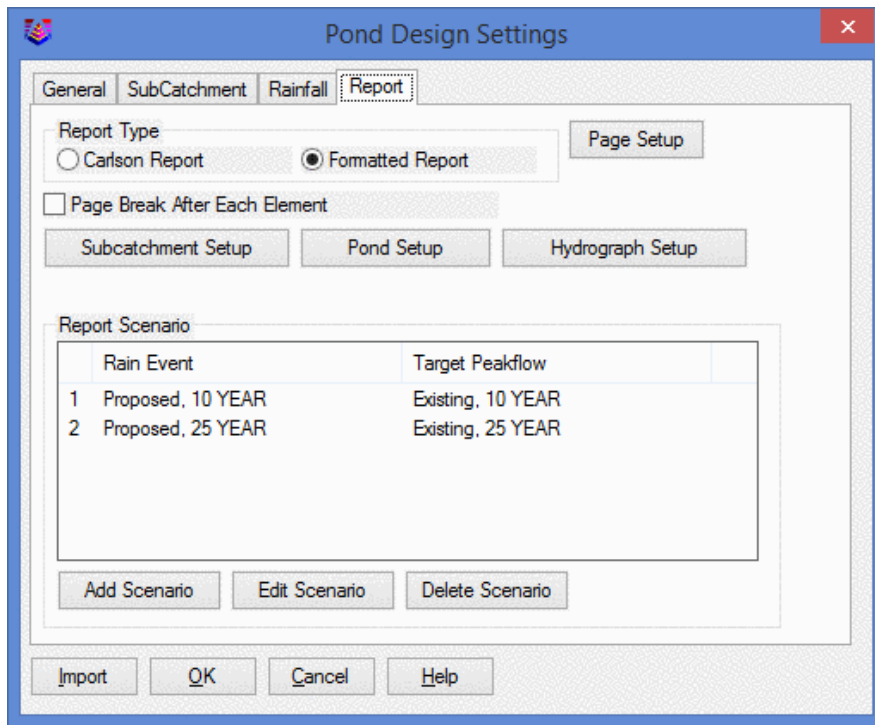
In the rainfall event spreadsheet, you can specify various storm events and their total precipitation.

The screenshot shows the 'Pond Design Settings' dialog box with the 'Rainfall' tab selected. The 'Storm Type' is set to 'TYPE II 24-HOUR' and 'Storm Duration (hr)' is 24.00. The 'AMC' section has radio buttons for 1, 2, and 3, with 2 selected. The 'Events' table lists storm events with their names and rain depths. To the right of the table are buttons for 'Rain Map', 'Add Row', 'Remove Row', 'Up', and 'Down'. At the bottom are 'Import', 'OK', 'Cancel', and 'Help' buttons.

	Event Name	Rain Depth (in)
1	1 YEAR	1.000
2	2 YEAR	2.000
3	10 YEAR	3.000
4	25 YEAR	5.000
5	50 YEAR	6.000
6	100 YEAR	8.000
7		

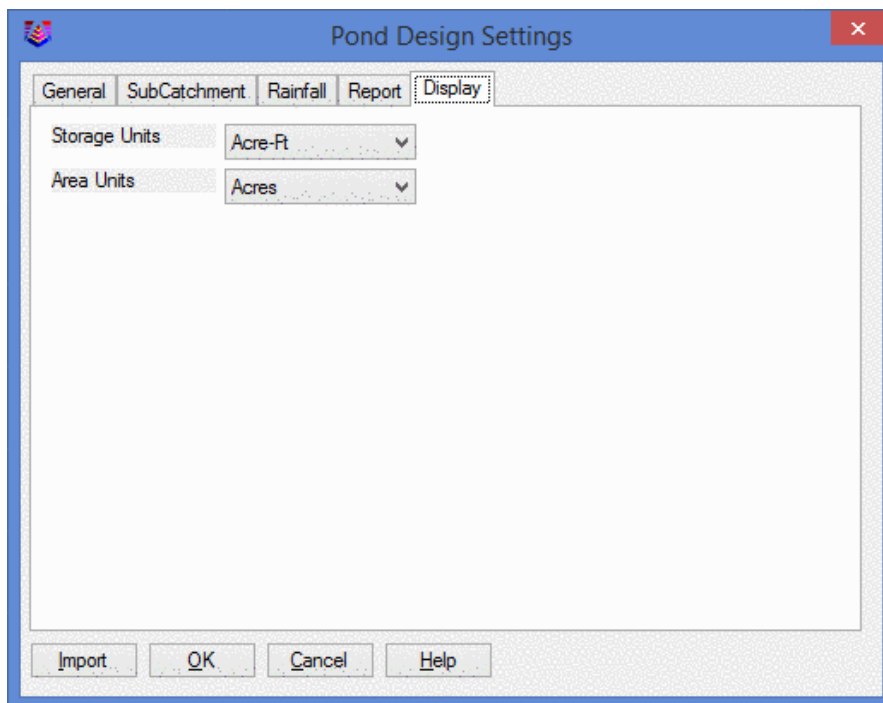
Report:

In report tab, you can select the report type: Carlson report or formatted report. The formatted report is display the report in a .pdf file format. The report scenario list allows you to report watershed runoff and pond sizes for multiple events.



Display:

In display tab, you can set up the pond storage units and subcatchment area units.



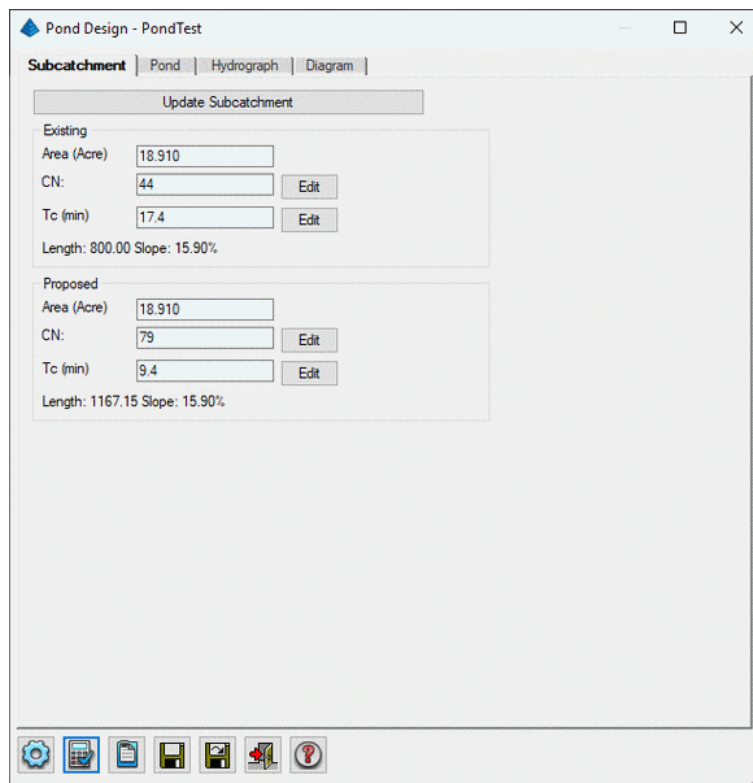
Pond Design

The pond design dialog opens when you start an existing pond project, or finish setting up the project settings for a new pond project. This dialog allows you to enter subcatchment parameters, and analyze the watershed runoff and pond sizes.

Subcatchment:

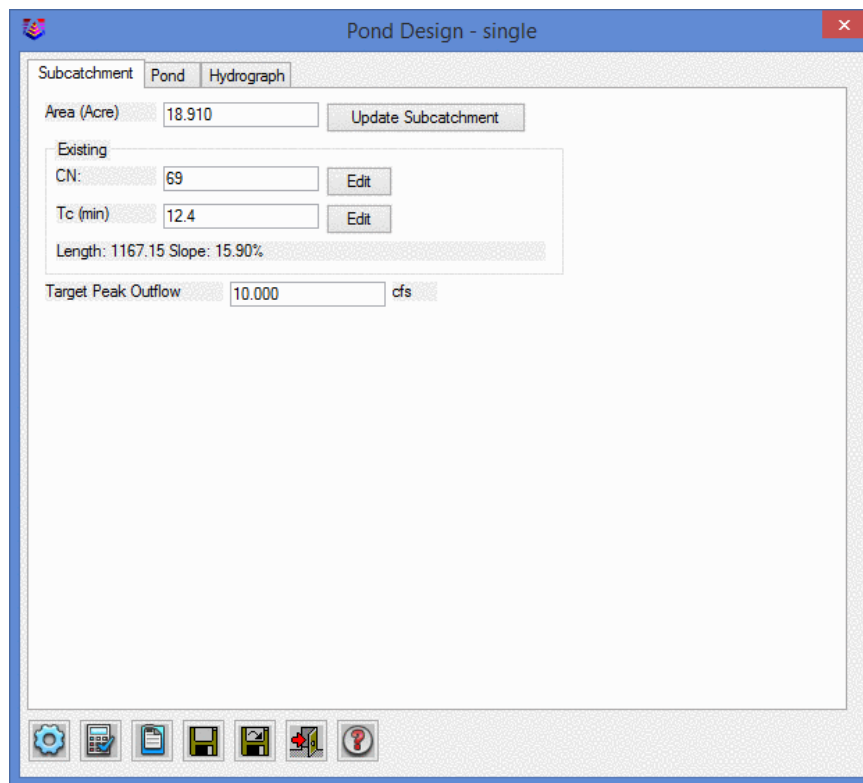
Subcatchment tab is where you enter the subcatchment data. The **Update Subcatchment** button allow the program to pull out the information from the existing and proposed watershed surface files and watershed layers files and fill all the values automatically. You can also enter values manually. The Edit buttons next to all the fields allow you to run the Curve Number program and Time of Concentration program to calculate the CN and Tc values. Please refer to the user manual on Curve Number command, and the Time of Concentration commands, both SCS method and Rational method.

When there's only one watershed condition in design, the target peakflow for outflow condition should be specified for pond sizing.



The screenshot shows the 'Pond Design - PondTest' dialog box with the 'Subcatchment' tab selected. The dialog has a title bar with standard window controls and a menu bar with 'Subcatchment', 'Pond', 'Hydrograph', and 'Diagram'. A prominent 'Update Subcatchment' button is at the top. Below it, the 'Existing' section contains input fields for 'Area (Acre)' (18.910), 'CN' (44), and 'Tc (min)' (17.4), each with an 'Edit' button. The 'Proposed' section contains input fields for 'Area (Acre)' (18.910), 'CN' (79), and 'Tc (min)' (9.4), also with 'Edit' buttons. Both sections show 'Length: 800.00 Slope: 15.90%' for existing and 'Length: 1167.15 Slope: 15.90%' for proposed. A toolbar with icons for settings, print, save, and help is at the bottom.

Parameter	Existing Value	Proposed Value
Area (Acre)	18.910	18.910
CN	44	79
Tc (min)	17.4	9.4
Length	800.00	1167.15
Slope	15.90%	15.90%



Pond:

Pond tab is where you specify the pond parameter and outlets information. The Stage-Storage Edit button opens the Stage-Storage Editor dialog to create and edit the pond data. The Stage-Discharge Edit button opens the Multiple Outlet Design dialog to create and edit the outlet data. There are two methods to estimate the pond size: TR-55 method and Linear Storage method. Please refer to the user manual on the topics of Stage-Storage Editor, Multiple Outlet Design, Detention Pond Sizing - TR-55 method and Linear Storage method.

Pond Design - PondTest

Subcatchment **Pond** Hydrograph Diagram

Stage-Storage: Peak 90.00 ft, 7.2314 acre-ft Edit
 Bottom Elev: 80.00 ft, Top Elev: 90.00 ft

Stage-Discharge: Peak 90.00 ft, 74.80 cfs Edit

Outlet Name	Invert (ft)	Discharge (cfs)
orifice	84.00	8.86
weir	88.00	65.93

Stage Increment: 1.000 ft
 Water Elev at T = 0: 80.000 ft
 Tailwater Elev: 0.000 ft

Pond Sizing Method: Linear Storage Method Estimate Pond Size

	Rain Event	Target Outflow Vol (ac-ft)	Pond Inflow Vol (ac-ft)	Pond Outflow Vol (ac-ft)	Estimate Pond Size (ac-ft)	Pond Storage (ac-ft)	Pond Elev (ft)	Pond Depth (ft)
1	1 YEAR	0.00	0.82	0.00	0.82	0.82	81.18	1.18
2	2 YEAR	0.02	1.87	0.00	1.82	1.87	82.67	2.67
3	10 YEAR	0.23	3.07	0.20	2.56	2.87	84.08	4.08
4	25 YEAR	0.62	4.34	1.45	2.47	3.03	84.30	4.30
5	50 YEAR	1.15	5.65	2.77	2.45	3.63	85.13	5.13
6	100 YEAR	2.56	8.33	5.44	3.16	5.11	87.17	7.17

Input-Edit Stage-Storage

Structure Name: demo1

Storage Method:
 User Defined Storage
 Irregular Shape
 Rectangular Shape
 Underground Pipe

Edit Detention Structure

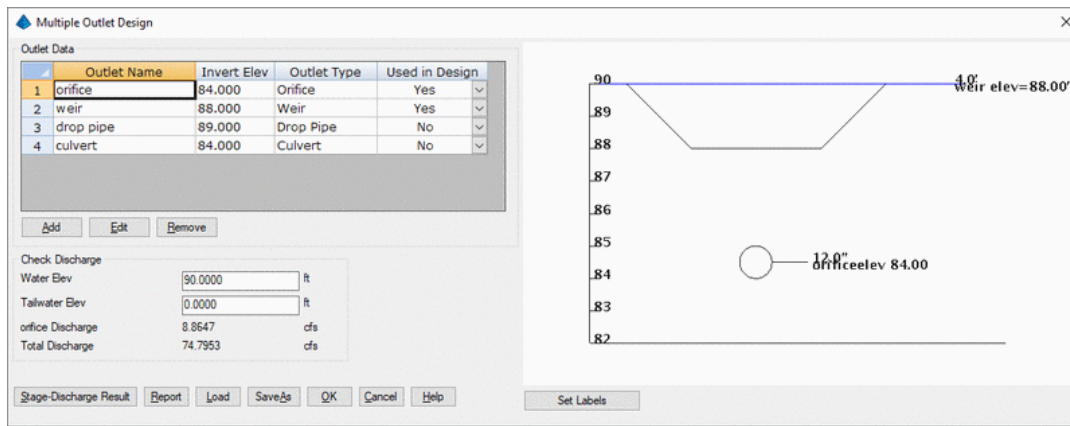
Storage Unit: Acre-Ft Area Unit: Acre

Elevation(Ft)	Stage(Ft)	Storage	Area
328.000	0.000	0.000	0.1558
330.000	2.000	0.343	0.1876
332.000	4.000	0.753	0.2216
334.000	6.000	1.250	0.2755
336.000	8.000	1.846	0.3214

Check Storage
 Elevation: Stage: Storage: Area:

Use Report Formatter

Load SaveAs Graph Report OK Cancel Help



Hydrograph:

When click on the **Compute** button, the results of all the rainfall events are shown on the Hydrograph tab. The peak flow results are shown in the spreadsheet, and the hydrograph of highlighted rainfall event are displayed on the top. The Pond Inflow hydrograph is the runoff hydrograph of proposed watershed, Pond Outflow hydrograph is the routing hydrograph, and the Target Outflow hydrograph is the runoff hydrograph of the existing watershed. When the routing hydrograph is less than the target outflow hydrograph, the project result passes.

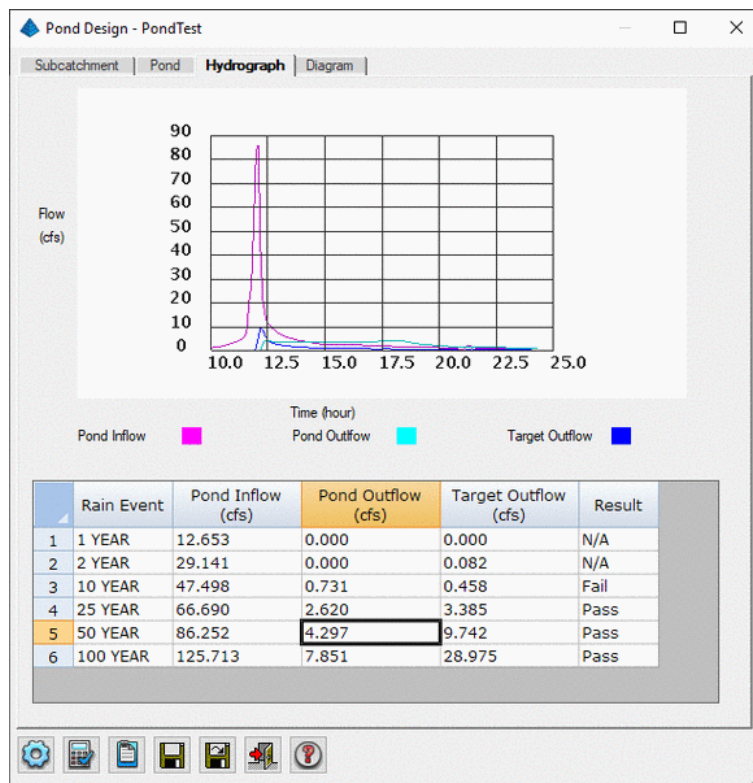
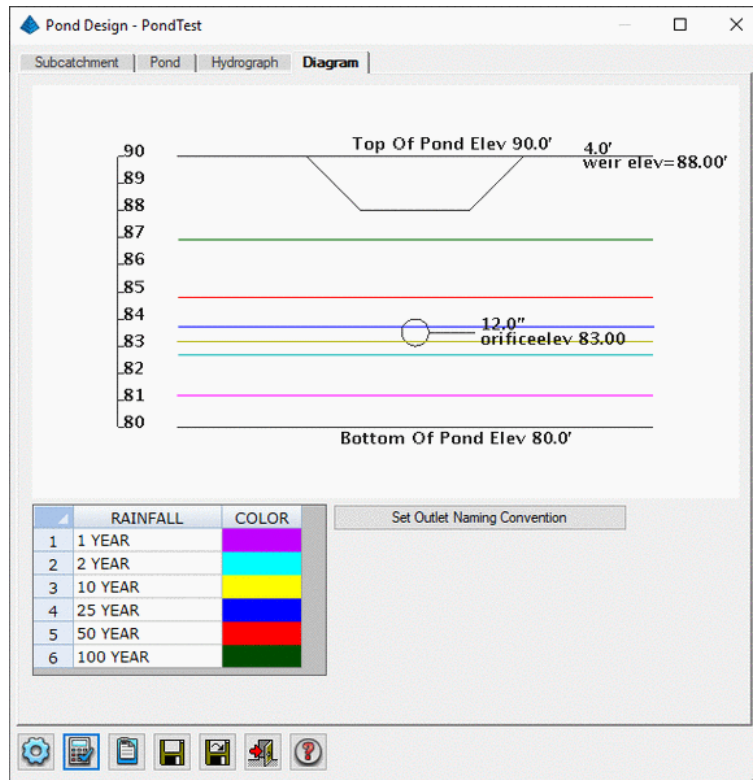


Diagram:

On this tab the outlets schematic view and the pond water levels of all the rainfall events are shown. Double-click

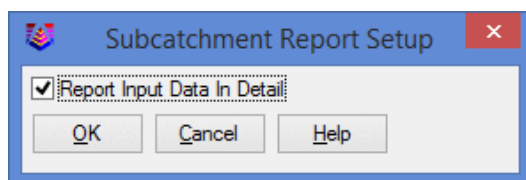
on the color cell to set new color for each rainfall event. The "Set Outlet Labels" button opens the label setup dialog.



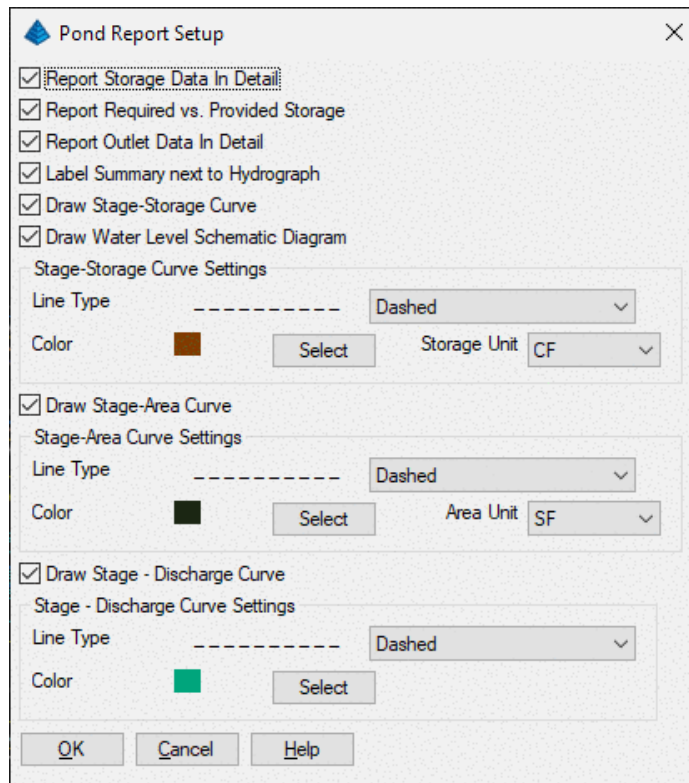
Report

The **Report** button generates report of the watershed information, pond size and outlet parameters in detail. You can setup report settings on the dialog of Pond Design Settings.

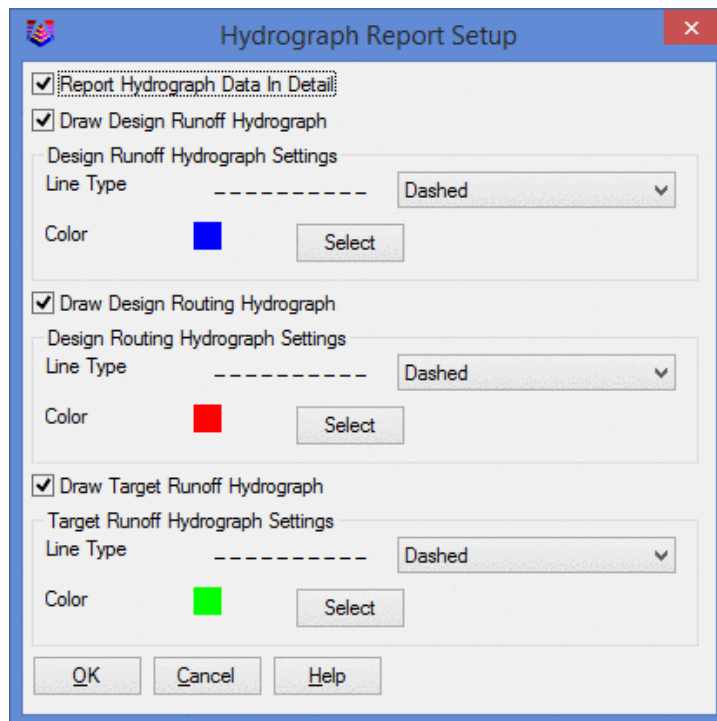
When select the option to Report Input Data In Detail on the **Subcatchment Report Setup** dialog, the area, ground cover, and runoff coefficient for every subarea will be listed in the report; otherwise only the weighted runoff coefficient and the total area will be shown.



The **Pond Report Setup** dialog has the options to report pond data and outlet data in detail, report required (estimated) storage vs. provided (designed) storage values, draw the stage-storage curves, the stage-area curves, and the stage-discharge curves, and draw water level schematic diagram.



The **Hydrograph Report Setup** dialog provides the options to report hydrograph data at every time interval; and the options to draw runoff hydrograph and routing hydrograph.



Libraries

There are four hydrology libraries used in the program: Rational Rainfall Library, SCS Rainfall Library, Curve Number (CN) Library, and Drainage Runoff (CF) Library. Please refer to the user manual on those topics.

Prompts

Design Detention Pond dialog: Fill in values.

Pulldown Menu Location: Structure > Design Detention Pond

Keyboard Command: pond_design

Prerequisite: a pond file (.pond)

Detention Pond Sizing

This command calculates the runoff and storage volumes for a detention pond. The program uses the method from the TR-55 program as described in the Urban Hydrology for Small Watersheds manual.

The command is run through the dialog box shown here. When the input values are filled in, click on the Calculate button to obtain the output values. The drainage area can be either entered directly or selected from AutoCAD by clicking on the Select Area button and then selecting the closed polyline from the screen. The peak inflow will use the value calculated in the Peak Flow-Graphical Method command. Likewise the runoff Q will use the value from the Curve Numbers & Runoff routine.

The output of this command, the storage volume value, can be applied to the Design Bench, Valley or Rectangular Pond routines. There is also an option to generate a TR-55 6A report.

Detention Pond Sizing

Input Values

Area Units

Sq ft Sq miles Acres

Drainage Area (sq ft) 1000

Select Area From Screen

Peak Inflow Discharge (cfs) 10

Desired Peak Outflow Discharge (cfs) 2

Runoff Q - Post-Dev (inches) 2

Storm Type

I IA II III

Output values

Runoff Volume: 0.0038 acft

Storage Volume: 0.0018 acft

Calculate Report Exit Help

More information regarding the various storm types can be found at the below links:

<https://www.lmnoeng.com/RainfallMaps/RainfallMaps.htm>

<https://www.wcc.nrcs.usda.gov/ftpref/wntsc/H&H/TRsTPs/TP149.pdf>

The below map, provided by the National Resource Conservation Service, marks where the various storm types occur in the USA. This map can also be found at the below link.

Fig. B-2. Approximate geographic boundaries for SCS rainfall distributions (SCS, 1986):



Figure B-2.—Approximate geographic boundaries for SCS rainfall distributions.

Prompts

Detention Pond Parameters dialog: Fill in values

Pull-down Menu Location: Structure > Detention Pond Sizing > TR-55 Method

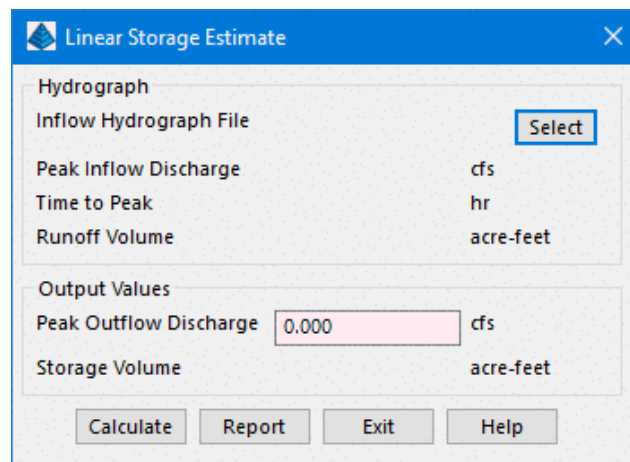
Keyboard Command: dpond

Prerequisite: None

Detention Pond Sizing - Linear Storage Estimate Method

This command calculates storage volumes for a detention pond by Linear Storage method. This method uses an inflow hydrograph and asks you to specify the peak outflow discharge from the reservoir. The program draws a straight line from the zero flow point on the hydrograph to the intersection of the specified outflow and the inflow hydrograph curve. The volume between this line and the hydrograph curve is then computed to be the storage required for the pond.

The output of this command can be applied to the Design Bench, Valley or Rectangular Pond routines. There is also an option to generate a report.



Prompts

Linear Storage Estimate dialog: Fill in values

Pulldown Menu Location: Structure > Detention Pond Sizing > Linear Storage Method

Keyboard Command: pondstorage

Prerequisite: None

Rectangular Pond Design

This program will draw rectangular ponds and calculate storage at any level in the pond corresponding to top of pond, emergency spillway, principal spillway and sediment (cleanout) level. Elevations can be "reverse-calculated" based on requested storage amounts. All calculations derive from input length-width and slope ratio values. Only one common ratio is used for the interior pond slopes (e.g. 1:1 or 2:1, etc.).

The **Draw Details** button will output scaled and fully annotated plan view, section A-A and section B-B drawings, complete with principal and emergency spillways. For simplicity, the principal spillway is considered to be a pipe spillway, and the emergency spillway is considered to be a flat-bottom weir spillway. If the pond in question has only one spillway, then the appropriate spillway elevation is entered in the dialog box, and the other spillway option is left blank.

The **Stage-Storage File** button will produce a table of storage values as .CAP file which can be plotted using the file option within the Draw Stage-Storage routine and used in hydrograph routing routines. The **Report** function creates a report that is shown in the standard report viewer and includes pond dimensions, storage volumes and "Required Freeboard". The **Draw Surface** function will draw the pond into the drawing using the Design Bench Pond command. The program will prompt for the target surface model for the pond out slopes to tie into, the location and rotation for the rectangular pond, the top of dam width and the out slope ratios. The net effect of the Rectangular Pond Design routine is that you can calculate necessary pond storages, plot the pond detail drawings, write out and import the report summary and plot the pond stage-storage curve.

There are ways to use the routine in "shortcut" form to draw ponds. Simply by completing 3 dialog entries (base width, base length and total depth) the user can draw the plan view, section A-A and section B-B. This is why the Pond Elevation items are considered "optional". The programs can also be used as a pond storage calculator. Any of the Pond Elevation options (excepting peak stage), when completed will lead to recalculated storage values. Storage values can likewise be altered and will lead to recalculated elevations. The act of pressing enter inside a dialog box activates the calculation process. If there is no need to plot the pond detail drawings, the cancel "button" in the dialog can be selected following calculations.

Prompts

The program begins by presenting the dialog. One effective way to fill out the dialog boxes is to pick the upper left box and work down and through the options by pressing the tab key after each entry. If all items are filled out as shown, the following prompts will appear:

Enter Scale Factor for Pond Drawing(s) <1>: *press Enter*

Draw Plan View: (<y>/n): *press Enter* **Pick Lower Left Corner: Plot Cleanout and Spillway Lines (<y>/n):** *press Enter*

Pick Location of Principal Spillway:

Draw Section A-A Horizontal (y<n>): *y*

Pick Left Location of Section A-A:

Pick Right Location of Section A-A:

Draw Section B-B Vertical (y/<n>): *y*

Pick One Side of Section B-B:

Pick one Side of Section B-B:

Pick Upper Left Corner for Section A-A:

Plot Cleanout and Spillway Lines (<y>/n): *press Enter*

Pick Upper Left Corner for Section B-B:

Plot Cleanout and Spillway Lines (<y>/n): *press Enter*

If no Section A-A or Section B-B identifier lines are drawn, no section A-A or Section B-B details will be drawn. Thus if you want Section A-A only, say "y" to Draw Section A-A Horizontal but "n" or Enter to Draw Section B-B Vertical. If you entered only length, width and depth in the original dialog, the resultant prompting would be:

Enter Scale Factor for Pond Drawing(s) <1>: *press Enter*

Draw Plan View? (<y>/n): *press Enter*

Pick Lower Left Corner:

Draw Section A-A Horizontal (y/<n>): y

Pick Left Location of Section A-A:

Pick Right Location of Section A-A:

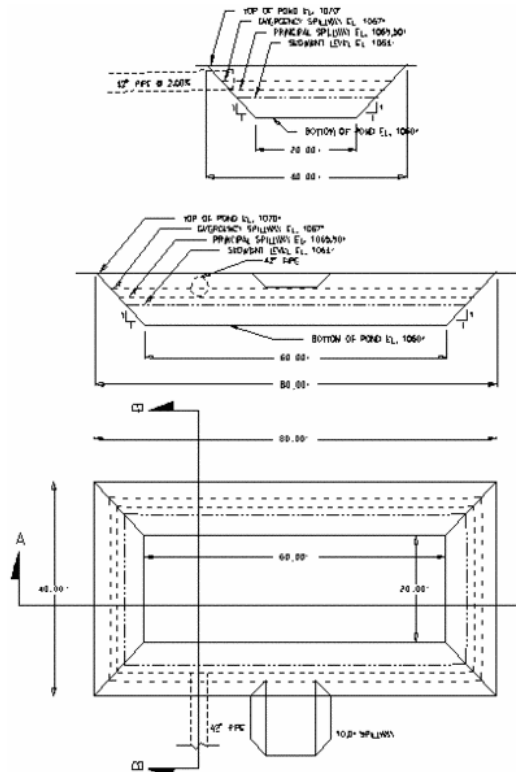
Draw Section B-B Vertical (y/<n>): y

Pick One Side of Section B-B:

Pick Other End of Section B-B:

Pick Upper Left Corner of Section A-A:

Pick Upper Left Corner of Section B-B:



Plots produced by the entries in the preceding dialog

Keep in mind that the scale factor, if other than 1, will enlarge or reduce the size of the detail drawings to suit the users needs, yet will annotate dimensions correctly in all cases.

The imported text based on the output ASCII file POND.TXT (located in \SCADXML\WORK by default) would appear as follows:

Top of Pond Elevation: 1070.00 feet Peak Stage (25th year-24 hour Storm Event): 1069.45 feet Includes 1.00 feet of Freeboard Emergency Spillway Elevation: 1067.00 Emergency Spillway Bottom Width: 10.00 Principal Spillway Invert Elevation: 1065.50 feet

Principal Spillway Diameter: 42.00 in.

Principal Spillway Slope: 2.00 % Sediment Pool (Cleanout) Elevation: 1064.00 feet

Bottom of Pond Elevation: 1060.00 feet

Storage Volume at Emergency Spillway: 0.2990 ac.ft.

Storage Volume at Principal Spillway: 0.2150 ac.ft.

Storage Volume at Sediment Pool: 0.1430 ac.ft.

The routines are fully metric and will substitute meters and cubic meters appropriately for feet and acre-feet. Pipe sizes, however, will default to diameters in inches.

Pull-down Menu Location: Structure in Hydrology

Keyboard Command: rpond

Prerequisite: None

Design Spillway

This command creates a spillway with 3D polylines in the drawing. The program uses a surface model of the area for the spillway, a spillway centerline and spillway dimensions (width, elevation, etc.). The surface model of the area can be defined by contour polylines, points and 3D polylines or can be created by the Design Bench or Valley Pond commands. The spillway dimensions can be calculated by the Design Channel commands to meet the desired discharge. The amount of cut required to make the spillway is calculated and reported.

Prompts

Source of surface model (File/<Screen>)? *press Enter* Use the File option to select a .grd file.

Pick Lower Left limit of surface area: *pick lower left*

Pick Upper Right limit of surface area: *pick upper right* Be sure to pick these limits well beyond the area of the spillway centerline in order to make room for the outslopes.

Make GRiD File Dialog

After selecting the limits of the disturbed area the program will generate a 3D grid that represents the surface. Specify the grid resolution desired and select OK.

Pick the spillway centerline: *select polyline that crosses the dam*

Pick a point within the pond: *pick a point* The program needs to know which end of the spillway centerline is within the pond.

Enter slopes as percent grade or slope ratio (Percent/<Ratio>)? *press Enter*

Enter the side slope ratio <1.0>: *press Enter*

Enter the flow slope ratio <100.0>: *press Enter*

Range of existing elevations along spillway centerline.

Enter spillway elevation <1476.5>: *1475.0* This is the entrance elevation of the spillway

Enter the spillway width <10.0>: *press Enter*

Spillway Report:

Spillway inlet elevation: 1445.0000

Spillway outlet elevation: 1445.0000

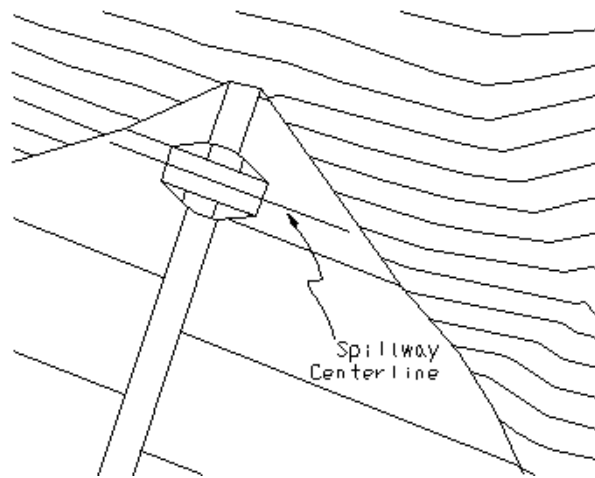
Spillway width: 10.0000

Side slope percent grade: 100.00, slope ratio: 1.00

Flow slope percent grade: 1.00, slope ratio: 100.00

Spillway EarthWork Volumes

Total cut: 55.593 C.Y., 1501.00150 C.F.



Spillway added to valley pond

Pulldown Menu Location: Structure in Hydrology

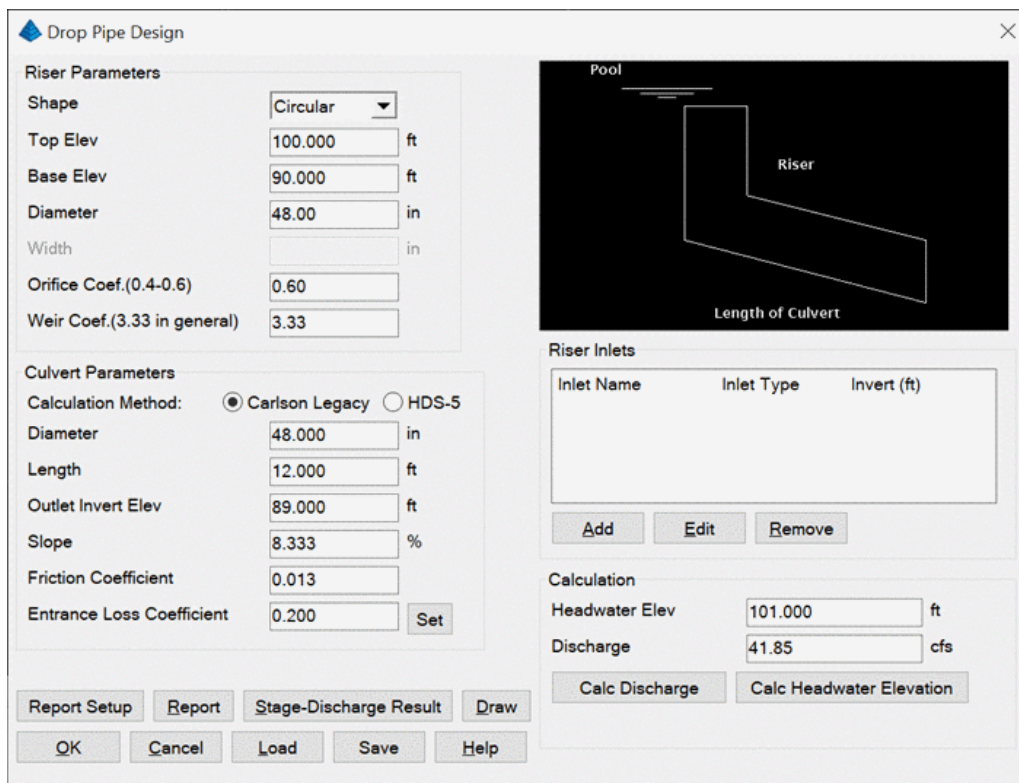
Keyboard Command: spill

Prerequisite: Surface entities that model the pond

Drop Pipe Spillway Design

This program calculates the spillway discharge at different water elevations. When the water is lower than the top of the riser, the Riser Inlets would take water when the water rise above the inlet's invert elevation. As the water elevation initially rises above the riser, the flow is controlled by weir flow. The program uses the perimeter of the riser for the weir length for the weir flow calculation. The riser can be either circular or box. At higher water elevations the flow is under orifice control. When the barrel flows full, the flow is controlled by full pipe flow. Given the water elevation and spillway dimensions, the program calculates the type of flow and discharge.

The Riser Inlets are orifice/weir openings on the riser and discharge into the riser box, and then to the culvert. There are two methods to calculate culvert discharge - Carlson Legacy and HDS-5.



The Calculate button will read the values in the dialog, calculate the flow and report this flow value at the bottom of the dialog. The Report button will generate a report of the input values and calculated flows. The File routine will create a stage-discharge (.STG) file. The Draw function will draw and label the drop pipe spillway in the drawing at the specified scale. The Graph button creates a stage-discharge graph.

Riser Parameters: Indicate the dimension of the vertical riser pipe, and the orifice and weir coefficients which are used in orifice and weir flow calculations.

Culvert Parameters: Indicate the culvert dimension and profile. The Friction Coefficient is the manning's n for calculating the pipe friction loss. The Entrance Loss Coefficient is used to calculate the energy loss at the culvert inlet end.

Headwater Elev: Indicate the water (pool) elevation.

Discharge: Discharge value at the end of the culvert.

Calc Discharge: Causes the routine to read the values in the dialog and calculate the Discharge flow at the current headwater elevation.

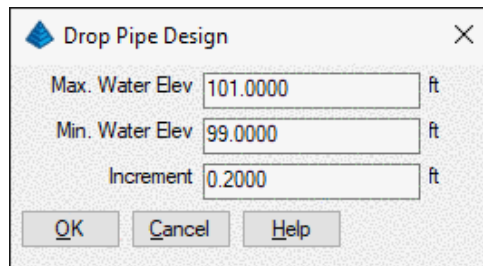
Calc Headwater Elevation: Causes the routine to read the values in the dialog and calculate the headwater elevation that's required to handle the known discharge flow.

Report Setup: Chooses between a Standard Report and a PDF Report. The Standard Report is a simple text report of the parameters and calculations. The PDF Report adds some text formatting and has the option to draw a schematic of the pipe detail and draw the stage-discharge graph.

Report: Creates a report for the current spillway.

Load/Save: Store and recall the parameters to a .drop file.

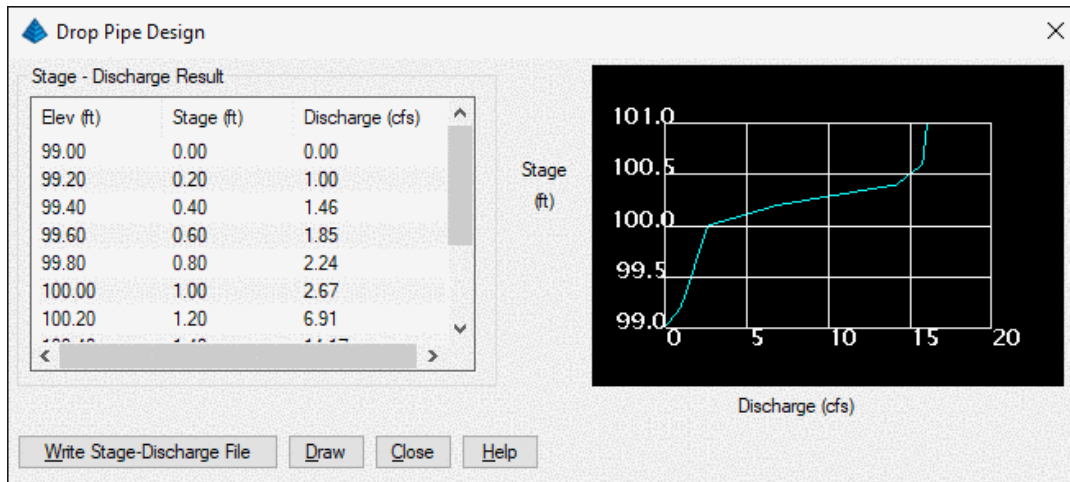
Stage-Discharge Result: Displays the following dialog box that allows you to indicate the water elevation range and increment to report:



The 'Drop Pipe Design' dialog box contains three input fields for elevation values and three buttons at the bottom.

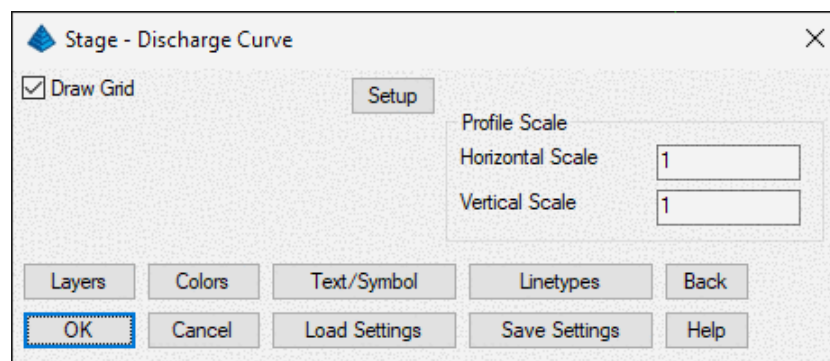
Max. Water Elev	101.0000	ft
Min. Water Elev	99.0000	ft
Increment	0.2000	ft

Buttons: OK, Cancel, Help



Write Stage-Discharge File: Creates a Stage-Discharge (.STG) file that can be used in other Carlson Hydrology commands.

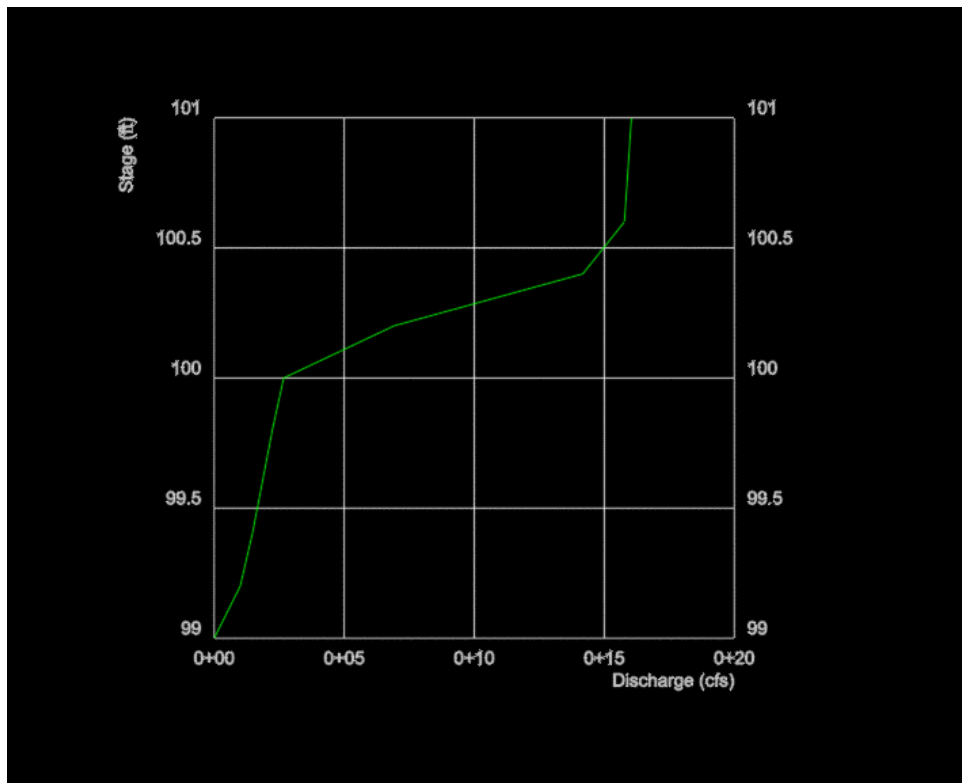
Draw: Allows a graphic representation of the Stage-Discharge data to be placed into the drawing.



The 'Stage - Discharge Curve' dialog box includes a 'Draw Grid' checkbox, a 'Setup' button, and a 'Profile Scale' section with 'Horizontal Scale' and 'Vertical Scale' input fields. At the bottom are buttons for 'Layers', 'Colors', 'Text/Symbol', 'Linetypes', 'Back', 'OK', 'Cancel', 'Load Settings', 'Save Settings', and 'Help'.

Profile Scale:
Horizontal Scale: 1
Vertical Scale: 1

Buttons: Layers, Colors, Text/Symbol, Linetypes, Back, OK, Cancel, Load Settings, Save Settings, Help



Draw Grid: When enabled, this option creates a grid that accompanies the Stage-Discharge curve. Click the **Setup** button to access the grid settings which are discussed in detail in the Draw Grid section of the Draw Profile command.

Output to Separate Drawing: When enabled, this option draws the hydrograph to a separate drawing. Click the **Set** button to specify the name/location of the external drawing.

Horizontal Scale: Specify the discharge scale (in CFS).

Vertical Scale: Specify the stage scale (in Ft).

The **Layers, Colors, Text Styles/Sizes** and **Linetypes** buttons provide access to settings for each of these features of the Stage-Discharge curve.

Load Settings: Loads a saved collection of Draw Stage-Discharge settings, saved in a (.PFS) file.

Save Settings: Saves all Draw Stage-Discharge settings in a (.PFS) file.

Pulldown Menu Location: Structure

Keyboard Command: spillway

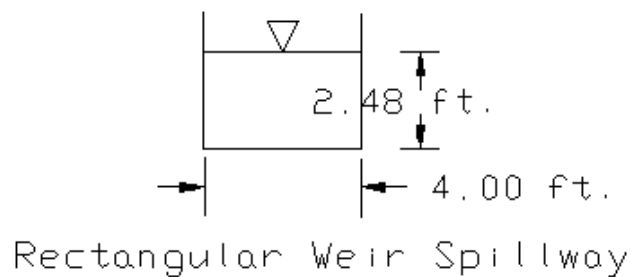
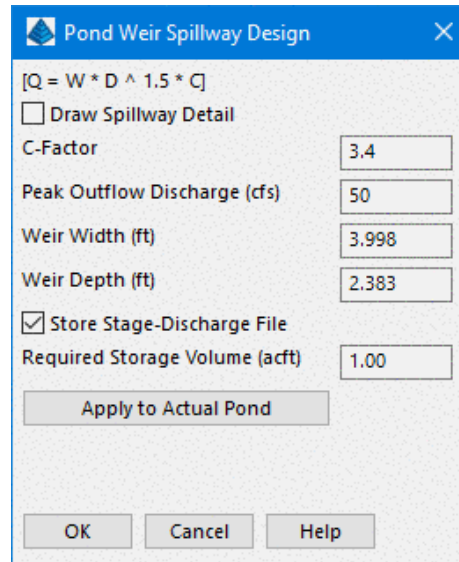
Prerequisite: None

Rectangular Weir Design

This program calculates the dimensions of a rectangular weir given the outflow discharge. The default discharge uses the value from the Detention Pond command. The weir width and depth are two free variables. Enter a value for one and press Enter. Then the value for the other is calculated.

The weir design may optionally be applied to a pond design. First enter a Required Storage Volume which can come from the Detention Pond command. Then click Apply to Actual Pond and choose a Storage Capacity File (.CAP). This .cap file can be created by Bench or Valley Pond Design and by the Stage-Storage command. The program then computes the elevation at the required storage volume and the corresponding elevation for the bottom of the weir given the weir depth.

When the Draw Spillway Detail option is checked, a drawing of the weir is created as shown below.



Pulldown Menu Location: Structure in Hydrology

Keyboard Command: weir

Prerequisite: None

Advanced Weir Design

The Advanced Weir Design uses the methodology described in HEC-22 Manual. The weir flow is determined as:

$Q = C_w L H^{1.5}$ for the Rectangular Weir without Contracted End

$Q = C_w (L - 0.2 H)^{1.5}$ for the Rectangular Weir with Contracted End

where: Q = discharge, ft^3/s (m^3/s)

C_w = weir coefficient, 3.33 in English units (1.84 in Metric units)

L = weir length, ft (m)

H = head above weir crest, ft (m)

$Q = C_w [\tan(\emptyset/2) H^{2.5} + 5/4 L H^{1.5}]$ for Trapezoidal Weir

where: Q = discharge, ft^3/s (m^3/s)

C_w = weir coefficient, 3.33 in English units (1.84 in Metric units)

L = weir length, ft (m)

\emptyset = internal angle of the two sides, degrees

H = head above weir crest, ft (m)

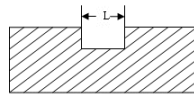
$Q = C_w \tan(\emptyset/2) H^{2.5}$ for V-Notched Weir

where: Q = discharge, ft^3/s (m^3/s)

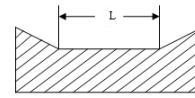
C_w = weir coefficient, 2.5 in English units (1.38 in Metric units)

θ = angle of v-notch, degrees

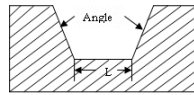
H = head above weir crest, ft (m)



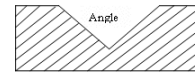
Rectangular Weir with Contracted End



Rectangular Weir without Contracted End



Trapezoidal Weir



V-Notched Weir

This command designs a weir structure and calculates its stage-discharge curve. Select Weir Design from the Structure menu in the Hydrology Module to display the design dialog. Select the Type of the weir, Rectangular, Trapezoidal or V-notched. Enter the dimension for the weir. In the Invert Elev box, type the absolute elevation at which the weir will be attached to a reservoir. The attachment point is at the bottom of the weir. In the Coefficient box, type a weir coefficient value. In the Number of Openings box, enter the number of weirs you want to combine.

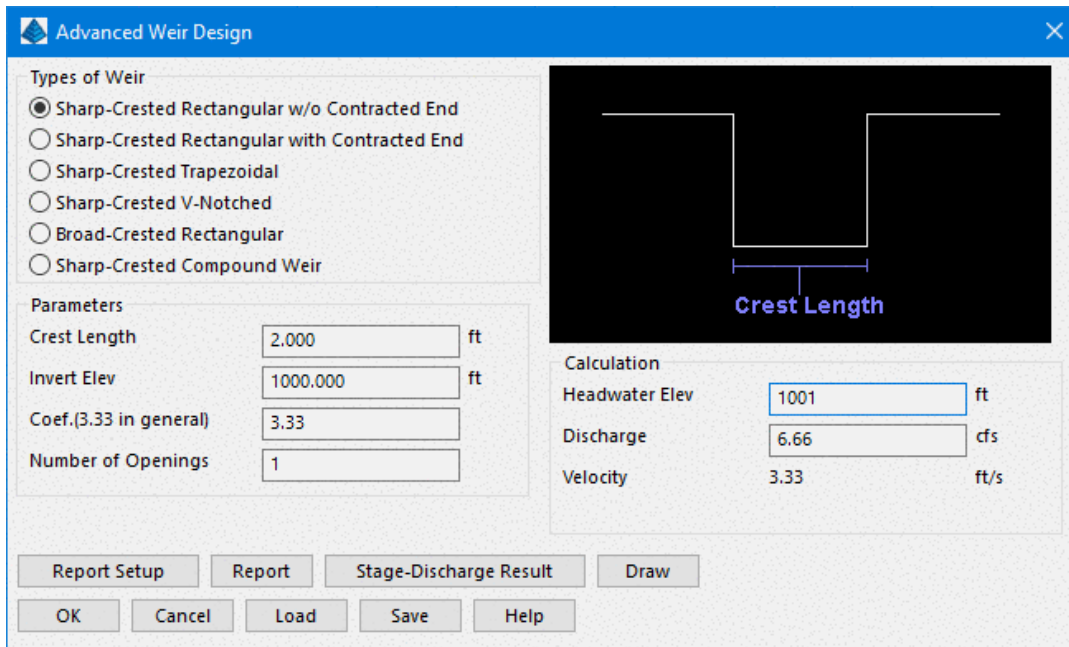
The program performs two-way calculation: calculate Discharge from Headwater, and calculate Headwater from Discharge. In the Headwater and Tailwater boxes, type the absolute headwater surface elevation and tailwater surface elevation respectively, and the maximum discharge and flow velocity through the orifice would be computed automatically. When Discharge value is entered, the Headwater elevation would be calculated automatically.

Click on the Stage-Discharge Result button to display the stage-discharge curve in the Stage-Discharge Result Dialog. This dialog allows you to write the stage-discharge data to a stage-discharge file(.STG), and draw the stage-discharge curve on the screen. From the Stage-Discharge Curve Draw Settings dialog, you can draw the curve into the drawing or write a .stg stage-discharge file.

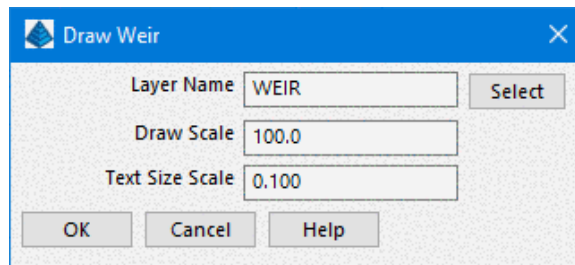
The Report Setup chooses between the Standard Report and the PDF Report. The Standard Report is a simple text report of the parameters and results. The PDF Report adds some text formatting and options to draw a schematic of the weir and draw a stage-discharge graph. The Report button generates the weir design report.

The Draw button creates a schematic of the weir with dimensions in the drawing.

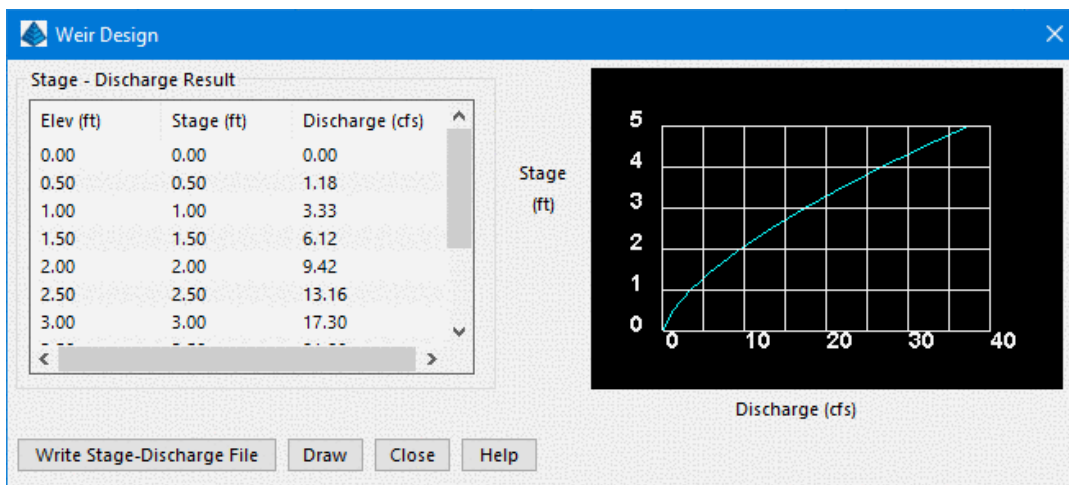
The Load and Save buttons store and recall the parameters to a .weir file.



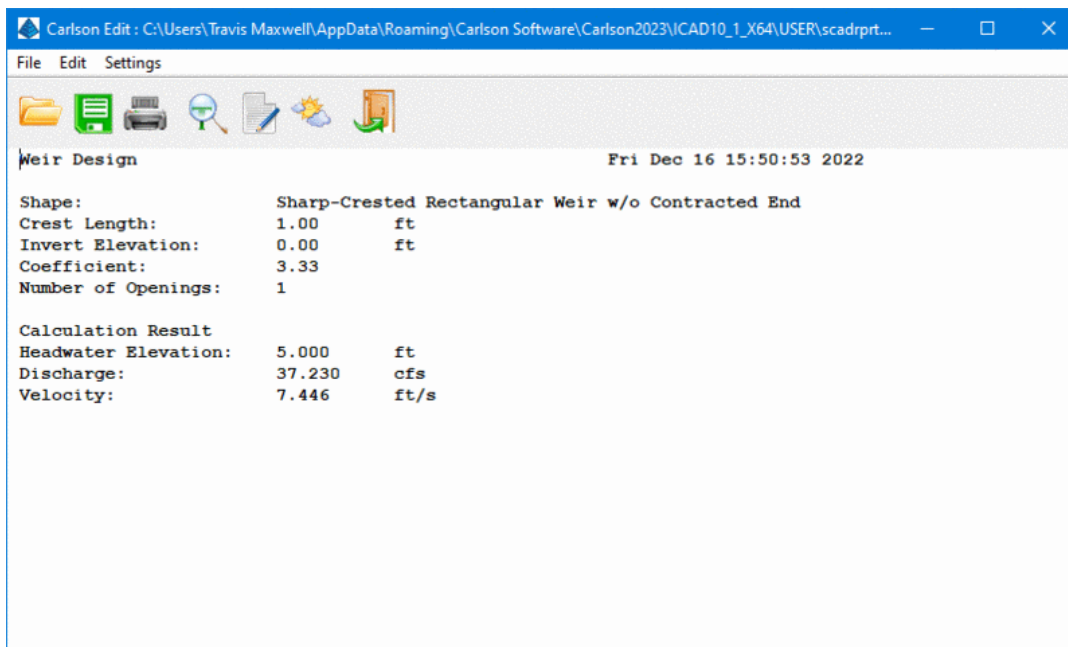
Advanced Weir Design Dialog



Stage-Discharge Limits Dialog



Stage-Discharge Result Dialog



Weir Design Report

Pulldown Menu Location: Structure > Advanced Weir Design

Keyboard Command: weir

Prerequisite: None

Orifice Design

The Orifice Design uses the methodology described in HEC-22 Manual. The orifice flow is determined as:

$$Q = C_o A_o (2 g H_o)^{0.5}$$

where: Q = discharge, ft³/s (m³/s)

C_o = orifice coefficient, unitless (0.40 - 0.60)

A_o = area of orifice, ft² (m²)

H_o = effective head on the orifice measured from the centroid of the opening, ft (m)

g = gravitational acceleration, 32.2 ft/s² (9.81 m/s²)

This command designs an orifice structure and calculates its stage-discharge curve. Select Orifice Design from the Structure menu in the Hydrology Module to display the design dialog. Select the Section Type of the orifice, Circular or Rectangular. Enter the dimension for the orifice. In the Invert Elev box, type the absolute elevation at which the orifice will be attached to a reservoir. The attachment point is at the bottom of the orifice. In the Coefficient box, type a roughness coefficient for the orifice. The coefficient ranges from 0.4 to 0.6 (HEC-22). In the Number of Openings box, enter the number of orifices you want to combine.

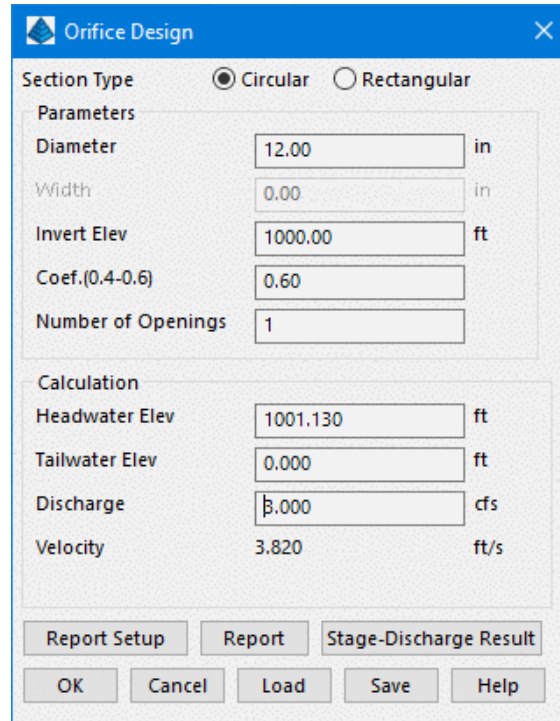
The program performs two-way calculation: calculate Discharge from Headwater, and calculate Headwater from Discharge. In the Headwater and Tailwater boxes, type the absolute headwater surface elevation and tailwater surface elevation respectively, and the maximum discharge and flow velocity through the orifice would be computed automatically. When Discharge value is entered, the Headwater elevation would be calculated automatically.

Click on the Stage-Discharge Result button to display the stage-discharge curve in the Stage-Discharge Result Dialog. This dialog allows you to write the stage-discharge data to a stage-discharge file(.STG), and draw the

stage-discharge curve on the screen. From the Stage-Discharge Curve Draw Settings dialog , you can specify how to draw the curve.

The Report Setup chooses between the Standard Report and the PDF Report. The Standard Report is a simple text report of the parameters and results. The PDF Report adds some text formatting and has the option to draw a stage-discharge graph. The Report button generates the orifice design report.

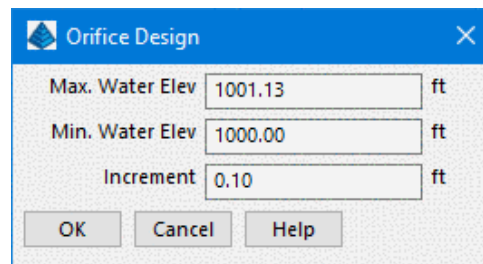
The Load and Save buttons store and recall the parameters to a .outlet file.



The Orifice Design dialog box is titled "Orifice Design" and has a close button (X) in the top right corner. It is divided into several sections:

- Section Type:** Two radio buttons are present: "Circular" (selected) and "Rectangular".
- Parameters:** A group box containing five input fields:
 - Diameter: 12.00 in
 - Width: 0.00 in
 - Invert Elev: 1000.00 ft
 - Coef.(0.4-0.6): 0.60
 - Number of Openings: 1
- Calculation:** A group box containing four input fields:
 - Headwater Elev: 1001.130 ft
 - Tailwater Elev: 0.000 ft
 - Discharge: 3.000 cfs
 - Velocity: 3.820 ft/s
- Buttons:** A row of five buttons: "Report Setup", "Report", "Stage-Discharge Result", "OK", "Cancel", "Load", "Save", and "Help".

Orifice Design Dialog

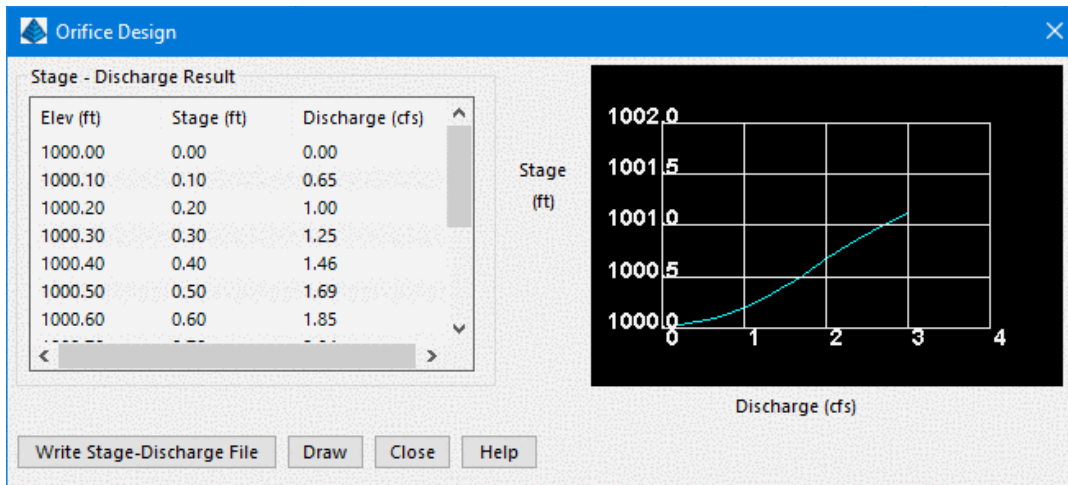


The Stage-Discharge Limits dialog box is titled "Orifice Design" and has a close button (X) in the top right corner. It contains three input fields:

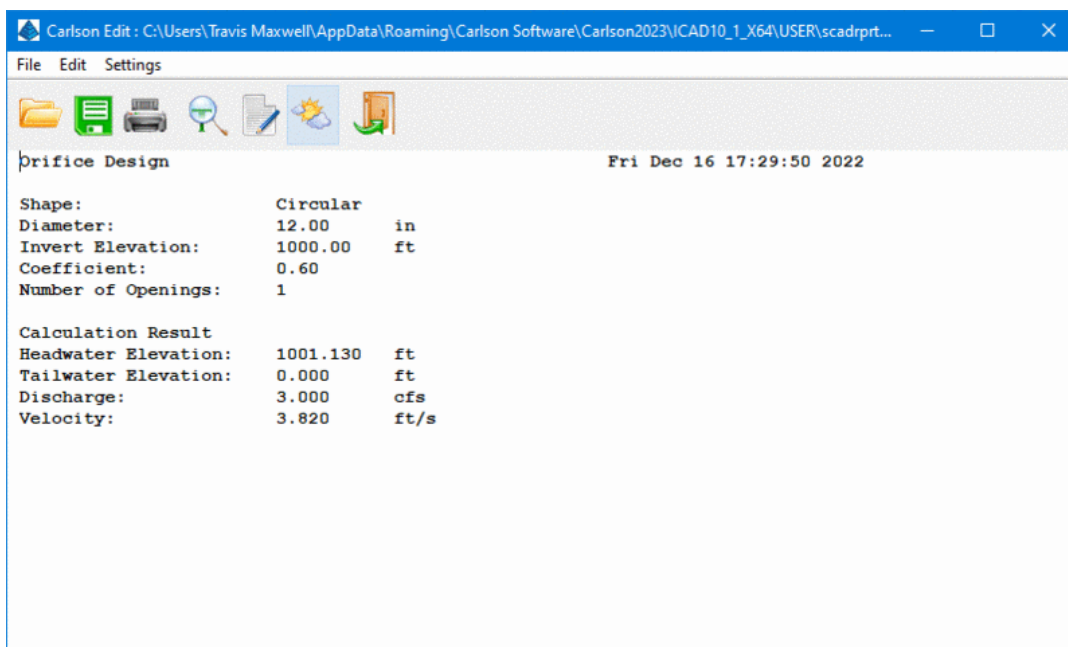
- Max. Water Elev: 1001.13 ft
- Min. Water Elev: 1000.00 ft
- Increment: 0.10 ft

At the bottom, there are three buttons: "OK", "Cancel", and "Help".

Stage-Discharge Limits Dialog



Stage-Discharge Result Dialog



Orifice Design Report

Pulldown Menu Location: Structure > Orifice Design

Keyboard Command: orifice

Prerequisite: None

Pond Exfiltration Design

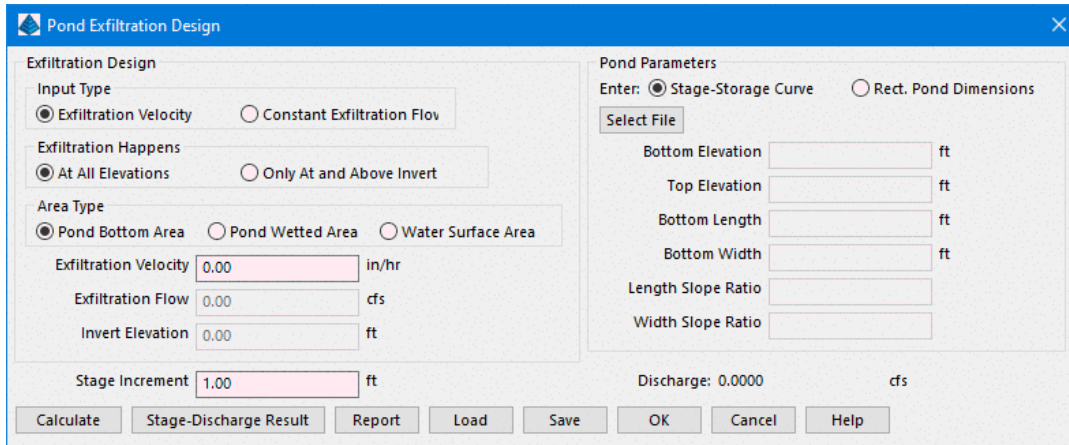
Exfiltration is classified as an outlet device and is incorporated into a pond's stage-discharge curve. But exfiltration is usually directed to the discarded outflow to prevent further routing. There are two design methods to determine exfiltration, shown below:

1. Constant exfiltration Flow: The flow value (cfs or cms) is applied whenever there is water in the pond, or only when the water level is at or above the specified invert elevation.
2. Exfiltration Velocity: The velocity (in/hr or mm/hr) is multiplied by the exfiltration area at a given elevation to determine the final exfiltration flow.

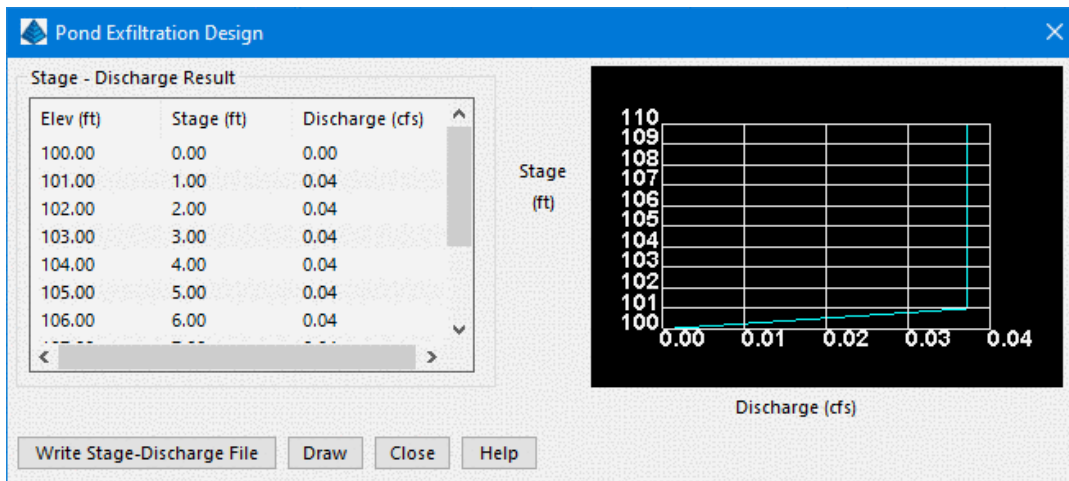
The exfiltration area may be defined in three ways: 1) Pond Bottom Area, which allows the exfiltration only through the bottom of pond; 2) Water Surface Area, which assumes the exfiltration is downward; 3) Pond Wetted Area, which allows the exfiltration occur through all exposed surfaces regardless of slope. You may also choose to exclude the area of the pond that lies at or below the specified invert elevation, which is useful in preventing exfiltration through impervious (lower) regions of the pond.

There are two ways to enter the pond parameters. If you elect to input the stage-storage curve, the program will determine the bottom and side areas automatically.

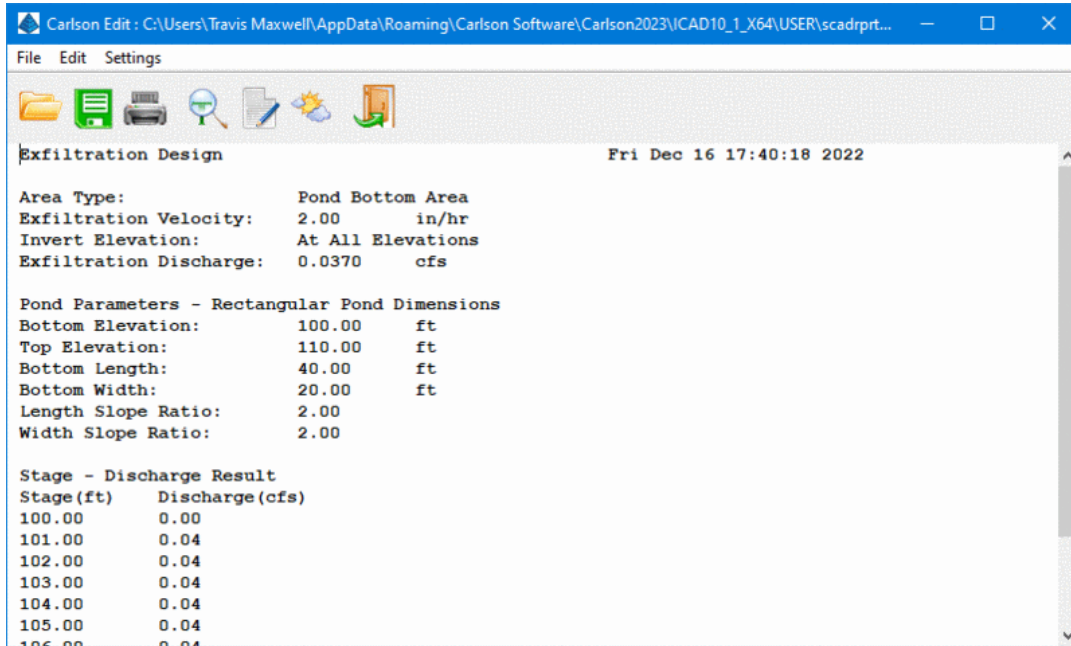
The Load and Save buttons store and recall the parameters to a .pxd file.



Pond Exfiltration Design Dialog



Stage-Discharge Result Dialog



Exfiltration Design Report

Pulldown Menu Location: Structure > Pond Exfiltration Design

Keyboard Command: exfilcalc

Prerequisite: None

Multiple Outlet Design

Multiple Outlet Design attaches multiple outlet structures to a detention pond and computes the stage-discharge data. All design parameters are stored in a combo outlet file (.COT). This command allows you to add, edit and remove outlet structures attached to a pond, and you can see how different combinations of structures affect the stage-discharge calculation. There are six types of outlet structures: culvert, drop pipe, orifice, weir, exfiltration and user defined stage-discharge curve. Please refer to the documentation of Culvert Design, Drop Pipe Spillway Design, Orifice Design, Weir Design and Stage-Discharge Editor for the details.

From the Structure menu in the Hydrology Module, choose Multiple Outlet Design command to open the design dialog. You need to specify an existing or a new combo outlet file (.COT) to load it into the design dialog. The dialog has a spreadsheet with a list of all the outlets. You can edit the Outlet Name, Invert Elevation and Outlet Type right in the spreadsheet. Or use the Edit button to bring up a dialog editor for the current highlighted outlet. The Used In Design toggle allows you to turn on and off whether to include the outlet in the discharge calculations.

Click on Add button to display the New Outlet Structure dialog. Enter the structure name and select a structure type, then click the OK button to display the outlet design dialog for that structure. Configure the structure in the design dialog and then click OK to exit the dialog and come back to the main dialog, which highlights the new structure and indicates its parameters and the stage-discharge result.

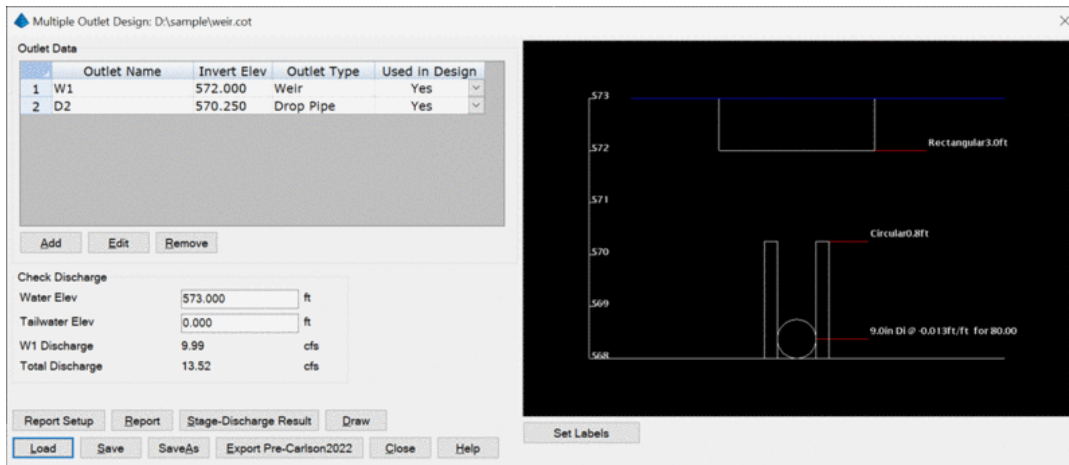
The Edit and Remove buttons allow you to edit and remove the highlighted outlet. The Stage-Discharge Result button computes the discharges at each stage from the minimum water elevation to the maximum, and displays the result in the Stage-Discharge Result Dialog.

The Report Setup chooses between the Standard Report and PDF Report. The Standard Report is a simple text report of the design parameters and the discharge results that is shown in the standard Carlson report display window, from where the information can be edited, saved, and printed to a printer or to the screen. The PDF Report adds some text formatting and options to draw a diagram of the outlets and draw a stage-discharge graph.

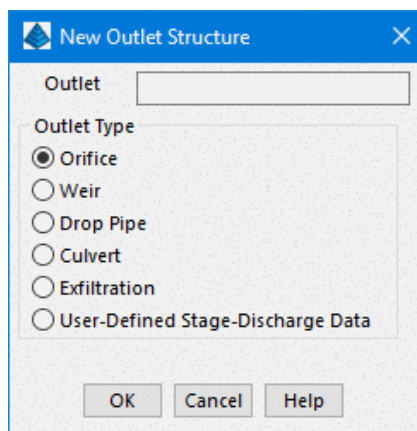
The Load button allows you to load other combo outlet files (.COT) for editing. The Save and SaveAs buttons save the current design parameters to an outlet file.

The Check Discharge section allows you to enter a Water Elevation and Tailwater Elevation and the program reports the total discharge from all the outlets along with the discharge for the currently highlighted outlet.

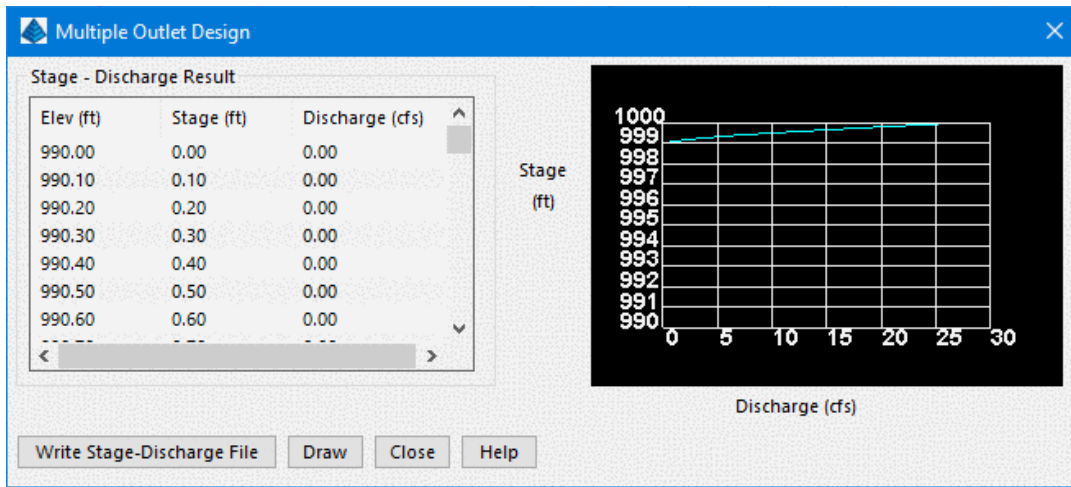
The schematic view of the pond and its outlets as well as the water elevation are shown on the right side. The Set Labels button opens the label settings dialog to set up the labels in the view.



Multiple Outlet Design Dialog



New Outlet Structure Dialog



Stage-Discharge Result Dialog

	Position	Prefix	Suffix	Sequence
<input checked="" type="checkbox"/> Label Pipe Length	Below Pipe	for	'	4
<input checked="" type="checkbox"/> Label Pipe Size	Below Pipe		in Di @	1
<input checked="" type="checkbox"/> Label Pipe Material	Below Pipe			3
<input checked="" type="checkbox"/> Label Pipe Slope	Below Pipe		%	2
<input checked="" type="checkbox"/> Label Pipe Name	Above Pipe			5
<input checked="" type="checkbox"/> Label Pipe Invert	Above Pipe	elev=	'	6

Slope Format
 Percent Ft/Rt

Pulldown Menu Location: Structure > Multiple Outlet Design

Keyboard Command: poutlet

Prerequisite: None

Stage-Storage Editor

This command defines a reservoir by entering stage/storage data in five ways: stage/storage (storage-area), stage/storage(by volum,e), stage/contour area data, rectangular/trapezoidal pond definition and underground pipe definition.

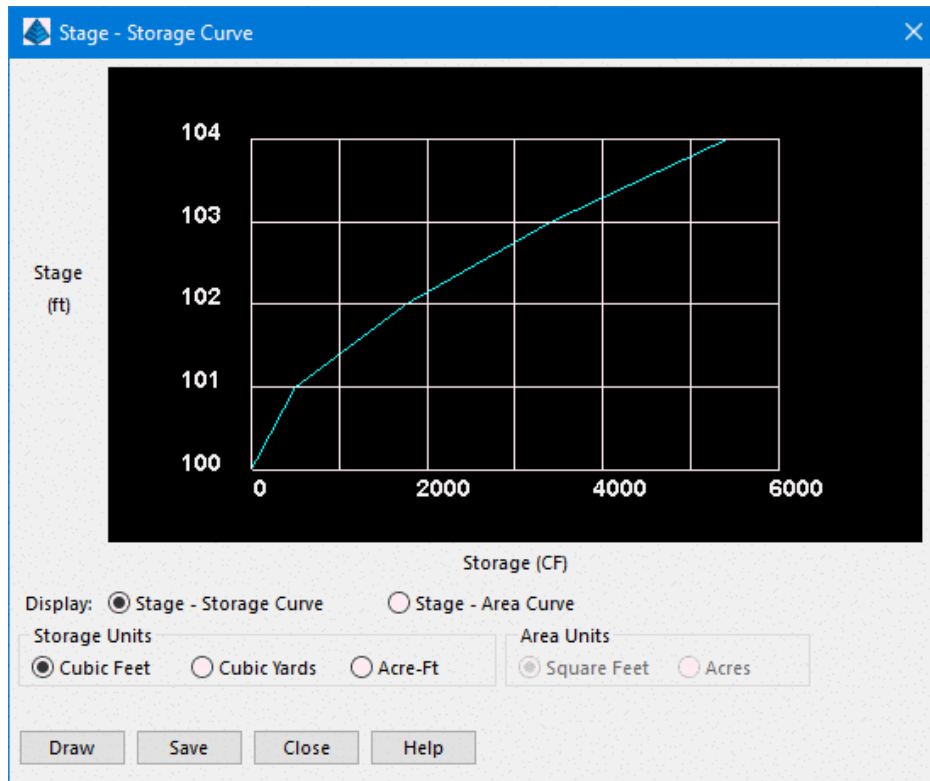
From the Structure > Stage-Storage menu in the Hydrology Module, run Input-Edit Stage-Storage command to open the design dialog. Enter the pond name in the Structure Name box. There are five Storage Methods: **User Defined Storage-Volume** allows you to manually enter stage-storage or stage-volume data, **Stage-storage or stage-Area**, **Irregular Shape** allows you to select the contours of a surface model from a drawing, **Rectangular Shape** defines a rectangular or trapezoidal pond, and **Underground Pipe** defines circular or box pipe shape reservoirs.

The **Biofiltration** is at the bottom of the pond that isn't 100% water capacity. The depth is from the bottom of the pond, and the void is smaller than 100%, in the example below it is 50%. The Biofiltration is calculated in the storage and shown in the stage-storage report.

Stage-Storage Data section displays the stage-storage curve data. Click on the **Edit Detention Structure** button to create/edit the stage-storage input. The **Check Storage** fields are a tool for entering the elevation or stage and reporting the storage and area at that level. Load, Save and SaveAs buttons allow you to load and save the stage-storage data. The Report button generates the stage-storage report in either **Standard Report** or **PDF Report** format, shown below. For a customized report, check the Use Report Formatter option and then pick Report.

Stage-Storage Editor

Stage-Storage Editor



Stage-Storage PDF Report

User Defined Storage-By Area

In the spreadsheet, you can enter elevations and corresponding pond surface area to define stage storage. You can also import an exiting spreadsheet .csv file via **Import CSV** button. Click on **OK** button to save the stage-storage data.

User Defined Storage-By Volume

In the spreadsheet, you can enter elevations and corresponding storage volumes to define stage storage. You can also import an exiting spreadsheet .csv file via **Import CSV** button. Click on **OK** button to save the stage-storage data.

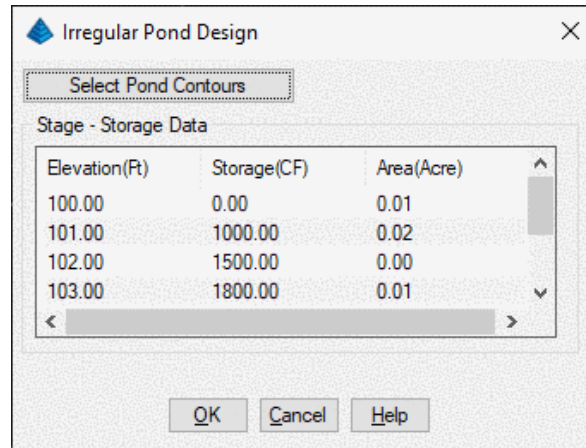
	Elevation (Ft)	Area (SF)
1	100	0
2	101	1000
3	102	1500
4	103	1800
5	104	2200
6		

Buttons: Insert Row, Delete Row, OK, Cancel, Help, Import CSV

User Defined Storage

Irregular Shape

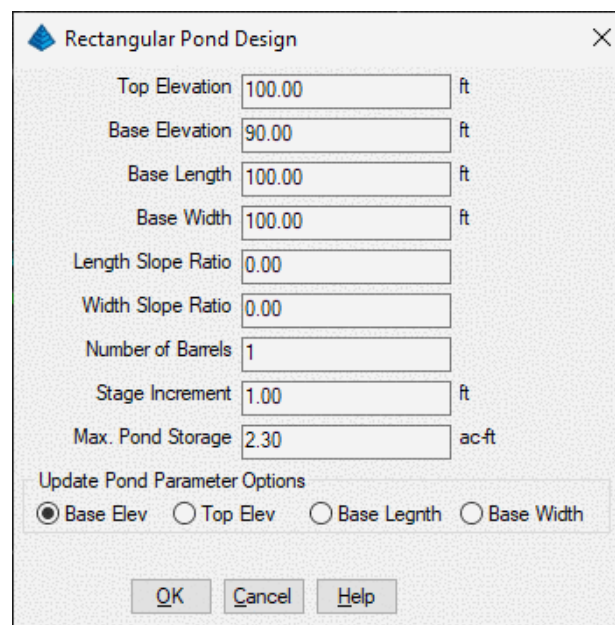
This method allows you to define an irregular shape pond from the contour polylines of a surface model, and generate the stage-storage data automatically. In order to use this method, you must have the surface drawing open, which contains the contours that you want to use to define the reservoir. Click on the **Select Pond Contours** and select as many contours as you need, the stage/storage and stage/area relationship will be then determined and displayed in the Stage-Storage Data table, starting from the lowest elevation to the maximum. Click on OK button to save the data.



Irregular Pond Design

Rectangular Shape

This method allows you to define a rectangular box or trapezoidal shape reservoir. Enter values in the Top Elevation, Base Elevation, Base Length and Base Width boxes. If you want to define a trapezoidal shape reservoir, enter the Length Slope Ratio and Width Slope Ratio. Multiple barrels are allowed in this type of pond. You also need to specify the Stage Increment. The **Max. Pond Storage** is the design capacity. When change the Max. Pond Storage value, the parameter selected in the Update Pond Parameter Options is updated automatically. Click on **OK** button to save pond parameters.



Rectangular Pond Design

Underground Pipe

This method allows you to specify a reservoir as a pipe. Pipes come in circular and rectangular shapes. Enter the pipe dimensions and the Invert Elev at which the pipe is located. Specify the Number of Barrels and the Stage

Increment. The **Stone Encasement** is around the pipe for additional storage. Click on OK button to save the pipe parameters.

Underground Pipe Design

Pipe Shape Circular Box

Diameter in

Width in

Length ft

Invert Elev ft

Number of Barrels

Use Stone Encasement

Stone Bottom Elev ft

Stone Width ft

Stone Depth ft

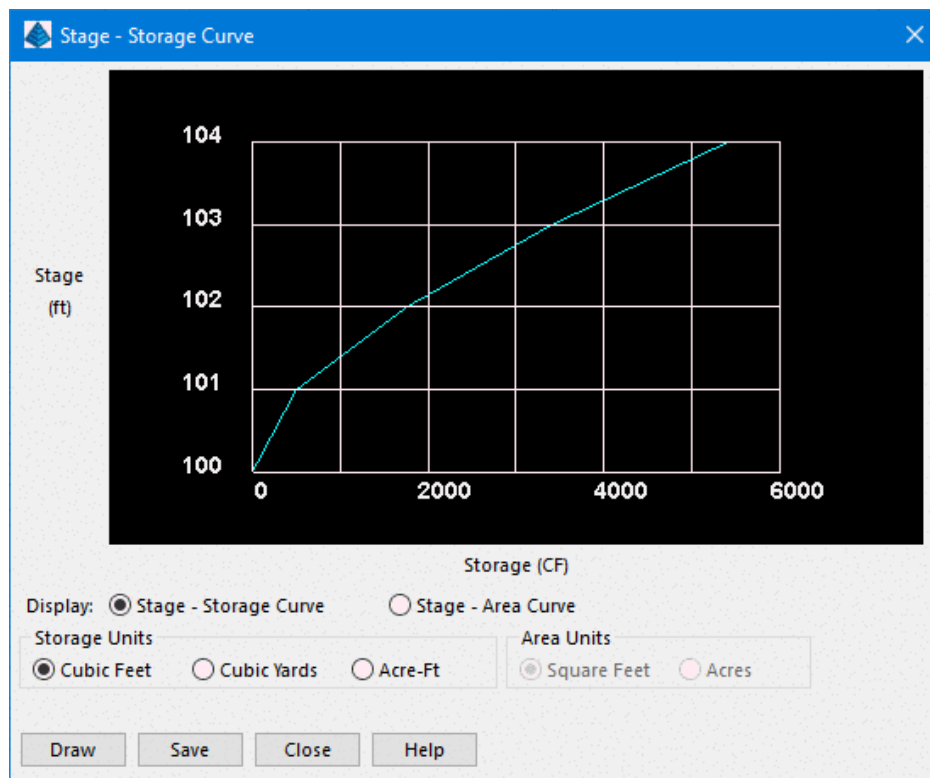
Stone Voids (%)

Stage Increment ft

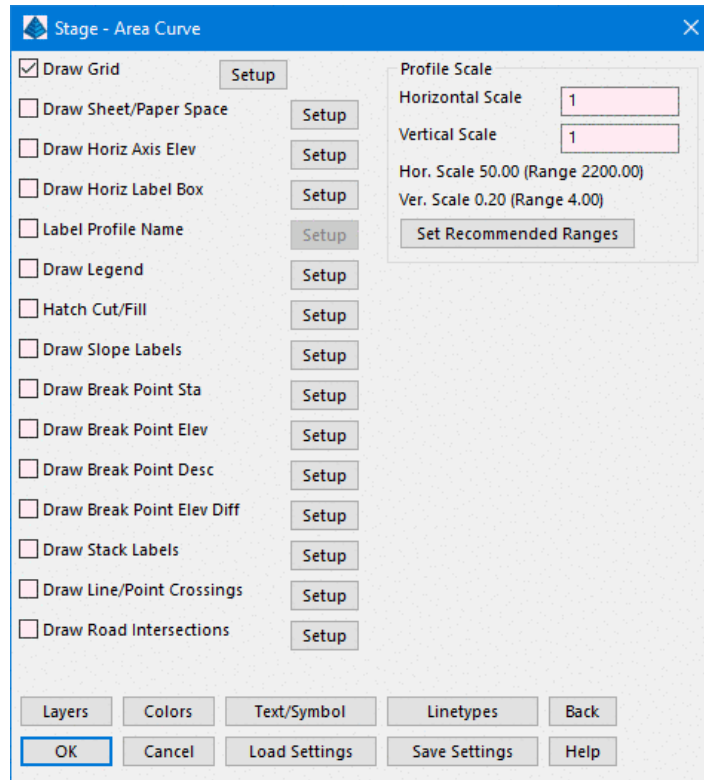
Underground Pipe Design

Stage-Storage Curve

When you click on the Graph button, the Stage-Storage Curve dialog displays. An image is shown for you to view the stage vs. storage, stage vs area plot for the reservoir. The graph can be plotted into the CAD graphic by clicking on Draw button. When you click on the Draw button, the Stage-Storage Curve Settings dialog displays from where you can define how to plot the text and graph on screen.



Stage-Storage Curve



Stage-Storage Curve Draw Settings

Pulldown Menu Location: Structure > Stage-Storage

Keyboard Command: edit_stage_store

Prerequisite: a stage-storage file (.CAP)

Calculate Stage-Storage

This command calculates stage-storage values for a pond that is already drawn in the drawing. Before running this routine, the surface model for the pond must be created as a triangulation file with Triangulate & Contour or as a grid file with Make 3D Grid File. A closed polyline for the perimeter of the pond is also required. It has the option to save a stage-storage capacity file, in one of 2 forms (Carlson for readability, or Sedcad form, for importing into Sedcad). The type of file stored is set in Configure, Hydrology Module.

Prompts

Select Pond Surface

Select the .tin, .flt or .grd file that models the pond surface.

Pick the top of dam polyline: *pick the closed polyline perimeter*

Choose method to specify storage elevations (<Automatic>/Interval/Manual)? *Manual*

Range of pond elevations: *1202 to 1220*

Sediment Elevation (Enter for none): *1206*

Enter stage elevation (Enter for none): *1206*

Enter stage elevation (Enter for none): *1210*

Enter stage elevation (Enter for none): *1215*

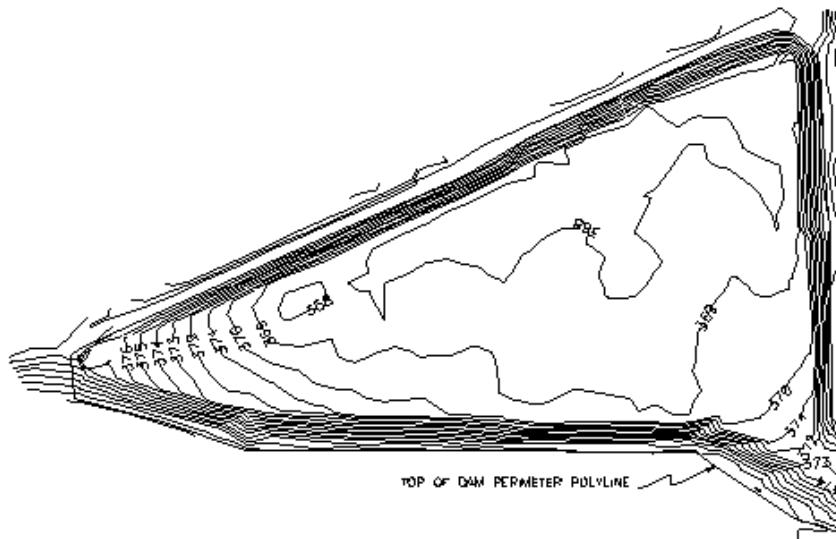
Enter stage elevation (Enter for none): *1220*

Enter stage elevation (Enter for none): *press Enter*

Pond Storage Volumes Report

Pond Storage Volumes		11/23/2004 21:52		
Sediment Elevation: 1206.000				
Sediment Volume: 873.529 C.Y.				
Water Elev	Storage(AcreFt)	(C.Y.)	(C.F.)	Area(Acre)
1206.00	0.00000	0.0	0.0	0.141
1210.00	0.67919	1095.8	29585.4	0.208
1215.00	1.96486	3170.0	85589.4	0.322
1220.00	3.92578	6333.6	171007.1	0.476

Write stage-storage to file [Yes/<No>]? *press Enter* When saving a stage-storage capacity file, to be drawn in the Draw Stage-Storage Curve command, it is a good idea to limit the number of stages to a reasonable number, such as 6 to 12 stages. More than 12 will plot off the page in a long list, unless you use the option "Skip Every 2nd Table Entry". If the stages are at odd intervals, the Draw Stage-Storage Curve command will interpolate additional stages, so reducing the number of stages used works best for plotting.



Typical Pond for Calculate Stage Storage

Pulldown Menu Location: Structure in Hydrology

Keyboard Command: postpond

Prerequisite: Surface entities that model the pond

Draw Stage-Storage Curve

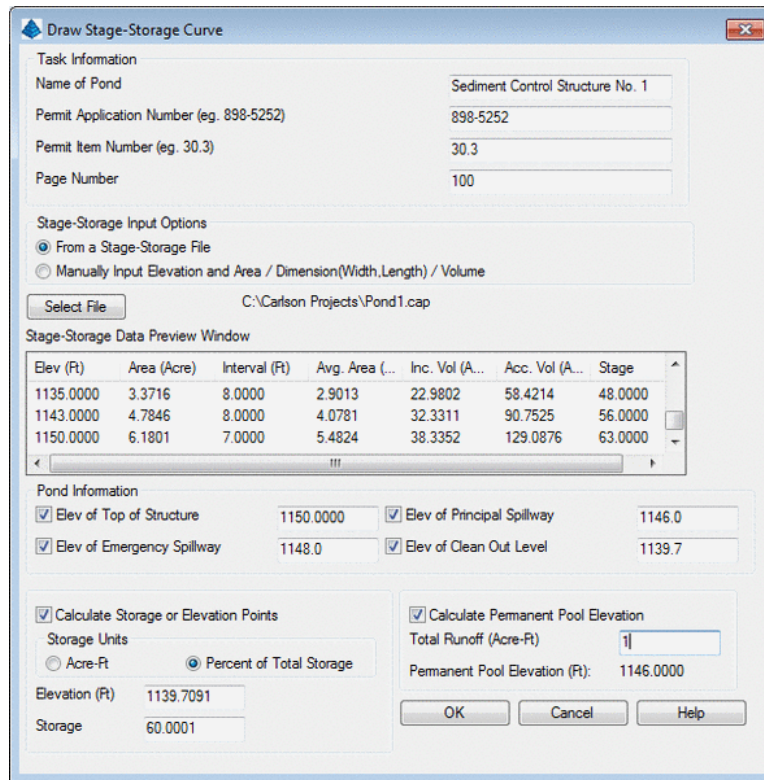
This routine draws a pond stage storage curve with pond elevation on the vertical axis and acre-feet of storage on the horizontal axis. It will plot and label the emergency spillway, principal spillway and cleanout levels and will produce a table of storage data. There is an option to plot the Stage-Area curve on the same graph. The program will read a *.CAP capacity file of pond storage, based on the surface area at each stage or elevation. CAP files (short for "capacity") are made by Design Bench Pond, Design Valley Pond, Rectangular Pond Design, Calculate Stage Storage and Calculate Pond/Pit Volumes.

In addition to CAP file-based inputs, you can enter pond dimensions directly by length-width, area at each stage, or volume at each stage. If stage-storage curves are loaded from file, which contains only volumes at different

stages, then the width and length columns are filled in as "N/A" (not applicable). Since volume-based entry does not include area information, no CAP files are stored with this option. However, the curves plot in all cases. Plots are sized to fit on standard letter sized 8.5" x 11" sheets at the selected scale for plotting. They are particularly suited for permit applications, so the program will prompt for permit number and page.

Prompts

The program is dialog-driven. The first dialog controls file loading and some pre-calculation options, and is shown below:



Task Information

This section allows for entry of the Name of the Pond, Permit numbers and the page number. These fields will be labeled on the final plot, ready to be inserted into a report.

Stage Storage Input Options

- **From a Stage-Storage File:** Using this option, load the CAP file with the Select File button. This will populate the preview window and fill in the top of structure elevation. If the goal is to set the emergency spillway at an elevation with storage at a certain acre-feet, enter the storage in the lower left, hit the Enter key, and calculate the appropriate elevation. A total runoff of the entered acre feet, subtracted from the acre-feet at the principal spillway, will set the recommended elevation of the "clean out level". If the pond silts up above that level, then the silt needs to be removed. For example, the minimum vertical separation between principal spillway and clean out level is 1.5 feet in some regions. To compute permanent pool elevations, enter the runoff quantities and hit the Enter key.

- **Elev of Top Structure:** This value is entered automatically, read in from the CAP file.
- **Elev of Emergency Spillway:** Enter this value either from the design, or from the Calculate Storage or

Elevation Points calculation.

- **Elev of Principal Spillway:** Enter this value either from the design, or from the Calculate Storage or Elevation Points calculation.
 - **Elev of Clean Out Level:** This is a value calculated usually by the Percent of Total Storage, or storage below the principal spillway, such as 60% of storage capacity.
 - **Calculate Storage or Elevation Points:** Select the units to be entered, enter either the elevation or storage, hit the Enter key to see the result.
 - **Calculate Permanent Pool Elevation:** Enter the total runoff and hit the Enter key to calculate the permanent pool elevation.
- **Manually input elevation and Area / Dimension (Width, Length) / Volume :** This option is used to manually enter the pond area, dimensions or volume at increasing stages (elevations), and all the options in the lower portion of the dialog turn off and are not available, since the pond characteristics are not yet known. Then prompting appears as shown below:

Input (A)rea, Length/Width (D)imensions or <V>olume: D

Stage No. 1

Elevation: 940

Width: 20

Length: 60

<Enter> for more, (R) to Revise, (E) to exit entry: If you made a mistake, you could enter R and then enter a revised Elevation, Width and Length. Otherwise, press Enter to continue.

Stage No. 2

Elevation: 945

Width: 30

Length: 70

<Enter> for more, (R) to Revise, (E) to exit entry: *press Enter*

Stage No. 3

Elevation <950.00>: The program defaults to the last interval.

Width: 40

Length: 80

<Enter> for more, (R) to Revise, (E) to exit entry: *E to exit*

A table appears, similar to the following:

Elev Width Length Area Interval Avg. Area Inc. Vol Acc. Vol Stage
(Ft) (Ft) (Ft) (Acre) (Ft) (Acre) (Acre Ft) (Acre Ft)

940	20.0	60.0	0.028	0.00	0.028	0.000	0.000	0.00
945	30.0	70.0	0.048	5.00	0.038	0.189	0.189	5.00
950	40.0	80.0	0.073	5.00	0.061	0.304	0.494	10.00

Areas are in acres. If the area method of entry were chosen instead, the user would have been prompted for area at each elevation (stage), and the summary table would be blank under the width and length columns. Similarly, if entry was by volume (in cubic feet), all width, length and area columns would be blank.

Calculate Storage or Elevation Points (y/<n>): y

Known (E)levation or known <S>torage:

Storage (e.g. 0.2 or %60 for 60% of total): %60

Storage: 0.30 Elevation: 946.759

Calculate Storage or Elevation Points (y/<n>): *press Enter.* This allows you to move on. The advantage of this option is the ability to find exact spillway and cleanout levels by experimenting with needed storages or desired elevations. For example, sediment cleanout levels are often set at 60% of total storage, which would be in this case 946.76.

Elevation of Top of Structure: 950

Elevation of Emergency Spillway: 948.5

Elevation of Principal Spillway (Enter if same): *press Enter*

Elevation of Cleanout Level: 946.76

Is Above Data OK (<y>/n): *press Enter.* 'n' leads to re-entry of above 4 items

Regardless of whether the stage-storage information was hand-entered or loaded from a CAP capacity file, both methods lead to the next dialog, which defines the drawing and labeling of the stage-storage/stage-area curve graph and text.

Draw Stage-Storage Setup

SPT5 SPT6

Symbol Name: SPT5 Symbol Name: SPT6

Scale of Drawing: 1.0 Draw Stage-Area Curve

Range of Elevation (Ft): 63.00 Total Storage Range (Acre-Ft): 129.09

Elev. Grid Interval: 5.0000 Storage Grid Interval: 10

Elev. Grid Decimals: 0.00 Storage Grid Decimals: 0.00

Draw Square Grids

Horiz. Axis Label: Accumulative Storage

Vert. Axis Label: Elevation MSL

Title of Plot: Stage Storage Curve

Data Title: Storage volume computations

Skip every 2nd table entry

Footnotes

None Certification Business Address

OK Cancel Help

- **Scale of Drawing:** This sets the size of the plot for CAD. A scale of 1.0, will draw the page 8.5 x 11 drawing units in the CAD window.
- **Elev. Grid Interval:** This sets the interval of the horizontal grid lines.
- **Elev. Grid Decimals:** Controls the number of decimals in the elevations.
- **Storage Grid Interval:** This sets the interval of the vertical grid lines.
- **Storage Grid Decimals:** Controls the number of decimals in the storage and area.
- **Draw Square Grids:** This setting sets the grid lines to even squares instead of rectangles. It can change the aspect ratio of the graph and may not always fit properly.

- **Axis Labels and Titles:** These are custom entered names that will appear on the graph.
- **Skip every 2nd table entry:** It is often beneficial to skip every other row, since the text rows may exceed the space allotted to the table.
- **Footnotes:**
 - None
 - **Certification:** The default is a KY certification that will read: I—, REG.—,DATE—hereby certify in accordance with 405 KAR 7:040E, Section 103 that this document is correct as determined by accepted engineering practices and includes all the information required of it by KRS Chapter 50 and KAR Title 405.
 - **Business Address:** This option will prompt for the company name and three lines of addresses, as shown below.

Pick Starting Position: pick lower left corner of stage storage curve on screen

Company Name: Carlson Software

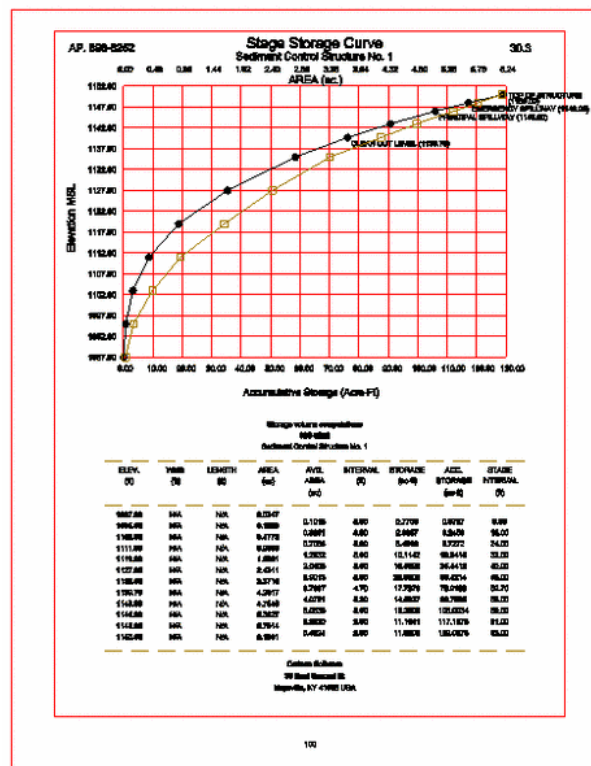
Address Line 1: 33 East 2nd Street

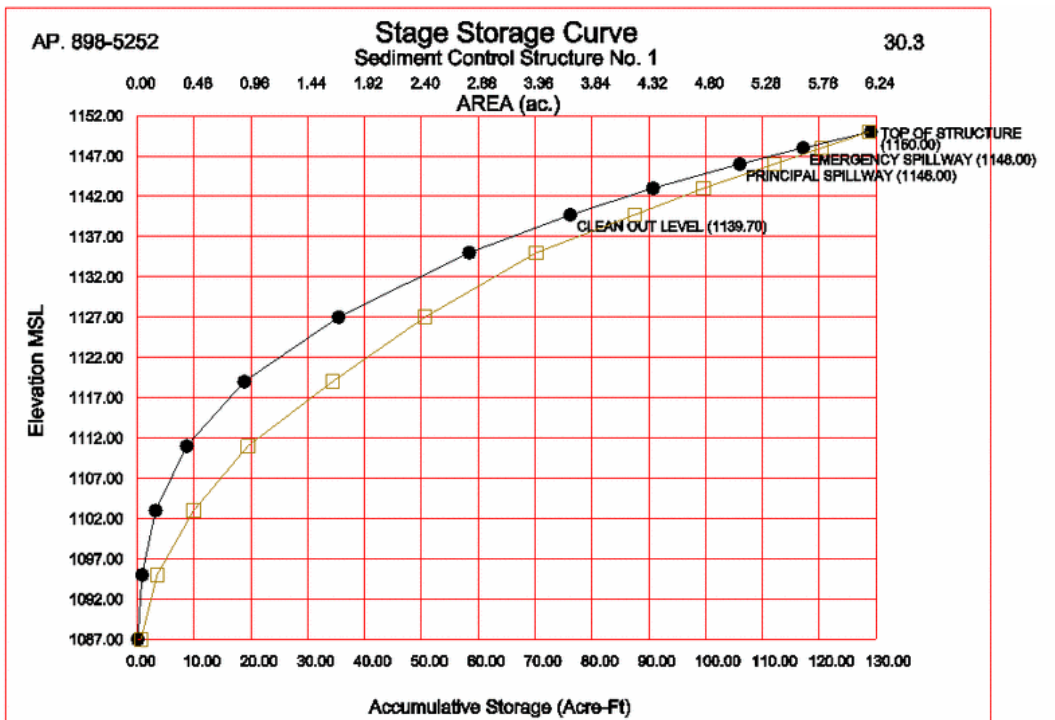
Address Line 2: Maysville, KY 41056

Address Line 3: Enter

Store Pond Capacity File (y/<n>): y. This prompt appears if you hand-enter stage-storage information within the routine and is followed by the normal save file dialog.

Note that if Drawing Setup is set to metric, the stage-storage curve is calculated in cubic meters and all entries are in meters. The final result of a typical combined Stage-Storage and Stage-Area plot is shown below:





Storage volume computations
898-5252
Sediment Control Structure No. 1

ELEV. (ft)	Width (ft)	LENGTH (ft)	AREA (ac)	AVG. AREA (ac)	INTERVAL (ft)	STORAGE (ac-ft)	ACC. STORAGE (ac-ft)	STAGE INTERVAL (ft)
1087.00	N/A	N/A	0.0347					
1095.00	N/A	N/A	0.1689	0.1018	8.00	0.7708	0.6767	8.00
1103.00	N/A	N/A	0.4773	0.3231	8.00	2.3687	3.2453	16.00
1111.00	N/A	N/A	0.9383	0.7068	8.00	5.4819	6.7272	24.00
1119.00	N/A	N/A	1.6501	1.2832	8.00	10.1142	16.8415	32.00
1127.00	N/A	N/A	2.4311	2.0406	8.00	16.5998	35.4412	40.00
1135.00	N/A	N/A	3.3716	2.9013	8.00	22.9802	58.4214	48.00
1139.70	N/A	N/A	4.2017	3.7867	4.70	17.7973	78.2188	52.70
1143.00	N/A	N/A	4.7846	4.0781	3.30	14.5337	90.7525	56.00
1146.00	N/A	N/A	5.3827	5.0638	3.00	15.2509	106.0034	59.00
1148.00	N/A	N/A	5.7814	5.2830	2.00	11.1841	117.1875	61.00
1150.00	N/A	N/A	6.1801	5.4824	2.00	11.9202	129.0876	63.00

Carlson Software
33 East Second St
Maysville, KY 41056 USA

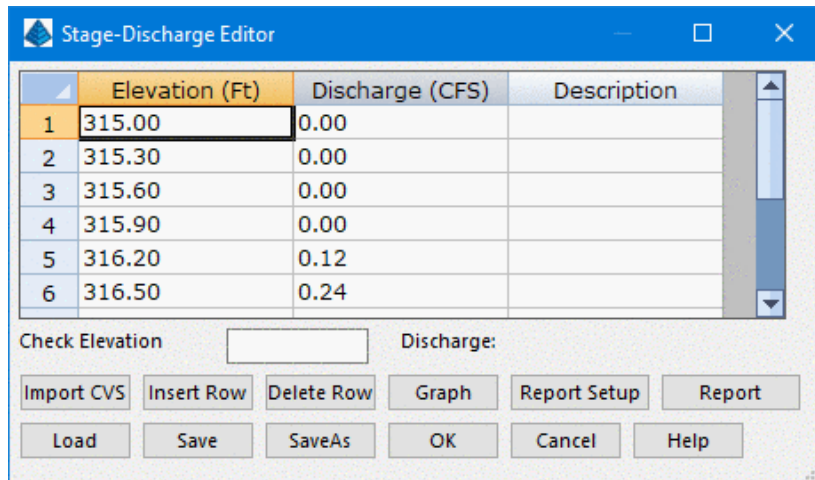
Pulldown Menu Locations: Structure in Hydrology, Surface in Mining

Keyboard Command: stage

Prerequisite: None

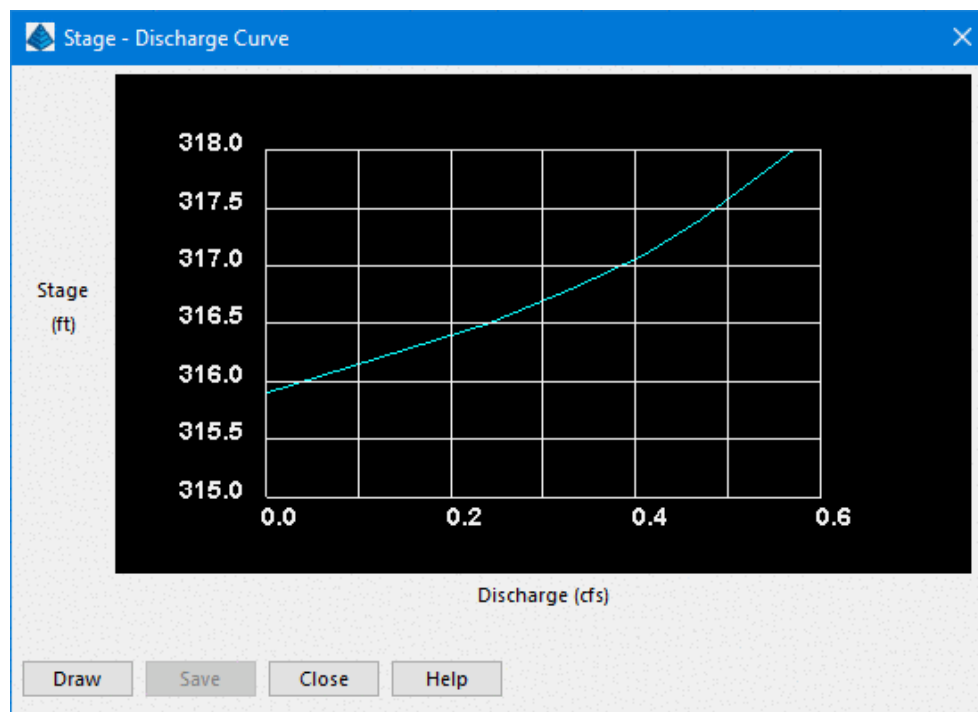
Stage-Discharge Editor

This command allows you to manually input and edit the discharge data at specific elevations. A rating curve is created between the minimum and maximum elevations. Insert Row and Delete Row buttons insert and delete rows at the cursor. Load, Save and SaveAs buttons allow you to load and save the stage-discharge data. Click on the Graph button to open the Stage-Discharge Curve dialog to view the stage-discharge data. After viewing the data, you can plot the graph into the CAD graphic by clicking on Draw button. When you click on the Draw button, the Stage-Discharge Curve Settings dialog displays from where you can define how to plot the text and graph on screen.

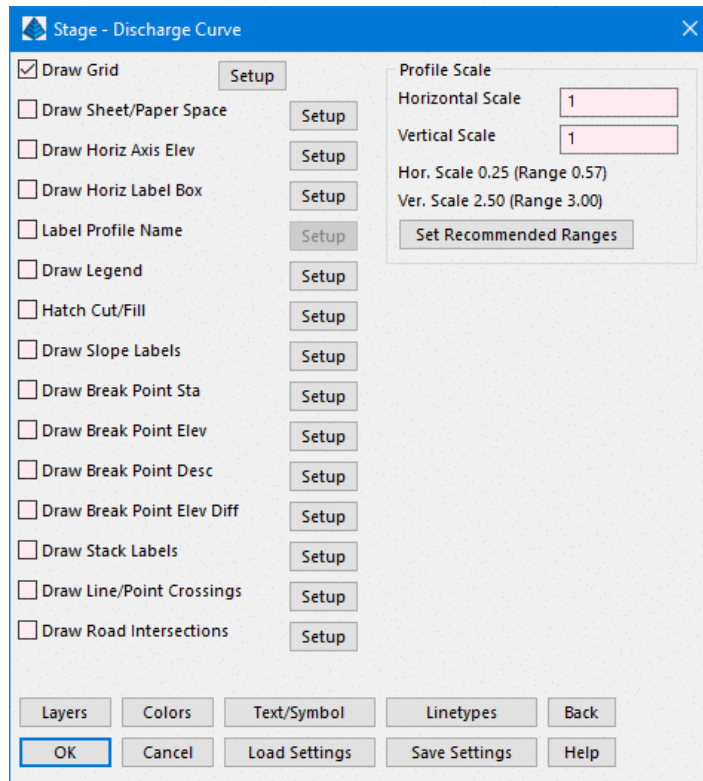


	Elevation (Ft)	Discharge (CFS)	Description
1	315.00	0.00	
2	315.30	0.00	
3	315.60	0.00	
4	315.90	0.00	
5	316.20	0.12	
6	316.50	0.24	

Stage-Discharge Editor



Stage-Discharge Curve



Stage-Discharge Curve Draw Settings

Pulldown Menu Location: Structure > Stage-Discharge

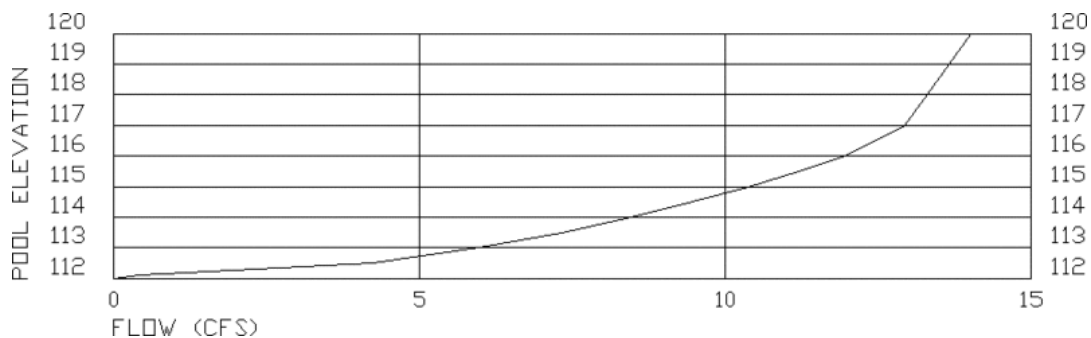
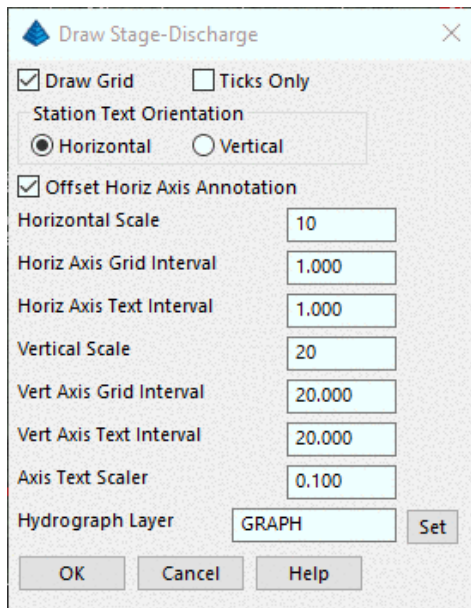
Keyboard Command: edit_stage_discharge

Prerequisite: a stage-discharge file (.STG)

Draw Stage-Discharge Graph

This program draws a stage-discharge graph with the stage (water elevation) on the Y-axis and the discharge on the X-axis. The data to graph is read from a stage-discharge (.stg) file which can be created by several routines including Design Channel, Drop Spillway, etc.

First you are prompted to select a STG file to draw. Then the program asks for the ending discharge for the graph which defaults to the highest discharge in the file. Next this dialog is displayed to enter the graph scale and intervals. The height of the annotation equals the horizontal scale times the Axis Text Scaler.



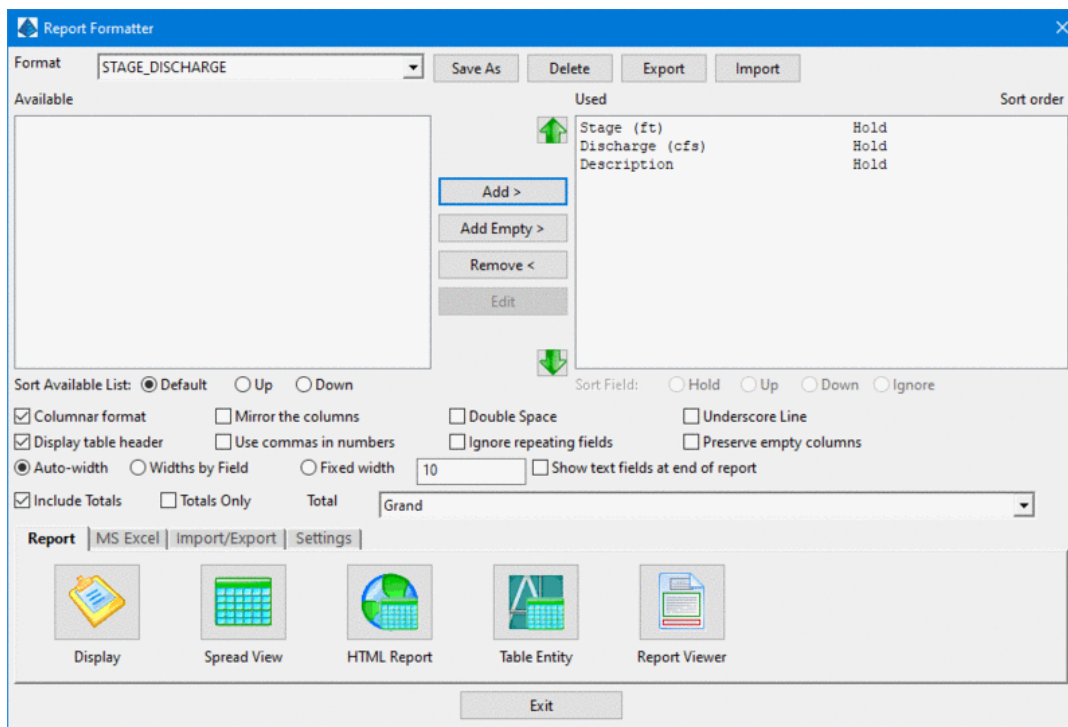
Pulldown Menu Location: Structure in Hydrology

Keyboard Command: stage2

Prerequisite: Stage-Discharge file (.stg)

Report Stage-Discharge

This command simply loads and presents a stage-discharge file, for review and printing. The Report Formatter is used for selecting which fields to report and field properties. Also, the Report Formatter has output to different formats such as Excel. The procedure is to load the file from the normal file loading dialog, then review, edit or print it as shown below:



Pulldown Menu Location: Structure in Hydrology

Keyboard Command: stg_report

Prerequisite: Creation of Stage-Discharge File in Pipe, Channel and Spillway Design routines

Merge Stage-Discharge Files

This command combines two or more stage-discharge files into a single, merged file. Ponds with two or more spillways (for example, a principal and emergency spillway) will outflow increasing volumes of water at higher elevations (stages) in the pond. In many ponds, a principal spillway allows baseline, non-storm flow to exit the pond, and an emergency spillway, placed at a higher elevation, permits storm flows to exit the pond as the water in the pond rises. Because flows increase for virtually all spillway types with increasing water elevation above the spillway, stage-discharge files (.STG files) will show increasing flow at increasing elevation. For ponds with more than one spillway, it is necessary to combine or merge the flows from the multiple spillways as they are encountered at higher elevation. The most typical application is to merge the flow from the principal and emergency spillways that are used on most pond designs. (See graphic in Draw Stage-Storage Curve.) These merged stage-discharge files are then used in the command Locate Structures, found under the Watershed pulldown menu. This command will also merge multiple selections of single stage-discharge files, or will accept a "pre-merged" stage-discharge file. In combination with the pond stage-storage files, the structures in a watershed layout will be used to compute hydrographs and determine the impact of pond placement and spillway design on reducing storm flows.

Prompts

The command is dialog-driven, in this order:

First Stage-Discharge File to Merge. (It is recommended to load the lowest elevation spillway file, typically the principal spillway stage-discharge file.)

Next Stage-Discharge File to Merge. (Here you would typically load the emergency spillway stage-discharge file.)

Merge Another Stage-Discharge File? Yes/No. Click Yes if there is another stage-discharge file, otherwise click No.

Choose the Output Stage-Discharge File. Name the output stage-discharge file.

Pulldown Menu Location: Structure in Hydrology

Keyboard Command: merge_stg

Prerequisite: Spillway design routines that create Stage-Discharge Files, such as Drop Pipe Spillway Design, Pond Weir Spillway Design, Open Channel (Manning's Eq) and Pipe Culvert Design.

Channel Lining Analysis

This routine is used to design small open channels that are constructed as part of a highway drainage system. The roadside channels capture sheet flow from the highway pavement and backslope and convey the runoff to larger channels or culverts with the drainage system. There two classes of channel lining materials: rigid or flexible channel linings. Flexible linings are able to adjust to some change in channel shape and are used in this routine.

Four types of channels are available in the routine. They are **Rectangular**, **Triangular**, **Trapezoidal**, **Irregular** and **Circular Pipe** type. Enter design flow to calculate channel depth, or enter channel Depth to calculate channel flow. When lining is used, the lining shear stress is calculated too. There are six types of channel linings in the lining analysis and please refer to HEC15 for details.

The screenshot shows the 'Design Channel' software window. It is divided into several sections:

- Channel Parameters:** Includes fields for Name (Pond), Channel Type (Circular), Bottom Width (ft), Left Side Ratio, Right Side Ratio, Diameter (ft) (2.000), Channel Depth (ft), Profile Slope (%) (1.0000), and Manning's n (0.0120). There is a 'Use Lining' checkbox and a 'Select' button.
- Calculation:** Includes Design Flow (cfs) (22.348) and Flow Depth (ft) (1.5000). Buttons for 'Calculate Discharge' and 'Calculate Depth' are present.
- Results Table:** A table with columns for Parameter, Value, and Unit.
- Report Type:** Radio buttons for 'Standard Report' and 'Formatted Report' (selected). Buttons for 'Report' and 'Report Setup' are present.
- Buttons:** 'OK', 'Cancel', 'Load', 'Save', and 'Help' are at the bottom.

Parameter	Value	Unit
Flow	22.348	cfs
Depth	1.500	ft
Area of Flow	2.527	sq ft
Wetted Perimeter	4.189	ft
Hydraulic Radius	0.603	ft
Velocity	8.842	ft/s
Critical Depth	1.686	ft
Top Width	1.732	ft
EGL	2.715	ft

The screenshot shows the 'Design Channel' software window. The 'Channel Parameters' tab is active, with the 'Liner Parameters' sub-tab selected. The 'Name' field contains 'carlson' and the 'Channel Type' is set to 'Circular'. The 'Diameter (ft)' is 2. The 'Design Flow (cfs)' is 22.3480 and the 'Flow Depth (ft)' is 1.500. A table of calculated parameters is displayed below the input fields.

Parameter	Value	Unit
Flow	22.348	cfs
Depth	1.500	ft
Area of Flow	2.527	sq ft
Wetted Perimeter	4.189	ft
Hydraulic Radius	0.603	ft
Velocity	8.842	ft/s
Critical Depth	1.686	ft
Top Width	1.732	ft
EGL	2.715	ft

Grass Lining: Grass lining analysis is based on HEC15, chapter 4 - Vegetative lining and bare soil design.

The screenshot shows the 'Channel Parameters' software window with the 'Liner Parameters' sub-tab selected. The 'Lining Type' is set to 'Grass'. A table of grass lining parameters is displayed below the dropdown menu.

	Value	Unit
Grass Cover Density	Good	
Grass Growth Form	Sod	
Grass Stem Height	0.500	ft
Soil Type	Noncohesive	
Soil D75	0.050	in

Grass Turf Lining: this is a turf reinforcement with gravel/soil mixture. The gravel provides a matrix of sufficient thickness and void space to permit establishment of grass roots within the matrix. Turf analysis is based on HEC15, chapter 4 - Vegetative lining and bare soil design.

Channel Parameters **Liner Parameters**

Lining Type:

	Value	Unit
Grass Cover Density	Good	
Grass Growth Form	Sod	
Grass Stem Height	0.500	ft
Gravel D75	1.000	in

Riprap, Cobble, or Gravel Lining: These linings are considered permanent flexible linings. They may be described as a noncohesive layer of stone or rock with a characteristic size, which is D50. Its analysis is based on HEC15, chapter 6 - Riprap, cobble, and gravel lining design.

Channel Parameters **Liner Parameters**

Lining Type:

	Value	Unit
D50	0.450	ft
Stone Weight	165.000	lb/ft ³

Gabion Lining: Gabions (rock filled wire containers) represent an approach for using smaller rock size than would be required by riprap. The smaller rock is enclosed in larger wire units in the form of mattresses or baskets, and the thickness of the gabion mattress may be less than the thickness of an equivalently stable riprap lining. The analysis is based on HEC15, chapter 7 - Gabion lining design.

Channel Parameters **Liner Parameters**

Lining Type:

	Value	Unit
D50	0.450	ft
Stone Weight	165.000	lb/ft ³
Mattress Thickness	0.750	ft

RECP, Rolled Erosion Control Product: RECPs are developed for erosion protection of channels. These products consist of materials that are stitched or bound into fabric. RECP analysis is based on HEC15, chapter 5 - Manufactured (RECP) lining design.

Channel Parameters **Liner Parameters**

Lining Type:

	Value	Unit
Shear Stress On RECP	1.000	lb/ft ²
Shear Stress Mid	1.500	lb/ft ²
Manning's n Lower	0.038	
Manning's n Mid	0.034	
Manning's n Upper	0.031	
Soil Type	Cohesive	<input type="text"/>
Soil Class	GM: Silty gravels, gravel-sand	<input type="text"/>
Soil Plasticity Index	5	
Void Ratio	0.500	
Soil D75	0.050	in

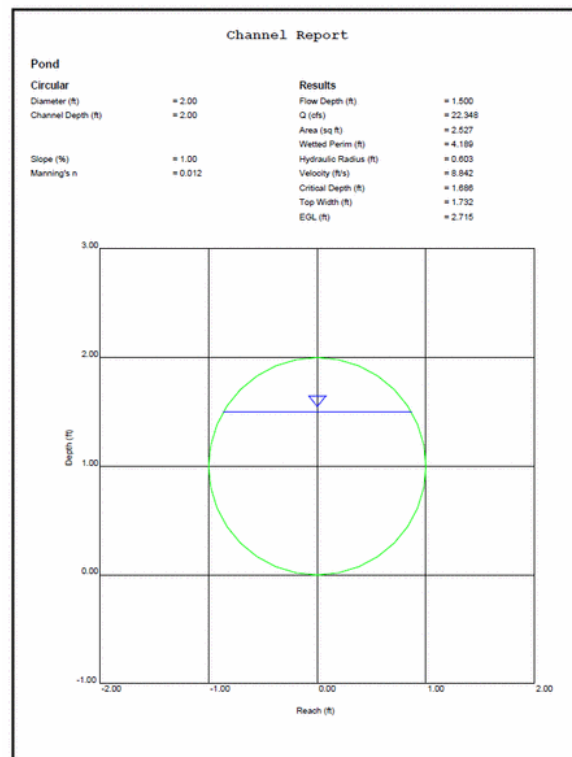
Turf Reinforcement with RECPs: This is a non-degradable RECP composed of UV stabilized synthetic fibers, filaments, netting and/or wire mesh processed into a 3D matrix. Its analysis is based on HEC15, chapter 5 - Manufactured (RECP) lining design.

Channel Parameters **Liner Parameters**

Lining Type:

	Value	Unit
Grass Cover Density	Good	
Grass Growth Form	Sod	
Grass Stem Height	0.500	ft
Gravel D75	1.000	in

Report: Standard Carlson report and PDF report are available. Use **Report Setup** dialog to add/remove report contents.



Pulldown Menu Location: Structure > Channel Design > Channel Lining Analysis

Keyboard Command: channel_lining

Prerequisite: none

Channel Design - NonErodible Mannings Equation

This will compute channel depth, flow and velocity based on channel parameters such as side slopes, base dimensions and Manning's n value. It handles triangular, trapezoidal, rectangular and irregular channels. Entry of a depth leads to calculation of flow and velocity. Entry of one of the other items (flow or velocity) will lead to calculation of the remaining items. In addition to functioning as a channel calculator, the program will output a typical section or detail of the channel as well as a report of the channel output. The routine also works in metric units. It applies to

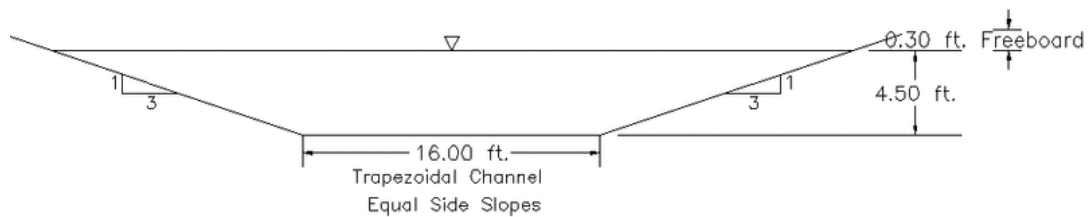
non-erodible channels, primarily. The user can select the Report button to output a report of the channel input and output values as well as a standard detail shown below for the above example.

Prompts

When the routine is selected, the dialog box shown below appears. Select, for example, a trapezoidal channel, equal sides, with side slopes of 3 (for 3:1) and a base dimension of 16. Enter the Manning's n value and channel slope as shown. Note that 0.1, in English units, means 0.1 foot drop per 100 foot of length. Then at the lower right, plug a value of 4.5 for the depth. This will calculate a flow of 862 CFS and a velocity of 6.5 fps, as shown below. If the channel is divided into two types of materials (paved lower portions and vegetated upper portion), you can specify a second Manning's n for the upper banks, as shown in the dialog.

To use the routine as a calculator, enter the known value in the lower right area of the dialog (flow, depth, or velocity), then press enter while still in the entered item. The other two items are then calculated. Note that the routine will default to the last values used during the current Carlson work session, and will capture the flow values calculated in Water Runoff under the Watershed Pull-down. When entering the Manning's n value, a table of n values can be brought up and an appropriate Manning's selected. Among the output options is "Draw Channel Detail", which will draw and annotate as shown below. Since Open Channels are often used as emergency spillways, the Write Stage/Discharge File will output an .STG file for use in the command Locate Structure under Watershed, for pond design and storm routing and hydrograph calculations.

Use Drawing Setup to activate Metric or English outputs. If English is configured, the formula $v = (1.486/n)(R^{2/3})(s^{1/2})$ is used, where n is the Manning's value, R is the water cross section divided by the wetted perimeter and s is the slope ratio. If Metric is configured, the formula becomes $v = (1.0/n)(R^{2/3})(s^{1/2})$ and outputs are in meters. To test metric, set to metric in Drawing Setup. Then for a rectangular, concrete open channel of 12.0 meters width, slope of 0.28%, Manning's n of 0.013 and depth of 2.5 meters, you should compute a velocity of 5.94 m/s and Q (flow) of 178 cubic meters/second.



Prompts

Channel Design (Manning's Eq) dialog: Fill in values.

Pulldown Menu Location: Structure in Hydrology

Keyboard Command: channel1

Prerequisite: None

Channel Design - Erodible Mannings Equation

This command uses the same Manning's equations as non-erodible channel design. In this case, the discharge and velocity are known. The velocity must be less than a maximum to prevent erosion. The program calculates the channel dimensions that meet the requirements.

First choose the channel and water type. Then either enter the Manning's n, velocity, and tractive force or select them from a table of channel types by clicking Select from Table. Also fill in the slope and discharge. Finally, choose either Calc Base or Calc Ratios to compute the channel dimensions. You may also fill in the channel dimensions and choose to Calc Discharge or Calc Depth. The Standard Parameters are used in drawing the channel detail. When OK is selected, the routine ends and the channel is drawn if Draw Channel Detail is checked.

When choosing Calc Discharge, Calc Base, Calc Ratios or Calc Depth, there will be a message **Error: unable to solve these parameters** on the top line if the design parameters never reach erosion conditions for any channel dimension. Consider an extreme error case with a discharge of 1 cfs, a slope of 0.1%, and a velocity of 5.0 fps.

There are no dimensions that meet these requirements. So, for this case, the channel dimensions can be set anyway to avoid erosion.

Prompts

Channel Design (Erodible) dialog: Fill in values.

Pulldown Menu Location: Structure in Hydrology

Keyboard Command: channel2

Prerequisite: None

Grass Channel Design

This feature allows you to design grass lined channels having one of three different cross sections:

1. Triangular
2. Parabolic
3. Trapezoidal

The design methodology that is used is that recommended in Chapter 7, Grass Waterways, Part 650 of the Engineering Handbook published by the United States Department of Agriculture Natural Resources Conservation Service. It uses the equations found in Appendix B of Chapter 7. The solution of the equations is an iterative process and results in several possible cross sections having varied dimensions. The designer must then use engineering judgment to choose the final channel cross section dimensions based on the site layout, machine crossing restraints, excavation equipment to be used, etc.

Basic Design Parameters

Use the entries on the Basic Design tab (see below) to specify the design methodology to use and the basic shape and limiting dimensions of the cross section.

The screenshot shows the 'Grass Channel Design Parameters' dialog box with the 'Basic Design' tab selected. The dialog is divided into several sections:

- Design Methodology:** Two radio buttons are present: 'Retardance Curves' (selected) and 'Calculate using grass parameters'.
- Cross Section to Use:** Three radio buttons are present: 'Triangular', 'Parabolic', and 'Trapezoidal' (selected).
- Design Requirements:** Four input fields with units:
 - Design peak flow: 0.00 cfs
 - Channel Centerline Slope: 0.000 percent
 - Maximum Channel Top Width: 1000.00 ft.
 - Freeboard: 0.00 ft.
- Construction Considerations:** Three input fields with units:
 - Side Slope Ratio: 1.75 :1
 - Min. Channel Bottom Width (optional): 0.00 ft.
 - Channel Depth Search Increment: 0.1 ft. (with a dropdown arrow)

At the bottom of the dialog, there are four buttons: 'Draw', 'Report', 'Calculate', and 'Exit'.

Design Methodology

You may use one of two available design methodologies:

1. **Retardance Curves** - retardance curves can be used when specific data on the condition, density, height and other information for the grasses being used to line the channel are not available. This method will generally result in a safe but relatively conservative design. In this method the designer must specify one of several available design curves (A through E) for the best and worst grass conditions. (see the Retardance Curve Design tab below)
2. **Calculate using grass parameters** - this method requires the designer to have a more detailed knowledge of the specific grass being used. This allows the designer to produce a safe design that may not be quite as conservative as the retardance curve method.

Cross Section to Use

Choose the type of cross section you prefer

1. **Triangular**
2. **Parabolic**
3. **Trapezoidal**

Design Requirements

Design Peak Flow - Enter the Design Peak Flow (cubic feet or cubic meters per second) as obtained from your watershed model or other source.

Channel Centerline Slope - Enter the percent slope of the reach of the channel you are designing.

Maximum Channel Top Width - You may specify a starting maximum channel top width. This value will limit the maximum top width that will be considered in the design calculations. This should be set to match the largest channel that your site could accommodate or the maximum that is desirable.

Freeboard - Specify how much freeboard you wish to add to the channel depth above the design water flow depth. This is a matter of engineering judgment and should, among other things, consider the consequences of overtopping and the probability that that may happen. You could estimate this value by doing a preliminary design using a more severe flow than the actual design peak flow.

Construction Considerations

Side Slope Ratio - specify the minimum slope ratio for the side slopes of the channel. For a parabolic channel this will be the side slope ratio calculated at the water surface. This should be set keeping in mind construction limitations and what types of vehicles or machinery must cross the channel, along with possible erosion due to runoff, etc.

Min. Channel Bottom Width (optional) - This field is only available in the case of a Trapezoidal cross section. If this value is not specified a minimum value of 1 foot (or 30 cm) will be assumed. This value should be set to accommodate construction equipment and can also effect ease of crossing by other vehicles or machinery.

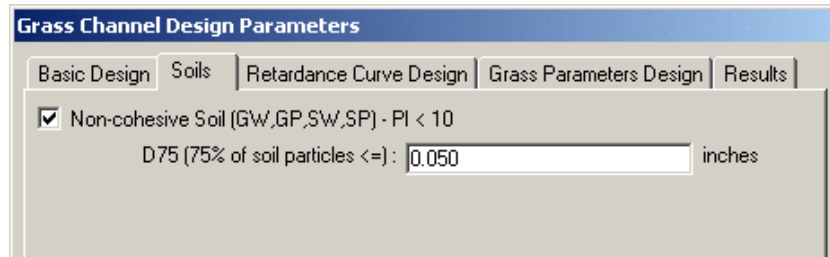
Channel Depth Search Increment - This value is used to increment the depth of the channel for the iterative solution and thus sets the significant figures used during the design calculations and the resulting design dimensions reported. This value should generally reflect the elevation accuracy expected for the construction of the channel cross section.

Soils

The **Soils** tab of the **Grass Channel Design Parameters** dialog allows you to specify the type of soil encountered along the reach of the channel being designed.

Non-cohesive Soils:

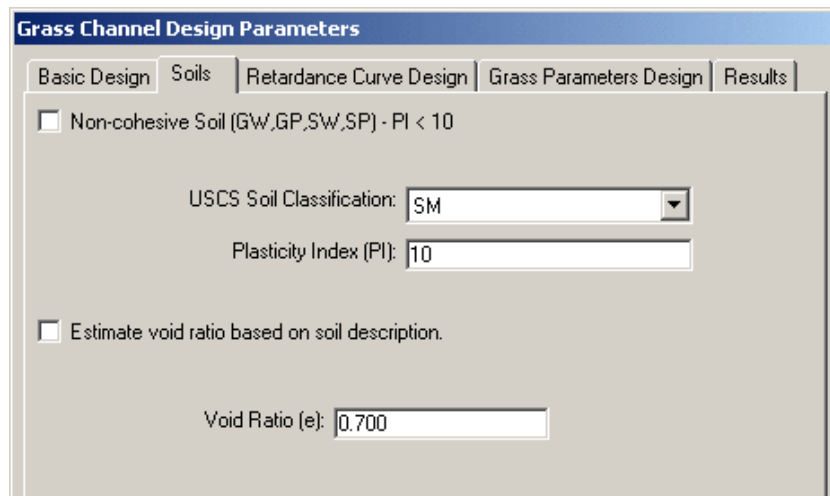
If the soil is non-cohesive then check the **Non-cohesive Soil** check box (see below). This means that the soil must have a Uniform Soil Classification System (USCS) classification of GW, GP, SW or SP (well or poorly graded gravel or sand) with a Plasticity Index (PI) that is less than 10.



When the **Non-cohesive Soil** check box is checked the **D75 (75% of soil particles <=)** edit box appears. Specify the opening size (in inches or millimeters) of the sieve that 75% of the soil particles will pass through.

Cohesive Soils

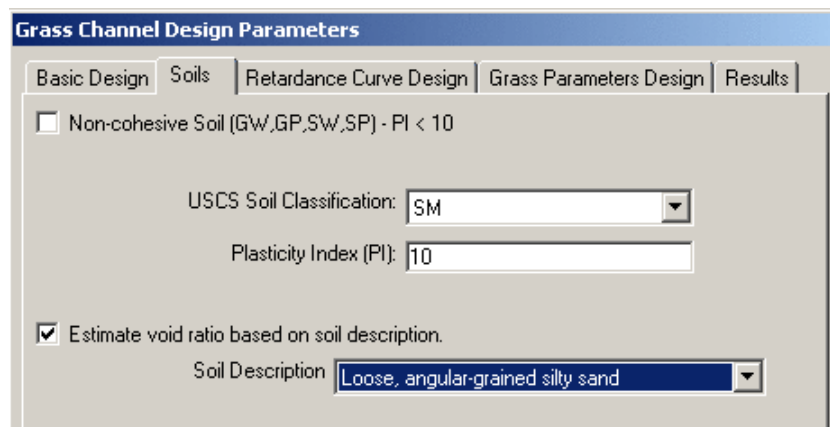
If the soils have a significant clayey or silty portion and have a $PI \geq 10$ then the **Non-cohesive Soil** check box should be unchecked (as shown below).



USCS Soil Classification - choose the correct USCS classification of the soil in this reach of the channel from the drop down list

Plasticity Index (PI) - type in the PI.

Estimate void ratio based on soil description - if this checkbox is checked the void ratio will be estimated from the soil description (see below). If it is unchecked the user must specify the void ratio.



Void Ratio (e) - enter the void ratio for the soil along this reach of the channel. (shows only if **Estimate void**

ratio based on soil description checkbox is *unchecked*) **Soil Description** - choose the description that best fits the soils encountered along this reach of the channel. (shows only if **Estimate void ratio based on soil description** checkbox is *checked*)

Retardance Curve Design

If you chose the **Retardance Curves Design Methodology** on the **Basic Design** tab, then you use this tab to specify the best and worst grass conditions for the design.

Note: If you have chosen the **Grass Parameters Design Methodology** and you click this tab, you will only see the message:

***** Using Grass Parameters Design Method *****

in the middle of the window.

Retardance Curve for Best Grass Condition

From the drop down list, choose A, B, or C retardance curves to be used for the best grass condition. This condition will control the minimum design depth of the water since grass in its best condition will offer more resistance to flow and cause the water to flow deeper. If you are unsure of which retardance curve to use, choose any one of the curves and read the grass descriptions in the window below the drop down list, then choose the one that fits the expected best site conditions.

Retardance Curve for Worst Grass Condition

Use the drop down list to choose the retardance curve (C, D, or E) to be used for the worst grass condition. This condition will control the maximum design depth of the water since grass in its worst condition will offer less resistance to flow and it or the soil may be damaged by flows above this maximum level. If you are unsure of which retardance curve to use, choose any one of the curves and read the grass descriptions in the window below the drop down list then choose the one that fits the expected worst case site conditions.

Grass Parameters Design

If you chose the **Calculate using grass parameters Design Methodology** on the **Basic Design** tab, then you use this tab to specify the grass parameters for the best and worst grass conditions for the design.

Note: If you have chosen the **Retardance Curve Design Methodology** and you click this tab, you will only see the message:

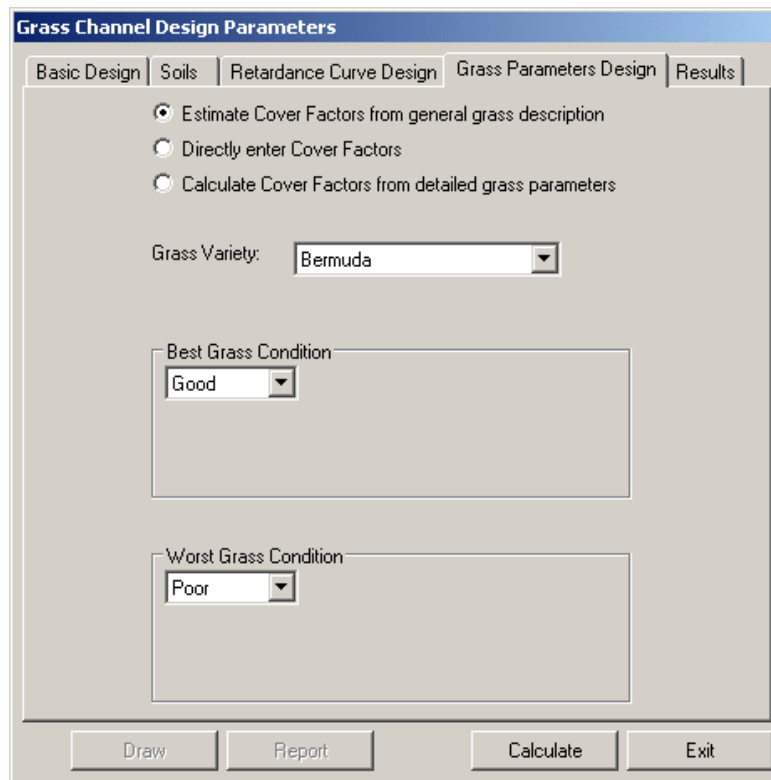
***** Using Retardance Method for Design *****

in the middle of this window

When using the Grass Parameters design method you may choose to use one of three methods to specify the cover factors (CF) for the best and worst grass conditions. Cover Factors describe the ability of the vegetal cover to reduce the maximum hydraulic stress on the soil and is related to the type and quality of the vegetal cover. The three available options are described below.

Estimate Cover Factors from general grass description

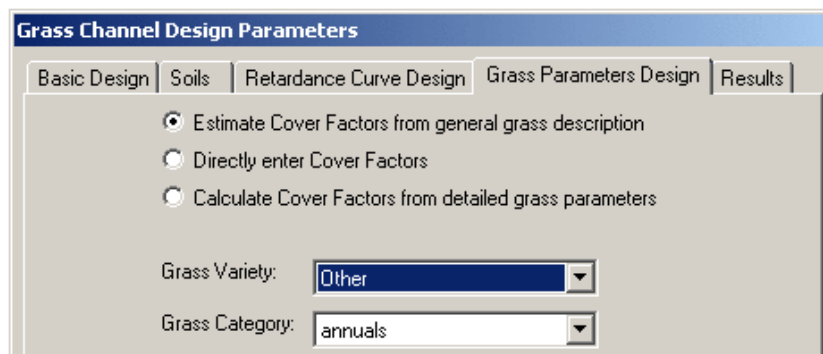
When you click the **Estimate Cover Factors from general grass description** radio button, the items that are visible in the dialog are as shown below:



The screenshot shows the 'Grass Channel Design Parameters' dialog box with the following settings:

- Radio buttons: Estimate Cover Factors from general grass description, Directly enter Cover Factors, Calculate Cover Factors from detailed grass parameters
- Grass Variety: Bermuda
- Best Grass Condition: Good
- Worst Grass Condition: Poor
- Buttons: Draw, Report, Calculate, Exit

Grass Variety - choose the grass variety that will be used to line the channel. If you cannot find the variety you wish to use in the list then choose **Other** (at the bottom of the list).



The screenshot shows the 'Grass Channel Design Parameters' dialog box with the following settings:

- Radio buttons: Estimate Cover Factors from general grass description, Directly enter Cover Factors, Calculate Cover Factors from detailed grass parameters
- Grass Variety: Other
- Grass Category: annuals

Grass Category - choose the general category of the grass to be used to line the channel. (this item only shows if you chose **Other** as the Grass Variety)

Best Grass Condition

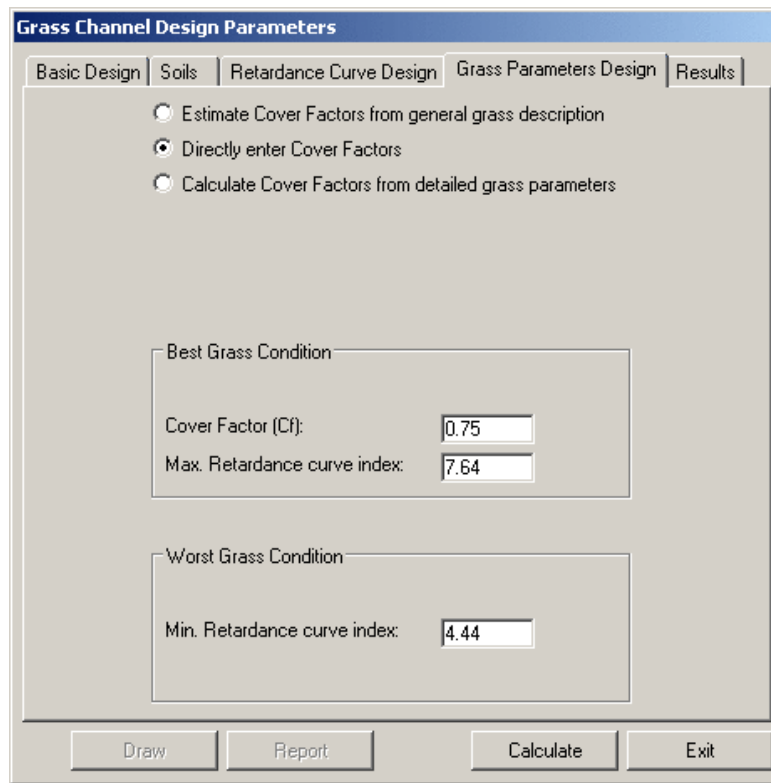
Choose the condition category that describes the expected best grass condition.

Worst Grass Condition

Choose the condition category that describes the expected worst grass condition.

Directly enter Cover Factors

When you click the **Directly enter Cover Factors** radio button the items visible in the dialog are as shown below:



Best Grass Condition

Cover Factor (Cf) - Enter the cover factor for the expected best grass condition.

Max. Retardance curve index - enter the maximum retardance curve index for the expected best grass condition.

Worst Grass Condition

Min. Retardance curve index - enter the minimum retardance curve index for the expected worst grass condition.

Note: The retardance curve index values for the various SCS retardance classes have been published in the publication quoted in the first paragraph of this help item and can be used to estimate the retardance curve index to use:

retardance Class retardance curve index (Ci)

A 10.0

B 7.64

C 5.60

D 4.44

E 2.88

Calculate Cover Factors from detailed grass parameters

If you click the **Calculate Cover Factors from detailed grass parameters** radio button, the items appearing in the dialog are as shown below:

Grass Variety - use the drop list to specify the grass variety to be used to line the channel. If none of the varieties listed are suitable you can choose **Other**.

Grass Category - choose the general category of the grass to be used to line the channel. (only appears if Grass Variety chosen is **Other**)

Best Grass Condition

Maximum Stem Density - enter the estimated stem density for the best grass condition

Maximum Stem Height - enter the estimated stem height for the best grass condition

Worst Grass Condition

Use the drop down list to choose the condition category that best fits the expected worst grass condition.

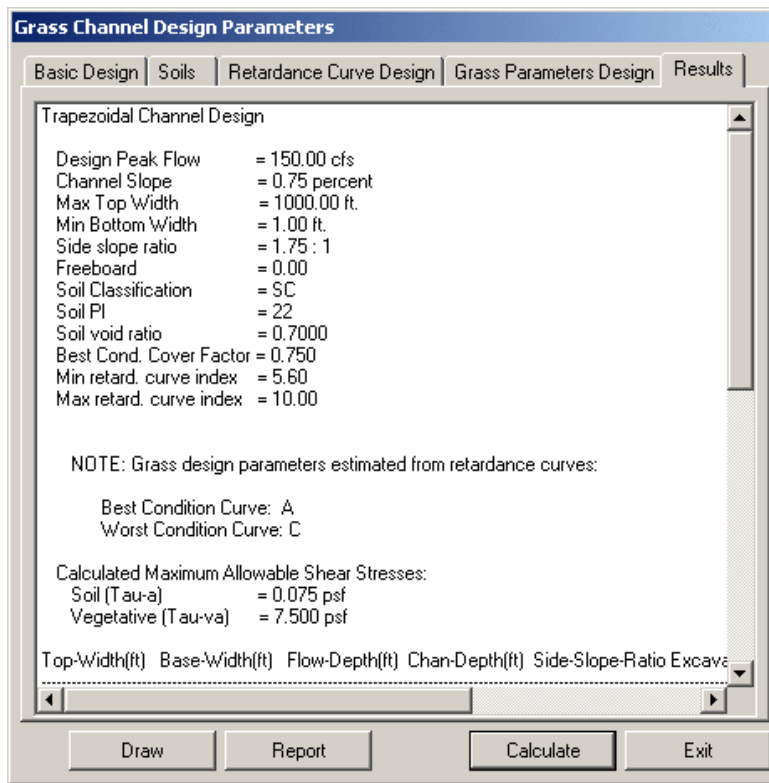
Minimum Stem Density - enter the estimated stem density for the worst grass condition

Minimum Stem Height - enter the estimated stem height for the worst grass condition

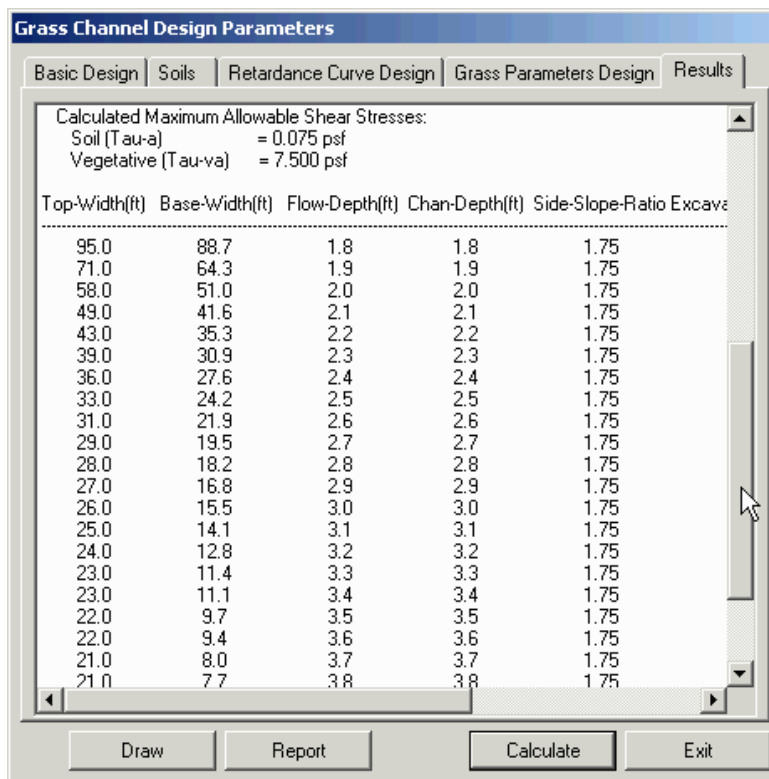
Once you have specified the basic design, soil, and grass parameters you can click the **Calculate** button. If no data entry errors are detected, the input data and the resulting channel design(s) will be listed on the **Results** tab.

Results

After the design calculations have been made, the input data and the resulting channel design information will be listed on the **Results** tab.



To view the channel design cross section information in this window, you merely left click on the scroll bar, hold the mouse button down and drag the scroll bar downward.



The Results listing starts with a recap of the design parameters and the resulting calculated maximum allowable stresses for the soil and the vegetation. Below that begins a list of channel configurations that satisfy the specified design parameters and thus will safely carry the design peak flow (with some freeboard - if specified). The channel geometry is described in six columns for a trapezoidal channel and in five columns for both triangular and parabolic (neither triangular nor parabolic have a bottom width column). The columns are labeled: Top-Width, Base-Width

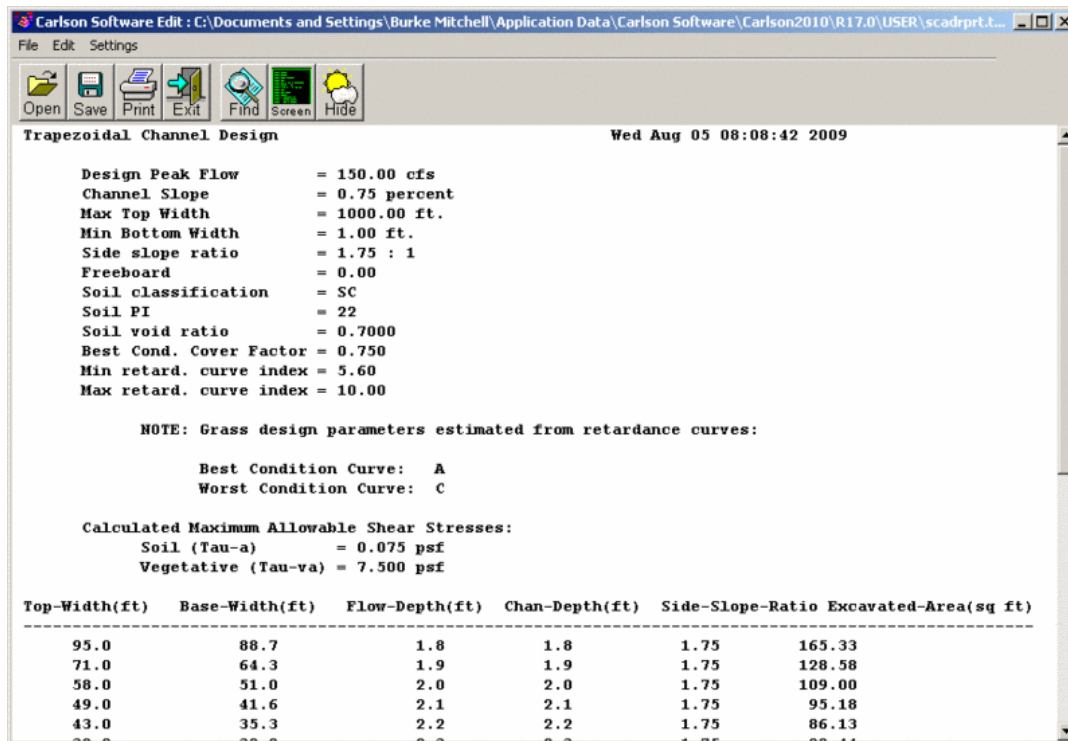
(only for trapezoidal), Flow-Depth, Chan-Depth, Side-Slope-Ratio, and Excavated-Area. The units will be specified according to the unit settings specified in Carlson Configure or CG Options. The Flow-Depth and the Chan-Depth will only differ if freeboard is not zero.

Note: If no safe design is possible for the specified input parameters the following message will appear below the column titles:

**** No valid channel cross section can be determined given specified parameters ****

Viewing, printing and/or saving the design report

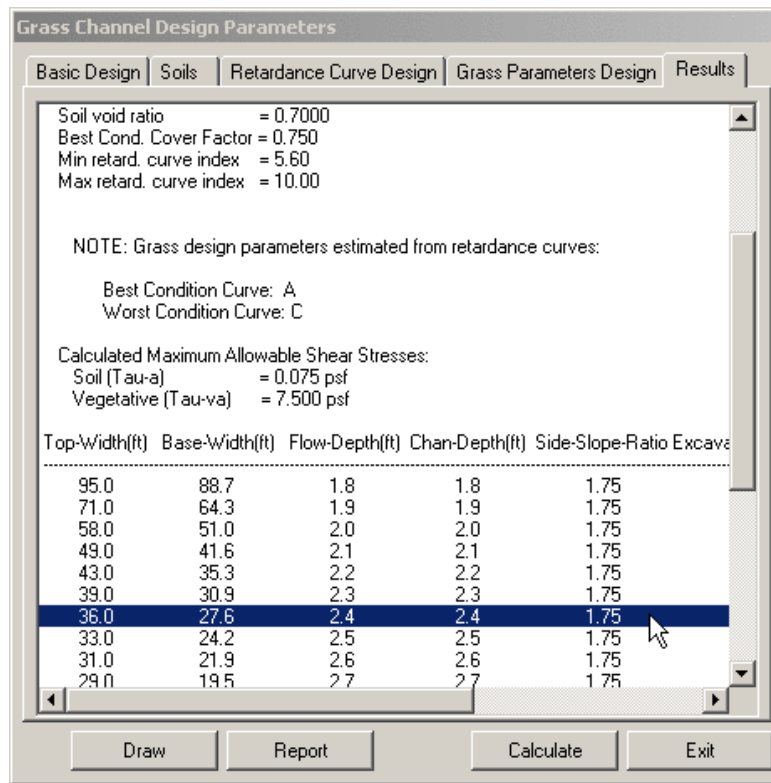
To view the design report click the **Report** button. The report will come up in the Carlson Editor, an example of which is shown below.



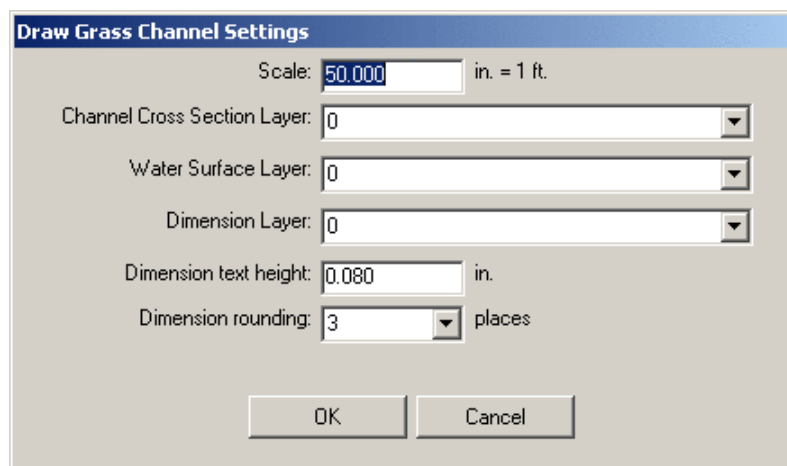
You can now view the input data and the results as well as save them to a text file and/or print them.

Draw the design channel cross section

Once the design calculations have been completed and the Results tab is showing, review the available channel geometries and choose the design you wish to use. To choose a design to be drawn to the current drawing, highlight the desired geometry on the Results tab as shown below:



After highlighting a specific design geometry by clicking on that line, click the Draw button to draw the channel cross section in the current drawing. This will bring up the **Draw Grass Channel Settings** dialog (see below).



Draw Grass Channel Settings

Scale - enter the scale at which the channel cross section is to be drawn. This may default to the current scale of your drawing but may be changed to a larger or smaller scale.

Channel Cross Section Layer - specify the layer on which the channel cross section lines are to be drawn. You may choose an existing layer by using the drop down list or you may specify a new layer by typing it in. If you specify a layer that does not currently exist, the layer will be created.

Water Surface Layer - specify the layer on which the water surface line is to be drawn. You may choose an existing layer by using the drop down list or you may specify a new layer by typing it in. If you specify a layer that does not currently exist, the layer will be created.

Dimension Layer - specify the layer on which the channel dimensions are to be drawn. You may choose an existing layer by using the drop down list or you may specify a new layer by typing it in. If you specify a layer that does not currently exist, the layer will be created.

Dimension text height - specify the height in plotted inches.

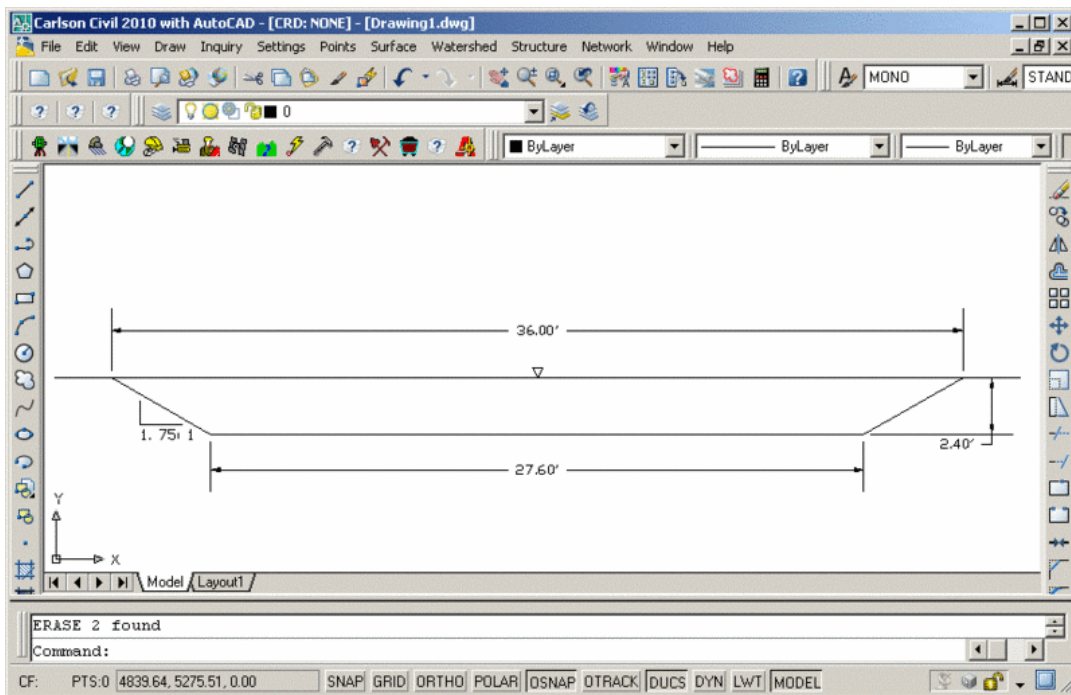
Dimension rounding - use the drop down list to specify the number of decimal places to be used for the dimensions.

Once the settings have been specified, click the **OK** button to proceed with the drawing or **Cancel** to return to the **Grass Channel Design Parameters** dialog

If you clicked the **OK** button you will be returned to the drawing with the following command line prompt:

Pick location for channel cross section drawing: pick a point on the screen for the upper left hand edge of the cross section.

The channel cross section will then be drawn (see an example below) and you will be returned to the **Grass Channel Design Parameters** dialog where you can choose to draw another cross section from the current design list or enter the design parameters for a redesign of the current reach or the design of another reach of this or another channel alignment.



When all designs are completed and you wish to close the dialog, click the **Exit** button.

Prompts

Use the dialog as described above.

When drawing the cross section, this prompt will appear:

Pick location for channel cross section drawing: pick a point on the screen for the upper left hand edge of the cross section.

Pulldown Menu Location: Hydrology Menu - Structure > Channel Design > Grass

Keyboard Command: cg_grass_chan

Prerequisite: None

Pipe Culvert Design

A culvert is a hydraulically short conduit, which can be used to convey stream flow underground through a roadway embankment or other flow obstructions, or used as an outlet structure attached to a detention pond. Culverts come

in circular and rectangular cross sections, and concrete, corrugated steel, aluminum and plastic materials.

The hydraulics of a culvert are complex since several flow control types may exist. The methods that Pipe Culvert Design uses are from FHWA Hydraulic Design Series No. 5 (HDS-5), Hydraulic Design of Highway Culverts. There are two flow controls: inlet control and outlet control. Under **Inlet Control**, the culvert's entrance characteristics determine its capacity, and the culvert is capable of conveying a greater discharge than the inlet will accept. With **Outlet Control**, the inlet can accept more flow than the culvert can carry because of the head loss due to the friction along the barrel or the high tailwater elevation. Furthermore, because culverts are generally not long enough to achieve uniform flow, the flow profile inside is often gradually or rapidly varied flow. In Pipe Culvert Design Settings tab, you may specify the flow control type to design the culvert. If you choose the Optimum option, both inlet control and outlet control calculations are performed, along with the gradually varied flow analysis. The worst-performing control condition is then used to evaluate the proposed design, i.e. the greater of the inlet control headwater and the outlet control headwater is the controlling headwater. Please refer to HDS-5 for details.

Gradually Varied Flow occurs for free surface flow conditions. This method computes water surface depth curve along the length of the barrel and are developed in an upstream direction by standard step method. In standard step analysis, the reach is divided into a number of sub0reaches. Computations are performed in steps from one section to the next, where the depth is incremented or decremented to yield a correct energy balance in the basic equation.

HY-8 Method version 7.5 was introduced in Carlson 2018. HY-8 automates the design methods described in HDS-5 and in HEC-14. Version 7.5 is the latest version and is available on FHWA website.

One significant difference in HY-8 is inlet control analysis. Carlson computes inlet control headwater strictly by the inlet control questions presented in HDS-5. HY-8 uses polynomial curve fits for inlet control headwater computation. The culvert inlets must have HY-8 Inlet Polynomial Coefficients in order to run HY-8 calculation.

The outlet control utilizes HDS-5 equations, and also conducts more complex water surface analysis based on seven flow profile types.

Please refer to HY-8 User Manuals for more details.

Culvert Inlet

Inlet Name: Square Edge, 30 to 75 Degree Wingwall Flares

Ke: 0.4000

Mitered Inlet

HDS-5 Inlet Coefficients

HY-8 Inlet Polynomial Coefficients

HDS-5 Inlet Coefficients

Equation Form: Form 1

K: 0.0026

M: 1.0000

C: 0.0347

Y: 0.8100

HY-8 Inlet Polynomial Coefficients

a: 0.0724927000

b: 0.5070870000

c: -0.1174740000

d: 0.0221702000

e: -0.0014895800

f: 0.0000380000

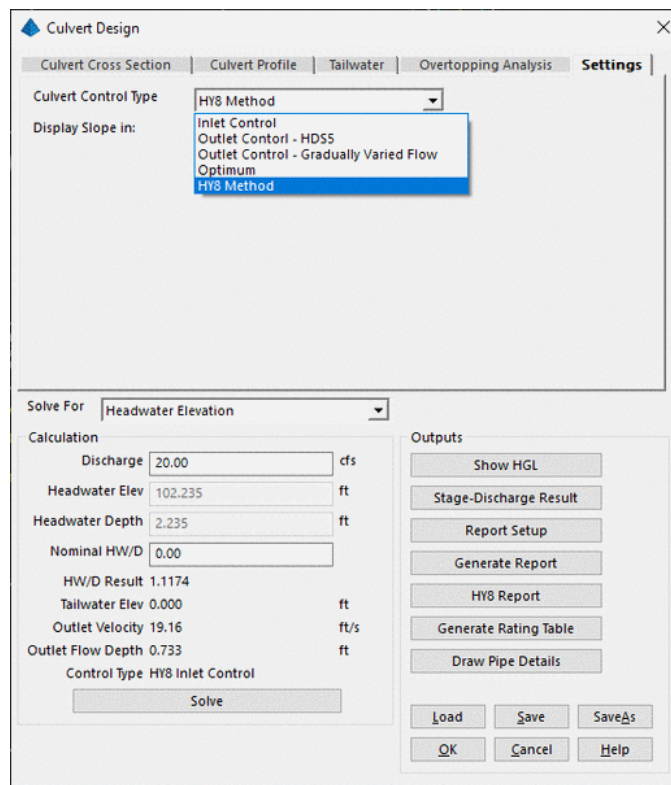
OK Cancel Help

Flow Control	Length Full	Flow Type		Flow Profiles	Outlet		Outlet Depth
		HW>D	HW<D		TW>D	TW<D	
Inlet	none	5	1	S2		n	Normal
Inlet	none	5	1	S1		t	Tailwater (TW)
Inlet	none	5	1	JS1		t	Jump, S1, TW
Inlet	none	5	1	M3, S3, H3, A3		t	Tailwater
Inlet	none	5	1	H3J, A3J		t	H3, Jump, TW
Inlet	part	5	1	S1	f		Full
Inlet	part	5	1	S1	f		Full
Inlet	part	5	1	JS1	f		Jump, S1, Full
Inlet	part	5	1	H3J, A3J	f		H3, Jump, Full
Outlet	none		2	M2, H2, A2		c	Critical
Outlet	none		3	M2, H2, A2		t	Tailwater
Outlet	none		3	M1		t	Tailwater
Outlet	part		3	M1	f		Full
Outlet	all	4		FF	f		Full
Outlet	most	6		FF		t	Tailwater
Outlet	most	6		FF		c	Critical
Outlet	part	7		M1		t	Tailwater
Outlet	part	7		M2, H2, A2		t	Tailwater
Outlet	part	7		M2, H2, A2		c	Critical

From the Structure menu in the Hydrology Module, choose Pipe Culvert Design to display the design dialog. Click on Load button to load an existing culvert file to view or modify it, or a new file to start a fresh design. From the Solve For list, select the value that you want to calculate. The available values are: Discharge, Headwater and Size. You may choose Size/Discharge or Size/Headwater to compute the exact discharge or headwater values after the size has been solved.

Settings

Under the Settings tab, Control Type is used to calculate culvert flow as described above. Users can also set Slope display format in either ft/ft or percentage.



Culvert Cross Section Tab

From the Shape list, select the type of culvert that you want to define. The available shapes are Circular and Box. From the Material list, select the material for the culvert such as Concrete or CMP/Aluminum. You can use the Library function next to Material to select the material and Manning's n from the pipe material library and to edit or add to this library. If you don't solve for the culvert size, enter the values in the Diameter box or Height and Width boxes depending on the culvert shape. In the Manning's n box, the Manning's n value is usually automatically set after the material type is chosen, but you can also type a different value manually. In the Number of Barrels box, enter the Number of Barrels for the culvert.

The Culvert Inlet list contains the inlet types available for the current culvert shape. It updates with different culvert shapes and materials. All the inlet types are specified in HDS-5. K_e is the entrance loss coefficient, which is depending on the culvert shapes and inlet types. It'll update with different chosen culvert shape and inlet type, you can also type the value in the box. The Library button next to the K_e field opens the Culvert Inlet Library dialog and allows users to create new Culvert Inlet Types.

Culvert Design

Culvert Cross Section | Culvert Profile | Tailwater | Overtopping Analysis | Settings

Shape: Circular

Material: Concrete

Diameter: 24.00 in

Width: in

Thickness: 2.00 in

Manning's n: 0.0130

Number of Barrels: 1

Culvert Inlet: Square Edge, 30 to 75 Degree Wingwall Flares

Ke: 0.4000

Solve For: Headwater Elevation

Calculation

Discharge: 20.00 cfs

Headwater Elev: 102.235 ft

Headwater Depth: 2.235 ft

Nominal HW/D: 0.00

HW/D Result: 1.1174

Tailwater Elev: 0.000 ft

Outlet Velocity: 19.16 ft/s

Outlet Flow Depth: 0.733 ft

Control Type: HY8 Inlet Control

Outputs

Show HGL

Stage-Discharge Result

Report Setup

Generate Report

HY8 Report

Generate Rating Table

Draw Pipe Details

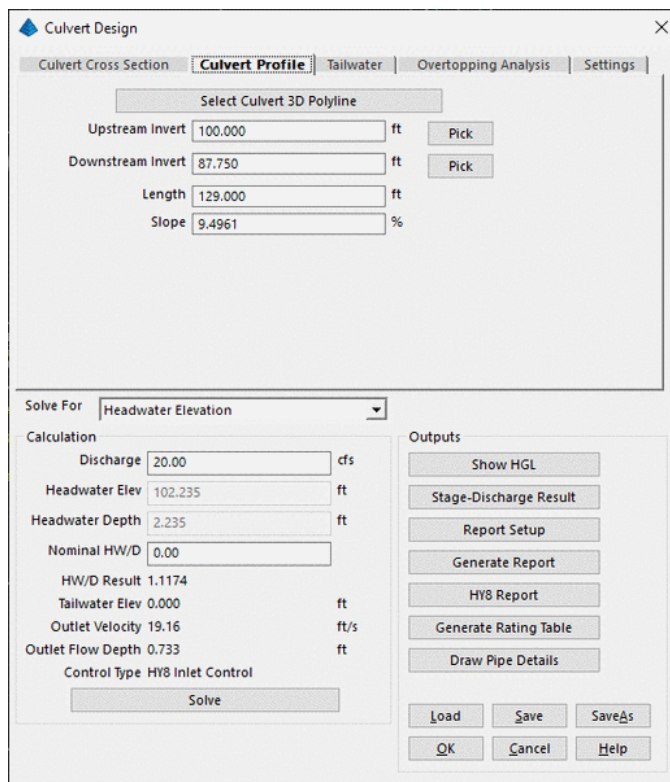
Load Save Save As

OK Cancel Help

Culvert Cross Section

Culvert Profile Tab

Inlet Invert is the elevation of the bottom of the culvert at the upstream end, while Outlet Invert is the elevation of the bottom of the culvert at the downstream end. In the Length box, enter the true culvert pipe length. The culvert slope will be calculated after entering the above three values and displayed in the Slope box. You may also change the outlet invert by entering a slope for the culvert. The most efficient way to get the culvert parameters filled is to select the 3D polyline that represents the culvert. Pick button allows you to pick a 3D point that represents the Inlet/Outlet invert.



Culvert Profile

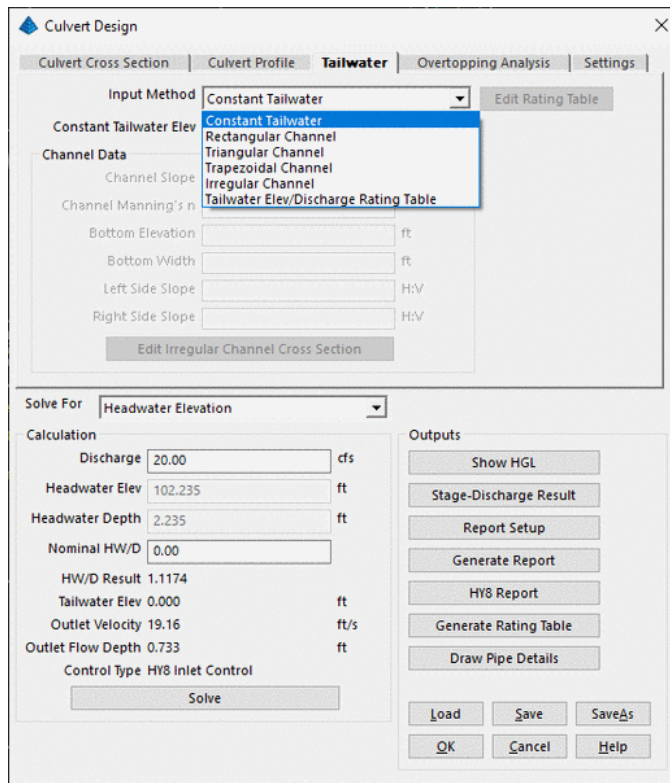
Tailwater Tab

There are six ways to calculate culvert tailwater: Constant Tailwater, Rectangular Channel, Triangular Channel, Trapezoidal Channel, Irregular Channel, and Tailwater Elev./Discharge Rating.

The Constant Tailwater method allows you to specify a single elevation for the culvert design regardless of the culvert discharge. This option is useful when the culvert is discharging into a large reservoir since the change in water surface elevation due to the culvert discharge is negligible.

The Rectangular/Triangular/Trapezoidal/Irregular Channel tailwater methods allow you to specify the channel downstream of the culvert. The culvert tailwater elevation will vary with discharge, and will be equal to the channel invert elevation plus the normal depth of flow in the channel for the current discharge. Channel Data will become available for input if chosen.

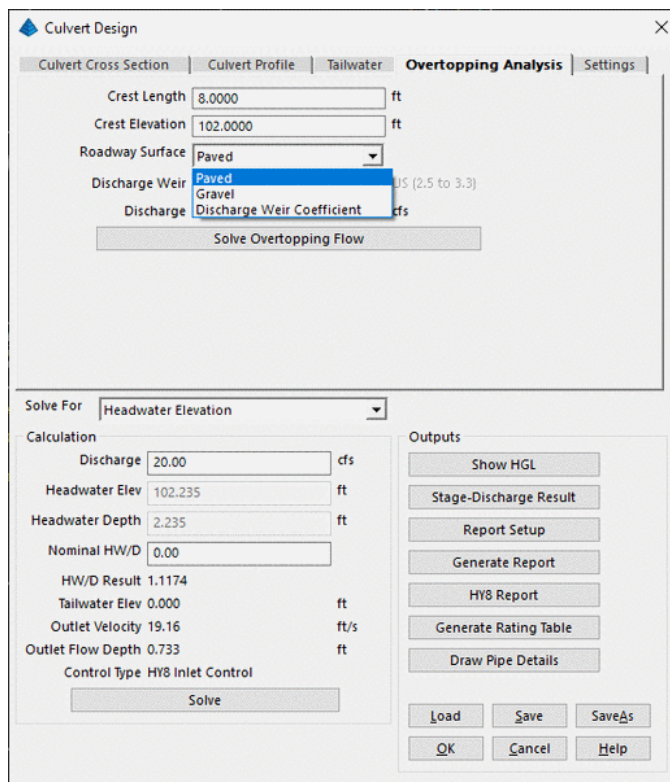
The Tailwater Rating method allows you to input a discharge/tailwater elevation table. Linear interpolation and extrapolation are used to calculate the tailwater for discharges not specified in the table.



Tailwater

Overtopping Analysis Tab

When the culvert headwater begins to rise above the elevation of the roadway, overtopping will occur. Overtopping flow is modeled as a special type of weir flow expressed by the general broadcrested weir equation. Enter values for Crest Length, Crest Elevation, Weir Coefficient in the dialog and overtopping flow can be calculated for the current culvert headwater elevation.



Overtopping Analysis

Calculation

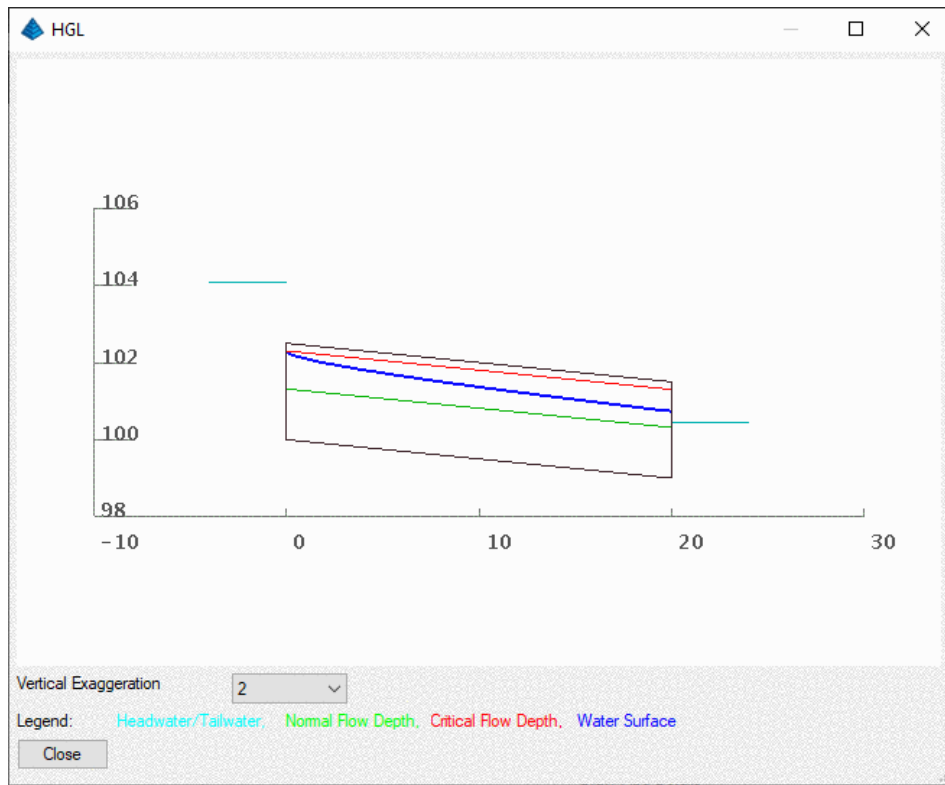
In the Discharge box, enter the flow rate in the culvert. In the Headwater Elev box, enter the water surface elevation at the upstream end of the culvert. You can also enter the Headwater Depth then the program will add the depth value to the inlet invert to get the headwater elevation. If you are solving for either discharge or headwater, the corresponding box will be disabled. Section Size Library allows you to specify as many as available pipe sizes for solving for culvert size. After the hydraulic calculation, the smallest, large enough, available pipe size will be chosen. Please refer to the Pipe Size Library command for defining pipe sizes.

When the Nominal HW/D Ratio is greater than 0, the headwater is always set by the ratio and pipe size. The HW/D result is the ratio of the actual headwater depth and the selected pipe size.

Click on Solve button, depending on Solve For selection, the discharge, headwater elevation, culvert size and the tailwater elevation are calculated and displayed in the dialog correspondingly. The Outlet Velocity and Flow Depth are calculated and shown, and the Control Type is also illustrated.

Outputs

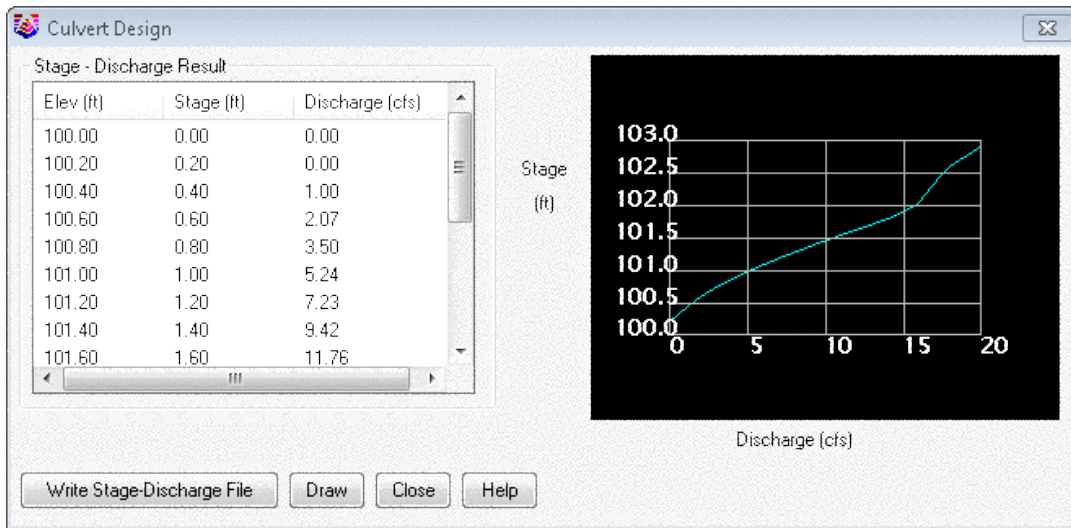
After culvert calculation, click on the **Show HGL** button to show the culvert water surface profile, normal flow level, critical flow level, as well as headwater elevation and tailwater elevation.



HGL Dialog

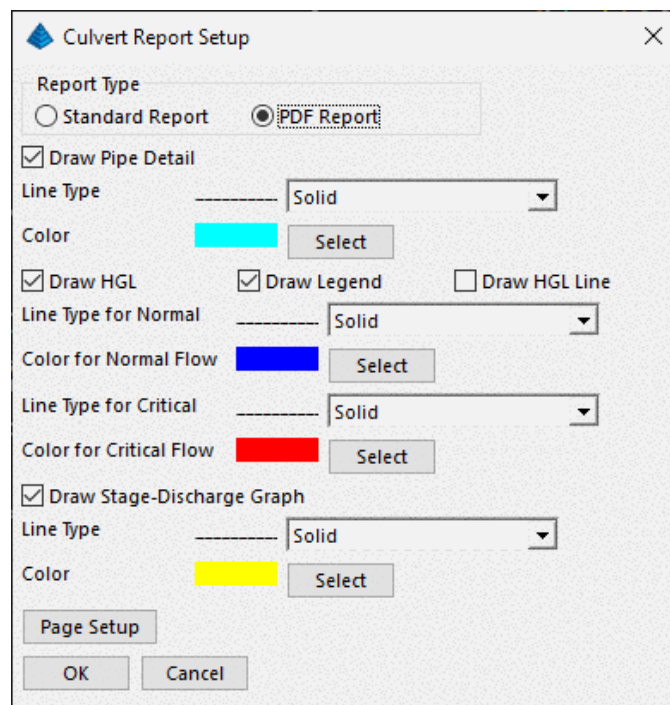
Click on the **Stage-Discharge Result** button to display the stage-discharge curve in the Stage-Discharge Result Dialog. From this dialog, you can view the stage-discharge curve, write the result to a stage-discharge file(.STG), and draw the graph into the CAD graphic. When you click on the Draw button, the Stage-Discharge Curve Settings dialog displays from where you can define how to plot the text and graph on screen.

Stage-Discharge Limits

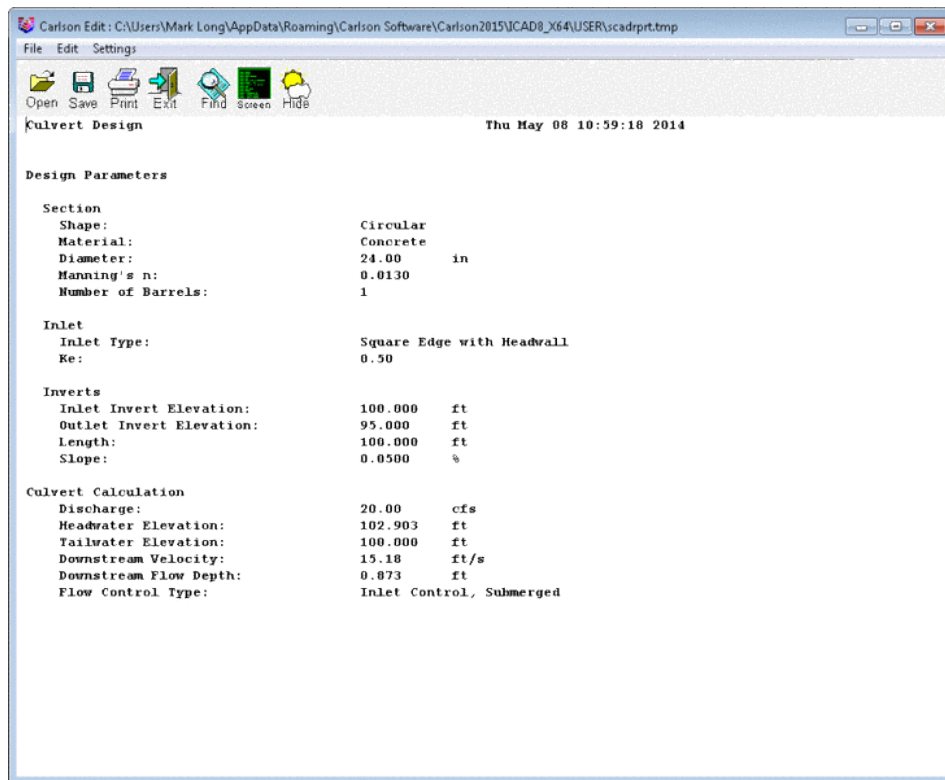


Stage-Discharge Dialog

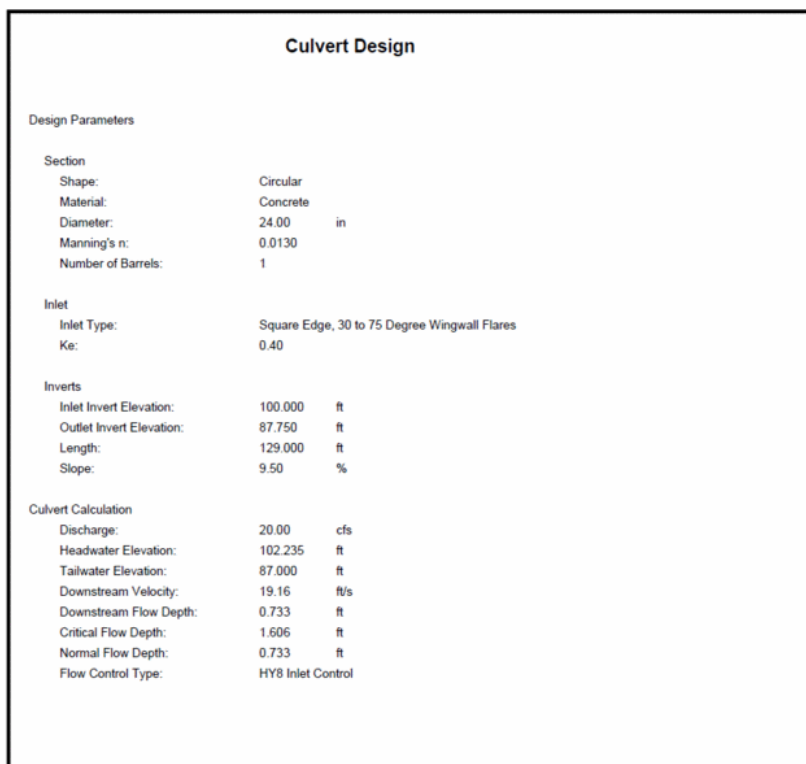
Report Setup button opens the Culvert Report Setup dialog where users can select the Report Type, what data and result to be in the report, as well as the graph color and line types.



Click on **Generate Report** button, the program will present a report screen that contains detailed information regarding the design parameters and the calculations. The report window provides the options of printing, drawing the report in AutoCAD or storing the report to a file. Shown below is an example.



Culvert Report



Culvert PDF Report

When HY8 method is used, click on **HY8 Report** button to generate the HY8 PDF report that based on the HY8 PDF report template. Shown below is an example.

HY-8 Culvert Analysis Report

Crossing Discharge Data

Storm Event:	
Discharge Selection Method:	Specify Minimum, Design, and Maximum Flow
Method: Minimum Flow:	25.00 cfs
Design Flow:	50.00 cfs
Maximum Flow:	100.00 cfs

Table 1 - Summery of Culvert Flows at Crossings: Culvert

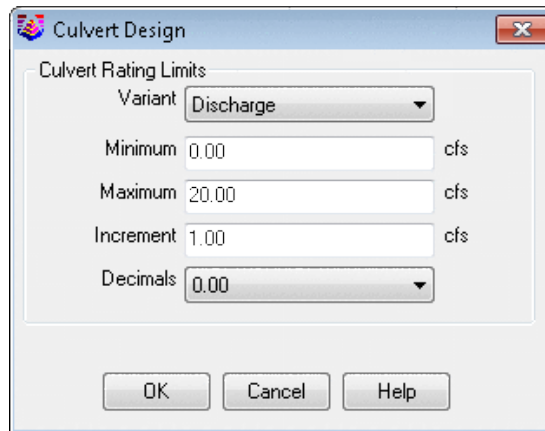
Headwater Elevation (ft)	Total Discharge (cfs)	Culvert Discharge (cfs)	Roadway Discharge (cfs)	Iterations
102.201	25.000	25.000	0.000	1
102.684	32.500	32.500	0.000	1
103.218	40.000	40.000	0.000	1
103.838	47.500	47.500	0.000	1
104.068	50.000	50.000	0.000	1
105.444	62.500	62.500	0.000	1
106.475	70.000	70.000	0.000	1
107.615	77.500	77.500	0.000	1
108.865	85.000	85.000	0.000	1
110.229	92.500	92.500	0.000	1
111.706	100.000	100.000	0.000	1
0.000	0.000	0.000	0.000	Overtopping

Table 2 - Culvert Summary Table: Culvert

Total Discharge (cfs)	Culvert Discharge (cfs)	Headwater Elevation (ft)	Inlet Control Depth (ft)	Outlet Control Depth (ft)	Flow Type	Normal Depth (ft)	Critical Depth (ft)	Outlet Depth (ft)	Tailwater Depth (ft)	Outlet Velocity (ft/s)	Tailwater Velocity (ft/s)
25.000	25.000	102.201	2.201	1.740	1-S3n	0.892	1.704	0.892	0.891	15.901	5.614
32.500	32.500	102.684	2.684	2.299	5-S2t	1.028	1.941	1.341	1.063	12.118	6.112
40.000	40.000	103.218	3.218	2.949	5-S2t	1.155	2.129	1.519	1.226	12.812	6.524
47.500	47.500	103.838	3.838	3.687	5-S2t	1.277	2.265	1.685	1.382	13.492	6.874
50.000	50.000	104.068	4.068	3.953	5-S2t	1.316	2.300	1.740	1.433	13.711	6.981
62.500	62.500	105.444	5.414	5.444	6-FFc	1.514	2.408	0.000	1.678	0.000	7.450
70.000	70.000	106.475	6.381	6.475	6-FFc	1.636	2.440	0.000	1.820	0.000	7.692
77.500	77.500	107.615	7.464	7.615	6-FFc	1.763	2.460	0.000	1.960	0.000	7.910
85.000	85.000	108.865	8.737	8.865	6-FFc	1.902	2.472	0.000	2.096	0.000	8.109
92.500	92.500	110.229	10.116	10.229	6-FFc	2.069	2.480	2.480	2.231	18.866	8.292
100.000	100.000	111.706	11.612	11.706	6-FFc	2.500	2.486	2.486	2.364	20.387	8.460

 Straight Culvert
 Inlet Elevation (invert): 100.000 Outlet Elevation (invert): 99.000
 Culvert Length: 20.000 Culvert Slope: 0.050

A **rating table** presents the discharge-headwater relationship in the tabular form. It can be displayed in a Microsoft spreadsheet or a standard report. Click on Generate Rating Table button to open Rating Limits dialog. In the Variant list, select the independent variable. The available variables are Discharge and Headwater. Enter data in the Minimum, Maximum and Increment boxes. Enter the Tailwater Elev and select the decimal setting from the Decimals list. When you finish entering data, click on OK button to calculate the rating table. A rating table example is shown below in the standard report format. The first column Discharge is an independent variable, and the other columns are computed variables.



Culvert Rating Table Limits

Carlson Report Viewer

1 of 1 | 100% | Find | Next

Wed Oct 27 15:17:25 2021

Culvert Rating Report

Discharge (cfs)	Headwater (ft)	Tailwater (ft)	Exit Velocity (ft/s)	Exit Flow Depth (ft)	Control Type
0.00	0.00	0.00	0.00	0.00	0.00 Inlet Control, Unsubmerged
1.00	100.51	0.00	7.95	0.17	0.17 Outlet Control, Gradually Varied Flow
2.00	100.73	0.00	9.79	0.23	0.23 Outlet Control, Gradually Varied Flow
3.00	100.91	0.00	11.04	0.28	0.28 Outlet Control, Gradually Varied Flow
4.00	101.06	0.00	12.03	0.33	0.33 Outlet Control, Gradually Varied Flow
5.00	101.20	0.00	12.86	0.36	0.36 Outlet Control, Gradually Varied Flow
6.00	101.33	0.00	13.56	0.40	0.40 Outlet Control, Gradually Varied Flow
7.00	101.45	0.00	14.20	0.43	0.43 Outlet Control, Gradually Varied Flow
8.00	101.56	0.00	14.77	0.46	0.46 Outlet Control, Gradually Varied Flow
9.00	101.67	0.00	15.27	0.49	0.49 Outlet Control, Gradually Varied Flow
10.00	101.78	0.00	15.75	0.51	0.51 Outlet Control, Gradually Varied Flow
11.00	101.88	0.00	16.18	0.54	0.54 Outlet Control, Gradually Varied Flow
12.00	101.99	0.00	16.60	0.56	0.56 Outlet Control, Gradually Varied Flow
13.00	102.09	0.00	16.98	0.59	0.59 Outlet Control, Gradually Varied Flow
14.00	102.19	0.00	17.34	0.61	0.61 Outlet Control, Gradually Varied Flow
15.00	102.29	0.00	17.68	0.63	0.63 Outlet Control, Gradually Varied Flow
16.00	102.39	0.00	18.01	0.65	0.65 Outlet Control, Gradually Varied Flow
17.00	102.49	0.00	18.31	0.67	0.67 Outlet Control, Gradually Varied Flow
18.00	102.59	0.00	18.60	0.69	0.69 Outlet Control, Gradually Varied Flow
19.00	102.69	0.00	18.88	0.71	0.71 Outlet Control, Gradually Varied Flow
20.00	102.80	0.00	19.16	0.73	0.73 Outlet Control, Gradually Varied Flow

Culvert Rating Table in Report Format

Draw Pipe Detail button plots a fully annotated standard detail, with user-controlled inlet and outlet slope entries and scaling.

Culvert Design

Layer Name: CULVERT

Draw Scale: 50.0000

Text Size Scale: 0.1000

Headwater Elev: 102.90 ft

Tailwater Elev: 100.00 ft

Draw Embankment

Embankment Data

Elevation: 105.00 ft

Slope Ratio at Inlet: 2.00

Slope Ratio at Outlet: 2.00

OK Cancel Help

Draw Pipe Detail Settings

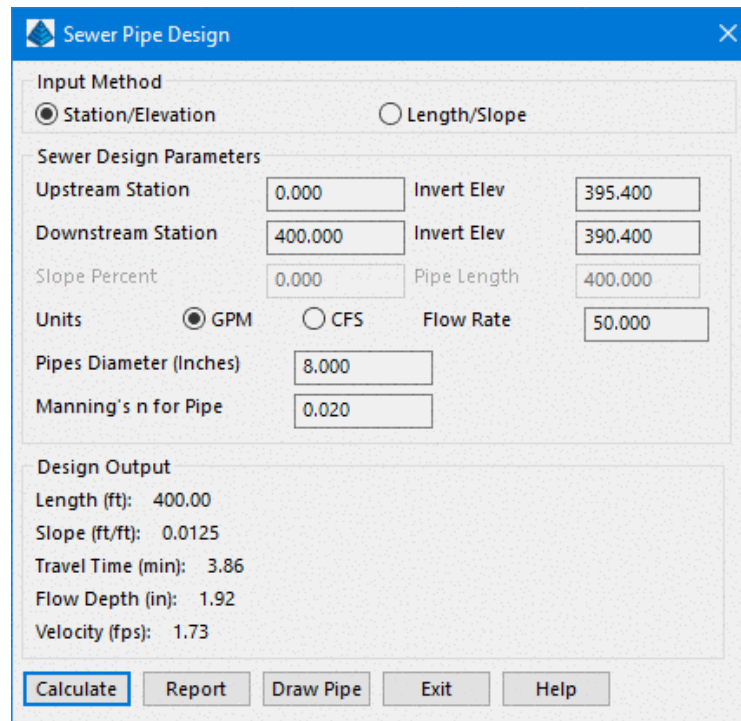
Pulldown Menu Location: Structure

Keyboard Command: culvert

Prerequisite: a culvert file

Sewer Pipe Design: Individual

This command calculates the travel time, flow depth, and velocity for a section of pipe. It calculates for one pipe section using the dialog shown here. Pipe sections can be entered as upstream/downstream stations and elevations, or as length and slope. Clicking Calculate will give you Design Output data based upon your input. Follow up Calculate by clicking Report, which will give you the Standard Report Viewer. Even after reviewing the information in the viewer and clicking on the viewers' Exit button, or its other output buttons, you still have the opportunity to change your original dialog box entries. This is because the command cycles from the viewer back into the dialog box for modifications. The Draw Pipe button draws the pipe details on the screen. The Exit within the dialog ends the command.



Sewer Pipe Design: Individual

Prompts

Sewer Pipe Design dialog Fill in variables. Click Calculate, then click Report.

Report results from the Standard Report Viewer:

Sewer Design

Upstream Station: 0.000 Invert Elev: 395.400

Downstream Station: 400.000 Invert Elev: 390.400

Flow Rate (GPM): 50.00

Pipe Diameter (in): 8.00

Manning's n: 0.020

Length (ft): 400.00

Slope (ft/ft): 0.0125

Travel Time (min): 3.86

Flow Depth (in): 1.92

Velocity (fps): 1.73

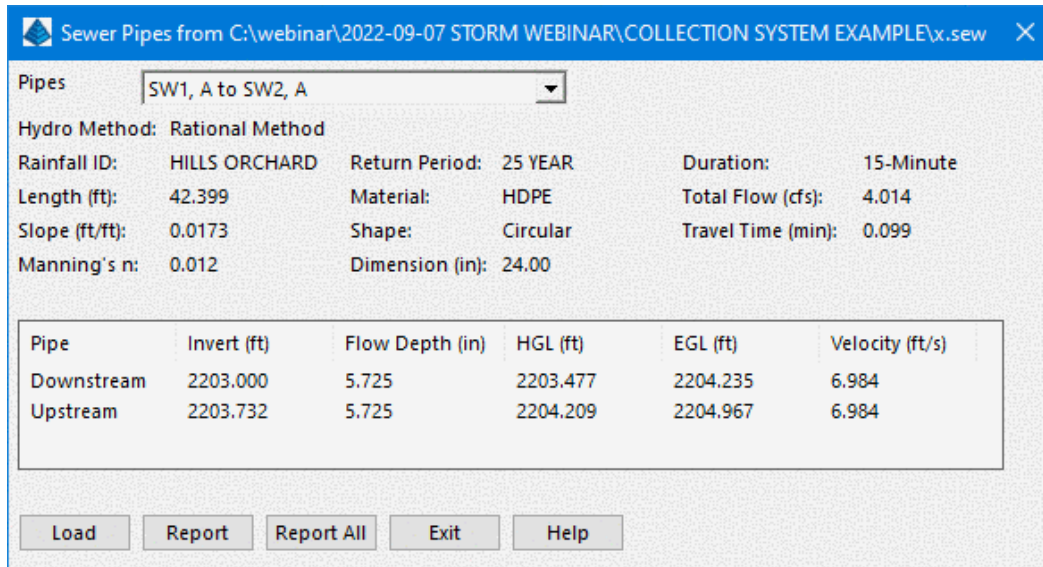
Pulldown Menu Location: Structure > Sewer Pipe Design > Individual

Keyboard Command: swrpipe

Prerequisite: None

Sewer Pipe Design: Sewer Network Segment

This command reads a sewer network and displays every pipe segment in the Sewer Pipe Design dialog. From the Structure > Sewer Pipe Design menu in the Hydrology Module, select Sewer Network Segment. The function reads a sewer network file, conducts the hydraulic calculations and displays pipe parameters and results in the dialog. Pipes list contains all pipe segments, you can select any one of them to display its data. The pipe parameters are shown in the middle part of the dialog, and the results are shown in the bottom table. Load button loads another sewer network file, Report button reports current pipe information to the standard Carlson report, and Report All button reports all pipe data to the standard report.



Sewer Pipe Design: Sewer Network Segment

Prompts

Sewer Network Segment dialog: Fill in values.

Pulldown Menu Location: Structure > Sewer Pipe Design > Sewer Network Segment

Keyboard Command: swrpipe3

Prerequisite: a sewer file (.SEW)

Sewer Pipe Design: Read Profile

This command calculates the travel time, flow depth, and velocity for a section of pipe. It reads the stations and elevations of a sewer or pipe profile (.PRO) created by the Design Sewer/Pipe Profile command in the Civil Design module. Pipe sections can be entered as upstream/downstream stations and elevations or as length and slope.

Prompts

Flow rate units [<GPM>/CFS]? *press Enter*

Flow rate <0.0>: *50*

Manning's n for pipe <0.020>: *.02*

Specify a Profile File dialog *select existing sewer or pipe .PRO file*

Number of decimal places <2>: *press Enter*

Report results from the Standard Report Viewer:

Profile Report

Sewer Profile

Station	Invert-IN		Invert-OUT			
Distance	Slope	Width(in)	Depth(in)	Time(min)	Velocity(fps)	
0+40.44	61.80		61.80			
285.53		2.50%	10.00	1.51	2.21	2.15
3+25.88	68.94		68.94			
274.20		2.40%	10.00	1.53	2.16	2.12
6+00.00	75.52		75.52			
200.02		1.50%	10.00	1.71	1.86	1.80
8+00.00	78.52		78.52			

Flow rate: 50.0 (GPM)

Manning's n for pipe: 0.020

Total travel time: 6.23 (min)

Pulldown Menu Location: Structure > Sewer Pipe Design in Hydrology

Keyboard Command: swrpipe2

Prerequisite: None

Lift Station Design

This command aids in the design of duplex sanitary or storm sewage lift stations. The program assumes a duplex station, with the second pump used solely for backup. That is, there are no provisions for multiple pump operation. The system head curve and pump curve are calculated using the least squares method of curve fitting through three points. To calculate the three points input the length of the force main (length of pressurized pipe), an assumed low-level wastewater surface elevation in the wet well, the elevation of the static lift in the force main, the sum of minor loss coefficients in the force main, and three flow rates that adequately cover the desired range of pump operation. The total dynamic head is calculated for each of the three flow rates by adding the static head, friction losses, velocity head, and minor losses that are calculated by the program from the input data. The next step is the calculation of the pump curve. The user should select one or more pumps from a manufacturer's catalog that will produce the desired operating conditions. The input data consists of the pump shutoff head (flow rate equal to zero), a head and flow rate near the desired operating point, and a head and flow rate beyond said operating point of the pump curve.

Lift Station Design Parameters

Length of Force Main Pressurized Pipe (ft)

Wet Well Wastewater Surface Elevation (ft)

Elevation of Static Lift (ft)

Force Main Diameter (in)

Sum of Minor Loss Coefficients

Hazen Williams Roughness Coefficient

Flow Rates (GPM) of System Head Curve

Flow Below Operating Condition

Flow Near Operating Condition

Flow Beyond Operating Condition

Pump Specifications

Pump Description

Shutoff Head (ft)

Head Near Operating Point (ft) Flow (GPM)

Head Beyond Operating Point (ft) Flow (GPM)

Report

Lift Station Design
Preliminary Results

Hydraulic Calculations

FLOW RATE (GPM)	VEL (FPS)	STATIC HEAD (FEET)	FRICTION LOSS (FEET)	VELOCITY HEAD (FEET)	MINOR LOSS (FEET)	TDH (FEET)
150.00	1.70	59.76	15.63	0.04	0.15	75.58
300.00	3.40	59.76	56.41	0.18	0.59	116.94
450.00	5.11	59.76	119.52	0.40	1.34	181.02

Design Characteristics

Length of Force Main= 4350.00 (ft)
Wet Well Wastewater Surface Elevation= 0.24
Elevation of Static Lift= 60.00
Pipe Diameter= 6.0 (in)

The system head curve and the pump curve are then intersected to produce preliminary operating point results. If the user is not happy with the results, click the Edit Input Values button and change any of the parameters. When the user attains the desired results then proceed with the wet well design by clicking OK.

Input for the wet well design includes type of wet well, wet well dimensions, invert elevation of the lowest line entering the wet well, and minimum wastewater depth in the wet well (usually specified by the pump maker). The lead pump's wet well volume is calculated using a formula from Metcalf & Eddy's Wastewater Engineering: Collection and Pumping of Wastewater: $V = CT/4$, where V equals required volume in gallons, C equals pump capacity (GPM), and T equals minimum time in minutes of one pumping cycle. After wet well design the program assigns a new low level wastewater surface elevation in the wet well, and then recalculates the system head curve and final operating point. At this point the user may change any or all of the input parameters. If no changes are needed then click OK to show the Final Results report.

Lift Station Design Parameters

Type of Wet Well: Circular Rectangular

Diameter: 6.00
 Width: 0.00
 Length: 0.00

Invert Elevation of Lowest Gravity Line Entering Wet Well: 3.5
 Minimum Water Depth in Wet Well: 1.000

Minimum Wet Well Capacity Formula:
 Metcalf and Eddy
 Cyclical Alternation of 2 Pumps @ 10 Starts/Hour

Report Data:
 Project: Jones Road Job Number: 1001
 Date: 03-01-95 Set Prep Name: JRF

OK Cancel Help

Final Results

LIFT STATION DESIGN
 FINAL RESULTS

PROJECT: Jones Road
 JOB NUMBER: 1001
 BY: JRF
 DATE: 03-01-95

WET WELL DESIGN
 WET WELL DIAMETER = 6.00
 PIPE INVERT ELEVATION = 3.50
 HIGH LEVEL ALARM ELEVATION = 3.25
 LAG PUMP ON ELEVATION = 2.25
 FIRST PUMP ON ELEVATION = 1.25
 BOTH PUMPS OFF ELEVATION = -3.61
 WET WELL INVERT ELEVATION = -4.61

HYDRAULIC CALCULATIONS
 FLOW VEL STATIC FRICTION VELOCITY MINOR TDH

Quit Edit Design Print

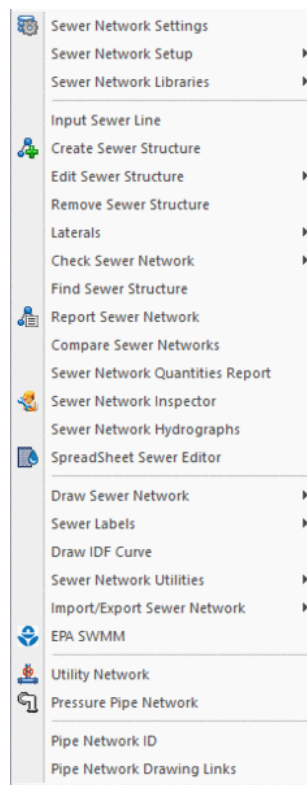
Pulldown Menu Location: Structure in Hydrology

Keyboard Command: LIFTSTA

Prerequisite: None

Network Menu

The Network pull-down menu has commands for layout and analysis of storm sewer networks.



Sewer Network Model

Carlson sewer network model is a powerful program for the design and analysis of networks. A storm sewer network is generally made up of pipes, structures and inlets. A sanitary/utility network, unlike a storm sewer, doesn't have inlets. There may be more than one pipe entering a structure, but only one can exit. The network type is specified via the Sewer Network Settings command.

Sewer structure types and inlet types are defined in their libraries. The default library files are stored in the user directory. Some users would like to share library data and put the library files on a central server and can specify the libraries via Sewer Network Settings command. The sewer network file saves sewer structure and sewer inlet parameters in the .sew file so the sewer network data can be run on any computers that haven't the library files. Once load the sewer structure and inlet data, the program would check the default or user-specify library files and add the new data to the library if it's not in the library.

Sewer Network Settings

This command sets up the working environment for the design and analysis of sewer, sanitary and utility networks and should be done before starting the construction of networks.

Network Type: Indicate if the sewer network conveys storm runoff flow (Storm Sewer) or man-made flow (Sanitary/Utility). If the Sanitary/Utility network type is specified, the controls found in the Drainage tab are disabled.

Active Files: Select a new or existing sewer (.SEW) file to become the active sewer file and a "surface file" for ground cover calculations:

- *Surface by Rim Elevations* - When this option is enabled, the ground surface elevation is derived as an interpolated grade from structure rim elevation to structure rim elevation and the Surface Model option is disabled, or,
- *Surface Model* - When the *Surface by Rim Elevations* option is disabled, select a valid surface model (.TIN, .FLT, .GRD) that blankets the sewer network. The ground surface elevation is obtained from the surface model along the pipe reach.

Link Sewer Network to Reference Surface and Centerlines: Indicate the method of how the sewer network should react if there are changes to either the specified Surface Model or any related Centerlines that are specified in either the Create Sewer Structure command or the Edit Sewer Structure command. The following types of corrections can be made:

1. If the reference Surface Model changes, the structure rim elevation can be updated and there are two options for updating the invert elevation, discussed in the *Elevation Update Method* section below.
2. Each structure has the option to assign a reference centerline and the structure will record the station and offset from this centerline. When the reference centerline of a structure changes, the structure can be moved to the position of the recorded station and offset along the newly modified centerline.

Depending on the desired level of end-user control vs. automation, one of the following corrective options can be specified:

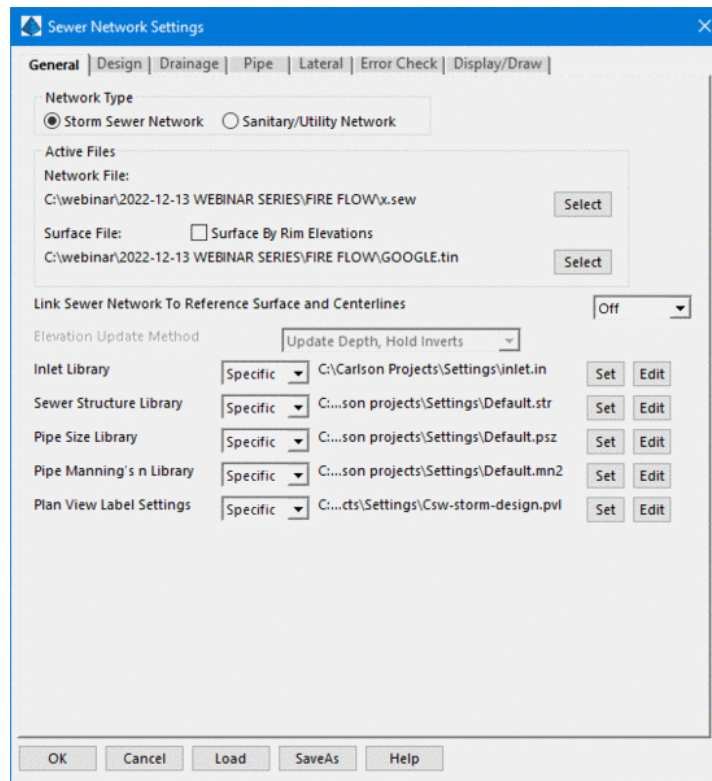
- *Off* - When a Surface Model or a Centerline changes, corrective action to the sewer network is not performed. Any corrections to the sewer network will need to be performed manually or through the use of the Check Reference Centerlines and Surface command which will compare the sewer network with the referenced file(s),
- *Prompt* - When a Surface Model or a Centerline changes, you will be offered a choice for a follow-up action of whether to update the structures or not.
- *Auto* - When a Surface Model or a Centerline changes, corrective action is automatically performed on the sewer network and any elevation updates will be calculated based on the *Elevation Update Method*.

Elevation Update Method: Indicate how the invert elevation(s) should change as a result of a change to a Surface Model (when the "linking" method is set to *Auto*):

- *Update Depth, Hold Inverts* - When this option is selected, the former invert elevations are retained and new depth of cover amounts are re-calculated, or,
- *Update Inverts, Hold Depth* - When this option is selected, the former depth of cover amounts are retained and new invert elevations are calculated

The Inlet Library, Sewer Structure Library, Pipe Size Library, Pipe Manning's N Library, and Plan View Label Settings can be set to the current project file CURRENT or SPECIFIC. If specific is specified the user can SET the location of the library files to be used.

The SaveAs button saves all the settings to a .SNS file and Load button loads all the settings from a previously saved .SNS file.



Direction: The network can be designed from Downstream to Upstream or vice versa. If the design direction is from Downstream to Upstream, the first structure created is generally the outfall and the current structure and its downstream pipe are highlighted in the plan view. Otherwise, the network is created from one of its entrances toward the eventual outfall and the current structure and one of its upstream pipes are highlighted.

Auto Set All Sewer Pipe Sizes: When enabled, this option disables the *Pipe Size* control found on the Pipe tab of the **Create/Edit Sewer Structure** command and sizes the pipe(s) automatically to the closest available pipe size as specified in the Pipe Size Library.

Auto Set All the Invert Elevations of the Sewer Network: When enabled, this option:

1. calculates initial *Invert* elevations found on the Structure tab and the Pipe tab of the **Create/Edit Sewer Structure** command, and,
2. enables the *Add Step Up to Minimize Excavation* control, and,
3. disables the *Auto Match Pipes at Junction* control.

Add Step Up to Minimize Excavation: When enabled, this option allows pipe inverts to be placed above the current invert elevation up to the **Max. Step Up** value to minimize excavation and create a potential "drop manhole."

Auto Match Pipes at Junction: When enabled, this option sets the pipe inverts automatically when creating a new pipe in order to match it to other pipes at the same junction.

Auto Set Invert In When Change Invert Out So Invert Out Not Higher: When enabled, this option will adjust the invert in to the pipe to be higher than the invert out.

Use The Connection Wizard When Add Pipes: This will open the pipe connection wizard when the user selects the ADD or PICK an available pipe connection on the docked dialog box under the Pipe Tab.

Minimize Pipe Sizes in Design: When enabled and if the sewer network is being designed, this option will ensure the pipes are not oversized as extra calculation iterations are performed.

Automatic Watershed Analysis: When enabled, this option performs an automatic watershed analysis of the referenced *Surface File* found on the General tab to determine the drainage area serviced by the new structure. This

action is the same as clicking the *Drainage Area - Calc* button found on the Drainage tab of the **Create/Edit Sewer Structure** command.

Auto Connect Structures: This option determines how to connect a newly created structure to the network:

- *On* - Automatically connects a pipe between the new structure and the closest structure, or,
- *Off* - Does not connect a pipe to the newly created structure (the structure would need to be manually connected to the network via the *Connection* control found on the Pipe tab of the **Create/Edit Sewer Structure** command or,
- *Prompt* - Provides an option as to whether or not a pipe should connect to the new structure.

HGL Computation Method: There are three methods to calculate HGL and EGL: FHWA Method, Virginia DOT Method and Just Full Capacity - Ohio Method. The differences are shown below.

1. FHWA Method

The complete description can be found in "Drainage of Highway Pavement (HEC12) form the FHWA Hydraulic Engineering Circular 12. In this method, when the HGL at the downstream is above the critical depth, use the HGL elevation as the initial elevation and conduct non-uniform flow calculation; when the HGL is below the critical depth and normal depth is above critical depth (subcritical flow), set the HGL at critical depth and conduct non-uniform flow calculation; when the HGL is below the critical depth and normal depth is below critical depth (supercritical flow), set the HGL at normal depth and conduct uniform flow calculation.

2. Virginia DOT Method

VDOT method is a simplified FHWA method. The first difference is the determination of the HGL at the downstream side of the pipe. In determining the HGL, begin with the actual tailwater elevation or an elevation equal to 0.8 times the diameter of the outlet pipe, whichever is higher. The second difference is that because the HGL is always greater than critical depth, non-uniform flow is conduct through all the pipes. Another difference is the velocity is normal flow velocity.

3. Just Full Capacity - Ohio Method

One difference between FHWA method and this method is the optimal pipe size. The optimal pipe size is based on the smallest diameter pipe in which the design discharge for the selected storm frequency will not exceed the just full capacity of the pipe. This storm frequency is called Just Full Capacity Frequency. Just full capacity with a free water surface is considered to occur at 93.8% of the pipe diameter for circular conduits. Another difference is the determination of the HGL at the downstream side of the pipe, this method uses the tailwater elevation or critical depth plus half of the diameter of the pipe, whichever is greater, to determine the starting HGL.

Friction Slope Averaging Method: Available methods are:

- *Arithmetic* - $S_{fav} = (S_{fu} + S_{fd})/2$, or,
- *Conveyance* - $S_{fav} = ((Q_u + Q_d)/(K_u + K_d))^2$, where
 - Q = Flow
 - K = Conveyance = $(1.486/n) * a * r^{2/3}$ (based on English units)
 - n = Manning's n-value
 - a = Flow area
 - r = Hydraulic radius
- *Geometric* - $S_{fav} = (S_{fu} * S_{fd})^{1/2}$

HGL Offset from Rim Elev: This value is used to check the hydraulic grade line result. If some of the hydraulic grade lines are within this value from the rim elevation, alerts are presented to indicate the potential problem(s).

Min. Time of Concentration: This value will replace any calculated Tc that is shorter than this value.

Max. Gutter Spread: Indicate the largest allowable gutter spread for determining inlet effectiveness.

Initial Network Structure Name: In creating a new network, this name is used as the first structure name, and then increment from it for every new structure.

Tributary Pipe Connect Action: When the tributary pipe line is from upstream to downstream and the downstream invert of the last pipe is lower than the manhole base elevation, there are five actions defined in the Sewer Network Settings to adjust the network pipe inverts.

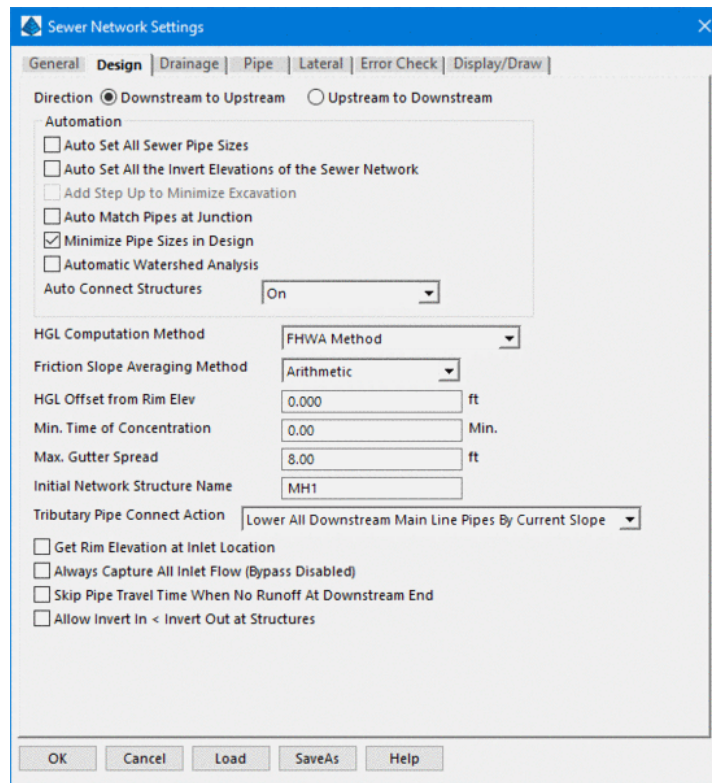
1. Lower All Downstream Main Line Pipes By Current Slope - Lower all downstream mainline pipes by the current slope to match the tributary pipe elevation, and create a step up for upstream mainline pipe.
2. Lower All Downstream Main Line Pipes By Current/Minimum Slope - Lower the downstream mainline pipes by the current/minimum slopes match the tributary pipe elevation, and create a step up for the upstream mainline pipe. Use a slope between the current and minimum slope to minimize the change of the manhole inverts.
3. Raise All Tributary Pipes By Current Slope - Raise the tributary pipes by the current slope to match the connection invert/crown.
4. Raise All Tributary Pipes By Current/Minimum Slope - Raise tributary pipes by the current/minimum slope to match the connection invert/crown. Use a slope between the current and minimum slope to minimize the change of the manhole inverts.
5. Raise Tributary Pipe At Downstream Side - Only raise the downstream invert of the last tributary pipe.

Get Rim Elevation at Inlet Location: When inlet is not placed at the structure location, use the surface elevation at inlet for structure rim elevation.

Draw Structure Symbol at Inlet Location: When inlet is not placed at the structure location, draw structure symbol at the inlet location in plan view.

Always Capture All Inlet Flow (Bypass Disabled): You may elect to let the inlets to capture all drainage runoff.

Skip Pipe Travel Time When No Runoff At Downstream End: When there's no runoff at the downstream catch basin, you can choose to skip its pipe travel time in composite peakflow calculation; otherwise the peakflow at downstream catch basin will be smaller than its upstream catch basin.



The drainage settings are used to set up the hydrology calculation method and rainfall information for the design of Storm Sewer networks.

Hydro Methods: Select either the Rational method (the SCS Rainfall group becomes disabled) or SCS method (the Rational Rainfall group becomes disabled) for Peak Discharge calculations.

Computation Methods: Indicate the computation method for the Storm Sewer network:

- *Peak Discharge* - The Rainfall Durations are disabled and the peak flow of each drainage area is calculated by the selected **Hydro Method** and the Storm Sewer network is designed to pass the peak flow, or,
- *Hydrograph* - The Rainfall Durations are enabled and a runoff hydrograph of each area is generated and routed through the network.

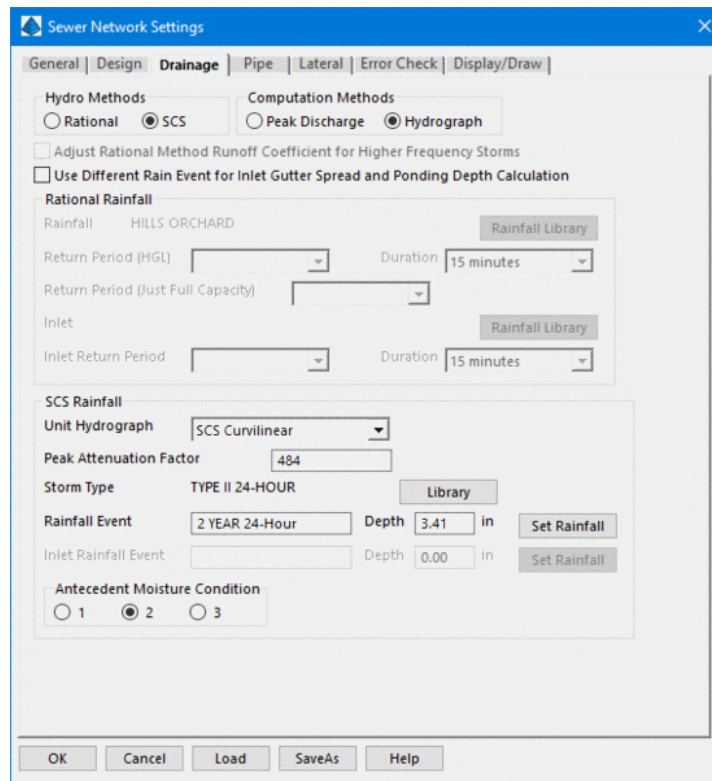
Adjust Rational Method Runoff Coefficient for Higher Frequency Storms this option would time the runoff coefficient with a constant to adjust the runoff coefficient. The constant for 25 year is 1.1, 50 year is 1.2 and 100 year is 1.25. The adjusted runoff coefficient is checked to make sure it's not greater than 1.0

Use Different Rain Event for Inlet Gutter Spread and Ponding Depth Calculation: When enabled, this option enables an alternate rainfall event for either Rational or SCS calculation methods to determine inlet efficiency.

Rational Rainfall: Specify the Rainfall, Return Period and Duration (if using the Hydrograph method) if using the Rational equation.

SCS Rainfall: Specify the Antecedent Moisture Condition, Storm Type and rainfall map if using the SCS methodology.

Unit Hydrograph: For SCS Rainfall, choose between SCS Curvilinear, SCS Triangular and Delmarva unit hydrographs. For SCS Triangular, the Peak Attenuation Factor controls the height of the hydrograph and needs to be a value between 300 and 600. The default of 484 matches the height used by the SCS Curvilinear method.



Cover: Enter the minimum and maximum depth of cover which is the distance from the surface elevation to the crown elevation along the pipes. Alternatively, enable either (or both) *Library* toggle(s) to use the value(s) specified in the Pipe Size Library.

Velocity: Enter the minimum and maximum flow velocity for the pipes. The minimum velocity is about 2 to 3 ft/s (0.6 to 0.9 m/s) when the pipe is flowing full for self-cleansing. The maximum velocity should be less than approximately 15 ft/s (4.5 m/s) to prevent erosion of the pipe interior by suspended sediment and debris. Alternatively, enable either (or both) *Library* toggle(s) to use the value(s) specified in the Pipe Size Library.

Slope: Enter the minimum and maximum slope range for the pipes in the network. The minimum slope should be sufficient to maintain the minimum velocity and the maximum slope is related to the maximum velocity. Alternatively, enable either (or both) *Library* toggle(s) to use the value(s) specified in the Pipe Size Library.

Normal Slope: The normal slope is the initial slope used to place a pipe in the network.

Max. Length: This is the maximum length between structures in a network before an alert is presented indicating a potential problem.

Drop Across Junction: This is the drop across the inside of a junction. Depending on the *Direction* specified on the Design tab, the Invert Up is either raised by this amount or the Invert Down is lowered by this amount.

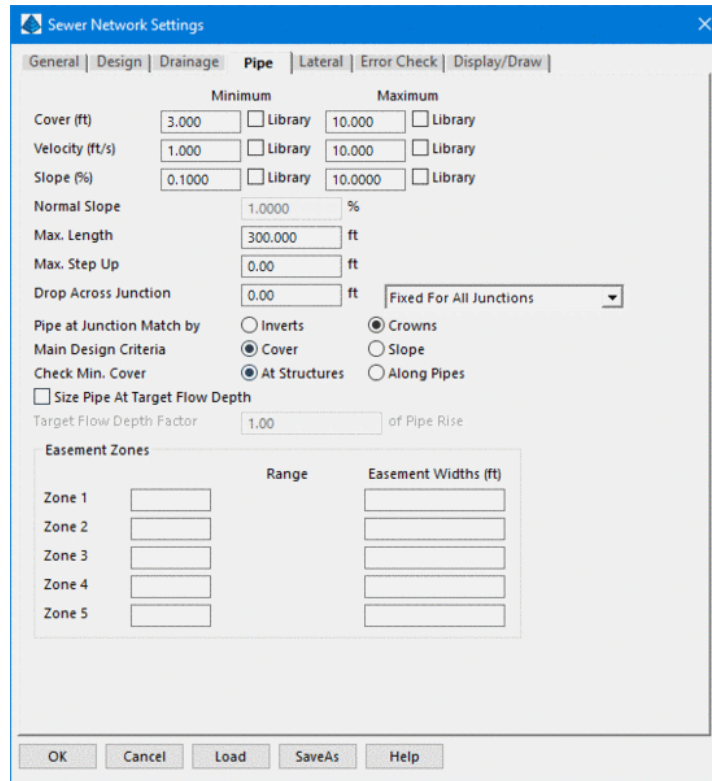
Pipe at Junction Match by: Indicate if pipes should be aligned by their inverts or crowns.

Main Design Criteria: Indicate which criteria should be maintained if both the *Minimum Cover* and *Minimum Slope* cannot be satisfied. If *Cover* is selected, the Slope will yield and vice versa.

Check Min. Cover: Define whether to check the minimum cover at the structures only or along the entire length of each pipe.

Size Pipe At Target Flow Depth: When this option is enabled in design mode, the sewer pipes will be set to the sizes that are sufficient to maintain the target flow depth.

Easement Zones: When this option is enabled, Utility Easements are drawn according to depth of Utility and corresponding Easement Width. Up to 5 widths can be defined which typically increase as depth increases.



Lateral Slope: Allows the user to set a minimum and maximum lateral slope.

Lateral Normal Slope: The normal slope is the initial slope used to place a lateral in the network.

Hold All Lateral Pipe Slopes: This option sets how the lateral should change if mainline edits are made, and will hold the lateral slopes should the mainline be adjusted. Therefore both the connection and the cleanout elevation would change to hold the slopes. For more than one cleanout, all lateral slopes are preserved after modifying the main line pipe.

Hold Slope of the Lateral Pipe Connected to Mainline: This option will adjust the first cleanout and lateral elevation if the mainline is adjusted.

Hold Lateral Inverts At Cleanouts: This option will adjust the lateral connection to the mainline pipe and the cleanout invert would remain at the original elevation.

Use Default Lateral Cleanout Symbol: Allows the user to select a symbol to be used as designation of the cleanout.

Use Default Lateral Connection Symbol: Allows the user to select a symbol to be used as designation of the connection

Initial Lateral Cleanout Name: The user can specify the initial cleanout name. A sequential numbers will start from this setting. For example CO1, CO2, CO3, etc.

Initial Lateral Connection Name: The user can specify the initial connection name. A sequential numbers will

start from this setting. For example CONN1, CONN2, CONN3, etc.

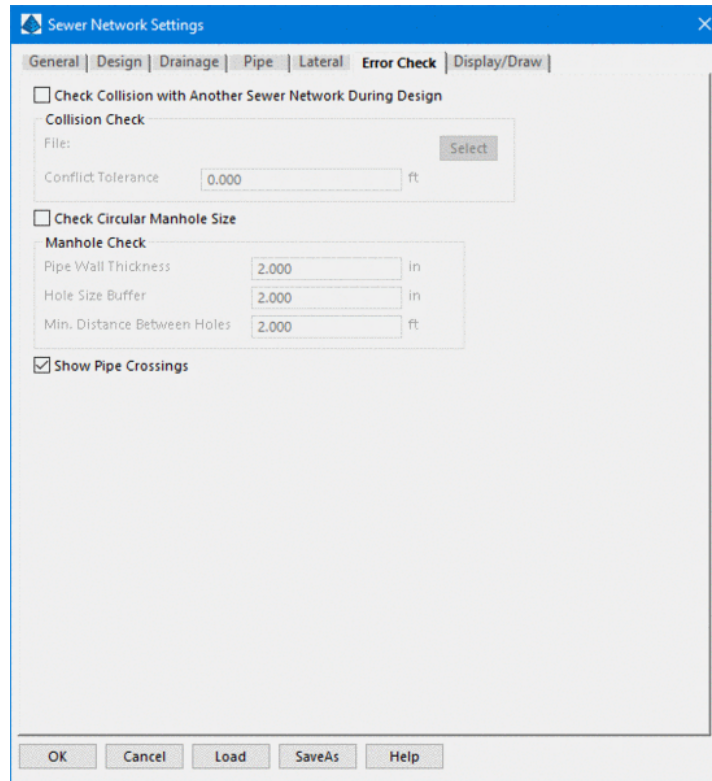
Default Lateral Connection Height: The user can specify the default connection height value for all lateral pipes.

The image shows a screenshot of the 'Sewer Network Settings' dialog box, specifically the 'Lateral' tab. The dialog has a blue title bar and a close button in the top right corner. The main area contains several input fields and checkboxes. At the top, there are tabs for 'General', 'Design', 'Drainage', 'Pipe', 'Lateral', 'Error Check', and 'Display/Draw'. Below the tabs, there are two columns for 'Minimum' and 'Maximum' values for 'Lateral Slope (%)', with values 0.1000 and 10.0000 respectively. Below that is a 'Lateral Normal Slope' field with the value 2.0000 and a '%' symbol. A section titled 'Lateral Edit Options For Mainline Edits' contains three radio buttons: 'Hold All Lateral Pipe Slopes' (selected), 'Hold Slope of the Lateral Pipe Connected to Mainline', and 'Hold Lateral Inverts At Cleanouts'. Below this are two checkboxes: 'Use Default Lateral Cleanout Symbol' and 'Use Default Lateral Connection Symbol', both with 'SPT 65' as a label and a 'Symbol' button. At the bottom, there are four text input fields: 'Initial Lateral Cleanout Name' (CO1), 'Initial Lateral Connection Name' (CONN1), and 'Default Lateral Connection Height' (0.00). The dialog box has a standard Windows-style button bar at the bottom with 'OK', 'Cancel', 'Load', 'SaveAs', and 'Help' buttons.

Check Collision with Another Sewer Network During Design: When enabled, this option allows for potential conflict detection with another sewer network. Should the placement of a structure or pipe encroach on the *Conflict Tolerance*, an alert will be displayed to caution against the potential collision.

Check Circular Manhole Size: When enabled, this option will check if the manhole is big enough to hold the incoming and outgoing pipes and will display an alert if any of the settings are violated.

Show Pipe Crossings: When enabled, this option allows you to display the pipe crossings of current network opposed to any sewer networks or pipes that are drawn in the drawing.



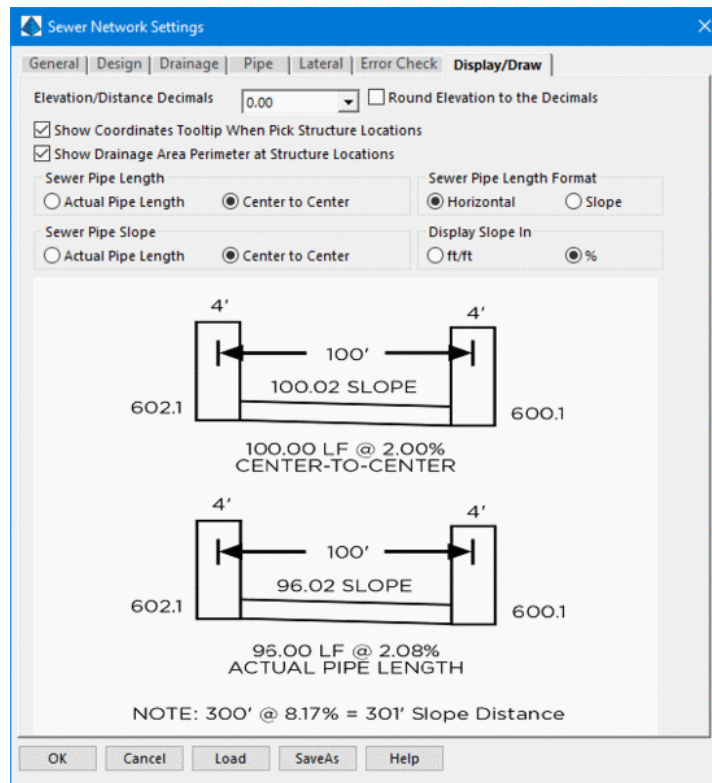
Pipe Length: Specify how the pipe length should be displayed.

Pipe Slope Annotation: Specify how the pipe slope should be displayed.

Display Slope In: Specify to display slope in the unit of either distance/distance or %.

Elevation Decimals: Indicate the desired amount of precision to display for elevation values.

Show Coordinates Tooltip When Pick Structure Locations: Show cursor's coordinates in a tooltip when pick new structure locations.



Pulldown Menu Location: Network > Sewer Network Setup

Keyboard Command: swrconfig

Prerequisite: None

Set Sewer File

This command sets a sewer network file as the current file. The other sewer network commands will reference this file. Either a new file can be created or an existing sewer file can be modified. The sewer network file stores all the sewer structure data (elevation, flow) and all the network connection data (slopes, pipe sizes). This file has a .sew file extension.

Pulldown Menu Location: Network

Keyboard Command: setswr

Prerequisite: none

Set Surface File

Use this command to set the grid or triangulation file to be used to compute sewer manhole surface elevations and minimum cover along pipe lengths, within the Sewer Network commands, and in particular the command Create Sewer Structure. A dialog will appear requesting the name of the surface file to be used.

Pulldown Menu Location: Network

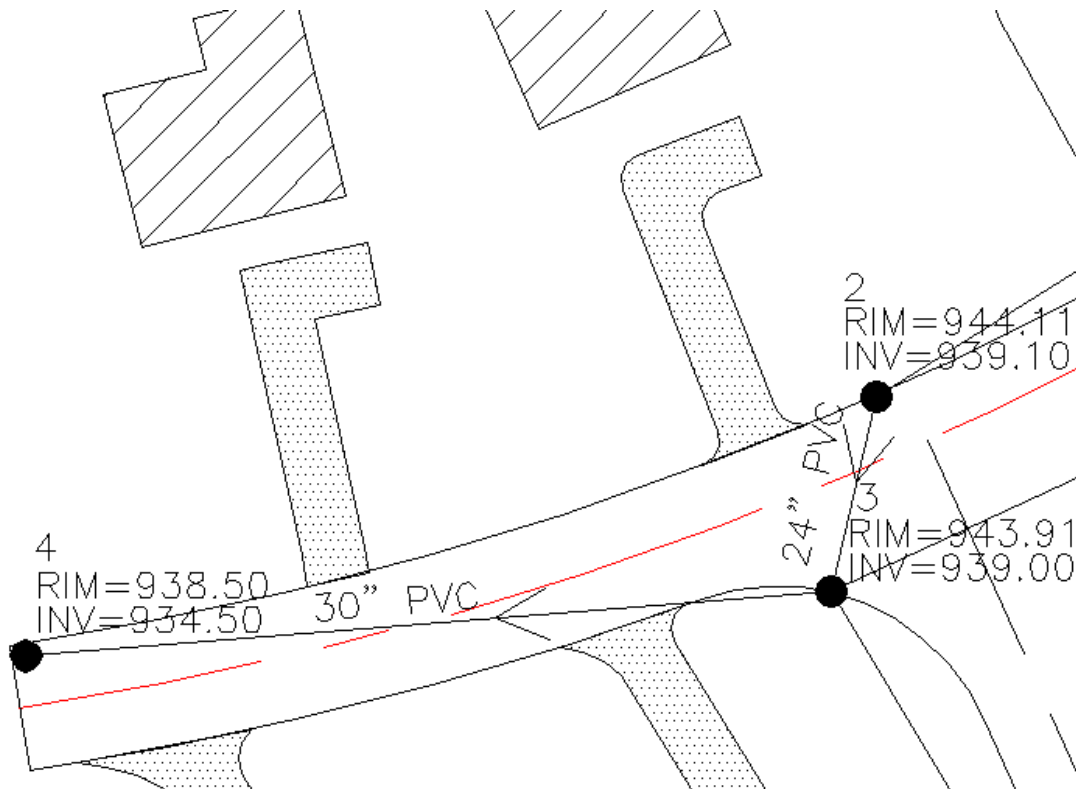
Keyboard Command: setgrd

Prerequisite: None

Plan View Label Settings

This command sets the drawing format for sewer pipeline and inlet/manhole annotations drawn for the sewer network. The settings are entered in a dialog with tabs for Structure Labels, Pipe Labels and General Settings.

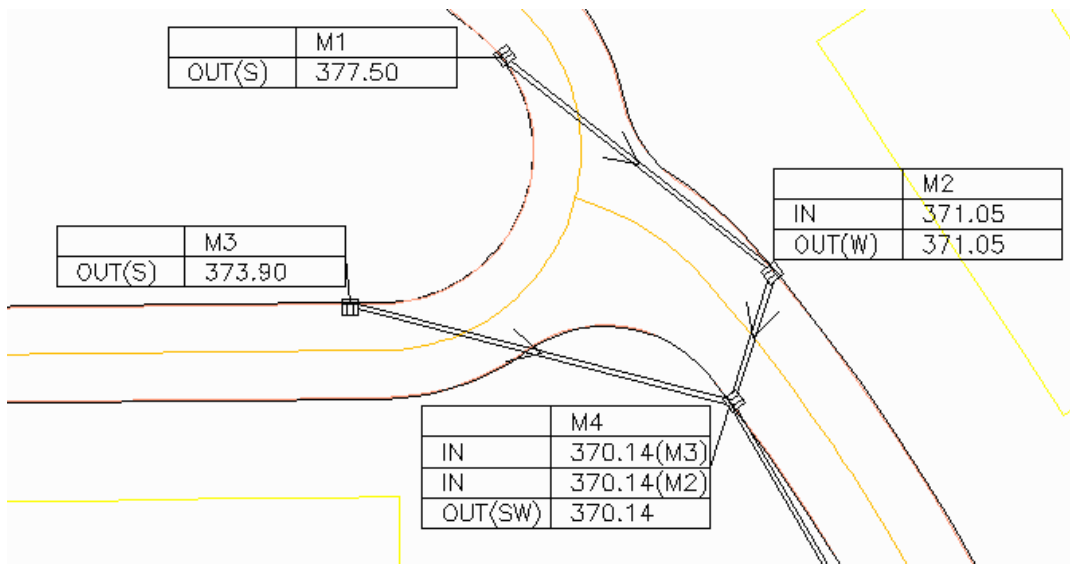
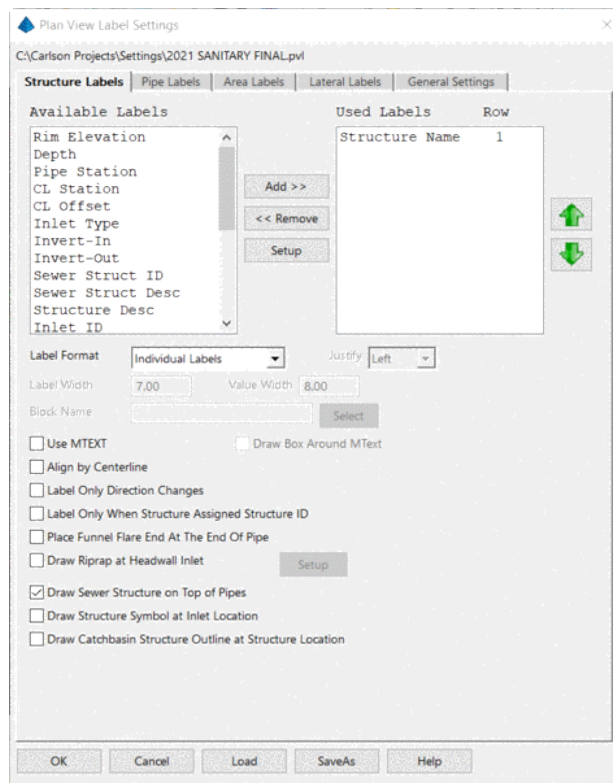
Shown below is the manhole or inlet name, along with rim elevation, invert elevation and the labeling of the sewer pipe itself. When the network entities are selected by the 3D Viewer Window command, they are automatically converted into 3D entities.



Under **Structure Labels**, you can choose whether to label the structure name, structure description, northing, easting, rim elevation, depth, pipe station, CL station, CL offset, inlet type, invert-in elevation, invert-out elevation, structure ID, Inlet ID or custom offsets. The custom offsets label elevations relative to either the structure rim elevation or invert elevation with a specified offset. The list of available fields is on the left and the list of fields to label is on the right. The order of the fields in the right side list is the order of the labels in the drawing. Use the Add, Remove and Up/Down arrow buttons to move fields between the lists and change their order. To edit the parameters for a field, highlight the field name on the right list and pick Setup. You can set the label prefix and suffix and whether to start a new row. If new row is off, then the field will be put on the same row as the previous field in the list. The pipe station is the accumulative length of the pipes from the outfall up to the structure. The CL station and offset values are calculated from the reference CL that is assigned to the structures in the sewer network. The invert elevations can be positioned either above the structure or along the associated pipe direction. For Inverts, the Add Quadrant option adds the bearing quadrant of the associated pipe direction to the invert label prefix.

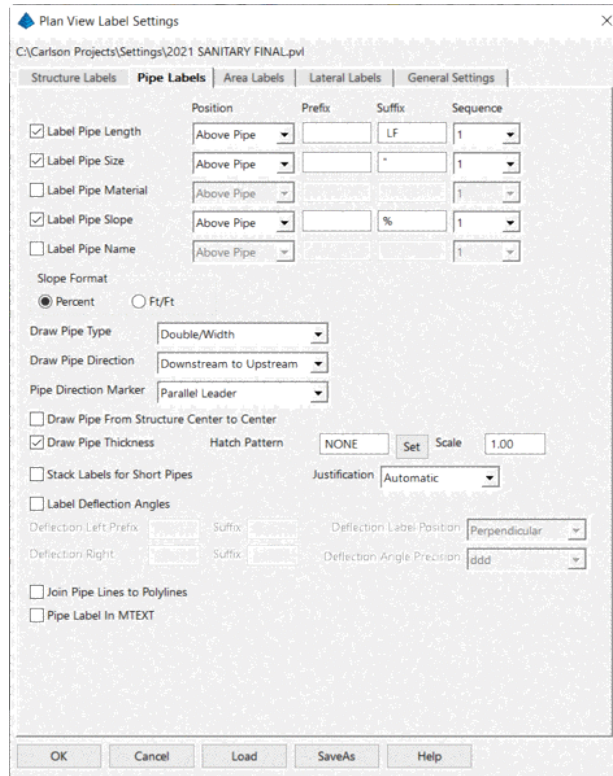
The options for **Label Format** are Individual Labels, Data Table and Attribute Block. For Individual Labels, the program draws regular text entities. The **Use MText** option will draw the labels as MTEXT entities. Otherwise they are drawn as TEXT entities. When draw in MText, the option **Draw Box Around MText** will draw a box around the MText Structure Labels. The **Data Table** method will put the labels in a block as shown below. There are settings for the size of the block columns and the block label justification. The **Attribute Block** method inserts a block and puts the labels into the block attributes. The **Align By Centerline** option will rotate the labels to be parallel with the pipe. Otherwise the labels are drawn horizontal to the current twist screen. The **Locate Only Direction Changes** will only label when the pipes to the structure have a deflection angle. This setting applies to utility networks that have a lot of nodes in straight lines and you only want to label the end nodes. The **Label Only When Structure Assigned Structure ID** option will only draw the structure labels when the structure has an ID other than None. This option is meant for clearing up unnecessary labels for cases like utility networks that have a lot of structures without ID's. The Place Funnel Flare End At The End Of The Pipe relocates the funnel to be part of the pipe length. The **Draw Riprap At Headwall Inlet** draws a hatch pattern using the pattern and size set under

Setup. In default, the pipe entities are drawn on top of sewer structure entities. The **Draw Sewer Structure on Top of Pipes** allows you to draw structures on top of pipes. **Draw Structure at Inlet Symbols at Inlet Locations** if the user has an inlet offset specified this will place symbol at the inlet location. **Draw Catch Basins Structure Outline at Structure Location** will draw the outline with the structure dimension at the structure location.

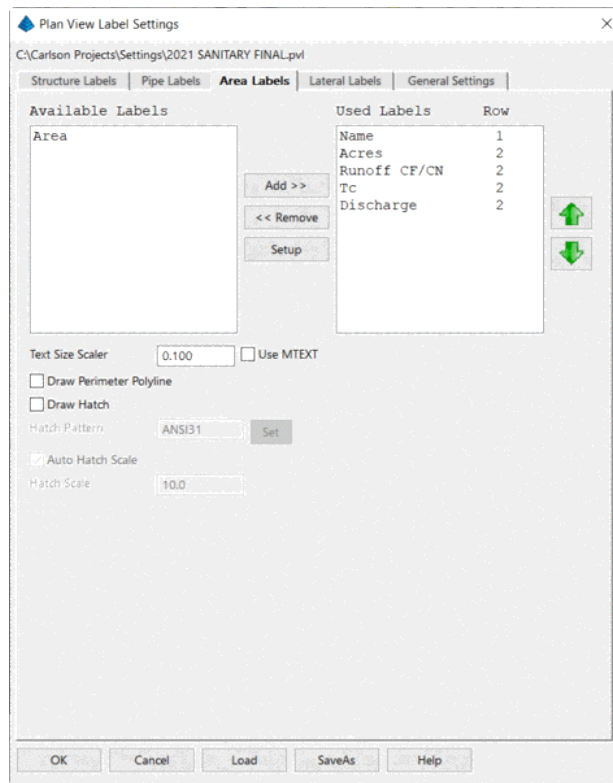


Under **Pipe Labels**, you can choose whether to label the pipe size, material, length, name or slope. For each label, there are settings for the prefix and suffix, for whether to put the label above or below the pipeline, and controls for label sequence of the fields. **Draw Pipe Direction** allows user to select Downstream to Upstream or Upstream to Downstream. The **Pipe Direction Marker** has two styles for flow direction arrows. The **Draw Pipe Line Type** sets the method for drawing the pipelines as 2D polylines, 3D polylines or parallel 2D polylines set apart with the width of the pipe. The pipe is drawn at outside border of the symbols, however the **Draw Pipe From Structure Center to Center** will draw pipes from structure center to center. The **Draw Pipe Thickness** will show the thickness of the pipe

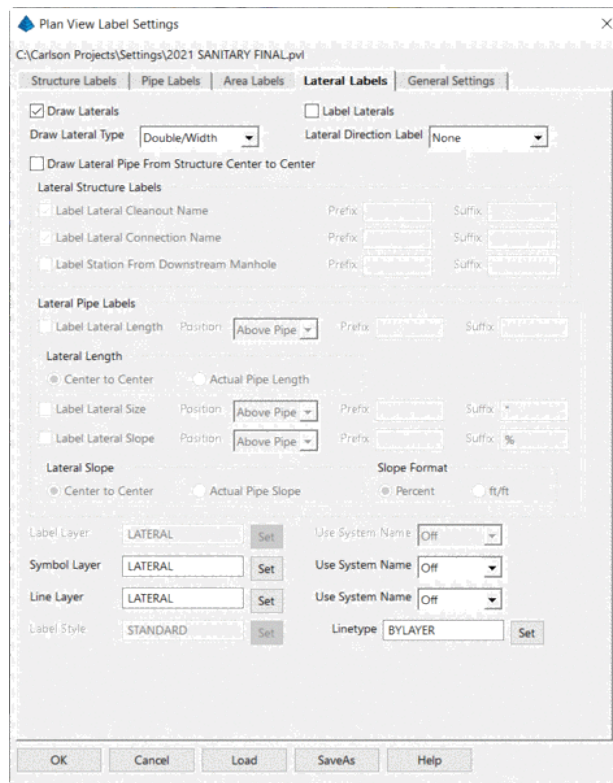
with the option to hatch. The **Stack Labels For Short Pipes** option will automatically make a stacked row of labels when the pipe segment is too short to fit on a single row. The **Deflection Angle** options have separate prefix/suffix settings for left and right, and the label rotation can be horizontal to the current screen view or perpendicular or parallel to the pipe. **Pipe Label in MTEXT** allows the user to label pipe runs using MTEXT.



For **Area Labels**, the program can label the drainage parameters for each inlet including area, runoff coefficient, T_c and discharge. There is also the ability to draw a perimeter polyline for the drainage area and to hatch the area.



For **Lateral Labels**, you can choose whether to Draw Laterals and Label Laterals. The Draw Lateral Type allows the user to select 2D Polyline, 3D Polyline, Double/Width, or Dashed/Width. Lateral Direction Label has three styles None, Parallel Leader, or Arrow On Pipe to show the direction of flow along the lateral. For the Lateral Structure Labels the user can specify a Prefix or Suffix for the Label Cleanout Name, Label Lateral Connection Name, and Label Station From Downstream Manhole by placing a check box with these two options if you choose to label. The check box toggle for Label Lateral Length the user can control the position of either Above Pipe or Below Pipe and also add a Prefix or Suffix. For Lateral Length the labels can be based on structure center-to-center or actual pipe dimension that removes the width of the structure and goes from the structure edges, and also specify what is to be labeled with a position, Prefix, or Suffix. The Lateral Slope the labels can be based on structure center-to-center or actual pipe slope, and toggle between the Slope Format of Percent or ft/ft. Layers can be assigned for the Labels, Symbols, and Linework and also the System Name as either Off, Prefix, or Suffix. Use the Label Style to assign text standard and set the Linetype.



Under **General Settings**, there are controls for the layers, styles, decimal places, sizes and linetypes.

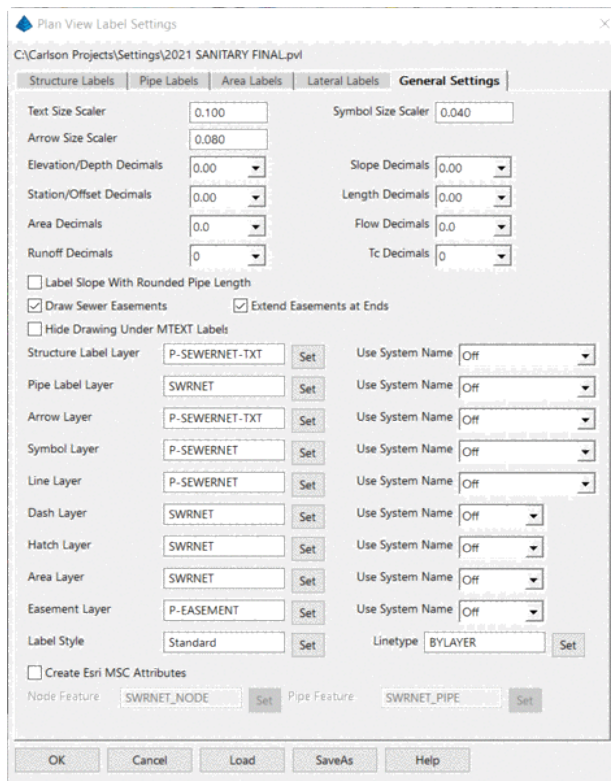
Label Slope With Rounded Pipe Length: would calculate and label pipe slope based on the rounded pipe length by the Length Decimals.

Draw Sewer Easements: would draw easement polylines, which width is based on the easement data defined in sewer network settings dialog.

Extend Easements at Ends: Will extend easement polylines past both ends of a network based on 1/2 of the easement total width.

Hide Drawing Under MText Labels: uses wipeout w or w/o a frame, which uses the Draw Box feature, to hide drawing under the MText labels.

The Use System Name has settings for adding to the layer names using any combo of the system name, line name, pipe size and pipe material. The linetype is on used when the pipe is draw as a 2D polyline. The Create ESRI MSC Attributes option will tag the network entities with ESRI format feature attributes which ESRI version 9.3 and higher will recognize as the specified feature names instead of plain CAD entities.



You are free to move the text anywhere desired for better appearance after it plots. The labeling will change automatically on the drawing if any of the sewer network information is edited or if the label settings are changed. This automatic redraw will put the labels back in their original positions if you moved the labels with standard drafting edit tools. If the Move Sewer Label command is used, the labels will stay at their modified position even after the automatic redraw. The labeling and manhole itself will be removed from the screen by the command Remove Sewer Structure, along with connecting pipe sizes and invert elevations of the immediate upstream and downstream manholes. The command Draw Sewer Network–Plan View will also redraw and label the sewer network that is "set" and current, according to the annotation parameters of this command.

Pulldown Menu Location: Network

Keyboard Command: swrsetup

Prerequisite: None

Save Sewer Network File

This routine re-saves the current sewer network file in another name, which can act as a backup file or "snapshot" of the sewer network design at a certain point in time. The file can be re-loaded later and re-used.

Pulldown Menu Location: Network

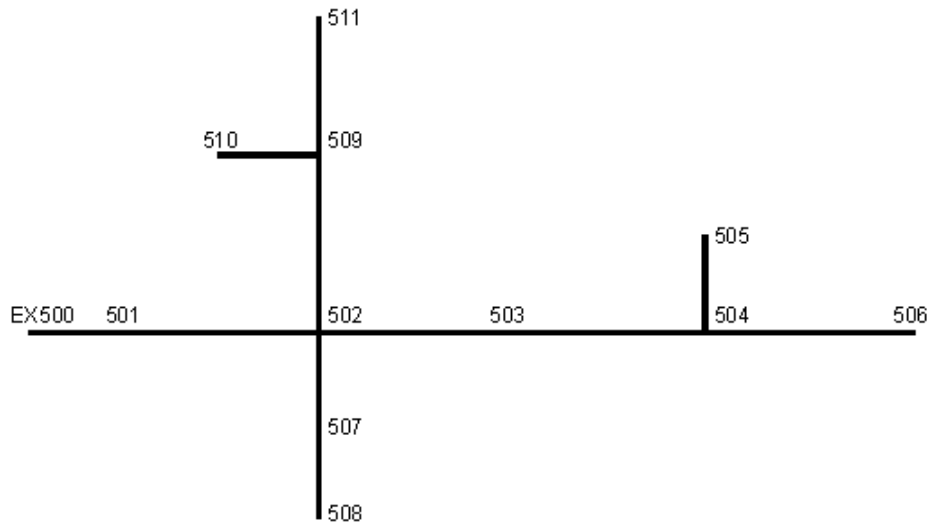
Keyboard Command: save_sewer

Prerequisite: Sewer network (.SEW) file

Import Haestad Network

This routine converts Haestad files into Carlson sewer files. Haestad sewer network files have fourteen pieces of information in an ASCII text file. Unfortunately, they do not contain two-dimensional coordinates of the manholes locations. Therefore, the routine must assume locations along some arbitrary datum. Examine the example output below. The program searches for the longest path along the sewer network, and places it on the X axis. The next

longest set(s) of network traces branch from the longest path. The process is repeated until the last upstream manhole on the network tree branches is encountered. While true coordinate locations of the manholes is unknown, additional hydraulic analysis of the system can be made on this Carlson pseudo-layout.



Pulldown Menu Location: Network->Sewer Network Utilities

Keyboard Command: ha2csnet

Prerequisite: Haestad file

Rational Rainfall Library

Rainfall is the source of the ground runoff. Along with the watershed conditions, the rainfall that neither infiltrates nor gets trapped in low areas and depressions contributes to the direct surface runoff, upon which the storm drainage system design is based.

The Rational Rainfall Library use IDF curves to provide average rainfall intensity data for particular storm events. With a known rainfall duration and frequency, an intensity is calculated via the IDF curve and applied to the Rational Method to obtain the peak flow for designing the sewer network. There are six methods to input rainfall data: TP-40 rainfall map, Hydro-35 rainfall map, rainfall accumulation, rainfall intensity, IDF equation coefficients and fixed rainfall intensity. Rainfall maps provided by government organizations are practical rainfall data resources for engineering design. TP-40 maps show precipitation depths in the US for storm durations from 1 hour to 24 hours and for recurrence intervals from 1 to 100 year. Hydro-35 maps are for the central and eastern US, and provide rainfall data for durations as short as 5 minutes. Please refer to HEC-12 for details on methodology for computing IDF curves from rainmaps. In addition to TP-40 and Hydro-35 method, you can input rainfall accumulations or intensities at various storm durations to define IDF curves. If you already have the IDF equation coefficients calculated, you can enter the coefficients directly to define the IDF curves. The IDF curves are interpolated linearly between the data points. If you choose to use fixed rainfall intensity, no IDF curve is calculated and this value is used directly. You may also input rainfall intensity data with lookup table method, where IDF curve is not used, rainfall intensity will be interpolated linearly from the lookup table.

The IDF equation for a given return period is defined as follows. The coefficients A, B and M are calculated by log-log regression of the rainfall intensity and (t + B).

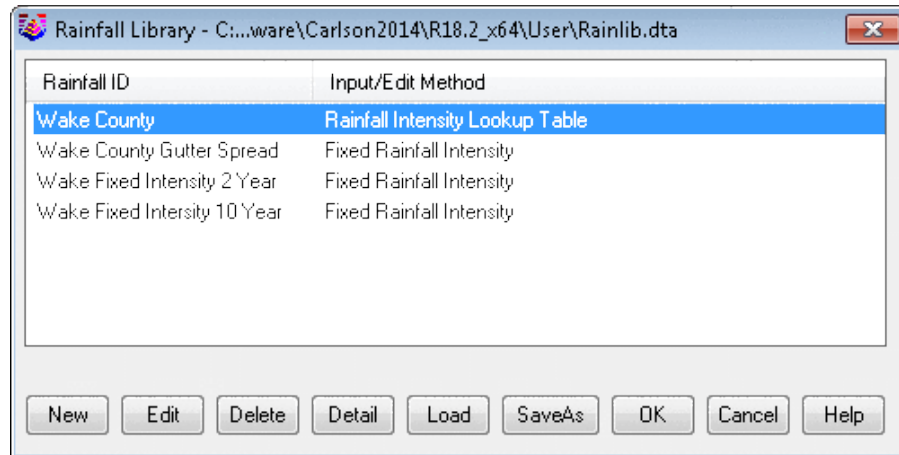
$$I = A / (t + B)^M$$

where: I = rainfall intensity (in/hr, or mm/hr in metric)

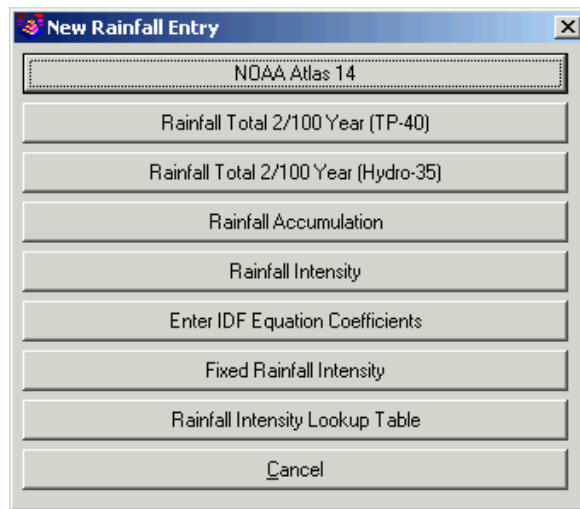
A, B, M = equation coefficients for a given return period

t = rainfall duration (min.)

The Rainfall Library stores rainfall data in a library file under the ...\\USER folder and is available for all projects. From the Network > Sewer Network Libraries menu in the Hydrology Module, select the Rainfall Library to open the library dialog to edit rainfall data. The dialog lists all rainfall entries by their ID and their input/edit method. New button creates a new rainfall through one of the seven methods: TP-40, Hydro-35, Rainfall Accumulation, Rainfall Intensity, Enter IDF Equation Coefficients, Fixed Rainfall Intensities, and Rainfall Intensity Lookup Table. Edit button allows you to modify an existing rainfall, and the Delete button removes the highlighted rainfall from the library. Load and SaveAs buttons allow you to load and save the rainfall data.



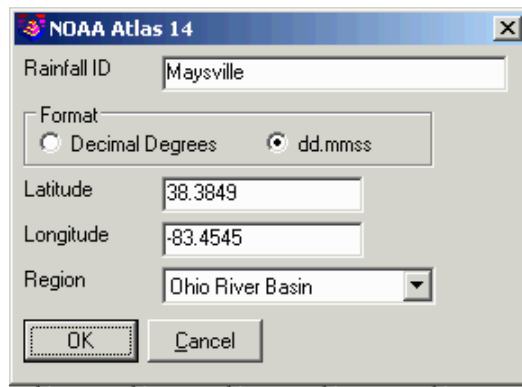
Rainfall Library Dialog



New Rainfall Dialog

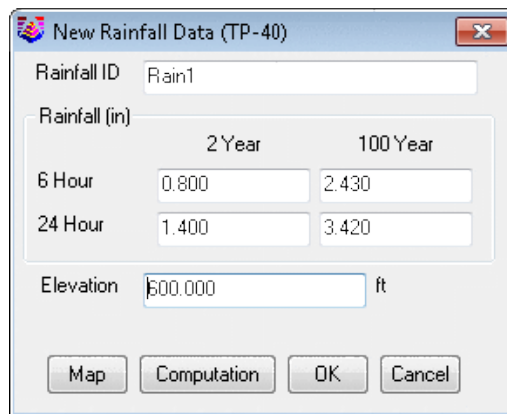
NOAA Atlas 14

This method fills out the rainfall intensity values using the NOAA Atlas 14 rainfall models. In the dialog, enter a name and lat/lon for your location and choose the NOAA Atlas 14 region. The first time that you run a region, the program downloads the rainfall models to your computer.

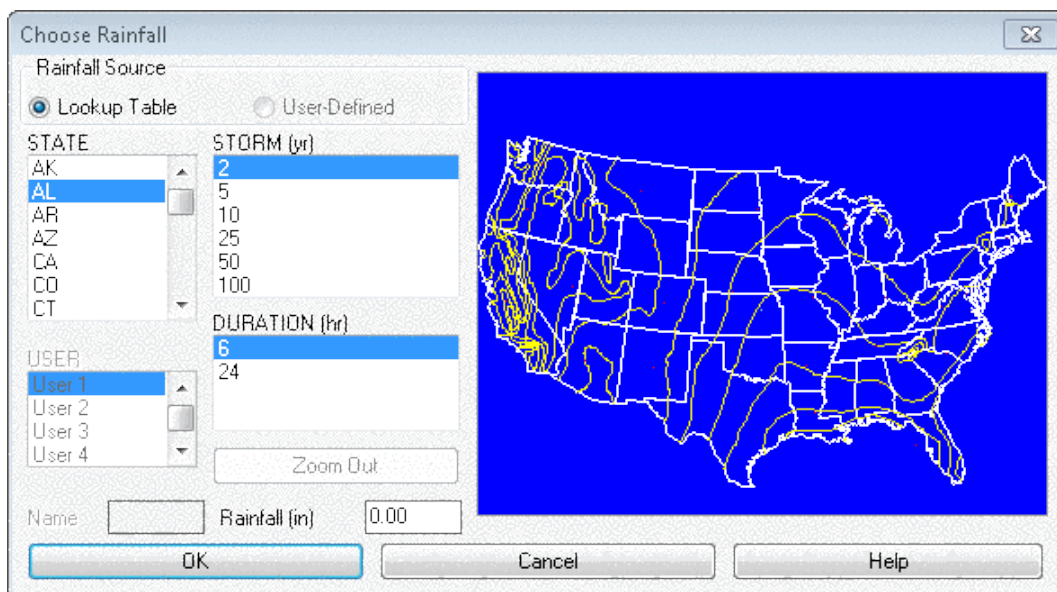


Rainfall Total 2/100 Year (TP-40)

The TP-40 method is used to define IDF curves for the Western states in the US. It requires rainfall accumulations of 6-hour and 24-hour storm durations for the 2-year and 100-year storms, and the elevation of the location. In the TP-40 dialog, type the rainfall name in the Rainfall ID box. The rainfall depth can be entered either manually or from the TP-40 maps. Click on Map button to open the rainfall map, pick a state on the map to zoom in to the state map, and then pick a location to get the rainfall depth of 6-hour and 24-hour duration for 2-year, 5-year, 10-year, 25-year, 50-year and 100-year storm events. In the Elevation box, enter the surface elevation at the design location. Computation button computes the rainfall intensities and the IDF coefficients, and displays the result in the Rainfall Intensity dialog. Click on OK button to commit the rainfall entry.



TP-40 Rainfall Data Dialog



TP-40 Rainfall Map

Rainfall Intensity (in/hr)					
	5 min	10 min	15 min	30 min	60 min
2 Year	1.887	1.464	1.236	0.857	0.542
5 Year	3.570	2.770	2.339	1.621	1.026
10 Year	4.843	3.757	3.173	2.199	1.392
25 Year	6.526	5.063	4.276	2.963	1.875
50 Year	7.799	6.051	5.110	3.541	2.241
100 Year	9.072	7.039	5.944	4.119	2.607

Coefficients of IDF Equation			
	A	B	M
2 Year	12.870	7.900	0.751
5 Year	24.351	7.900	0.751
10 Year	33.036	7.900	0.751
25 Year	44.518	7.900	0.751
50 Year	53.203	7.900	0.751
100 Year	61.889	7.900	0.751

Intensity Calculation
 Return Period: min
 Time of Concentration: min
 Intensity: 0.489 in/hr

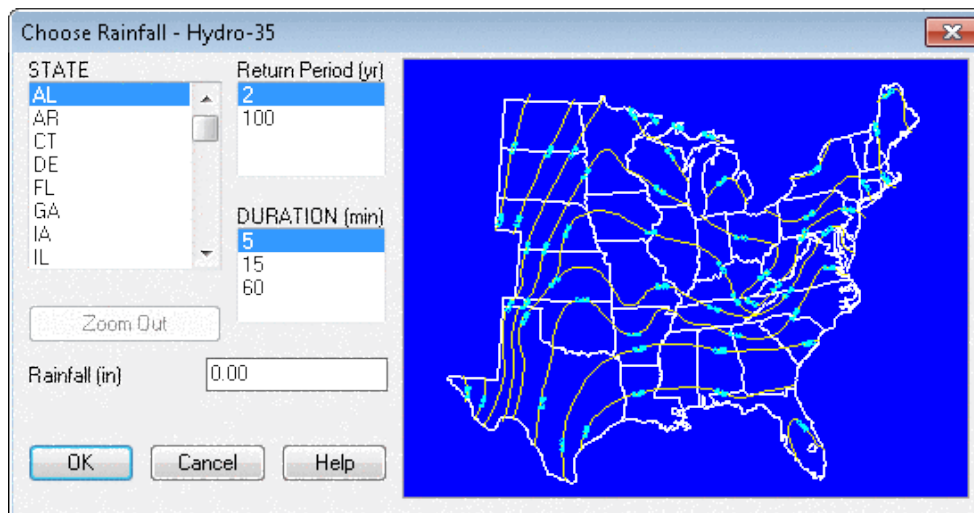
Rainfall Intensity Results

Rainfall Total 2/100 Year (Hydro-35)

The Hydro-35 method is used to define IDF curves for the Central and Eastern states in the US. It requires rainfall accumulations of 5-min, 15-min and 60-min storm durations for the 2-year and 100-year storms. In the Hydro-35 dialog, type the rainfall name in the Rainfall ID box. The rainfall depth can be entered either manually or from the Hydro-35 maps. Click on Map button to open the rainfall map, pick a state on the map to zoom in to the state map, and then pick a location to get the rainfall depth. Computation button computes the rainfall intensities and the IDF coefficients at 2-year, 5-year, 10-year, 25-year, 50-year and 100-year return periods, and displays the result in the Rainfall Intensity dialog. Click on OK button to commit the rainfall entry.

Rainfall ID	<input type="text" value="Rain2"/>	
Rainfall (in)	2 Year	100 Year
5 min	<input type="text" value="0.350"/>	<input type="text" value="0.790"/>
15 min	<input type="text" value="0.670"/>	<input type="text" value="1.570"/>
60 min	<input type="text" value="1.300"/>	<input type="text" value="2.720"/>
<input type="button" value="Map"/> <input type="button" value="Computation"/> <input type="button" value="OK"/> <input type="button" value="Cancel"/>		

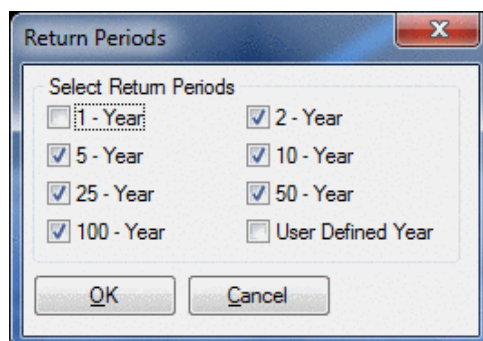
Hydro-35 Rainfall Data



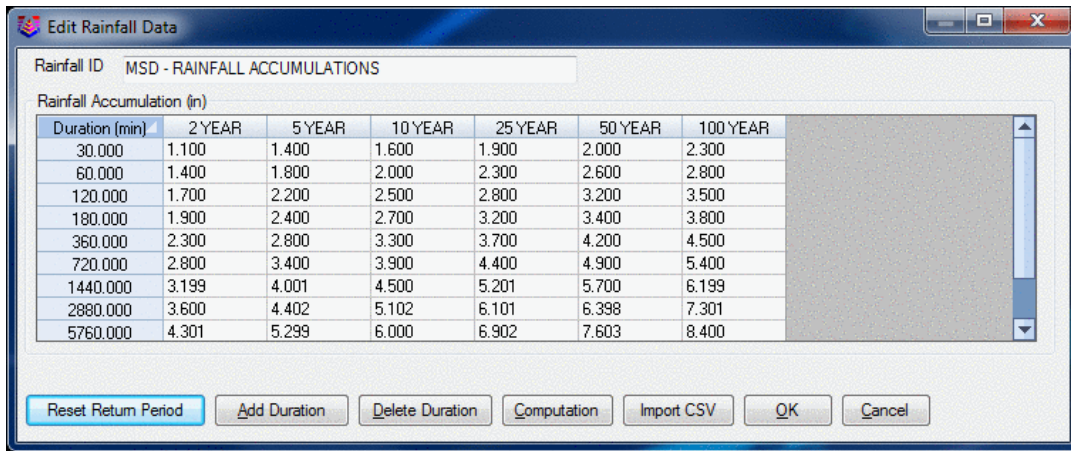
Hydro-35 Rainfall Map

Rainfall Accumulation

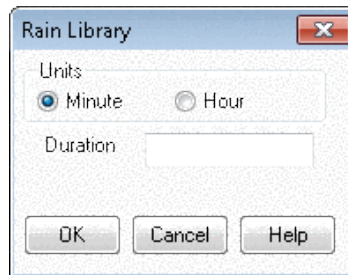
This method allows you to enter the rainfall accumulations of various durations for any of 1-year, 2-year, 5-year, 10-year, 25-year, 50-year, 100-year, and a User-Defined return periods for computing the IDF curves. The **Reset Return Period** button allow the user to add or remove Return Periods as needed. **Add a Duration** button adds new duration entry to the spreadsheet. For accuracy, three or more durations are required. **Delete a Duration** button deletes the highlighted duration entry. **Computation** button computes the rainfall intensities and the IDF coefficients, and displays the result in the Rainfall Intensity dialog. The **Import CSV** button allows you to import the rainfall depth values in a comma separated values text file. In the CSV file, the first column is the duration value in minutes, the rest columns are rainfall depths associated to the chosen return periods. Click on OK button to commit the rainfall entry.



Return Periods Dialog



Customed Rainfall Accumulations



New Duration Entry

Rainfall Intensity

This method allows you to enter the rainfall intensities of various durations for any of 1-year, 2-year, 5-year, 10-year, 25-year, 50-year 100-year, and User Defined return periods for computing the IDF curves. The **Reset Return Period** button allow the user to add or remove Return Periods as needed. **Add a Duration** button adds new duration entry to the spreadsheet. For accuracy, three or more durations are required. **Delete a Duration** button deletes the highlighted duration entry. **Computation** button computes the rainfall intensities and the IDF coefficients, and displays the result in the Rainfall Intensity dialog. The **Import CSV** button allows you to import the rainfall intensity values in a comma separated values text file. In the CSV file, the first column is the duration value in minutes, the rest columns are rainfall intensities associated to the chosen return periods. Click on OK button to commit the rainfall entry.



Customed Rainfall Intensities

Enter IDF Equation Coefficients

When you have the coefficients calculated, you can use this method to enter the coefficients to obtain the actual IDF curve equation. In the following spreadsheet dialog, enter the known coefficients A, B and M to create the IDF equation. **Computation** button computes the rainfall intensities and displays the result in the Rainfall Intensity dialog. Click on OK button to commit the rainfall entry.

	A	B	M
2-Year	22.170	5.090	0.670
5-Year	29.910	5.090	0.670
10-Year	35.760	5.090	0.670
25-Year	43.500	5.090	0.670
50-Year	49.350	5.090	0.670
100-Year	55.210	5.090	0.670

IDF Equation Coefficients

Fixed Rainfall Intensity

The method creates a rainfall entry with the fixed rainfall intensity. When you choose to use this storm event, this fixed intensity will be used in your hydrology calculation directly.

Return Period	A	B	M
2-Year	22.170	5.090	0.670
5-Year	29.910	5.090	0.670
10-Year	35.760	5.090	0.670
25-Year	43.500	5.090	0.670
50-Year	49.350	5.090	0.670
100-Year	55.210	5.090	0.670

Fixed Rainfall Intensity

Rainfall Intensity Lookup Table

The method allows you to enter the rainfall intensities of various durations for any of 1-year, 2-year, 5-year, 10-year, 25-year, 50-year, 100-year, and User Defined return periods and creates a rainfall intensity lookup table. The rainfall intensity will be interpolated linearly from the table.

Rainfall ID: Rain7

Rainfall Intensity (in/hr)

Duration (min)	2-Year	10-Year	25-Year	100-Year
5.000	5.580	7.080	7.780	8.640
10.000	4.460	5.660	6.190	6.860
15.000	3.740	4.780	5.240	5.780
30.000	2.580	3.460	3.880	4.430
60.000	1.620	2.250	2.580	3.050
120.000	0.940	1.330	1.540	1.850
360.000	0.400	0.580	0.680	0.840
1440.000	0.140	0.210	0.240	0.300

Buttons: Reset Return Period, Add Duration, Delete Duration, Computation, OK, Cancel

Rainfall Intensity Lookup Table

Users in Kentucky and some states

Users in Kentucky and some states have started using rainfall data from NOAA Atlas 14 and require project specific rainfall information. The rainfall data can be obtained here: http://hdsc.nws.noaa.gov/hdsc/pfds/orb/ky_pfds.html. After selecting the data type and the location, the precipitation frequency estimate table from NOAA Atlas 14 would be shown as follows. You can then enter the data by either the Rainfall Accumulation method or Rainfall Intensity method into Carlson. You are not require to enter all the data, three durations of rainfall data is enough for creating the IDF curves for each return period.

NOAA's National Weather Service
Hydrometeorological Design Studies Center
Precipitation Frequency Data Server (PFDS)

www.nws.noaa.gov

Home Site Map News Organization Search

KENTUCKY

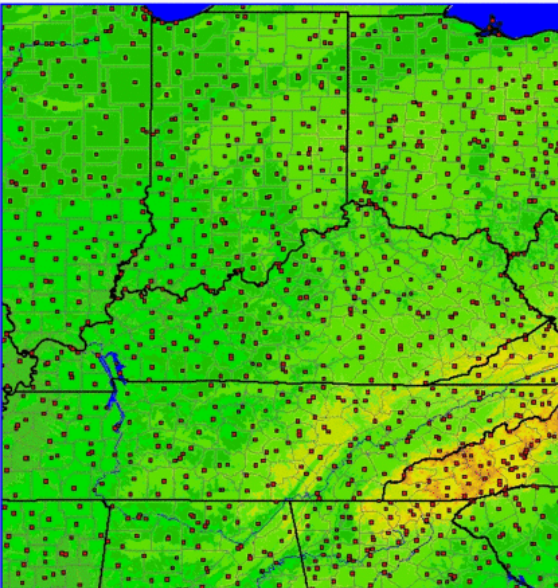
1. DATA DESCRIPTION:
 Data type:
 Units:
 Time series type:

2. SELECT LOCATION:
 Choose one of the following options:

2.1 Select site from list:

2.2 Enter location:
 Latitude (decimal degrees):
 Longitude (decimal degrees):

2.3 Click on map to select location information:
 Latitude:
 Longitude:



Precipitation Frequency Estimates (inches)																		
ARI+ (years)	5 min	10 min	15 min	30 min	60 min	120 min	3 hr	6 hr	12 hr	24 hr	48 hr	4 day	7 day	10 day	20 day	30 day	45 day	60 day
1	0.34	0.54	0.67	0.92	1.15	1.36	1.46	1.78	2.16	2.67	3.23	3.70	4.43	5.13	7.06	8.76	11.06	13.35
2	0.40	0.63	0.80	1.10	1.38	1.63	1.75	2.12	2.57	3.18	3.84	4.40	5.27	6.08	8.36	10.36	13.03	15.69
5	0.46	0.74	0.93	1.32	1.70	1.99	2.13	2.58	3.12	3.84	4.63	5.26	6.31	7.19	9.80	12.07	14.95	17.88
10	0.51	0.82	1.04	1.50	1.96	2.29	2.46	2.96	3.57	4.37	5.26	5.92	7.12	8.03	10.86	13.30	16.30	19.42
25	0.58	0.92	1.17	1.73	2.31	2.71	2.91	3.50	4.22	5.10	6.12	6.79	8.21	9.12	12.18	14.81	17.93	21.23
50	0.63	1.01	1.27	1.92	2.60	3.06	3.28	3.95	4.75	5.69	6.79	7.47	9.07	9.95	13.14	15.90	19.07	22.50
100	0.68	1.09	1.38	2.10	2.90	3.43	3.67	4.43	5.31	6.30	7.48	8.14	9.93	10.76	14.06	16.92	20.11	23.63
200	0.74	1.17	1.47	2.29	3.22	3.82	4.09	4.95	5.91	6.92	8.19	8.80	10.79	11.56	14.93	17.87	21.06	24.65
500	0.81	1.27	1.60	2.56	3.67	4.37	4.68	5.67	6.76	7.79	9.14	9.68	11.95	12.59	16.02	19.02	22.17	25.83
1000	0.86	1.36	1.70	2.76	4.02	4.82	5.16	6.26	7.45	8.47	9.89	10.35	12.85	13.36	16.80	19.85	22.95	26.64

Point Precipitation Frequency Estimates From NOAA Atlas 14

Default Libraries

A default library is included in the install and placed in the Carlson Projects\Settings folder. This library is for Ohio and includes 4 IDF curves for different regions. Use the Load function to select this library. There are also many default libraries for North Carolina under Carlson Projects\Settings\NorthCarolina.

Pulldown Menu Location: Network > Sewer Network Libraries > Rational Rainfall Library

Keyboard Command: rainlib

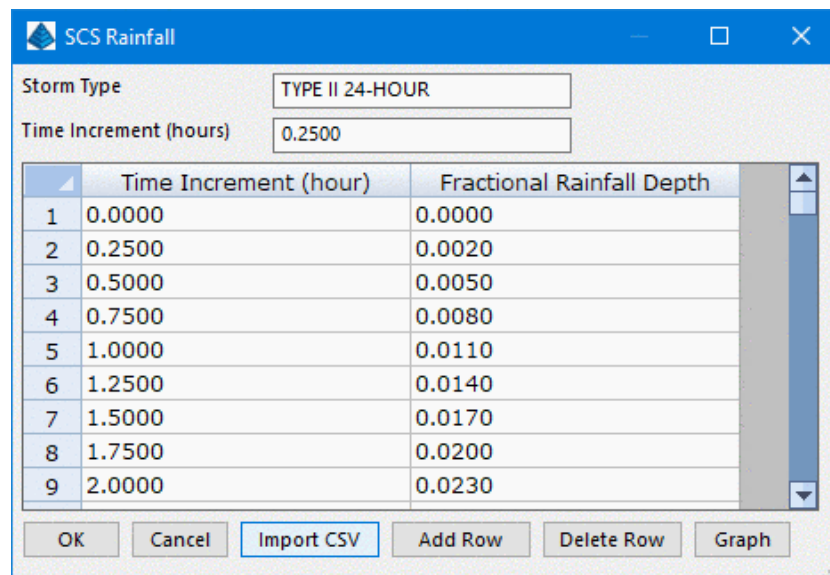
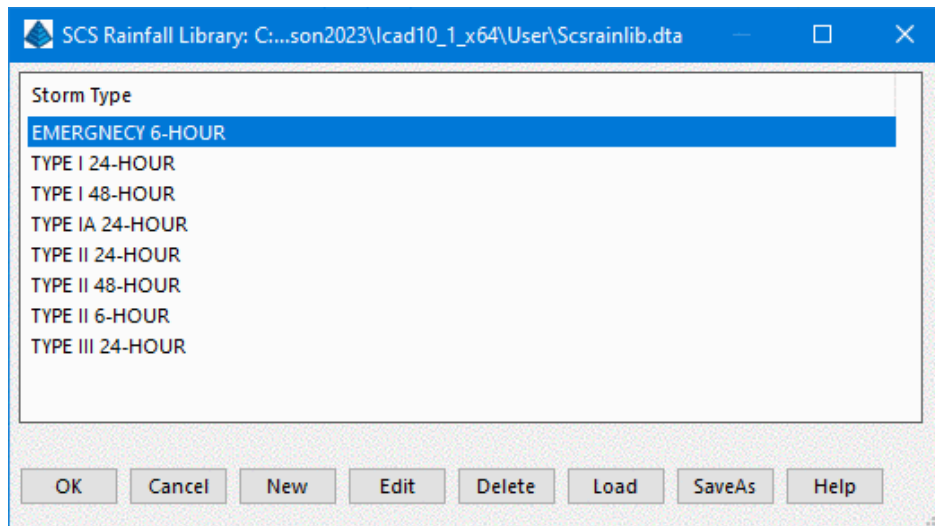
Prerequisite: None

SCS Rainfall Library

The SCS Rainfall Library command allows you to store commonly used storm types for SCS method. The SCS has developed several standard distributions that cover the entire US, commonly known as the Type I, IA, II, and III storms. For situations where a standard rainfall distribution is not available or appropriate, you can create a

localized rainfall distribution table in the SCS Rainfall Library. The default library file is in the Carlson Install Directory\USER folder and is available for all projects in sewer network design.

From the Network > Sewer Network Libraries menu in the Hydrology Module, select SCS Rainfall Library to open the library dialog. New button creates a new storm type entry, Edit button allows you to modify the highlighted entry, and Delete button removes the highlighted entry from the library. Load and SaveAs buttons allow you to load and save the library data. The Import CSV or (comma-separated values) allows the user to import rainfall depth data in csv or txt file formats. In the CSV file, only the fractional rainfall depth at every time increment is required. The fractional rainfall depth data should be between 0 and 1, and delimited by comma. The time increment value is entered on the dialog.



We have collected most of the rainfall distributions that are published on the NOAA website. The rainfall distribution data are all in 24-hour duration, and 90% frequency, the fractional rainfall depth value is between 0 and 1. You can

load the SCS rainfall library file (.srn) of the desired region to the library to use it. Available regions are as follows:

California:

Data Files: Region 1 - NOAA California Region 1.srn

Region 2 - NOAA California Region 2.srn

Region 3 - NOAA California Region 3.srn

Region 4 - NOAA California Region 4.srn

Region 5 - NOAA California Region 5.srn

Region 6 - NOAA California Region 6.srn

Region 7 - NOAA California Region 7.srn

Region 8 - NOAA California Region 8.srn

Region 9 - NOAA California Region 9.srn

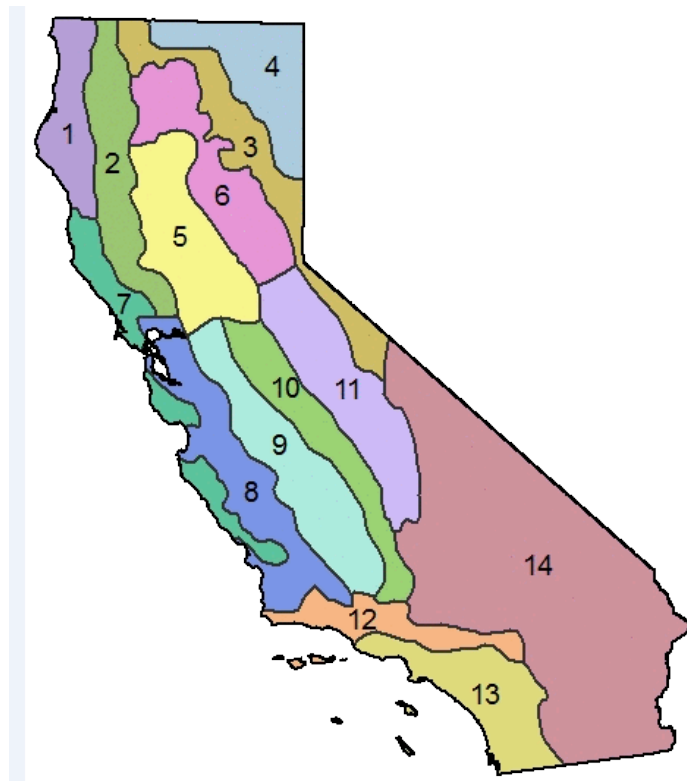
Region 10 - NOAA California Region 10.srn

Region 11 - NOAA California Region 11.srn

Region 12 - NOAA California Region 12.srn

Region 13 - NOAA California Region 13.srn

Region 14 - NOAA California Region 14.srn



Hawaiian Islands Region:

Data Files:

Region - NOAA Hawaiian Islands.srn

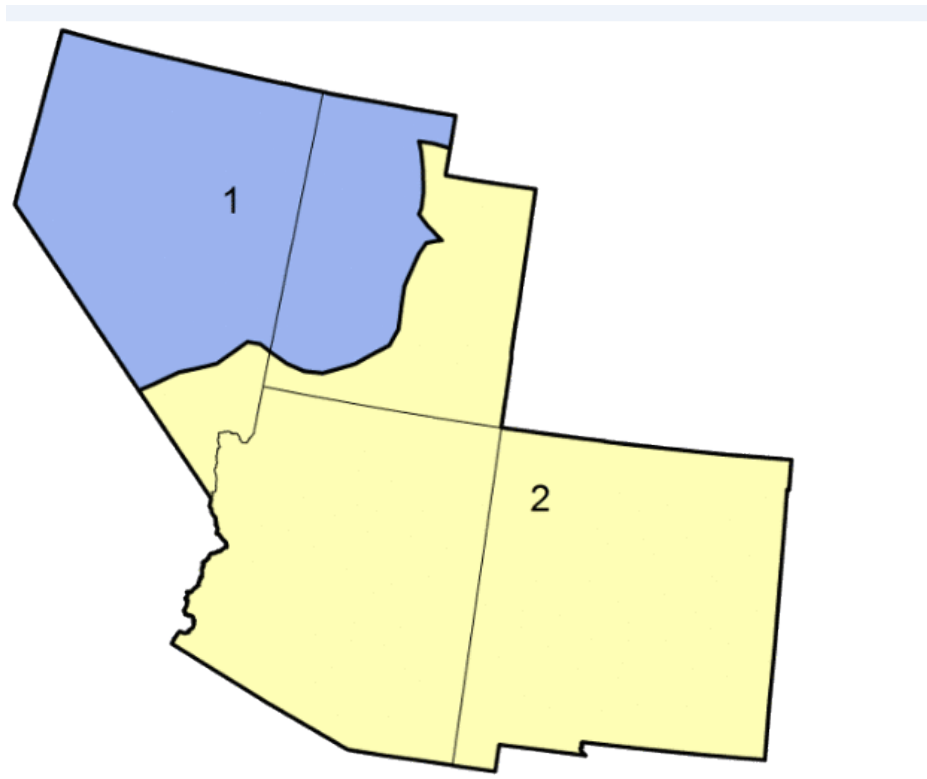


Semi-arid:

Data Files:

Semi-arid SW Convective Area Region: NM, AZ - **NOAA Semi-arid SW Convective Area.srn**

Semi-arid SW General Precipitation Region: UT,NV - **Semi-arid SW General Precipitation.srn**



Midwest:

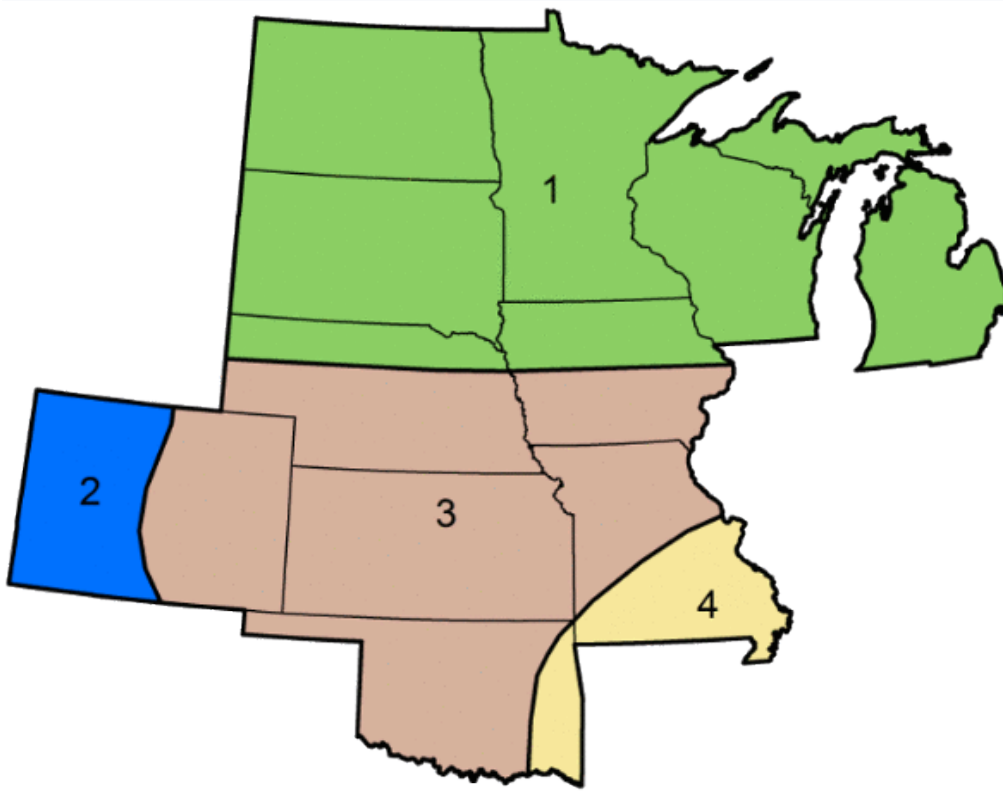
Data Files:

Region 1: SD, ND, MN, WI, MI - **NOAA Midwestern Region 1.srn**

Region 2: Southwest corner of CO - **NOAA Midwestern Region 2.srn**

Region 3: OK, NE, KS, IA, CO - **NOAA Midwestern Region 3.srn**

Region 4: Missouri - **NOAA Midwestern Region 4.srn**

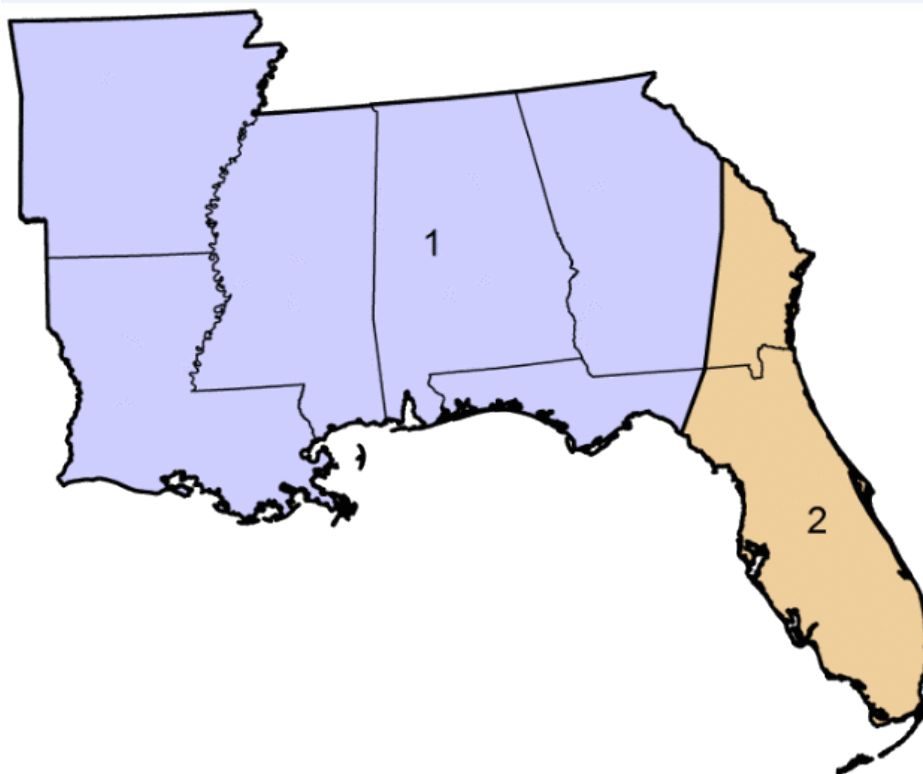


Southeast:

Data Files:

Region 1: AR, LA, MS, AL, GA - NOAA Southeast Region 1.srn

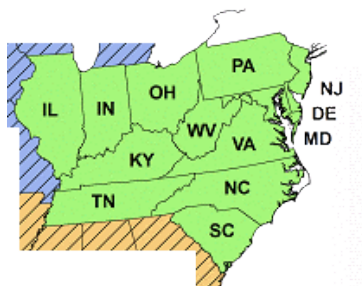
Region 2: FL - NOAA Southeast Region 2.srn



Ohio River Basin Region: SC, NC, TN, KY, WV, IL, IN, OH, VA, MD, Washington D.C., Delaware, NJ, PA

Data Files:

Region - **NOAA Ohio River Basin.srn**

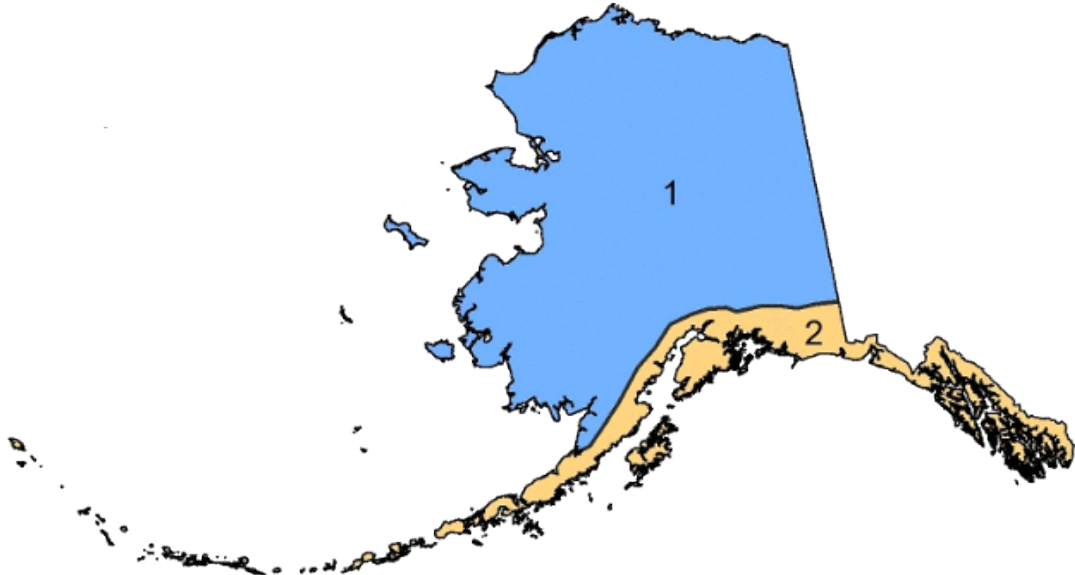


Alaska

Data Files:

Region 1: Upper Part of Alaska - **NOAA Alaska Region 1.srn**

Region 2: Juneau, Aleutian Islands - **NOAA Alaska Region 2.srn**



Puerto Rico and the US Virgin Islands

Data Files:

Region - NOAA Puerto Rico and the U.S. Virgin Islands.srn



Selected Pacific Islands

Data Files:

Region - NOAA Selected Pacific Islands.srn



Prompts

SCS Rainfall Library dialog: Fill in values.

Pulldown Menu Location: Network > Sewer Network Libraries > SCS Rainfall Library

Keyboard Command: scsrainlib

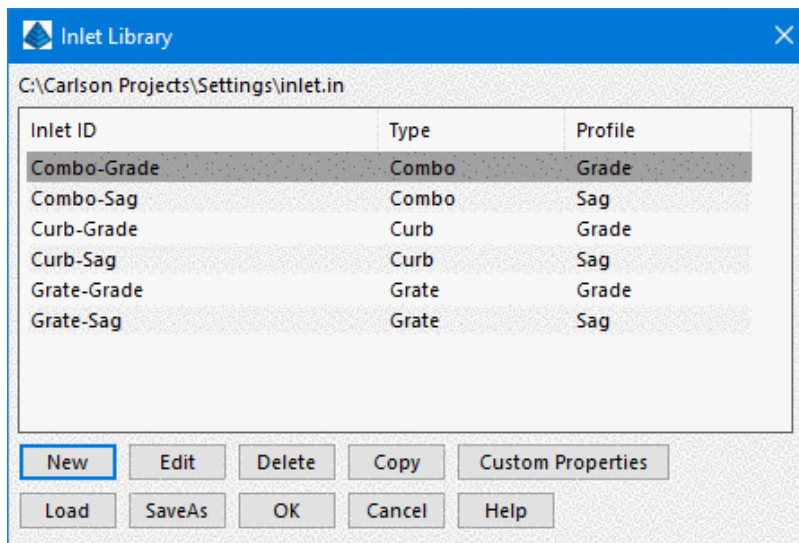
Prerequisite: None

Inlet Library

In storm sewer systems, inlets are the surface components that gather the ground runoff and convey the flow to a subsurface storm sewer network. The inlet capacities should be sufficient to intercept the flows that the sewer system can handle. Inlets can be located "on-grade" or in sag locations. Inlets located "on-grade" may be designed to intercept either all or part of the runoff in the gutter whereas inlets that are installed in sag points should be sized to capture the entire runoff approaching them. The longitudinal and cross-slope of the roadway along with the *Manning's "n"* value of the gutter influence the performance of the inlet. These parameters can be obtained automatically in the watershed modeling of the sewer network design. The details of the inlet design procedures can be found in the HEC-22 Manual.

The Inlet Library command allows you to make, edit and store inlets. The working Inlet library data file is stored (by default) in the `%AppData%\Carlson Software\...\User\Inlet.dta` file and is available across all Carlson Software projects.

The Custom Properties function defines additional fields for each inlet. Values for these fields can be entered into the Inlet Library. Then these fields are available in the Report Sewer Network command.



When you create or edit an inlet, the New/Edit Inlet dialog box displays (shown below).

Inlet ID: Indicate the name of the inlet.

Inlet Type: Indicate the type of inlet:

- Grate
- Curb
- Slotted
- Combo (Grate/Curb)
- Ditch

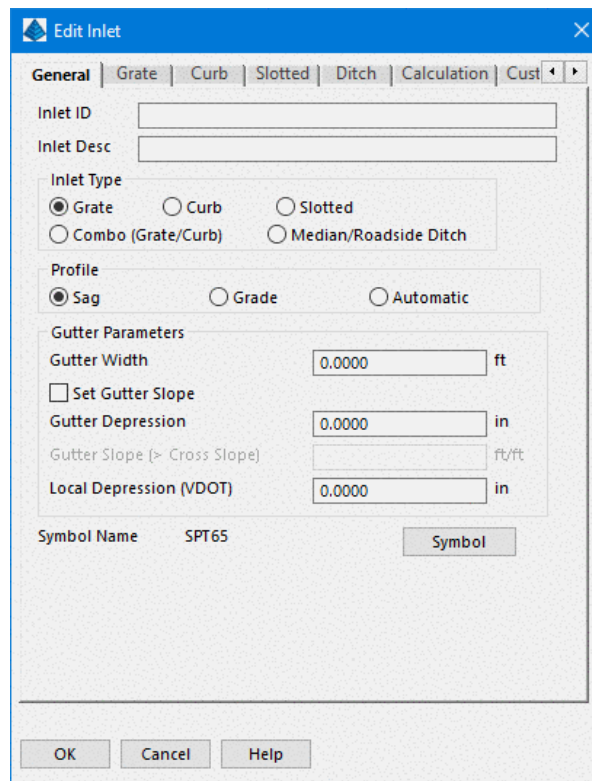
Profile: Indicate whether the inlet is located in a sag portion of a vertical curve or "on grade."

Gutter Parameters: A gutter is a section of pavement adjacent to the roadway, which conveys flow during a storm runoff event. Gutters can have uniform and composite sections. A uniform gutter has the same cross-slope value as the cross-slope of the roadway adjacent to the gutter. A composite gutter section is depressed in relation to the

adjacent pavement cross-slope. Therefore, the Gutter Depression should be entered when using a Composite gutter. The Local Depression is the depression for curb inlets that are off the gutter section.

Design Inlet Length: When enabled, the inlet length is calculated based on Design Constraints of the maximum allowable ponding width (spread) on the roadway and the interception efficiency. When disabled, the spread and efficiency are computed. When the inlet is at a sag location, the interception efficiency is always 100%.

Symbol Name: Click on Symbol button to select a symbol for displaying the inlet in the plan view on the screen.

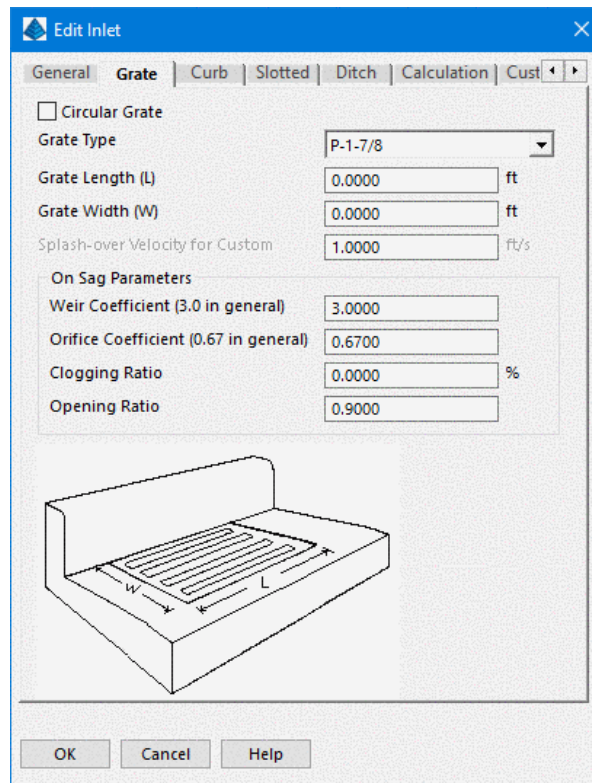


When either the Grate or Combo inlet type is specified, the controls on the Grate Inlet tab are enabled allowing you to create or edit the Grate details. Grate inlets perform well on grades where clogging with debris is not a problem. Their capacity decreases as the pavement longitudinal slope increases. Grate inlets are not generally recommended for use in sag locations because of their tendency to clog. There are seven types of standardized grates:

P-1-7/8 (P-50), P-1-1/8 (P-30), 45d Tilt Bar, Reticuline, P-1-7/8-4 (P-50x100), Curved Vane, 30d Tilt Bar

Please refer to HEC-22 for additional details. You can also define your own type of grate by selecting **Other** in the Grate Type list. If you use a non-standard grate on grade, you must specify a splash-over velocity. Enter the grate dimensions in the Grate Length and Grate Width boxes. When a grate inlet is used, its width can't exceed the Gutter Width.

If the grate inlet is in a sag location, sag-related parameters need to be specified. A grate inlet in a sag operates as a weir at small ponding depths and like an orifice at large depths and these are dependent on the size of the grate. The Clogging Ratio is the percent area of the inlet covered by debris whereas the Opening Ratio is the ratio of the open area to the total area, which can be obtained from HEC-22. You need to specify the Opening Ratio if you use a non-standard grate.

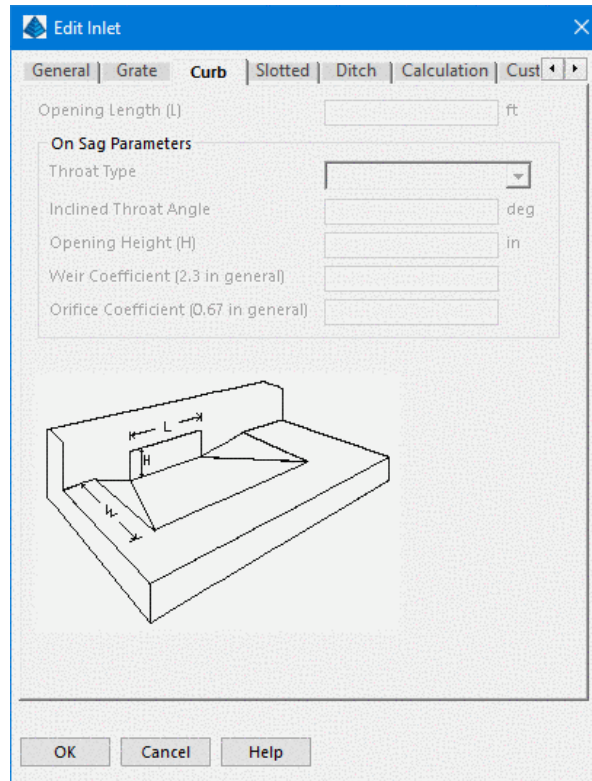


When either the Curb or Combo inlet type is specified, the controls on the Curb Inlet tab are enabled allowing you to create or edit the curb inlet. Curb inlets are less inclined to clog than are grate inlets and have little interference on traffic operation. When placed "on grade," their interception capacity decreases more significantly than that of Grate Inlets as the pavement longitudinal slope increases. As such, Curb Inlets are suitable for use in sag locations or on relatively flat roadways.

When the Curb Inlet is located in a sag, you need to specify the Throat Type:

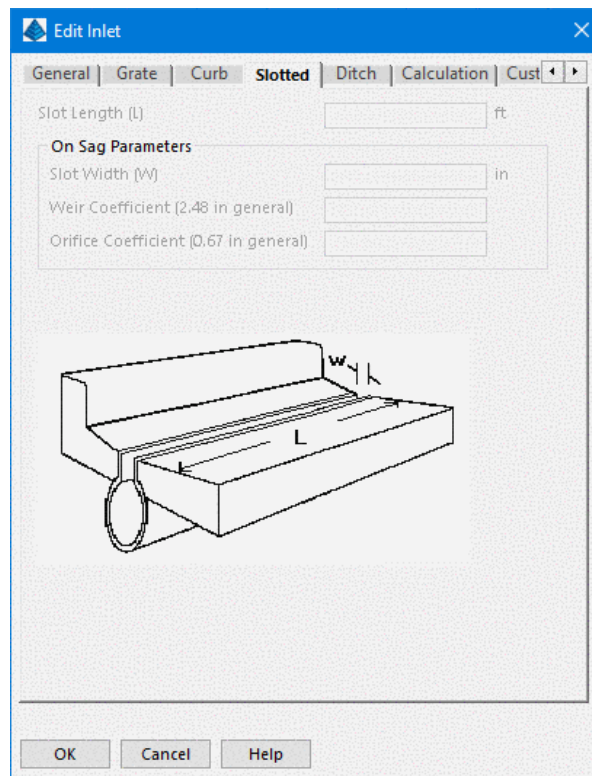
- Horizontal
- Inclined
- Vertical

Please refer to HEC-22 for details on throat types. Additional parameters including the Opening Height, Weir and Orifice Coefficients must also be specified. A curb inlet in a sag behaves as a weir when the depth of water ponding at the curb is less than or equal to the height of the curb opening and like an orifice when ponding depth is greater than 1.4 times the height.



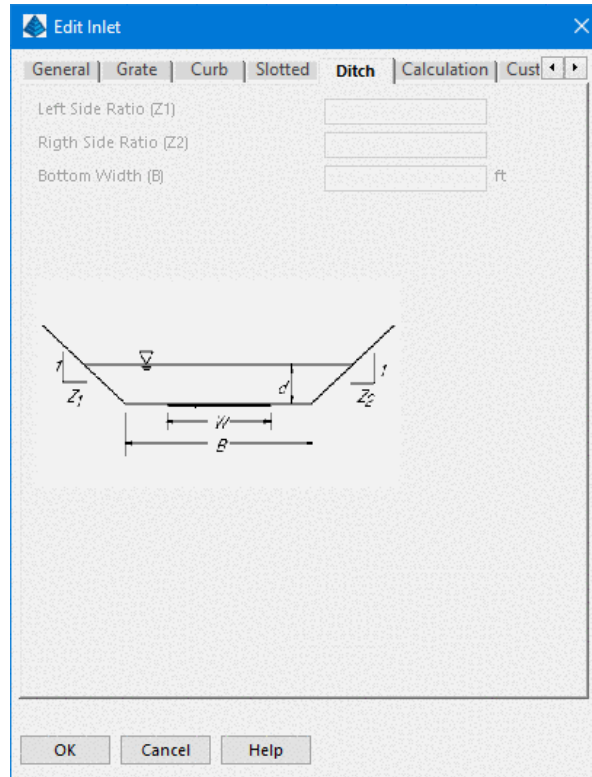
When a Slotted inlet type is specified, the controls on the Slotted Inlet tab are enabled allowing you create or edit a Slotted inlet. Slotted inlets are very effective in intercepting sheet flows due to their long lengths and are suitable to place on roadways. They are very sensitive to clogging and therefore not recommended for use in sags and other locations where debris loadings are considerable. When the slot width is at least 1.75 inches (45 mm) and placed on the pavement along the length of the gutter, the slotted inlet operates similarly to a Curb Inlet.

When installed in a sag, Slotted Inlets perform as weirs to depths up to 0.2 ft (0.06m) and like orifices when the depth is greater than 0.4 ft (0.12m). Refer to HEC-22 for more information.



When the Combo inlet type is specified, the controls on the Grate Inlet tab and Curb Inlet tab are enabled and offer the advantages of both inlet types. The grate is usually the same length as the curb opening and placed alongside it; also called an Equal Length inlet. When the curb opening is longer than the grate, it effectively intercepts trash and debris which may clog the grate and is otherwise known as a Sweeper Inlet.

When a Ditch inlet type is specified, the controls on the Ditch Inlet tab are enabled allowing you create or edit a Ditch inlet. Ditch inlets are typically placed only in sag locations making their capture efficiency 100%. Care should be taken when utilizing Ditch inlets such that the opening does not create a dangerous traffic hazard. In addition to specifying the Bottom Width of the Ditch Inlet, also specify the Left and Right Side ratios as an H:1 ratio.



The Calculation tab allows you to determine how the active inlet will perform against the following parameters:

Flow to Inlet: Specify an assumed amount of flow that will be traveling to the inlet.

Longitudinal Slope: Specify the slope that leads into the inlet.

Pavement Cross Slope: Specify the slope of the pavement that abuts the inlet.

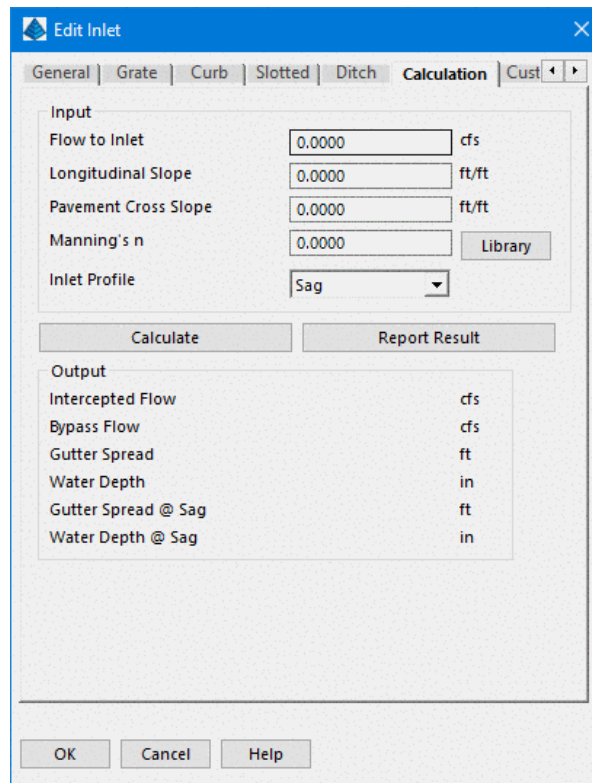
Manning's n: Indicate the *Manning's n* value for the pavement that abuts the inlet or click the **Library** to retrieve the *Manning's n* value from the Pavement Manning's n Library.

Intercepted Flow: The amount of flow that is intercepted by the inlet is reported.

Bypass Flow: The amount of flow (if any) that by-passes the inlet is reported.

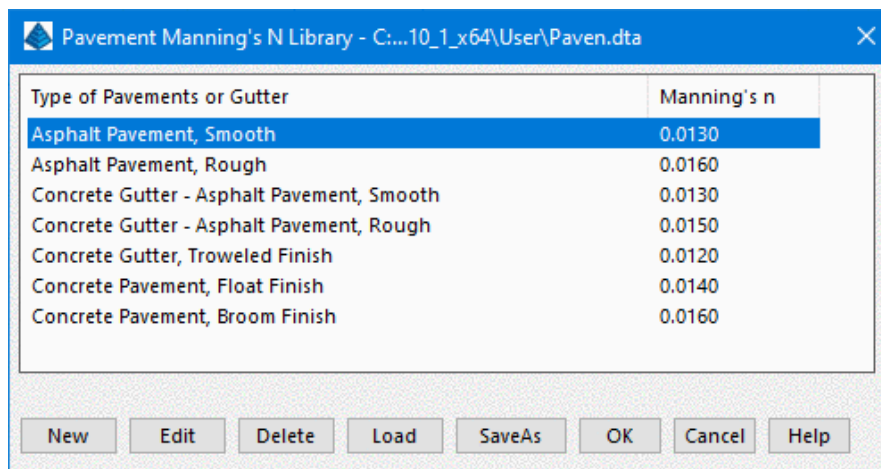
Gutter Spread: The gutter spread (distance from the curb to the outermost edge of the flow of water) is reported.

Water Depth: The water depth in the gutter is reported.



Pavement Manning's n Library

The working Pavement Manning's n library data file is stored (by default) in the `%AppData%\Carlson Software\...\User\Paven.dta` file and is available across all Carlson Software projects. Selecting a desired pavement value and clicking the **OK** button returns the selected *Manning's n* value back to the Calculation tab.

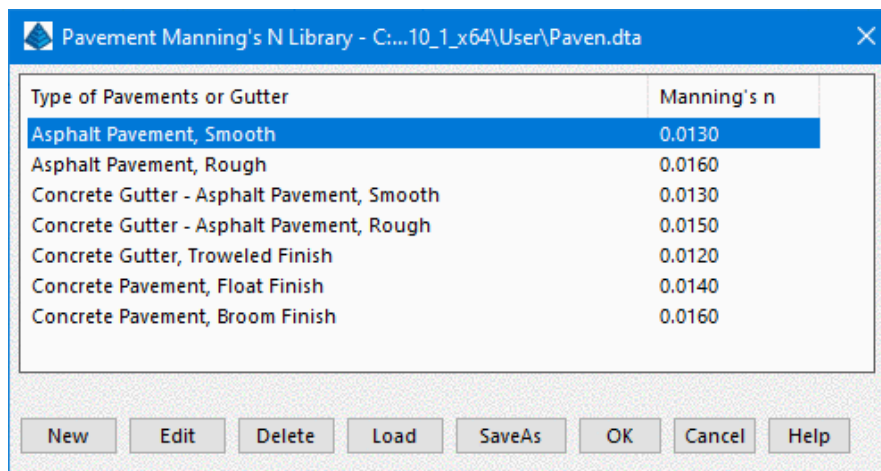


Clicking the **New** or **Edit** button allows a new pavement type and its corresponding *Manning's n* value to be specified.

Pavement Manning's n New Entry

Type of Pavement: Indicate the desired type of pavement.

Manning's n Value: Indicate the *Manning's n* value associated with the pavement type.



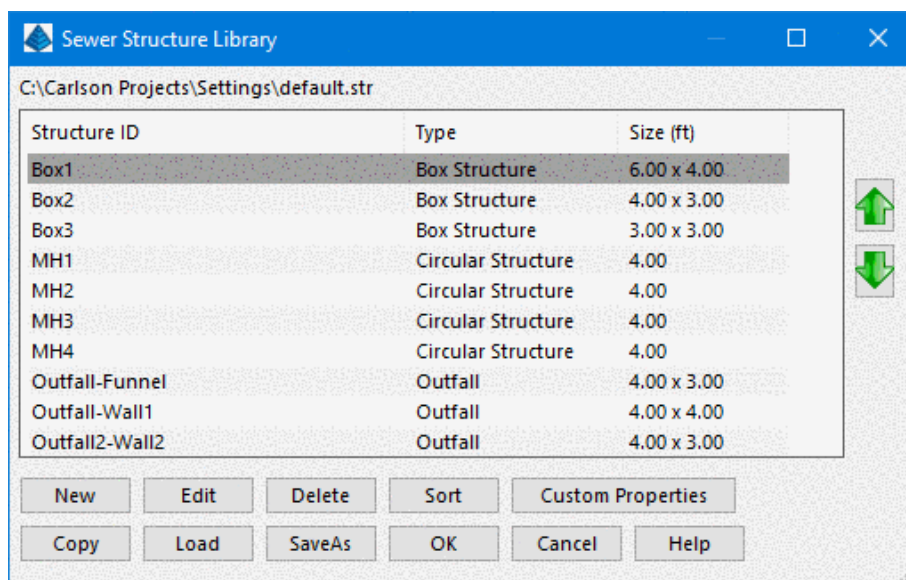
Pulldown Menu Location: Network > Sewer Network Libraries

Keyboard Command: inletlib

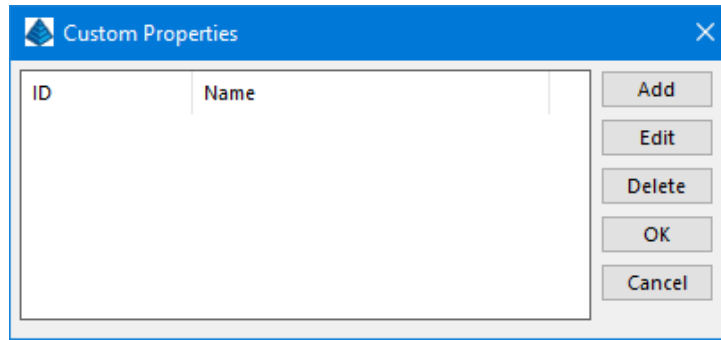
Prerequisite: None

Sewer Structure Library

In a sewer network, there are a few types of structures used at the junctions. The Sewer Structure Library command allows you to create, edit and store sewer structures. The details of the design and construction of sewer structures can be found in the HEC-22 manual.



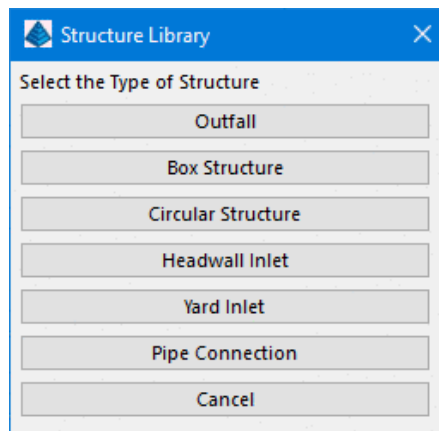
The working structure data file is stored (by default) in the **Carlson Software Install\User** directory and is available across all Carlson Software projects. Existing structures can be edited, sorted or removed from the Structure Library and the Library itself can be saved or re-loaded as desired. The Copy function makes a new structure with the dimensions copied from an existing structure.



The Custom Properties function defines additional fields for each structure. Values for these fields can be entered into the Structure Library. Then these fields are available in the Report Sewer Network command.

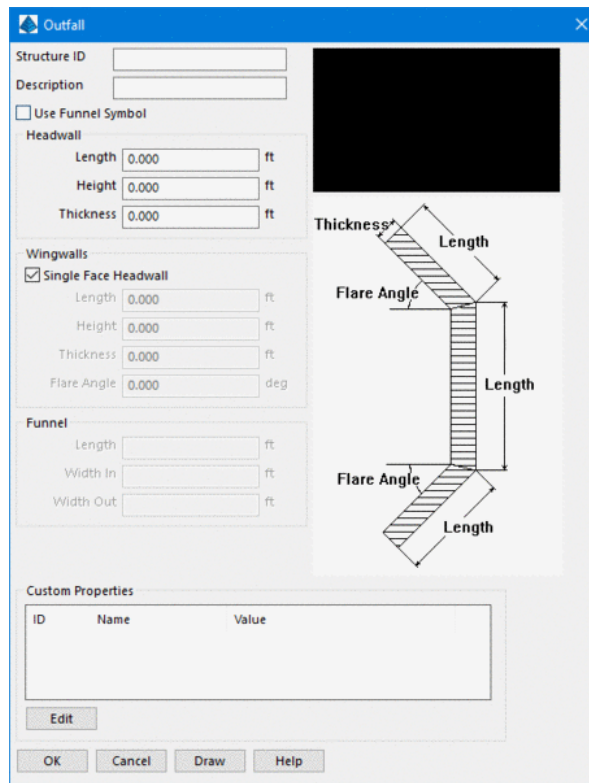
Clicking New gives you the ability to create one of the five types of sewer structures allowed in the Carlson Hydrology module:

- Outfalls are outlet structures at the sewer outlet location. A regular outfall may consist of a headwall and two wingwalls while a funnel outfall consists of a funnel shape structure.
- Box Structures are usually used to support inlet openings and connect them to the underground piping system.
- Circular Structures are manholes that provide access to the sewer network for inspection and maintenance. Manholes are usually installed where pipes change direction (horizontally, vertically or slope), where two or more pipes join or where the pipe sizes change.
- Headwall Inlets are an open-end section similar to a culvert entrance and is used to capture flows at low points.
- Pipe Connections used for lateral connection to main network.

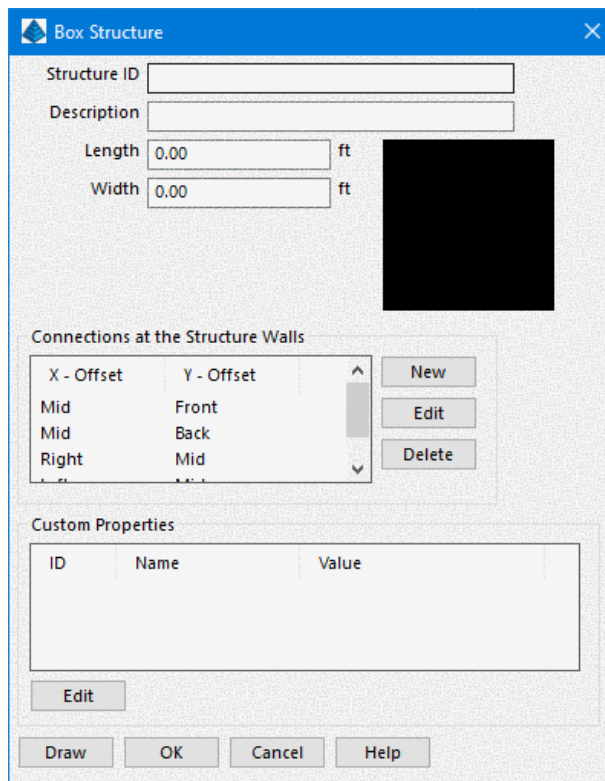


For each structure type, a Draw option allows you to place a "profile aspect" detailed image of the structure (complete with annotation) into your drawing.

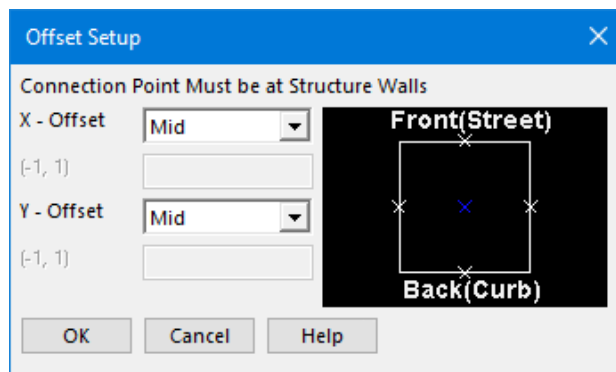
In the Structure ID box, type the structure name. For the regular outfalls, enter the dimensions of the headwall and wingwalls. For the funnel type outfalls, enable the *Use Funnel Symbol* toggle and enter the length and widths of the funnel.



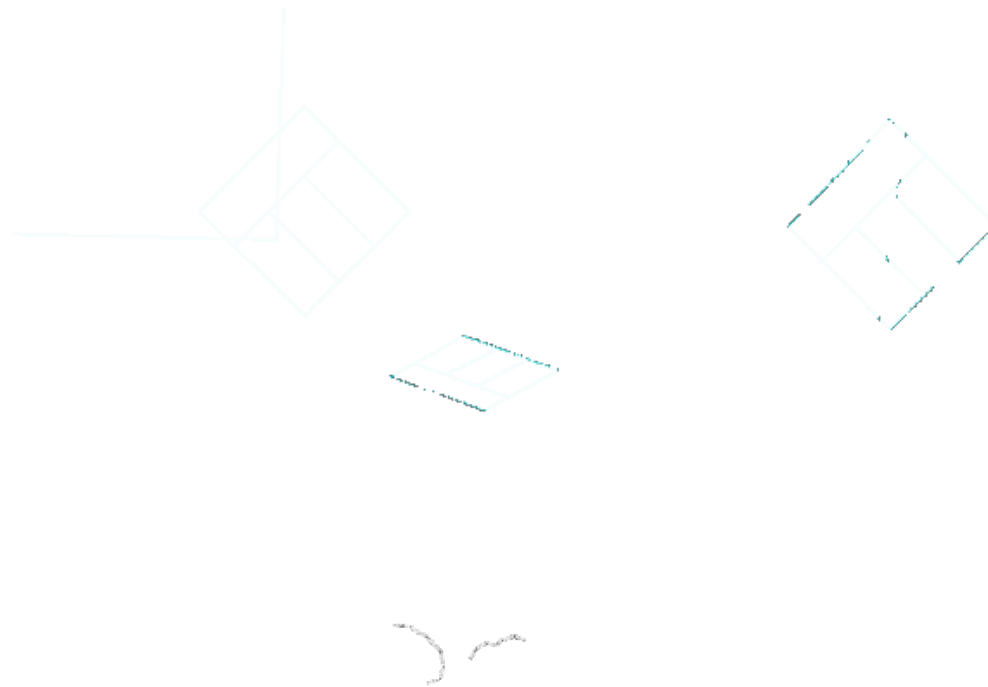
In the Box Structure dialog, type the name of the structure in the Structure ID box, enter the length and width of the structure.



In addition to specifying the physical dimensions of the box structure, you also have the ability to indicate one or more "offset locations" that may better position the box structure around the pipes in situations where the roadway direction is skewed relative to the orientation of the pipes:



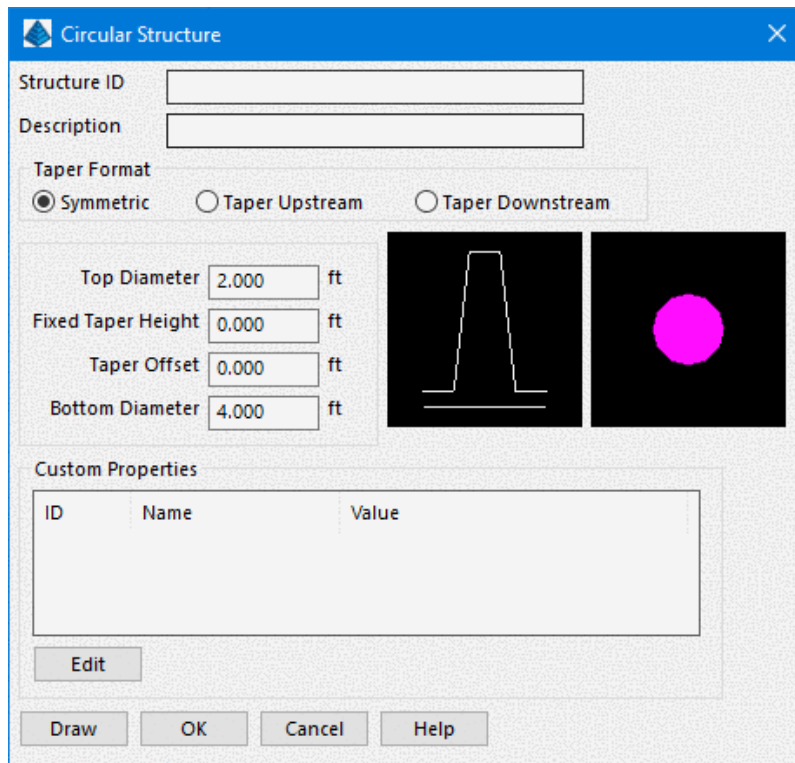
Consider the following example showing a Box Structure with custom offset:



"X" offsets can vary between -1 (Left) through 1 (Right) as a decimal factor relative to the structure length.

"Y" offsets can vary between -1 (Back) through 1 (Front) as a decimal factor relative to the structure width.

In the Structure ID box, type the structure name. Select a Taper Format and enter the Bottom Diameter, Top Diameter, Taper Offset and Taper Height of the structure. A graphic box on the right side of the dialog displays the graphic of the currently defined manhole.



Circular Structure

Structure ID:

Description:

Taper Format

Symmetric Taper Upstream Taper Downstream

Top Diameter: ft

Fixed Taper Height: ft

Taper Offset: ft

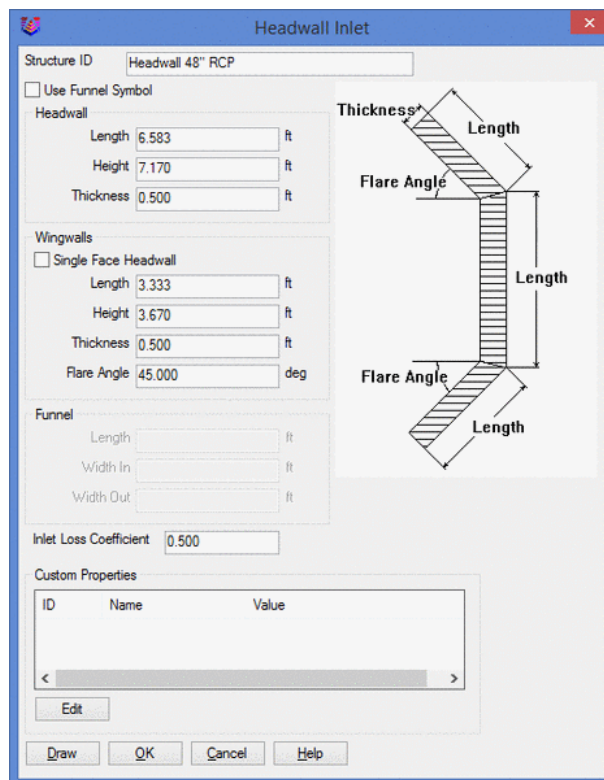
Bottom Diameter: ft

Custom Properties

ID	Name	Value

The headwall inlet is very similar to the outfall structure of headwall type. In the Structure ID box, type the structure name. Enter the dimensions of the headwall and wingwalls. Inlet loss coefficient is used in hydrograph routing calculation.

For the funnel type inlet, enable the *Use Funnel Symbol* toggle and enter the length and widths of the funnel.



Headwall Inlet

Structure ID:

Use Funnel Symbol

Headwall

Length: ft

Height: ft

Thickness: ft

Wingwalls

Single Face Headwall

Length: ft

Height: ft

Thickness: ft

Flare Angle: deg

Funnel

Length: ft

Width In: ft

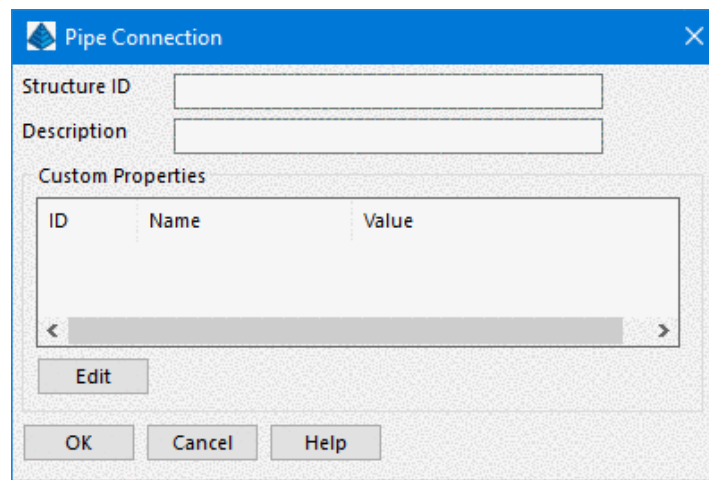
Width Out: ft

Inlet Loss Coefficient:

Custom Properties

ID	Name	Value

Type the Structure ID and add Custom Properties if required.



Pulldown Menu Location: Network > Sewer Network Libraries > Sewer Structure Library

Keyboard Command: strlib

Prerequisite: None

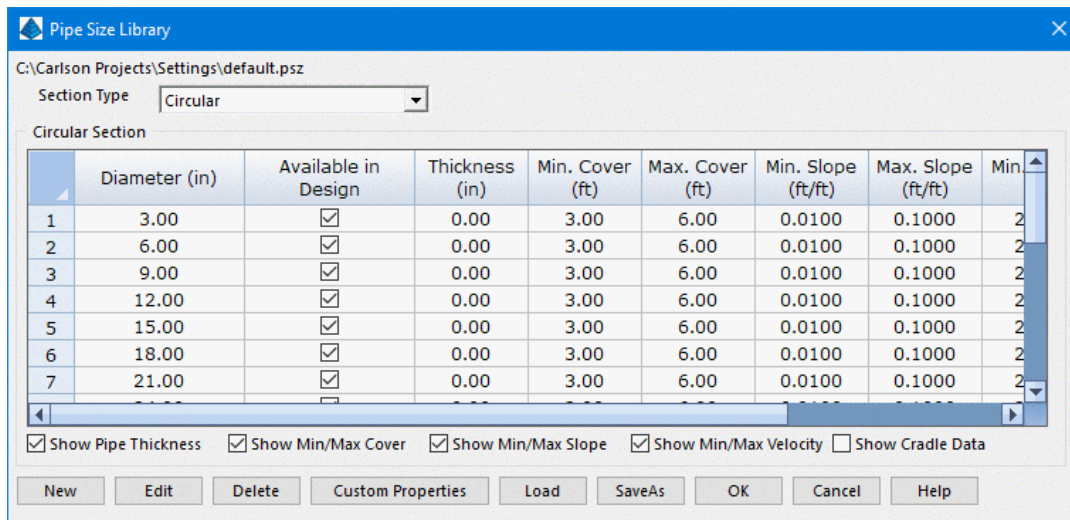
Pipe Size Library

The Pipe Size Library command allows you to store the dimensions of the widely used pipes. There are four pipe sections: box, circular, horizontal ellipse and vertical ellipse. The working structure data file is stored (by default) in the `%AppData%\Carlson Software\...\User\PipeSize.dta` file and is available across all Carlson Software projects.

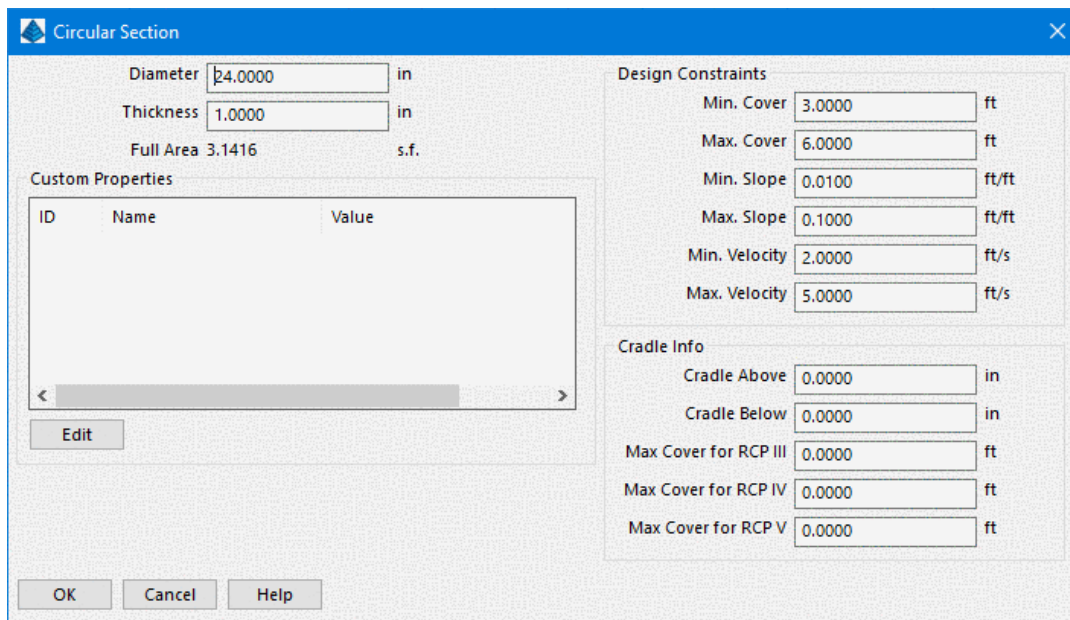
From the Network > Sewer Network Libraries menu in the Hydrology Module, select Pipe Size Library to open the library dialog. The Section Type list contains four section types. You can select one pipe shape to display all the pipe sizes of that shape in the spreadsheet. In the right column of the spreadsheet, the Available in Design check boxes are listed next to the pipe sizes indicating that whether the pipe sizes are available in the pipe size design or not. You may choose to display the Min/Max Cover, Min/Max Slope, Min/Max Velocity and Cradle Data on the spreadsheet. These parameters are used in sewer network design and analysis.

The Custom Properties function defines additional fields for each pipe. Values for these fields can be entered into the Pipe Size Library. Then these fields are available in the Report Sewer Network command.

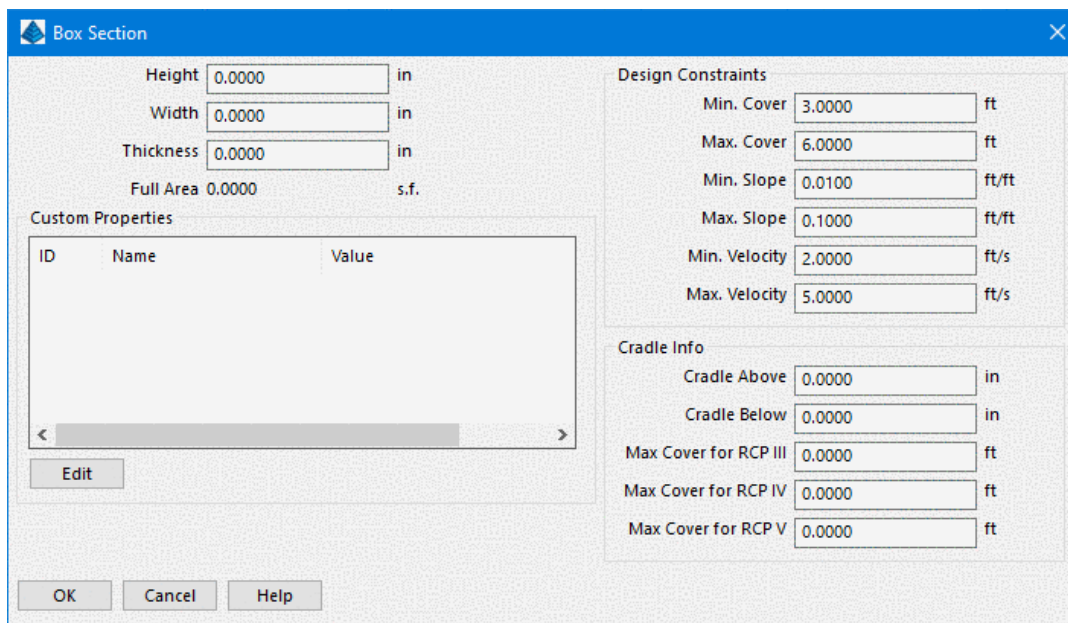
New button adds a new pipe size, Edit button allows you to modify an existing pipe size, and Delete button removes the highlighted pipe size from the library. Load and SaveAs buttons allow you to load and save the pipe size data.



In the Circular Section dialog, type the value in the Diameter box, and then the full cross-section area is calculated and displayed. You may fill in the Design Constraints and Cradle Info for sewer network analysis and sewer profiles. Click on OK button to commit the pipe size entry.



In the Box Section dialog, type the values in the Height and Width boxes, and then the full cross-section area is calculated and displayed. You may fill in the Design Constraints and Cradle Info for sewer network analysis and sewer profiles. Click on OK button to commit the pipe size entry.



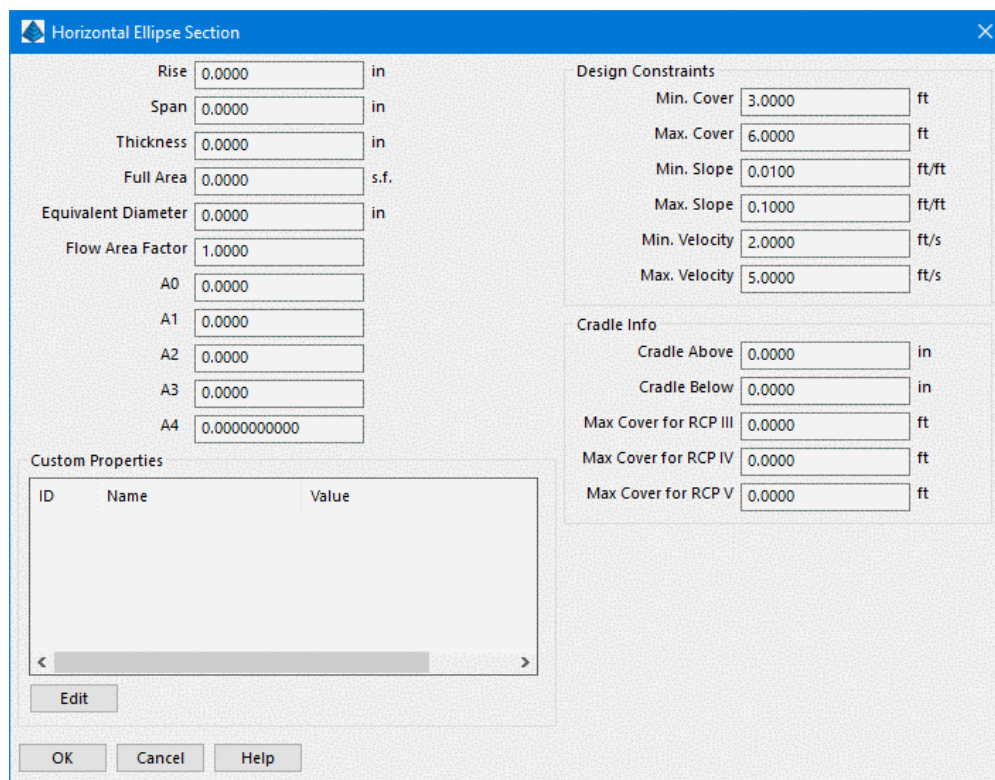
In the Horizontal and Vertical Ellipse Section dialog, type the values in the Rise, Span and Full Area boxes. The Equivalent Diameter is the diameter of a circular section that is equivalent to the ellipse. The Flow Area Factor is the ratio of the calculated full area to the specified full area. A0, A1, A2, A3 and A4 are the five coefficients to the 4th order polynomial equation, which describes the relationship between the wetted perimeter (in) of the ellipse and the depth/rise ratio, as below.

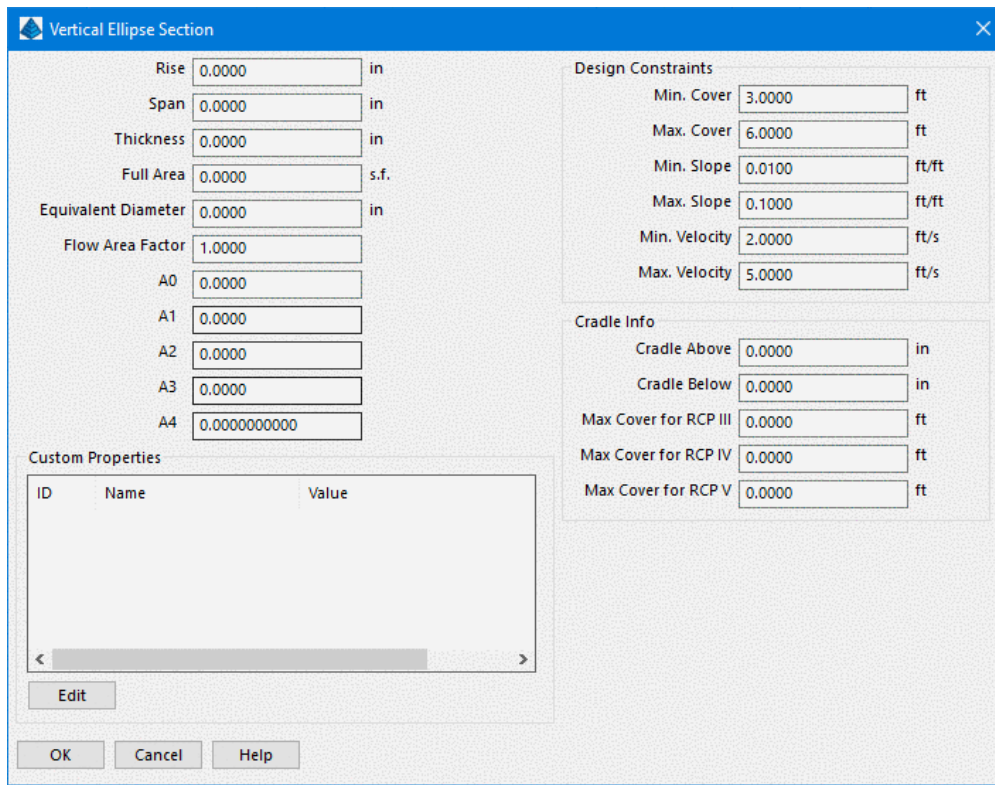
$$p = A_0 + A_1 (d/r) + A_2 (d/r)^2 + A_3 (d/r)^3 + A_4 (d/r)^4, \text{ where:}$$

p = wetted perimeter (in or mm)

d = water depth in the pipe (ft or m)

r = pipe rise (ft or m)





Default Libraries

There are several default libraries for North Carolina under Carlson Projects\Settings\NorthCarolina. Use the Load function to select a library.

Pulldown Menu Location: Network > Sewer Network Libraries > Pipe Size Library

Keyboard Command: pszlib

Prerequisite: None

Curve Number Library

The Curve Number Library command allows you to store commonly used drainage area cover descriptions, soil types and their curve numbers for SCS method. The library file is in the ... \USER folder and is available for all projects in sewer network design.

From the Network > Sewer Network Libraries menu in the Hydrology Module, select Curve Number Library to open the library dialog. The Cover Description list displays all the stored drainage area cover types, the A, B, C and D lists display the curve numbers for soil type A, B, C and D. New button creates a new curve number entry, Edit button allows you to modify the highlighted entry, and Delete button removes the highlighted entry from the library. Load and SaveAs buttons allow you to load and save the library data.

Cover Description	A	B	C	D
Open Space-Poor (<50% grass)	68	79	86	89
Open Space-Fair (50% to 75% grass)	49	69	79	84
Open Space-Good (>75% grass)	36	61	74	80
Paved Lots Roofs & Driveways	98	98	98	98
Paved Streets w/Curbs & Sewers	98	98	98	98
Paved Streets w/ Open Ditches	83	89	92	93
Gravel Streets (including ROW)	76	85	89	91
Dirt Streets (including ROW)	72	82	87	89
Commercial/Business Districts	89	92	94	95
Industrial Urban Districts	81	88	91	93
Residential 1/8 acre or less	77	85	90	92
Residential 1/4 acre	61	75	83	87
Residential 1/3 acre	57	72	81	86
Residential 1/2 acre	54	70	80	85
Residential 1 acre	51	68	79	84
Residential 2 acres	46	65	77	82
Newly Graded Areas	77	86	91	94
Fallow - Bare Soil	77	86	91	94
Fallow - CR (Poor)	76	85	90	93

Drainage Runoff Library

Drainage Runoff New Entry

Prompts

Curve Number Library dialog: Fill in values.

Pulldown Menu Location: Network > Sewer Network Libraries > Curve Number Library

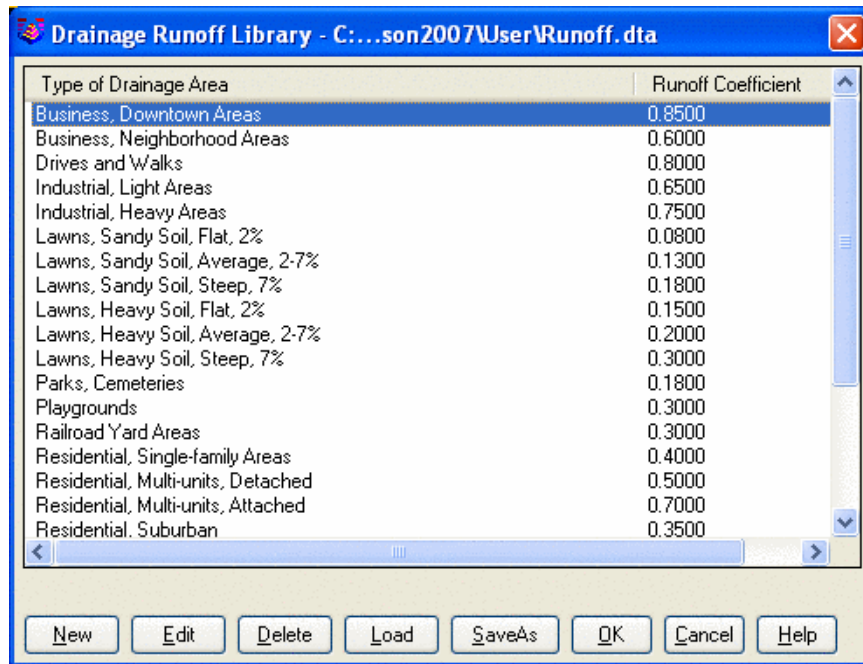
Keyboard Command: cnlib

Prerequisite: None

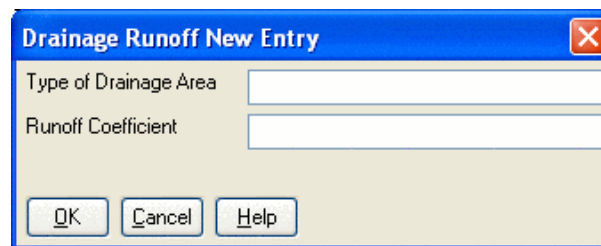
Drainage Runoff Library

The Drainage Runoff Library command allows you to store commonly used drainage area types and their runoff coefficients. The library file is in the ... \USER folder and is available for all projects in sewer network design.

From the Network > Sewer Network Libraries menu in the Hydrology Module, select Drainage Runoff Library to open the library dialog. The Type of Drainage Area list displays all the stored drainage area types and the Runoff Coefficient list displays the corresponding runoff coefficients. New button creates a new drainage runoff entry, Edit button allows you to modify the highlighted entry, and Delete button removes the highlighted entry from the library. Load and SaveAs buttons allow you to load and save the library data.



Drainage Runoff Library



Drainage Runoff New Entry

Default Libraries

There are several default libraries for North Carolina under Carlson Projects\Settings\NorthCarolina. Use the Load function to select a library

Prompts

Drainage Runoff Library dialog: Fill in values.

Pulldown Menu Location: Network > Sewer Network Libraries > Drainage Runoff Library

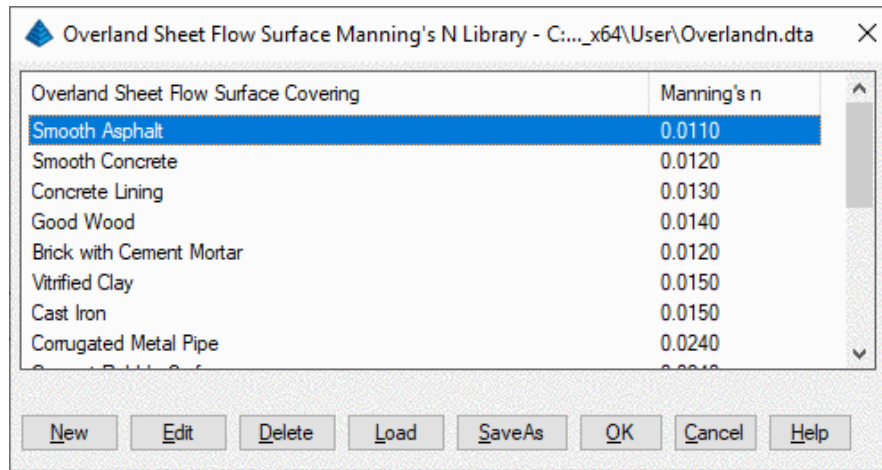
Keyboard Command: runofflib

Prerequisite: None

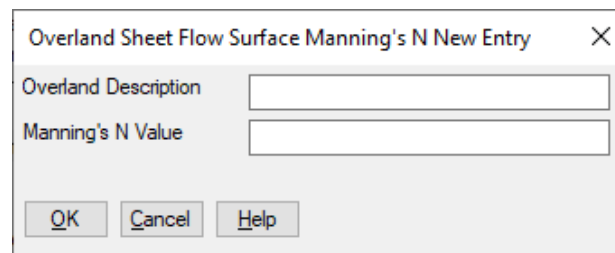
Overland Sheet Flow Surface Mannings N Library

This library has a table of overland flow manning's values that are used in TR-55 sheet flow Time of concentration calculation. The library file is in the ... \USER folder and is available for all projects.

From the Network > Sewer Network Libraries menu in the Hydrology Module, select Overland Sheet Flow Surface Manning's N Library to open the library dialog. It shows the list of surface covers and the corresponding Manning's N values. New button creates a new manning's n entry, Edit button allows you to modify the highlighted entry, and Delete button removes the highlighted entry from the library. Load and SaveAs buttons allow you to load and save the library data.



Overland Sheet Flow Surface Manning's N Library



Manning's N New Entry

Prompts

Overland Sheet Flow Surface Manning's N Library dialog: Fill in values.

Pulldown Menu Location: Network > Sewer Network Libraries > Overland Sheet Flow Surface Manning's N Library

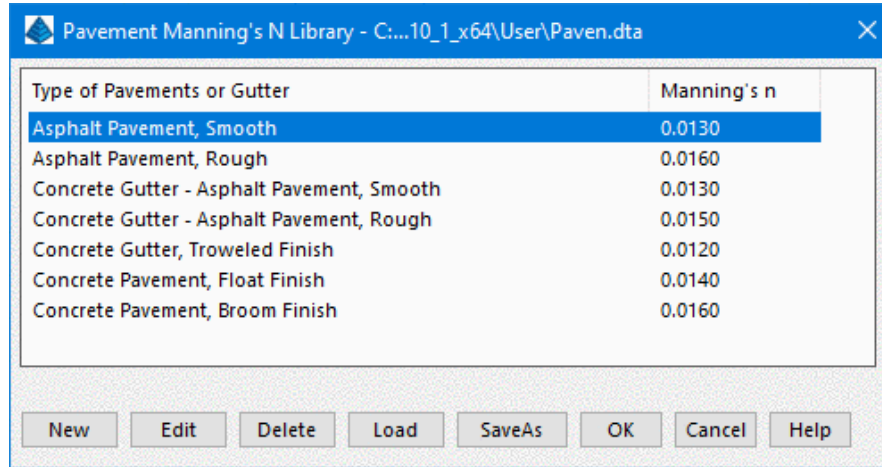
Keyboard Command: overlandnlib

Prerequisite: None

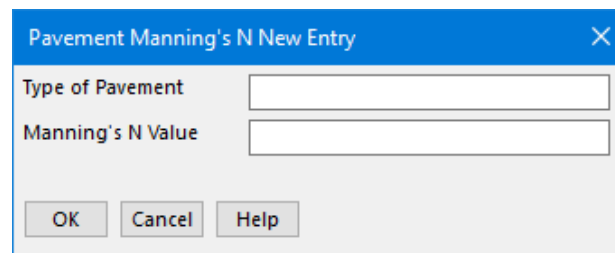
Pavement Mannings N Library

The Pavement Manning's N Library command allows you to store commonly used pavement types and their manning's n values. The library file is in the ... \USER folder and is available for all projects.

From the Network > Sewer Network Libraries menu in the Hydrology Module, select Pavement Manning's N Library to open the library dialog. The Type of Pavements of Gutter list displays all the stored pavement types and the Manning's N list displays the corresponding manning's n values. New button creates a new pavement manning's n entry, Edit button allows you to modify the highlighted entry, and Delete button removes the highlighted entry from the library. Load and SaveAs buttons allow you to load and save the library data.



Pavement Manning's N Library



Pavement Manning's N New Entry

Prompts

Pavement Manning's N Library dialog: Fill in values.

Pulldown Menu Location: Network > Sewer Network Libraries > Pavement Manning's N Library

Keyboard Command: pavenlib

Prerequisite: None

Pipe Manning's n Library

The Pipe *Manning's n* Library command allows you to store commonly used pipe types and their Manning's n values. The working pipe Manning's n data file is stored (by default) in the **%AppData%\Carlson Software\...\User\Pipen.dta** file and is available across all Carlson Software projects for both culvert design and sewer network design.

The Custom Properties function defines additional fields for each pipe material. Values for these fields can be entered into the Pipe Material Library. Then these fields are available in the Report Sewer Network command.

The New button creates a new pipe Manning's n entry while the Edit button allows you to modify the highlighted entry while the Delete button removes the highlighted pipe entry from the library. The Load and SaveAs buttons allow you to load and save the library data.

Pipe Material	Manning's n	Material Type
Cast Iron Pipe	0.0150	Steel
Clay	0.0150	Concrete
Concrete	0.0130	Concrete
Corrugated Metal	0.0250	Steel
Ductile Iron	0.0150	Steel
HDPE	0.0120	Plastic
PVC	0.0200	Plastic
RCP III	0.0130	Concrete
RCP IV	0.0130	Concrete
RCP V	0.0130	Concrete

Pipe Material Name: Indicate the desired pipe material name.

Material Type: Select the closest available pipe material.

Manning's n Value: Indicate the appropriate Manning's n value for the pipe.

Pipe Material New Entry	
Pipe Material Name	Cast Iron Pipe
Material Type	Steel
Manning's N Value	0.015
<input type="button" value="OK"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/>	

Pulldown Menu Location: Network > Sewer Network Libraries

Keyboard Command: pipenlib

Prerequisite: None

Input Sewer Line

This command creates a sewer network by placing structures on polyline vertices. Before running this command, draw a polyline connecting where you want to create structures. An advantage of this command over Create Sewer Structure is that the polyline defines the connections between the structures. Also before running this routine, go to Sewer Network Settings to assign the Network File, Surface File and settings.

In the options dialog, set the default values for the structures and pipes. See the Create Sewer Structure topic for information on these settings.

After the dialog, the program prompts to select the polyline. Then temporary arrows are drawn to show the flow direction that the program will use. There is a prompt to reverse this direction if needed. Then the program creates structures at each vertice in the polyline. The design settings such as Min Cover and Min Slope from Sewer Network Settings are used to set the pipe slopes and invert elevations. After creating the structures, the Edit Sewer Structure dialog shows the new structures. You can review and edit the structure properties.

Prompts

Input Sewer Line dialog

Select Network Polyline: *pick a polyline*

Switch flow direction [Yes/<No>]?: *press Enter*

Edit Sewer Structure dialog

Pulldown Menu Location: Network

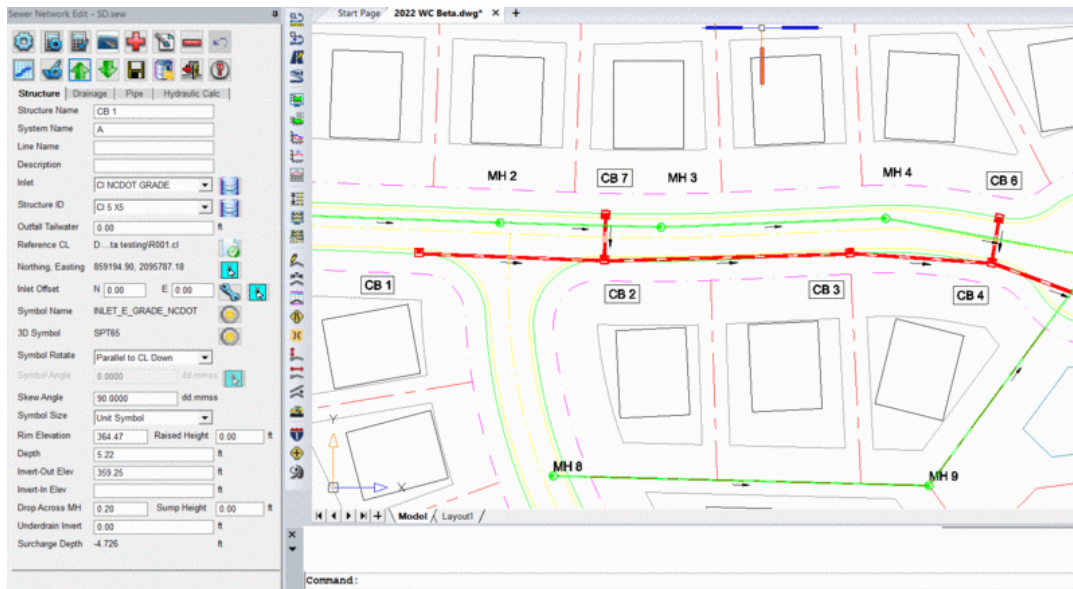
Keyboard Command: plswr

Prerequisite: polyline and Sewer Network Settings

Edit Sewer Structure

The Create Sewer Structure or Edit Sewer Structure command is a very powerful program for the design and analysis of networks. A storm sewer network is generally made up of pipes, structures and inlets. A sanitary/utility network, unlike a storm sewer, doesn't have inlets. There may be more than one pipe entering a structure, but only one can exit. The network type is specified via the Sewer Network Settings command. Depending on the type of sewer network being created or edited, you'll either have the full-featured editing control (described below) for Storm Sewer networks or a more stream-lined Create/Edit Sanitary/Utility Structure command.

This command allows you to construct a graphical representation of a pipe network in the active drawing with associated data including (but not limited to) pipe, structure, inlet, watershed and rainfall details. The Edit/Create Sewer Structure command takes the form of a "docked dialog box" as illustrated below. This flexible interface allows direct access to the *Command*, graphic window, pull-down menus and toolbar... all while the docked dialog box is open.



The storm sewer network is solved using the standard step gradually varied flow methods. This is an iterative procedure used to determine the energy and hydraulic terms at the end of each pipe. The direction of computation is from the most downstream pipe of the network to the most upstream pipe. The following steady state energy equation is used between the upstream and downstream ends of every pipe. Please refer to HEC-22 manual for details. You can design the sewer system with one rainfall return period, and analyze it with another return period.

$$Z_u + V_u^2 / 2g = Z_d + V_d^2 / 2g + H_f$$

where: Z_u = upstream water surface elevation

$V_u^2 / 2g$ = upstream velocity head

Z_d = downstream water surface elevation

$V_d^2 / 2g$ = downstream velocity head

H_f = friction loss

The Manning's equation is applied to determine the friction slope.

$$Q = (M/n) A R^{2/3} S_f^{1/2}$$

where: Q = discharge

M = 1.49 for English unit, 1.0 for Metric units

n = Manning's roughness coefficient

A = cross-sectional area

R = hydraulic radius

S_f = friction slope

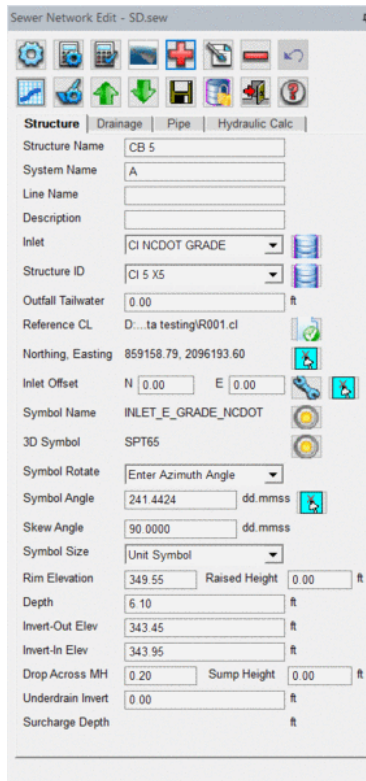
Then the friction loss along the pipe is computed by the following equation:

$$H_f = S_f L$$

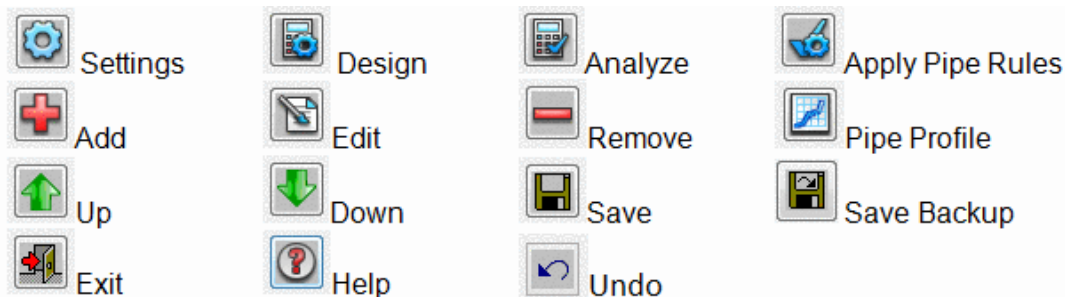
where: L = pipe length

With the friction loss calculated, the elevation of the upstream water surface can be determined.

First set up the working environment by running command Sewer Network Settings under Network > Sewer Network Setup menu. Then select Edit/Create Sewer Structure. If you are creating a sewer structure, pick a location in the plan view where you want to locate the structure, otherwise click on an existing structure symbol. After a structure has been located, the dock dialog displays, and the current structure symbol is highlighted in the drawing. Following is an example of the dock dialog.



The top of the docked dialog box has thirteen buttons, the buttons are as followed. Hover mouse over each icon to see its name.



Common icon controls include the following:

Settings: This function performs as same as the Sewer Network Settings command except for the settings found on the General tab.

Design: When you select any Automation settings in the Sewer Network Settings, i.e. Auto Set All Sewer Pipe Sizes, Auto Set All the Invert Elevation of the Sewer Network and etc., this function would design the profile of sewer lines, such as pipe inverts and sizes, depending on the design settings, and then performs the hydraulic calculation. If you don't choose any Automation settings, this button would work just as the **Analyze** button.

When designing pipe sizes, the program first estimates the design flow for each pipe in the system and makes an initial selection of the size, from the pipe size library, required for each pipe.

Typically, pipe slope is set to the actual invert slope. If the pipe invert elevations are to be designed, pipe slope is assumed as the same as the normal slope. The Manning's equation is then used to solve the required pipe size given the pipe Manning's n coefficient, design discharge and slope. The calculated size is then rounded up to an available

size in the pipe size library. When designing pipe invert elevations, the criterion of minimizing ground cover at all locations along pipe lines is used.

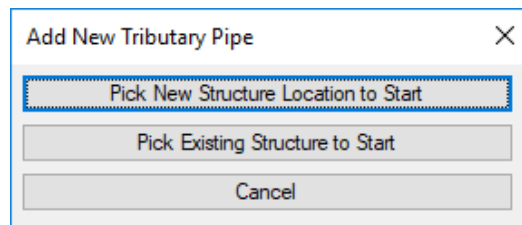
After initial design, the program analyzes gradually varied flow with the standard step method for a few iterations. It uses the actual velocity from the previous calculation to determine the total time of concentration, and therefore the actual flow and hydraulic grade line, modifies the pipe sizes and invert elevations based on the design constraints, and then performs next iteration of computation, until the result is stable and meets the design constraints. Any violations of the design settings will be shown in a warning message dialog window.

Analyze: This function conducts the hydraulic calculation on the existing sewer network.

The program analyzes gradually varied flow with the standard step method and reports the results such as hydraulic grade line, energy grade line, flow velocity, drainage flow rate and inlet interception, *etc.* Any violations of the design settings will be shown in a warning message dialog window.

Add Tributary Pipe Line: This function adds a tributary pipe connecting to an existing structure of the main network.

When adding a tributary pipe line from upstream to downstream, you would choose **Pick New Structure Location to Start**, and then keep picking to create new structures, and in the end pick an existing structure to connect to the main network and finish the tributary line. When choose **Pick Existing Structure to Start**, you would start the tributary pipe line from downstream to upstream, and in the end you hit **Enter** to finish the tributary line.



When the tributary pipe line is from upstream to downstream and the downstream invert of the last pipe is lower than the manhole base elevation, there are five actions defined in the Sewer Network Settings to adjust the network pipe inverts.

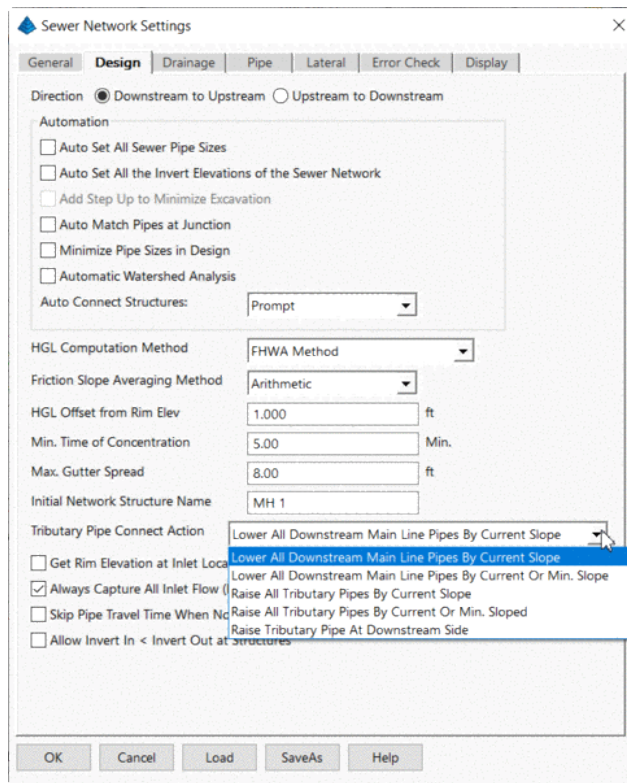
Lower All Downstream Main Line Pipes By Current Slope: This will lower all downstream mainline pipes by the current slope to match the tributary pipe elevation, and create a step up for upstream mainline pipe.

Lower All Downstream Main Line Pipes By Current/Minimum Slope: This will lower the downstream mainline pipes by the current/minimum slopes match the tributary pipe elevation, and create a step up for the upstream mainline pipe. Use a slope between the current and minimum slope to minimize the change of the manhole inverts.

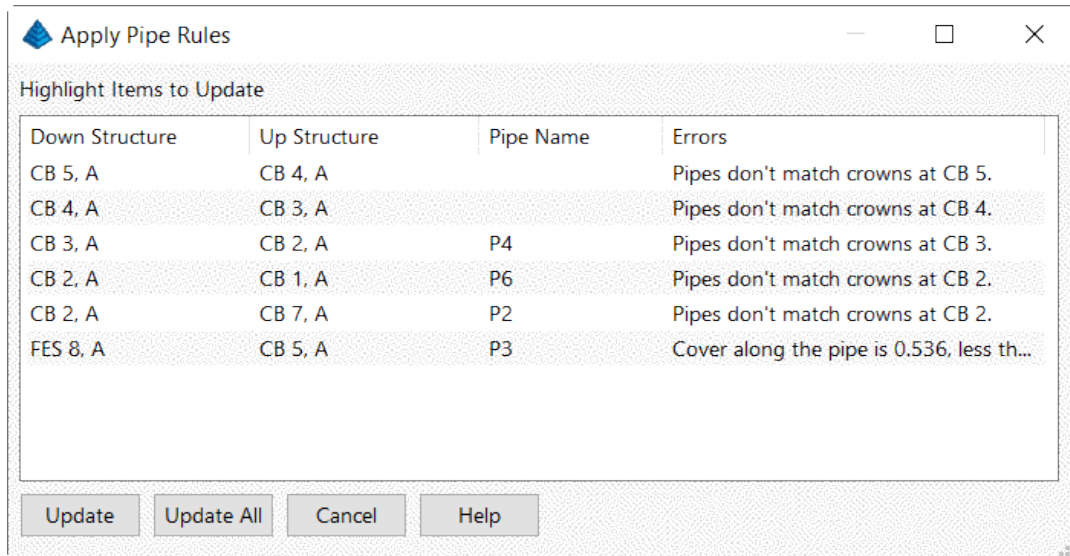
Raise All Tributary Pipes By Current Slope: This option will raise the tributary pipes by the current slope to match the connection invert/crown.

Raise All Tributary Pipes By Current/Min Slope: This option will raise tributary pipes by the current/minimum slope to match the connection invert/crown. Use a slope between the current and minimum slope to minimize the change of the manhole inverts.

Raise Tributary Pipe At Downstream Side: This option will only raise the downstream invert of the last tributary pipe.

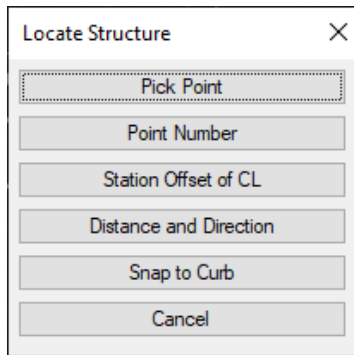


Apply Pipe Rules: Click this button to correct detected pipe problems against those specified in the Pipe Size Library. Use standard Windows **click**, **shift+click** and/or **ctrl+click** functionality to select multiple pipes at the same time.



Pipe Profile: This function will open the Spreadsheet Sewer Editor and allow edits to be made while reviewing multiple pipe runs.

Add: This function allows you to create a new structure. The new structure will connect to the nearest structure found in the network if you select the **Auto Connect Structures** option. There are five ways to pick a location: **Pick Point** in the drawing, enter **Point Number**, enter **Station and Offset** of the centerline, enter **Distance and Direction** from current structure, and **Snap to Curb** and then pick a curb polyline to place the new structure right at the curb line.



Edit: This function allows you to pick an existing structure symbol in the plan view to make the structure active for editing.

Remove: This function removes the structure that you pick and also removes the corresponding pipes and then reconstructs the network.

Up: Moves to the upstream structure and makes it active.

Down: Moves to the downstream structure and makes it active.

Save: Prompts to save any pending network changes.

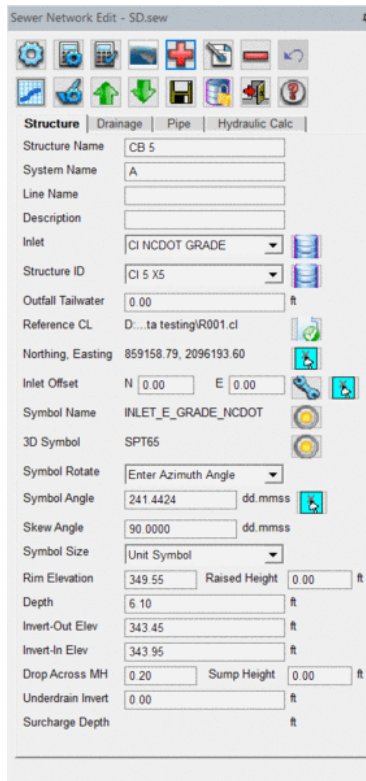
Save Backup: Prompts to save the network data in a backup file.

Undo: Can undo the changes made to the sewer network. One click reverts one change.

Exit: Dismisses the docked dialog box.

Just below the icons are four tabs. The tabs and each function are described below tabs:

1. Structure - Controls information related to sewer structures (manholes, inlets, rim elevation, *etc*).
2. Drainage - Controls drainage information including watershed information.
3. Pipe - Controls information related to sewer pipes (connection points, size, material, *etc*).
4. Hydraulic Calc - Controls and reports information related to junction losses and the hydraulic calculations.



Structure Name: Provide a unique value to identify the structure in the network.

System Name: A name for current network. All the structures within a sewer line must have the same System Name (there can be more than one sewer line in a sewer (.SEW) file).

Line Name: Provide a unique value to identify the lines names in the network.

Description: Provide an optional note that further describes the structure.

Inlet: Select a pre-defined inlet to be associated with the structure or click the *Library* button to select an Inlet or make changes to the Inlet Library. In addition to associating an Inlet with the Structure, the *Library* button also populates the **Symbol Name** control with the default value stored in the Inlet Library. For a box/circular structure without an inlet, the structure is not a catch basin, but a junction box.

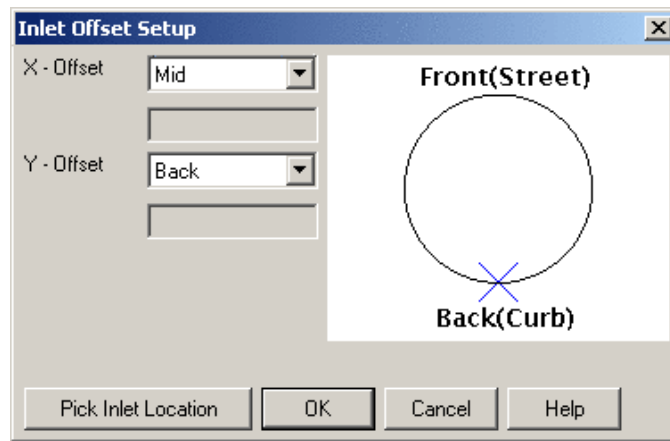
Structure ID: Select a pre-defined structure or click the *Library* button to select a Structure or make changes to the Structure Library. For a box/circular structure without an inlet, the structure is not a catch basin, but a junction box.

Outfall Tailwater: Assign a tailwater elevation for the current outfall. There can be multiple outfalls in a sewer file and each outlet has a tailwater value. This feature becomes active if current structure is an outfall.

Reference CL: Use the *Select* button to determine the station and offset location of the structure relative to a Centerline File or polyline in the drawing.

Northing, Easting: Use the *Location* button to identify the coordinate location where the Structure should be located.

Inlet Offset: Specify the placement of the inlet relative to that of the structure or click the *Setup* button for common offset configurations:



X-Offset: Specify the Easting/Departure of the inlet relative to that of the structure.

Y-Offset: Specify the Northing/Latitude of the inlet relative to that of the structure.

Pick Inlet Location: Graphically identify the placement of the inlet relative to that of the structure.

Symbol Name: This is the plan view symbol for inlets. Use the *Symbol* button to select a symbol for the Structure or to over-ride the value associated with the current **Inlet**.

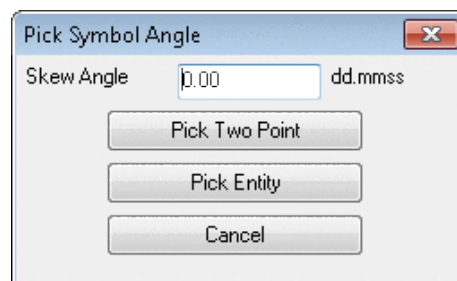
3D Symbol: This is the 3D symbol for inlets when drawing sewer network in 3D. Use the *Symbol* button to select a symbol for the Structure or to over-ride the value associated with the current **Inlet**.

Symbol Rotate: Indicate the method to orient the **Symbol Name** in the drawing. If a **Reference CL** has been specified and depending on how the symbol was originally drawn, consider using one of the *Parallel to CL...* or *Perp to CL...* options to orient the symbol relative to the direction of the centerline.

Symbol Angle: When the **Symbol Rotate** value is set to *Enter Azimuth Angle*, use this field to specify the rotation angle for the symbol, or pick using the screen pick icon



If selecting the icon for symbol rotation the follow options are available for use.



Symbol Size: Indicate how the size of the symbol should be determined:

- *Inlet Library Width* - The symbol is sized according to the *Gutter Width* specified in the Inlet Library.
- *Drawing Scaler* - The symbol block is scaled based on the **Drawing Scaler** derived in Drawing Setup.
- *Unit Symbol* - The symbol is sized based on the size of the symbol block itself.

Rim Elevation: Indicate the rim elevation for the structure this can user input or based off a surface model. Use raised height option to add additional height to a rim elevation.

Depth: Indicate the distance between the Rim Elevation and the base (bottom) elevation of the structure.

Invert-Out Elev: Indicate the invert elevation of the pipe that exits the structure.

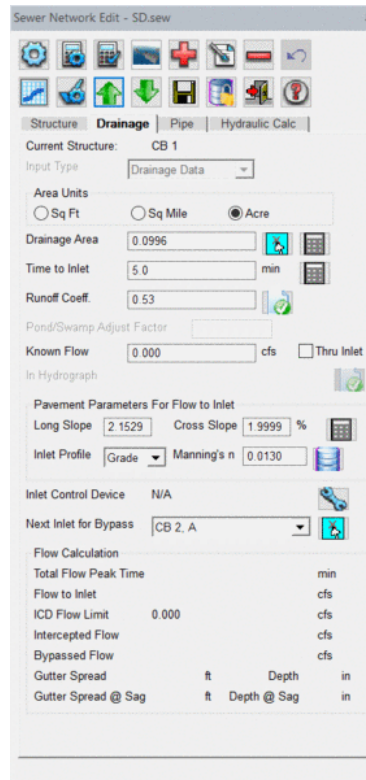
Invert-In Elev: Indicate the invert elevation of the pipe that enters the structure.

Drop Across MH: Indicate the amount of vertical drop across the manhole/structure.

Underdrain Invert: Indicate the invert elevation of an Underdrain Invert system.

Sump Height: Indicate the sump distance below the manhole bottom. Use Sump Height to add additional depth for a structure

Surcharge Depth: The distance between the hydraulic grade line and the Rim Elevation. If the value is negative, the hydraulic grade line is below the rim elevation; otherwise, the water has blown out the structure.



Current Structure: The **Structure Name** from the Structure tab that is current. Use the *Up*, *Down* or *Edit* buttons to navigate through the network to set an alternate structure current.

Input Type: If the **Computation Method** option located in the Drainage tab of the **Sewer Network Settings** is set to *Peak Discharge*, then this option is disabled. If the **Computation Method** is set to *Hydrograph*, two input types are available:

- *Drainage Data* - will let the program generate runoff hydrograph, or,
- *Hydrograph* - specify a known runoff hydrograph directly.

Area Units: Choose a unit to display the area values.

Draw: When this button is clicked, a watershed analysis is performed on the surface model and the drainage area that contributes to this inlet is hatched in the drawing.

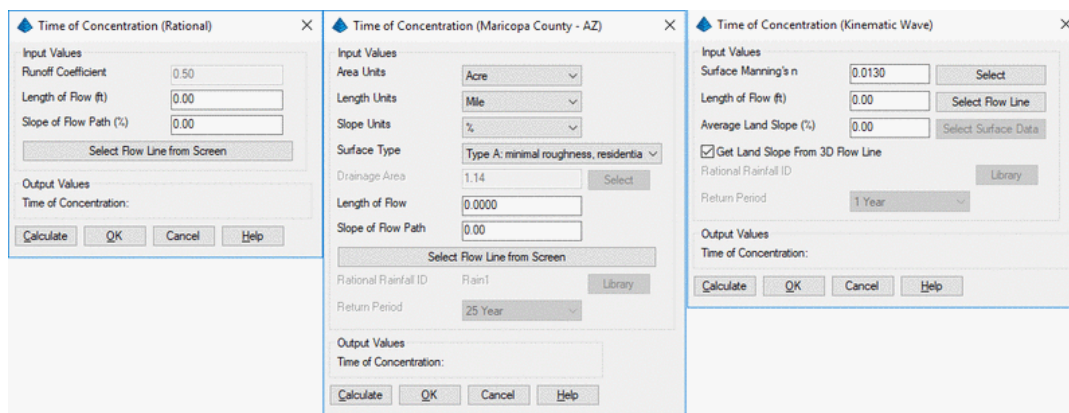
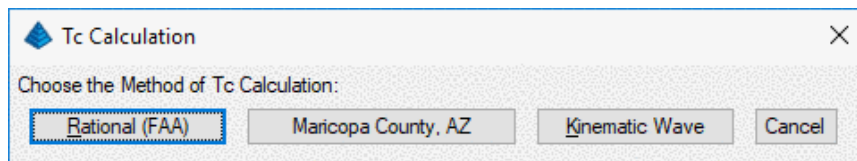
Drainage Area: The drainage area that contributes to this inlet only.

Pick: Use this button to select a closed polyline that represents the boundary of the drainage area. The area will be calculated and displayed.

Calc: This function triggers the watershed analysis routine to calculate the drainage area and weighted runoff coefficient (curve number for SCS method) and displays the values.

Time to Inlet: Enter the time to inlet value or click on the **Set** button to calculate the Time of Concentration. There are three methods to calculate Time to Inlet. Choose one of the method and open its dialog to enter the flow path

data to calculate Time of Concentration.



Runoff Coeff: Indicate the runoff coefficient or click the **Select** button to derive a weighted runoff coefficient from land uses and their respective areas. Please refer to the documentation on Define Runoff Layers for details.

Pond/Swamp Adjust: When the **Hydro Methods** option located in the Drainage tab of the **Sewer Network Settings** is set to *SCS*, indicate the Pond and swamp adjustment factor used in *SCS* peak flow calculations.

Known Flow: You may elect to let any known flow enter into the downstream pipe directly or go through the inlet by enabling the **Thru Inlet** toggle. This provides flexibility to input the calculated drainage runoff directly to the program and to also enter known sanitary flow, infiltration/exfiltration flow, *etc.*, into the sewer network.

Thru Inlet: When enabled, the known flow will enter the sewer network through the inlet rather than simply being considered system flow, and inlet calculation is conducted. Depending on the inlet capacity, some known flow may be bypassed and flow to the downstream inlet. Otherwise, all the known flow is added to the network flow directly.

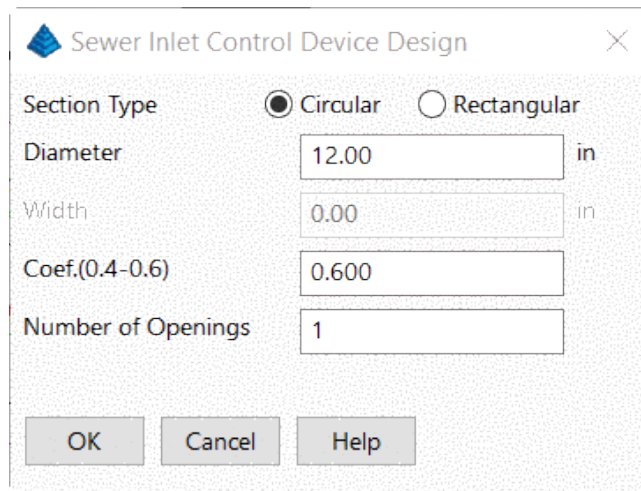
In Hydrograph: When the **Input Type** is set to *Hydrograph*, use the **Select** button to specify the Runoff Hydrograph file name (.HYD) that enters into the structure.

Long Slope: Enter the longitudinal slope of the pavement. This edit field is only available if the inlet is "on-grade."

Cross Slope: Enter the pavement cross slope.

Inlet Profile: Choose between **Grade** or **Sag** type inlet.

Inlet Control Devices: Add inlet control devices, options include **Circular** and **Rectangular**.

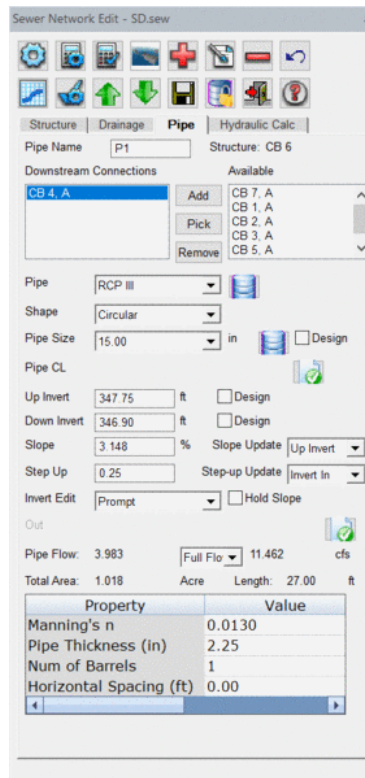


Calc: This function triggers the watershed analysis routine to analyze the surface model and extract the longitudinal slope and cross slope.

Manning's n: Indicate the *Manning's n-value* for the pavement or click on the **Library** button to retrieve this value from the Pavement Manning's n Library.

Next Inlet for Bypass: Indicate the inlet that shall receive any by-pass flow from current catch basin.

Flow Calculation: After you issue either the **Design** or the **Analyze** command on the network, the inlet calculation results are displayed. The inlet results help you to determine if the inlet is sufficient for conveying the ground flow into the network.



Pipe Name: Provide a unique value to identify the pipe in the network.

Downstream (Upstream) Connections: A list showing the connection that exits the structure or the connection(s) that enters the structure, depending on the *Direction* setting found on the Design tab of the **Sewer Network Settings**.

Available: A list that contains all of the structures that are not connected to the current structure, *i.e.* the potential structures that can be connected to the current structure.

Add: Select an available structure to connect to the current structure and click on the *Add* button to form the connection.

Pick: Select a structure symbol in the plan view graphic to form a connection between it and the current structure. If the connection cannot be completed, a warning message appears.

Remove: Highlight a connection in the *Downstream (Upstream) Connections* list and click on the *Remove* button to remove the connecting pipe between the structures.

Pipe Material: Select the pipe material or click on the **Library** button to select a pipe and its corresponding *Manning's n* value from the Pipe Manning's n Library.

Pipe Shape: The pipes can have four different cross-sectional shapes:

- Circular
- Box
- Horizontal Ellipse
- Vertical Ellipse

Refer to the Pipe Size Library for additional details.

Pipe Size: If the *Auto Set All Sewer Pipe Sizes* option found on the Design tab of the **Sewer Network Settings** dialog box is disabled, select the desired pipe size from the list of available pipe sizes or click on the **Library** button to access the Pipe Size Library.

Design: When enabled, the program calculates an appropriate pipe size based on the flow and design settings and selects the closest available pipe size from the *Pipe Size Library*.

Pipe CL: This option allows you to design a curvilinear (non-straight) pipe. The pipe centerline should start from one structure and end at the other structure exactly. Use the **Select** button to identify a Centerline File or polyline in the drawing that represents the pipe path. When one of the structures is relocated, the pipe centerline association will be removed. The pipe bending loss is computed for pipe friction loss when the pipe is curvilinear.

Down Invert/Up Invert: If the *Auto Set All the Invert Elevations of the Sewer Network* option found on the Design tab of the **Sewer Network Settings** dialog box is disabled, indicate the upstream and/or downstream invert elevation of the current pipe or enable either **Design** toggle to have the program calculate the design invert elevation(s).

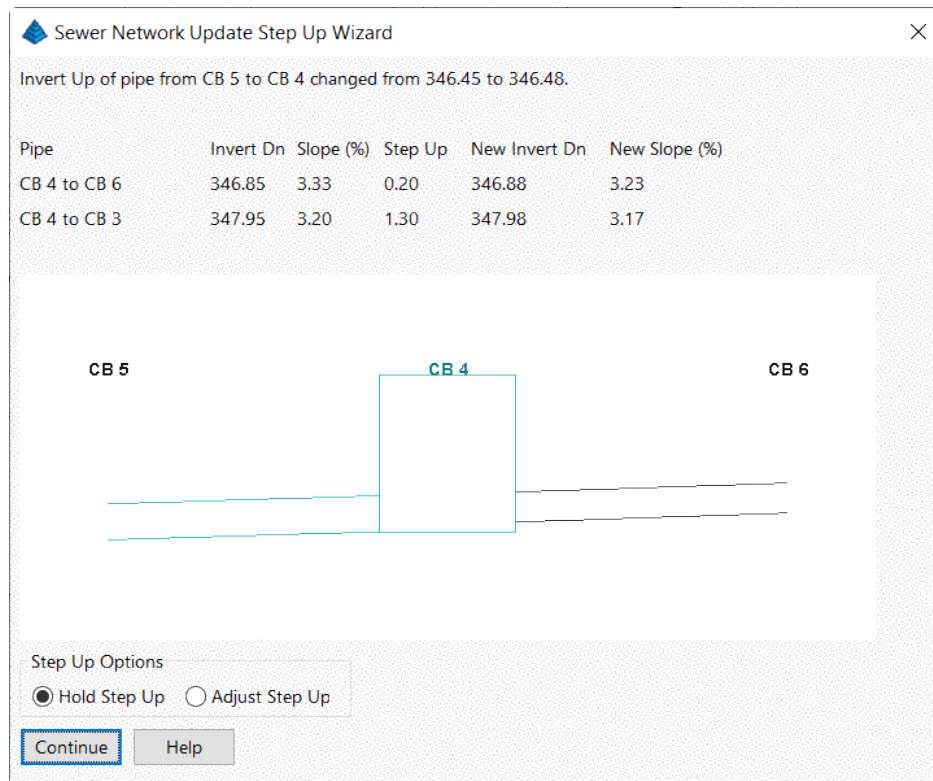
Slope: If neither the *Up Invert Design* toggle nor the *Down Invert Design* toggle is enabled, indicate the pipe slope. Changing the Slope value will alter the *Up Invert* value unless the **Hold Upstream** toggle is enabled.

Slope Update: Allows the user to specify which invert will be adjusted if user changes slope manually.

Step Up: Indicate the "step up" amount of the *Downstream Invert* elevation with respect to the "invert out" elevation of the structure to which it connects. In general the invert out elevation is lower than the invert in elevation of the structure as water flows from upstream to downstream. In Carlson 2018, we introduced **negative step up** in network design, when the invert out is lower than the invert in at the junction.

Slope Update: Allows the user to specify which invert will be adjusted if user changes step-up.

Invert Edit: When a structure has multiple invert in's the step-up wizard will show. Step-up Options are to Hold the original Step-Up or to Adjust Step-Up to a new value.



Hold Slope: This option will hold the pipe slope when editing inverts and step up values..

Out: When the computation method is hydrograph, you are able to generate the outflow hydrograph at the outlet of each pipe. Use the **Select** button to specify the file name.

After you issue either the **Design** or the **Analyze** command on the network, the pipe calculation results are shown.

Pipe Flow: Flow that is being carried by the pipe:

- **Full Flow:** Flow that is being carried by the pipe when flowing full.
- **Max Flow:** The maximum flow that can be carried by the pipe.

Total Area: Total of all the drainage areas that contribute to the flow inside the pipe.

Length: The length of the pipe based on the *Pipe Length* setting found on the Display tab of the **Sewer Network Settings**.

Properties and values may also be entered in the following

Manning's n: Indicate the Manning's n coefficient is used to calculate the friction loss of the pipe.

of Barrels: Indicate the number of pipe barrels that can be found between the *Upstream* and *Downstream* structures.

Hori. Spacing: Indicate the amount of horizontal spacing between each of the *Barrels*.

The energy losses through a pipe junction are specified in the Hydraulic Calc tab. There are six methods to calculate the junction losses: Approximate Method, Dynamic Method, Fixed Head Loss and Energy-Loss Method.

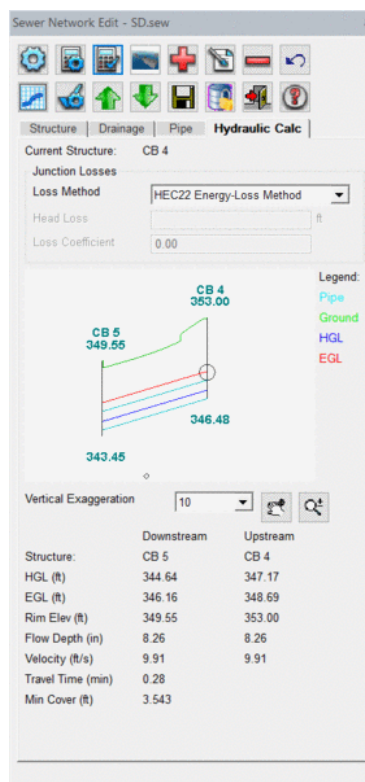
Junction Losses: Indicate one of the following methods for junction loss consideration:

- *Approximate Method* - uses the difference between the downstream velocity head and the upstream velocity head multiplied by the junction **Loss Coefficient**.
- *Dynamic Method* - uses the downstream velocity head multiplied by the junction **Loss Coefficient**.
- *Fixed Head Loss* - uses the actual **Head Loss** you specify.
- *HEC22 Energy-Loss Method* - similar to the Dynamic Method, uses the downstream velocity head multiplied by the adjusted junction loss coefficient. The adjusted junction loss coefficient is defined as the initial head loss coefficient based on relative size of structure multiplied by the correction factors for pipe diameter, flow depth, plunging flow and benching. Calculations use the HEC-22 2009 3rd edition method. Please refer to the HEC-22 manual for details.
- *Total Loss Method* - uses uniform pipe velocities to compute the contraction loss, expansion loss and the greatest bend loss and sums them to get the total head loss. If surface inflow into the structure is greater than 20% of the total flow, then the total head loss should be increased by 30%.
- *Total Loss Smoothed Method (Inlet Shaping)* - is similar to **Total Loss Method**. The difference is that the total head loss should be decreased by 50% if the manhole or junction incorporates partial diameter inlet shaping or channel smoothing.

After you run either the **Design** or the **Analyze** command on the network, the hydraulic calculation results are displayed. The hydraulic grade line, energy grade line, flow depth, flow velocity at both downstream and upstream ends are reported, Travel Time, which is how long the flow takes to travel through the pipe, and **Min. Cover, which is** the minimum distance from the surface elevation to the crown elevation along the length of the pipe.

A graphic box also shows the hydraulic and energy grade lines, pipe outlines and the ground surface, which help you to observe the design result easily.

VDOT Velocity is the velocity based on VDOT calculation method that assumes an 80% full tailwater condition for the pipe run. It's not available for other methods. The **Travel Time** is also calculated from VDOT velocity when VDOT method is used. **Velocity** is the actual velocity based on the true tailwater condition.



Prompts

Select sewer structure to edit: Pick a manhole symbol.

Pulldown Menu Location: Network

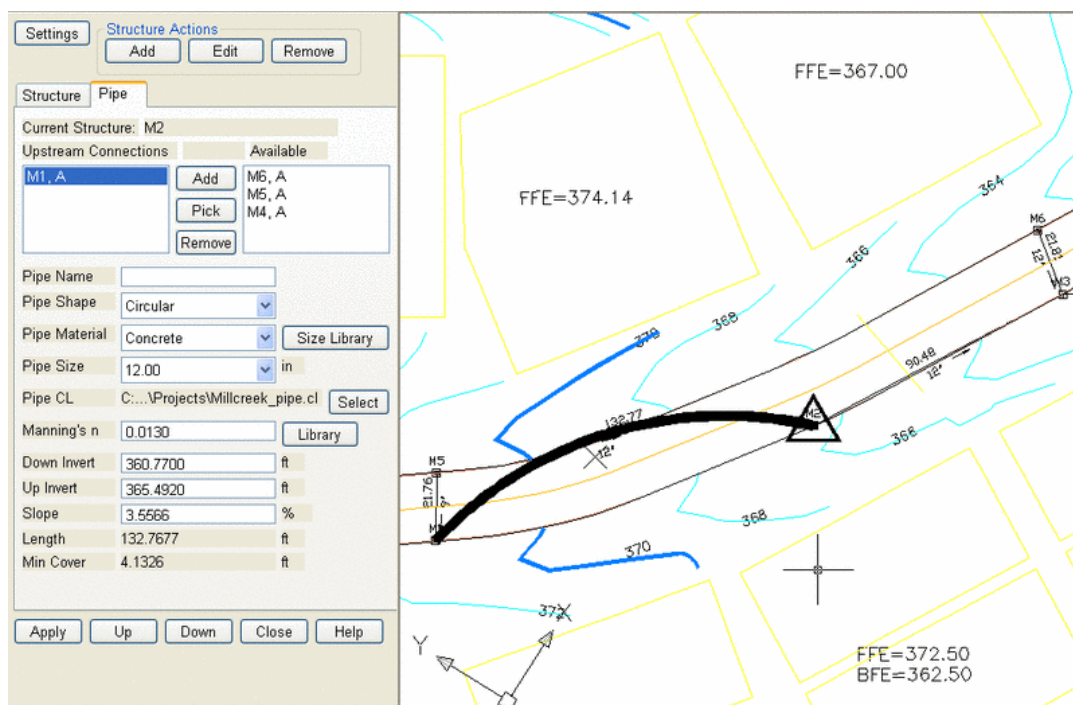
Keyboard Command: editswr

Prerequisite: A sewer (.SEW) file

Edit/Create Sanitary/Utility Structure

This is an alternative version of Edit/Create Sewer Structure for the design of sanitary and utility networks. A sanitary or utility network is generally made up of pipes and structures. There may be more than one pipe entering a structure, but only one can exit. The network type is set in the Sewer Network Settings dialog, and the default type is storm sewer. When you run Edit/Create Sewer Structure, the program performs differently based on the network type. If you work on a storm sewer network, please refer to the documentation on the Edit/Create Sewer Structure.

This command constructs a graphical representation of a pipe network in active drawing, which contains all your design data, such as pipe and structure data. The following is an network example. The Edit/Create Sanitary/Utility Structure dock dialog is on the left, while the network plan view in the active drawing is on the right with current structure and pipe highlighted. When you modify the edit fields on the dock dialog and click on Apply button, the network plan view will be updated automatically. Furthermore, you are allowed to work on the active drawing while the dock dialog is open. This command doesn't perform hydrology calculation, so no flow data is needed.



Edit/Create Sanitary/Utility Structure Dialog and Dynamic Editing in the Active Design Drawing

1. Dock-Dialog Components

First set up the working environment by running command Sewer Network Settings under Network > Sewer Network Setup menu. Then select Edit/Create Sewer Structure. If you are creating a sewer structure, pick a location in the plan view where you want to locate the structure, otherwise click on an existing structure symbol. After a structure has been located, the dock dialog displays, and the current structure symbol is highlighted in the drawing. Following is an example of the dock dialog.

Edit/Create Sewer Structure

This dialog window has two tabs, Structure and Pipe, which are used to enter structure and pipe parameters. The following is the description of the functionalities of the buttons for designing sanitary/utility network.

Settings: This function performs as same as the Sewer Network Settings command except for the generate settings of setting up the network file and surface file etc. Please refer to the documentation on Sewer Network Settings.

Add: Adds a new structure to the network at the location you pick in the plan view. The just created structure will become the active structure for editing.

Edit: This function allows you to pick an existing structure symbol in the plan view to make the structure active for editing.

Remove: This function removes the structure that you pick, and also removes the corresponding pipes and then reconstruct the network.

Apply: Save the changes of the network.

Up: Moves to the upstream structure and makes it active.

Down: Moves to the downstream structure and makes it active.

Close: Quit the sewer network dock dialog.

2. Structure

The structure data is entered through the Structure tab.

Structure		Pipe	
Structure Name	<input type="text" value="M2"/>		
System Name	<input type="text" value="A"/>		
Structure ID	<input type="text" value="Box1"/>	<input type="button" value="Library"/>	
Reference CL		<input type="button" value="Select"/>	
Northing, Easting	<input type="text" value="1012.468, 1072.840"/>	<input type="button" value="Location"/>	
Symbol Name	<input type="text" value="INLET4"/>	<input type="button" value="Symbol"/>	
Symbol Rotate	<input type="text" value="Parallel to CL Up"/>		
Symbol Angle	<input type="text" value="0.0000"/>	<small>dd.mmss</small>	
Symbol Size	<input type="text" value="Drawing Scaler"/>		
Rim Elevation	<input type="text" value="365.9025"/>	<input type="text" value="ft"/>	
Depth	<input type="text" value="5.1325"/>	<input type="text" value="ft"/>	
Invert-Out Elev	<input type="text" value="360.7700"/>	<input type="text" value="ft"/>	
Sump Height	<input type="text" value="0.0000"/>	<input type="text" value="ft"/>	

Sanitary/Utility Network Edit Structure

Structure Name: An identical name of the structure in the network.

System Name: A name for current network. All the structures in the same network have the same system name.

Structure ID: This is the ID of a predefined structure in the structure library. The **Library** button next to it allows you to select or define a sewer structure in the structure library. Once you select the structure, the dimension of the structure are retrieved from the library. Please refer to the documentation of the Sewer Structure Library for details.

Reference CL: The reference centerline is used to locate the structure by station/offset of the centerline points, and align the structure symbol in the graphic. The **Select** button allows you to select a centerline from either a centerline file or a polyline.

Location: This button allows to relocate the structure by pick a position in the drawing.

Symbol Name: This is the name of the symbol that represents the structure in the network plan view. The **Symbol** button allows you to select a symbol from a list of symbols.

Symbol Rotate: There are 10 options to rotate the structure symbol for displaying in the drawing.

Symbol Angle: When the Symbol Rotate value is set to Enter Azimuth Angle, this edit field is enabled for entering an angle.

Symbol Size: Three options to determine the size of the structure symbol.

Rim Elevation: The rim elevation for the structure, it's usually the surface elevation.

Depth: The distance between the rim elevation and the base elevation of the structure.

Invert-Out Elev: The invert elevation of the pipe that exits the structure.

Sump Height: The distance between the base elevation and the invert-out elevation.

3. Pipe

The pipe data is entered through the pipe tab. The **Downstream/Upstream** list contains the connection that exits the structure or the connections that enter the structure, depending on the design direction. The **Available** list contains all the structures that are not connected to the current structure, i.e. the potential structures that can be connected to

the current structure. There is two ways to add a connection. The first one is clicking on the **Add** button to connect the highlighted structure in the Available list to the current structure. The other one is clicking on the **Pick** button and then select a structure symbol in the plan view to connect it. If the connection is unable to be performed, a warning message pops up. By selecting the available structure to **Add** or **Pick** the **Pipe Connection Wizard** dialog will open to allow the user to visualize the connection to be made. See the following section for more information on the **Connection Pipe Wizard**. The **Remove** button allows you to remove the highlighted connection from the Downstream/Upstream list.

Sewer Network Edit Pipe

Pipe Name: An identical name of the pipe in the network.

Pipe Shape: The pipes can have four different cross-sectional shapes: circular, box, horizontal ellipse and vertical ellipse.

Pipe Material: There are nine material options.

Pipe Size: The value can be chosen from a list of pipe sizes stored in the size library.

Pipe Size Library: This button allows you to store commonly used pipe sizes. Please refer to the documentation on the Pipe Size Library for details.

Pipe CL: This option allows you to design a non-straight pipe. The pipe centerline should start from one structure and end at the other exactly. If you change the location of one of the structures, the centerline would be deserted and the pipe would become straight.

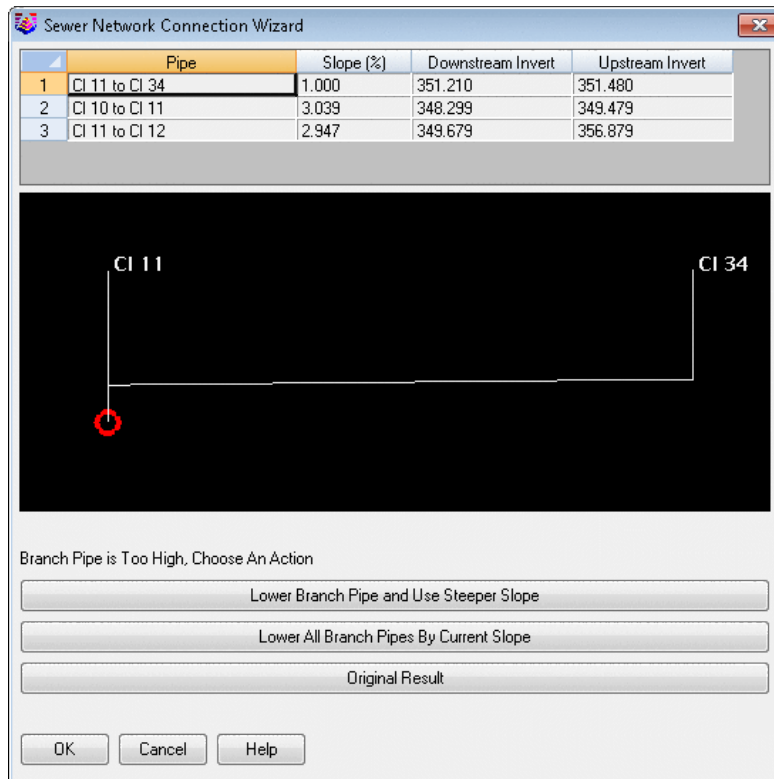
Manning's n: The Manning's n coefficient is used to calculate the friction loss of the pipe. The **Library** button allows you to select a Manning's' n value. Please refer to the documentation on the Manning's N Library for details.

Down Invert/Up Invert: They are the downstream invert and upstream invert elevations of the pipe.

Slope: Pipe slope.

Min. Cover: The minimum distance from the surface elevation to the crown elevation all along the pipe, is calculated automatically.

Selecting the available structure to **Add** or **Pick** the **Pipe Connection Wizard** dialog will open to allow the user to visualize the connection to be made. If the normal slope based from the **Sewer Network Settings** file CAN be created the following dialog box will appear.

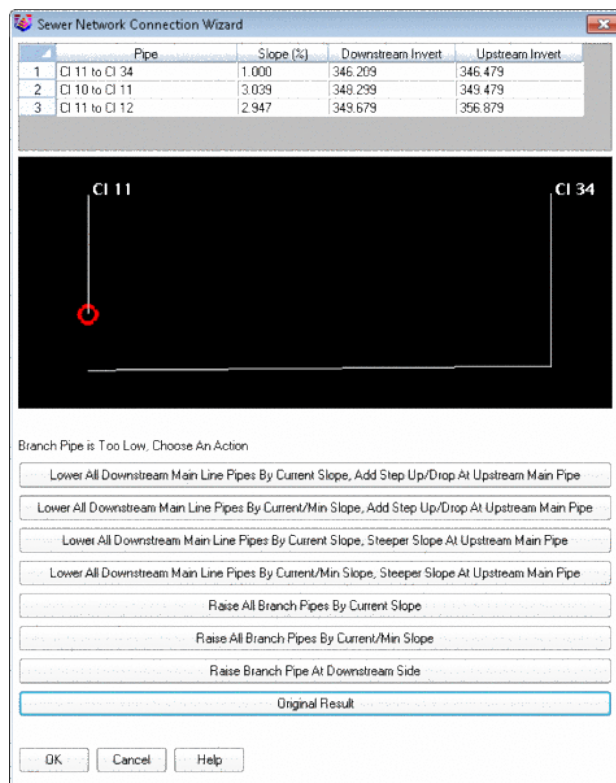


Lower Branch Pipe and Use Steeper Slope: This will change the slope of the branch pipe to match the invert of other upstream connection.

Lower All Branch Pipes By Current Slope: This will lower both inverts and maintain the current slope.

Original Results: This will reset changes made to the original results.

If the normal slope based from the **Sewer Network Settings** file CANNOT be created the following dialog box will appear and offer other design changes for the connection.



Lower All Downstream Main Line Pipes By Current Slope, Add Step Up/Drop At Upstream Main Pipe: This will lower all downstream pipes by the current slope to the branch pipe elevation, therefore creating a step up/drop structure for upstream pipes.

Lower All Downstream Main Line Pipes By Current/Minimum Slope, Add Step Up/Drop At Upstream Main Pipe: This will use the Sewer Network Settings to lower all downstream pipes by the current/minimum slopes to the branch pipe elevation, therefore creating a step up/drop structure for upstream pipes.

Lower All Downstream Main Line Pipes By Current Slope, Steeper Slope At Upstream Main Pipe: This will use the Sewer Network Settings to lower all downstream pipes by the current slope to the branch pipe elevation increasing the upstream pipe connection.

Lower All Downstream Main Line Pipes By Current/Minimum Slope, Steeper Slope At Upstream Main Pipe: This will use the Sewer Network Settings to lower all downstream pipes by the current/minimum slopes to the branch pipe elevation increasing the upstream pipe connection.

Raise All Branch Pipes By Current Slope: This option will raise branch pipes by the current slope

Raise All Branch Pipes By Current/Min Slope: This option will raise branch pipes by the current/minimum slope.

Raise Branch Pipe At Downstream Side: This option will raise the downstream invert of the branch pipe.

Original Results: This will reset changes made to the original results.

Prompts

Select sewer structure to edit: *pick a manhole symbol*

Sewer Structure Data dialog: Fill in values

Pull-down Menu Location: network > Edit/Create Sewer Structure

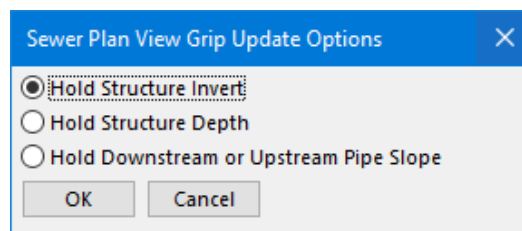
Keyboard Command: editswr/putswr

Prerequisite: a sewer file (.SEW), a surface file (.TIN, .GRD, .FLT)

Edit/Create Sewer Structure - Plan View

The Edit Plan View command allows the user to select a plan view structure and make fast, easy edits to a sanitary\storm network (*.SEW). These edits will automatically update the network file (*.SEW), update the profile drawn, and update other design features such as Energy Grade Line (EGL) and Hydraulic Grade Line (HGL) data as long as the user applies the feature to Save to Profile File when drawing the profile itself.

Command Prompt: *Select sewer structure to move [Options]: Press O for Options*



Sewer Plan View Grip Update Option allows the user to define what happens to the relocated structure. Hold Structure Invert would maintain the original invert elevation to the new location. Hold Structure Depth would hold the original depth to the new location. Hold Downstream or Upstream Pipe Slope maintains the original pipe slopes to the new location.

Prompts

Select sewer structure to move [Options]:

Pull-down Menu Location: Network

Keyboard Command: sewer_plan_grip

Prerequisite: A sewer (.SEW) file

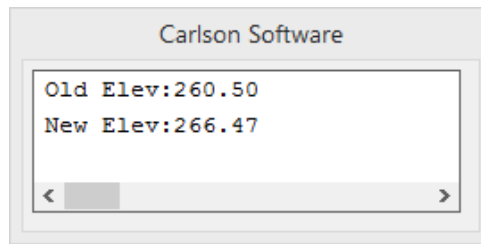
Edit/Create Sewer Structure - Profile View

This command allows you to select a structure or a pipe on a profile and make fast, easy edits to a sanitary\storm network (*.SEW). These edits will automatically update the network file (SEW), update the profile drawn, and update other design features such as Energy Grade Line (EGL) and Hydraulic Grade Line (HGL) data as long as the feature to Save to Profile File when drawing the profile is active.

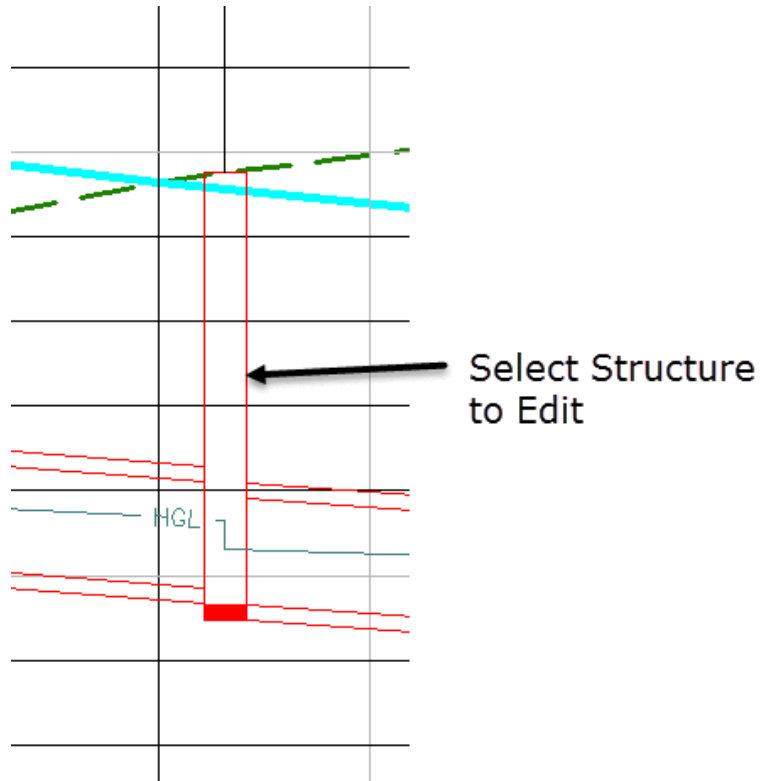
When making profile edits you can select either a structure or a pipe to edit. When a structure is selected, you can make elevation edits to the selected structure. When a pipe is selected, there are options to modify the pipe invert in or invert out by selecting on the appropriate end of the pipe, or by selecting the middle of the pipe to hold the pipe slope and move the entire pipe for vertical revisions.

First select the structure to edit. The following dialog box will open showing the Old Elev: and the New Elev: for the structure invert elevation. You can then enter new elevation or pick a point for elevation on screen. You can also edit the structure rim elevation by typing R for Rim at the command line. Structures can only be edited in the

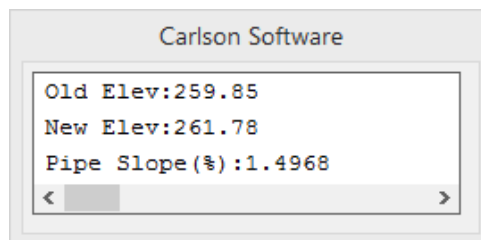
profile vertically.



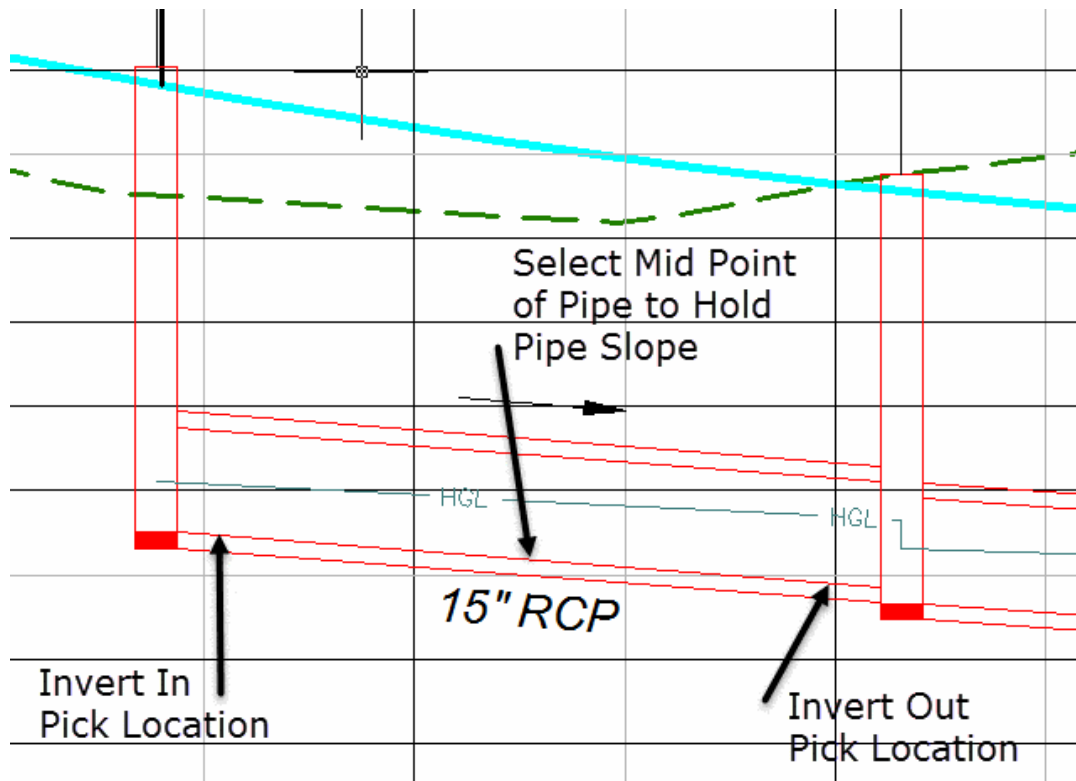
Structure Edits can be made by selecting the structure to edit and screen picking or entering new elevations.



Select the pipe to edit. The following dialog box will open showing the invert elevations and pipe slope. You can then enter new elevation or pick a point for elevation on screen. At the command line, you can also enter keywords for Size, Slope or Step up to change the pipe size, pipe slope or step-up.



The pipe edit mode depends on where you select the pipe. Selecting near the mid point holds the pipe slope and you can edit the elevations together. Selecting near an invert edits only that end of the pipe and you can change the slope.



Prompts

Select sewer entity to move (Enter to End): *pick a structure*

Enter new elevation or pick a point for elevation [Rim Elevation](Enter to End): *pick elevation*

Select sewer entity to move (Enter to End) [Undo/Exit]: *pick a pipe*

Enter new elevation or pick a point for elevation [Size/STepup] (Enter to End): *pick elevation*

Select sewer entity to move (Enter to End) [Undo/Exit]: *press Enter*

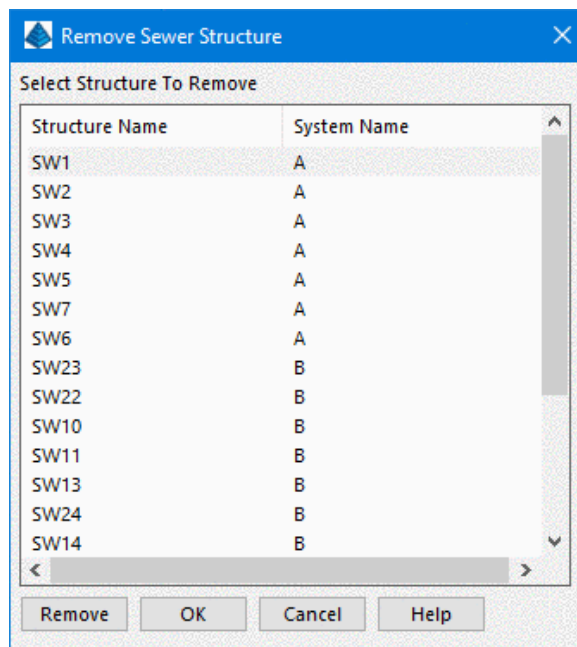
Pulldown Menu Location: Network > Edit Sewer Structure

Keyboard Command: sewer_prof_grip

Prerequisite: A sewer (SEW) file and saved (PRO) profile pipes.

Remove Sewer Structure

This command removes a structure from the sewer network. The structure to remove can be selected from a list of structure names or screen picked. To screen select a manhole, pick on the manhole symbol. The manhole symbol and labels are erased from the screen and the manhole is removed from the sewer network file.



Prompts

Select structures to erase by screen pick or name list [**<Pick>/List**]? *press Enter for Pick*

Select sewer structure to remove: *pick a manhole symbol*

Select sewer structure to remove: *press Enter to end*

Pulldown Menu Location: Network

Keyboard Command: rmswr

Prerequisite: Sewer network manholes

Lateral Design Overview

Lateral design is available in both the storm sewer and sanitary sewer design programs. A lateral pipe is very much like a regular pipe in sewer network. A lateral is the underground pipe that connects a residence or business to the main line pipe. The lateral connection has a property of offset elevation relevant to the mainline sewer invert at the point of connection. Both the cleanout and connection can be either a box or circular structure. The lateral data is stored in the main network / current network file, although designed using a different command. A prerequisite for Lateral Design the user must have completed either a Storm Network or Sanitary Sewer Network as the main line, and set the file as the Active File to be used in the Sewer Network Settings dialog box.

There are five options to locate a Lateral Connection Point. If connecting to a structure, select "Pick A Structure". If the lateral is to connect at the main, there are 4 options.

Pick a Pipe.

Using this option, a connection is made where a pipe is selected.

Pick a Pipe and Exact Location on Main.

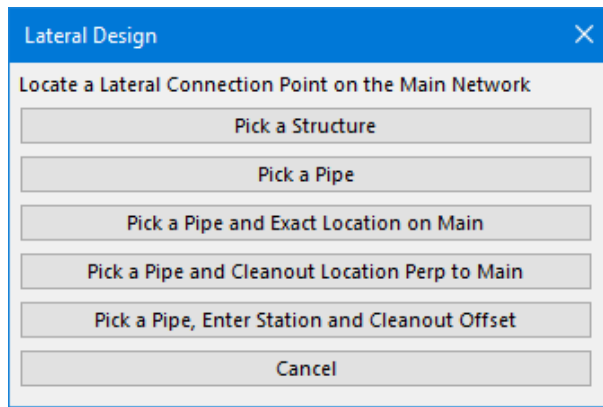
Using this option, a pipe is selected, then the connection location is selected.

Pick a Pipe and Cleanout Location Perpendicular to Main

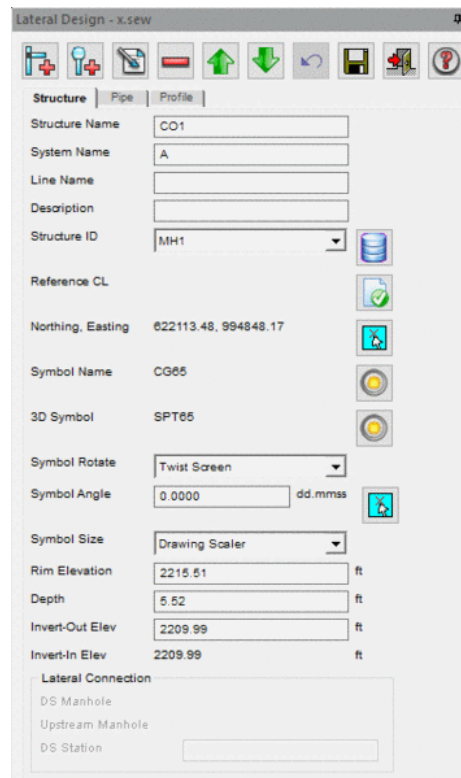
Using this option, a target pipe is selected, then the cleanout is located perpendicular to the selected pipe.

Pick a Pipe, Enter Station and Cleanout Offset.

Using this option, a target pipe is selected and the connection is made at a specified Station and offset.



The Pick a Structure allows the user to select a structure within the Main Network File and opens the following Docked Dialog. This dialog box works similar to the regular pipe in sewer network. By hovering the mouse over the icons at the top of the dialog box the user can see the command associated with each. Starting from left to right the commands are as followed Add Connection, Add Cleanout, Edit, Remove, Up, Down, Save, Exit, and Help. The Structure Tab gives the user information of the selected structure. The dialog box specifies the downstream and upstream manhole connections as well as the station along the main network pipe measured from the downstream structure. The Connection Height allows the user to specify the entrance height of the connection on the main network structure. Depending on the type of the connection point some of the dialog box will only supply data while others may be edited. In the following example we selected to connect to a structure, therefore data present related to the actual manhole with only the connection height available for editing.

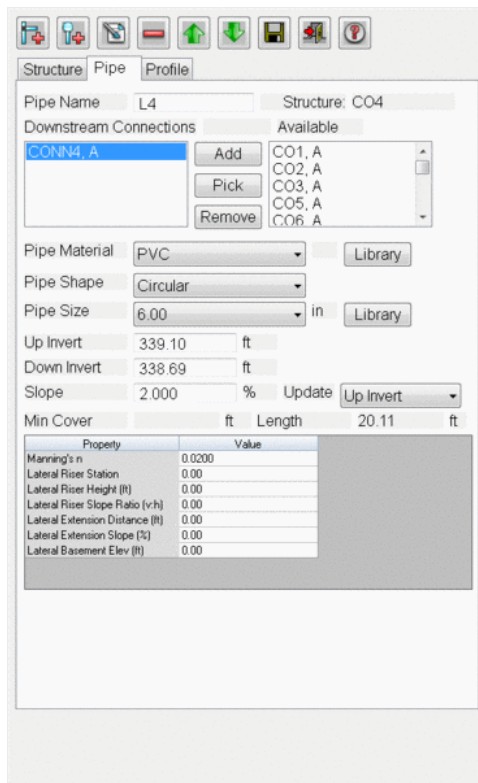


When one of the 4 Pick a Pipe options is selected other fields in the docked dialog box are available for selection allowing the user to specify a Structure Name, System Name, Description, Symbol Rotate, Symbol Angle, and Symbol Size. Rim Elevations and Depth are calculated and the downstream and upstream manholes shown. The user is given the downstream station for the connection that can be adjusted if desired, as well as a connection height.

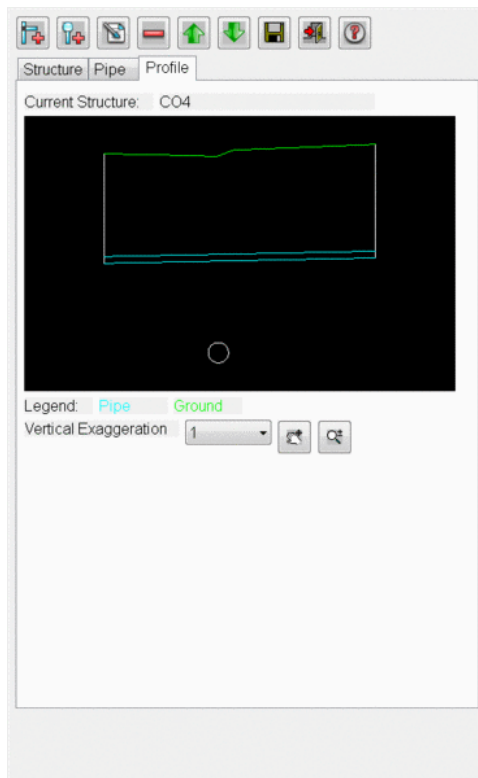
The screenshot shows a software dialog box titled "Pick a Pipe" with three tabs: "Structure", "Pipe", and "Profile". The "Pipe" tab is active. The dialog contains the following fields and controls:

- Structure Name: CONN147
- System Name: A
- Description: (empty)
- Structure ID: None (dropdown menu) with a "Library" button
- Reference CL: (empty) with a "Select" button
- Northing, Easting: 801043.24, 2032895.27 with a "Location" button
- Symbol Name: Spt6 with a "Symbol" button
- 3D Symbol: (empty) with a "Symbol" button
- Symbol Rotate: Twist Screen (dropdown menu)
- Symbol Angle: 0.0000 dd mmss
- Skew Angle: 90.0000 dd mmss
- Symbol Size: Unit Symbol (dropdown menu)
- Rim Elevation: 347.46 ft
- Depth: 9.22 ft
- Invert-Out Elev: 338.24 ft
- Invert-In Elev: (empty) ft
- Lateral Connection section:
 - DS Manhole: MH20, A
 - Upstream Manhole: MH21, A
 - DS Station: 107.97
 - Connection Height: 0.00 ft

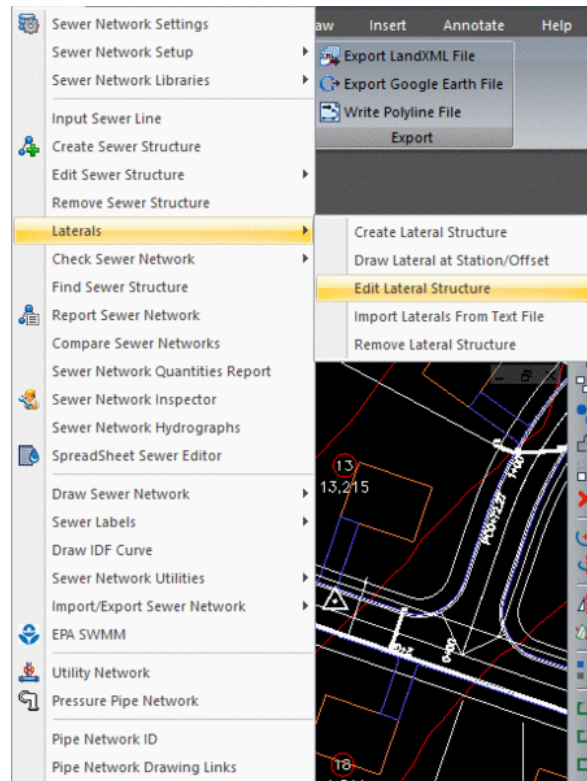
The Pipe Tab of the docked dialog will display the pipe lateral data once a cleanout location is located. This is the same for both types of connections. This dialog box allows the user to design pipe data associated with the lateral pipe, and is used to assign Pipe Name, to assign Downstream Connections, Pipe Material, Pipe Shape, Pipe Size, set the Inverts Upstream and Downstream, or to adjust the Pipe Slope and which invert to Update. The properties box at the bottom displays other properties and values assigned.



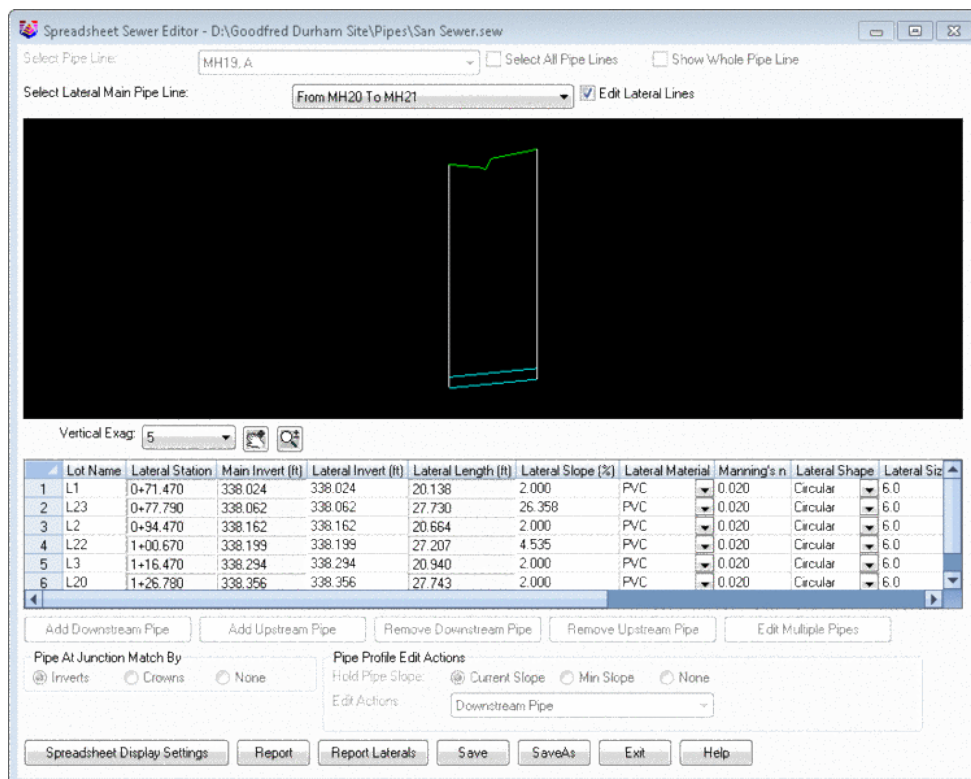
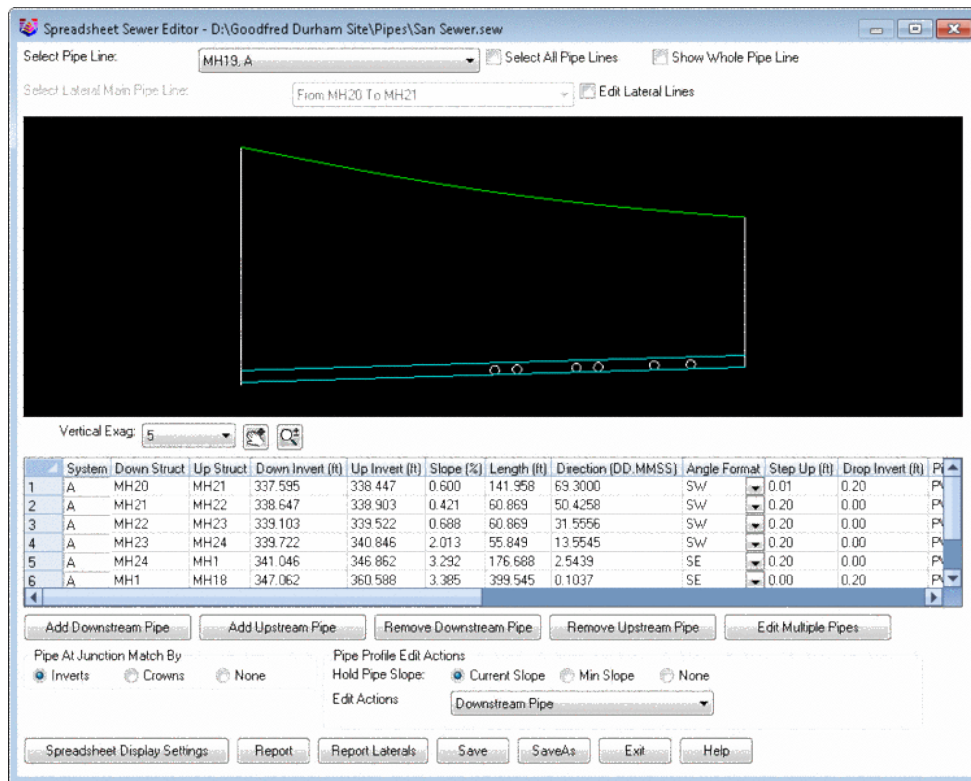
The Profile Tab provides a visual aid showing the lateral pipe and other associated pipe crossings for consideration during design. A Vertical Exaggeration can also be applied for an increased aid.



Editing a lateral is similar to a regular pipe. The invert elevation of both ends of the lateral can be modified the same as a regular pipe. To edit a lateral structure the user can select from the following image in the network menu or double click the lateral structure to edit. After selecting the docked dialog box opens and edits can be made on the Structure or Pipe Tab.



The Spreadsheet Sewer Editor can also be used to review and edit lateral connections. Toggle the Edit Lateral Line to revise or adjust laterals.



With the toggle on the user Selects Lateral Main Pipe Line, and the laterals associated with that main line pipe can be edited.

Editing a lateral is similar to a regular pipe. The invert elevation

Pulldown Menu Location: Network

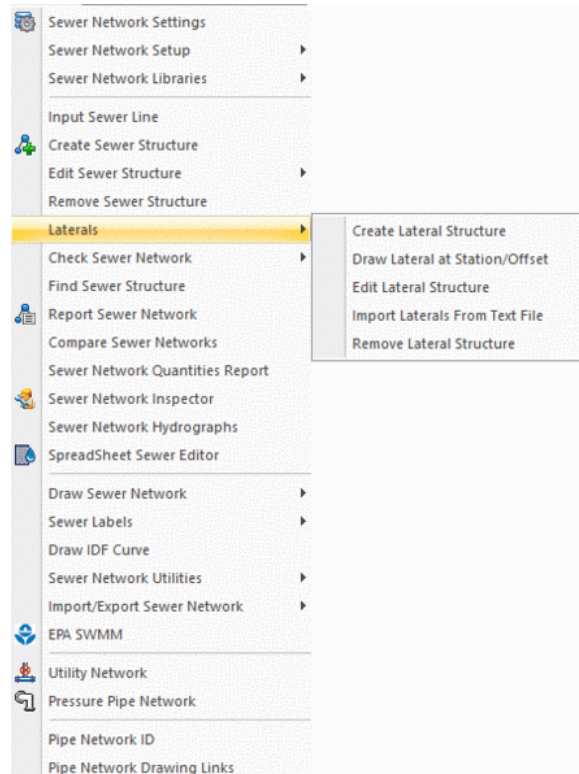
Keyboard Command: editlat

Prerequisite: A sewer (.SEW) file

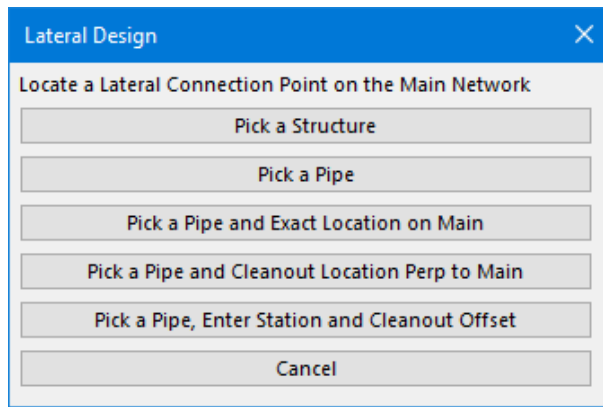
Create/Edit Lateral Structure

Lateral design is available in both the storm sewer and sanitary sewer design programs. A lateral pipe is very much like a regular pipe in sewer network. A lateral is the underground pipe that connects a residence or business to the main line pipe. The lateral connection has a property of offset elevation relevant to the mainline sewer invert at the point of connection. Both the cleanout and connection can be either a box or circular structure. The lateral data is stored in the main network / current network file, although designed using a different command. A prerequisite for Lateral Design the user must have completed either a Storm Network or Sanitary Sewer Network as the main line, and set the file as the Active File to be used in the Sewer Network Settings dialog box.

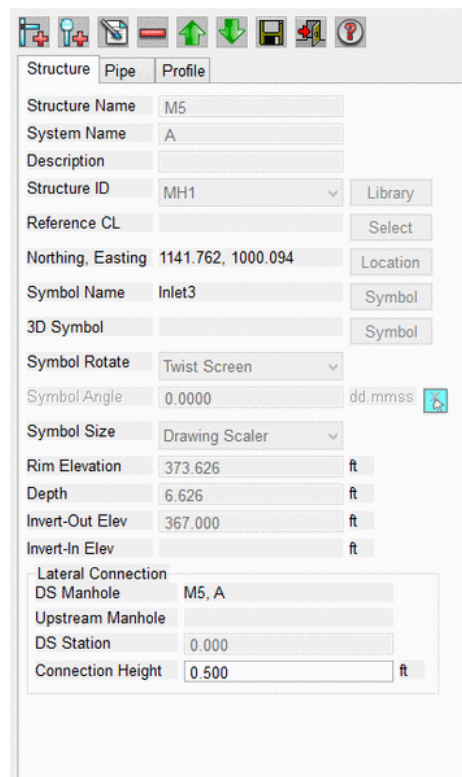
There are two commands to open the Lateral Network dialog: Create Lateral Structure and Edit Lateral Structure. To Create a Lateral Structure, you are required to locate a lateral connection on the existing network; To edit a lateral structure the user can run the command from the the network menu or double click the lateral structure to edit.



There are five options to locate a Lateral Connection Point. **Pick a Structure**, this can be any structure created in the main line file such as a manhole or a box structure. Multiple lateral connections are allowed at a manhole or a box structures. **Pick a Pipe** is to create a pipe connection along an existing pipe, where the only one lateral is allowed. **Pick a Pipe and Exact Location on Main** is to create a pipe connection at an exact location on the main pipe. **Pick a Pipe and Cleanout Location Perp to Main** allows users to pick the main pipe first, and then pick where the cleanout is, the program would create a lateral pipe perpendicular to the main pipe.

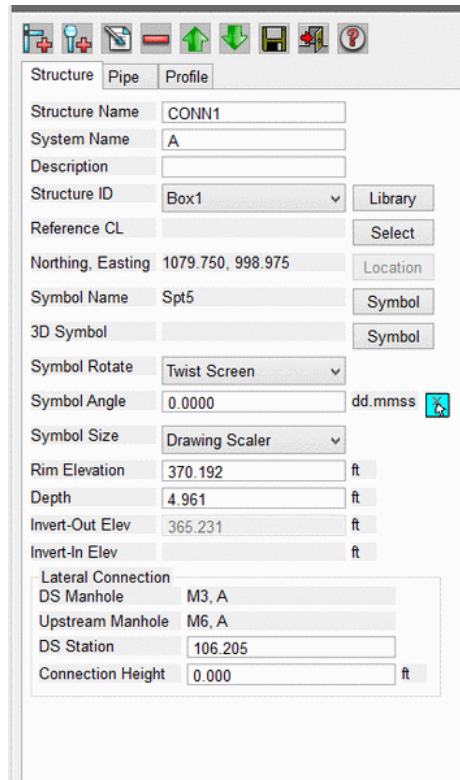


The **Pick a Structure** options allows the user to select an existing structure within the Main Network File and opens the following Docked Dialog. By hovering the mouse over the icons at the top of the dialog box the user can see the command associated with each. Starting from left to right the commands are as followed **Add Connection, Add Cleanout, Edit, Remove, Up, Down, Save, Exit,** and **Help**. The Structure Tab gives the user information of the selected structure. The dialog box specifies the downstream and upstream manhole connections as well as the station along the main network pipe measured from the downstream structure. The Connection Height allows the user to specify the entrance height of the connection on the main network structure. Depending on the type of the connection point some of the dialog box will only display data while others may be edited. In the following example we selected to connect to a structure, therefore data present related to the actual manhole with only the connection height available for editing. The **Connection Height** is the value how much the lateral invert elevation is higher than the main pipe invert elevation at the lateral connection.



When the **Pick a Pipe** option is selected other fields in the docked dialog box are available allowing users

to specify a Structure Name, System Name, Description, Symbol Rotate, Symbol Angle, and Symbol Size. Rim Elevations and Depth are calculated and the downstream and upstream manholes shown. The user is given the downstream station for the connection that can be adjusted if desired, as well as a connection height.

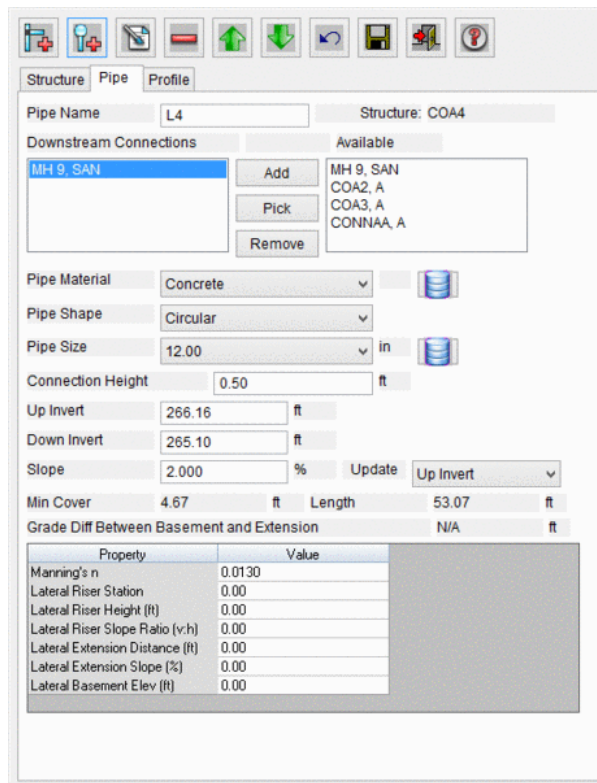


The image shows a software dialog box with three tabs: Structure, Pipe, and Profile. The 'Pipe' tab is selected. The dialog contains the following fields and controls:

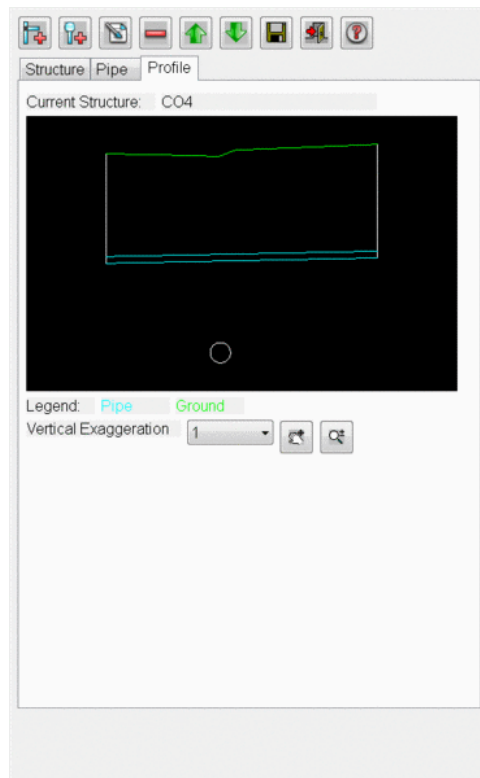
- Structure Name: CONN1
- System Name: A
- Description: (empty)
- Structure ID: Box1 (dropdown menu) with a Library button
- Reference CL: (empty) with a Select button
- Northing, Easting: 1079.750, 998.975 with a Location button
- Symbol Name: Spt5 with a Symbol button
- 3D Symbol: (empty) with a Symbol button
- Symbol Rotate: Twist Screen (dropdown menu)
- Symbol Angle: 0.0000 with a unit dropdown set to dd.mmss and a help icon
- Symbol Size: Drawing Scaler (dropdown menu)
- Rim Elevation: 370.192 ft
- Depth: 4.961 ft
- Invert-Out Elev: 365.231 ft
- Invert-In Elev: (empty) ft
- Lateral Connection section:
 - DS Manhole: M3, A
 - Upstream Manhole: M6, A
 - DS Station: 106.205
 - Connection Height: 0.000 ft

The Pipe Tab of the docked dialog displays the lateral pipe data once a cleanout location is located. This is the same for both types of connections. This dialog box allows the user to design pipe data associated with the lateral pipe, and is used to assign Pipe Name, to assign Downstream Connections, Pipe Material, Pipe Shape, Pipe Size, set the Inverts Upstream and Downstream, or to adjust the Pipe Slope and which invert to Update.

The properties box at the bottom displays other properties and values assigned.



The Profile Tab provides a visual aid showing the lateral pipe and other associated pipe crossings for consideration during design. A Vertical Exaggeration can also be applied for an increased aid.



Laterals can be created using the Draw Lateral by Station Offset command. For this command, "Station Number of reference" is the distance from the nearest downstream storm structure to the lateral connection.

Command: drswrlat

Select the downstream sewer structure (Esc to exit):

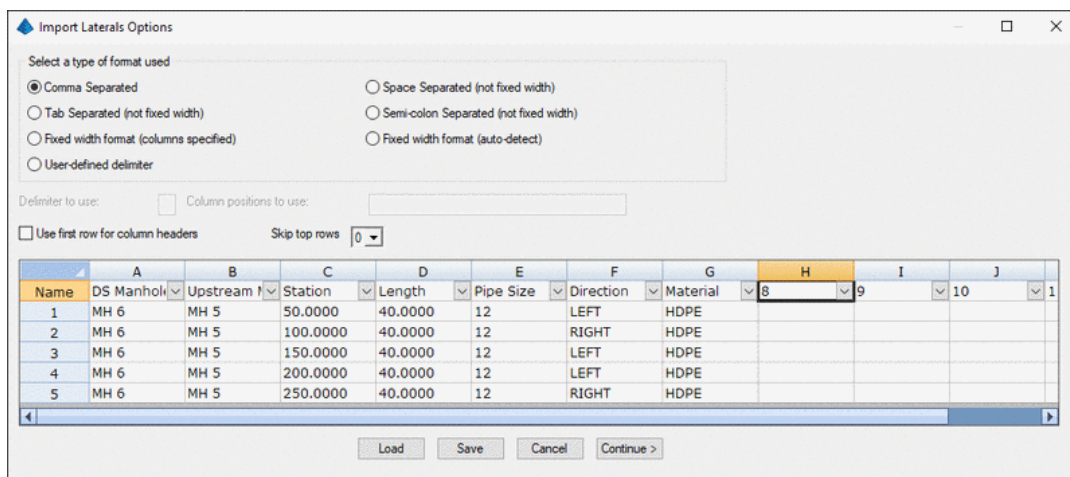
Enter Lateral Station (> 0): 100

Enter Lateral Offset: 15

The command will draw a lateral at the lineal distance of 100 feet upstream of the selected downstream sewer structure. The service connection will be located 15' from the pipe main, perpendicular to the proposed Utility.

This command imports the lateral pipe data from text file to the current sewer network file. The text file has the lateral data with one lateral pipe per row in the text file with the fields separated by a comma or other delimiter. The text file requires main pipe downstream manhole, main pipe upstream manhole name, station from downstream manhole name, lateral pipe length, pipe size, direction that lateral pipe on the left or right side of the main pipe by looking at upstream direction, and pipe material.

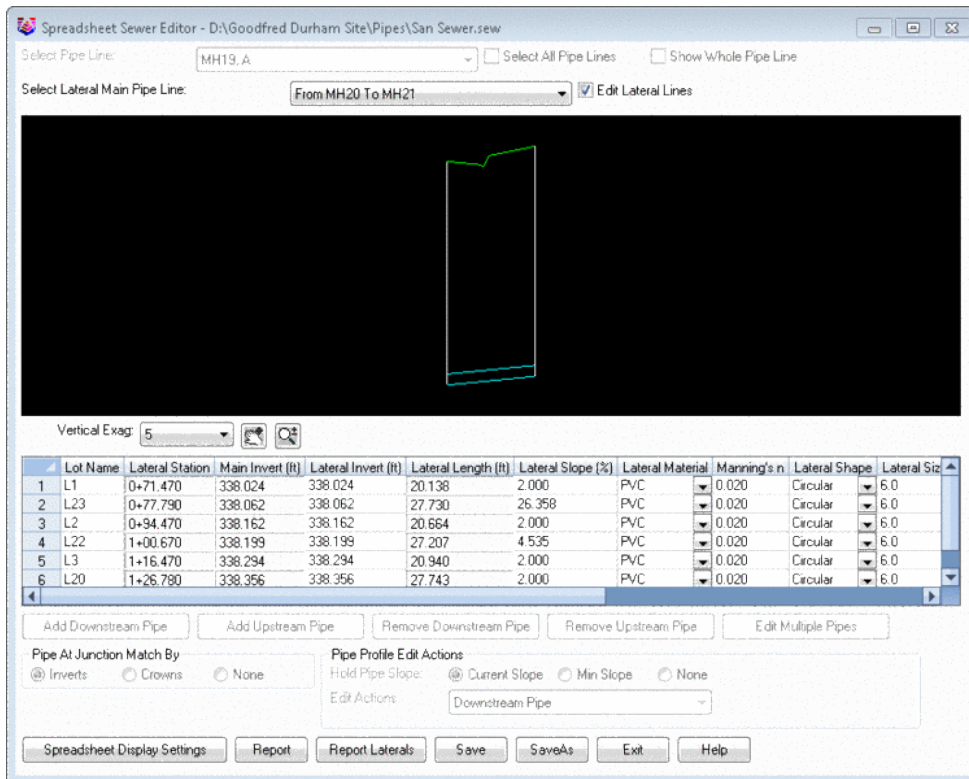
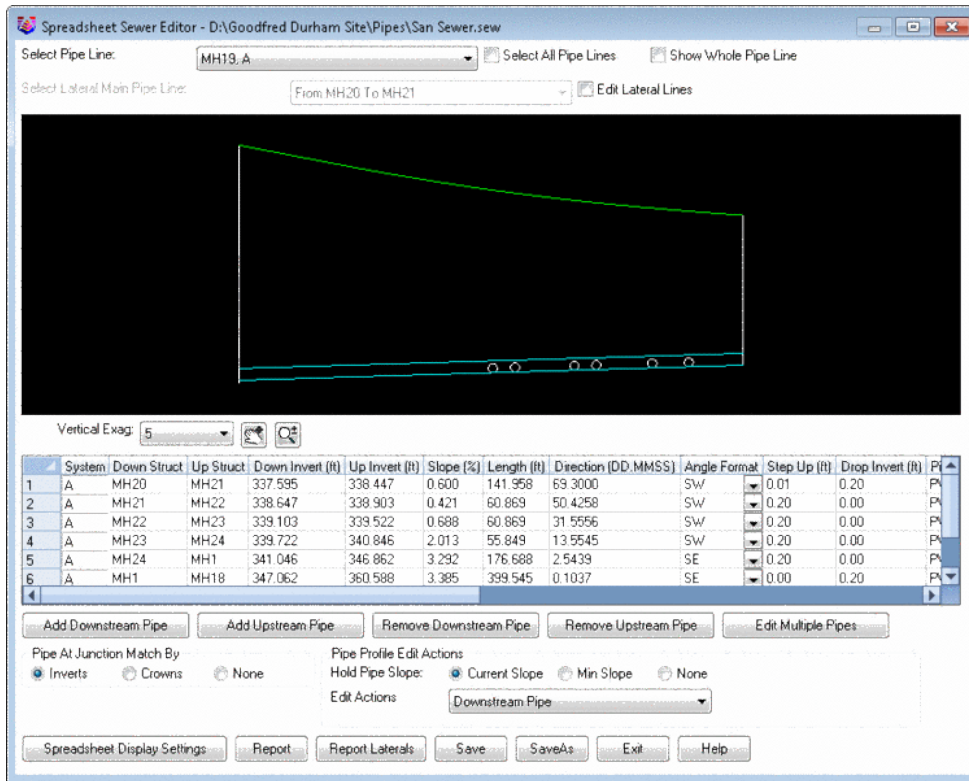
The program prompts for the lateral file to import. Then there is a dialog to specify the format of the input text files. For each column of data in the input files, choose which field to assign.



Example lateral data file:

DS Manhole, Upstream MH, Station from DS Manhole, Lateral Length, Pipe Size, Direction, Pipe Material
MH 6,MH 5,50.0000,40.0000,12,LEFT,HDPE
MH 6,MH 5,100.0000,40.0000,12,RIGHT,HDPE
MH 6,MH 5,150.0000,40.0000,12,LEFT,HDPE
MH 6,MH 5,200.0000,40.0000,12,LEFT,HDPE
MH 6,MH 5,250.0000,40.0000,12,RIGHT,HDPE

The Spreadsheet Sewer Editor can also be used to review and edit lateral connections. Toggle the Edit Lateral Line to revise or adjust laterals. With the toggle on the user Selects Lateral Main Pipe Line, and the laterals associated with that main line pipe can be edited.



Pulldown Menu Location: Network

Keyboard Command: putlat

Prerequisite: A sewer (.SEW) file

Import Laterals From Text File

This command imports the lateral pipe data from text file to the current sewer network file. The text file has the lateral data with one lateral pipe per row in the text file with the fields separated by a comma or other delimiter. The text file requires main pipe downstream manhole, main pipe upstream manhole name, station from downstream manhole name, lateral pipe length, pipe size, direction that lateral pipe on the left or right side of the main pipe by looking at upstream direction, and pipe material.

The program prompts for the lateral file to import. Then there is a dialog to specify the format of the input text files. For each column of data in the input files, choose which field to assign.

Name	DS Manhole	Upstream MH	Station	Length	Pipe Size	Direction	Material	8	9	10	1
1	MH 6	MH 5	50.0000	40.0000	12	LEFT	HDPE				
2	MH 6	MH 5	100.0000	40.0000	12	RIGHT	HDPE				
3	MH 6	MH 5	150.0000	40.0000	12	LEFT	HDPE				
4	MH 6	MH 5	200.0000	40.0000	12	LEFT	HDPE				
5	MH 6	MH 5	250.0000	40.0000	12	RIGHT	HDPE				

Example lateral data file:

DS Manhole, Upstream MH, Station from DS Manhole, Lateral Length, Pipe Size, Direction, Pipe Material

MH 6,MH 5,50.0000,40.0000,12,LEFT,HDPE

MH 6,MH 5,100.0000,40.0000,12,RIGHT,HDPE

MH 6,MH 5,150.0000,40.0000,12,LEFT,HDPE

MH 6,MH 5,200.0000,40.0000,12,LEFT,HDPE

MH 6,MH 5,250.0000,40.0000,12,RIGHT,HDPE

Pulldown Menu Location: Network > Laterals

Keyboard Command: csv2lat

Prerequisite: structure and pipe text files.

Remove Lateral Structure

Look at the command prompt:

Select structures to erase by screen pick or name list [<Pick>/List]?

Pulldown Menu Location: Network

Keyboard Command: rmlat

Prerequisite: A sewer (.SEW) file

Check Sewer Network Parameters

This command reads a sewer network file and audits the sewer network for any invalid data fields or values that don't follow the specified design constraints. If any problems are found with the sewer data, a report is displayed indicating all invalid values.

Pulldown Menu Location: Network > Check Sewer Network

Keyboard Command: chkswr

Prerequisite: A sewer file (.SEW) and corresponding Inlet Data and Pipe Size Data.

Check Reference Centerlines and Surface

This command is a manual alternative to the *Auto* linking method found on the General tab of the **Sewer Network Settings**. This routine compares the sewer network structure locations to that of the reference surface and centerline(s). The current sewer surface file is used as the elevation reference for the structure rim elevations.

Each structure has the option to assign a reference centerline and the structure will record the station and offset from this centerline. If this routine finds a difference between the structure location and the referenced centerline, an option will be provided to either update the sewer structures to their new and relative station/offset or simply leave the structures where they are located.

For reference surface changes, the structure rim elevation can be updated and there are two options for updating the invert elevation. One method is to hold the structure depth and change the invert elevation. The other method is to hold the invert elevation and change the depth.

Pulldown Menu Location: Network > Check Sewer Network

Keyboard Command: chkswrref

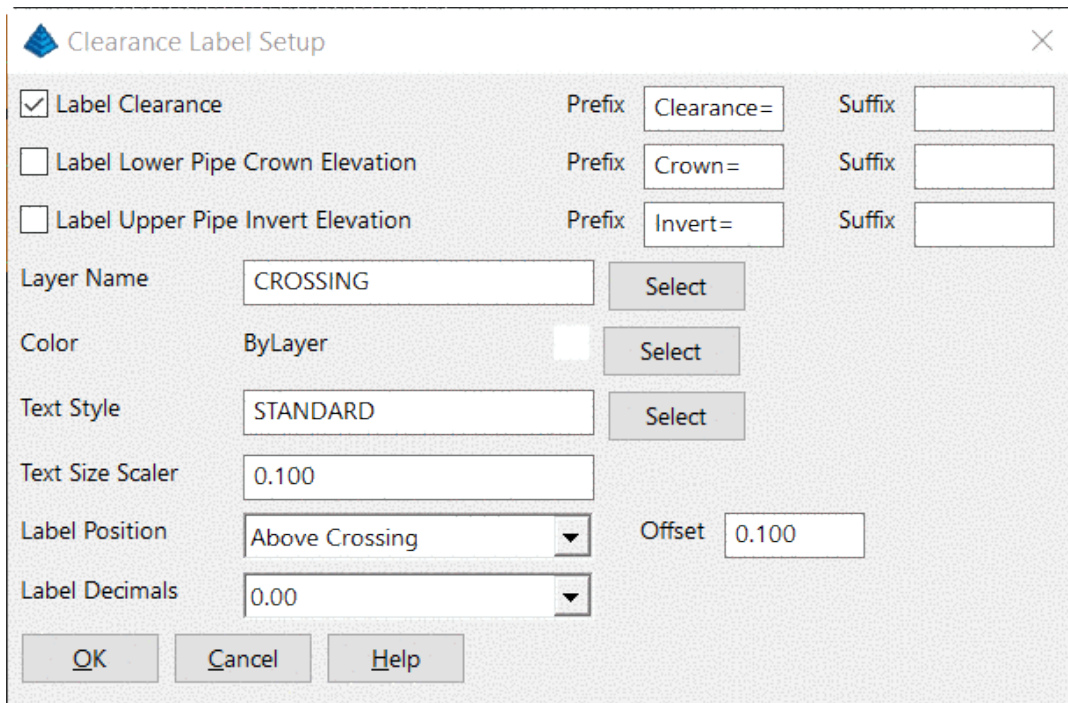
Prerequisite: Sewer (.SEW) file

Check/Label Clearance

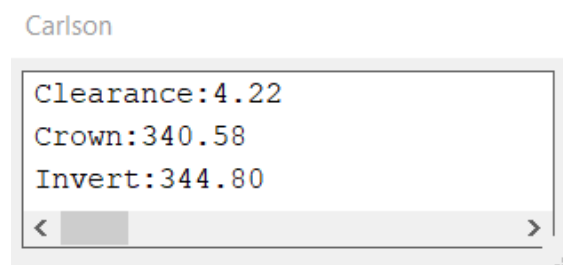
This command check clearance of a crossing between any sewer network pipes, utility network pipe, and 3D pipe polylines. It displays clearance value, as well as the invert and crown elevations on the command line. Users can also label the results next to the crossing on screen.

From the Network > Check Sewer Network menu in the Hydrology Module, select Label Clearance at Crossings. In the drawing, first have all the networks drawn in plan view. Hovering over a pipe crossing in plan-view will give real time information on screen dialog box.

The Clearance Label Setup Dialog is used to draw the clearance labels on screen. The settings are saved and no need to bring it up next time to check and label clearance. Clear the Draw Labels box and the program won't label the results on screen.



Label Setup Dialog



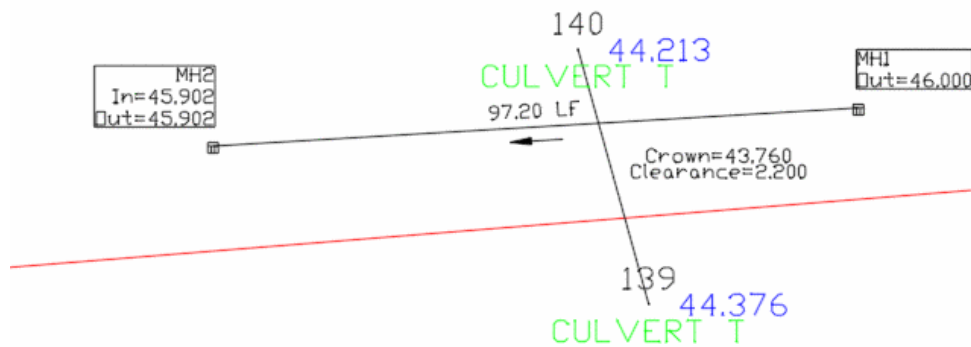
Real Time Clearance Result Dialog Box on Screen

```

number of detected entities: 0
Command: LABEL_CLEARANCE
Pick crossing location (0 for Options,Enter to End): 0
Pick crossing location (0 for Options,Enter to End):
Clearance=2.200, Crown=43.760, Invert=45.960
Pick crossing location (0 for Options,Enter to End):
Command:

```

Clearance Result on Command Line



Clearance Labels on Screen

Pulldown Menu Location: Network > Check Sewer Network

Keyboard Command: label_clearance

Prerequisite: sewer network/utility network/3D pipe polyline crossings in plan view

Collision Conflicts Check

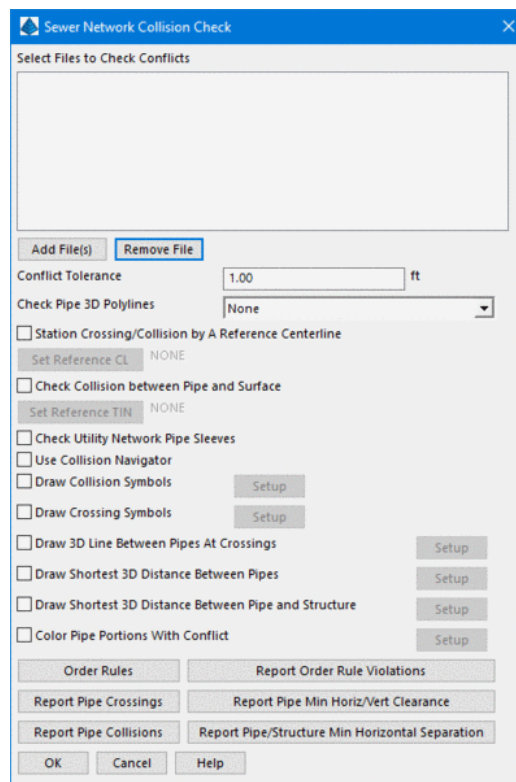
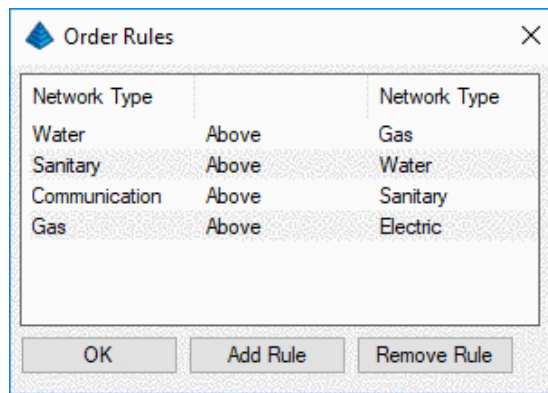
When there are two or more sewer networks in one area, it's very important to know if there are any collisions among pipe lines. The Collision Conflicts Check command performs a three dimensional check on unlimited sewer .SEW, utility .UTIL networks and 3D pipe polylines. If any portions of the these systems are too close to each other, i.e their distance is closer than the safety buffer, the conflicting pipes and their collision locations would be reported. The program also checks for any collisions within the same sewer network. Besides checking unlimited sewer networks, the Check Pipe 3D Polyline option will check for collisions with 3D polylines that have been tagged as pipes with the Assign Pipe Data To Polyline command.

From the Network > Check Sewer Network menu in the Hydrology Module, select Collision Conflicts Check. In the dialog, select two or more network files that you want to check the collisions and/or select the Check Pipe 3D Polyline option. The program can also check collisions for 3D Pipes only and allow you to pick 3D pipe polylines on screen.

In the Conflict Tolerance box, specify the value of the distance buffer that any two of the sewer systems shouldn't violate. Optionally, use the Station Crossing/Collision by Reference Centerline switch and set the Reference CL to report any crossings and collisions by Station and Offset from the reference centerline. The Use Collision Navigator toggle provides an Data Problem Log dialog to navigate you to the location of every collision, otherwise the collisions would reported in Carlson standard report format.

Draw Collision Symbols and Draw Crossing Symbols option will draw a symbol at each collision or crossing. Besides drawing symbols, there are options to label the clearance and pipe elevations.

The **Order Rules** button displays the Order Rules dialog, allowing the user to specify and report violations of the "depth order" of a set of various utilities. For example, if Gas is supposed to be below Water, as indicated in the dialog below, but is actually above, the program will note this as a violation and report this in the Report Order Rule Violations formatted report.

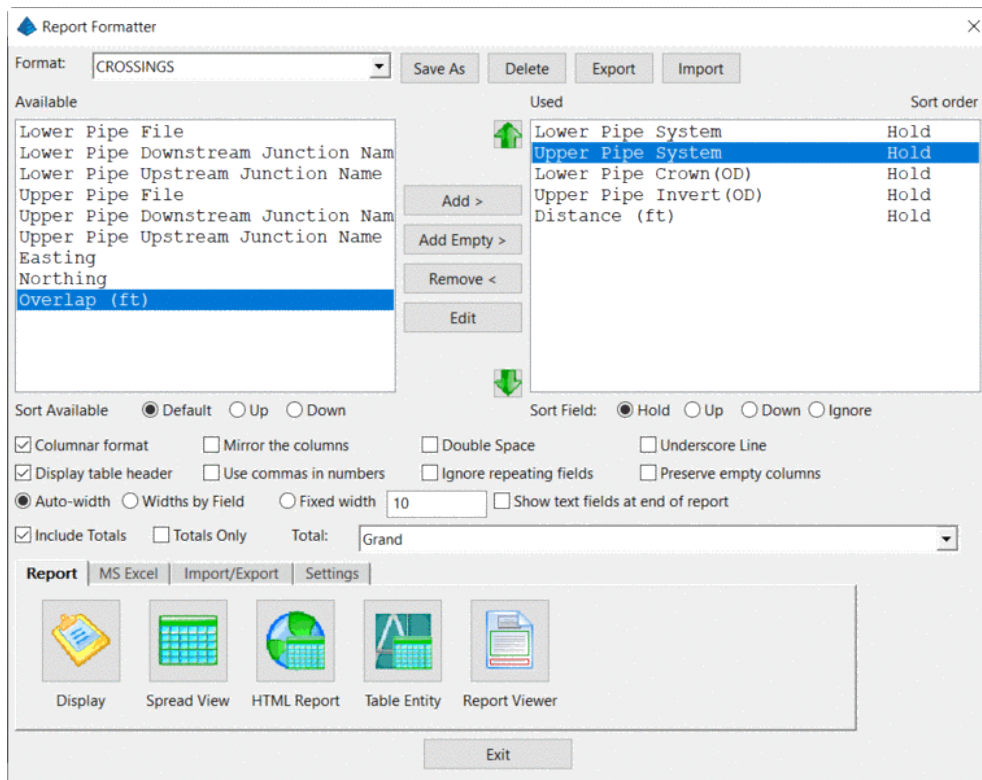


Sewer Network Collision Conflict Check

The **Report Crossings** and **Report Collisions** buttons would show all the pipe crossings and collisions in a formatted report.

Report Pipe Crossings Report Pipe Crossings

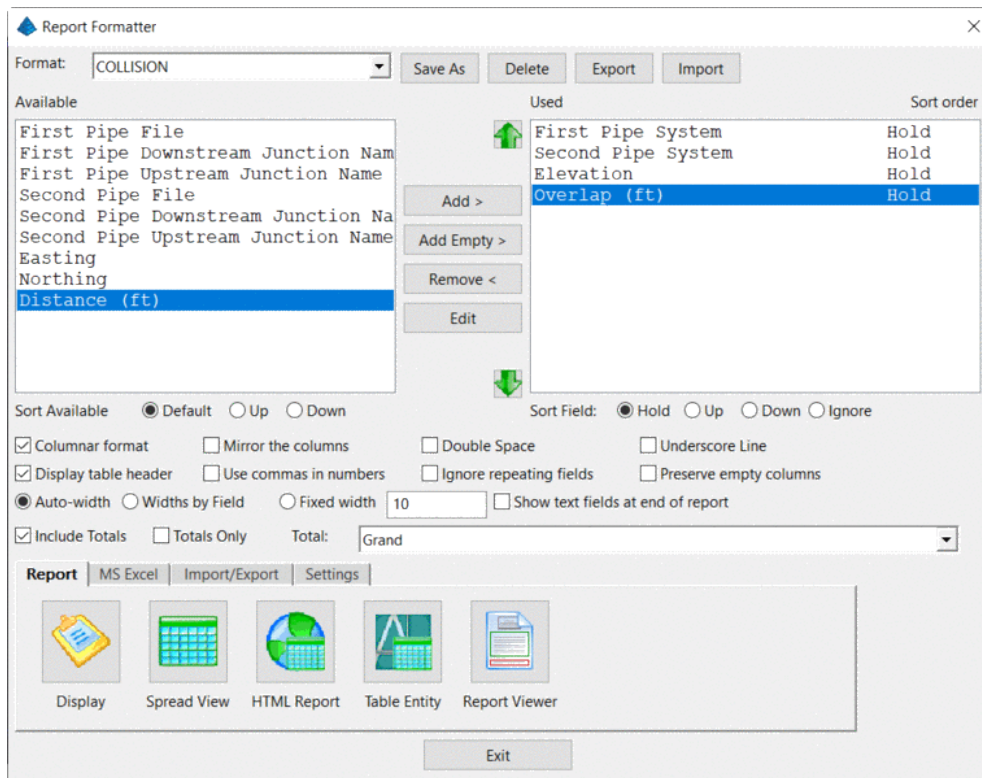
Opens the Report Formatter Options Dialog Box. The user can select from the Available List and Add the to the Used list to customize the report, and select output options. For more information on using the Report Formatter see Report Viewer in this manual.



Report Collisions

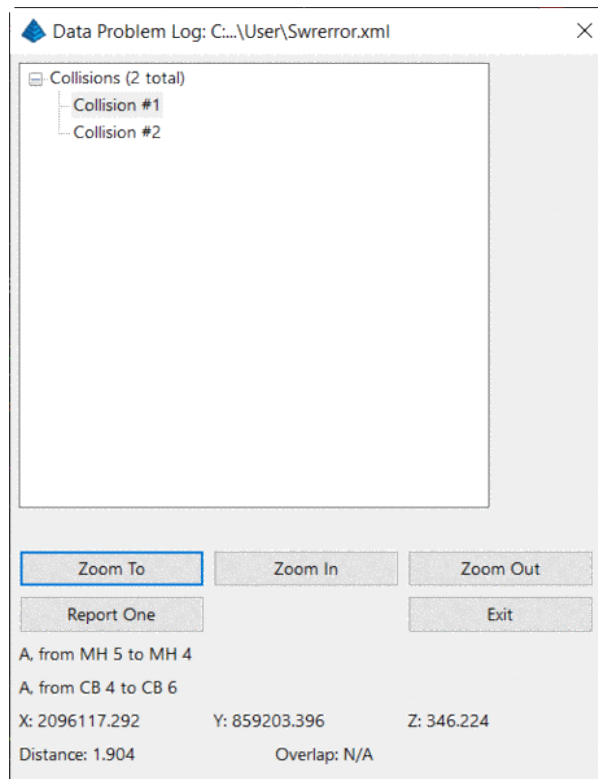
Report Pipe Collisions

Opens the Report Formatter Options Dialog Box. The user can select from the Available List and Add the to the Used list to customize the report, and select output options. For more information on using the Report Formatter see Report Viewer in this manual.

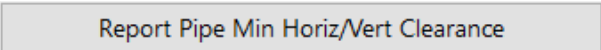



Data Problem Log dialog

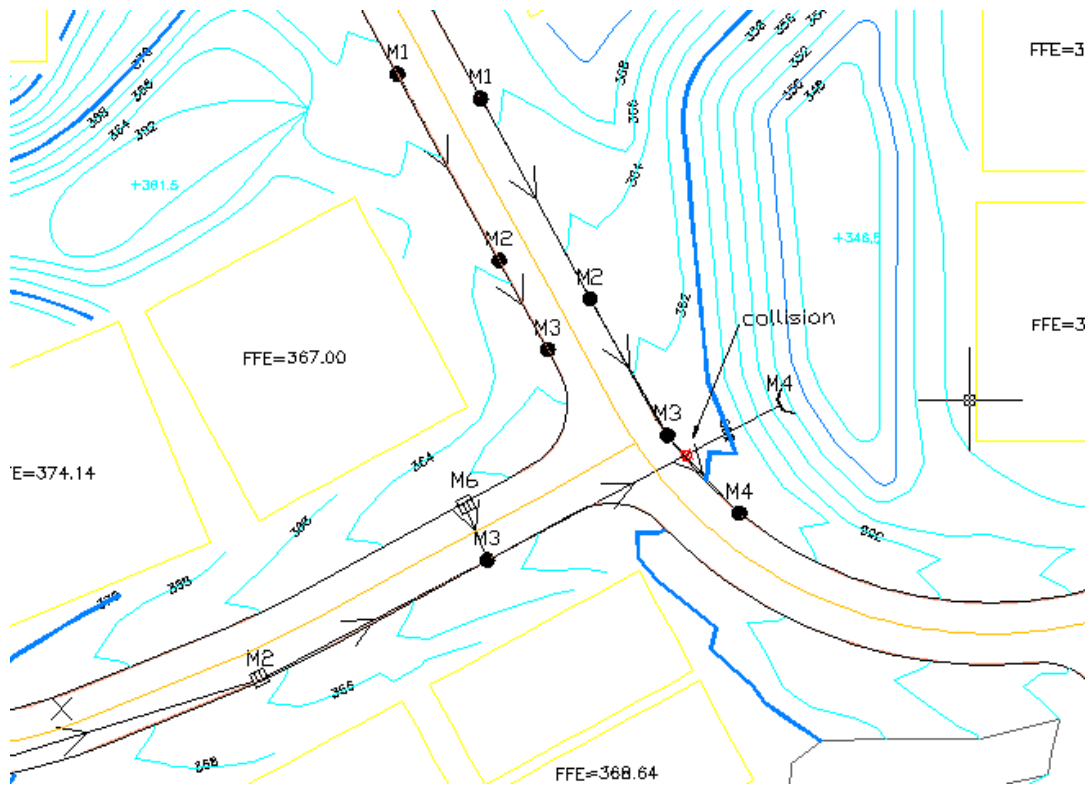
When the **Use Collision Navigator** toggle is on, an Data Problem Log is generated if there are any collisions. Clicking to the "+" sign beside the Collisions Total will display the individual collisions. When a collision item is selected, click on **Zoom To** button and the drawing is centered to the exact location of the highlighted collision. **Zoom In** button zooms in on the highlighted collision for a closer inspection. **Zoom Out** zooms out away from the collision. **Report All/One** button between One and All depending on whether a single collision or all collisions are selected from the Log. An error Report is generated listing the positions of the entities in conflict.



Collision Navigator

Report Pipe Min Horiz/Vert Clearance  button would report the Horizontal Separation and the Vertical Clearance of pipes in a formatted report.

Report Pipe/Structure Min Horizontal Clearance  button would report the pipe and structures horizontal separation in a formatted report.



Collision Example

Sewer Network Collision Report

System Name: A, File: C:\Carlson2008_08\Projects\millcreek\MillCreek.sew
 System Name: B, File: C:\Carlson2008_08\Projects\millcreek\MillCreek2.sew
 System Name: C, File: C:\Carlson2008_08\Projects\millcreek\MillCreek3.sew
 Conflict Tolerance: 1.00 ft

Collision 1:
 System A, Pipe from M4 to M3 and System B, Pipe from M4 to M3 collide at (1219.903, 927.383, 355.137).

Pipe Crossings:

#	Lower Pipe	Upper Pipe	Easting	Northing	Crown	Invert	Distance
1	B, from M4 to M3	A, from M4 to M3	1219.827	927.876	356.238	354.569	Collide

Collision Report Example

Pulldown Menu Location: Network > Check Sewer Network

Keyboard Command: chkmswr

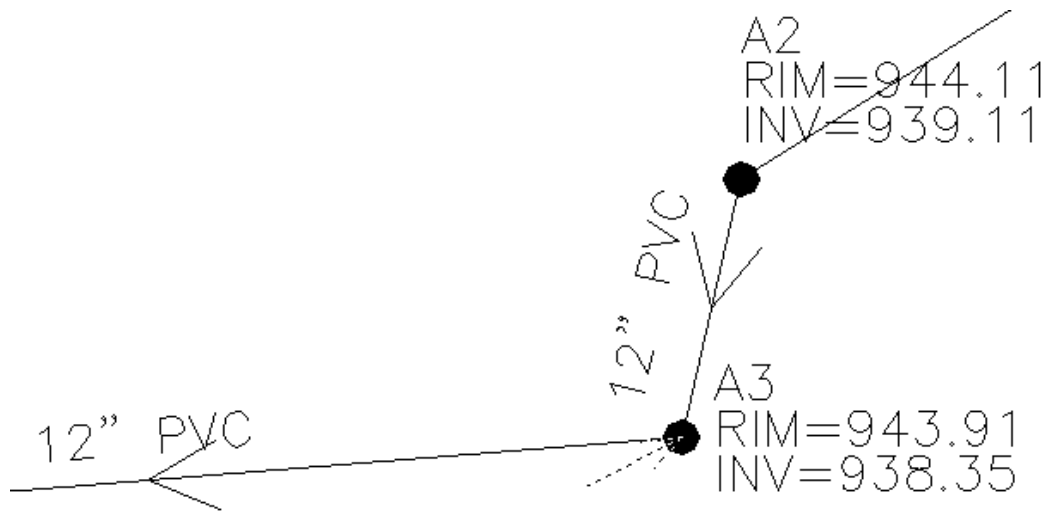
Prerequisite: two to four sewer files (.SEW)

Find Sewer Structure

This command will find a sewer structure by name.

Prompts

Structure Name to Find: A3



The program will then display a temporary arrow locating the structure, and zoom to it at the current zoom resolution.

Pulldown Menu Location: Network

Keyboard Command: findswr

Prerequisite: Sewer network (.SEW) file

Report Sewer Network

This command reads a sewer network file and reports its design parameters and hydraulic results. Select Report Sewer Network from the Network menu in the Hydrology Module, the report dialog displays. The **Select Sewer Line(s) To Report** list displays all upstream entrance structure names, which represents pipe lines from every entrance to the outfall. Select one entrance to report one pipe line. If you want to report the whole sewer network, turn on the Report All Sewer Lines toggle. Select one or multiple return periods for reporting. The **Settings** dialog allows the user to set decimal variables to Station, Elevation / Depth, Pipe Length, Pipe Slope, and Other for precision.

The sewer network report has four categories: **Inlet**, **Pipe**, **Structure** and **Drainage**, in order to report all information thoroughly.

Report Subareas lists subarea type, area, runoff Coefficient and increment Runoff Coefficient for every catchbasin.

Custom Report allows you to choose which portion of data you would like to report from the combination of drainage and pipe categories.

Report HGL Computation lists the result of the friction losses of every pipe and the junction losses of every structure.

Report Inlet Computation reports the drainage runoff and pavement parameters, and the result of the gutter and inlet calculation for every surface inlet.

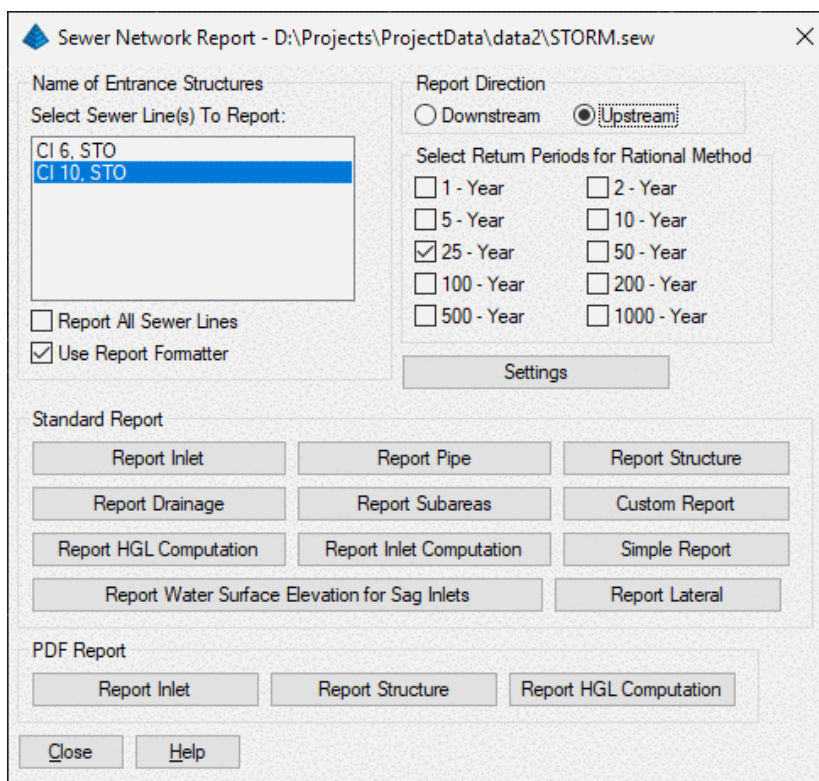
Simple Report lists some portion of information from all four categories.

Report Water Surface Elevation for Sag Inlets lists the orifice and weir calculation of the sag inlets.

Report Lateral lists the data for lateral connections.

Check the **Use Report Formatter** toggle to report the sewer network data in a Microsoft Excel spreadsheet or as a Table Entity in the plan; otherwise in the standard Carlson report window from where the information can be edited, printed to a printer or to the screen, and saved.

The **PDF report** allows you to report sewer data and results by various PDF report templates. Currently we support **Inlet**, **Structure** and **HGL Computation** PDF report based on PDF report templates come with the install.



Sewer Network Report Dialog

Structure Name	Subarea Type	Area (Acres)	Area (SF)	Area (%)	Runoff C	Increm. C
CI 5	Streets Drives and Walks	0.09	3921.75	50.66	0.95	0.48
CI 5	Lawns, Sandy Soil, Average, 2-7%	0.09	3818.90	49.34	0.35	0.17
CI 7	Lawns, Sandy Soil, Average, 2-7%	0.08	3399.37	59.39	0.15	0.09
CI 7	Roofs Inclined	0.02	750.00	13.10	1.00	0.13
CI 7	Streets Drives and Walks	0.04	1574.11	27.50	0.95	0.26
CI 8	Streets Drives and Walks	0.05	2097.05	6.64	0.95	0.06
CI 8	Roofs Inclined	0.02	995.63	3.15	1.00	0.03
CI 8	Lawns, Sandy Soil, Average, 2-7%	0.65	28492.04	90.21	0.15	0.14

Sewer Network Report Example: Subarea Report

Inlet Report - 5 YEAR

Line No	Inlet ID	Q = CIA (cfs)	Q carry (cfs)	Q capt (cfs)	Q Byp (cfs)	Junc Type	Curb Inlet		Grate Inlet			Gutter					Inlet			Byp Line No			
							Ht (in)	L (ft)	Area (sqft)	L (ft)	W (ft)	So (ft/ft)	W (ft)	Sw (ft/ft)	Sx (ft/ft)	n	Depth (ft)	Spread (ft)	Depth (ft)		Spread (ft)	Depr (in)	
1	1 (STORM)	0.00	0.00	0.00	0.00	Outfall						0.01			0.02	0.013							Off
2	2 (STORM)	1.80	0.00	1.14	0.67	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.02	0.02	0.013	0.17	8.33	0.17	8.33	0.0	Off	
3	3 (STORM)	1.80	0.00	1.14	0.67	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.02	0.02	0.013	0.17	8.33	0.17	8.33	0.0	Off	
4	4 (STORM)	1.80	0.00	1.14	0.67	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.02	0.02	0.013	0.17	8.33	0.17	8.33	0.0	Off	
5	5 (STORM)	1.80	0.00	1.14	0.67	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.02	0.02	0.013	0.17	8.33	0.17	8.33	0.0	Off	
6	6 (STORM)	0.30	0.00	0.22	0.08	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.01	0.01	0.013	0.07	6.56	0.07	6.56	0.0	Off	
7	7 (STORM)	1.80	0.00	1.14	0.67	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.02	0.02	0.013	0.17	8.33	0.17	8.33	0.0	Off	
8	8 (STORM)	1.80	0.00	1.14	0.67	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.02	0.02	0.013	0.17	8.33	0.17	8.33	0.0	Off	
9	25 (STORM)	1.80	0.00	1.14	0.67	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.02	0.02	0.013	0.17	8.33	0.17	8.33	0.0	Off	
10	51 (STORM)	1.80	0.00	1.14	0.67	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.02	0.02	0.013	0.17	8.33	0.17	8.33	0.0	Off	
11	52 (STORM)	0.24	0.00	0.22	0.02	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.02	0.02	0.013	0.08	3.92	0.08	3.92	0.0	Off	
12	54 (STORM)	1.14	0.00	0.84	0.30	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.02	3.00	0.02	0.02	0.013	0.13	5.99	0.13	5.99	0.0	Off	
13	55 (STORM)	0.00	0.00	0.00	0.00	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.02	0.02	0.013	0.00	0.00	0.00	0.00	0.0	Off	
14	56 (STORM)	0.93	0.00	0.93	0.00	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.04	3.00	0.50	0.50	0.013	0.33	0.67	0.33	0.67	0.0	Off	
15	57 (STORM)	1.53	0.00	0.85	0.69	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.01	0.01	0.013	0.15	12.28	0.15	12.28	0.0	Off	
16	58 (STORM)	1.29	0.00	0.74	0.55	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.01	0.01	0.013	0.14	11.52	0.14	11.52	0.0	Off	
17	59 (STORM)	0.03	0.00	0.03	0.00	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.01	0.01	0.013	0.03	2.76	0.03	2.76	0.0	Off	
18	60 (STORM)	1.44	0.00	1.34	0.10	Catch Basin	0.0	3.00	6.00	3.00	2.00	0.01	3.00	0.02	0.02	0.130	0.37	24.74	0.37	24.74	0.0	Off	
Project File: Rain1												Number of lines: 18					Run Date: 05/13/2020						

Structure Report

Struct No.	Structure ID	Junction Type	Rim Elev (ft)	Structure			Line Out			Line In			
				Shape	Length (ft)	Width (ft)	Size (in)	Shape	Invert (ft)	Size (in)	Shape	Invert (ft)	
1	1 (STORM)	Outfall	141.67	Headwall	0.00	0.00				54	Circular	136.25	
2	2 (STORM)	Catch Basin	159.13	Box	6.00	6.00	54	Circular	136.57	48	Circular	150.00	
3	3 (STORM)	Catch Basin	181.50	Box	6.00	6.00	48	Circular	160.36	48	Circular	161.85	
4	4 (STORM)	Catch Basin	177.95	Box	6.00	6.00	48	Circular	164.43	48	Circular	164.68	
5	5 (STORM)	Catch Basin	176.28	Box	6.00	6.00	48	Circular	167.29	48	Circular	167.90	
6	6 (STORM)	Catch Basin	187.06	Box	6.00	6.00	48	Circular	178.00	48	Circular	179.00	
7	7 (STORM)	Catch Basin	209.36	Box	6.00	6.00	48	Circular	192.32	48	Circular	199.28	
8	8 (STORM)	Catch Basin	230.66	Box	6.00	6.00	48	Circular	212.19	24	Circular	221.28	
										15	Circular	225.38	
										48	Circular	212.29	
9	25 (STORM)	Catch Basin	234.45	Box	6.00	6.00	48	Circular	216.05	15	Circular	229.24	
										36	Circular	217.03	
										42	Circular	216.30	
10	51 (STORM)	Catch Basin	235.00	Box	6.00	6.00	42	Circular	217.10	36	Circular	217.35	
										36	Circular	217.35	
11	52 (STORM)	Catch Basin	235.28	Circular	0.00	4.00	36	Circular	218.03	15	Circular	221.79	
										30	Circular	218.27	
12	54 (STORM)	Catch Basin	235.20	Circular	0.00	4.00	30	Circular	218.60	30	Circular	218.85	
13	55 (STORM)	Catch Basin	236.67	Circular	0.00	4.00	30	Circular	221.65	36	Circular	221.90	
14	56 (STORM)	Catch Basin	231.43	Circular	0.00	4.00	36	Circular	222.55	30	Circular	222.81	
15	57 (STORM)	Catch Basin	230.76	Circular	0.00	4.00	30	Circular	223.15	24	Circular	223.40	
Project File: Rain1							Number of Structures: 18				Run Date: 05/13/2020		

Structure Report

Struct No.	Structure ID	Junction Type	Rim Elev (ft)	Structure			Line Out			Line In		
				Shape	Length (ft)	Width (ft)	Size (in)	Shape	Invert (ft)	Size (in)	Shape	Invert (ft)
16	58 (STORM)	Catch Basin	230.60	Circular	0.00	4.00	24	Circular	224.23	18	Circular	224.39
17	59 (STORM)	Catch Basin	230.31	Circular	0.00	4.00	18	Circular	225.05	15	Circular	225.30
18	60 (STORM)	Catch Basin	230.54	Circular	0.00	4.00	15	Circular	225.65			

Project File: Rain1 Number of Structures: 18 Run Date: 05/13/2020

Hydraulic Grade Line Computations - 5 YEAR

Line	Size (in)	Q (cfs)	Downstream										Len (ft)	Upstream										Check Ave Sf (%)	Energy loss (ft)	JL coeff (K)	Minor loss (ft)
			Invert elev (ft)	HGL elev (ft)	Depth (ft)	Area (sqft)	Vel (ft/s)	Vel head (ft)	EGL elev (ft)	Sf (%)	Invert elev (ft)	HGL elev (ft)		Depth (ft)	Area (sqft)	Vel (ft/s)	Vel head (ft)	EGL elev (ft)	Sf (%)								
1	54	127.07	136.25	139.57	3.32	12.57	9.92	1.53	141.16	0.52	63.94	136.57	139.95	3.38	12.83	10.11	1.59	141.16	0.50	0.51	0.33	0.00	0.00				
2	48	126.75	150.00	151.59	1.59	4.66	27.20	11.50	163.09	0.00	148.03	160.36	161.95	1.59	4.66	27.20	11.50	163.09	0.00	6.96	10.30	0.00	5.68				
3	48	126.60	161.85	168.37	6.52	12.57	10.07	1.58	169.95	0.77	128.78	164.43	169.37	4.94	12.57	10.07	1.58	169.95	0.77	0.77	0.99	0.00	0.32				
4	48	126.45	164.68	170.42	5.74	12.57	10.06	1.57	171.99	0.77	130.13	167.29	171.42	4.13	12.57	10.06	1.57	171.99	0.77	0.77	1.00	0.00	0.51				
5	48	126.59	167.90	172.67	4.77	12.57	11.19	1.95	174.25	0.77	252.41	178.00	181.37	3.37	11.31	10.07	1.58	174.25	0.74	0.75	1.90	0.00	1.00				
6	48	78.79	179.00	182.68	3.68	12.09	8.77	1.20	183.34	0.26	88.81	192.32	195.01	2.69	8.98	6.51	0.66	183.34	0.48	0.37	0.33	0.00	0.00				
7	48	78.43	199.28	200.45	1.17	3.07	25.53	10.13	210.58	0.00	151.64	212.19	213.36	1.17	3.07	25.53	10.13	210.58	0.00	8.45	12.81	0.00	2.84				
8	48	74.49	212.29	216.46	4.17	12.57	8.57	1.14	217.01	0.27	626.80	216.05	218.66	2.61	8.69	5.93	0.55	217.01	0.46	0.36	2.28	0.00	0.71				
9	42	56.28	216.30	219.61	3.31	9.42	6.77	0.71	220.16	0.27	160.18	217.10	219.92	2.82	8.31	5.98	0.56	220.16	0.32	0.30	0.47	0.00	0.36				
10	36	25.98	217.35	220.36	3.01	7.07	4.16	0.27	220.57	0.15	136.00	218.03	220.51	2.48	6.25	3.68	0.21	220.57	0.15	0.15	0.20	0.00	0.17				
11	30	25.37	218.27	220.85	2.58	4.91	5.17	0.42	221.27	0.38	64.93	218.60	221.10	2.50	4.91	5.17	0.42	221.27	0.37	0.37	0.24	0.00	0.27				
12	30	24.81	218.85	221.52	2.67	4.91	6.99	0.76	221.92	0.36	172.48	221.65	223.35	1.70	3.55	5.05	0.40	221.92	0.56	0.46	0.80	0.00	0.64				
13	36	24.91	221.90	224.09	2.19	5.53	6.44	0.65	224.41	0.18	129.71	222.55	224.16	1.61	3.87	4.50	0.32	224.41	0.44	0.31	0.40	0.00	0.48				
14	30	18.73	222.81	224.75	1.94	4.09	5.36	0.45	225.08	0.23	68.92	223.15	224.83	1.68	3.50	4.58	0.33	225.08	0.33	0.28	0.19	0.00	0.38				
15	24	12.73	223.40	225.31	1.91	3.09	5.07	0.40	225.57	0.27	165.52	224.23	225.72	1.49	2.51	4.12	0.26	225.57	0.38	0.33	0.54	0.00	0.08				
16	18	6.71	224.39	225.89	1.50	1.77	4.32	0.29	226.11	0.41	113.08	225.05	226.28	1.23	1.55	3.80	0.22	226.11	0.40	0.40	0.46	0.00	0.06				
17	15	1.34	225.30	226.35	1.05	1.10	1.88	0.05	226.37	0.04	48.34	225.65	226.35	0.70	0.71	1.22	0.02	226.37	0.12	0.08	0.04	0.00	0.01				

Project File: Rain1 Number of lines: 17 Run Date: 05/13/2020

Notes:

Prompts

Report Sewer Network dialog: Fill in values.

Pulldown Menu Location: Network > Report Sewer Network

Keyboard Command: reportswr

Prerequisite: a sewer file (.SEW), a surface file (.TIN, .GRD)...\\USER\\RainLib.dta, ...\\USER\\inlet.dta, ...\\USER\\pipesize.dta (mpipesize.dta in Metric unit), ...\\USER\\swrStruct.dta

Sewer Network Quantities Report

This command reports pipe and structure quantities by depth range for the current sewer network file. In the options dialog, you can choose to report all the pipes or selected pipes.

Use **Report Formatter** is used to customize reports and output to various formats like Excel.

Under **Setup Depth Zones**, you can define the different depth ranges to report and assign costs for the structures and the pipe length per depth range.

Report Depth Zones reports the structures and pipe length for each depth range.

Report Stations Depth Summary reports the pipeline stations for the different depth ranges.

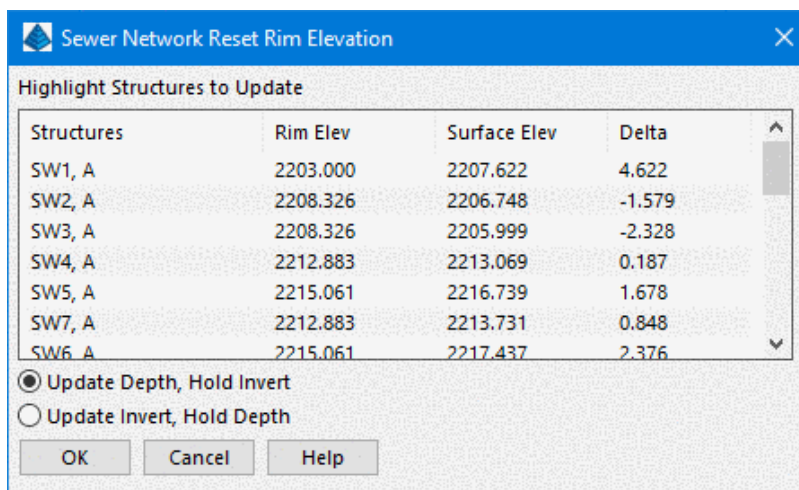
Report Manhole Depth Summary reports the depth of each structure and the sum of these depths.

Report Pipe Length By Size reports the total pipe length per pipe size.

The **Draw Plan View Zone Map** option draws the pipe lines in different colors by depth and has the option to label the zones along the pipes. The colors and line widths for the map are set in the Setup Depth Zones dialog.

Pipe Details report includes the length, depth, size, material and other parameters for every pipe.

Depth Details report includes the pipe parameters broken out by depth zone.



Pipe Network Quantities Report

Depth	Manholes	Linear(ft)	% of Total
0-5.0	4	338.9	60.2
5.0-10.0	1	223.8	39.8
>10.0	0	0.0	0.0
Total	5	562.7	

Stations	Depth	Starting Structure
0+00.0 to 3+25.3	0-5.0	M1
3+25.3 to 5+49.1	5.0-10.0	M1
5+49.1 to 5+62.7	0-5.0	M1

Pipe Size(in)	Linear(ft)	% of Total
9	288.2	51.2
12	274.5	48.8

Manhole	Depth
M5	3.99
M4	8.31
M3	4.73
M2	4.01
M1	4.00
Total:	25.04

Pulldown Menu Location: Network

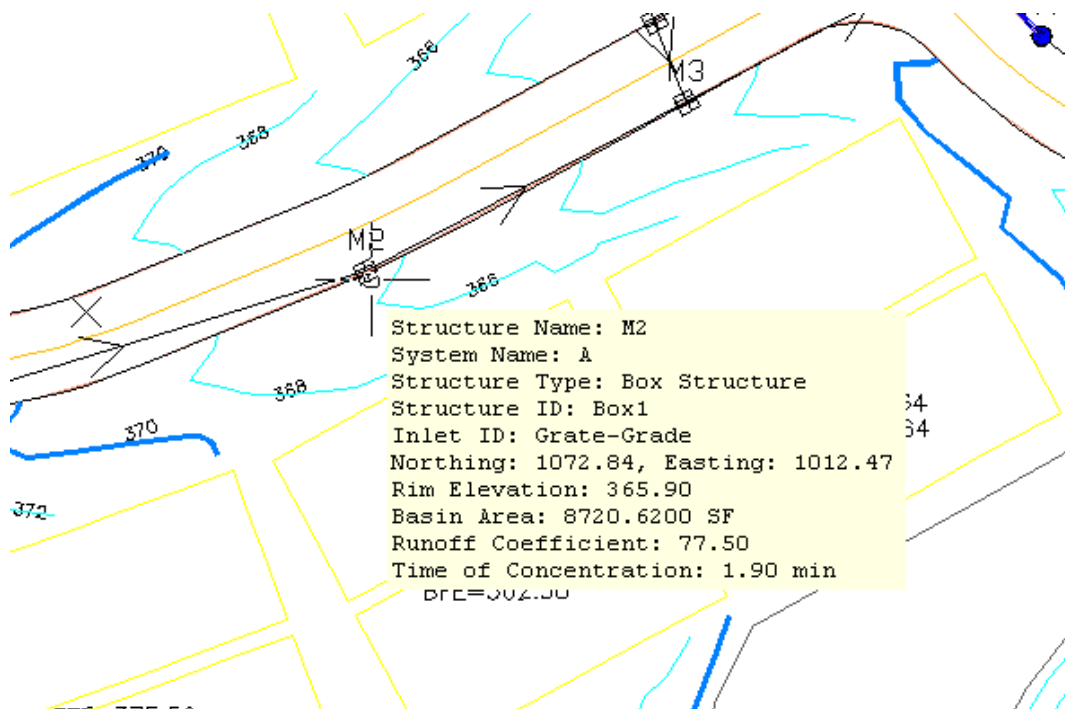
Keyboard Command: swrdepth

Prerequisite: .sew file

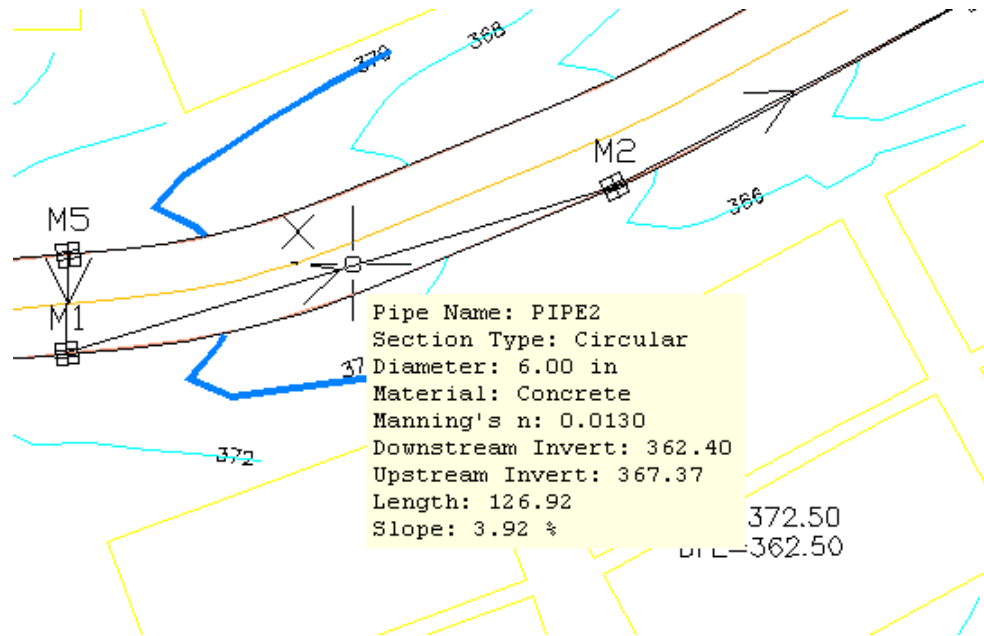
Sewer Network Inspector

The command allows you to inspect the active sewer network on screen.

Select Sewer Network Inspector from Network menu. Move your cursor to the sewer network in plan view. When the cursor hovers over a structure or a pipe, its parameters will be shown on screen. After finishing inspection, press Enter key to exit.



Sewer Network Inspector Example of Structure



Sewer Network Inspector Example of Pipe

Prompts

Pulldown Menu Location: Network > Sewer Network Inspector

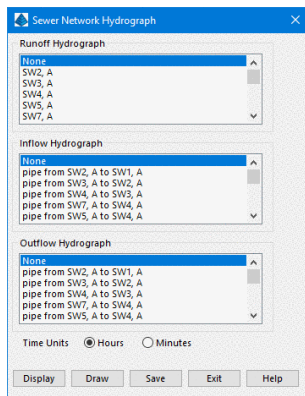
Keyboard Command: swrinspect

Prerequisite: active sewer network in plan view

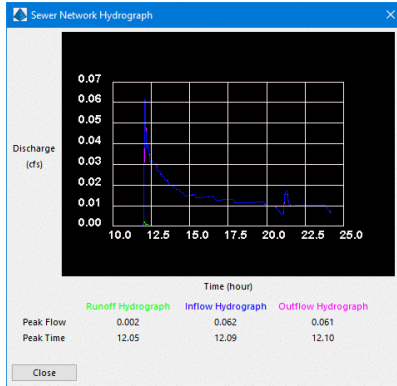
Sewer Network Hydrographs

This command generates the runoff hydrograph for every catch basin, inflow hydrograph for every pipe at upstream and outflow hydrograph for every pipe at downstream. If the sewer network uses Rational method to calculate the peak flow, the runoff hydrograph is calculated by Universal Rational method; otherwise, the runoff hydrograph is calculated by SCS method. The storage-indication method is used to route the inflow hydrograph through the whole network.

Select Sewer Network Hydrographs from the Network menu in the Hydrology Module. The command reads the current active sewer network file and conducts the hydraulic computation. If the computation is successful, a dialog opens with the lists of inflow and outflow hydrographs identified by the structures and pipes. Select a runoff, inflow and outflow hydrograph and click on Display button to show the hydrographs in the Hydrograph dialog.



Sewer Network Hydrograph by Carlson



Hydrograph Example by Carlson

Prompts

Sewer Network Hydrographs dialog: Fill in values.

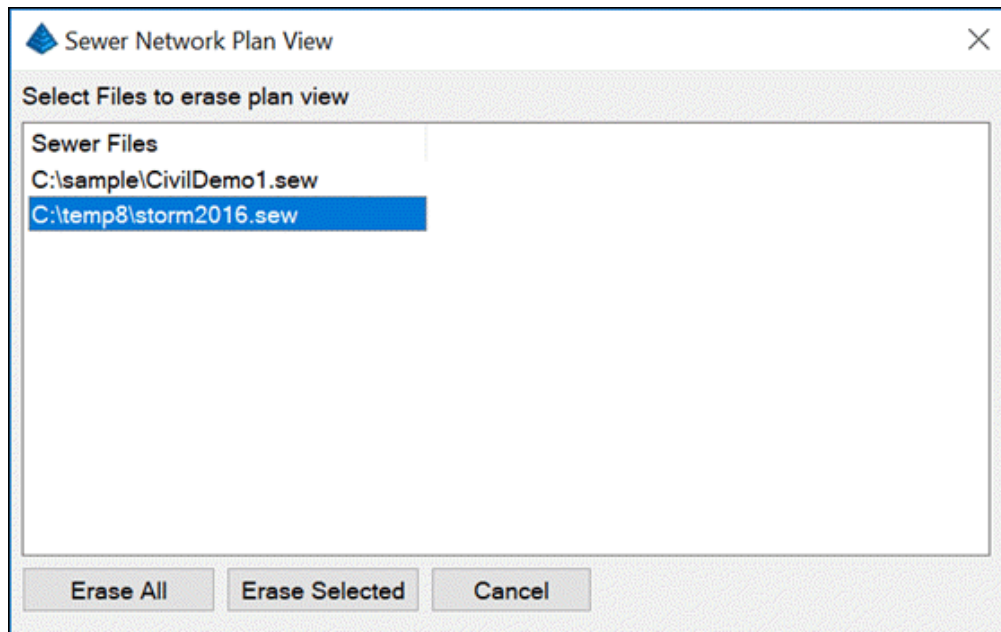
Pulldown Menu Location: Network > Sewer Network Hydrographs

Keyboard Command: swrhyd

Prerequisite: a sewer file (.SEW), a surface file (.TIN, .GRD)... \USER\RainLib.dta, ... \USER\inlet.dta, ... \USER\pipesize.dta (mpipesize.dta in Metric unit), ... \USER\swrStruct.dta, ... \USER\paven.dta, ... \USER\pipen.dta, ... \USER\runoff.dta

Erase Plan View

This command erases entities created by Draw Sewer Network > Plan View. This command looks in the current drawing for sewer network plan view entities and shows a list of all the sewer network models that are found. Select a sewer network model from the list and pick Erase Selected to erase its plan view entities. Or pick Erase All to erase all sewer network plan view entities. The sewer network model data is not erased.



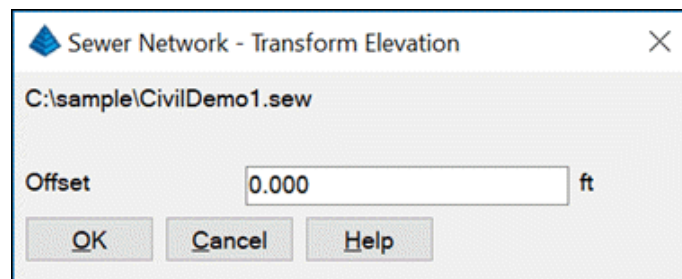
Pulldown Menu Location: Network > Draw Sewer Network

Keyboard Command: swrdel

Prerequisite: sewer network plan view entities

Transform Elevations

This command changes all the elevation in the current sewer network by the same amount. Both the rim elevations and invert elevations are changed. This command applies when a site design has been adjusted vertically by the same amount and you want to update the sewer network to match.



Pulldown Menu Location: Network->Sewer Network Utilities

Keyboard Command: set_swr_elev

Prerequisite: sewer network structures

Identify Sewer Network Entity

This command prompts to select an entity in the drawing and reports the sewer network model that the entity belongs to.

Prompts

Select sewer network entity: *pick an entity*

Network: C:\sample\CivilDemo1.sew

Select sewer network entity ([Enter] to End): *press Enter*

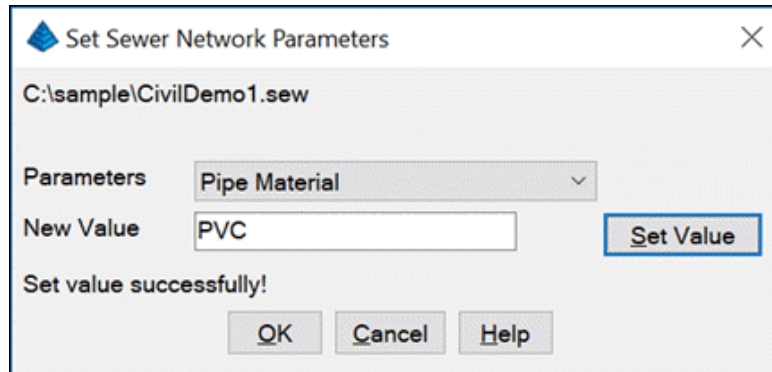
Pulldown Menu Location: Network > Sewer Network Utilities

Keyboard Command: id_swrnet

Prerequisite: sewer network entities

Set Parameter For Entire Network

This command sets a parameter to the same value for all elements in the current sewer network. In the dialog, choose the parameter from the list and fill in the value in the edit box. Pick the Set Value button to apply this value to the sewer network.



Pulldown Menu Location: Network->Sewer Network Utilities

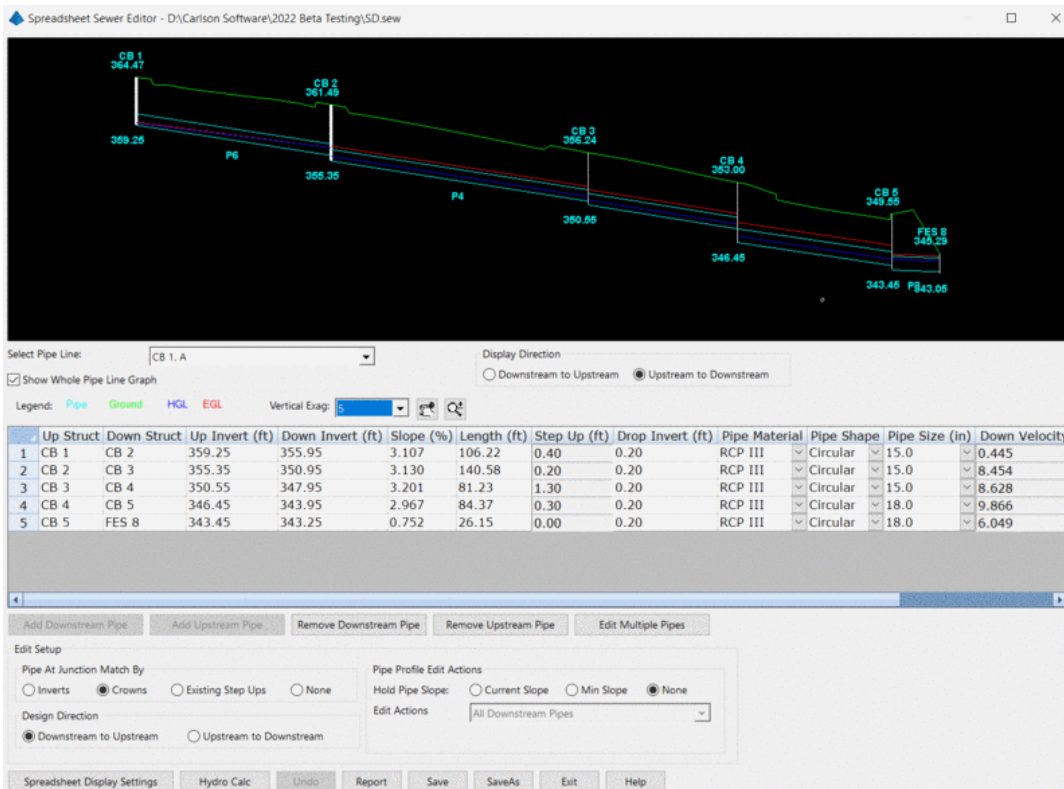
Keyboard Command: set_swr_value

Prerequisite: sewer network structures

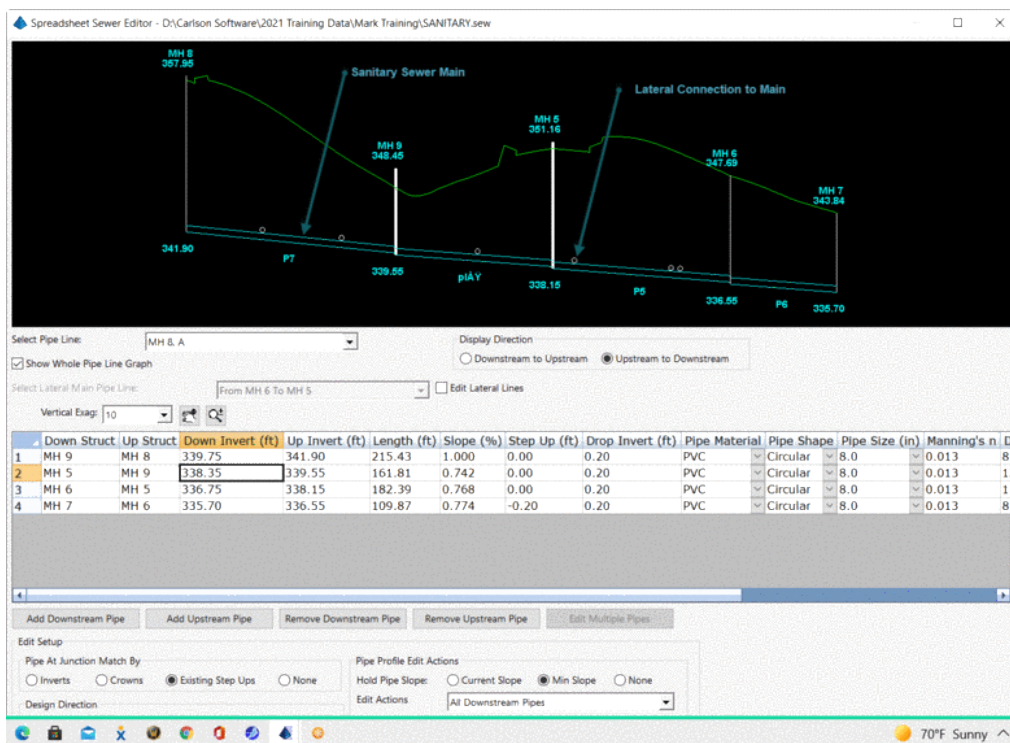
Spreadsheet Sewer Editor

This command is an alternative method to the Edit Sewer Structure command for editing existing sewer networks in a tabular format. Data changes can be applied more quickly in the spreadsheet editor and the content and position of displayed data is customizable.

Storm Drainage Sewer Editor



Sanitary Sewer Editor with Laterals



Show Lateral Lines: Select Lateral Main Pipe Line

Select Pipe Line: Select the sewer reach that will be displayed in the dialog box or enable the **Select All Pipe Lines** toggle to display the entire sewer network.

Show Whole Pipe Line: When enabled, the entire sewer network reach of the selected pipe is displayed.

Display Direction: To Display the network in either downstream to upstream or upstream to downstream direction, in order to match the **Design Direction**.

Design Direction: When downstream to upstream is selected, change a pipe's downstream invert elevation, its upstream invert will be updated automatically by pipe slope; otherwise change a pipe's upstream invert to update its downstream invert by pipe slope.

Vertical Exag: Indicate the desired amount of vertical exaggeration to be applied to the selected pipe display.

Edit Multiple Pipes: To globally change the value within a given column to a specific value, "click and drag" or use standard Windows **shift+click** functionality to select a range of cells within a given column and then click *Edit Multiple Pipes*.

Pipe at Junction Match By: Indicate the method by which pipes are vertically aligned at each junction:

- **Inverts:** Sets the pipe inverts at the same elevation at a given junction.
- **Crowns:** Sets the pipe crowns at the same elevation at a given junction.
- **Existing Setup:** The program will hold step-up values when edits are made.

- **None:** No vertical adjustment is performed.

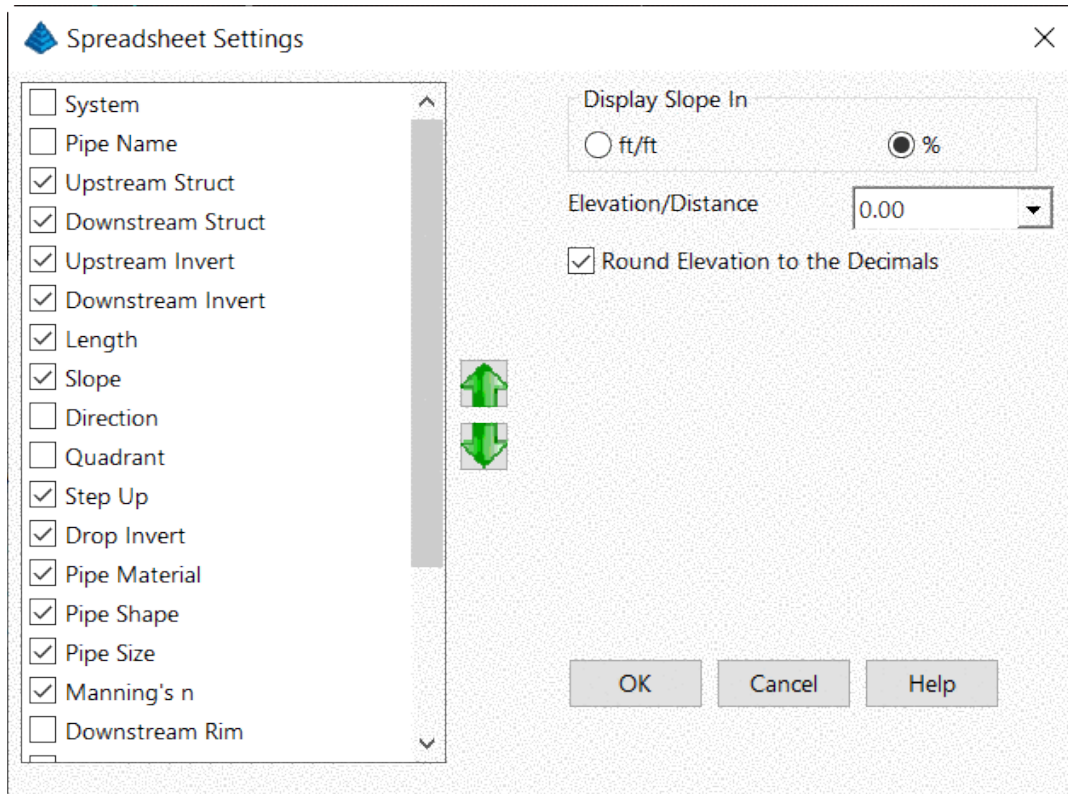
Pipe Profile Edit Actions: Indicate what should happen when *Hydro Calc* is clicked:

Hold Pipe Slope: Indicate how to manipulate the slope of the identified pipe(s):

- **Current Slope:** The slope of the current pipe is held.
- **Min Slope:** The minimum pipe slope is held.
- **None:** No vertical adjustment is performed.

Edit Actions: Set the desired method of how the sewer network should be adjusted when the *Hydro Calc* button is clicked.

Spreadsheet Display Settings: Enable the options for the value(s) you would like displayed in the Sewer Spreadsheet Editor and position each them using the Move Up and Move Down buttons.

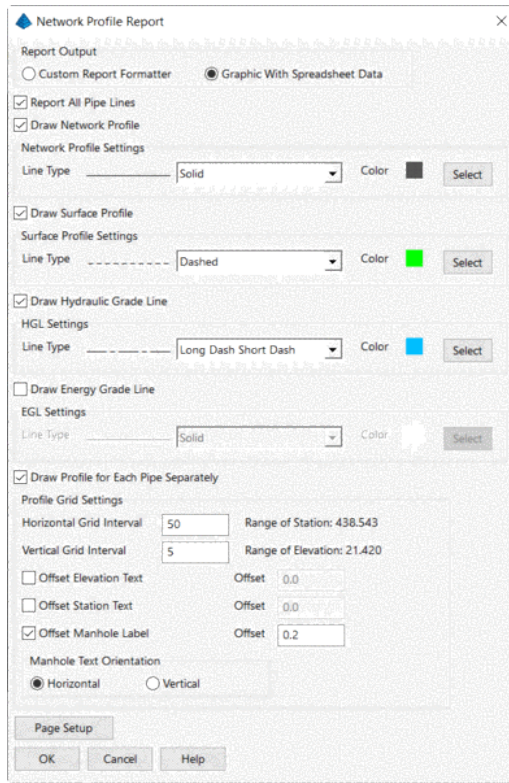


Display Slope In: Indicate the unit of measure to be used for the display of the pipe slope.

Elevation/Distance: Decimal Place Value for rounding, use drop down arrow for settings Round Elevation to the Decimal place.

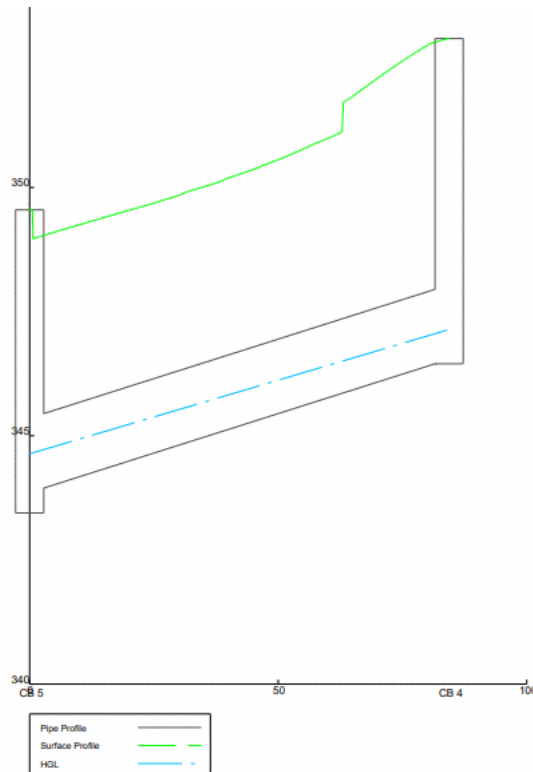
Hydro Calc: The hydraulic calculations are performed on the network and the results are displayed.

Report: This option allows data or graphical reports of the network to be generated.



Report Output: Indicate the desired type of report output:

- **Custom Report Formatter:** Produces a user-customized report format using the standard Carlson Report Formatter functionality.
- **Graphic with Spreadsheet Data:** Produces a PDF-based report of the sewer network (see an example below) along with data results in a tabular format.
- **Report All Pipe Line:** If checked will include all pipe runs in the network report. If unchecked on the main dialog box what is displayed will be reported by using the Select Pipe Line.
- **Draw Network Profiles:** Will include the following image in the report.



Draw Profile for Each Pipe Separately: Draws each pipe run on its own page in the report.

Draw Network Profile, Draw Surface Profile, Draw Hydraulic Grade Line, Draw Energy Grade Line: When enabled, the appropriate item is included in the graphical portion of the report and allows the **Linetype** and **Color** to be specified for each, respectively.

Draw Profile for Each Pipe Separately: When enabled, each pipe is detailed into the report as illustrated above. When disabled, each reach of the sewer network is drawn into the report.

Profile Grid Settings: Set the desired Grid Setting to be used in report.

Manhole Text Orientation: Can be toggled between Horizontal or Vertical for display in the report

Page Setup: Allows the look and size of the eventual PDF report to be customized using the standard Carlson PDF Report Generator.

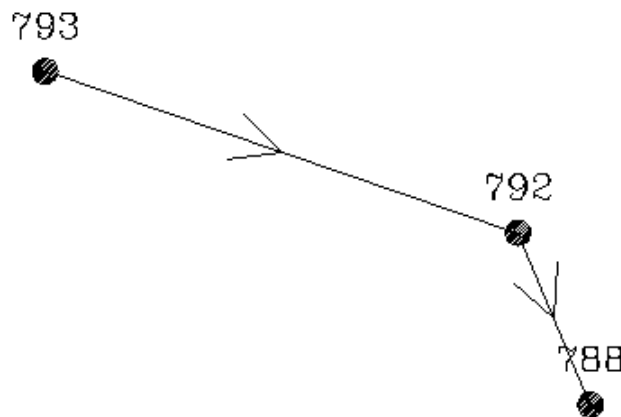
Pulldown Menu Location: Network

Keyboard Command: sizeswr

Prerequisite: A sewer (.SEW) file, a surface (.TIN, .GRD, .FLT) file

Draw Sewer Network Plan View

This command draws and labels the manhole symbols and pipe connections for the current sewer network (.SEW) file. An arrow can be drawn on the connections to indicate the direction of flow. The format for the labels is defined in the Plan View Label Settings command.



Example drawn sewer network

Pulldown Menu Location: Network

Keyboard Command: drawswr

Prerequisite: Sewer network (.SEW) file

Draw Sewer Network Data Table

This command draws the sewer network structure and pipe data tables on screen.

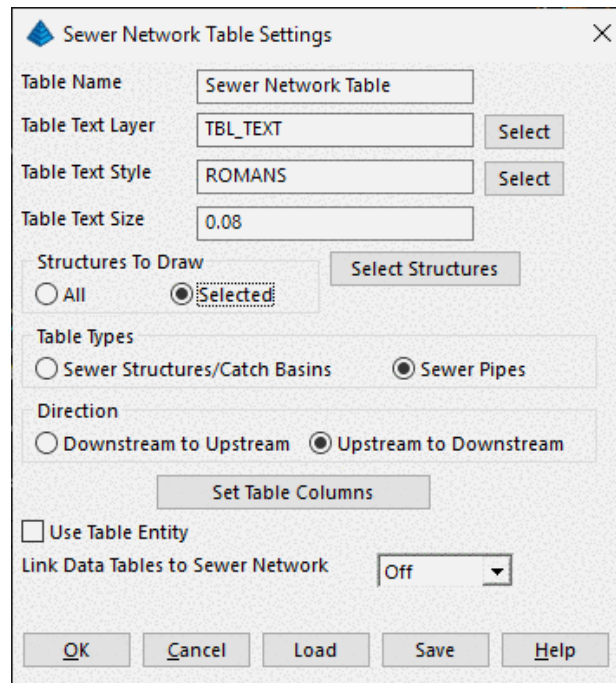
Select Data Table from Network > Draw Sewer Network menu. In the Table Settings dialog, specify the table name that will be shown as the table title, the layer name, the text style and the text size. Choose the table

type, either the sewer structures or sewer pipes, and also the direction how you want to display the sewer network: downstream to upstream or upstream to downstream. Click on Set Table Columns button to open a spreadsheet dialog, from where you can specify how you want to display the data table: which fields to display and in what order, the label, width, alignment and precision of each field. Choose Use Table Entity option to draw the data tables as the table entities, otherwise the tables are drawn as the traditional table block. After entering all parameters, click on OK button to draw the data table.

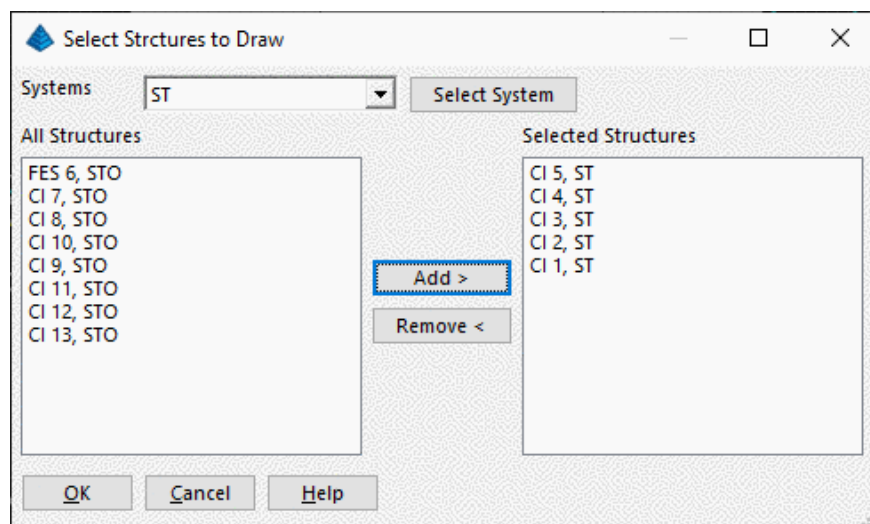
Structure to Draw if toggled to All, then all pipes will be in the data table. **Selection** toggle allows the user to select specific data to be included in the data table.

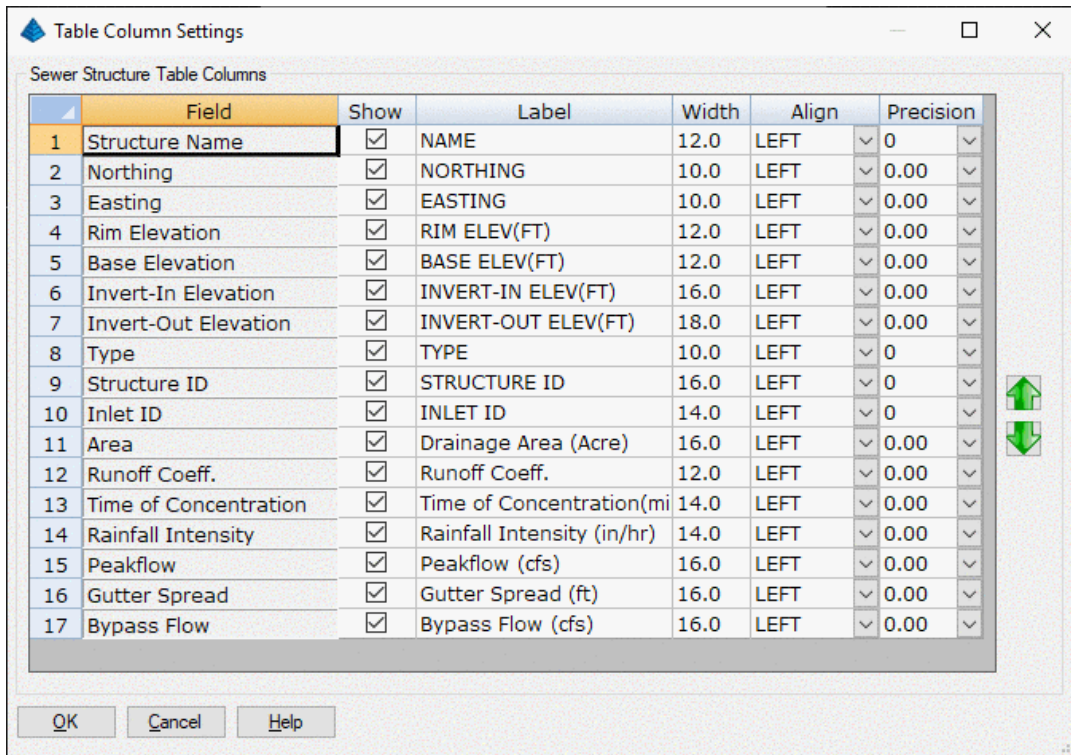
In the **Sewer Structure/Catch Basins** table, multiple invert-in elevations are reported.

When select the option **Link Data Tables to Sewer Network**, the data tables are redrawn automatically when the sewer network is revised.

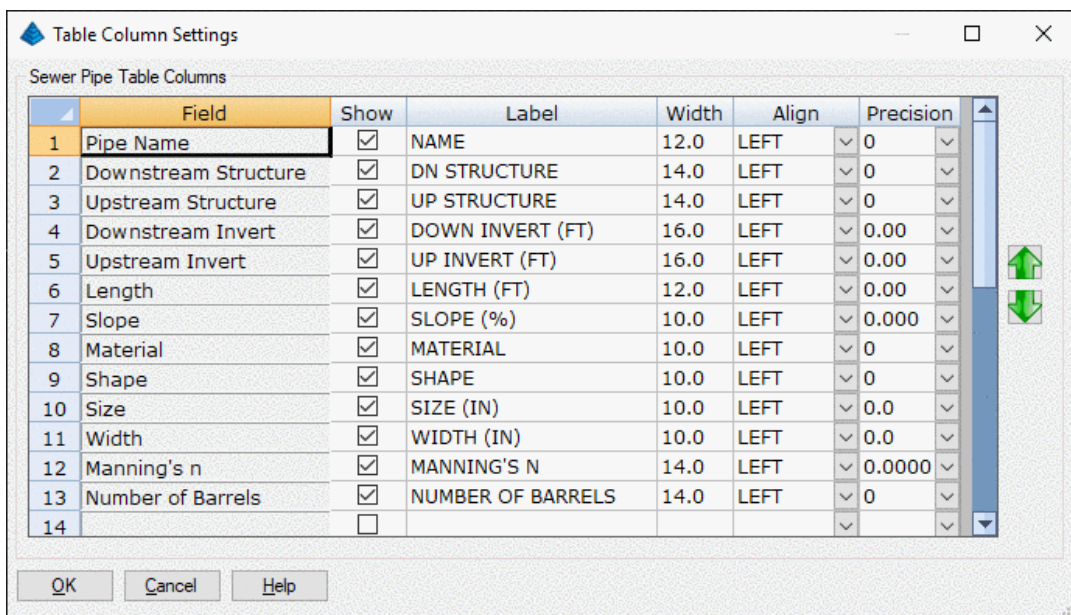


Sewer Network Table Settings Dialog





Column Settings Dialog for Sewer Structure Tables



Column Settings Dialog for Sewer Pipe Tables

Sewer Network Table									
NAME	EASTING	NORTHING	RIM ELEV(FT)	BASE ELEV(FT)	INVERT-IN ELEV(FT)	INVERT-OUT ELEV(FT)	TYPE	STRUCTURE ID	INLET ID
M4	914.96	1242.30	357.01	353.26	353.26	N/A	Outfall	Outfall-Wall1	
M3	967.29	1151.24	362.31	358.56	358.56	358.56	Box	Box1	Grate-Grade
M6	984.86	1164.17	362.21	358.70	N/A	358.70	Box	Box1	Grate-Grade
M2	1012.47	1072.84	365.90	362.15	362.40	362.15	Box	Box1	Grate-Grade
M1	1095.63	976.95	370.87	367.37	367.37	367.37	Box	Box1	Grate-Grade
M5	1106.89	995.57	370.70	367.53	N/A	367.53	Box	Box1	Grate-Grade

Sewer Structure Data Table Example

Sewer Network Table											
NAME	DN STRUCTURE	UP STRUCTURE	DOWN INVERT (FT)	UP INVERT (FT)	LENGTH (FT)	SLOPE (%)	MATERIAL	SHAPE	SIZE (IN)	WIDTH (IN)	MANNING'S N
PIPE5	M4	M3	353.26	358.56	105.03	5.044	Concrete	Circular	8.0	0.0	0.0130
PIPE4	M3	M6	358.56	358.70	21.81	0.624	Concrete	Circular	8.0	0.0	0.0130
PIPE3	M3	M2	358.56	362.15	90.48	3.971	Concrete	Circular	8.0	0.0	0.0130
PIPE2	M2	M1	362.40	367.37	128.92	3.917	Concrete	Circular	6.0	0.0	0.0130
PIPE1	M1	M5	367.37	367.53	21.76	0.728	Concrete	Circular	6.0	0.0	0.0130

Sewer Pipe Data Table Example

Prompts

Sewer Network Table Settings dialog: Fill in values

Pulldown Menu Location: Network > Draw Sewer Network > Data Table

Keyboard Command: swrtbl

Prerequisite: a sewer network file

Draw Sewer Network Centerlines

This command creates a centerline that connects each pipeline between the selected structures. The centerline can be drawn as a polyline or saved to a centerline (.cl) file. The direction of the centerline can go either upstream or downstream.

Account for Structures Connections Point for Box Structures: When this checkbox is unchecked, The centerline is from structure center to center; when the box checked on, centerline is from pipe point at the wall to structure center, and then to next pipe point at the wall.

Pulldown Menu Location: Network

Keyboard Command: drwswrcl

Prerequisite: Sewer network (.SEW) file

Draw Sewer Network Profile

This command will read the active sewer network (.SEW) file, which contains invert elevations, rim elevations, pipe sizes and structure dimensions, and draw the network as a profile, using the standard prompting in Draw Profile. In the options dialog, you can select the structure names for the start and end of the profile and set the profile direction as either upstream or downstream. The **Save To Profile File** option will save the sewer profile to a .pro file in addition to drawing the profile. When the Save option is active, there is an option available to **Link Profiles To**

Sewer Network. This Link option will update the sewer profile drawing whenever the sewer network is modified. Also, this link option will update the sewer network if the profile is changed. For example, if Profile Editor is used to change an invert elevation and the link option is active, then the invert elevation will also be updated in the sewer network. If you have a road and want to use it to reference the sewer network, you can choose the **Station by Another Reference Centerline** to do that. When not using a reference centerline, the **Starting Station** value is used for the beginning station of the sewer profile.

Draw Tributary Pipe Connections draws ellipses at the profile structures for any branch pipes that connect to the structure.

Draw Pipe Lateral Connections draws ellipses at the profile structures or pipes for lateral pipe connections.

Label Flow Along Pipes option pulls out the total flow that runs through each pipe and labels it on the profile.

Draw Hydraulic and **Energy Grade Line** options record the hydraulic flow calculation result and draw hydraulic and energy grade lines with network profile. You may choose to draw HGL/EGL for multiple storm events.

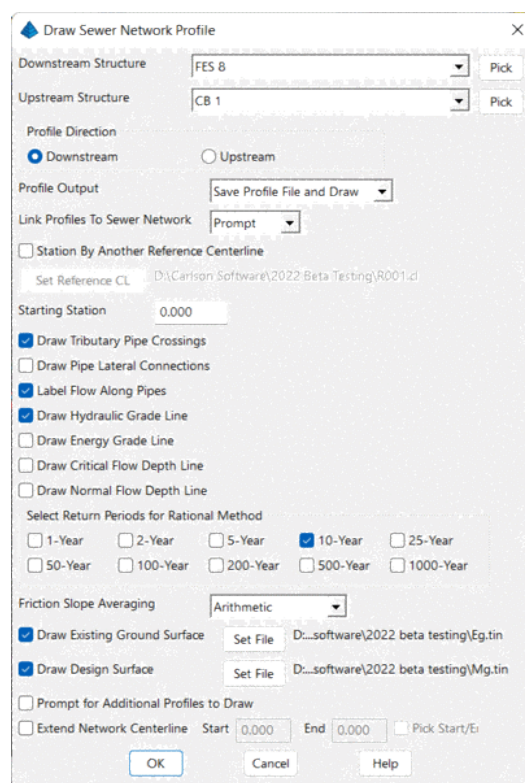
Draw Critical Flow Depth Line: Critical depth is defined as the depth of flow where energy is at a minimum for a particular discharge.

Draw Normal Flow Depth Line: Normal depth is the depth of flow that would occur if the flow was uniform and steady, and is usually predicted using the Manning's Equation.

Draw Existing Ground Surface and **Draw Design Surface** options would extract a profile from the surface to draw with the network profile. The **Set** button sets the surface files and the program stores the file names in the settings for redraw. If the files are not set on the dialog, the program would prompt for a surface file.

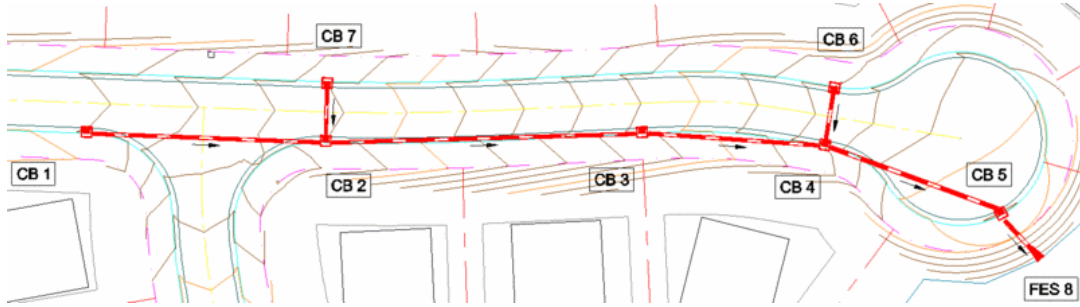
Prompt for Additional Profiles to Draw allows you to draw other profiles, such as a road profile, along with the network as a reference.

Extend Network Centerline lengthens the sewer network centerline by the specified amounts for extracting the surface profile for the existing and design surfaces. The **Pick Start/End** option allows users to pick start/end points that are not on the extension of sewer pipe lines.



Draw Sewer Network Profile Dialog

Consider the Storm Drainage Line shown in the plan view below:



When this network main storm drainage line is entered using Locate Sewer Structure, starting at the upstream rim elevation of 364.47 and running downhill to 1328.54, a new .SEW file is created. Prompting asks you to select a starting structure. If you created 8 structures named CB 1 through CB 5, you could choose CB 1 to plot all 5. This file will then plot, in profile view, as shown below (this example was drawn with ground surface and hydraulic grade lines). If you pick Draw Existing Ground Surface, you will be prompted for the grid or triangulation file for the ground surface, and similarly if you turn on Draw Design Surface.



Profile view of Sewer Network

Prompts

Draw Sewer Network Profile dialog: Fill in values

Pulldown Menu Location: Network->Draw Sewer Network

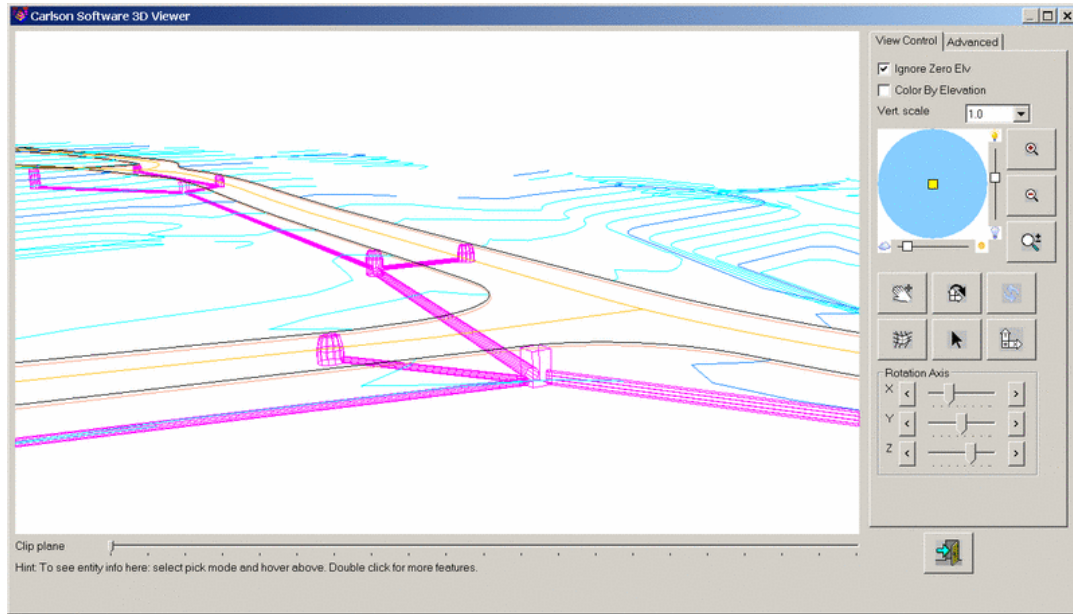
Keyboard Command: drswrpro

Prerequisite: Sewer network (.SEW) file

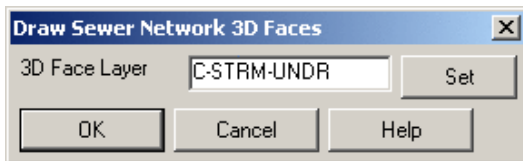
Draw Sewer Network-3DFaces

The command will read the active sewer file as set through the Set Sewer File command and draws the sewer pipelines as 3D faces for viewing in 3D. A primary use for this routine is to perform visual checks to help verify if

there are sufficient clearances with other 3D objects.



The 3D faces are drawn directly on the plan view in a layer of your choosing. The structures are drawn using the dimensions as defined in the Sewer Structure Library.



3D Face Layer: Key-in or use the **Select** button to identify the target layer for the 3D Face entities.

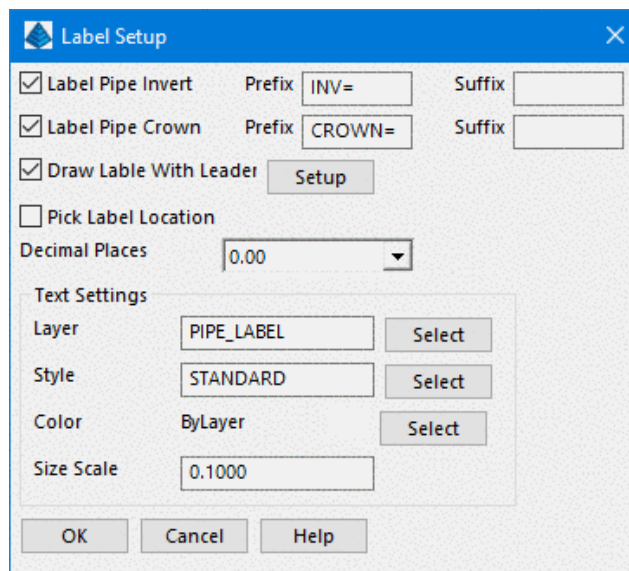
Pulldown Menu Location: Network > Draw Sewer Network

Keyboard Command: swr3d

Prerequisite: Sewer network (.SEW) file

Move Sewer Label

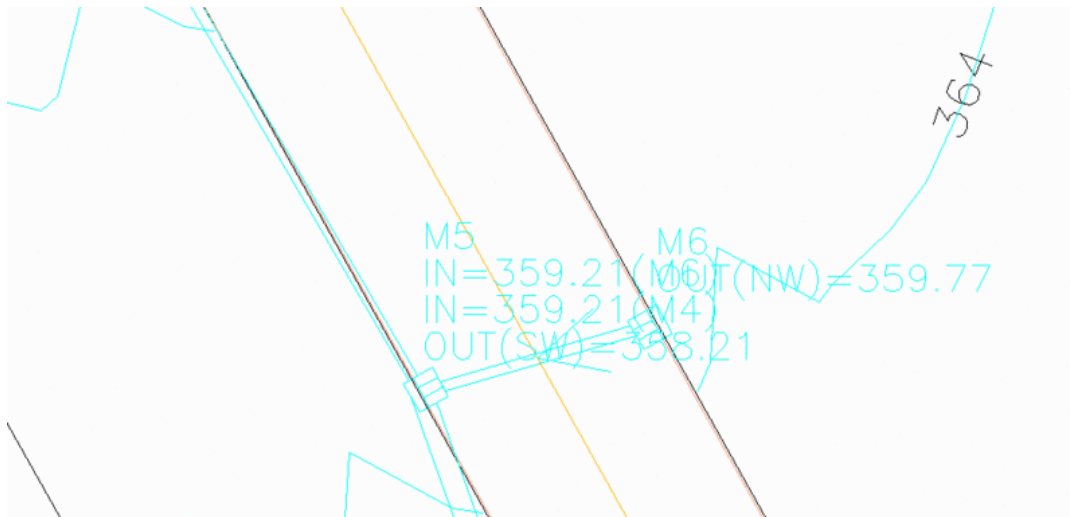
This command moves the selected plan view sewer network labels with the option to draw a leader from the labels to the sewer network reference location. Both structure and pipe labels can be moved. The purpose is to clean up label overlaps. To move a label, pick any one of the text labels and the program will pick up all the other associated labels. Then pick the new location and while the pointer is moved, the program shows an outline of the label area. The program remembers the moved locations for each label so that when the plan view labels are redrawn, the moved locations are retained.



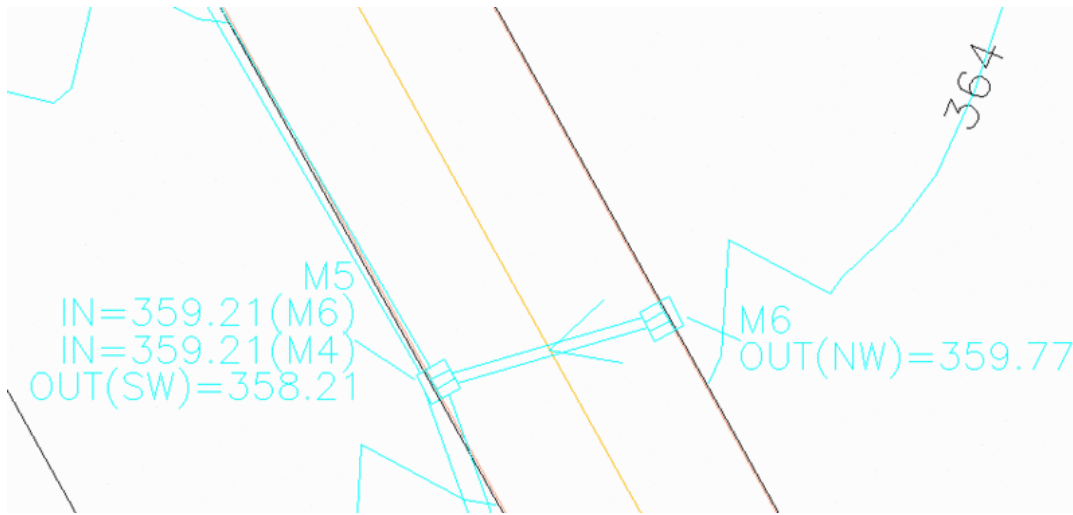
At the command prompt, there are keyboard options to show command Options, change the Angle and to Restore. The Options function brings up a dialog for command options for when to draw the label leader and whether to include an arrow head. The Minimum Leader Length Scaler value is multiplied by the current Horizontal Scale defined in Drawing Setup to set the minimum distance from the label to the structure for a leader to be drawn. There are separate leader scalers for structure and pipe labels. The Place Label Above or Below the Structure option allows for creating a leader that is vertical from the structure and using the Text Justify mode. Otherwise, the leader always puts the label to the left or right of the structure.

The Angle command line function allows you to rotate the label. The Restore function puts the labels back to their default position.

For pipe labels, there is an extra option at the command line when picking the new location to make the label horizontal. Otherwise the pipe label is aligned with the pipe segment.



These graphics show the sewer labels before and after Move Sewer Label was used to clean up the label overlaps.



Prompts

Select sewer label to move [Options/Angle/Restore]: *Pick a sewer label text entity*

Pick label position: *pick a point*

Select sewer label to move or Enter to end [Options/Angle/Restore]: *press Enter to end*

Pulldown Menu Location: Network

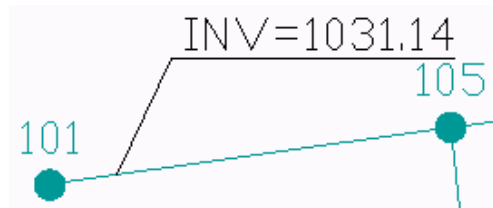
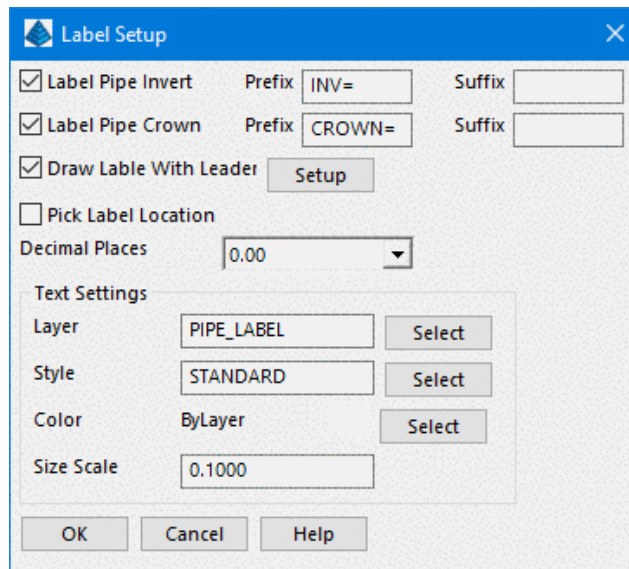
Keyboard Command: move_swr_label

Prerequisite: Plan view sewer network labels

Pipe Elevation Label

The command labels pipe elevations in plan view at specified points. In the options dialog, there are options whether to label the pipe invert elevation and pipe crown elevation. Also, there are options for the label prefix, suffix, layer, size, style, color, decimal places and whether to draw a leader from the label to the pipe. The Pick Label Location option allows for manually picking the location for the label. Otherwise, the label is automatically placed next to the pipe position. After clicking OK from the options dialog, the program prompts for points to label. For each point, the program finds the closest perpendicular point along a pipe in the current sewer network and labels that pipe position.

The elevation labels are for the inside pipe elevations without using the pipe wall thickness. A reason for labeling the pipe elevations is for checking for collisions.



Prompts

Pick pipe location (Enter to end): *pick a point*

Pick pipe location (Enter to end): *press Enter*

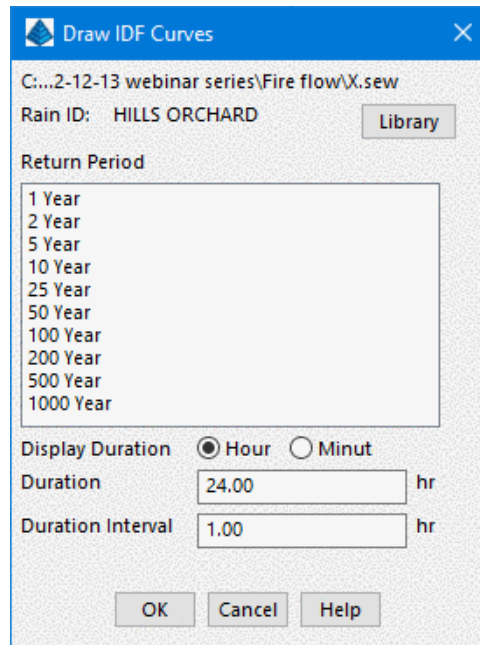
Pulldown Menu Location: Network > Sewer Labels

Keyboard Command: swrlabpipe

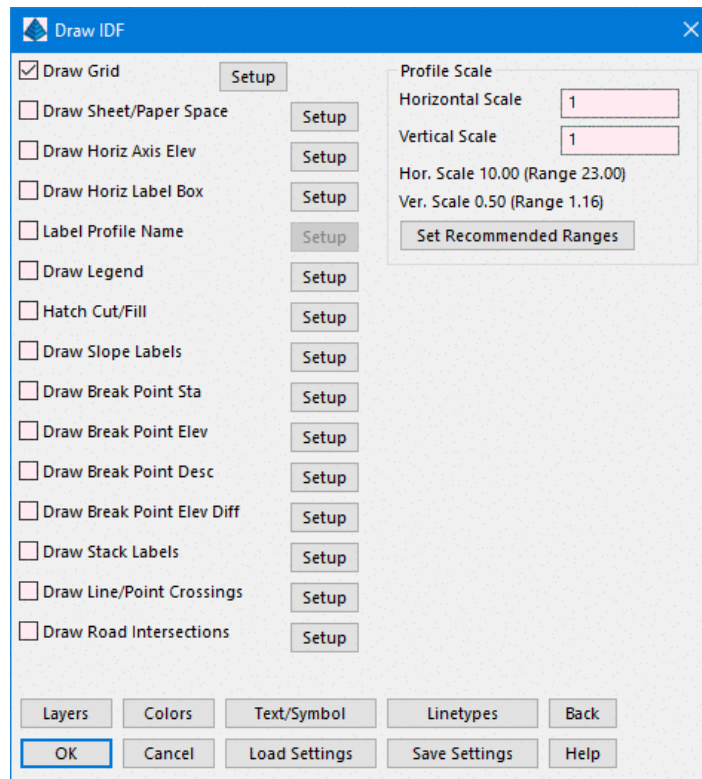
Prerequisite: sewer network in plan view

Draw IDF Curve

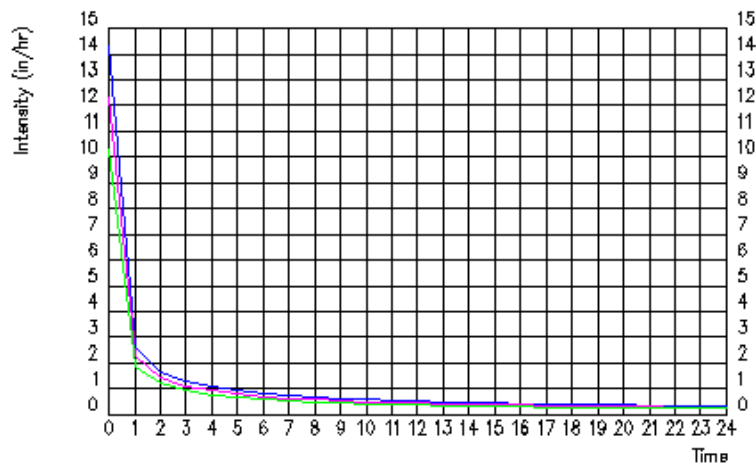
This command plots the Intensity-Duration-Frequency curves for the rainfall associated with the current sewer network. From the Network menu in the Hydrology Module, choose Draw IDF Curve. On the top of the dialog, the sewer network file name and the rainfall ID are shown. The Library button allows you to specify other rainfall data from the rainfall library. In the Return Period list, select one or more return periods. Select the Display Duration in either Hour or Minute, and enter the values in the Duration and Duration Interval boxes. Click on OK button to plot. The Draw IDF Settings dialog allows you to specify how to plot IDF curves on screen.



Draw IDF Curves



Draw IDF Settings



IDF Curve Example

Prompts

Draw IDF Curve dialog: Fill in values.

Pull-down Menu Location: Network > Draw IDF Curve

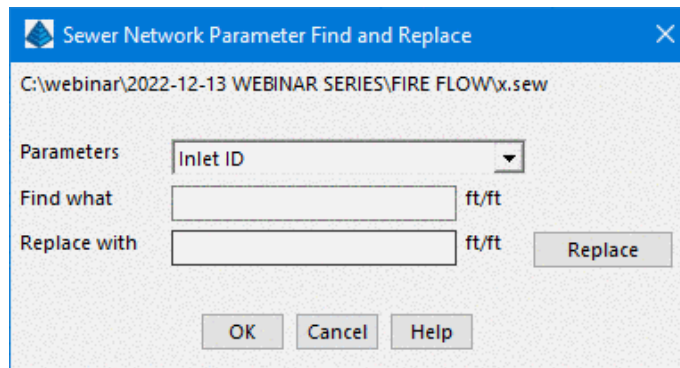
Keyboard Command: drwidf

Prerequisite: a sewer file (.SEW), ... \USER \RainLib.dta

Find And Replace Data Values

The Edit Sewer Structure command allows you to modify the parameters of a sewer network. However, it can be tedious when the network is large and when you need to change just one parameter of most of the sewer structures or pipe lines. For example, you'll have to go through all pipe lines if you want to change pipe manning's n to 0.014 for all pipes whose manning's n is currently 0.013. Here, the Find and Replace Data Values command would help you to find and replace the pipe manning's n values for all pipes at once.

From the Network > Sewer Network Utilities menu in the Hydrology Module, select Find and Replace Data Values to open the dialog. The sewer file name is displayed on the top of the dialog. In the Parameters list, choose what parameter you want to replace. In the Find what Box, type the value that you want to change, and in the Replace with box type the new value. Click on Replace button to find all pipes with manning's n values of 0.013 and replace their manning's n with 0.014. A message will be displayed on the dialog showing how many values have been replaced. Click on OK button to commit the changes or Cancel button to abandon the changes.



Find and Replace Sewer Network Parameters

Prompts

Find and Replace Data Values dialog: Fill in values.

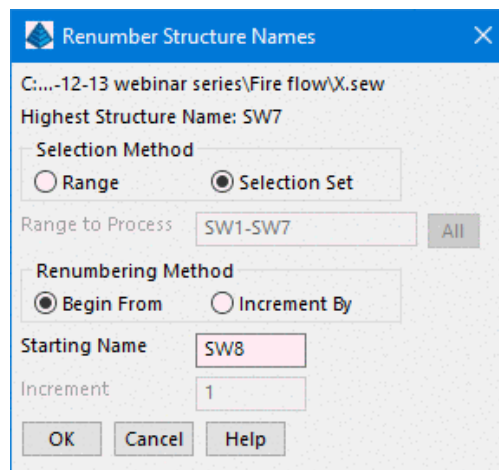
Pulldown Menu Location: Network > Sewer Network Utilities > Find and Replace Data Values

Keyboard Command:swr_find_replace

Prerequisite: a sewer file (.SEW)

Renumber Structure Names

The Renumber Structure Names routine provides an easy method to quickly rename/renumber all or part of a sewer network. Once a sewer file has been specified or set as Active via the Set Sewer File command, the following dialog box is presented.



Sewer Network File: The active sewer file that will be acted upon.

Highest Structure Name: The ID of the structure with the largest numerical value.

Selection Method: Indicate the method of which structures should be renumbered. *Range* allows the structures to be identified via the dialog box; *Selection Set* allows the structures to be identified via a graphical selection method.

Range to Process: When the **Selection Method** is set to *Range*, manually identify the structure(s) that should be part of the renumber process or click on the **All** button to select all structures in the network.

Renumbering Method: Indicate how structures should be renumbered. *Begin From* assigns the **Starting Name** value to the lowest structure name and increments by one until the network has been renumbered; *Increment By* adds the **Increment** value to each structure.

Note:

- Caution should be exercised when specifying a negative value for the **Increment** setting as duplicate structure numbers may result if the Increment value forces the renumbered structures to values less than 0.
- For network structures that do not contain a numerical value (*e.g.* MHA, MHB, MHC, MHD, *etc.*), a trailing numerical value of 1 is assumed (*e.g.* MHA1, MHB1, MHC1, MHD1, *etc.*) before the Increment process commences.

Pulldown Menu Location: Network > Sewer Network Utilities

Keyboard Command: renumswr

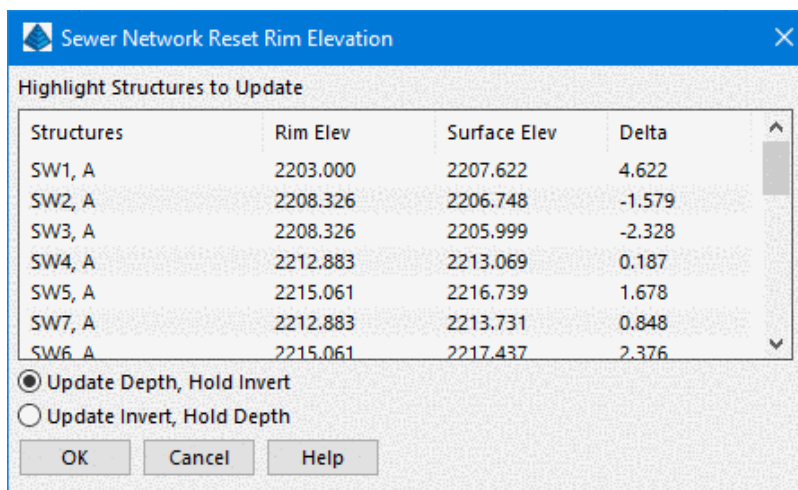
Prerequisite: A sewer (.SEW) file

Set Rim Elevation to Match Surface

This command compares the rim elevations for the current sewer network with the current surface model. If any differences are found, the program shows a dialog box with the rim elevations, surface elevations, structure names and elevation differences. You can highlight structures from this list to update their rim elevations to match the surface model. There are two update methods:

Update Depth, Hold Invert: Keeps the invert elevations while updating the rim elevations so that the depths are adjusted.

Update Invert, Hold Depth: Adjusts the invert elevations by the same amount as the rim elevations so that the depths are maintained.



Pulldown Menu Location: Network->Sewer Network Utilities

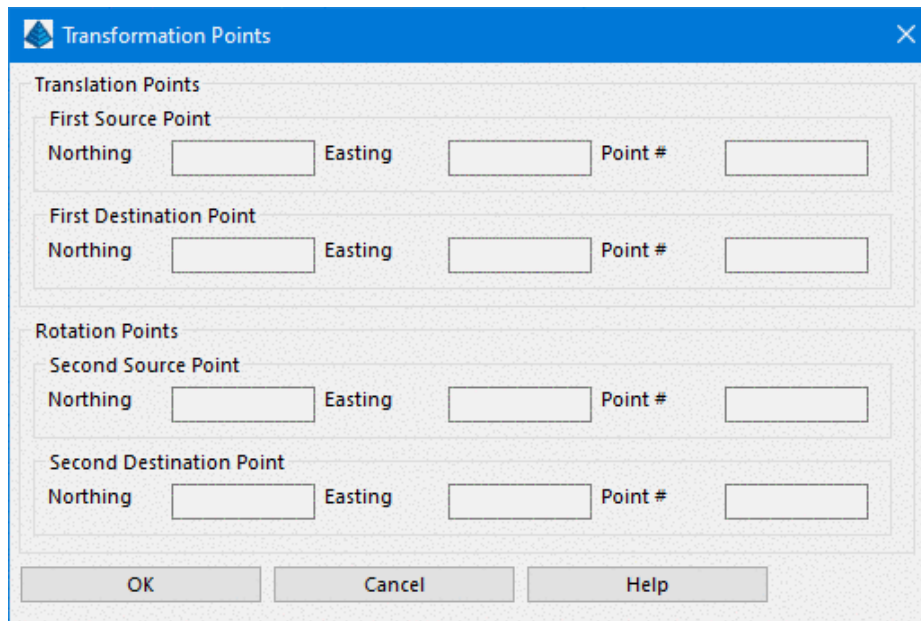
Keyboard Command: swrsurf

Prerequisite: sewer network structures

Transform Coordinates

This command translates and rotates the x,y coordinates of the structures in the sewer network model which can be used in case the coordinate system of the job is changed after creating the sewer network. The transform is defined by two pairs of source and destination points. The first pair of points defines the translation. The second pair defines the rotation by the angular difference between the angle of the two source points and the angle of the two destination points. The rotation pivot point is the first destination point. The points can be entered by point number from the current coordinate file or by typing in the coordinates. This transformation works similar to the Align Points command in the Points menu.

The program reports the amount of translation and rotation at the command line. The program creates a backup of the original sewer network model by copying the original .sew file to a .bak file of the same name.



Pulldown Menu Location: Network > Sewer Network Utilities

Keyboard Command: swrtransform

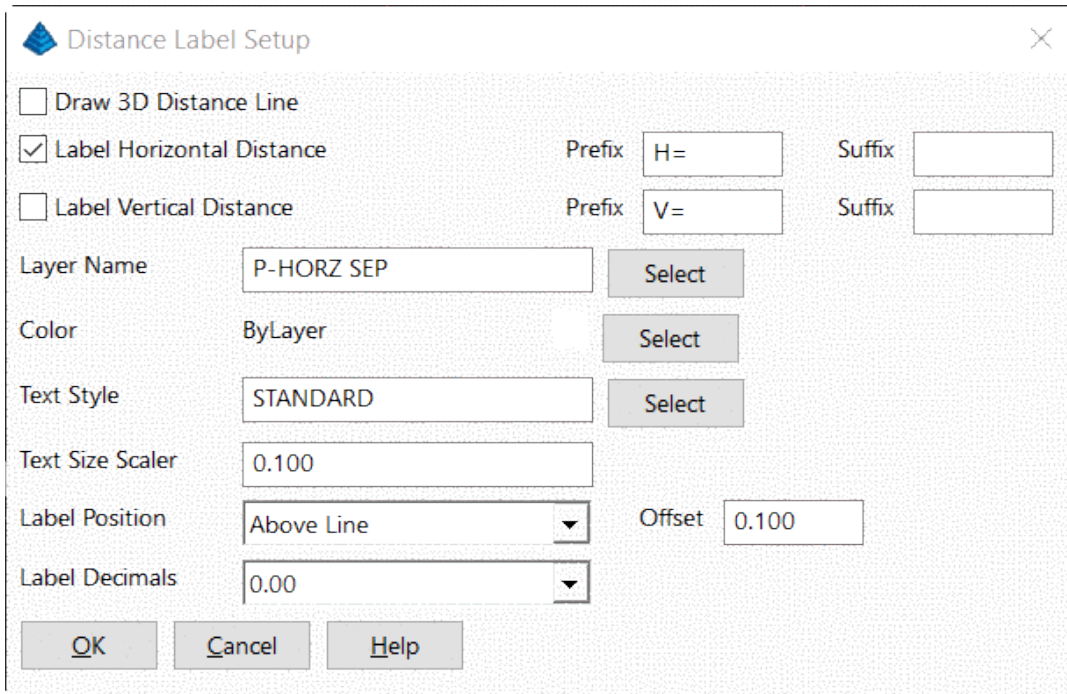
Prerequisite: Sewer Network File

Check/Label Clearance

This command checks for horizontal separation between any sewer network pipes, utility network pipes, and 3D pipe polylines. It displays horizontal separation value, as well as the Vertical Distance on command line and real time dialog box. Users can also label the results in plan-view.

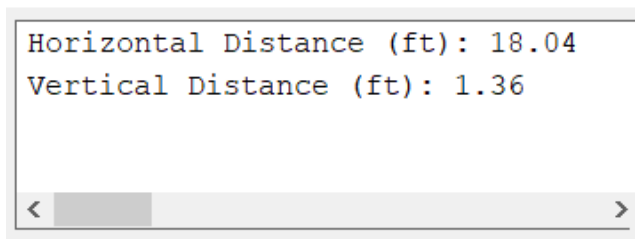
From the Network > Check Sewer Network menu in the Hydrology Module, select Label Separation for Parallel Pipes. In the drawing, first have all the networks and pipes drawn in plan view. Hovering between two sets of pipes in plan-view will give real time information on screen dialog box and picking in the plan view will place a label at the location picked.

The Label Separation for Parallel Pipes is used to draw the horizontal separation labels on screen. The settings are saved and no need to bring it up next time to check and label clearance. Clear the Draw Labels box and the program won't label the results on screen.



Distance Label Setup

Carlson



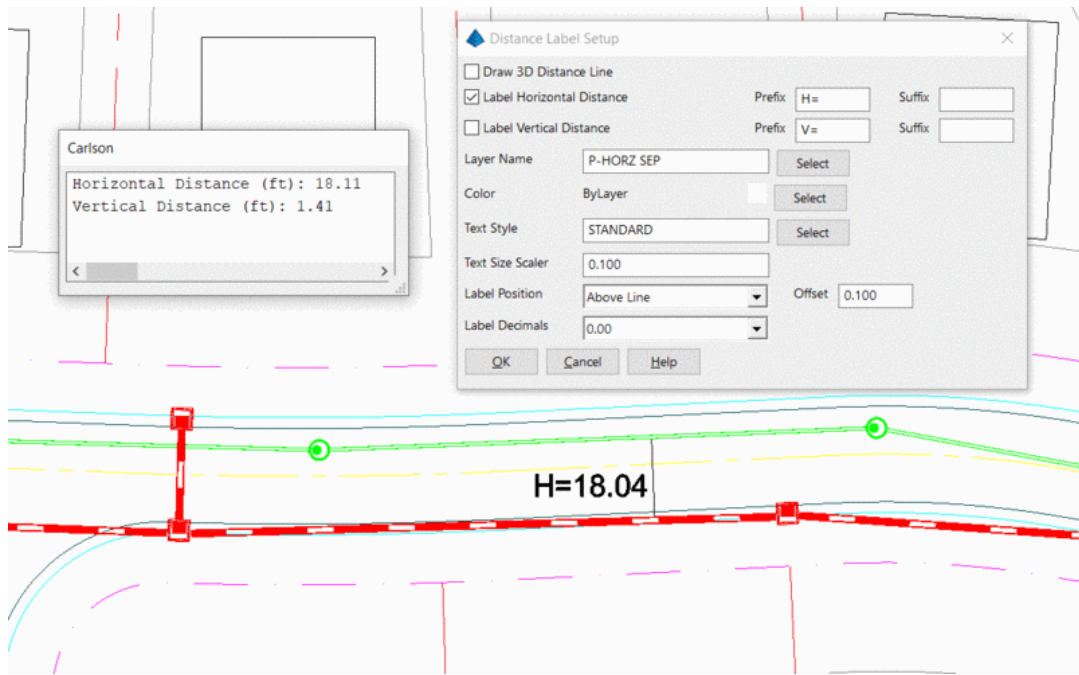
Real Time Clearance Result Dialog Box on Screen

```

Command: label_parallel
Inspect Distance Between Pipes
Pick location to report (0 for options, Enter to end): 2096003.13,859194.67
Hori. Dist=18.04, Vert. Dist=1.36
-----
Pick location to report (0 for options, Enter to end):

```

Clearance Result on Command Line



Horizontal Separation Labels and Dialogs on Screen

Pulldown Menu Location: Network > Check Sewer Network

Keyboard Command: label_parallel

Prerequisite: sewer network/utility network/3D pipe polyline separation in plan view

Compare Sewer Networks

This command reports the differences between two sewer network models which can be used to compare revisions of the same design. The program prompts for a sewer network file (.sew) to compare with the current sewer network. Then a report is displayed with any differences in the structures or pipes.


```

Compare Sewer Networks                               Tue Nov  2 10:12:20 2021
Network 1: D:\Carlson Software\2022 Beta Testing\SD.sew
Network 2: D:\Carlson Software\2021 Training Data\Mark Training\STORM.sew

Structure Changes

1 FES 8
  Name has changed from CI 8 to FES 8.
  Structure has changed from CI 5X5 to FES 24".
  Location has changed from (2096119.386, 859215.762) to (2096210.919, 859139.198).
  Base Elevation has changed from 347.500 to 343.250.
  Rim Elevation has changed from 353.664 to 345.290.

2 CB 5
  Name has changed from CI 5 to CB 5.
  Structure has changed from CI 5X5 to CI 5 X5.
  Location has changed from (2096191.138, 859157.567) to (2096193.600, 859158.794).
  Base Elevation has changed from 342.000 to 343.450.
  Rim Elevation has changed from 348.953 to 349.550.
  Drop Across Manhole has changed from 0.200 to 0.000.
  Inlet has changed from COMBO-GRADE to CI NCDOT SAG.
  Drainage Area has changed from 15929.253 to 15469.700.
  Runoff Coefficient has changed from 0.470 to 0.480.

3 CB 4
  Name has changed from CI 4 to CB 4.
  Structure has changed from CI 5X5 to CI 5 X5.
  Base Elevation has changed from 347.000 to 346.450.
  Rim Elevation has changed from 353.664 to 353.000.
  Inlet has changed from COMBO-GRADE to CI NCDOT GRADE.
  Drainage Area has changed from 1960.074 to 2027.583.

4 CB 6
  Name has changed from CI 6 to CB 6.
  Structure has changed from FES 24" to CI 5 X5.
  Location has changed from (2096204.173, 859128.891) to (2096119.386, 859215.762).
  Base Elevation has changed from 341.800 to 347.750.
  Rim Elevation has changed from 344.776 to 353.000.
  Inlet has changed from to CI NCDOT GRADE.
  Bypass Catchbasin has changed from to CSMH3.
  Drainage Area has changed from 0.000 to 44364.527.
  Tc has changed from 0.000 to 9.000.
  Runoff Coefficient has changed from 0.000 to 0.650.
  Curve Number has changed from 0.000 to 50.000.

```

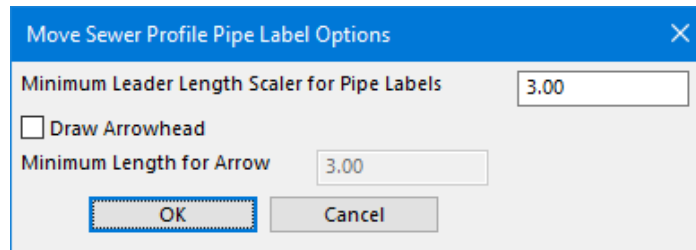
Pulldown Menu Location: Network
Keyboard Command: diffswr
Prerequisite: 2 Sewer network (.SEW) files

Move Sewer Profile Pipe Labels

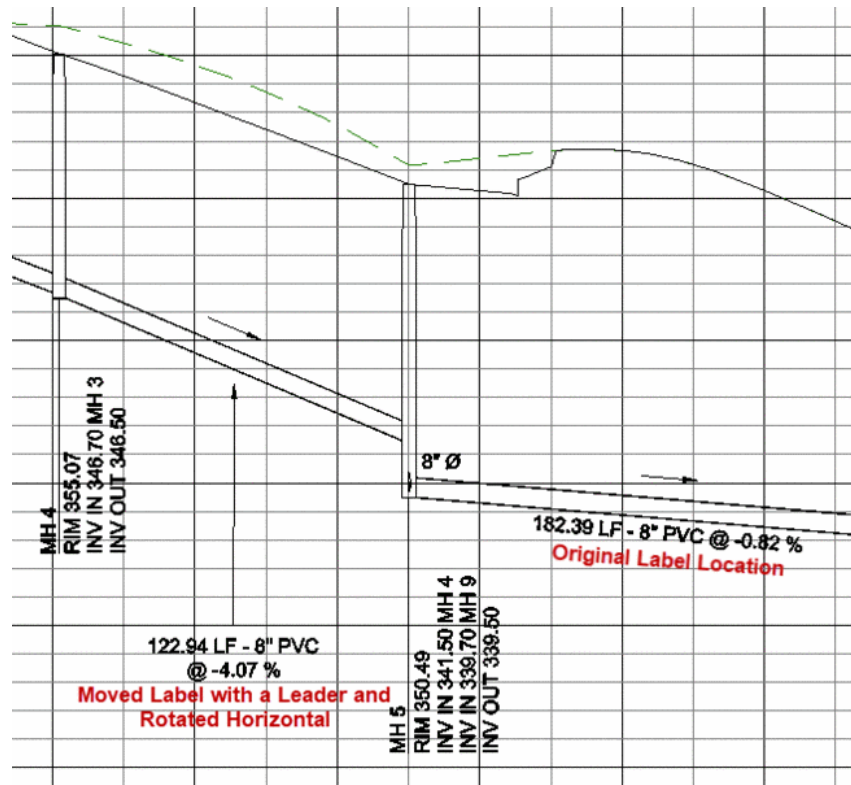
This command will allow the user to move profile pipe labels. See Command Prompt.

Command Prompt `Select profile Pipe label to move [Options/Restore]:`

Press O for Options



Select Pipe text within a Profile and move to new location (Press H to make label Horizontal)



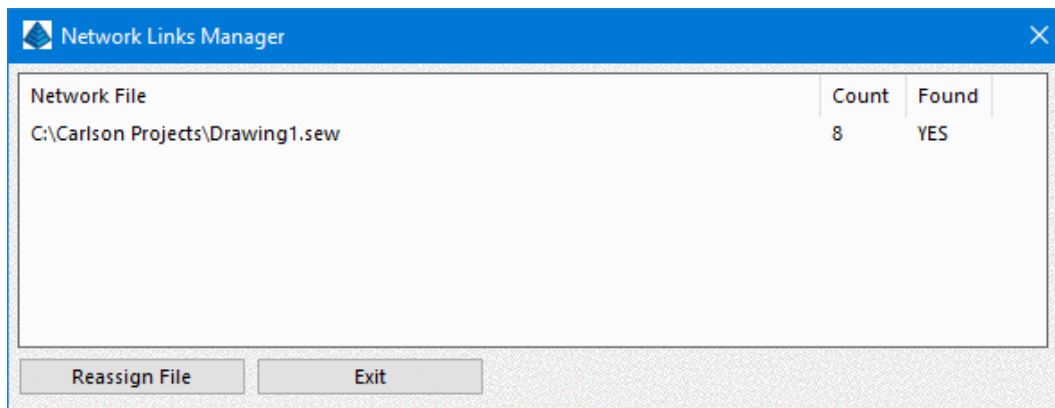
Pulldown Menu Location: Network > Sewer Labels > Move Profile Pipe Labels

Keyboard Command: move_swrprof_pipelabel

Prerequisite: Sewer network (.SEW) file

Pipe Network Drawing Links

This command finds all the Sewer, Utility or Pressure Pipe network entities in the drawing and reports the network file names that created the entities, how many entities were found for each network file, and whether that network file is found on the computer. The Reassign File button applies when the network file has been moved to another folder. This function prompts to select the new folder for the file and then updates the entities in the drawing to use the new folder location.



Pulldown Menu Location: Network

Keyboard Command: netlink_mgr

Prerequisite: Network entities

Pipe Network ID

This command prompts to pick an entity from the drawing and then reports which Sewer, Utility or Pressure Pipe network file created the entity.

Prompts

Select pipe entity to identify: *pick an entity*

Utility Network: D: \projects\utilnet\utilnet1\FREEDOM_2.util

Select pipe entity to identify (Enter to end): *press Enter*

Pulldown Menu Location: Network

Keyboard Command: pipe_id

Prerequisite: Network entity

Edit/Create Structure With Inverts

The Create Structure With Inverts command creates structures with rim and invert elevations but without pipe connections. So these structures are standalone without connections and can't be used for hydraulic calculations. The main purpose for these structures is for plan view labeling. The labels are controlled by the Plan View Label Settings command.

After picking the location for the structure, the data for the structure is entered in the dialog shown here. You can have any number of inverts that can be entered by either elevation or depth in the spreadsheet. See the Create Sewer Structure command in the manual for details on the other fields in this dialog.

The Edit Structure With Inverts command allows you to modify the data in existing structures. To run this command, first pick on a structure symbol or label and then the dialog shown here is used to edit the data.

Pulldown Menu Location: Network->Sewer Network Utilities

Keyboard Command: putswrstr, editswrstr

Prerequisite: None

Review Sewer Network Links

This command shows a list of all the sewer network links that the program knows about in the current drawing. These links are between the sewer network files and the drawing entities. You can use the Remove button to remove links for any obsolete sewer networks or if you don't want to link a certain sewer network.

Pulldown Menu Location: Network->Sewer Network Utilities

Keyboard Command: swrnetdict

Prerequisite: Sewer network (.SEW) file

Review Sewer Profile Links

This command shows a list of all the sewer network profile links that the program knows about in the current drawing. These links are between the sewer network files and the sewer profiles in the drawing. You can use the Remove button to remove links for any obsolete sewer profiles or if you don't want to link a certain sewer profile.

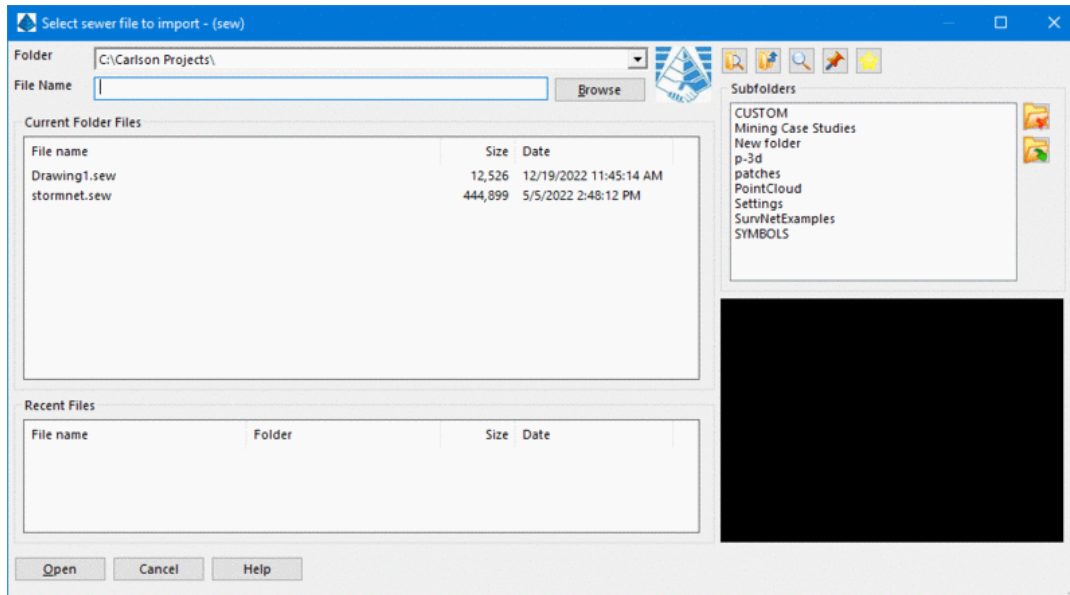
Pulldown Menu Location: Network->Sewer Network Utilities

Keyboard Command: swrproduct

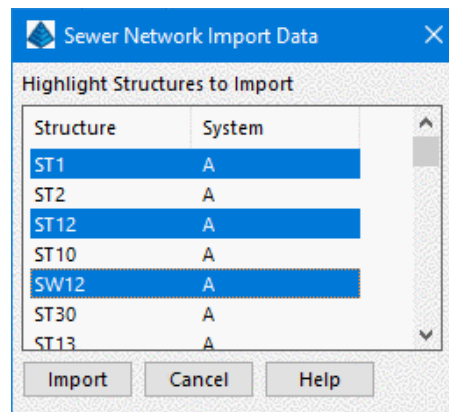
Prerequisite: none

Import Carlson Sewer Network

In certain scenarios, it may be desirable to combine the contents of one Carlson sewer project file into another sewer file and form a more comprehensive Carlson sewer project file. The Import Carlson Sewer Network routine will import selected contents of a Carlson sewer file (.sew) into the active sewer file as defined via the Set Sewer File command or the Sewer Network Settings command. If an active sewer file has not been set, you will first be prompted to select an active sewer network file and will then continue with the command as described below.



Browse to the folder location where the Carlson sewer file resides that you wish to import into the current sewer file.



Use standard Windows **click**, **shift+click** and/or **ctrl+click** functionality to select the structure(s) that should be imported.

Note:

- Any structure names found in the incoming sewer file that are also found in the active sewer file will not be imported.
- After being imported, the network can be drawn into the drawing via the Draw Sewer Plan command.

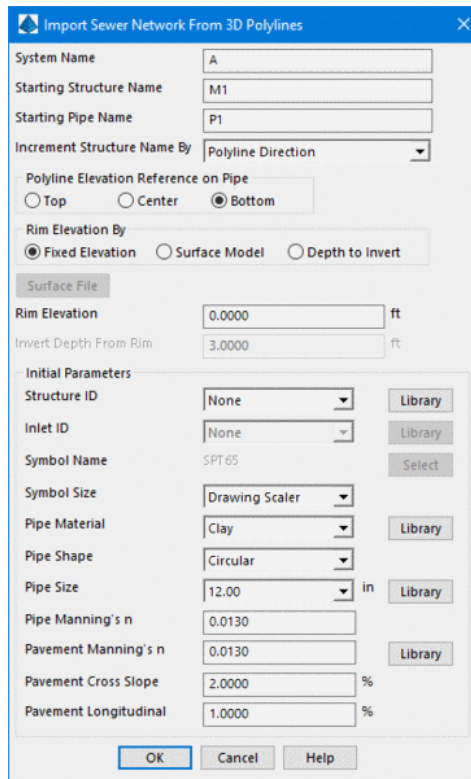
Pulldown Menu Location: Network > Sewer Network Utilities

Keyboard Command: impswr

Prerequisite: Existing Carlson Software sewer file (*.sew)

Import Network From 3D Polylines

This function assigns pre-determined sewer parameters for pipes and structures from selected 3D polylines that have been previously drawn to the screen and converts them to a Sewer (*.SEW) file. This command is very similar to the Import Sewer Network from 2D Polylines command and the Import Network from Centerline/Profile command. Sewer structures are inserted at the polyline vertices with the base elevation of the structure taken from the polyline vertex elevation. Pipes are created to replace the polyline segments and the invert elevations of the pipes also come from the polyline vertex elevations.



System Name: Indicate the system designation for the incoming sewer entities. A *.SEW file can have multiple systems (e.g. A, B, STRM1, STRM2, etc) within it.

Starting Structure Name: Indicate the designation that is to be assigned to the first structure (i.e. a junction) to be created by the routine.

Starting Pipe Name: Indicate the designation that is to be assigned to the first pipe that will be created by the routine.

Rim Elevation By: Set toggle to desired option. Fixed Elevation, Surface Model when enabled, this option allows you to click the **Surface File** button indicate a valid Carlson surface file (generally created through the Triangulate & Contour command) as the source for the new rim elevations, and Depth to Invert

Rim Elevation: Indicate a single desired rim elevation to be applied to all new structure locations.

Invert Depth from Rim: Indicate the desired depth from the rim elevation.

Structure ID: Indicate the type of structure to be used for the new sewer data or click on the **Library** button to access the Sewer Structure Library command to create a new structure definition or edit an existing structure definition. If this is an utility network (i.e. waterline, gas line and etc.) where there's no structure, you can choose None as the Structure ID. Therefore all the junctions would become an elbow pipe connections.

Inlet ID: Indicate the type of inlet to be used for the new sewer data or click on the **Library** button to access the Inlet Library command to create a new inlet definition or edit an existing inlet definition.

Symbol Name: Select the symbol to be used. Click on the **Select** button to access the Symbol Library. If this is an utility network (i.e. waterline, gas line and etc.) where there's no structure, you can choose SPT0 for no symbols.

Symbol Size: For the size of the symbol three options are available, Inlet Library Width, Drawing Scaler, and Unit Symbol

Pipe Material: Indicate the type of material to be used for the new pipes or click on the **Library** button to access the Pipe Material and Manning's *n*-Library command to create a new pipe material or edit an existing pipe material definition.

Pipe Shape: Indicate the general pipe shape that is to be applied to the new pipes.

Pipe Size: Indicate the desired pipe size for the new pipes or click on the **Library** button to access the Pipe Size Library command to create a new pipe size or edit an existing pipe size definition.

Pipe Manning's n: Indicate the desired Manning's *n*-value to be applied to the new pipes.

Pavement Manning's n: Indicate the desired Manning's *n*-value to be applied for storm sewer network runoff considerations or click on the **Library** button to access the Pavement Manning's *n*-Library command to create a new Pavement Manning's *n*-value or edit an existing Pavement Manning's *n*-value definition.

Pavement Cross-slope: Indicate the desired pavement cross-slope for ponding width calculation purposes.

Pavement Longitudinal Slope: Indicate the desired pavement longitudinal slope for ponding width calculation purposes.

Prompts

Select 3D polylines.

Select objects: *Select the desired 3D polylines that are to be converted and press Enter when complete. If several polylines are met at one vertex, only one structure is created, and the first polyline is the main flow line, others are the upstream branches.*

The sewer network will be drawn on the screen as the routine completes and can be manually drawn later through the Draw Sewer Network > Plan View command.

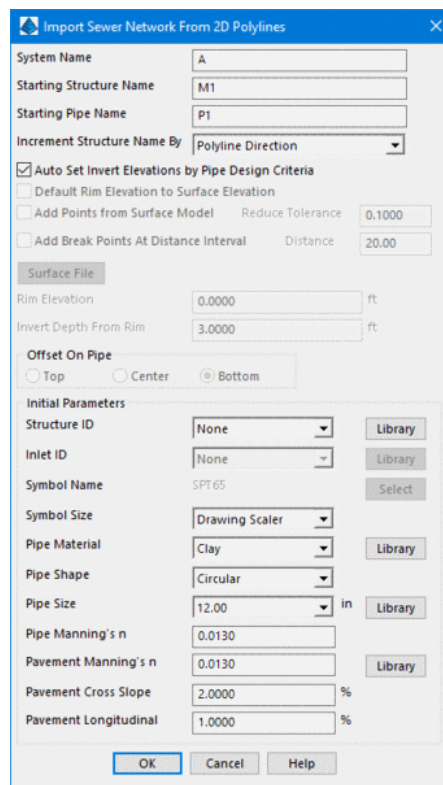
Pull-down Menu Location(s): Network > Sewer Network Utilities

Keyboard Command: swr3dp

Prerequisite: 3D polylines drawn on screen

Import Sewer Network From 2D Polyines

This function assigns pre-determined sewer parameters for pipes and structures from selected 2D polylines that have been previously drawn to the screen, places the pipes at a pre-determined offset from the structure rim elevation and converts them to a Sewer (*.SEW) file. This command is very similar to the Import Sewer Network from 3D Polyines command and the Import Network from Centerline/Profile command. Sewer structures are inserted at the polyline vertices with the base elevation of the structure taken from the polyline vertex elevation. Pipes are created to replace the polyline segments and the invert elevations of the pipes also come from the polyline vertex elevations.



System Name: Indicate the system designation for the incoming sewer entities. A *.SEW file can have multiple systems (*e.g.* A, B, STRM1, STRM2, *etc*) within it.

Starting Structure Name: Indicate the designation that is to be assigned to the first structure (*i.e.* a junction) to be created by the routine.

Starting Pipe Name: Indicate the designation that is to be assigned to the first pipe that will be created by the routine.

Default Rim Elevation to Surface Elevation: When enabled, this option allows you to click the **Select File** button indicate a valid Carlson surface file (generally created through the Triangulate & Contour command) as the source for the new rim elevations.

Add Points from Surface Model: When enabled, this option inserts additional junction points along the 2D polyline where it crosses the triangulation legs of the surface model.

Reduce Tolerance: Identify the smallest tolerance to which new points should be added to the 2D polyline.

Rim Elevation: Indicate a single desired rim elevation to be applied to all new structure locations.

Offset Invert from the Rim: Indicate a vertical offset distance rim elevation for the placement of the pipe.

Offset on Pipe: Indicate a the location on the pipe to be used for the pipe offset:

- **Top** - The top of the pipe is placed at the vertical offset from the rim elevation.
- **Center** - The center of the pipe is placed at the vertical offset from the rim elevation.
- **Bottom** - The bottom of the pipe is placed at the vertical offset from the rim elevation.

Structure ID: Indicate the type of structure to be used for the new sewer data or click on the **Library** button to access the Sewer Structure Library command to create a new structure definition or edit an existing structure definition.

Inlet ID: Indicate the type of inlet to be used for the new sewer data or click on the **Library** button to access the Inlet Library command to create a new inlet definition or edit an existing inlet definition.

Symbol Name: Use the **Select** button to access the symbol selector of the Symbol Library command to access a desired symbol for the structure.

Pipe Material: Indicate the type of material to be used for the new pipes or click on the **Library** button to access the Pipe Material and Manning's *n*-Library command to create a new pipe material or edit an existing pipe material definition.

Pipe Shape: Indicate the general pipe shape that is to be applied to the new pipes.

Pipe Size: Indicate the desired pipe size for the new pipes or click on the **Library** button to access the Pipe Size Library command to create a new pipe size or edit an existing pipe size definition.

Pipe Manning's n: Indicate the desired Manning's *n*-value to be applied to the new pipes.

Pavement Manning's n: Indicate the desired Manning's *n*-value to be applied for runoff considerations or click on the **Library** button to access the Pavement Manning's *n*-Library command to create a new Pavement Manning's *n*-value or edit an existing Pavement Manning's *n*-value definition.

Pavement Cross-slope: Indicate the desired pavement cross-slope for ponding width calculation purposes.

Pavement Longitudinal Slope: Indicate the desired pavement longitudinal slope for ponding width calculation purposes.

Prompts

Select 2D polylines.

Select objects: *Select the desired 2D polylines that are to be converted and press Enter when complete. If several polylines are met at one vertex, only one structure is created, and the first polyline is the main flow line, others are the upstream branches.*

The sewer network will be drawn on the screen as the routine completes and can be manually drawn later through the Draw Sewer Network > Plan View command.

Pulldown Menu Location: Network > Sewer Network Utilities

Keyboard Command: swr2dp

Prerequisite: 2D polylines drawn on screen

Import Network from Centerline/Profile

The Import Network from Centerline/Profile command allows a sewer file (*.SEW) file to be created by combining the horizontal components of a centerline (*.CL) file with the vertical component of a sewer profile (*.PRO) file. The process to create the sewer file is as follows:

1. The stationing contained within the centerline file is processed into memory.
2. The stations within the sewer profile file are located along the path of the centerline to form the structure/junction locations.
3. The pipe information stored within the sewer profile file are placed linearly between the structure locations.

The resultant sewer file can be further edited with the Edit Sewer Structure command or the Spreadsheet Sewer Editor command.

Note:

- Any stations within the *.PRO file that occur after the last station in the *.CL file are not written into the *.SEW file.
- Once the *.SEW file has been created, you can activate it for editing via the Set Sewer File command and/or draw it graphically via the Draw Sewer Network > Plan View command.

Pulldown Menu Location: Network > Sewer Network Utilities

Keyboard Command: swrc1pro

Prerequisite: Centerline (*.CL) file, Sewer Profile (*.PRO) file

Import Network From Text File

This command creates a sewer network file (.sew) from data in Bentley .sdb database file.

The program prompts for .sdb file to import and then the sewer network file to output. The data in Inlets, InletDesign, Manholes, ManholeDesign, Pipes, PipeDesign and Areas tables are imported and saved in the new sewer network file. The program would draw the sewer network plan view after it finishes importation.

ID	InletClass	Type	Length	Width	EffectiveLen	EffectiveWid	PerimeterFa
DBI 22, Sta. 397+00 Lt.	Type 3	Median Drop	3.35	2.94	0	3.35	2.94
DBI 26, Sta. 399+03.41 Rt.	Type 13 G RT	Grate	2.48	1.48	0	2.48	1.48
CBI 47, Sta. 399+03.41 Lt.	Type A Ah Dn LT	Curb Opening	10	0	0	10	0
CBI 48, Sta. 399+80.91 Lt.	Type A Sg Dn LT	Curb Opening	10	0	0	10	0
CBI 49, Sta. 399+80.91 Rt.	Type A Sg Dn RT	Curb Opening	10	0	0	10	0
CBI 50, Sta. 400+58.41 Lt.	Type A Bk Dn LT	Curb Opening	10	0	0	10	0
CBI 51, Sta. 400+58.41 Rt.	Type A Bk Dn RT	Curb Opening	10	0	0	10	0
CBI 52, Sta. 401+85 Lt.	Type A Bk Dn LT	Curb Opening	10	0	0	10	0
CBI 54, Sta. 403+35 Lt.	Type A Bk Dn LT	Curb Opening	10	0	0	10	0
CBI 56, Sta. 406+35 Lt.	Type A Bk Dn LT	Curb Opening	10	0	0	10	0
CBI 58, Sta. 409+30 Lt.	Type A Bk Dn LT	Curb Opening	10	0	0	10	0
CBI 60, Sta. 412+30 Lt.	Type A Bk Dn LT	Curb Opening	10	0	0	10	0
CBI 62, Sta. 414+50 Lt.	Type A Bk Dn LT	Curb Opening	10	0	0	10	0
CBI 64, Sta. 415+30 Lt.	Type A Bk Dn LT	Curb Opening	10	0	0	10	0
CBI 65, Sta. 416+00 Lt.	Type A Bk Dn LT	Curb Opening	10	0	0	10	0
CBI 53, Sta. 401+85 Rt.	Type A Bk Dn RT	Curb Opening	10	0	0	10	0
CBI 55, Sta. 403+35 Rt.	Type A Bk Dn RT	Curb Opening	10	0	0	10	0
CBI 57, Sta. 406+35 Rt.	Type A Bk Dn RT	Curb Opening	10	0	0	10	0
CBI 59, Sta. 409+30 Rt.	Type A Bk Dn RT	Curb Opening	10	0	0	10	0
CBI 61, Sta. 412+30 Rt.	Type A Bk Dn RT	Curb Opening	10	0	0	10	0
CBI 63, Sta. 414+50 Rt.	Type A Bk Dn RT	Curb Opening	10	0	0	10	0
CBI 66, Sta. 416+00 Rt.	Type A Bk Dn RT	Curb Opening	10	0	0	10	0
DBI 27, Sta. 401+85 Lt.	Type 3	Median Drop	3.35	2.94	0	3.35	2.94
DBI 28, Sta. 407+58 Lt.	Type 3	Median Drop	3.35	2.94	0	3.35	2.94
DBI 29, Sta. 408+25 Lt.	Type 3	Median Drop	3.35	2.94	0	3.35	2.94
DBI 30, Sta. 410+50 Lt.	Type 3	Median Drop	3.35	2.94	0	3.35	2.94
DBI 31, Sta. 411+00 Lt.	Type 3	Median Drop	3.35	2.94	0	3.35	2.94
DBI 19, Sta. 393+93 Rt.	Type 14	Median Drop	1.9	1.9	0	1.68	1.68
CBI 44, Sta. 394+50 Rt.	Type A Ah Dn RT	Curb Opening	10	0	0	10	0
CBI 43, Sta. 394+50 Lt.	Type A Ah Dn LT	Curb Opening	10	0	0	10	0
CBI 46, Sta. 397+00 Rt.	Type A Ah Dn RT	Curb Opening	10	0	0	10	0
CBI 45, Sta. 397+00 Lt.	Type A Ah Dn LT	Curb Opening	10	0	0	10	0

Pulldown Menu Location: Network > Import/Export Sewer Network

Keyboard Command: sdb2swr

Prerequisite: Bentley database file .sdb.

Import ADAC

This routine converts ADAC storm sewer network files into Carlson sewer (.sew) files. The ADAC data is stored in XML files.

Pulldown Menu Location: Network->Sewer Network Utilities

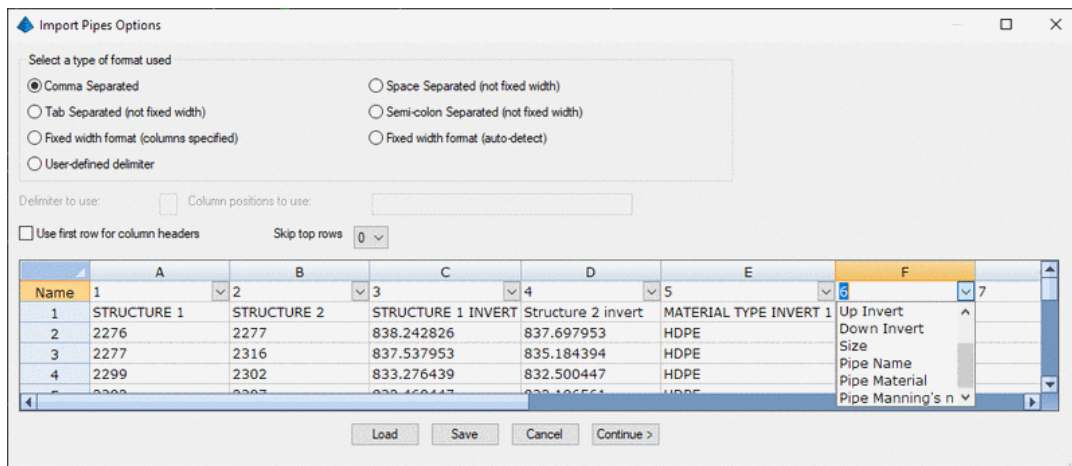
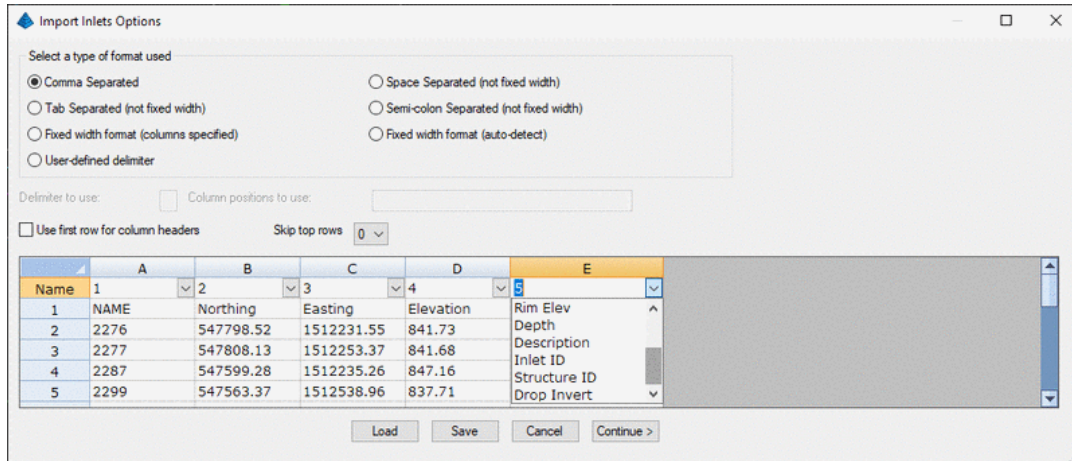
Keyboard Command: import_adac

Prerequisite: ADAC format XML file

Import Network From Bentley

This command creates a sewer network file (.sew) from data in two text files. One text file has the structure data with one structure per row in the text file with the fields separated by a comma or other delimiter. The structure file requires name, northing, easting and rim elevation. Other available fields are description, rim elevation, depth, description, inlet ID, structure ID and drop invert. The second file has the pipe data with one pipe per row. The pipe fields require the names of the connecting structures. The other fields include the upstream/downstream structure name, up/down inverts, pipe size, pipe name, pipe material and pipe manning's n.

The program prompts for the structure file and pipe file to import and then the sewer network file to output. Then there are dialogs to specify the format of the input text files. For each column of data in the input files, choose which field to assign.



Example structure data file:

Name,Northing,Easting,Rim

CB349,207062.75480,409437.63270,187.80000

DCB368,207194.02290,409287.13550,178.75000

DCB367,206993.48600,409306.21250,178.50000

CB347,206876.38490,409351.77020,176.50000

Example pipe data file:

Upstream_Node,Downstream_Node,Upstream_Invert,Downstream_Invert,Diameter

CB349,CB347,178.3000,166.1000,24.0000

DCB368,DCB367,174.0000,172.3500,15.0000

DCB367,CB347,172.3500,166.1000,15.0000

Pulldown Menu Location: Network > Import/Export Sewer Network

Keyboard Command: csv2swr

Prerequisite: structure and pipe text files.

Import StormCAD Network

This routine converts StormCAD storm sewer network files into Carlson sewer (.sew) files. The StormCAD data is stored in MDB database files. The routine prompts for the Pipes Database to read and then the Storm Database to read. The pipes data is required for the structure data. The storm data is optional for the drainage data.

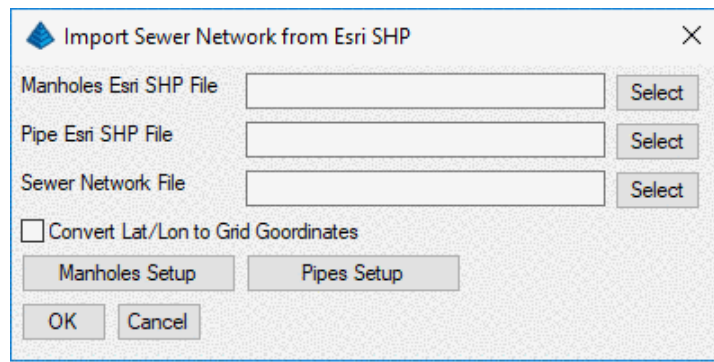
Pulldown Menu Location: Network->Sewer Network Utilities

Keyboard Command: swrimport

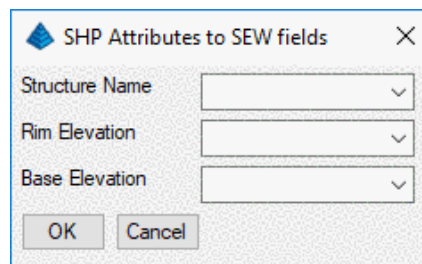
Prerequisite: StormCAD file

Import Sewer Network From Esri Shapefiles

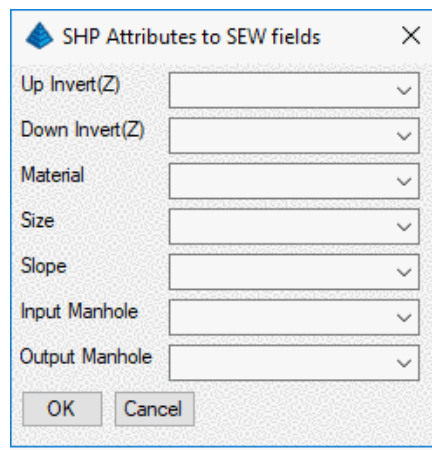
This command converts Esri sewer network shapefiles into Carlson sewer network file (.sew) . In order to create a Carlson sewer network file(.sew), a manholes SHP and a pipes SHP are required. The Manholes SHP contains manhole data, and the SHP type has to be point/pointZ/pointM. The Pipes SHP contains pipe data, and the SHP type has to be polyline/polylineZ/polylineM. There is also a setting to convert the ESRI SHP lat/lon coordinates to grid coordinates which requires the grid projection to be already defined in the Drawing Setup command.



There are **Manholes Setup** and **Pipe Setup** dialogs that match the SHP attributes to the .sew fields. This command uses Structure name as a key identifier and this field is required.



This command uses input manhole and output manhole as key identifiers to identify a pipe, and it uses these manholes to create relationship between manholes and pipe. The rest of fields matching are optional.



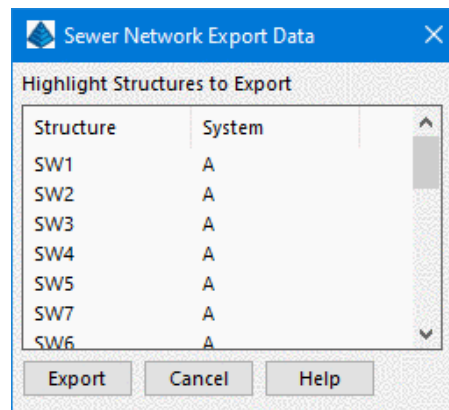
Pulldown Menu Location: >Network>Import/Export Sewer Network

Keyboard Command: shptoswr

Prerequisite: Esri Shapefiles

Export Carlson Sewer Network

In certain scenarios, it may be desirable to save all or parts of the active sewer file to an alternate sewer file name or location. The Export Carlson Sewer Network routine will export selected contents of the active Carlson sewer file (.sew) as defined via the Set Sewer File command or the Sewer Network Settings command and create a new sewer output file. If an active sewer file has not been set, you will first be prompted to select an active sewer network file and will then continue with the command as described below.



Use standard Windows **click**, **shift+click** and/or **ctrl+click** functionality to select the structure(s) that should be imported.

Browse to the folder location where you would like to write the new Carlson sewer file and specify the new sewer file name.

Pulldown Menu Location: Network > Sewer Network Utilities

Keyboard Command: expswr

Prerequisite: Existing Carlson Software sewer file (*.sew)

Export ADAC Network

This routine converts Carlson sewer (.sew) files into ADAC storm sewer network files. The ADAC data is stored in XML files.

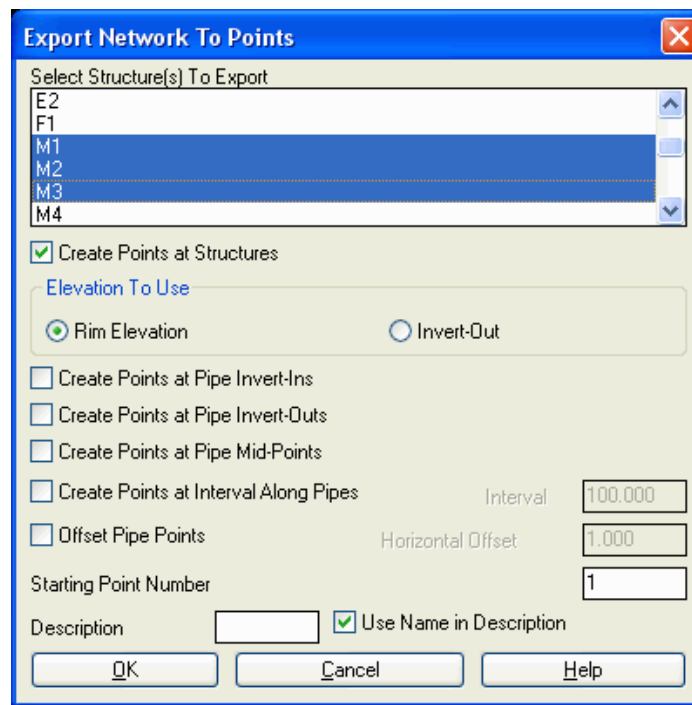
Pulldown Menu Location: Network->Sewer Network Utilities

Keyboard Command: export_adac

Prerequisite: Carlson .sew file

Export To Points

This command creates points in the current coordinate file for the selected structures of the current sewer network. In the options dialog, you can select multiple structures from the list of structure names. You can create points at the structure locations and the pipe locations. For pipes, you can create points at the invert-in and invert-out connection points, at the pipe mid-points and at an interval along the pipe. The Offset Pipe Points option offsets horizontally perpendicular to the pipe. The elevation for the structure points can be either the rim elevation or the invert-out of the structures. The point numbers will be incremented from the specified Starting Point Number. The Use Name in Description option puts the structure name into the point description.



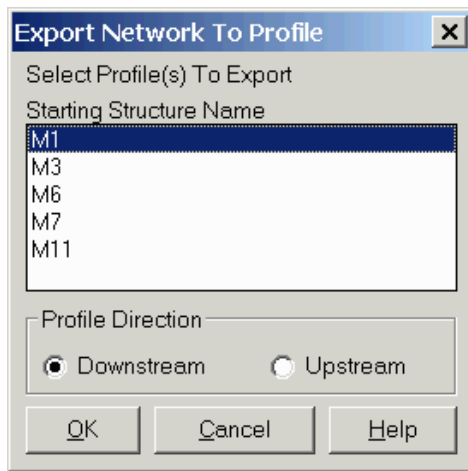
Pulldown Menu Location: Network->Sewer Network Utilities

Keyboard Command: swr2pts

Prerequisite: Sewer network (.SEW) file

Export To Profiles

This command creates a profile (.pro) from the current sewer network. The profile is created from the specified upstream structure through all the downstream connections to the end of the pipeline. The profile direction can be either upstream or downstream for the stations.



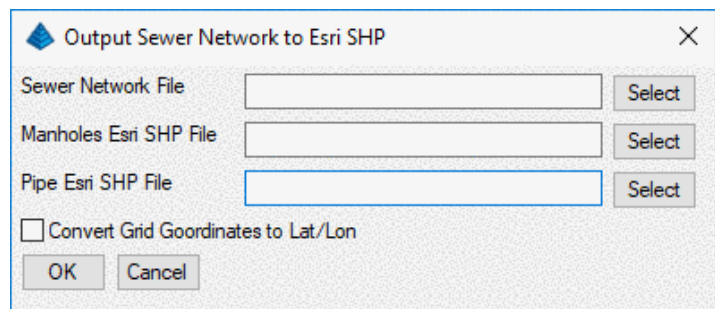
Pulldown Menu Location: Network->Sewer Network Utilities

Keyboard Command: swr2pro

Prerequisite: Sewer network (.SEW) file

Export To Esri shapefiles

This command output Carlson sewer network (.sew) to Esri Shapefiles. The command uses a .sew file as an input, then it output manholes and pipes data to two different SHP files. It is optional to convert the northings/eastings from the drawing to lat/lon by converting the drawing coordinates to lat/lon.



Pulldown Menu Location: Import/Export Sewer Network

Keyboard Command: swrtoshp

Export Pre-Carlson 2020 Sewer Network

This routine saves the current sewer network file in the older file format that was used before Carlson 2020, so the project can be used in Carlson 2019 or older releases.

Pulldown Menu Location: Network > Import/Export Sewer Network

Keyboard Command: swrsave_old

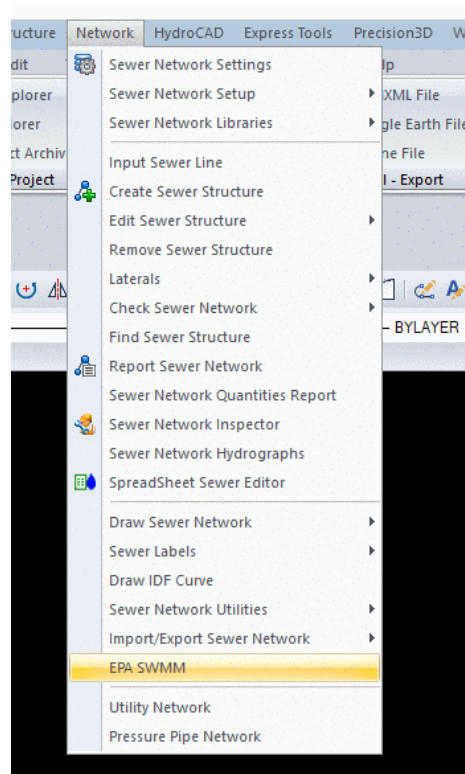
Prerequisite: Sewer network data

EPA_SWMM

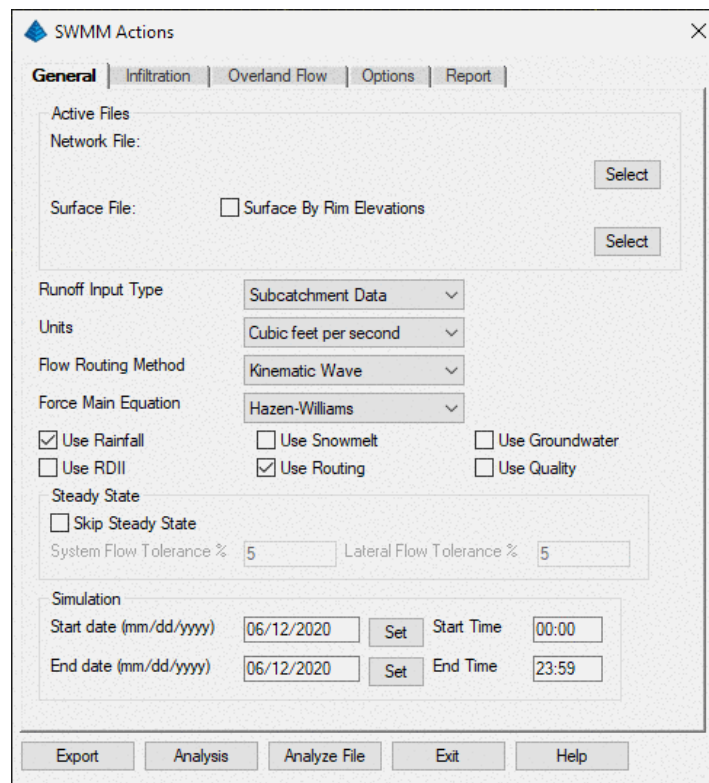
EPA SWMM (Environmental Protection Agency Storm Water Management Model) is a software package developed and provided by the EPA for modeling and reporting of storm water management systems. Many Carlson users are required to use SWMM analysis and reporting in lieu of Carlson's analysis system. This command allows the user to model the storm water system within Carlson then export a set of files that can be used to perform an analysis within the EPA SWMM program. Users may find EPA SWMM software and documentation here:

<https://www.epa.gov/water-research/storm-water-management-model-swmm>

The command, **swmm**, may be typed in the Command Ribbon or by selecting EPA SWMM from the Hydrology Network dropdown.



When executed, the **SWMM Actions** dialog appears. The General tab is presented first.



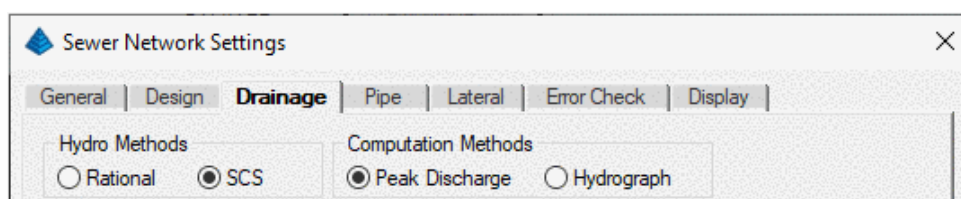
Active Files: Select an existing sewer (.SEW) file to become the active sewer file and a "surface file" for ground cover calculations:

- *Surface by Rim Elevations* - When this option is enabled, the ground surface elevation is derived as an interpolated grade from structure rim elevation to structure rim elevation and the Surface Model option is disabled, or,
- *Surface Model* - When the *Surface by Rim Elevations* option is disabled, select a valid surface model (.TIN, .FLT, .GRD) that blankets the sewer network. The ground surface elevation is obtained from the surface model along the pipe reach.

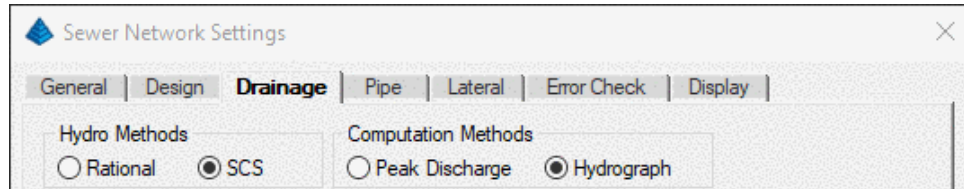
Use the **Select** buttons to browse to the correct surface and network files.

To create a .sew file, input the storm water management system using the **Create Sewer Structure** command found in the Network dropdown of the Hydrology module. It's important to note that Carlson only transfers rain event data generated by the SCS Method. If the SCS Method is inappropriate for the system, the user may input other types of rain events and watershed analysis using other methods directly into SWMM.

The **Runoff Input Type** allows for 2 options: **Subcatchment Data** and **Inlet Flow Hydrograph**. The **Subcatchment Data** specifies that the node inflow will be generated from the watershed computed flow and the **Computation Methods** must be set to **Peak Discharge** in the **Sewer Network Settings**:

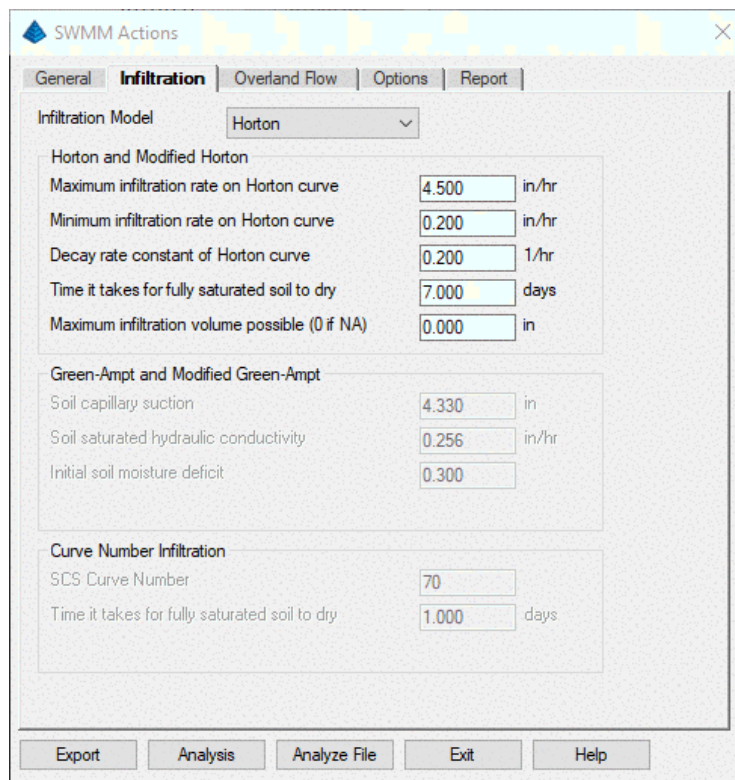


The **Inlet Flow Hydrograph** option specifies that the node input flow is from the Hydrograph:

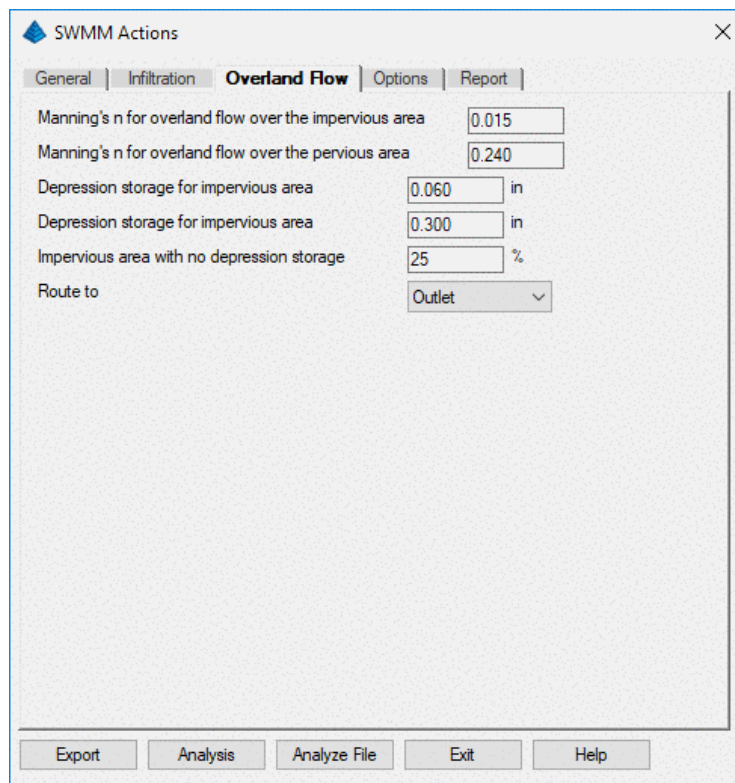


Set the desired **Units, Flow Routing Method and Force Main Equation**. Choose the physical processes to include in the analysis. Choose whether to skip **Steady State** analysis or not. If skipped, set the **System Flow Tolerance** and the **Lateral Flow Tolerance** percentages. Finally, set the **Simulation Start and End** dates and times.

Selecting the **Infiltration** tab allows the user to choose one of five Infiltration Models and exposes input parameters specifically needed for the Model selected. Parameters for Models not selected are grayed out.



Selecting the **Overland Flow** tab allows the user to set Manning's friction coefficients (n) for impervious and pervious areas, depression storage depths, the percentage of Impervious areas with no depression storage, and where Overland Flow is routed, Pervious, Impervious or Outlet.



Selecting the **Options** tab allows the user to enter additional parameters needed by SWMM for analysis.

Sweep start and Sweep end are used reduce the accumulated buildup of specific pollutants with street sweeping.

Dry Days is the number of days with no rainfall prior to the start of the simulation.

Report Step is the interval for reporting of computed results.

Wet Step is the interval for computing runoff during rainfall and continuing until standing ponds and LID devices (Low Impact Development) are dry.

Dry Step is the time step length used for runoff computations when there is no rainfall or ponding and storage devices are dry - essentially to compute pollutant buildup. Dry Step must be greater than or equal to the Wet Step.

Routing Step is the time step length used for routing flows and quality constituents through the conveyance system.

Lengthening Step is used with Dynamic Wave Routing to artificially lengthen a conduit so that the travel time of a wave is greater than this time step (Courant stability criterion).

Variable Step is a safety factor applied to variable time steps computed using Dynamic Wave Routing.

Minimum Step is the shortest time step allowed when variable time steps are used with Dynamic Wave Routing.

Inertial Damping is used to control inertial terms in the Saint Venant momentum equation used in Dynamic Flow Routing. Choosing **None** uses these terms at all times, **Partial** reduces the terms as flow approaches critical and removes them when flow becomes supercritical, **Full** removes these terms from the equation.

Normal Flow Limited specifies which condition is checked to determine if conduit flow is supercritical and should be limited to normal flow using Dynamic Wave Routing. Use Slope to determine if water surface slope is greater than conduit slope. Use Froude to check if the Froude Number is greater than 1 or Both to check both conditions.

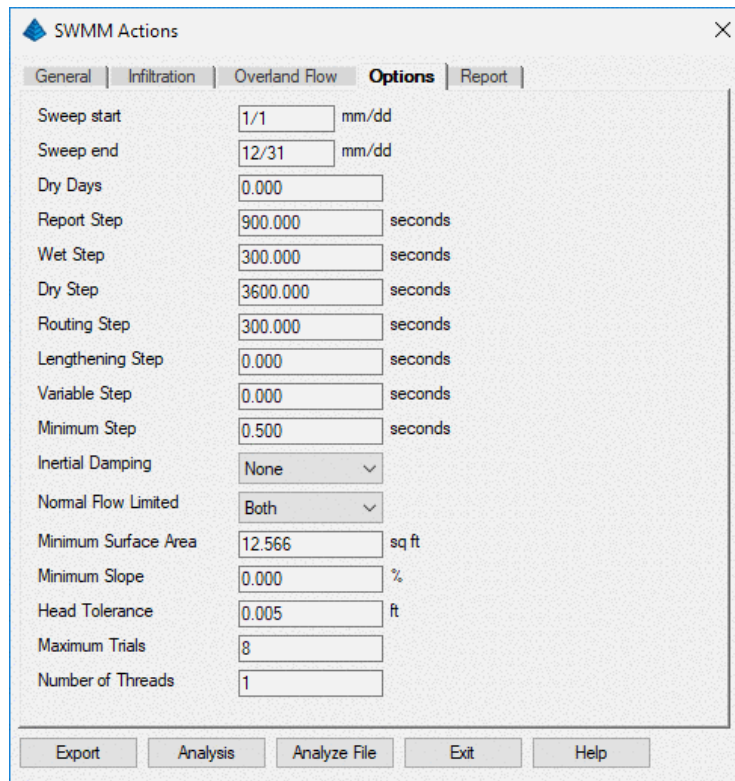
Minimum Surface Area is the minimum surface area used when computing water depth changes under Dynamic Wave Routing.

Minimum Slope is the minimum slope allowed for a conduit. Zero sets no minimum. SWMM uses a minimum of .001 ft drop during computations.

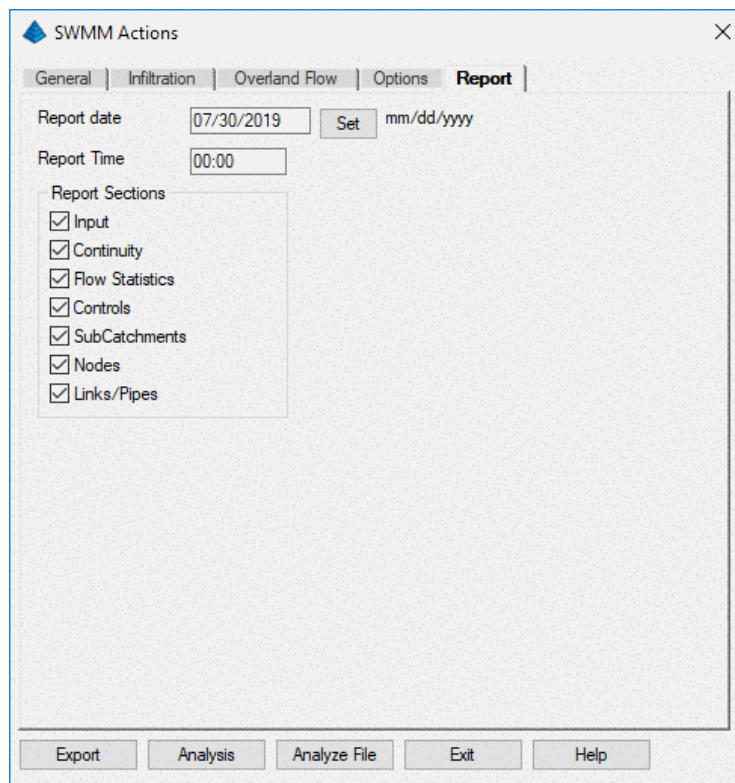
Head Tolerance is the difference at each node between successive trials. If the difference in head for a time step is less than this value, the flow solution has converged.

Maximum Trials is the maximum number of trials allowed during a time step to reach convergence when updating hydraulic heads at the system's nodes.

Number of Threads allows the user to increase the number of parallel processing threads used during Dynamic Wave Flow Routing if the computer being used has multi-core processors installed.



Selecting the **Report** tab allows the user to specify the report date and time as well as to select the Report Sections to be included.



The **Export** button exports an .inp file to be opened in EPA SWMM software.

The **Analysis** button creates a report within Carlson software for review and error checking.

The **Analyze File** button allows the user to open an existing .inp file and displays the Carlson analysis report.

Exit closes the SWMM Actions dialog.

Pulldown Menu Location: Network

Keyboard Command: swmm

Prerequisite: SWMM version 5.1

Utility Network

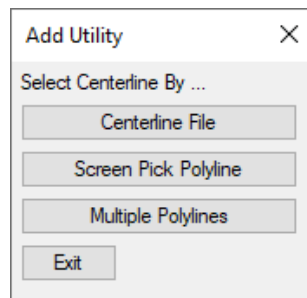
Utility Network is a command that synthesizes underground utility design for Projects which include all types of underground utilities. Interactive design for all utility centerlines, profiles and utility sizes and types is accomplished using a system similar to Carlson Road Network.

The command consists of a docked dialog on the left of the screen identifying all parameters of a Utility Network. The tool is very similar to Carlson Road Design Tool by design where utility horizontal alignments are defined by polylines or existing centerline files within the drawing. These alignments, along with a design surface specified in Settings, are used to determine proposed grade within each pipe alignment. Proposed grade, along with depth of utility, are used to set elevation for utilities that are installed a set distance below ground. For Utilities such as Gravity Sanitary and Stormwater Collection which typically have varying cover depths, the Storm/Sewer Network tool is recommended.

Once the Utility Network is setup, plan and profiles can be created for any Utility in the Network including symbols and notation. Additional output includes export of networks as 3d polylines and 3dfaces. When creating any profile such as a road profile, utility crossings will be included in the profile for any Utility in Utility Network. When all underground utilities are setup as a Utility Network, a comprehensive below ground utility collision check can be performed. If pressure calculations are needed, utility network files can be opened in Pressure Networks and setup with a pressure model.

To open the Utility Network Main dialog, enter "utilnet" at the command line or select Utility Network from the main menu "Utility" within the Hydrology module. Utility Network will dock a dialog to the side of the drawing which allows full use of the command line and menu items for other tasks.

The main console contains all utility alignments assigned to a project. An alignment is added by selecting the add(+) button which opens the Add Utility dialog.



If a tin file representing final surface elevation has not already been specified in Settings, a prompt will appear requesting the tin file name. If centerline files have not already been created, select a polyline representing horizontal alignment. Name and save the selected centerline file. Lastly, if the project contains multiple underground utilities to add to the model, Multiple Polylines can be selected and setup at one time using global settings.

The next dialog will request the method used to establish Utility depth.

Minimum Depth, Minimum Length and options for reference surface break points can be specified. Options for surface break points include the following: **None**, **At Minimum Depth**, **Min Depth Streamlined**, **Min/Max Streamlined and Maximum Deflection Angle**. The **Min Depth Streamlined** method uses fewer profile grade breaks to keep the profile below the Min Depth and will not follow the surface up for short bumps. The **Min/Max Streamlined** is similar with the addition of keeping the profile above the specified Max Depth. The **Min Length** controls the shortest distance between profile grade breaks. The **Transition Depth Between Stations** method linearly transitions the depth between two depths set at two stations. The **Maximum Deflection Angle** option is used when designing rigid pipe such as Ductile Iron which has a standard length and maximum deflection angle between pipe segments.

Once the Utility has been added, the Edit Utility dialog is displayed. Finish the addition by defining additional parameters. If adding multiple polylines, these will be named (Utility Name)1, (Utility Name)2, etc. Once Utilities are created by any manner, all will appear in the Utility Network discreetly and may then be further edited individually if desired.

Utility Name is a user defined label for the utility. Each name must be unique in the utility network.

Description is an optional setting that is used in reports and can be labeled in the Draw Profile and Draw Sections commands.

Type setting chooses the type of utility as gas, water, sanitary, storm, communication or other. The setting is used in reporting and used to set the default color for the linework.

Pipe Size is the initial Outside Diameter for the utility line. If the utility alignment contains only one size, enter the

value here. If the utility changes size along the alignment, select the edit button and specify size/station.

Pipe Thickness (in) is the difference in the Outside Diameter minus the Inside Diameter divided by 2. By providing pipe thickness, the profile of the utility can show both Interior and Exterior diameters when specified in Utility Profile Labels in the Draw Profile dialog.

Min and **Max Cover** is the minimum and maximum depths for the utility profile. These values are used to control depth of Utility and are used by Report for checking the min/max cover.

Layer and **Linetype** used to draw the horizontal alignment are specified for the Plan View options.

3d Polyline layer is the layer used if a 3d polyline is created for the Utility and 3D Faces Layer is used to draw 3 Dimensional faces.

Profile Elevation Reference On Pipe sets the reference point of a Utility used to set depth.

Pipe Shape specifies the shape of the Utility. **Circular** is typically used for most utilities, but other options may be needed. A **Rectangular** shape can be used to specify a dry utility corridor.

The **Centerline** button is used to specify the centerline file name for the utility. The **Edit** button on the same row opens the horizontal alignment editor.

The **Profile** button is used to specify the profile file name for the utility. The **Edit** button on the same row opens the vertical profile editor. This profile editor is similar to the **Road Profile Editor** which is documented in the Roads chapter. However, the Utility Network Vertical Profile editor has additional functions needed for Utility design.

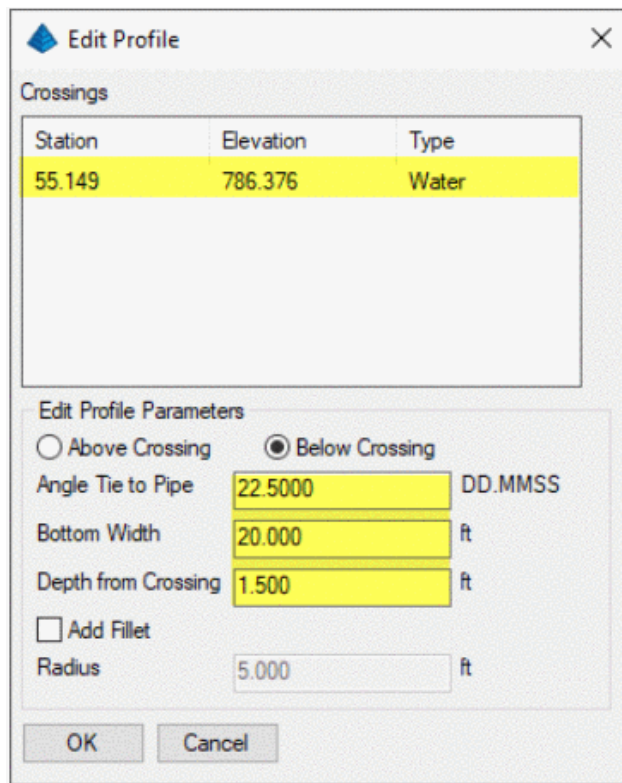
Set Depth can be used to make the utility profile a fixed depth from the reference surface. This function has an option to add stations in the utility profile at each grade break in the surface so that the utility profile can exactly follow the surface. The Min Depth Streamlined method uses fewer profile grade breaks to keep the profile below the Min Depth without setting depth of utility at every grade change which can result in a "jagged" alignment. The Min/Max Streamlined is similar with the addition of keeping the profile above the specified Max Depth. The Min Length controls the shortest distance between profile grade breaks. The Maximum Deflection Angle option, when used with a minimum nominal pipe segment length is used when each pipe segment must be shown in profile view at its nominal length. For example, a common nominal length for 6 inch Ductile Iron Pipe is 20 feet which has a gasketed fitting that must connect to the next pipe segment at an angle less than 5 degrees from center bore.

Please note that changes to depth does not have to be applied to the entire alignment. If changes are only needed for stations past a starting station, select the starting station, select set depth and check the option for Update from current location. The starting station will be displayed once Update from current location.

Reduce removes profile data points without changing the profile elevations more than the specified cutoff tolerance. **Translate** is used to raise or lower the vertical alignment by a set offset distance. The entire utility can be adjusted with this command or a specific station range can be specified.

Setup calls the **Utility Profile Settings** dialog box. These are profile settings when profiles are drawn within the Utility Network Editor.

Edit Profile at Crossing allows for editing the profile when it must pass over or under another utility. The tool automatically provides the stations where it recognizes a crossing. This tool will also recognize pipe crossings from Networks created in the Storm/Sewer Network Editor.



Choose if the profile is to pass over or under the crossing. Input the **Angle Tie to Pipe** to drop or rise. Input the **Bottom Width** of the crossing excavation (flat after drop or rise). Input the **Depth From Crossing** (clearance).

Edit Connections is an optional list of stations along the utility line where something related to the network is located. name and symbol or model is specified. The symbols specified are drawn in both plan and profile views. The **Model** is a 3D Solid such as a water system valve to draw with the Draw 3D Faces function.

Add From Points allows the user to add connections by importing coordinate file points by range or group.

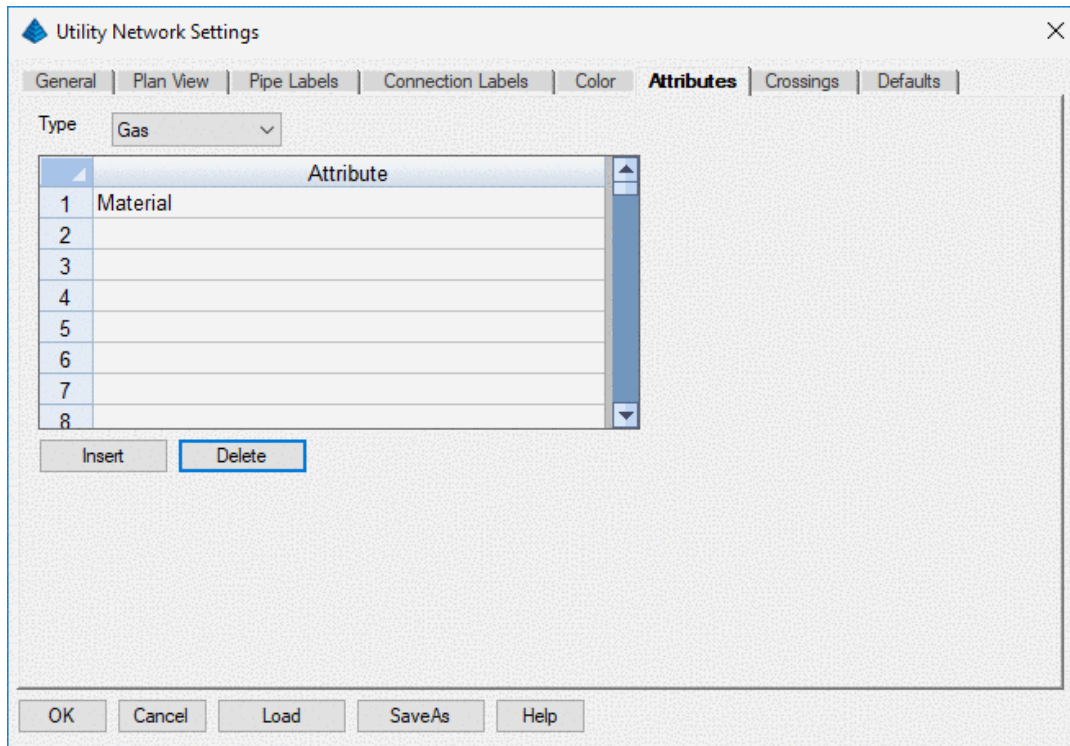
For the **Insert** function, the program first prompts for the centerline. You can either select an existing .CL file, or pick a polyline from the drawing. When you select a 3D polyline, the polyline is used both for the centerline and profile.

Delete button removes the currently highlighted utility line from the network.

The **Report** function has three type of reports. **Data Files** report includes the parameters for each utility line. The **Connection** report includes the input parameters for all the connections. The **Laterals** report includes the input parameters from the laterals. The **Min/Max Cover** report checks each utility profile and reports Min Cover Setting, Average Cover, Min Cover Station and Depth, Max Cover Station and Depth. The **Quantities** report includes the utility name, type, length, shape, pipe size and user-defined attributes.

Edit Laterals is an optional list of laterals connected to the utility line. Each lateral is defined by a station at the connection along the utility line, the pipe size for the lateral and the "cleanout" coordinates. Please Note that the "cleanout" can be any type of service connection such as a cleanout or water meter. Use the **Pick** button to screen pick the connection and cleanout points, or to pick an existing line for the lateral. The Connection and Cleanout Symbols are used with drawing the utility network in plan view.

Edit Attributes is an optional list of attribute names and values such as pipe material or construction phase to assign to the utility. The list of available attributes are defined in the Settings function. These attributes are used in reports and are entered in a spreadsheet.



The **Draw** button has four functions:

The **Draw 3D Polylines** function creates a 3D polyline for each utility line. These polylines are tagged with their pipe size in the same format as the Assign Pipe Data To Polyline command.

The **Draw 3D Faces** function draws each utility line as a 3D pipe using 3D faces. These 3D faces can then be used in the 3D Viewer commands.

The **Draw Plan View** function draws a 2D polyline for each utility line and labels the utility name and connection points.

The **Draw Profile** function draws the profile for the currently highlighted utility. Any symbols defined on the Connection List for the utility will be drawn on the profile. Besides picking the Draw button, you can also run Draw Profile by picking the utility in the list, right-click and pick Draw Profile.

The **Save**, **SaveAs** and **Load/New** functions are used to store and recall the utility network to .util files.

General Settings

For the Settings function, the **Reference Surface** is a triangulation file that is used for surface elevation reference when setting utilities to a specified depth. The **Auto Zoom To Current Selection** and **Highlight Current Centerline** apply to highlighting centerline alignments to indicate the current selected Network alignment. When Auto Zoom is on, modespace will zoom to the extents of the current alignment. **Redraw Profiles When Network Changed** dropdown allows the user to automatically redraw changed profiles, prompt for redraw, or not redraw automatically when exiting the command. Use **Load** to load an existing Utility Network Settings file (.UNS). Use

Save As to save the current Utility Network Settings to a .UNS file for later use.

Plan View Settings

The **Text** and **Symbol Size Scalers** and **Text Style** are used for the Draw Plan View function. When **Draw Easements** is selected, an easement will be drawn as part of the pipe network where the width is determined by utility depth. Entering a value for the first **Zone** sets the maximum depth for the first Zone and the Easement Width, then continue to set maximum depths for the Next Zone and the Easement Width. Easements widths increase as depth increases due to maintenance requirements for deeper utilities.

On the **Pipe Labels** tab, changes to pipe labeling functionality can be set for each type of utility by selecting Utility Type. **Type** is selected using the dropdown arrow. Available labels appear on the left. Click **Add** to select label information wanted in the drawing, **Remove** to not display this information, **Setup** to add a Prefix and/or Suffix to the label. Use the green arrows to reorder the Used Labels list. **Label Location** lets you place the label at Ends of the pipe, at an Interval, Both the Ends and at an Interval, or None. Fill in the desired interval in the **Label Interval** box. Choose **End Label Style** as Parallel, Horizontal or using MLeader arrow and text box. Choose a **Labels Above Symbol** number from 0 - 9. The labels equal to and less than the number selected are placed above the symbol with the remaining labels being located below when using Horizontal or Parallel Styles.

On the **Connection Labels** tab you can control how Connection Labels are placed. Available labels appear on the left. Click **Add** to select label information wanted in the drawing, **Remove** to not display this information, **Setup** to add a Prefix and/or Suffix to the label. Use the green arrows to reorder the Used Labels list. Type in a layer name or choose an existing drawing layer with the Set button. Choose **Label Style** as Parallel, Horizontal or using MLeader arrow and text box. Choose a **Labels Above Symbol** number from 0 - 9. The labels equal to and less than the number selected are placed above the symbol, the remaining labels below when using Parallel or Horizontal Styles. Use **Load** to load an existing Utility Network Settings file (.UNS). Use **Save As** to save the current Utility Network Settings to a .UNS file for later use.

On the **Color** tab, set the default colors to use when creating line work for each type of utility within the utility network.

On the **Attributes** tab, set the optional attribute names to assign to each utility type to be used in the Report > Quantities function. The attributes are specified by Utility Type. Select the Utility **Type** using the dropdown arrow, then enter Attributes, one per line. Use Insert to insert another Attribute between existing entries, use Delete to remove an entry. Attribute Types include Gas, Water, Sanitary, Storm, Electric, Communication, Irrigation, Fire, Utility Duct, Other and Connection.

Using the **Crossings** tab, you may specify if vertical pipe clearance is checked within the Utility. To check clearances, **Check Crossings** box must be selected. Additionally, you can check other Utility Networks for collision with the current network by clicking the **Add File** button, then browsing to the desired network file. Remove additional networks with the **Remove File** button. Disable **Check Crossings** by unchecking the box. **Do not specify the current network in the additional networks to check.**

The **Profile** tab provides the ability to add an existing and proposed surface that will be used when vertical profiles are drawn. Please note these are only used to produce existing and proposed profiles and are not used as part of the design.

The **Defaults** tab has default a symbol and model to use when adding connections.

Although all types of Utilities can be setup together in a Utility Network, they can also be setup separate from each other when additional layer control is needed.

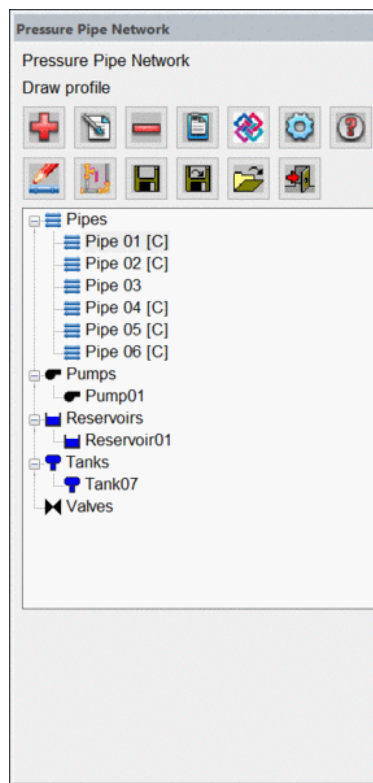
Pulldown Menu Location: Network

Keyboard Command: utilnet


Prerequisite: None


Pressure Pipe Network


Executing the command docks a dialog to the side of the drawing.






The top of the docked dialog box has thirteen buttons. Hover over each icon to see its name. Each button is described as follows:

1.  Add Pressure Pipe Element
 - (a) Pipe
 - (b) Pump
 - (c) Reservoir
 - (d) Tank
 - (e) Valve


2.  Edit Pressure Pipe Element
 - (a) Pipe
 - i. Edit Connection
 - ii. Edit Laterals
 - iii. Edit Attributes
 - (b) Pump
 - (c) Reservoir
 - (d) Tank
 - (e) Valve

3.  Remove Pressure Pipe Element

4.  Report

5.  Output IFC File
6.  Settings
 - (a) General
 - (b) Plan View
 - (c) Pipe Labels
 - (d) Connection Labels
 - (e) Color
 - (f) Attributes
 - (g) Crossings
 - (h) Defaults

7.  Help

8.  Draw
 - (a) 3D Polylines
 - (b) 3D Faces
 - (c) Plan View
 - (d) Profile

9.  EPANet

10.  Save

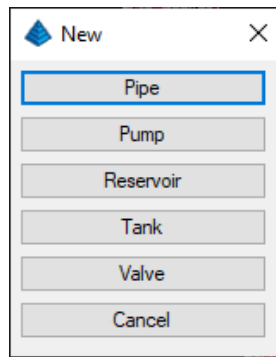
11.  SaveAs

12.  Load/New

13.  Exit

Add Pressure Pipe Element

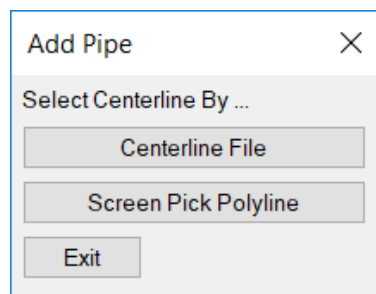
Clicking the **Add** button displays the **New** dialog box prompting the user to select an element to create.



Add Pipe
Add Pump
Add Reservoir
Add Tank
Add Valve

Add Pipe

Add Pipe dialog box prompts the user to select a Centerline file or to screen pick a polyline for a pipe. To add other entities, click the component heading, right-click to add. **Add Pipe** may also be accessed by right-clicking "Pipes" in the Pressure Pipe Network tree or by right-clicking on any existing pipe in the network.



Edit Pipe

Once the choice has been selected, the **Edit Pipe** dialog box appears. This box can also be accessed by clicking the **Edit** button or by right-clicking on an existing pipe in the network and choosing "Edit Pipe".

The **Pipe Name** is the name of the pressure pipe line. Each name must be unique in the pressure pipe network.

The **Pipe Dim1** is the exterior diameter of the pipe. Carlson currently supports only round pipe in a pressure pipe network.

The **Pipe Thickness** is the thickness of the pipe's wall.

The **Min Cover** is the minimum depth for the pressure pipe profile. This value is used to create the profile and keep it below the reference surface. Also, the Report function has a report for checking the min cover.

The **Roughness Coefficient** is used to calculate friction loss in the pressure pipe network and is specific to the pipe material and the **Analysis Method** described below. Click the Set button to display a list of pipe materials with associated roughness coefficients.

The **Plan View Linetype** is used for drawing the pressure pipe line in the Draw Plan View function.

The **Layer** settings are used for drawing the pressure pipe line in the drawing.

The **Profile Elevation Reference on Pipe** controls the pipe position in the pressure pipe profile with respect to the reference surface and the Min Cover value.

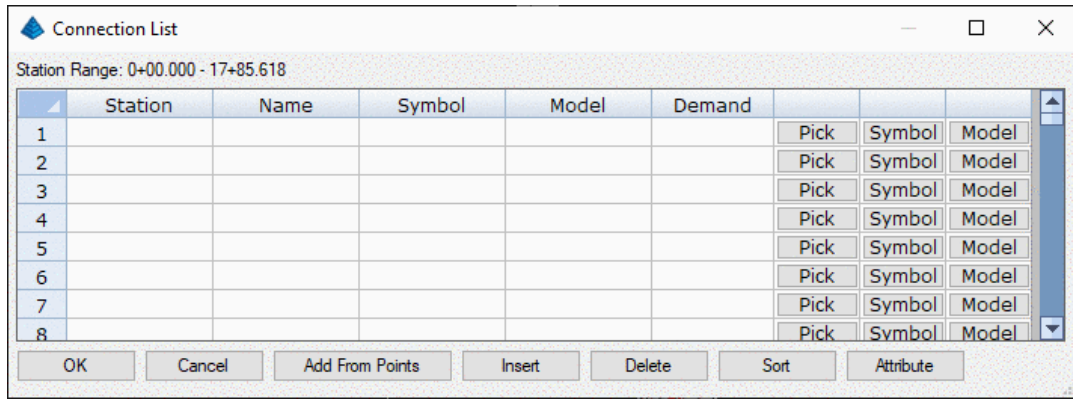
The **Centerline** defines the horizontal alignment of the pressure pipe line.

The **Profile** defines the vertical alignment for the pressure pipe line.

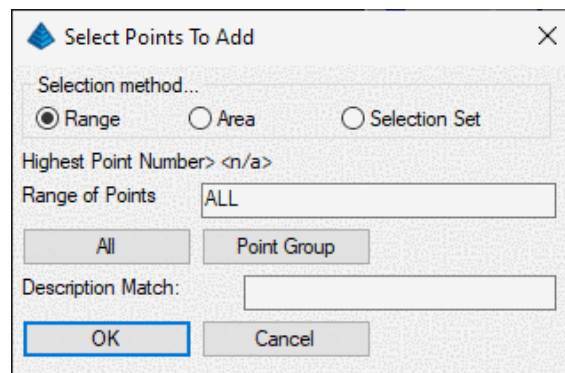
Edit Connections

The **Edit Connections** is an optional list of stations along the pressure pipe line where you can set a name,

a 2-D drawing Symbol, a 3-D drawing Model, and a Demand for pressurized water at this Station.



The **Add From Points** button displays the following dialog and allows for the creation of pipe connections from a list of points.

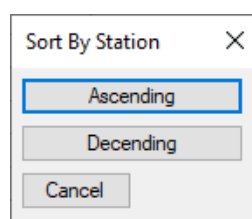


The **Add Multiple** button allows creating many connections at a time. A dialog prompts for the number of connections, starting station, station interval, name, symbol and demand. All the new connections will be created with the same name, symbol and demand which you can edit in the spreadsheet.

The **Insert** button allows the user to enter a new Station, a name for the new Station, 2-D Symbol, 3-D Model, and water Demand at the new Station in volume per time.

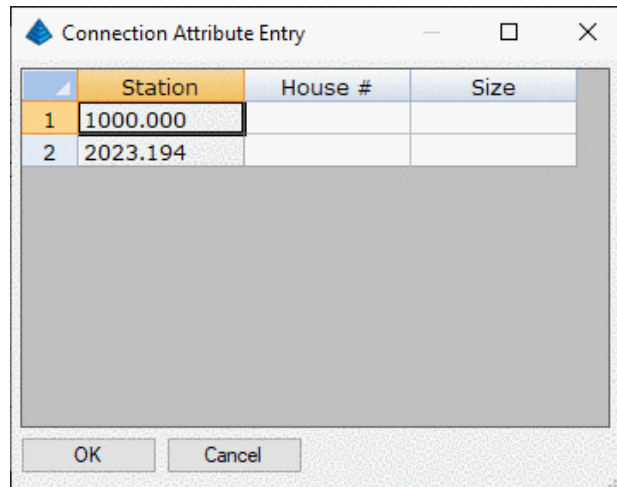
The **Delete** button removes the currently highlighted connection from the Connection List.

The **Sort** button displays the following dialog:



To sort the connection list by ascending station values click Ascending. To sort the stations in descending order, click Descending.

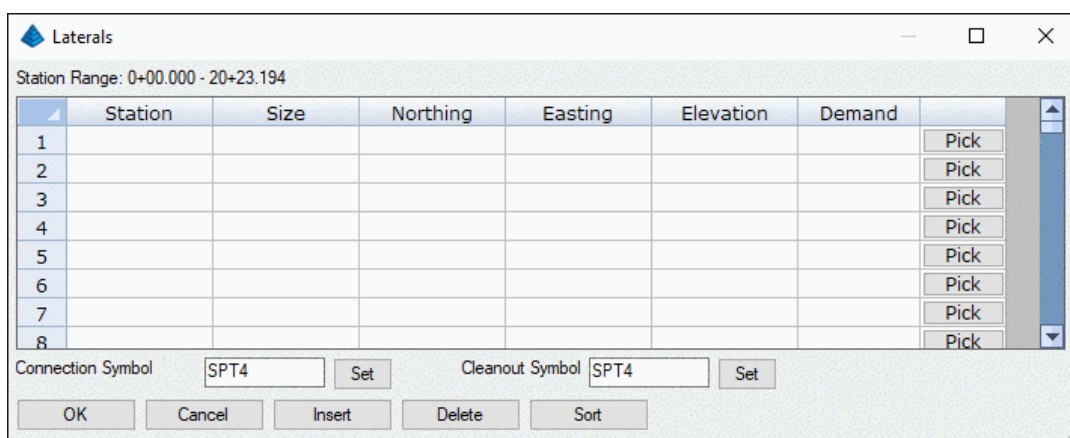
The **Attribute** button is only visible if connection attributes have been defined in the **Attributes** tab in the **Settings** dialog. When the **Attribute** button is clicked a dialog similar to the following is displayed:



A separate row for each defined station will be displayed. The number of columns displayed is based upon the number of connection attributes that have been defined in the **Attributes** tab in the **Settings** dialog. Enter the desired attribute value in each of the fields.

Edit Laterals

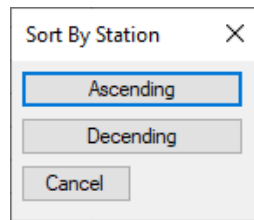
The **Edit Laterals** is an optional list of stations along the pressure pipe line where Laterals are connected. A Lateral is defined by a Station, pipe size, Northing and Easting location of the Cleanout, Elevation of the Cleanout, and water Demand of the Lateral. Use the **Pick** button to screen pick the connection and cleanout points, or to pick an existing line for the lateral. The Connection and Cleanout Symbols are used for drawing the lateral in plan view.



The **Insert** button allows the user to enter a new lateral on the pipe being edited.

The **Delete** button removes a lateral from the pipe being edited.

The **Sort** button displays the following dialog:

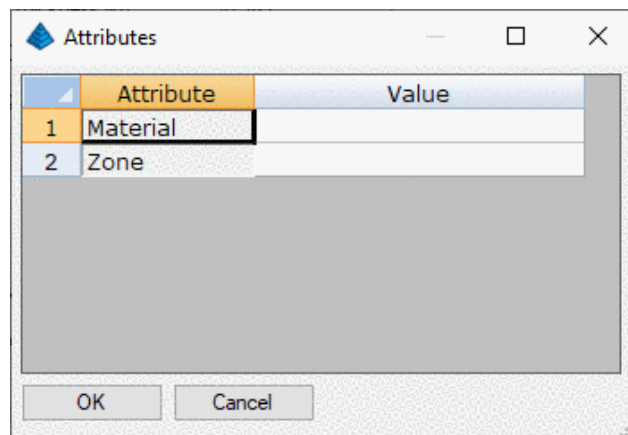


To sort the laterals station list by ascending station values click **Ascending**. To sort the stations in descending order, click **Descending**.

The **Add Multiple** button allows creating many laterals at a time. A dialog prompts for the number of laterals, starting station, station interval, size, offset and demand. All the new laterals will be created with the same size, offset and demand which you can edit in the spreadsheet.

Edit Attributes

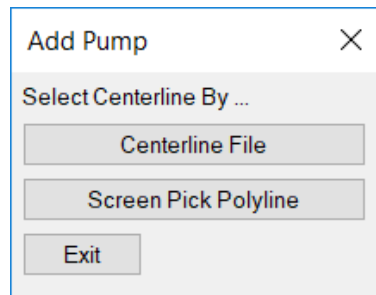
The **Edit Attributes** button allows the assignation of attributes, previously defined on the **Attributes** tab in the **Settings** function, to the pipe. These **Attributes** are used in reporting and drawing of labels. The **Water** attributes will be displayed in the **Attributes** dialog for adding or updating. If no **Water** attributes have been defined in **Settings**, then a message will be displayed.



Add Pump

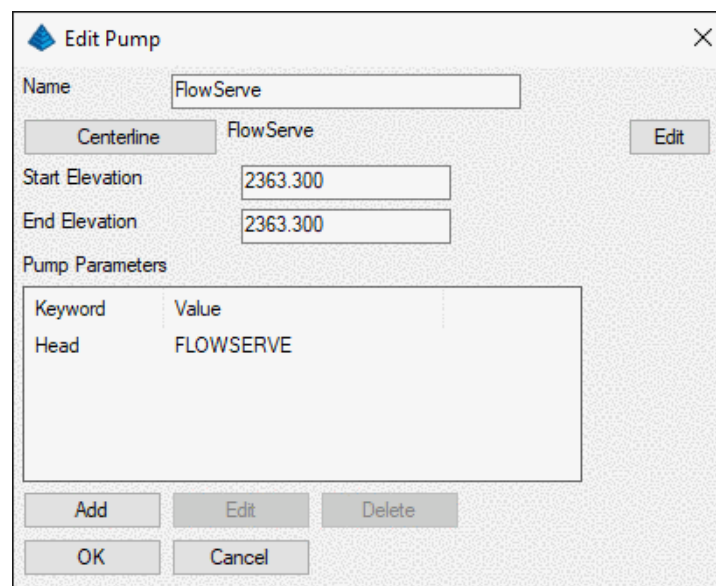
To add a Pump, highlight the heading **Pumps** in the Pressure Network tree, right-click and select **Add Pump**. Highlighting an existing pump and right-clicking allows the choices **Add Pump**, **Edit Pump**, and **Delete Pump**.

When **Add Pump** is selected, the **Add Pump** Dialog Box is displayed, allowing the user to select a Centerline file or the selection an existing polyline on-screen.

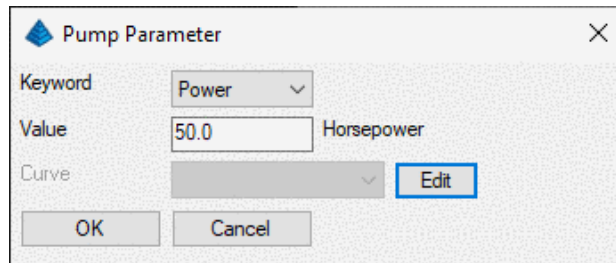
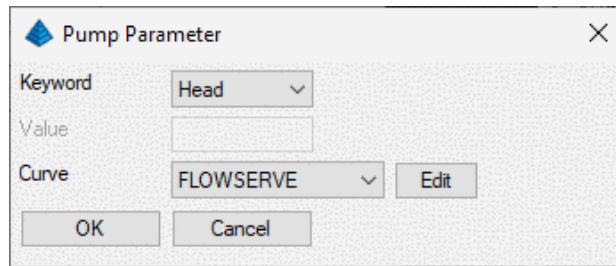


Edit Pump

Once this choice is made, the **Edit Pump** dialog box appears.



Name is the user defined name of the pump. Change the centerline file by clicking the **Centerline** button. Clicking the **Edit** Button next to the centerline file name allows the editing of the chosen centerline file. The **Start Elevation** and **End Elevation** fields are used to set the pump's elevation in the system. These values are typically equal, but not necessarily. The **Pump Parameters** define the pump's operation. The **Add**, **Edit** and **Delete** buttons below the **Pump Parameters** list allow adding and editing of pump parameters. The **Edit** and **Delete** buttons are only enabled when a pump parameter is selected in the **Pump Parameters** list. To add a new pump parameter press the **Add** button which will display the following dialog:



A pump must be defined using either a **Power** or a **Head** parameter but never both. When the **Power** keyword is selected the **Value** field is enabled where a constant energy in Horsepower or Kilo-Watts is entered, producing the same energy requirement for any combination of water volume and pressure. When the **Head** keyword is selected the **Curve** field is enabled and the **Value** field is disabled. A pump **Curve** plots the relationship between flow volume and head. Curves can be created or edited using the **Edit** button. For more details about curves, see the **Add/Edit EPANet Curves** section. **Speed** modifies the pump speed, which is 1.0 for normal speed. Double speed would be 2.0 and half speed would be 0.5. To shut the pump off, set the speed to be 0.0. **Pattern** allows the selection of a pattern curve which allows variable pump speeds through time. Once a **Keyword** and **Value** or **Curve** are selected, click **OK** to save the parameter.

Highlight a pump parameter in the **Pump Parameters** list and click **Edit** and the pump parameter dialog will be displayed for editing. Double-clicking on a pump parameter will also bring up the pump parameter dialog for editing.

Highlight a pump parameter in the **Pump Parameters** list and click **Delete** to remove the parameter.

Add Reservoir

To add a **Reservoir**, highlight the **Reservoirs** heading in the Pressure Pipe Network tree, right-click and select **Add Reservoir**. Highlighting an existing Reservoir and right-clicking allows the choices of **Add Reservoir**, **Edit Reservoir**, or **Remove Reservoir**.

When Add **Reservoir** is selected, the Add **Reservoir** Dialog Box is displayed.

Select the **Reservoir** location by clicking the arrow in the dropdown box next to "Pipe". **Reservoirs** may also be connected to pumps. A pump may be connected to only one reservoir and will disappear from the pipe and pump list if already assigned to a reservoir. Select whether the reservoir is at the **Start** or **End** of the pipe or pump. **Name** the reservoir. Set the elevation of the **Hydraulic Head**.

Edit Reservoir

To edit a reservoir, select an existing reservoir name, right click and choose **Edit Reservoir**, causing the **Edit Reservoir** dialog box to appear.

Change values as needed, click OK to save changes.

Add Tank

To add a **Tank**, highlight the **Tanks** heading in the Pressure Pipe Network tree, right-click and select **Add Tank**. Highlighting an existing Reservoir and right-clicking allows the choices of **Add Tank**, **Edit Tank**, or **Remove Tank**.

When Add Tank is selected, the Add **Tank** Dialog Box is displayed.

Clicking the arrow in the drop down box beside Pipe allows the selection of the component the tank attaches to. **Start** and **End** buttons are used to set the tank's location with respect to the component. **Name** the tank. The **Set Parameters By Pressure** button computes the tank parameters using a specified pressure. The **Bottom Elevation** is the tank's minimum gravity head, the bottom of an empty tank. **Initial Water Level** is the starting water level elevation of the analysis. The **Minimum Water Level** is the least amount of water desired in the tank entered as an elevation. **Maximum Water Level** is the maximum amount of water desired in the tank entered as an elevation. **Nominal Diameter** is the average cross-sectional area of the tank used to convert a change in water volume to the change in water level elevation in the tank. The **Minimum Volume** is the least amount of water allowed in the tank which should be equivalent to the **Minimum Water Level**. There is an option to assign a **Volume Curve** to the tank to model tanks of different shapes with cross-sections that vary by elevation.

Edit Tank

To edit a tank, select an existing tank name, right click and choose **Edit Tank**, causing the **Edit Tank** dialog box to appear.

Change values as needed, click OK to save changes.

Add Valve

To add a **Valve**, highlight the **Valves** heading in the Pressure Pipe Network tree, right-click and select **Add Valve**. Highlighting an existing Reservoir and right-clicking allows the choices of **Add Valve**, **Edit Valve**, or **Remove Valve**.

When **Add Valve** is selected, the **Add Valve** Dialog Box is displayed, allowing the selection of a Centerline file or the selection of an existing polyline on-screen.

Edit Valve

After selection, the **Edit Valve** dialog box appears.

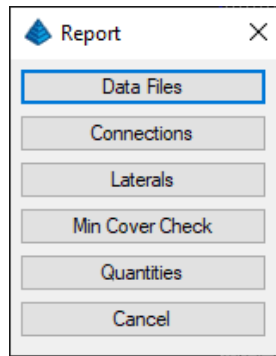
Name the Valve. Clicking the **Centerline** button allows a choice of a new **Centerline** file. Clicking the **Edit** button here allows the editing of the **Centerline**. Enter the **Start Elevation** and **End Elevation**. Assign a valve **Type**. A **Pressure Reducing Valve**, **Pressure Sustaining Valve**, or **Pressure Breaker Valve** require a **Setting** pressure. A **Flow Control Valve** requires a **Setting** of limiting volume. A **Throttle Control Valve** requires a **Setting** of a loss coefficient supplied by the manufacturer of the valve. A **General Purpose Valve** requires the application of a **Headloss Curve**. To edit or create a new curve click **Edit** and for more details about curves, see the **Add/Edit EPANet Curves** section. A **Diameter** in inches or mm of the valve is required for all valves. A **Loss Coefficient** may be applied to any valve or may be zero.

The **Edit** button edits the highlighted element from the pressure pipe network.

- Pipe
- Pump
- Reservoir
- Tank
- Valve

The **Remove** button deletes the highlighted element from the pressure pipe network. If a valid element is selected, then you will be prompted to verify the element removal with the following dialog:

The **Report** button displays the Report dialog box.



The **Data Files** Report displays the files and parameters for each component of the pressure pipe network.

The **Min Cover Check** Report displays the Reference Surface and .util files used, and, for each pipe, the minimum cover as well as the average cover elevation.

The **Quantities** displays the Carlson **Report Formatter**, allowing the user to create custom reports using pressure pipe network information, import additional information, present the information in a variety of different formats, and export the report for other use.

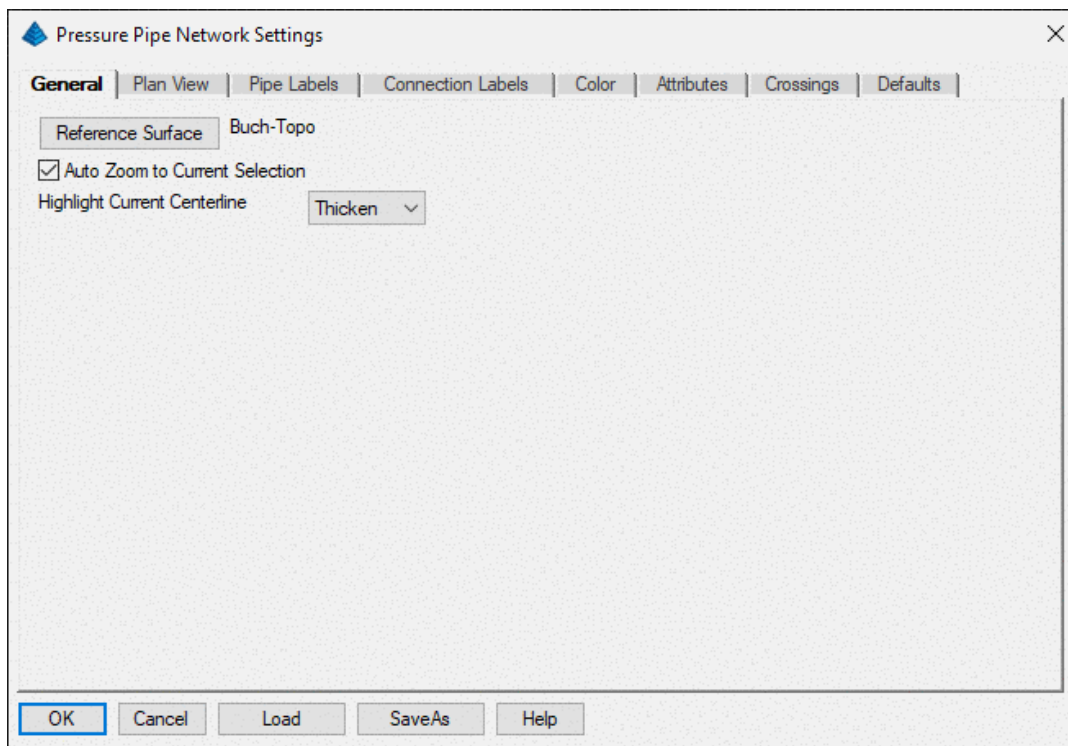
Output IFC File

The **Output IFC File** button creates a file that can be used by other software to import the 3-D model of the pressure pipe network.

Settings

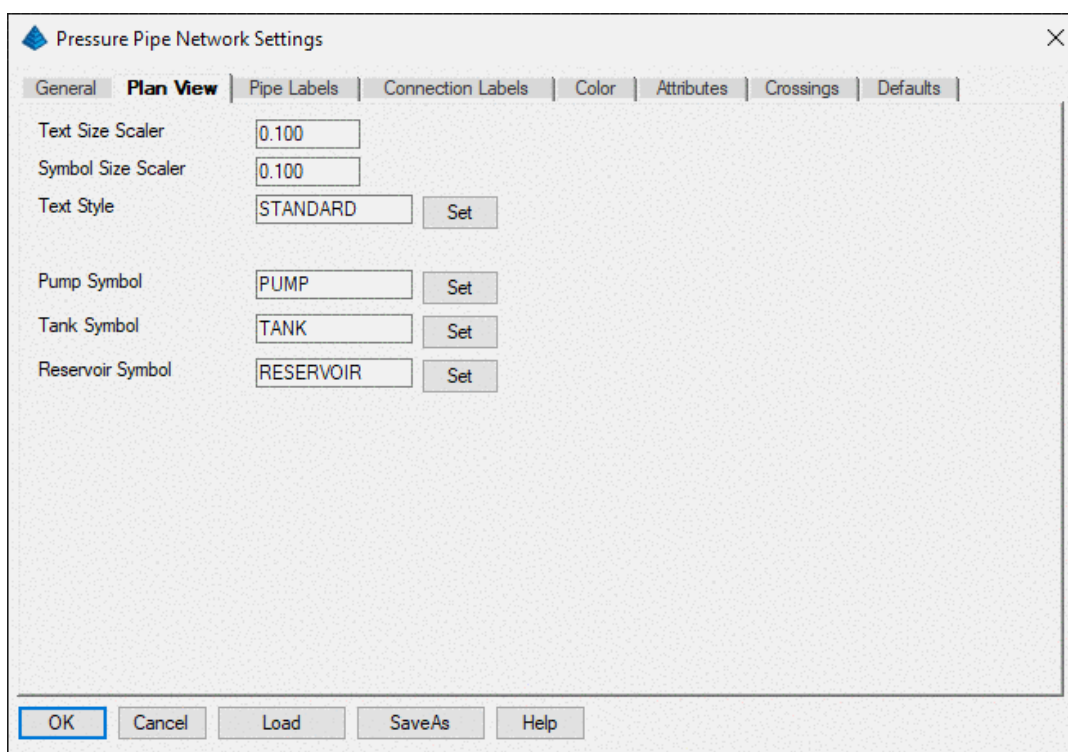
The **Pressure Pipe Network Settings** button displays a multi-tabbed dialog box, defaulting to the "General" tab. All pressure pipe network settings from all tabs can be saved to a file using the **SaveAs** button. The saved settings are stored in a .uns file. Any previously stored settings can be loaded using the **Load** button. The new settings will not be stored into the .util file until you press the **Save** button on the main docked dialog.

Setting General Tab



The **Reference Surface** is a triangulation file that is used for the min cover calculations and as a reference when editing pressure pipe profiles. The **Auto Zoom to Current Selection** check box finds and zooms to the component highlighted in the pressure pipe network tree. **Highlight Current Centerline** provides visual choices for in-drawing indication of the pressure pipe network component which has been selected.

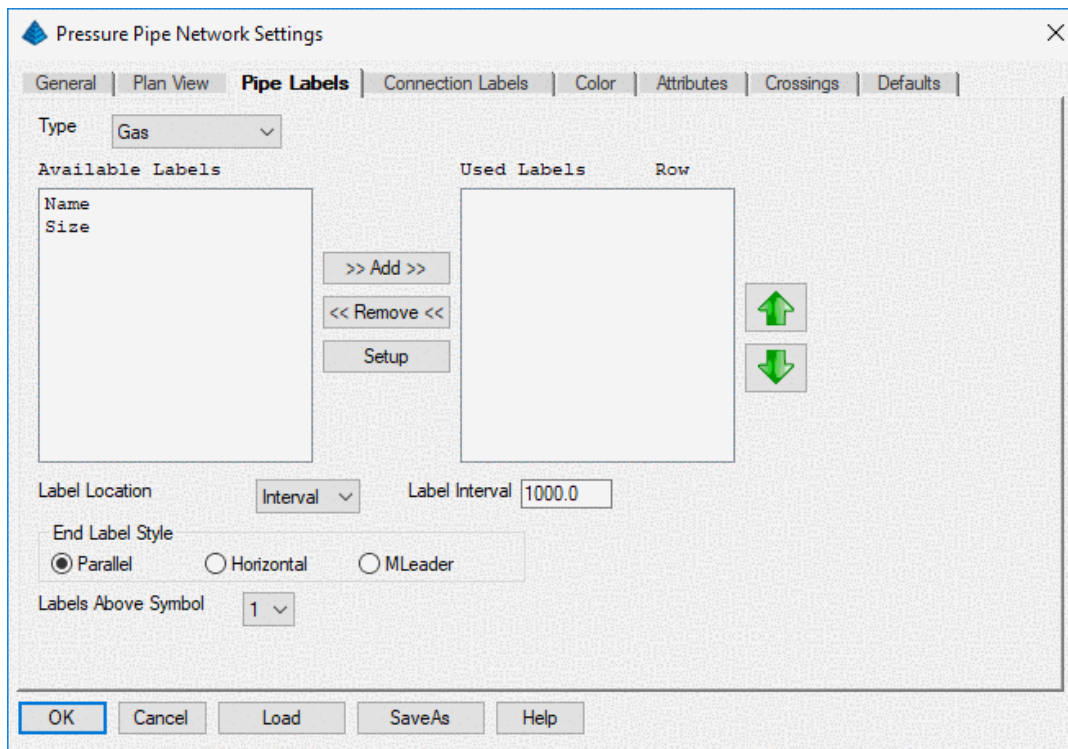
Setting Plan View Tab



On the "Plan View" tab, the **Text** and **Symbol Size Scalers** are used to set the size of pressure pipe network text

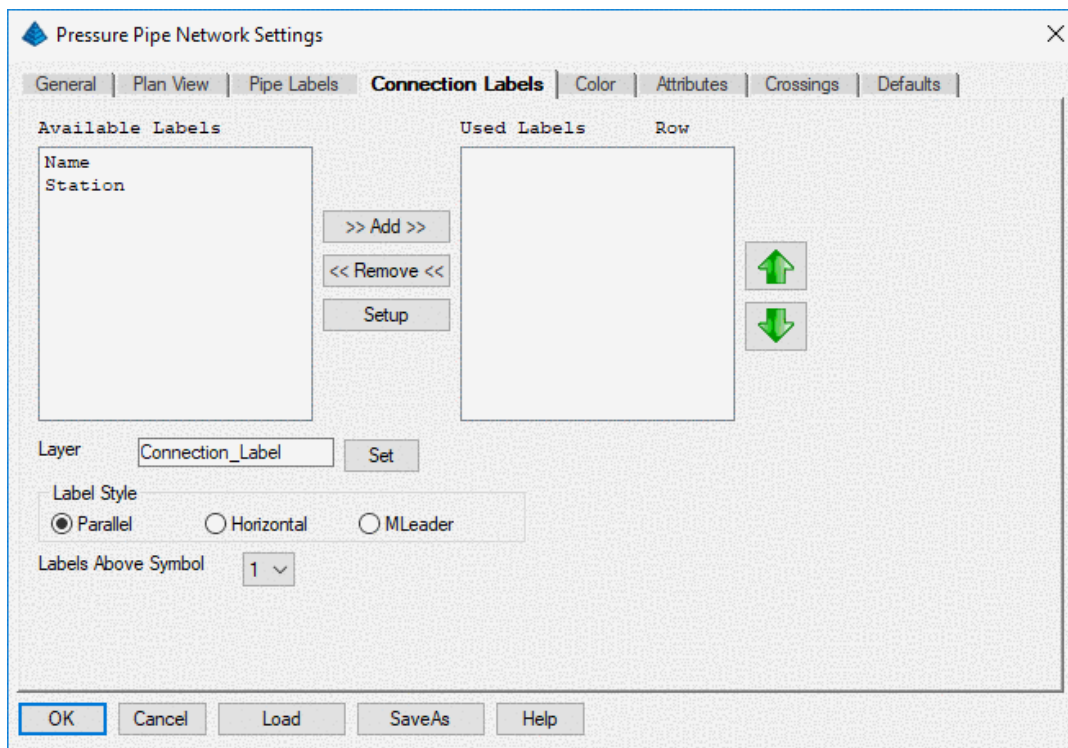
and symbols for the Draw Plan View function. Clicking the "Set" button next to the **Text Style** field controls the font used for the pressure pipe network. **The Label Name** dropdown box allows the user to place the pressure pipe network component label at either end, at intervals defined in the **Label Interval** field, or choose "None" for no labels on components. Clicking the "Set" button beside **The Pump, Tank, and Reservoir Symbols**, selects the 2-D plan view symbols used for these components.

Settings Pipe Labels Tab



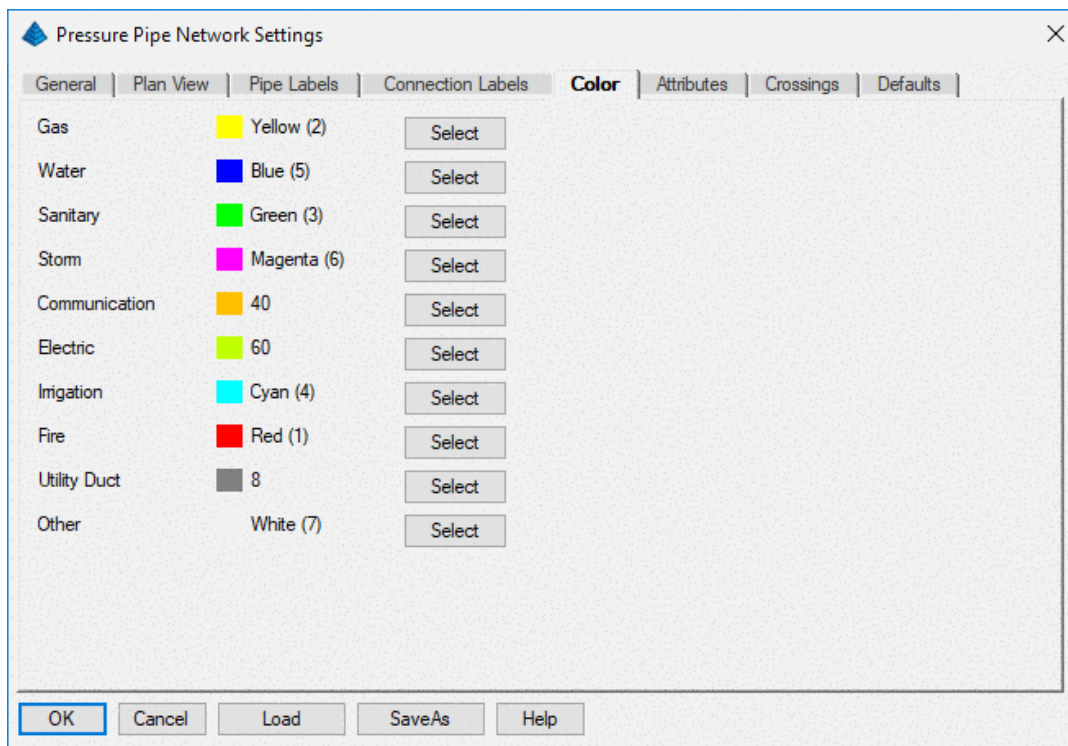
On the "Pipe Labels" tab, choose the Pressure Pipe **Type** from the dropdown, then select the **Available Labels**, moving them to be included using **Add**, then use the green arrows to reorder selections. **Label Location** may be None, Ends, Interval or Both. If Interval or Both is selected, set the **Label Interval**, then the **End Label Style**. **Labels Above Symbol** places the first X number of labels in the **Used Labels** box above the line, the rest below.

Settings Connection Labels Tab



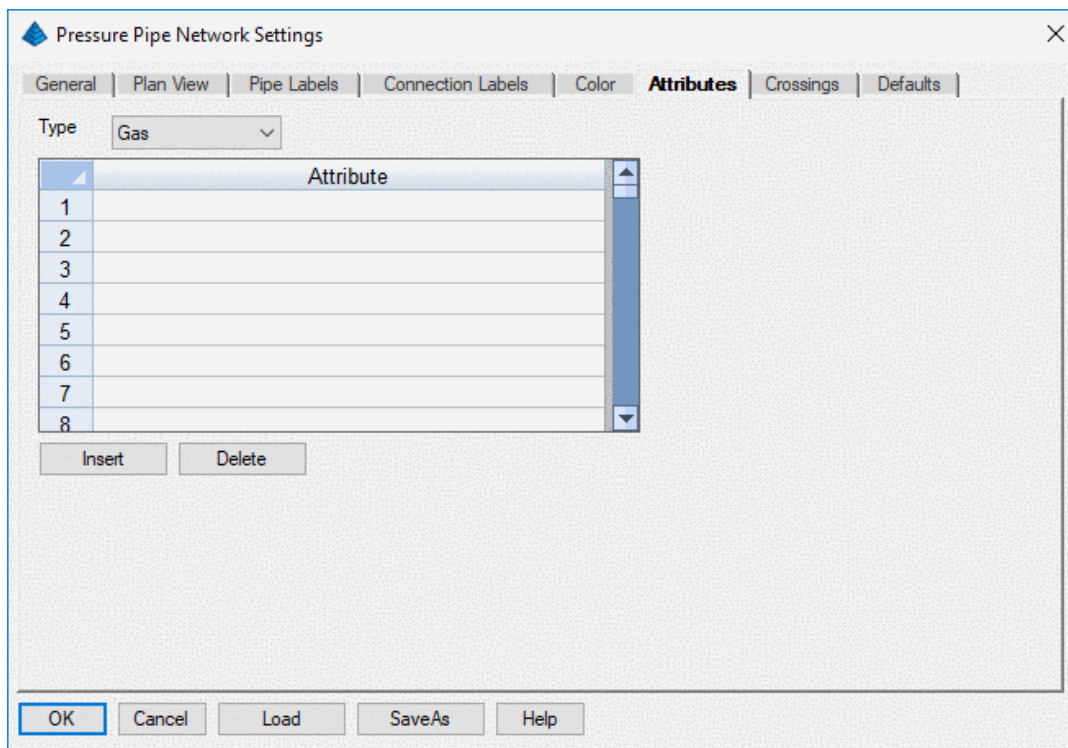
On the "Connection Labels" tab, select the **Available Labels**, moving them to be included using **Add**, then use the green arrows to reorder selections. Choose the layer for the connection labels. Choose the **End Label Style**. Labels Above Symbol places the first X number of labels in the **Used Labels** box above the line, the rest below.

Setting Color Tab



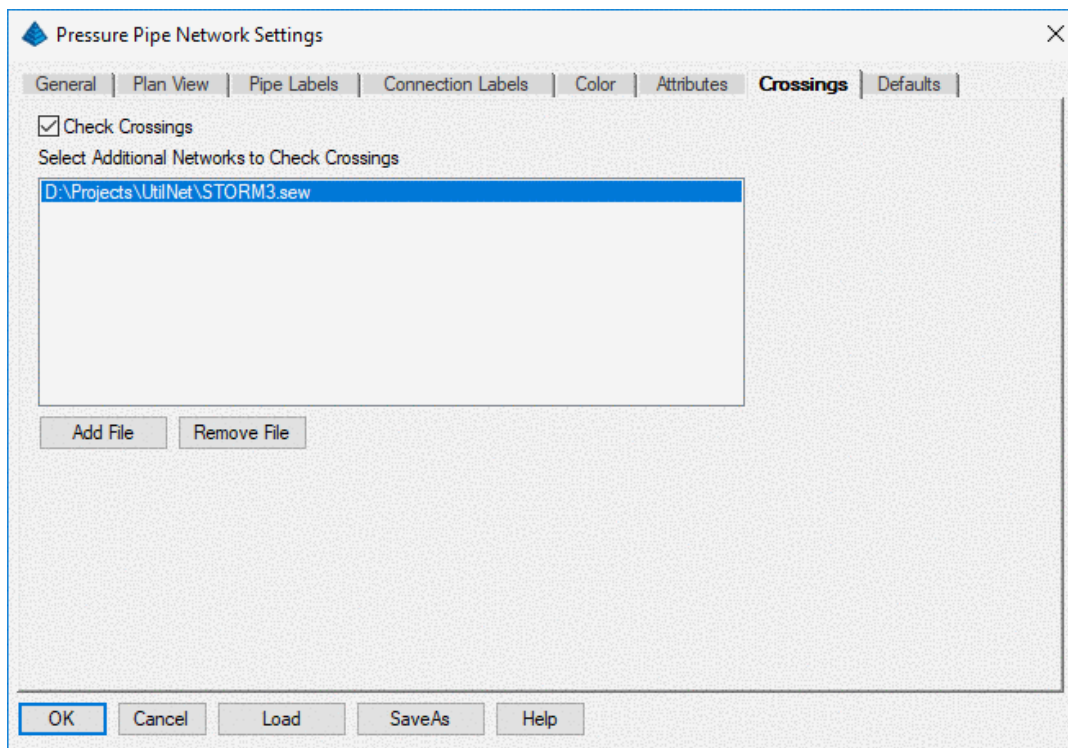
The "Color" tab sets the default colors to use when creating the layers used for drawing this and other Utility Networks.

Settings Attributes Tab



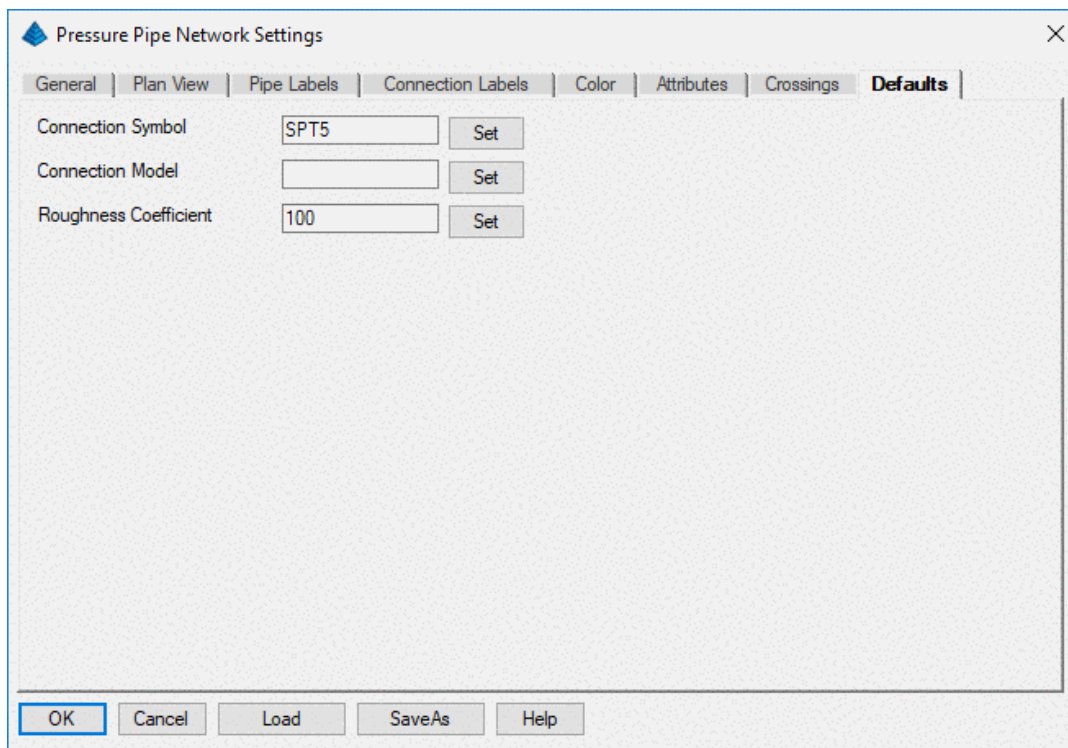
On the "Attributes" tab, choose the **Type** then add any number of **Attribute** values or descriptions to associate with any components in the pressure pipe network. These can be displayed on the **Quantities Report**.

Settings Crossings Tab



On the "Crossings" tab, select the **Check Crossings** box to activate the ability to check for crossings and collisions. Add files to be checked, .SEW or .UTIL files.

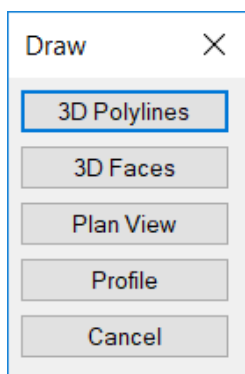
Settings Defaults Tab



On the "Defaults" tab, the initial 2-D **Connection Symbol** and 3-D **Connection Model** are defined using the "Set" buttons. These may be changed when adding or editing a Connection. Set a Manning n Roughness Coefficient for the connection.

The **Help** button displays this document..

Clicking the **Draw** button displays the **Draw** dialog box.



Draw 3D Polylines

The **3D Polylines** function creates a 3D polyline for each pressure pipe line. These polylines are tagged with their pipe size in the same format as the Assign Pipe Data to Polyline command.

Draw 3D Faces

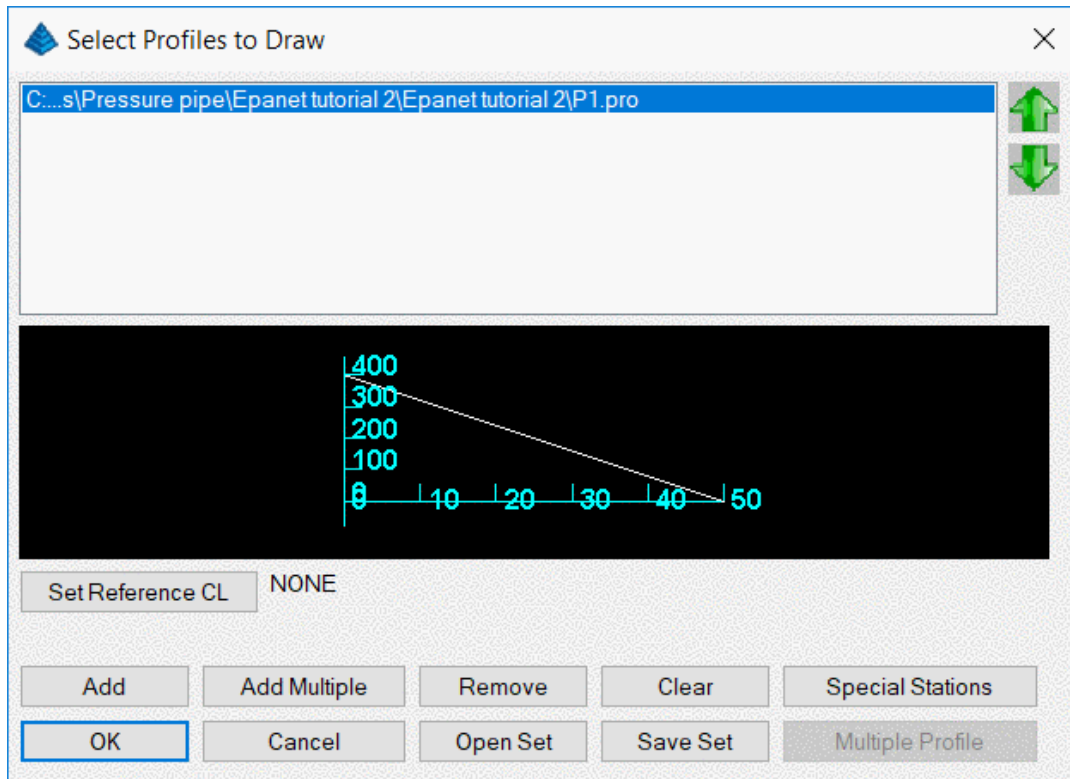
The **3D Faces** function draws each pressure pipe line as a 3D pipe using 3D faces. These 3D faces can then be used in the 3D Viewer commands.

Draw Plan View

The **Plan View** function draws a 2D polyline for each pressure pipe line and labels the line with its' name and connection points.

Draw Profile

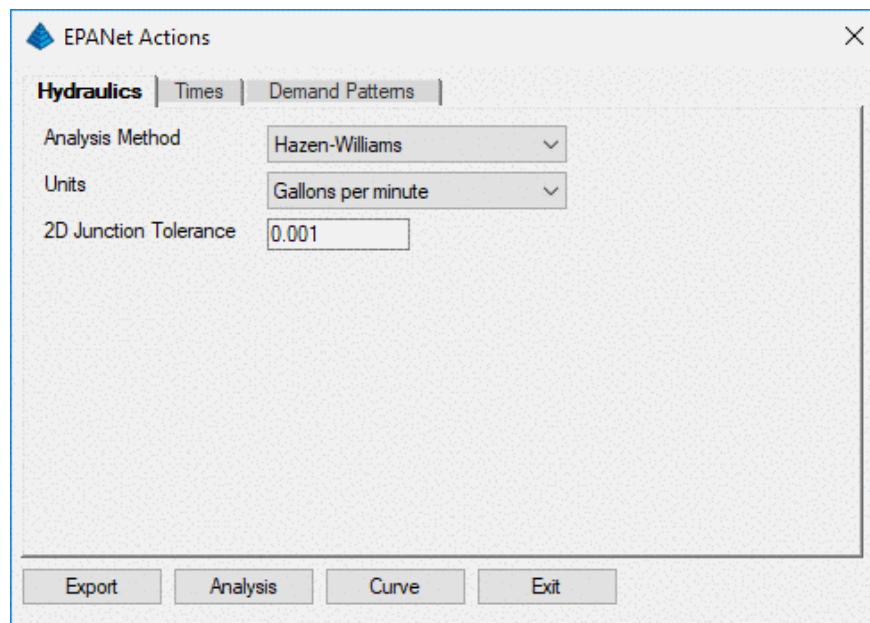
The **Profile** function displays the **Select Profiles to Draw** dialog box.



Set Reference Centerline allows the selection of a Centerline file to choose a Profile file. **Add** allows the selection of a Profile file. **Add Multiple** allows the selection of more than one Profile file choice at a time. **Remove** deletes a Profile from the list. **Clear** removes all Profiles from the list. **Special Stations** allows Station marks and elevation information to be displayed at other positions on the profile than at the beginning, end, and connection points by adding a new Station. **Save Set** saves the list of profiles to a user defined file name for recall. **Open Set** opens a previously saved Set. **Multiple Profile** allows the manipulation of Profile files containing more than one Profile.

Selecting the **EPANet** button displays EPANet Actions dialog box and defaults to the "Hydraulics" tab.

EPANet Hydraulics Tab



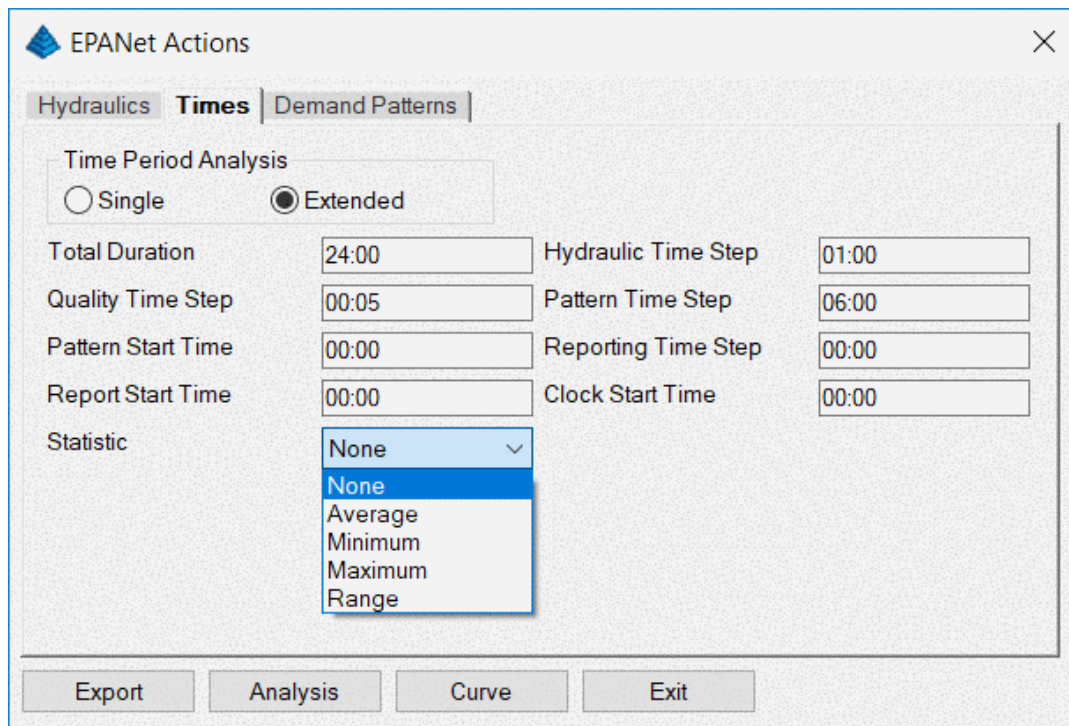
The **Analysis Method** dropdown box allows the selection of one of three methods supported by EPANet.

The **Units** dropdown box allows the selection of the desired volume per time unit.

The 2D Junction Tolerance allows the user to insert a 2D distance. Like pressure pipe network entities within the 2D tolerance will be joined even if the 3D elevation does not exactly match.

EPANet Times Tab

The "Times" tab allows setting parameters for either a single time period or an extended analysis.



If **Time Period Analysis - Single** is selected, all other fields on the tab are deactivated, setting up a "snapshot" analysis of the pressure pipe network.

If **Time Period Analysis - Extended** is selected, all fields on the tab are available for edit. **Total Duration** is the total length of time to be analyzed. Water quality is not currently being analyzed in Carlson 2024. **Quality Time Step** is not currently used. **Pattern Start Time** is the time the **Demand Patterns** created and listed on the "Demand Patterns" tab starts. **Report Start Time** is the first time shown on the analysis. **Hydraulic Time Step** is the time unit incremented as the analysis proceeds through the **Total Duration**. **Pattern Time Step** is the time unit used to step from one **Demand Pattern** to the next. **Reporting Step Time** defaults to the **Hydraulic Time Step** unless and different increment is assigned in this field. **Clock Start Time** is when the analysis begins. The **Statistic** controls how information is presented in the **Analysis** report. **None** reports data at each **Reporting Time Step** increment or, if zero, the **Hydraulic Time Step**. **Average** reports the average values found for the **Total Duration**. **Minimum** reports the minimum value, **Maximum** reports the maximum value and **Range** reports minimum and maximum values across the **Total Duration**.

EPANet Demand Patterns Tab

The "Demand Patterns" tab allows the weighting of water demand assigned to connections and laterals when creating the pressure pipe network based on time of day.

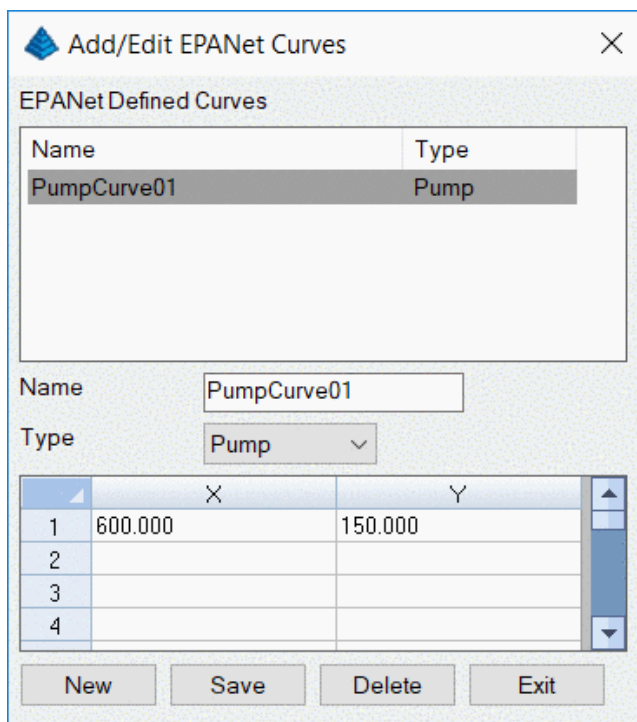
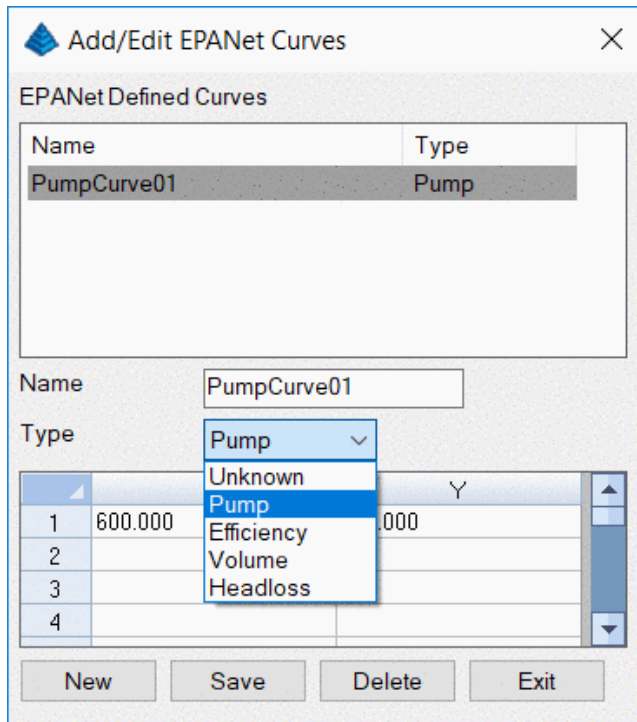
Period	Multiplier
1	0.500
2	1.300
3	1.000
4	1.200
5	
6	

Select an existing pattern to be used to weight the demand during different times of the day. To add a new Pattern, pick the Add button. Another dialog is shown to enter the parameters. Type a new **Pattern ID**, a new **Description**, desired **Multipliers** and click OK.

In the example above, the Clock and the Pattern start at midnight and the demand is reduced by the **Multiplier** 0.50 until the **Pattern Time Step** increment comes due at 6am, increasing the demand by 1.3. At noon, demand drops to its' default values, then at 6pm demand is increased by a factor of 1.20. Note that the **Pattern Time Step** and the number of **Multipliers** are synchronized.

Add/Edit EPANet Curves

Next, develop a pump curve by clicking the **Curve** button. By specifying the **Type**, the values in the X and Y columns are defined.



The Curve Type **Pump** develops a single-point or multi-point **Pump Curve** plotting volume per unit time (flow rate) in the X column, head in the Y column. An **Efficiency Curve** plots a pump's flow rate in the X column vs. energy use efficiency as a percentage in the Y column. A **Volume Curve** plots flow rate in the X column vs. the change in elevation (head) in a tank and is used to model tanks with non-uniform cross sections. A **Headloss Curve** plots the flow rate in the X column vs. the headloss in feet or meters in the Y column as water flows through a General Purpose Valve.

The **Analysis** button produces a report that can be viewed, saved, printed and search showing demands, flow rates, head, energy consumption, pressures, velocities and headlosses in the Network, either as a snapshot or with a **Demand Pattern** applied.

The **Export** button creates a file which can be imported into EPANet.

Save

The **Save** button saves any changes made to the current .util file.

SaveAs

The **SaveAs** button prompts for a new .util file and saves the current pressure pipe network and settings .

Load/New

The **Load/New** button prompts for an existing .util file or the file name for a new .util file and loads the existing file or initializes a new file.

Exit

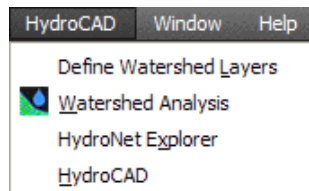
The **Exit** button exits the Pressure Pipe Network command. If changes have been made, a prompt will be displayed asking if you want to save or discard the changes.

Pulldown Menu Location: Network

Keyboard Command: epanet

Prerequisites: Centerline files, Profile files and/or polylines representing pipes, pumps, and valves.

HydroCAD



The HydroCAD Menu is provided as a starting point for 3 stormwater design tools. The first tool is the Watershed Layer tool. This is the same tool used in Stormwater Networks that provides a method to calculate runoff coefficients. Please note changes to the runoff layer settings changes the settings for all Carlson Tools. For example, when designing a collection system with the Rational Method after designing a detention pond with HydroNet, the watershed definition must be changed from the Curve Number Method to the Rational Method. Stormwater Collection System design may use the Rational Method whereas stormwater detention calculations use the Curve Number Method. The modeling method type must be changed when It is recommended that settings files be saved so that when switching methods, the only requirement is to load your settings files. For additional information on Watershed Layer Setup, see the Layer Setup process covered in Storm Networks.

The second tool is the watershed analysis tool. This tool generates watershed delineations and travel paths given surface geometry. For additional information, see detailed instruction in the Storm Network Dialoge

The third tool opens Hydrocad which is an external program. Carlson provides this button for access to the Hydrocad application. For additional information on Hydrocad, visit their website at <https://www.hydrocad.net/>. Note that Hydrocad is very similar to Carlson's HydroNet program, uses a drag and drop dialogue and provides direct modeling support for most Stormwater detention systems.

Documentation on the Define Watershed Layers and Watershed Analysis commands is under the Watershed Menu topics.

HydroNet Explorer

HydroNet Explorer is a powerful stormwater modeling program that designs and analyze simple and complex watershed networks. In the program, you define a hydrologic network by building a collection of hydrologic nodes. There are four types of nodes: subcatchment, pond, reach and link.

The general idea for the use of the HydroNet Explorer is that you have already prepared a drawing for analysis.

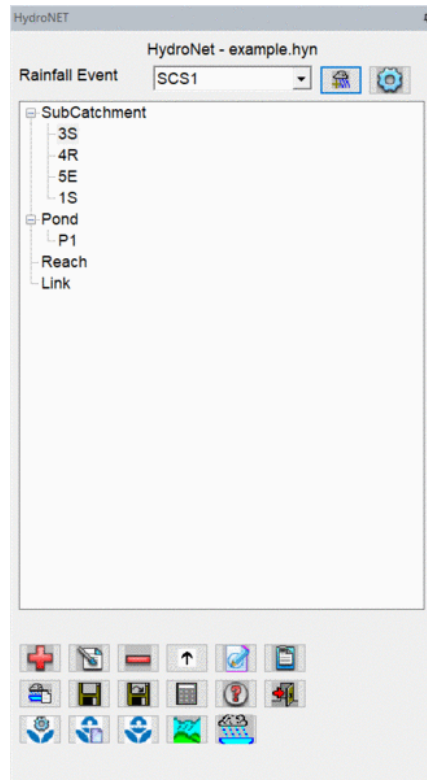
This preparation would include:

Soils: The boundaries of the Hydrologic Soils Groups should be drawn on the drawing layer specified in the Watershed Layers dialog, with the A, B, C, or D labels on the layer specified for that. These areas do not have to be closed polylines, as long as the linework encloses each area that is part of the study.

Ground Covers: The various Ground Covers in the study area should be drawn as closed polylines on the layers specified in the Watershed Layers dialog. These do not need to be labeled.

Watersheds: The Watersheds (subcatchments) for the site should be drawn and labeled. It is best to do this with closed polylines, but it is not essential, as you can use an alternative method of picking within the area and having the software define it from drawing linework.

When you run the HydroNet Explorer command, you are prompted to create a new .HYN file or open an existing one. This is the file that stores all of the data about the components in the project.



Once the HydroNet Explorer is open, the first thing you should do is check the settings for the current project. Pick the icon with the GEAR on it. In the HydroNet Settings dialog, review the various tabs containing the settings for the different aspects of the project.

On the **General** tab, choose Runoff Method either Rational or SCS TR-20, if choosing SCS-TR20 select SCS Unit Hydrograph (options include Delmarva, SCS Curvelinear, SCS Triangular (Set Peak Attenuation Factor), Clark, Snyder) and then select Pond Routing Method (Stor-Ind), and Reach Routing Method (options include Stor-Ind, Stor-Ind w. Trans, Muskingum - Cunge, Modified Puls, and Channel Lag). Set time span. There are two methods to calculate watershed runoff: SCS TR-20 and Rational method. Set the surface to be used to provide the average slope for each subcatchment when using the Curve Number/Lag method.

On the **Subcatchment** tab, set the default starting number for subareas, the default method for Tc calculation, and also set whether Curve Numbers should be calculated in Carlson Hydrology (check the box) or in HydroCAD (uncheck the box).

On the **Pond**, **Reach** and **Link** tabs, set the default starting number for each of these components.

HydroNet Settings

General | SubCatchment | Pond | Reach | Link | Rainfall | Report

Runoff Method: SCS TR-20

SCS Unit Hydrograph: SCS Triangular Peak Attenuation Factor: 484

Pond Routing Method: Stor-Ind

Reach Routing Method: Muskingum-Cunge

Time Span

Start Time (Hours): 0.00

End Time (Hours): 24.00

Time Increment (Hours): 0.10

Surface File: C:\Carlson Projects\HydroCAD_EG.tin Set

Use Plan View Location for Schematic Location

Report Slope In: ft/ft %

Save Settings Load Settings OK Cancel Help

HydroNet Settings

General | SubCatchment | Pond | Reach | Link | Rainfall | Report

Default Number: 15

Area Units

Square Feet Acres

Tc Calculation Method

Curve Number/Lag TR-55

Process Curve Numbers/Runoff Coefficients

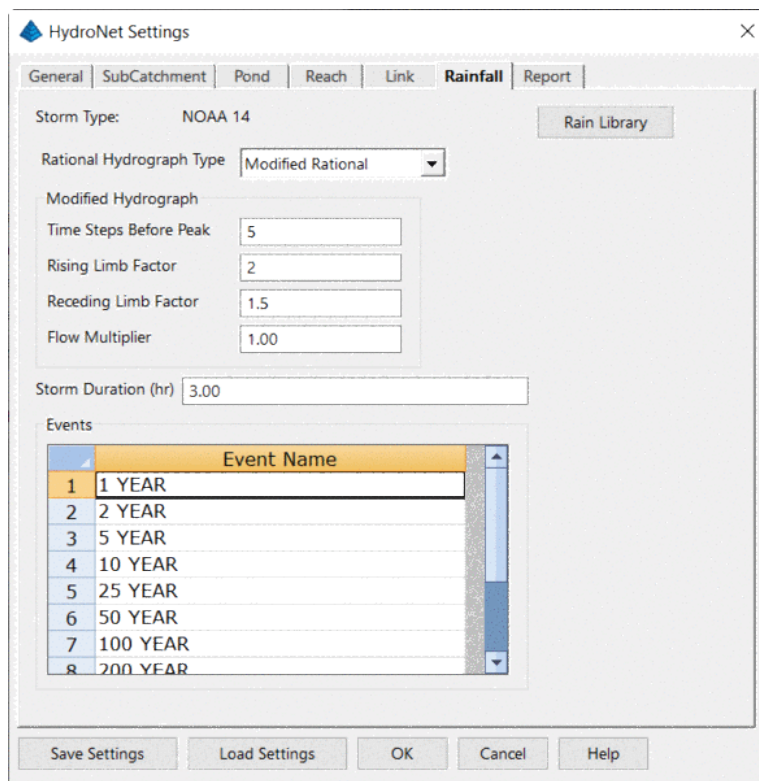
Use 3D Flowline Slope as Land Slope in Tc Calculation

Save Settings Load Settings OK Cancel Help

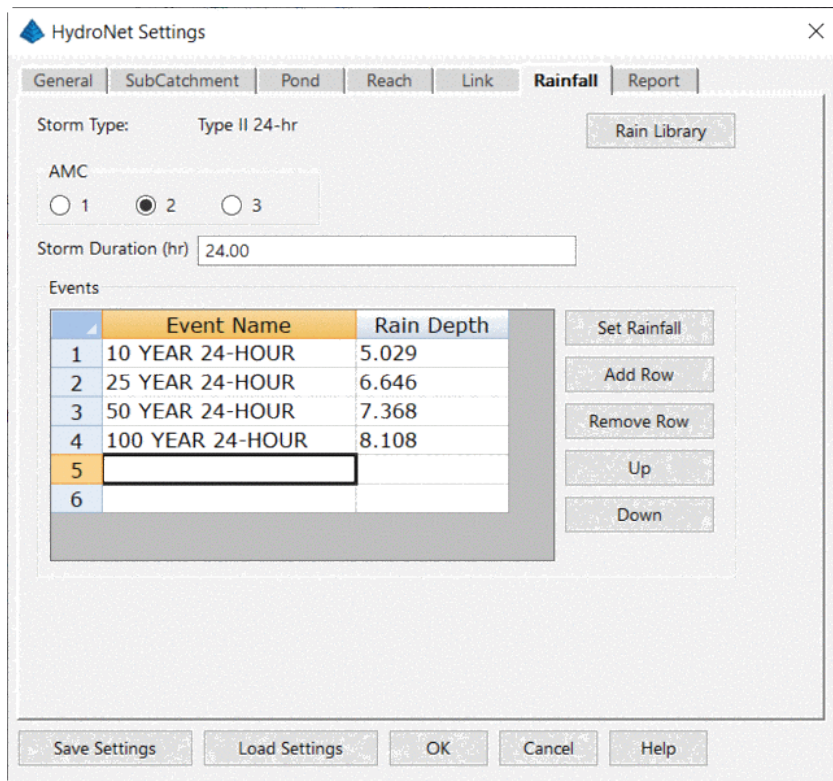
Rainfall Tab

If you want rainfall to be added to the calculations in Carlson Hydrology, set up the rainfall parameters here. Otherwise the rainfall data can be added in HydroCAD. In this dialog, you can enter multiple rain events for reporting. Detailed information about setting up Rainfall events in Carlson is under the Watershed section of the Carlson documentation. There is also an icon at the top of the HydroNet Explorer that you can use to import a Rainfall event from another HydroNet/HydroCAD project file.

There are three Rational Hydrograph Types in HydroNet: Modified Rational, Dekalb Rational and Universal Rational methods. Storm Duration is required for Modified Rational method.



Rational Rainfall



SCS Rainfall

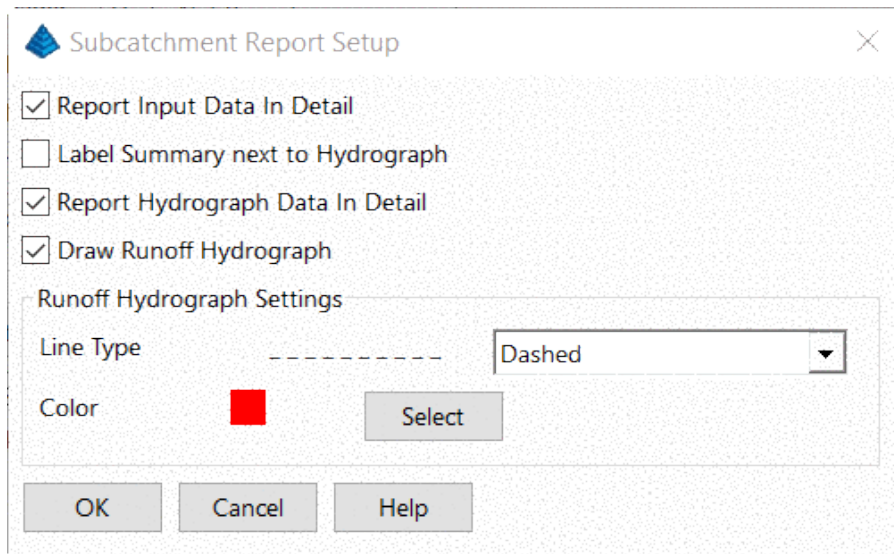
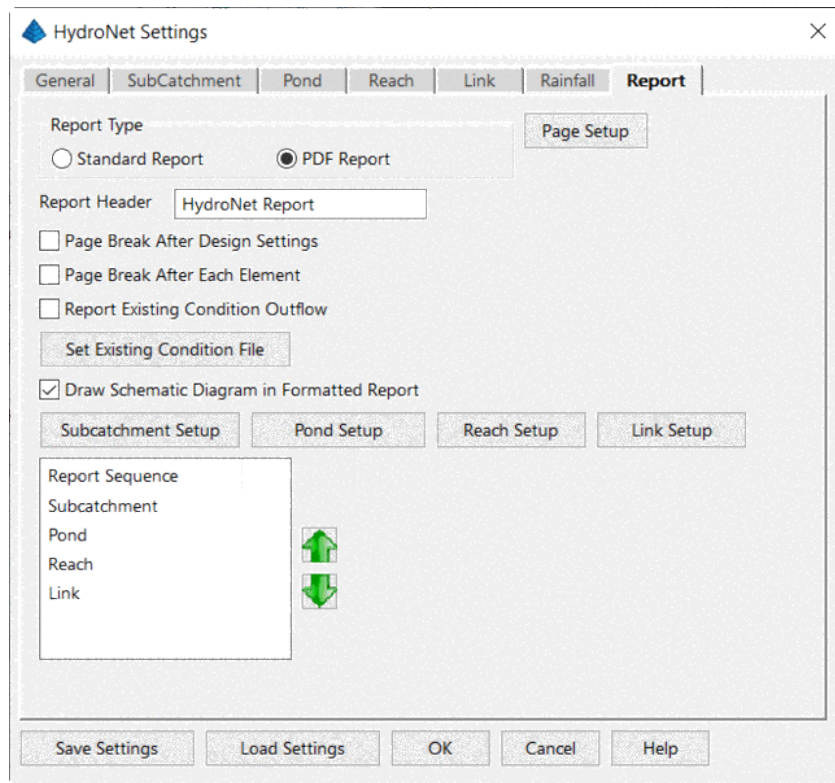
Report Tab

On the Report tab, establish the type of report and the details of the report that you want for each of the 4 component types. Each setup button allows for reporting options in detail, drawing option such as colors, and linetypes.

Report Existing Condition Outflow is to reference pre-developed runoff values when reporting proposed analysis results. **Set Existing Condition File** to to select a HydroNet file that has pre-developed watershed condition.

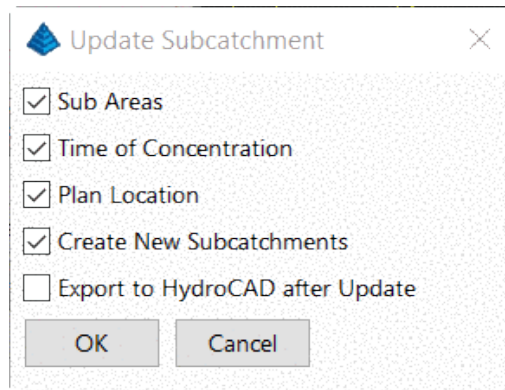
Subcatchment Setup, Pond Setup, Reach Setup, and Link Setup are used to define detailed reporting options with adjustable **Report Sequence** output. Use green arrows to adjust order.

Report Sequence: This is the order that the hydraulic components show in the report. The green Up and Down button allow users to change the report sequence.



Subcatchment contains the watershed conditions. Once the watershed layer file is defined by the command Define Watershed Layers, the program can automatically pull out the watershed data and fill all the values. There can be unlimited numbers of subcatchment in the network, which are divided by the watershed boundary polylines.

You can now use the HydroNet Explorer to automatically analyze the drawing and add all subcatchments defined in the drawing. Pick the Update button, and check all of the options. With the dialog set up as shown, when you pick OK, all subcatchments found on the specified layer are added to the Explorer and exported to HydroCAD. The detailed data for each subcatchment can now be viewed and/or edited. Double click on any subcatchment to edit.



To add Subcatchments manually, pick on the Subcatchment item in the Explorer, and either right click and pick add from the submenu or pick the Add icon.

In the Subcatchment dialog, enter the Number for the Subcatchment. If you have the Subcatchment labeled in the drawing, make the Number match the label. Pick the Edit button next to the Area. In the Sub Areas dialog, pick the Select Subcatchment button. Carlson Hydrology searches for a subcatchment on the layer specified in the Watershed Layers dialog that has a label that matches on the specified layer that matches the Number. If it finds one, it highlights it and asks you to confirm that this is the Subcatchment you are meaning to use. If you pick Yes, the SubAreas are calculated from all of the additional data in the drawing. Pick OK. Back in the Subcatchment dialog, if you specified the CN/Lag method for Tc, the Average Slope of the Subcatchment has been calculated and displayed, and the longest polyline within the subcatchment on the specified layer as been selected and it's length displayed. Define Baseflow (cfs) if needed. Pick OK. The new Subcatchment is listed in the Explorer.

Pond acts as a storage and routing element in the network, and Reach is a routing element. The upstream node of pond and reach generates a hydrograph. The procedure to add Ponds and Reaches is similar to Subcatchments, either right click and pick Add, or select the category heading and pick the Add button below. The detailed documentation on inputting data for these component types is found in the Watershed section of the Carlson Hydrology documentation.

A reach is any length of a stream, river, channel, or culvert. The term is often used by hydrologists when referring to a small section of a stream or river rather than its entire length. User can select Pick Inlet/Outlet Locations where you will be prompted to pick Reach Inlet Location or Select Reach 3D Polyline where you will be prompted to select a 3D polyline. Reach properties include Invert In, Invert Out, Length, and Slope. Reach Type can be set to Channel or Culvert with editing options for both.

Reach

Number: 2R

Name:

Inflow Available: [] Add Remove

Inflow Used: 1R

Outflow: None

Pick Inlet/Outlet Locations Select Reach 3D Polyline

Reach Properties

Invert In: 978.000 ft

Invert Out: 960.000 ft

Length: 576.000 ft

Slope: 3.125 %

Reach Type: Channel Culvert

Inlet Location: (0.00, 0.00)

Outlet Location: (0.00, 0.00)

Edit Channel Edit Culvert

Plan View Location

Northing: 0.000 Easting: 0.000 Location

Report Report Input Save Hydrograph OK Cancel Help

Reach Type can be set to Channel or Culvert with editing options for both.

Design Channel

Channel Type: Trapezoidal Define

Bottom Width (ft): 1.000

Left Side Ratio: 3.000

Right Side Ratio: 3.000

Diameter (ft):

Channel Depth (ft): 2.000

Profile Slope (%): 0.0000

Manning's n: 0.0130 Select

Calculation

Design Flow (cfs): Calculate Discharge

Flow Depth (ft): Calculate Depth

Parameter	Value	Unit
-----------	-------	------

OK Cancel Load Save Help

Design Channel

Culvert Design

Shape: Circular

Material: RCP III Library

Diameter: 12.00 in Library

Width: in

Thickness: 0.00 in

Manning's n: 0.0130

Number of Barrels: 1

Culvert Inlet: Square Edge, 30 to 75 Degree Wingwall Flares

Ke: 0.4000 Library

Culvert Control Type: Optimum

Solve For: Size

Calculation

Discharge: 1.68 cfs

Headwater Elev: 90.863 ft

Headwater Depth: 0.863 ft

Tailwater Elev: 0.000 ft

Nominal HW/D Ratio: 1.2

HW/D Result: 0.86

Outlet Velocity: 4.47 ft/s

Outlet Flow Depth: 0.483 ft

Control Type: Outlet Control, Gradually Varied Flow

Solve

Load Save OK Cancel Help

Design Culvert

Link is used to model a hydrograph or a sewer network. This program assumes that the hydrologic network is a

linear system, and allows two hydrographs to be superposed by adding the two given flows at a specific time step. In the network two or more hydrographs are combined at a node by allowing multiple links to discharge to one downstream node. The procedure to add is similar to Subcatchments, either right click and pick Add, or select the category heading and pick the Add button below.

If you are using HydroCAD in conjunction with Carlson Hydrology, once the elements of the study are added to the HydroNet Explorer, pick the Export to HydroCAD button to transfer the data to HydroCAD.



Any changes made in the drawing that affect any of the components of the study can be instantly updated and sent to HydroCAD with the Update button in the HydroNet Explorer. Also, each individual component can be updated alone with the update button within it's specific dialog box.



The components listed in the HydroNet Explorer can also be drawn into the drawing file with the Draw Layout in CAD button (paintbrush). Set the desired parameters in the HydroNet Draw dialog box.

HydroNet Draw Setup

General Subcatchment Pond Reach Link

Layer Name HYDRONET Select

Line Type BYLAYER Select

Text Style STANDARD Select

Draw Scale 50.000

Diagram Size Scale 0.100

Symbol Size Scale 0.100

Text Size Scale 0.100

Use MText

Draw Connection Lines

OK Cancel Load Save Help

HydroNet Draw Setup


General Subcatchment Pond Reach Link

Available Labels		Used Labels		Row
Name	>> Add >>	Number		1
	<< Remove <<	Area		2
	Setup	CN		3
		Tc		4
		Peak Discharge		5

Symbol Options

Default Symbol Text Box Custom Symbol

Symbol Name SPT5 Select

Symbol Color Green (3)  Select

OK Cancel Load Save Help

Users have the ability to access EPASWMM from within the HydroNet Manager as well as the EPASWMM command found under the Network dropdown. You must have EPASWMM on your computer to use EPASWMM.



The first icon, EPASWMM settings, calls Carlson's EPASWMM configuration dialogs. The second icon, Export to EPASWMM, creates a .inp file to be imported at a later time into EPASWMM. The third icon, Analyze with EPASWMM, directly runs EPASWMM in the background, creating and displaying the desired report.

Users may export data from Hydronet Explorer to HEC-HMS, Hydrologic Engineering Center - Hydrologic Modeling System. This software is created by the US Army Corps of Engineers and is used to simulate the effects of precipitation and runoff of areas with tributaries. Using three main components - Basin Model, Meteorologic Model and Control Specifications - each with many parameters, the user can simulate a wide range of hydrologic problems, from a large river's tributary system and resulting river flow to small, urban systems, like a storm sewer network.

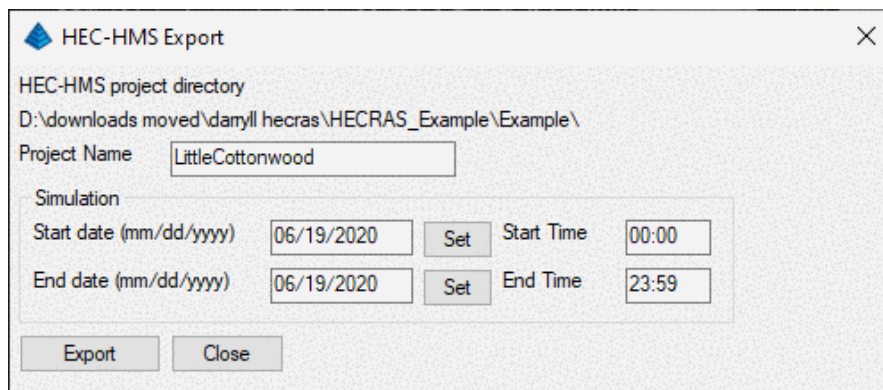
Much more information about the abilities, how to use HEC-HMS and how to download the software is available here:

<https://www.hec.usace.army.mil/software/hec-hms/>



To export, click the HEC-HMS export icon

Choose the Project Name, Start Date, Start Time, End Date and End Time. Click Export.



HEC-HMS can also be accessed using simplified inputs not requiring modeling under the Watershed => Hydrograph Routing => Watershed Hydrograph => SCS Method or by typing "RUNOFFHYD" in the Command Ribbon. Modify parameters as needed, calculate and click the "HEC-HMS" button to export data for HEC-HMS input.

Pulldown Menu Location: HydroCAD

Keyboard Command: hydronet2

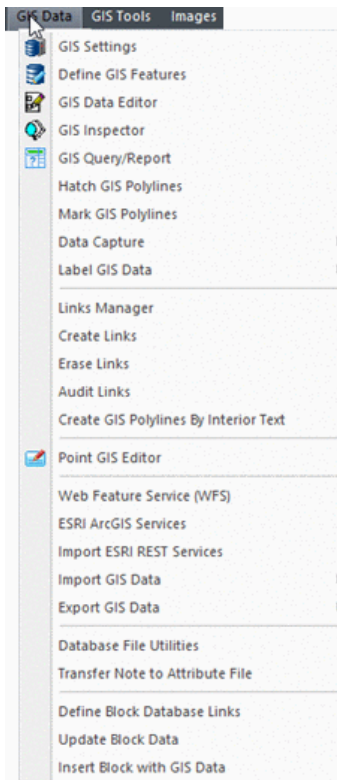
Prerequisite: Soils, soil labels, watersheds, watershed labels, a TIN, and Ground Covers, all on different layers for the different areas.

GIS Module

7

GIS Data Menu

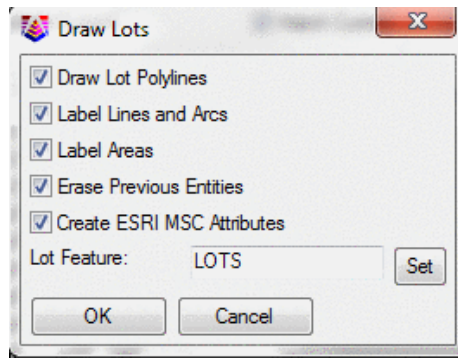
The GIS Data menu shown below has commands for managing and reporting data attached to drawing entities.



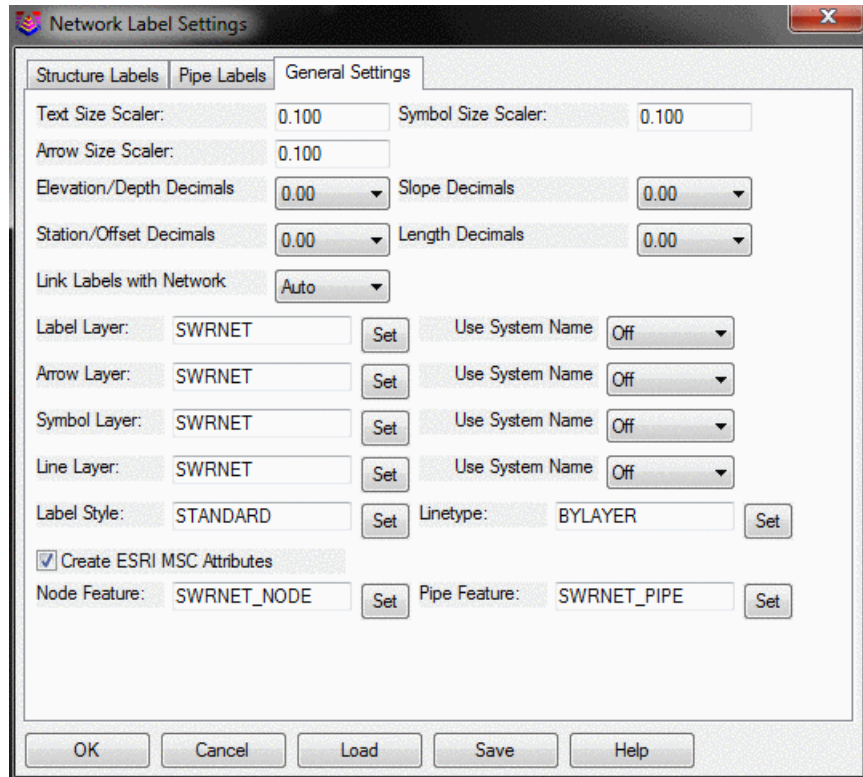
Carlson GIS and Esri

In addition to exchanging data with Esri using SHP files, Carlson GIS supports Feature Data within a drawing that is compatible with Esri ArcGIS software. This data is stored in a format created by Esri known as Mapping Specifications for CAD, or MSC. Starting with ArcGIS 9.3.1, Esri creates a DWG with MSC data using the Export To CAD function in the ArcGIS Toolbox. You can select which features to export to the DWG. The DWG with MSC contains the geometry of the features, the feature attribute definitions and the attribute values. Essentially, the DWG with MSC is a complete geodatabase. In Carlson GIS, set the data format to Esri under GIS Settings and the GIS commands such as GIS Data Editor will work with the Esri data. You can both add and edit GIS data in Esri format in the DWG. Then to bring the data into ArcGIS, choose the Add Data function in ArcGIS and pick the DWG file. With the MSC data format within the DWG, ArcGIS will recognize the features in the DWG instead of everything defaulting to generic CAD.

Several Carlson programs can create this data, such as LotNet, SewerNet, Mining Timing and Surface Mine Reserves. When lots, pipe networks or mining areas are drawn, there are new options to create Esri MSC in the process. When selected, the drawing can then be saved and loaded into ArcGIS as data, and the Features and Attributes created are available to ArcGIS. Here is where MSC is created from the Draw Lots command in Lot Manager.

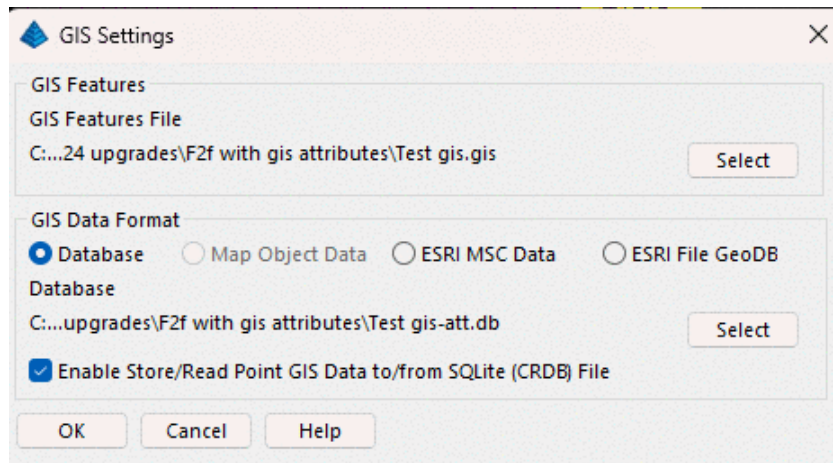


Here is where it is specified to create MSC when plan view of a Sewer Network is drawn.



GIS Database Settings

This command is the initial step to setup Carlson GIS routines. The user need to provide the GIS Features File name and Data Format type for the current drawing.

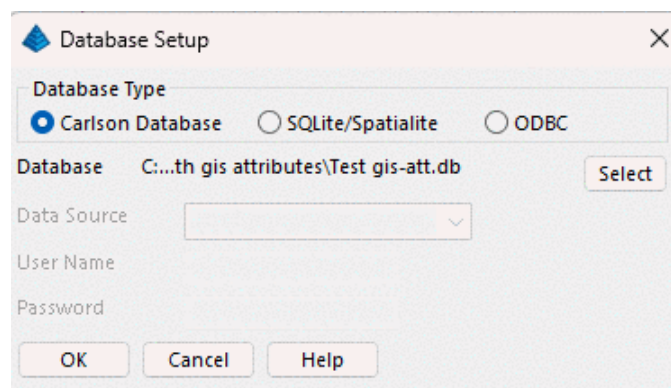


The GIS Features file (.GIS) includes the structure of the GIS features and their attribute, and it is defined in the Define GIS Features command.

The Data Format defines the type of data format and its stored location.

Currently, the GIS Data Formats include Database, Map Object Data, ESRI MSC and ESRI GeoDatabase.

The Database includes 3 types, Carlson Database(.DB), SQLite/Spatialite(.sqlite) , and ODBC Database. Carlson Database stores data in an external database in a SQLite format and the data schema is Carlson defined. The SQLite/Spatialite stores data in SQLite format(.sqlite) with spatial capabilities, and the data schema is user defined. The SQLite/Spatialite DB can be added into QGIS layer and it is also supported by ArcGIS 10.2 and above. This database can be read/write between Carlson GIS and ArcGIS/QGIS. The ODBC connection use Data source name to connect to third party database. The data schema is user defined, and it can be used by both Carlson GIS and other third party. Currently, only PostgreSQL has been tested. It is recommended to use Carlson DB if no external usage is needed.



For Carlson Database and SQLite/Spatialite need to specify the database name. For the ODBC connection, Data Source needs to be setup ahead of time. Please see How to Setup ODBC Data Source. Only import_odbc function allows to save GIS Data in ODBC database.

Map object data is attached to an individual object and stored in the object data tables in the drawing. Map Object Data is limited to Autocad Map 3D only. The Esri MSC Data stores the GIS data within the drawing file in a format that both Carlson and Esri use. Esri added support for MSC from ArcGIS V9.3. The ESRI MSC structure makes

the DWG file a type of geodatabase with all the feature definitions, GIS data and geometry stored in the file.

The Enable Store/Read Point GIS Data to/from SQLite (CRDB) File option provides an easier transferal method to and from data used in SurvCE/PC and Carlson Office products. The CRDB file format is based on SQLite structures. The CRDB format contains the same data as a CRD format but with extra information like GIS data and photos links. If Enable Store/Read Point GIS Data to/from SQLite (CRDB) File is OFF, the GIS data will be stored in a VTT format file.

If Spatialite DB/ODBC DB/ESRI FileGeoDB are selected, then there are special setup for the projection in the Settings/Drawing Setup dialog. Please see how to setup Projection for these GIS Data Format in Drawing Setup.

Pulldown Menu Location: GIS Data

Keyboard Command: gis_config

Prerequisite: None

Define Template Database

This command creates the Feature/Attribute data structure, template database or schema, for GIS functionality. The structure is stored in a Carlson Features File with a GIS file extension.

A feature, such as a manhole, can have multiple attributes, such as Number of rungs, Type of material, Number of inlets, etc. Features can be organized into Categories: Utilities, Roads, Properties. The Category designation is an arbitrary way of organizing the features. Features and attributes can be imported from:

- Carlson Coordinate File (.CRDB)
- Carlson Database (.DB)

- Carlson GIS Features file (.GIS)

- Carlson Field to Finish file (.FCL)
- Carlson SurvCE Features file (.VTT)
- Esri MSC data contained within the drawing (.DWG)
- Esri File GeoDatabase (.GDB)
- ESRI Shapefile(.SHP)

- Google Earth file (.KML)
- Map Object Data
- StarNet (.CSV)
- TDS Features file (.FEA)
- Text File (.CSV)
- Topcon GIS
- Trimble Features file (.FXL or .FCL)

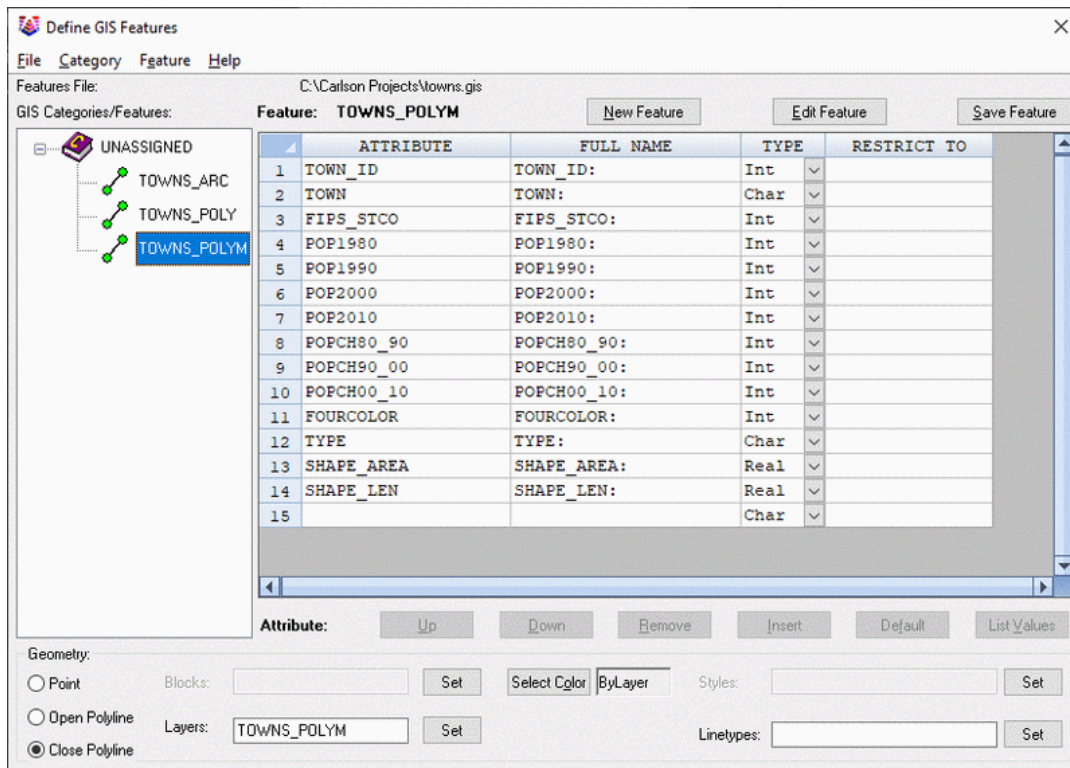
Note: Some data source such as ESRI Shape file, ESRI File GeoDatabase, ODBC Database and SQL/Spatialite Database will automatically add the new feature and its attributes when import GIS Data.

Features and attributes can of course also be defined "from scratch" in the Define GIS Features dialog box. The workflow is outlined below.

1. Set the GIS file name which will contain the features and attributes structure. Use the File menu to create a new (.GIS) file or open an existing one for editing. Please note that the system will only process correctly if a FCL and a GIS file both have the exact same file name
2. Set up one or more Category Names, using the Category menu. GIS feature codes can be categorized (e.g. STRUCTURES, UTILITIES, ROAD FEATURES, etc.). At least one category must be created.
3. Define Features, using the Feature menu. To edit a feature, double-click on the feature name.g. A category such as UTILITIES might have features such as manholes, light poles, fire hydrants, water valves, etc.
4. Define the attributes for the Feature. Each attribute has:

- a) a Name
- b) a Full Name, or Prompt
- c) a Type - Integer, Character or Real
- d) a system Default Value. Press the Default button to add a system variable from a list. For example, press Default > Elevation. The system will add \$EQN=\$LAT, a system variable that will insert the current latitude from the point measurement
- e) optionally, a List of values to pick from. Use the List Values button or type in each value followed by Enter within the field required, to build a list. The list will as a pick list when the attribute screen is shown.
- f) whether the attribute is required
- g) whether the attribute Value can be field Edited, appears as Read Only, or is Hidden
- h) whether the value used is restricted to the list

The order of attributes appearing on the data-entry screen is controlled by the Up and Down buttons.



Geometry Settings

At the bottom of the dialog box the user can specify the geometry settings for each feature. Each feature can be defined as a open/close polyline or point. Each feature can also have a layer, block text style and line type defined. Please note that GIS features can only be defined on polylines and not lines.

Note: If the GIS Data Format is ESRI MSC, then it is required to setup Layers. If the Layer name is miss-

ing, then the system will use the feature name as the Layer name

Note: If the GIS Data Format is ESRI File GeoDatabase and the feature definition is to create a GeoDatabase schema from scratch, then it is important to have an attribute as primary key. The attribute has to have type as Int and it is required. Plus, the Field Edit has to be Hidden because ESRI File GeoDatabase will automatically generate the primary key's value. Therefore, this attribute will not be available in GIS Data Editor.

Pulldown Menu Location: GIS Data

Keyboard Command: def_template

Prerequisite: None

GIS Data Editor

This routine creates, reviews and appends GIS data linked to entities stored in the drawing.

The GIS Smart Prompting dialog has a spreadsheet format for editing the data fields. The GIS table to process is selected in the pull-down list in the upper right of the dialog. The GIS tables that are available depend on the tables that are defined in the current data source. Use the *GIS Settings* and *Define GIS Features* commands to setup the tables. Once you select a table to process, the fields for that table are displayed in a spreadsheet format. If a field is related to a field in another table in the database, a "+" character is shown next to the field name. Picking the "+" will open another dialog box with the related data in the other table. The data in this related table is not editable, only the data in the initial linked table.

The bottom portion of the dialog has features for attaching images to the entity. (Note: The bottom portion is only limited to single file type database and ESRI MSC Data.) Existing image files (BMP, JPG or GIF) can be linked by choosing the New option. The Update option will replace the current image with a newly selected image. The Delete option will remove the current, attached image. The Capture button will take a shot in the field using a configured camera and then attach the image to the entity. Different digital cameras can be used by picking Pick or Set Camera.

The *GIS Data Editor* command is an excellent way to simply review the data associated with an entity. If the entity has GIS data, the banner line at the top of the dialog will display "Entity has GIS Data". If not, the banner line will display "Entity has no GIS Data". Even when the entity has no data, the default values for the prompts will appear. Pressing OK will assign this data to the entity. To avoid assigning data to the entity (if it has none), press Cancel. Alternately, you can use the commands *GIS Inspector Settings*, followed by *GIS Data Inspector*, to review the data with no possibility of editing or inputting data in the process.

There are three methods for selecting the drawing entities to process: S for Select, P for Pick and N for Number:

Select Object method: With this method, you pick the drawing entity to process the data attached to that entity. When selecting a Carlson point, the point number is used to link to the database.

Pick method: For this method, you pick inside a closed polyline to process the data attached to that polyline.

Number method: Here you simply input the point number from the current CRD file to process.

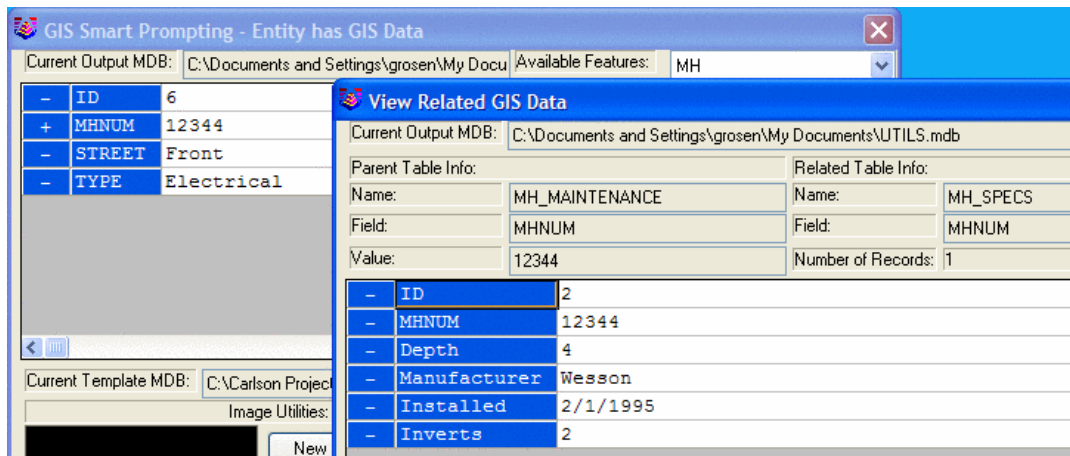
Map Object Data has different dialog that only allow user to select GIS Feature table and add/edit/delete data.

Prompts

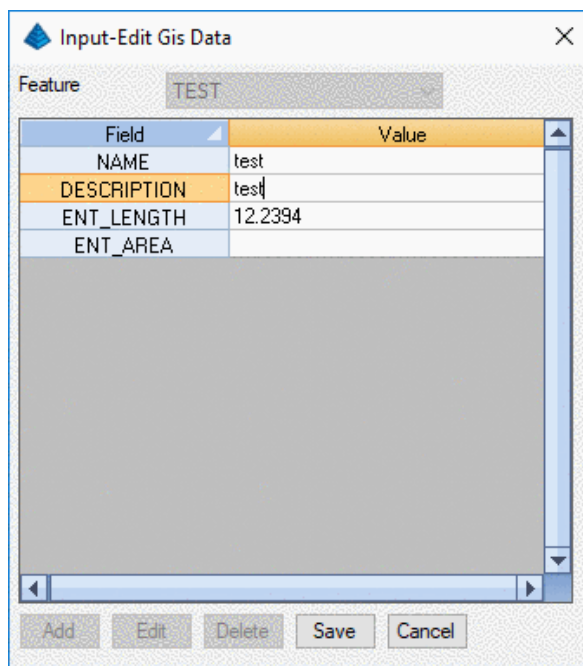
Select object (Number/Pick/<Select>): P

Pick a point inside polygon (Select/Number/<Pick>): pick a point

GIS Smart Prompting dialog make selections



Map object data/ODBC Database editor dialog



Pulldown Menu Location: GIS Data

Keyboard Command: gisdata

Prerequisite: GIS Feature file name and GIS Data Format must be defined in GIS Settings dialog

GIS Inspector

This command displays all or portions of the data attached to drawing entities in real-time. Initially, the command sets up the fields to be displayed when using the GIS Inspector.

When you move the cursor over an entity with GIS data, selected fields are displayed in a tooltip box next to the cursor. For data attached to closed polylines, you can move the cursor anywhere inside the polyline to show the data. Polylines that are closed will highlight with a solid fill as you inspect each one. Open polylines, such as road centerlines, will highlight with a solid fill generated along the length of the polyline. The solid fill color for all highlighting is set in GIS Inspector Settings.

The routine starts by prompting you to select entities. The entities that you select will be used by GIS Inspector. In the case of a large drawing, this selection allows you to limit the entities for inspector to a local area instead of

having to process the whole drawing. Then after reading the entities, you can move the cursor around the drawing to inspect the GIS data. You can also use the arrow, page up and page down keys to pan and zoom the display. Pressing enter ends the routine.

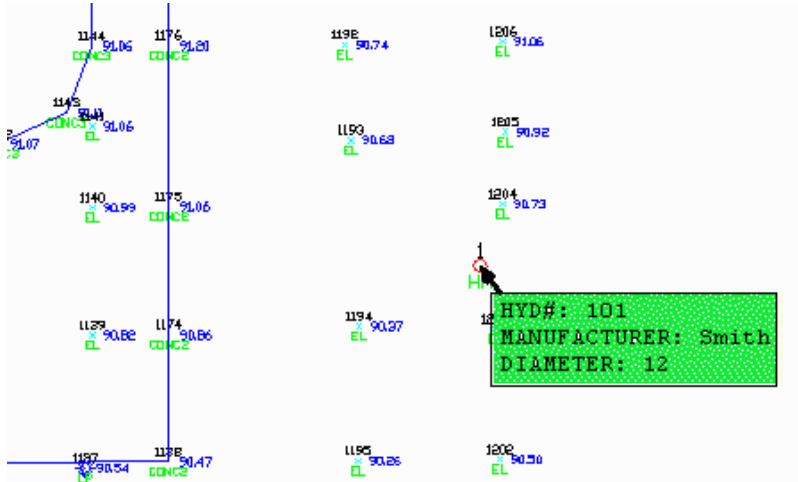
Prompts

Select objects: *select entities with attached data*

Arrow keys=Pan; PageUp/Down=ZoomOut/In;

Zoom=Pick left-lower and right-upper corner;

Move pointer over entity with Gis Data (Enter to End): *move cursor over entities with data; press Enter to end*



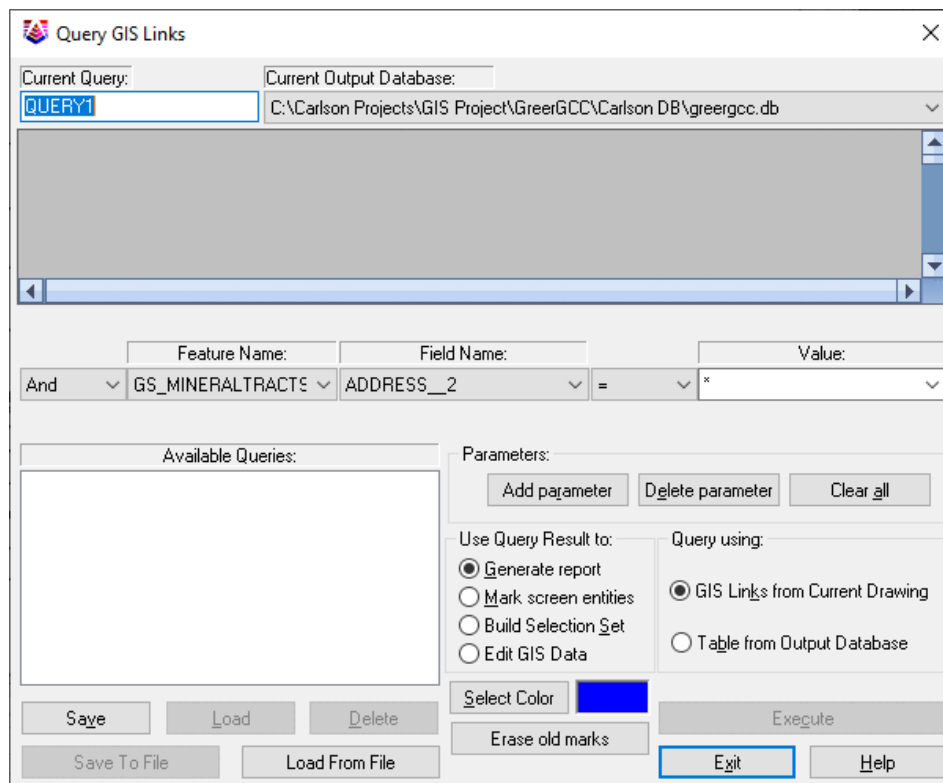
Pulldown Menu Location: GIS Data

Keyboard Command: gis.inspector

Prerequisite: MDB GIS Prompting must be created in *Define Template Database* and entities must have linked GIS information.

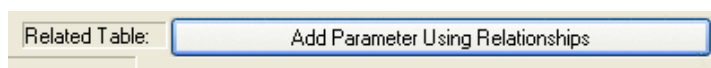
GIS Query/Report

This command applies a user-defined query on a data table or related tables with the database. Records in the table that pass the query can be reported or the associated entities can be highlighted in the drawing. The *Query Using* option in the main dialog box sets the source of the data table to process as either GIS data attached to selected drawing entities or from the current Output GIS Data Format source.



The query is defined in the dialog shown here. To add a query, enter a new query name in the in the space underneath Current Query. If there is already a name there, just highlight and type over it with a new name, then hit Clear All to clear out existing query lines and get full access to all Feature Names.

The top portion of the dialog contains a list of the query parameters. To add a parameter, select a Feature Name from the pop-up list. The available features will either be all the features found in the GIS links of the drawing or all the features from the GIS Data Format source depending on the *Query Using* option. Once the feature is specified, the Field Name pop-up list contains all the available fields in the feature. Choose a field from this list. Next choose the operator (=, >, etc.) from the operator list. The Value pop-up list contains all the different values for that field that are found in the current data set. You can either select one of these values or type in another value into this field. If a Field Name relates to another Feature, when you select that Field, an additional button will appear allowing you to add a query parameter from the related feature.

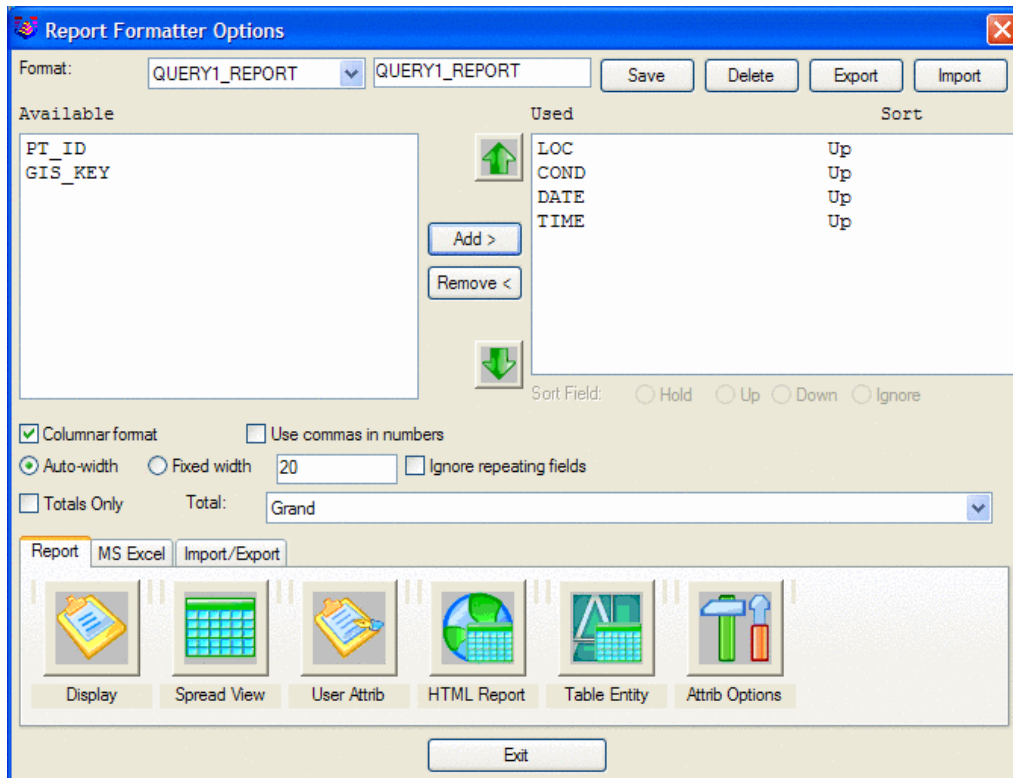


When all the parameter values are set, pick the Add Parameter button. Once a feature is selected and add a parameter is added, the Feature Names list becomes unavailable because any additional query parameters must come from that feature, or relate through that primary feature.

When all the parameters are defined for the query, you can save these settings by filling out a name Current Query field and then picking the Save button. This query can be recalled later by highlighting the query name and clicking the Load button. The Delete button removes the highlighted query. The Save, Load and Delete functions operate on the current set of queries active in the program. The Save To File and Load From File functions read and write the collection of queries to a .QRY file for managing different sets of queries and sharing with others.

Pick the Execute button to process the query. The Mark Screen Entities option will set the color of entities with GIS data that match the query to the specified color. The Build Selection Set option creates a selection set of the entities that pass the query. To use this selection set in other commands, enter "P" for previous at the "Select objects:"

prompt. With the Generate Report option, the program will bring up the Report Formatter which allows you to choose the fields to include in the report and the report format. If the Highlight Screen Entities option is on, then the program will highlight the entities with GIS data that pass the query. Point entities are highlighted by drawing a box around the point and polylines are highlighted by solid fill. Shown here is the report for all manholes with a Condition of Good.



Pulldown Menu Location: GIS Data

Keyboard Command: gis_query

Prerequisite: MDB file with data or entities with linked GIS information

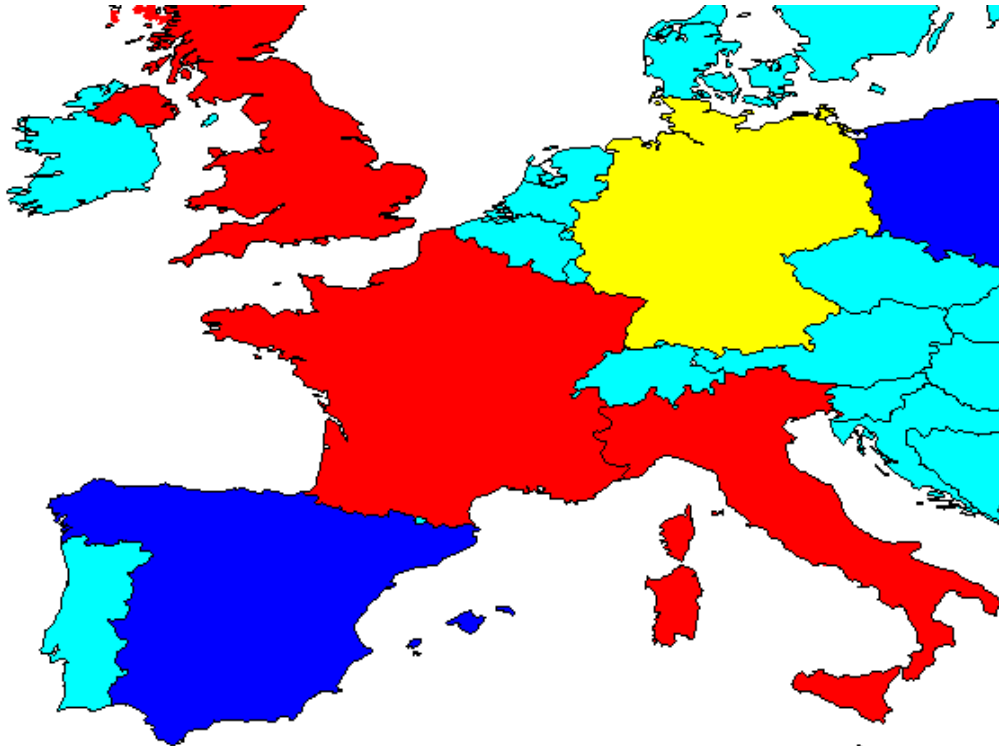
Hatch GIS Polylines

This command hatches closed polylines with attached GIS data based on the value of a specified database field. The program starts by selecting closed polylines in the drawing with GIS data. Then a dialog appears for specifying the database field to process. This dialog displays a list of all the GIS table names found in the selected polylines. First choose a table to process and then choose a field to process by clicking on the down arrow underneath Current field. Next you can specify the color, hatch pattern and layer for each zone. The Auto button can be used to quickly fill out the hatch zones. The Show All The Distinct Values option chooses between processing as series of ranges or individually for each data value. Applications include hatching all commercial property red and residential yellow, coloring buildings by type of construction material, coloring properties by type of ownership, etc.

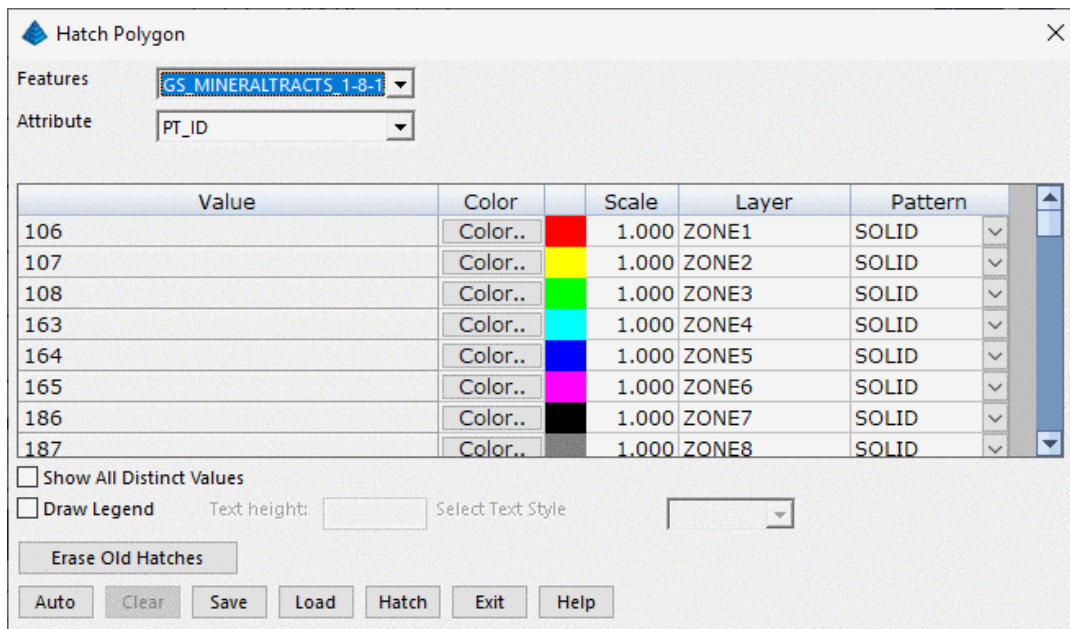
The Draw Legend option will create a legend of the hatch zones. The Erase Old Hatches button will erase any existing hatches inside the selected polylines. The settings can be saved to and recalled from a GIS settings file (.GSF) using the Save and Load buttons. Once all the settings are ready, pick the Hatch button to hatch the polylines. When using solids for the hatch pattern, the Solid Fill Float command in the View menu can be used to make the polylines appear on top of the solids.

Prompts

Select objects: *select closed polylines with GIS data*



Example country polylines hatched by population range field



Pulldown Menu Location: GIS Data

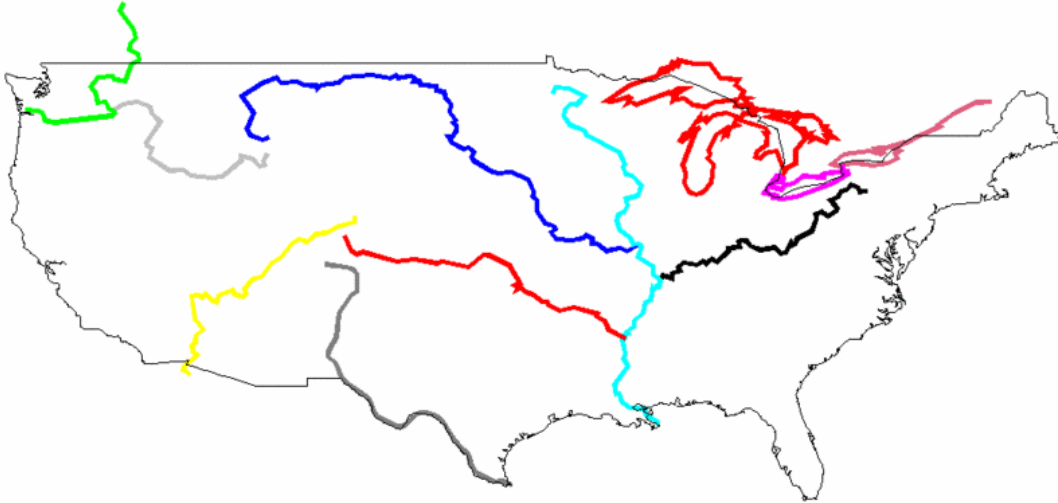
Keyboard Command: hatch_polygon

Prerequisite: Closed polylines with linked GIS data

Mark GIS Polylines

This command draws new polylines as markers along selected existing polylines. The new polyline markers are drawn with width and color based on a query of the attached GIS data. The program starts by selecting open or closed polylines in the drawing with GIS data. Then a dialog appears for specifying the database field to process. This dialog displays a list of all the GIS table names found in the selected polylines. First choose a table to process and then choose a field to process. Next you can specify the color and width for each data value. The Auto button can be used to quickly fill out the color and widths. The Show All The Distinct Values option chooses between processing as series of ranges or individually for each data value.

The Draw Legend option will create a legend of the marks. The Erase Old Marks button will erase any existing polyline markers from the selected polyline. The settings can be saved to and recalled from a GIS settings file (.GSF) using the Save and Load buttons. Once all the settings are ready, pick the Draw button to draw the marker polylines.



Mark Polygon [X]

Features:

Attribute:

Value	Color	Width	Layer
106	Color.. [Red]	1.000	ZONE1
107	Color.. [Yellow]	1.000	ZONE2
108	Color.. [Green]	1.000	ZONE3
164	Color.. [Cyan]	1.000	ZONE4
165	Color.. [Blue]	1.000	ZONE5
186	Color.. [Magenta]	1.000	ZONE6
187	Color.. [Black]	1.000	ZONE7
188	Color.. [Grey]	1.000	ZONE8

Show All Distinct Values

Draw Legend Text height: Select Text Style:

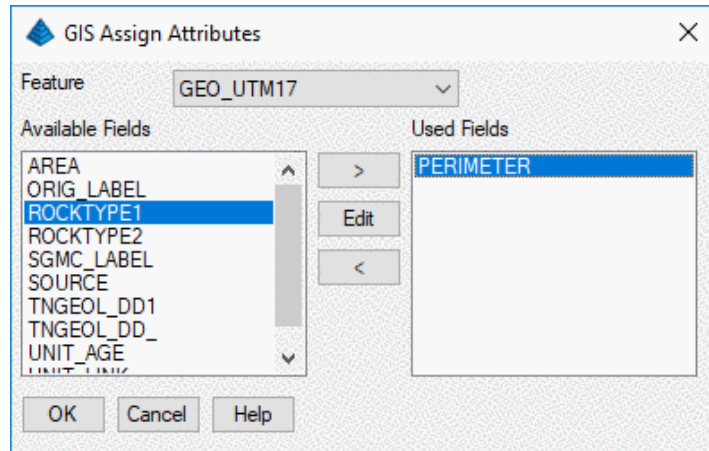
Pulldown Menu Location: GIS Data

Keyboard Command: trace_polyline

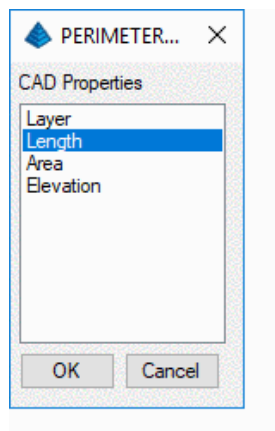
Prerequisite: Closed polylines with linked GIS data

Data Capture From CAD Properties

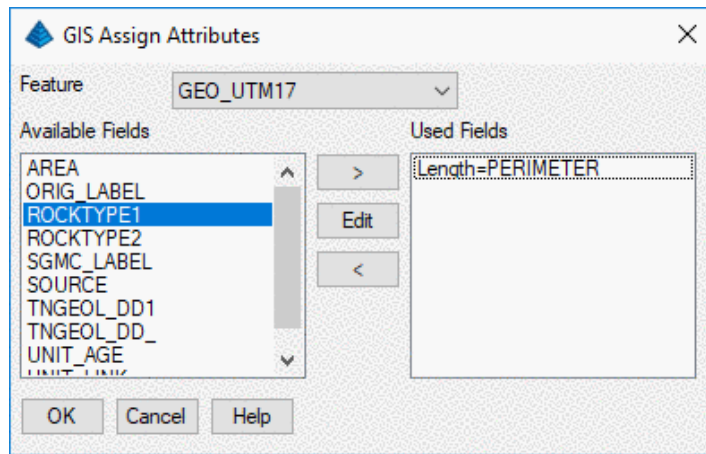
This command capture CAD properties data from selected entities and assigned them to entities's GIS values. The user needs to select GIS Feature from the drop down button and add the desired fields from the right combo box to the left combo box. Then the user need to match the GIS field with CAD Properties.



The available CAD properties are area, length, layer and elevation. Only closed polyline can assign Area to the GIS data.



After the selection, the entry in the Used Fields will have the format of *<CAD property>=<GIS attribute>*. The following picture shows "Length=Perimeter"



Prompts

Select linework entities: select linework

Pulldown Menu Location: GIS Data > Data Capture

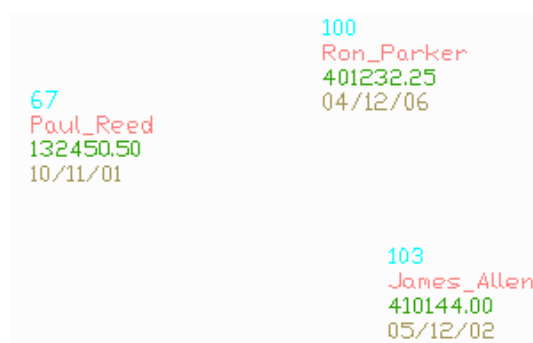
Keyboard Command: giscaddata

Prerequisite: Linework and defined GIS feature

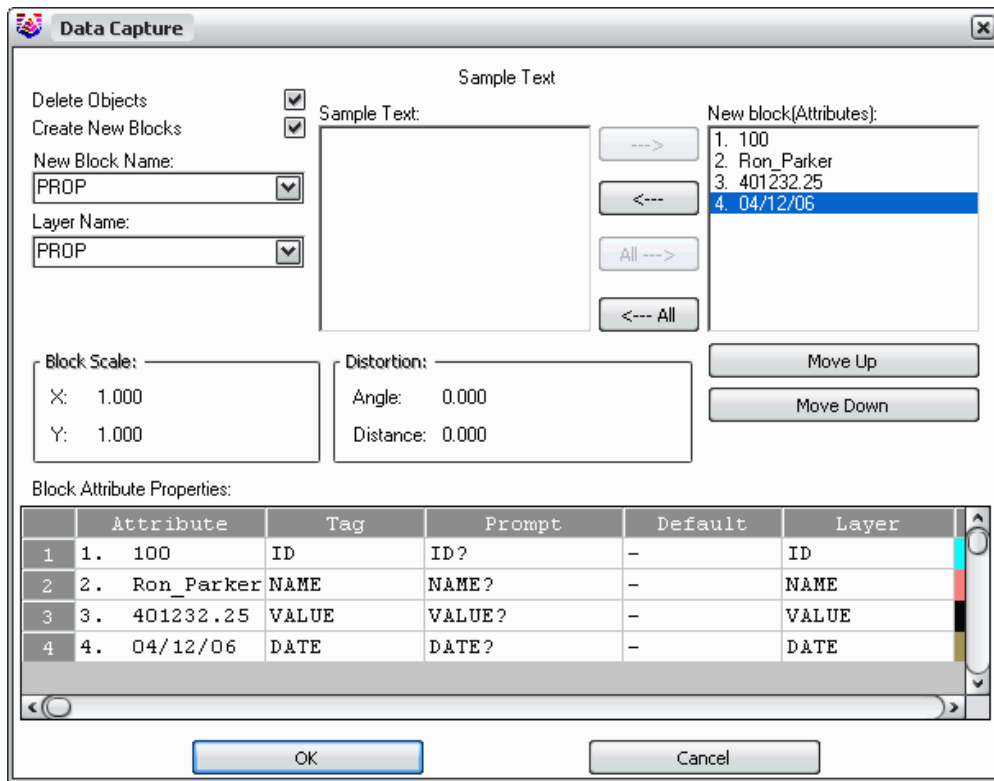
Data Capture Text By Sample

This command automates data transfer from text entities to a database table. Many drawings are warehouses of information. The Data Capture Text By Sample command applies to drawings that contain text annotations which show the information about the objects in the drawing.

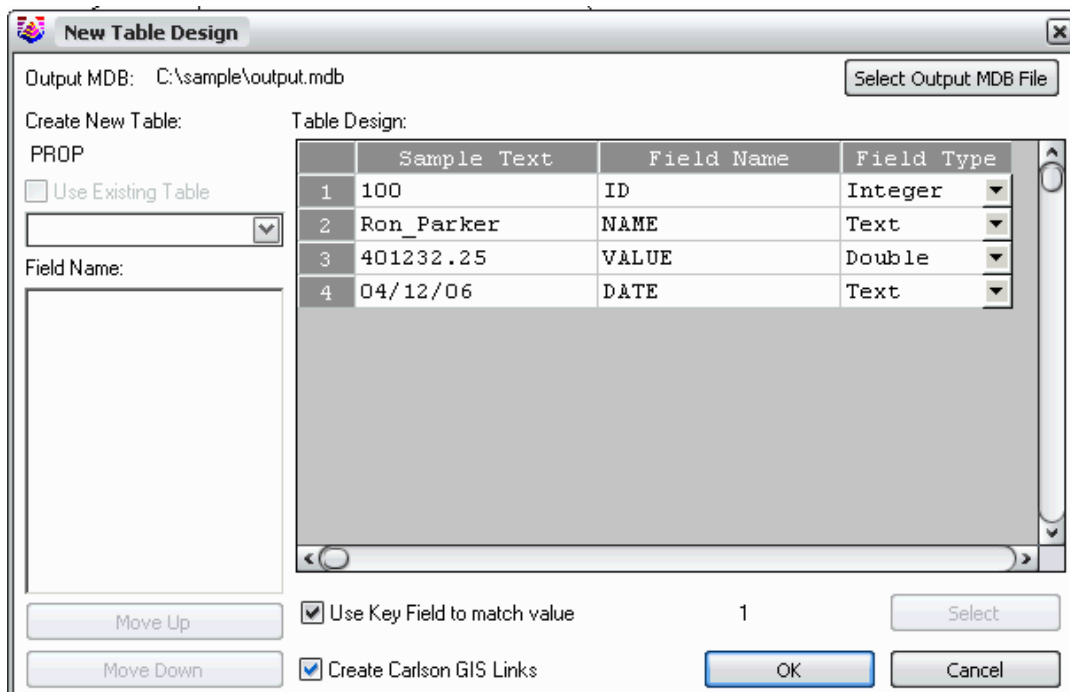
The command starts by prompting you to select one sample of the text to capture. This sample selection should consist of one instance each of the data fields to process. In the example below, the sample selection includes the property id, owner's name, property value and date.



Next, there is a dialog for specifying the text fields to capture and their order. Optionally you can convert the text entities into blocks by turning on the Create New Blocks option. Each sample text field is available to make into a block attribute by moving these fields to the block attribute list. In the New Block Name and Layer Names fields, you can specify the name and layer for the blocks. The Delete Objects option will erase the text entities. In the Block Attribute Properties spreadsheet, you can set the block tags, prompt, default and layer for each attribute.



The second dialog setups up the database table to create from the text data. The destination database is set with the Select Output MDB File button. You can either create a new table in the database or select an existing table. Under Table Design, you can set the Field Name and Field Type for each attribute when a new table is being created. When adding to an existing table, the Field Name and Field Type are read from the existing table. For with an existing table, the Field Name list in the lower left will show the field names from the existing table, and the order of these fields should be set to match the order of corresponding the text fields. The Use Key Field To Match Value option will use the selected record as the key field for the database table. The key field must have unique values for each record in the table. To set the key field, highlight the record row # in the spreadsheet and then pick the Select button. The Create Carlson GIS Links option will link the database table with the drawing entity for each record.



After the dialogs, there is a command line prompt to select the text entities to import. The program uses the sample text layout to find matches sets of text data in the text to process selection set.

Prompts

Select sample text to capture.

Select objects: *pick one instance of each type of text field*

Data Capture dialogs

Select entities to process [Select/<All>]: *all*

Created 3 GIS Links.

Done

Pulldown Menu Location: GIS Data > Data Capture

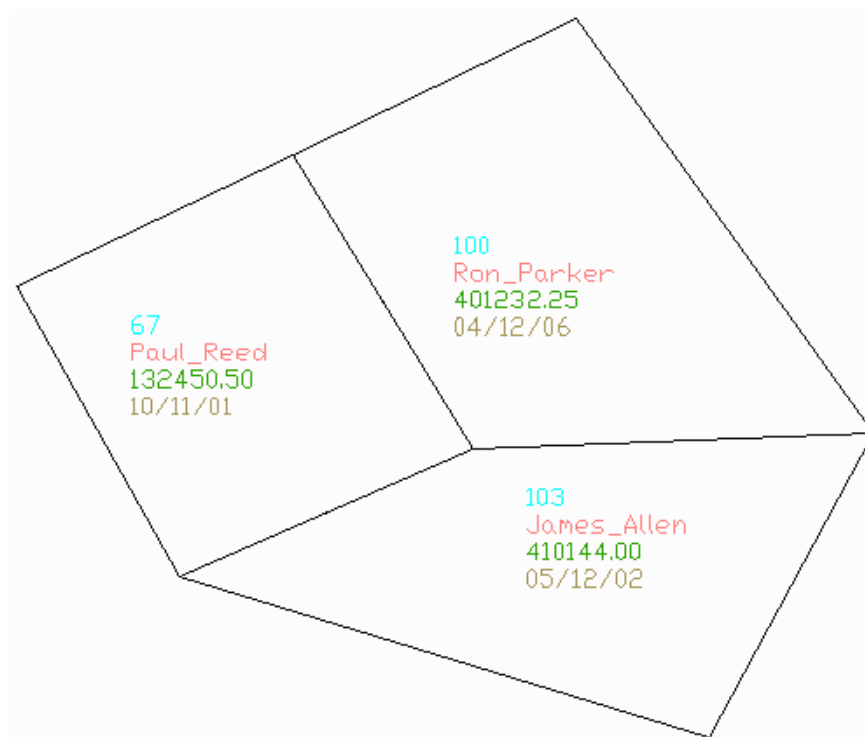
Keyboard Command: `cgis_gettxtbysmp`

Prerequisite: Text entities

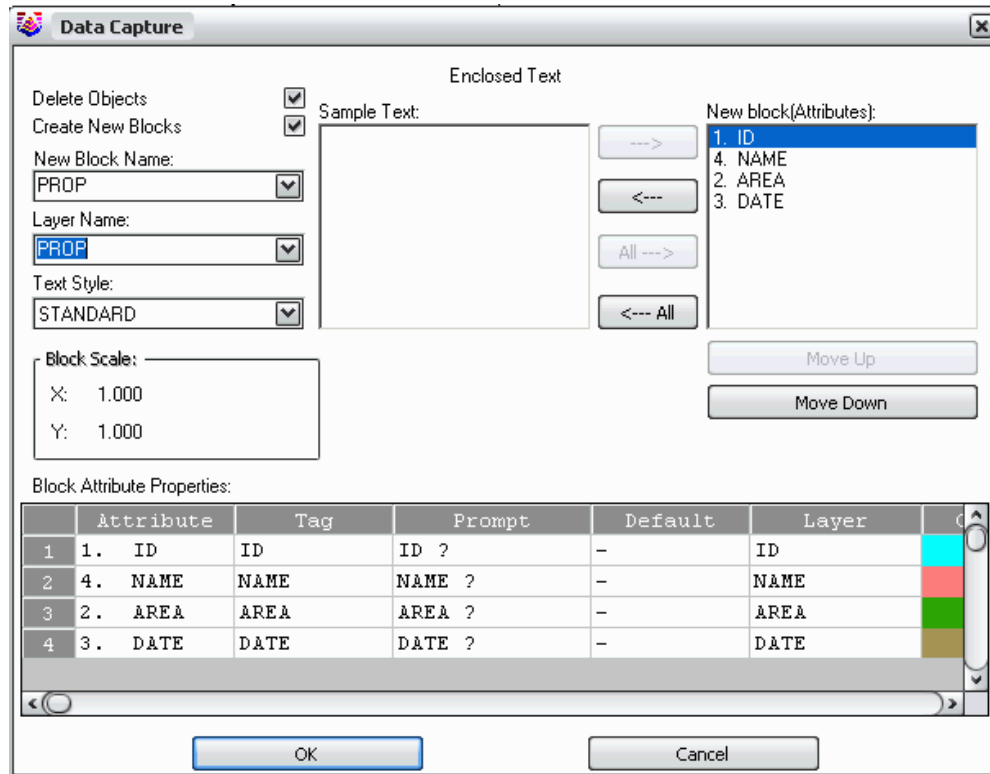
Data Capture Enclosed Text

This command automates data transfer from text entities to a database table. Many drawings are warehouses of information. The Data Capture Enclosed Text command applies to drawings that contain text annotations which are within linework perimeters.

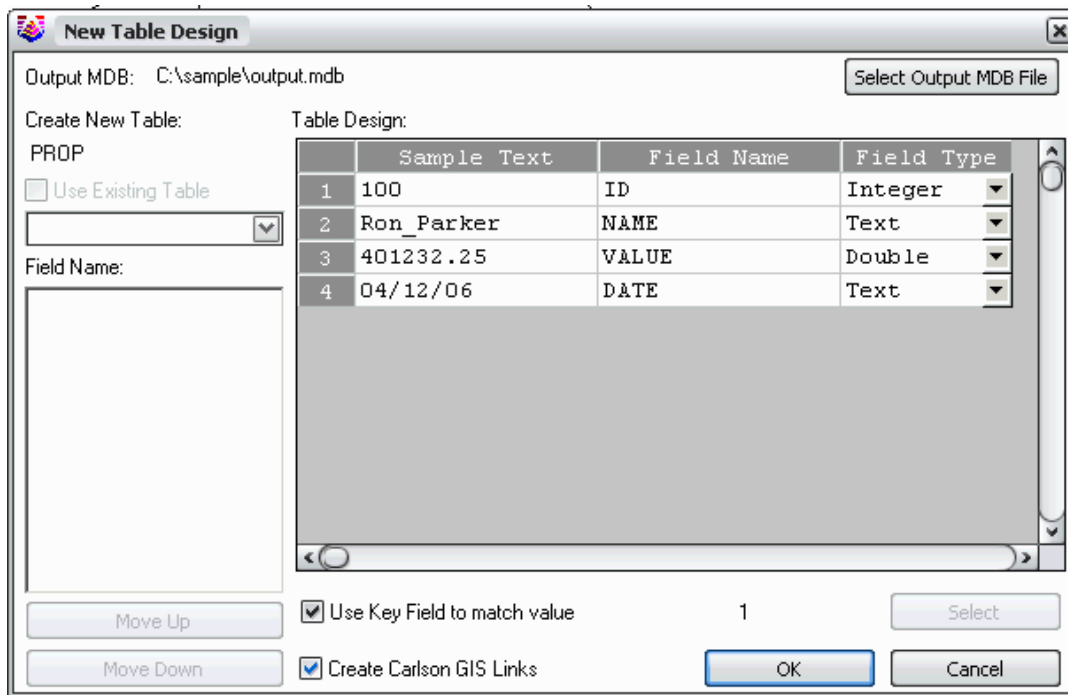
The command starts by prompting whether to identify the text to process by Group or by Layer. For the Group method, you select one sample of the text to capture. This sample selection should consist of one instance each of the data fields to process. In the example below, the sample selection includes the property id, owner's name, property value and date. For the Layer method, the program will find all the different layers for the text entities and you can select which layers to use. For the Layer method, each type of text field should be on separate layers.



Next, there is a dialog for specifying the text fields to capture and their order. With the Group method, the text fields will show the values of the selected text. With the Layer option, the text fields will show the layer names. There is an option to convert the text entities into blocks by turning on the Create New Blocks option. Each sample text field is available to make into a block attribute by moving these fields to the block attribute list. In the New Block Name and Layer Names fields, you can specify the name and layer for the blocks. The Delete Objects option will erase the text entities. In the Block Attribute Properties spreadsheet, you can set the block tags, prompt, default and layer for each attribute.



The second dialog setups up the database table to create from the text data. The destination database is set with the Select Output MDB File button. You can either create a new table in the database or select an existing table. Under Table Design, you can set the Field Name and Field Type for each attribute when a new table is being created. When adding to an existing table, the Field Name and Field Type are read from the existing table. For with an existing table, the Field Name list in the lower left will show the field names from the existing table, and the order of these fields should be set to match the order of corresponding the text fields. The Use Key Field To Match Value option will use the selected record as the key field for the database table. The key field must have unique values for each record in the table. To set the key field, highlight the record row # in the spreadsheet and then pick the Select button. The Create Carlson GIS Links option will link the database table with the drawing entity for each record.



After the dialogs, there is a command line prompt to select the text entities to import. For each text entity, the program finds the boundary perimeter from the drawing linework that encloses the text. These boundaries are used to separate the text labels into the groups that make up the table records.

Prompts

Select text to capture [Group/<Layer>]: *L for layer*

Data Capture dialogs

Select entities to process [Select/<All>]: *All*

Created 3 GIS Links.

Done

Pulldown Menu Location: GIS Data > Data Capture

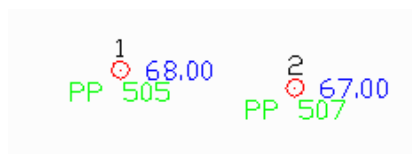
Keyboard Command: *cgis_getencltxt*

Prerequisite: text entities within linework perimeters

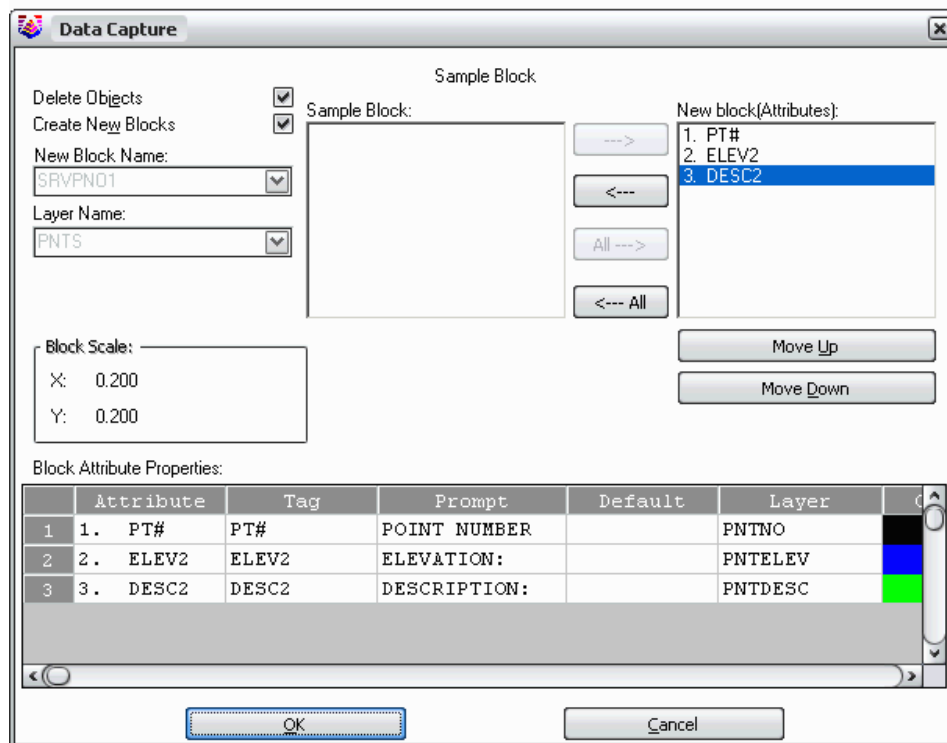
Data Capture Block Attributes

This command automates data transfer from block attributes to a database table. Many drawings are warehouses of information. The Data Capture Block Attributes command applies to drawings that contain blocks with attribute values.

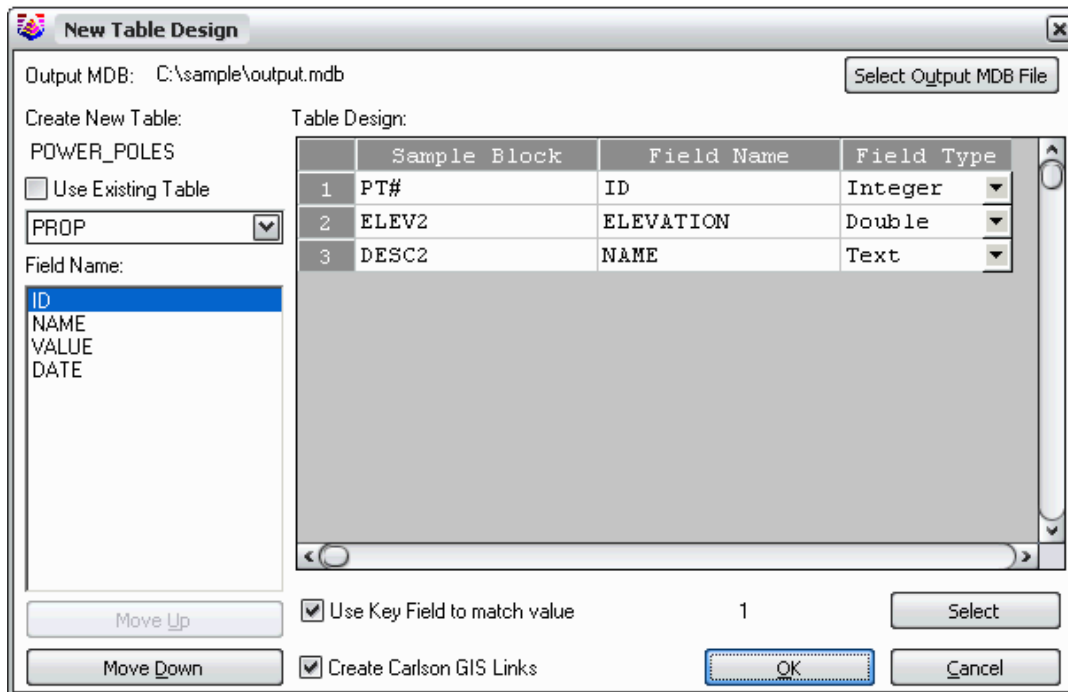
The command starts by prompting to select one sample of the block to capture. The program reads the attribute names from this sample block. This example uses Carlson point blocks that have attributes for point number, elevation and description.



Next, there is a dialog for specifying the attributes to capture and their order. Each sample attribute is available to capture by moving these fields to the block attribute list. There is an option to create a new block out of the selected attribute by turning on the Create New Blocks option. In the New Block Name and Layer Names fields, you can specify the name and layer for the blocks. The Delete Objects option will erase the source block entities. In the Block Attribute Properties spreadsheet, you can set the block tags, prompt, default and layer for each attribute.



The second dialog setups up the database table to create from the block data. The destination database is set with the Select Output MDB File button. You can either create a new table in the database or select an existing table. Under Table Design, you can set the Field Name and Field Type for each attribute when a new table is being created. When adding to an existing table, the Field Name and Field Type are read from the existing table. For with an existing table, the Field Name list in the lower left will show the field names from the existing table, and the order of these fields should be set to match the order of corresponding the text fields. The Use Key Field To Match Value option will use the selected record as the key field for the database table. The key field must have unique values for each record in the table. To set the key field, highlight the record row # in the spreadsheet and then pick the Select button. The Create Carlson GIS Links option will link the database table with the drawing entity for each record.



After the dialogs, there is a command line prompt to select the blocks to import. Only blocks that match the sample block names will be processed.

Prompts

Select sample block to capture: *select a block*

Processing entities...Please wait...

Select entities to process [Select/<All>]: *All*

Created 2 GIS Links.

Done

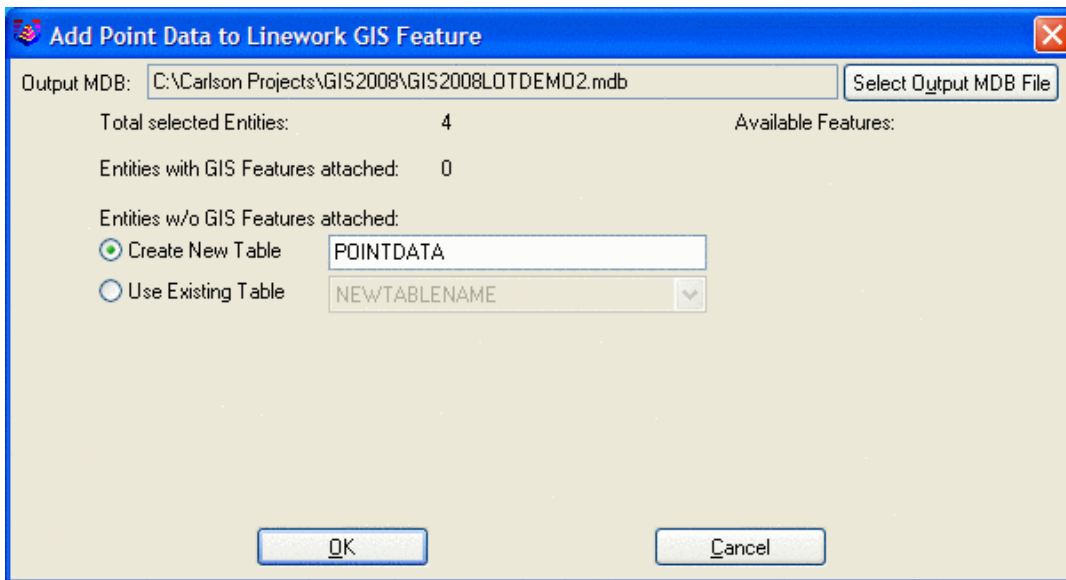
Pulldown Menu Location: GIS Data > Data Capture

Keyboard Command: `cgis_getblockattr`

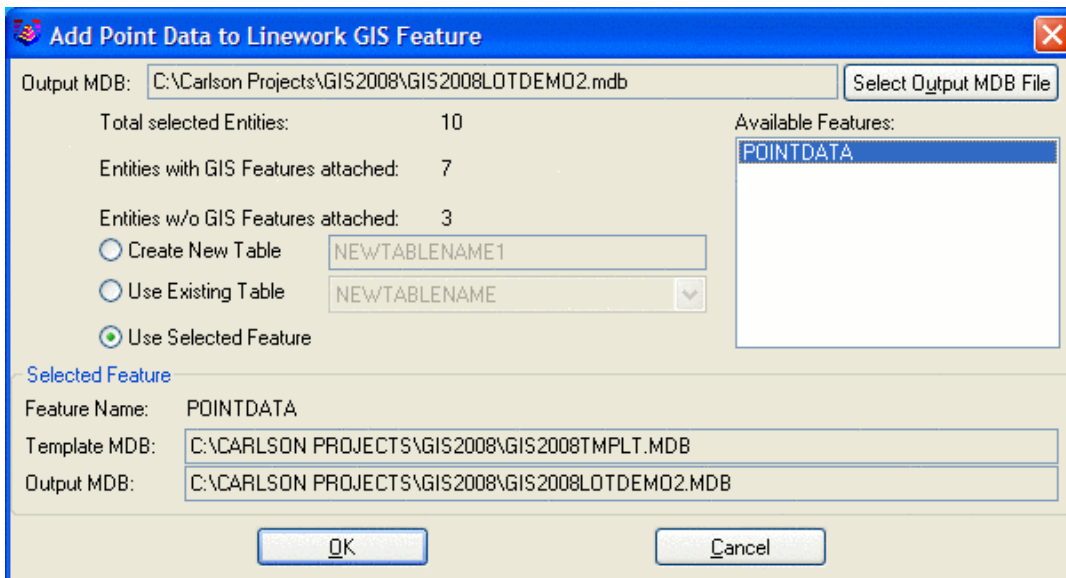
Prerequisite: Blocks with attributes

Data Capture Add Point Data to Linework

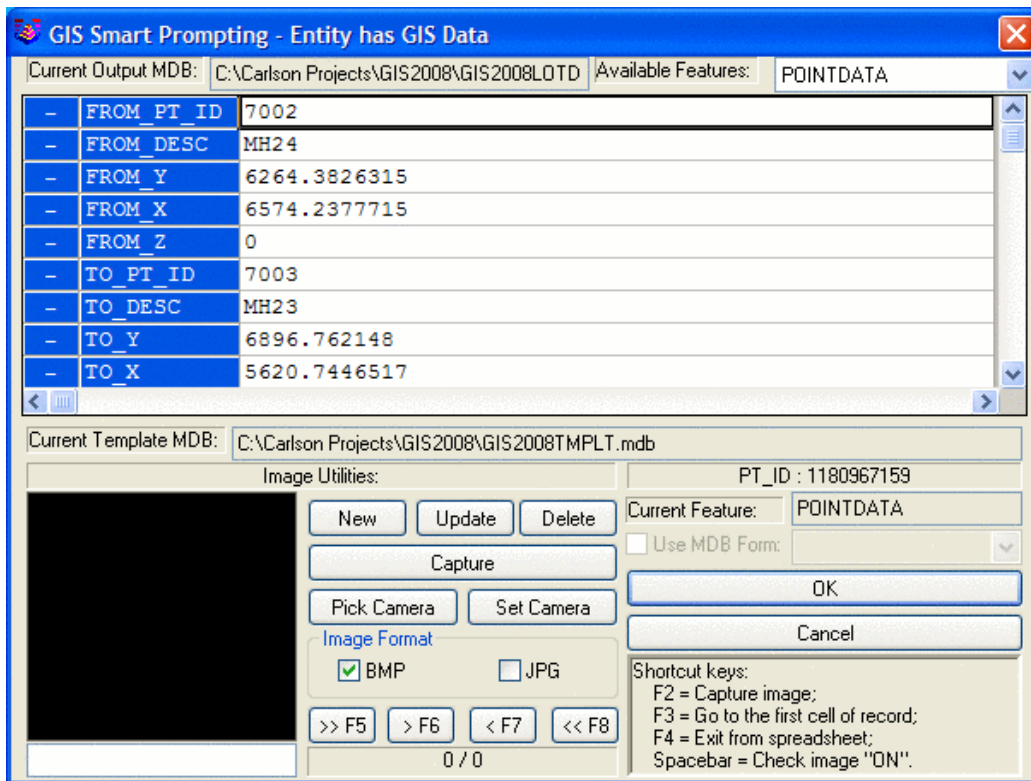
This command captures point data (Number, Northing, Easting, Elevation, Description) from Carlson Points at the ends of existing linework and writes that data to a table in the Output database. If the selected linework has no GIS data already attached, the following dialog is displayed:



If any of the selected linework has GIS data already attached, the following dialog is displayed:



In either case, you are asked to specify what table is to be used to store the point data. Once accomplished, using Input/Edit GIS data displays the point data now associated with the linework.



Prompts

Select linework entities: *select linework*

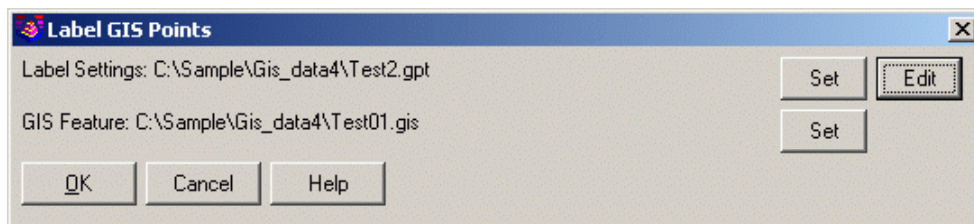
Pulldown Menu Location: GIS Data > Data Capture

Keyboard Command: gisptdata2linework

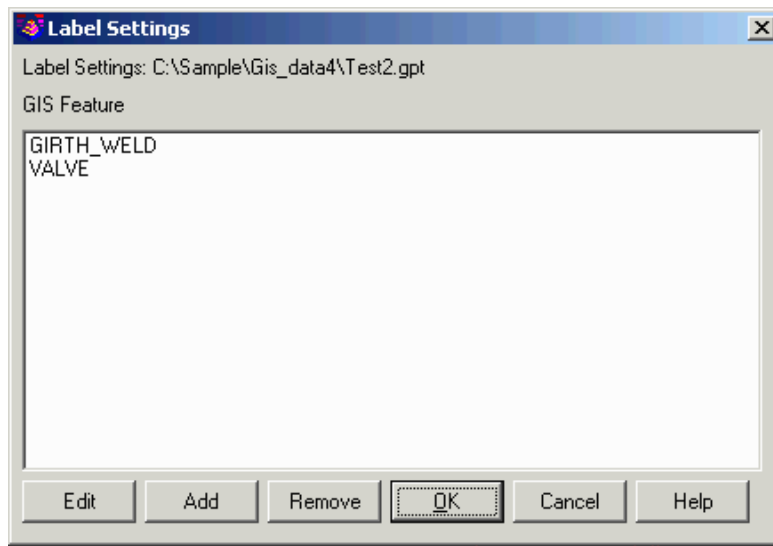
Prerequisite: Linework and Carlson Points

Label GIS Point Data

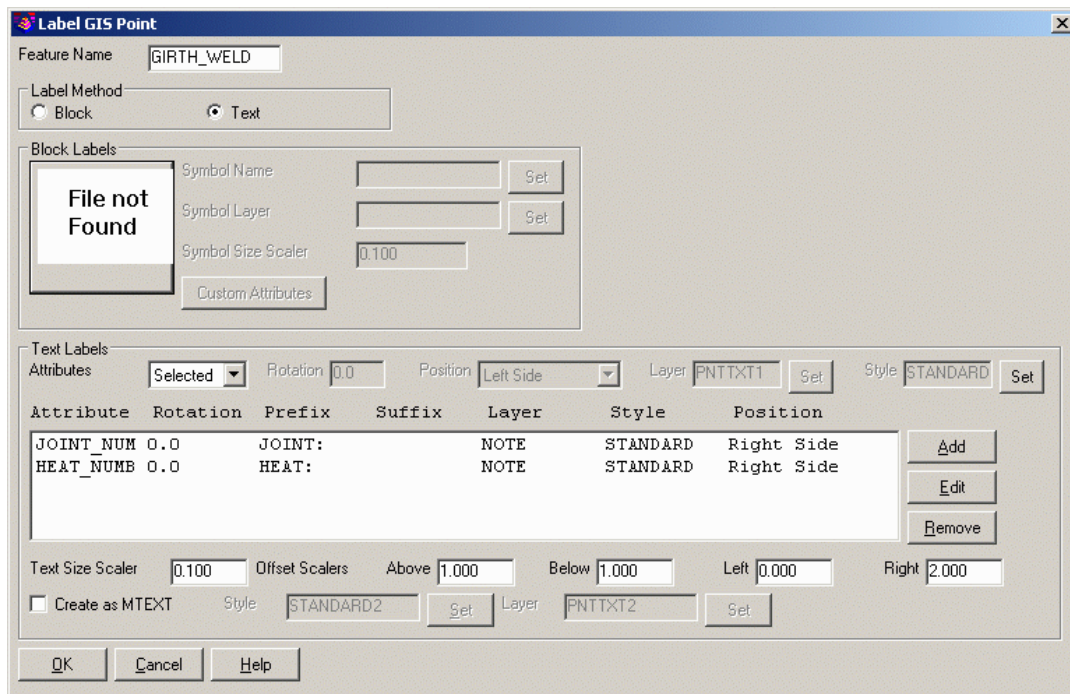
This command labels attributes for point features. The labels can be drawn as text entities or as blocks with attributes. Before running this command, use the Define GIS Features command to define the GIS point features and the attributes.



This command starts with a dialog to set the Label Settings file (.gpt) and the GIS Features file (.gis). The Label Settings file is used only for this command. Pick the Edit button to access the editor for the label settings.



The Label Settings dialog shows a list of the features with defined labels. Use the Edit, Add and Remove functions to manage the list.



The Edit and Add functions bring up the label editor dialog.

Feature Name: This name needs to match the feature name for the points as set by Define GIS Features.

Label Method: This settings chooses between creating text labels or blocks with attributes.

Block Labels

Symbol Name: This is the block drawing name. This drawing must have block attributes defined for filling out with the GIS attribute values.

Symbol Layer: The blocks will be drawn on this layer.

Symbol Size Scaler: Controls the size of the blocks. This size is multiplied by the current drawing horizontal scale to set the block size.

Custom Attributes: This function has another dialog for mapping the GIS attributes to the block attributes.

Text Labels

Attributes: Chooses whether to label all the attributes or specific attributes. When using All, there is one set

of settings for the Rotation, Position, Layer and Style for all the labels. For using Selected, there is a list of the attributes to label and the Add, Edit and Remove buttons are used to manage the list.

Text Size Scaler: Controls the size of the text entities.

Offset Scalers: Control how far to offset the labels from the point.

Create as MText: This groups all the labels into a single MText entity instead of individual Text entities for each attribute.

Prompts

Label GIS Points dialog

Select points to label.

Select objects: *pick Carlson points*



Pulldown Menu Location: GIS Data > Label GIS Data

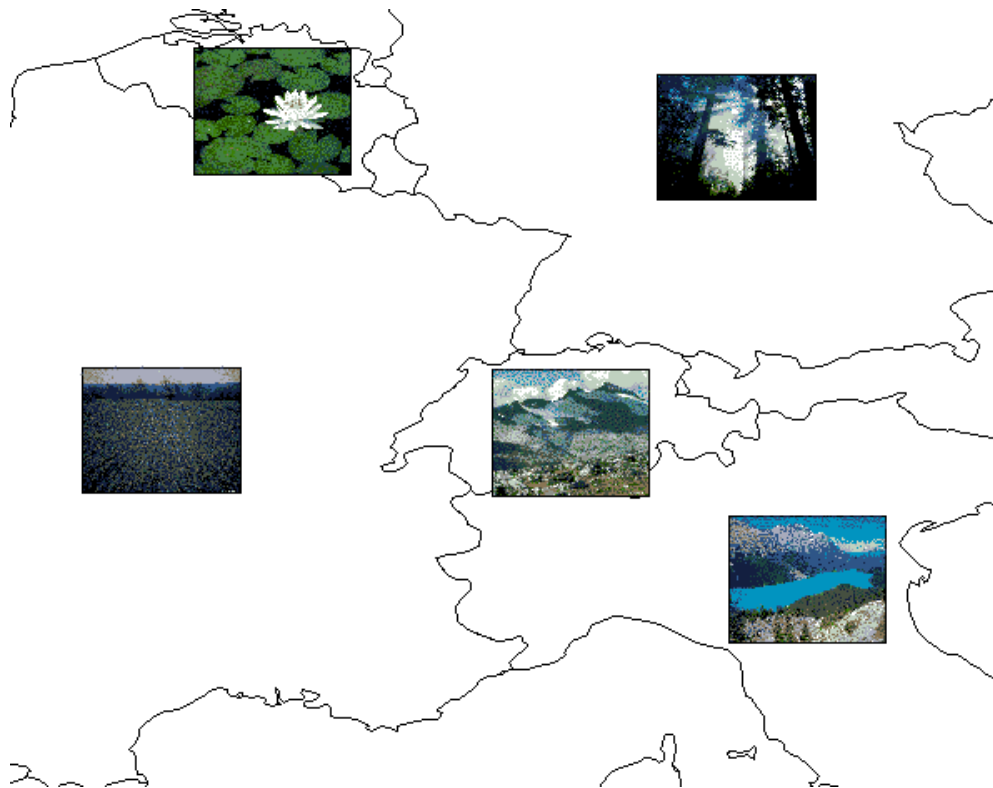
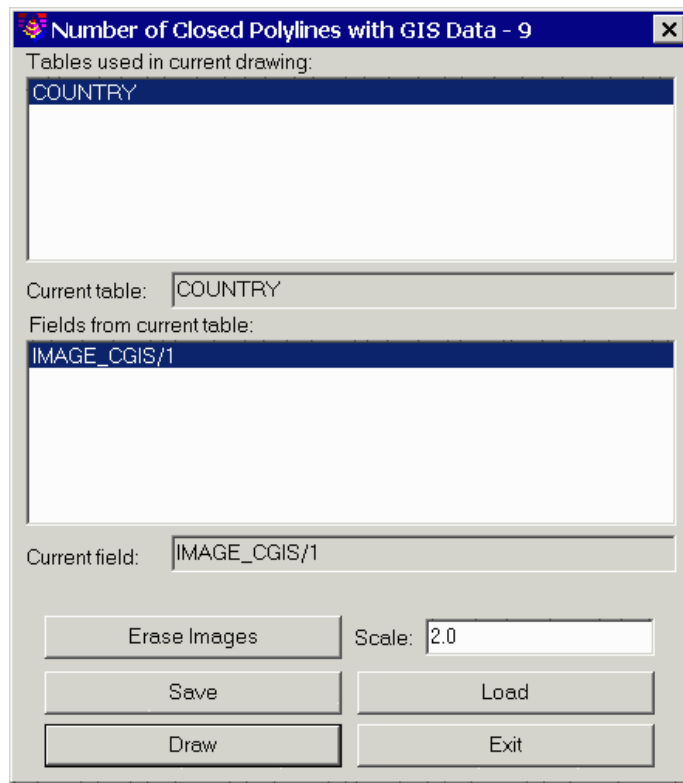
Keyboard Command: labgispt

Prerequisite: Carlson points with GIS attributes, Define GIS Features (.gis) file

Label GIS Polyline: Closed Polyline Image

This command draws images inside the selected closed polylines with attached GIS image files. Images can be assigned to polylines by the GIS Data Editor command.

The program starts by selecting closed polylines in the drawing with GIS data. Then a dialog appears for specifying the image to draw. This dialog displays a list of all the GIS table names found in the selected polylines. First choose a table to process. Then the image fields defined for this table are displayed in the lower list. Only one image can be drawn inside the polyline. The Erase Images button will erase any existing images inside the selected polylines. The settings can be saved to and recalled from a GIS settings file (.gsf) using the Save and Load buttons. Once all the settings are ready, pick the Draw button to draw the images. The images are drawn in the centroid of the polylines.



Example of images drawn inside closed polylines

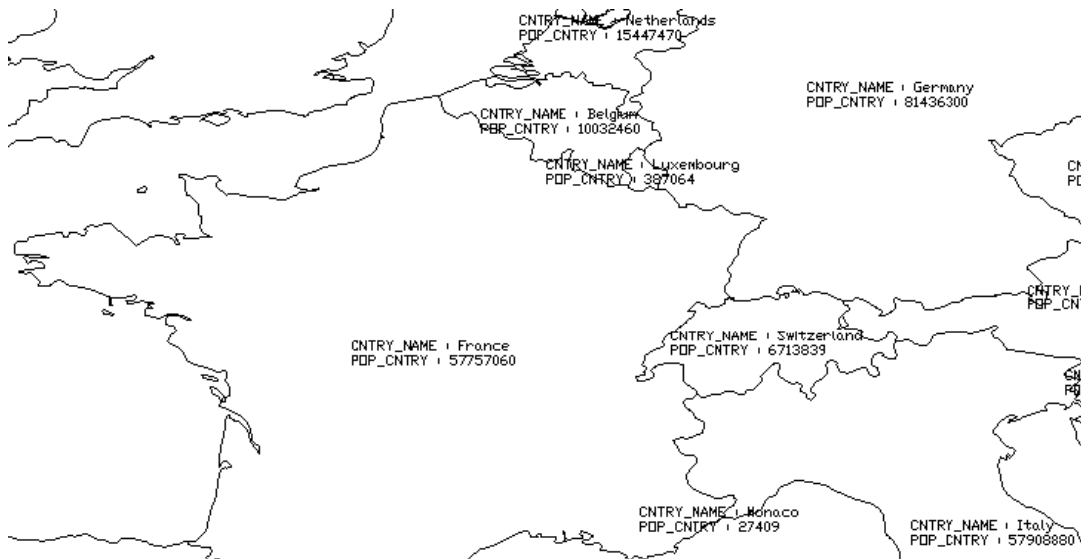
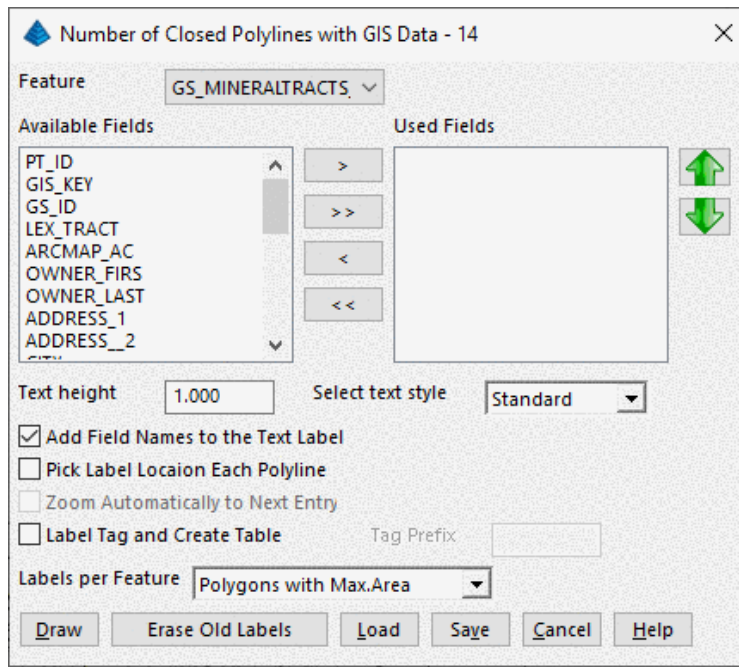
Pulldown Menu Location: GIS Data >Label GIS Data

Keyboard Command: display_polygon_image

Prerequisite: Closed polylines with linked GIS images

Label GIS Polyline: Closed Polyline Data

This command draws text labels for the specified fields inside the selected closed polylines with attached GIS data. The program starts by selecting closed polylines in the drawing with GIS data. Then a dialog appears for specifying the fields to label. This dialog displays a list of all the table names found in the selected polylines. First choose a table to process. Then the fields defined for this table are displayed in the lower left list. To add a field to the label, highlight the field name and pick the > button. The fields names in the lower right list are the fields to be labeled in order. Use the Up and Down buttons to change the field order. The settings can be saved to and recalled from a GIS settings file (.GSF) using the Save and Load buttons. Once all the settings are ready, pick the Draw button to create the labels. The labels are drawn center justified in the centroid of the polylines.



Label Closed Polyline Data result of country name and population fields

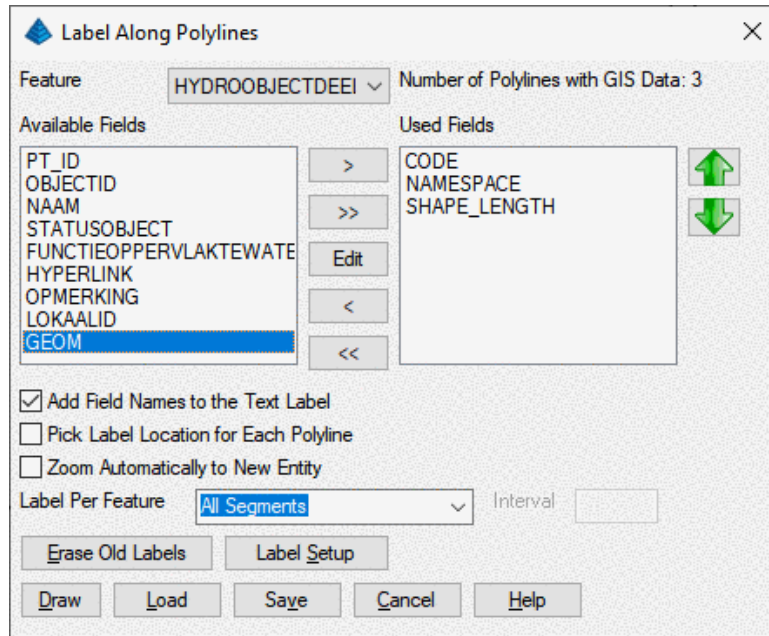
Pulldown Menu Location: GIS Data > Label GIS Data

Keyboard Command: label_polygon_text

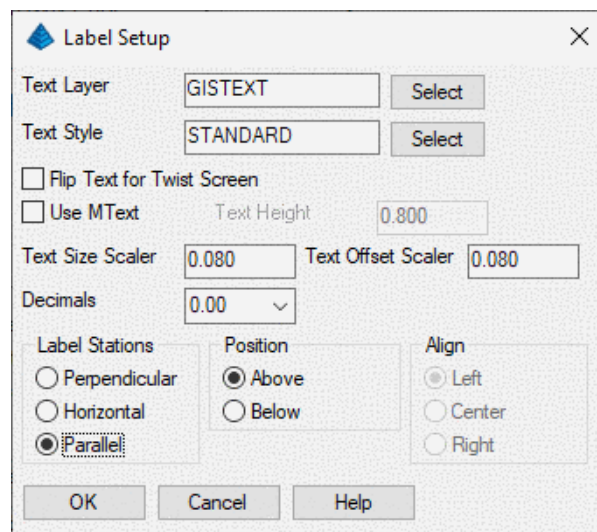
Prerequisite: Closed polylines with linked GIS information

Label GIS Polyline: Label Along Polylines

This command draws text labels for the specified fields along the selected polylines with attached GIS data. The program starts by selecting polylines in the drawing with GIS data. Then a dialog appears for specifying the fields to label. This dialog displays a list of all the table names found in the selected polylines. To label a field, highlight the field from the Available Fields list and pick the > button. Then use the Up/Down buttons to order the fields in the Used Fields list. The Erase Labels option will erase any existing field labels for the selected polylines. The settings can be saved to and recalled from a GIS settings file (.GSF) using the Save and Load buttons. Once all the settings are ready, pick the Draw button to create the labels. The labels are drawn along the polylines, and it is optional to label on all segments, maximum length segment or at interval. Also, it is optional to let the user pick label location for each polylines.



Use the Label Setup button to set parameters for the label in the used box.



Pulldown Menu Location: GIS Data > Label GIS Data

Keyboard Command: label_arc_text

Prerequisite: Polylines with linked GIS information

Links Manager

This command displays the GIS links for the selected entities and includes functionality of the GIS Data Editor, Create Links and Erase Links commands. The command starts by prompting you to select the entities to process. Then the program displays a dialog with a list of all the selected entity types (POINT, LINE, etc.).

To review and edit GIS data, highlight an entity type and the program will then list all the entities for that type. If a table is attached to the entity, the table name is displayed. Otherwise the entity reports "NO TABLE". The current table is shown in a spreadsheet editor at the bottom of the dialog. When you highlight an entity from the list, the linked record in the table is shown in the first column of the spreadsheet. Also the drawing is zoomed to the entity and a highlight box is drawn around the entity. You can use the arrow and magnify glass buttons to pan and zoom the display. The arrow, page up and page down keys also pan and zoom the display.

To add GIS data, highlight an entity from the list to process that has NO TABLE. Then select a table from the table pulldown list. The available tables for this list come from the Template MDB database shown in the top of the dialog. The spreadsheet then shows a record in the first column. Fill in the values and then pick the Create GIS Link button.

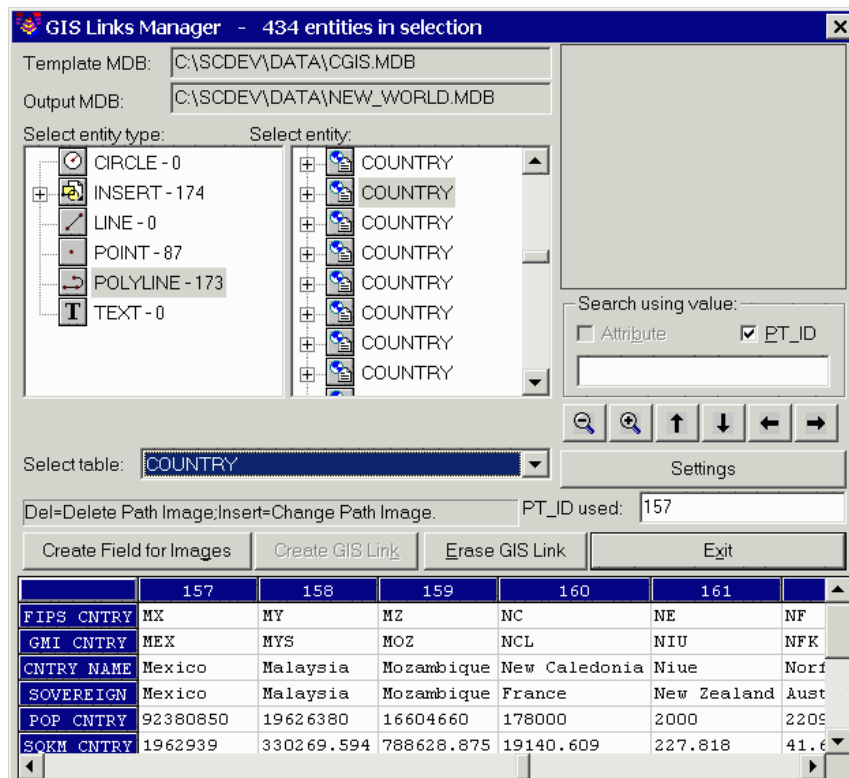
To erase a GIS link, highlight an entity from the list and pick the Erase GIS Link button.

The Create Field for Images button will add a field to the current table for image files. To add an image file, highlight the image field in the spreadsheet and press the Insert (Ins) key.

Under the Settings button you can set filters for the GIS links to process based on layer names, colors, polyline type and link status. You can also set the zoom factors for the screen display.

Prompts

Select objects: Select the entities, with linked GIS information, to process.



Pulldown Menu Location: GIS Data

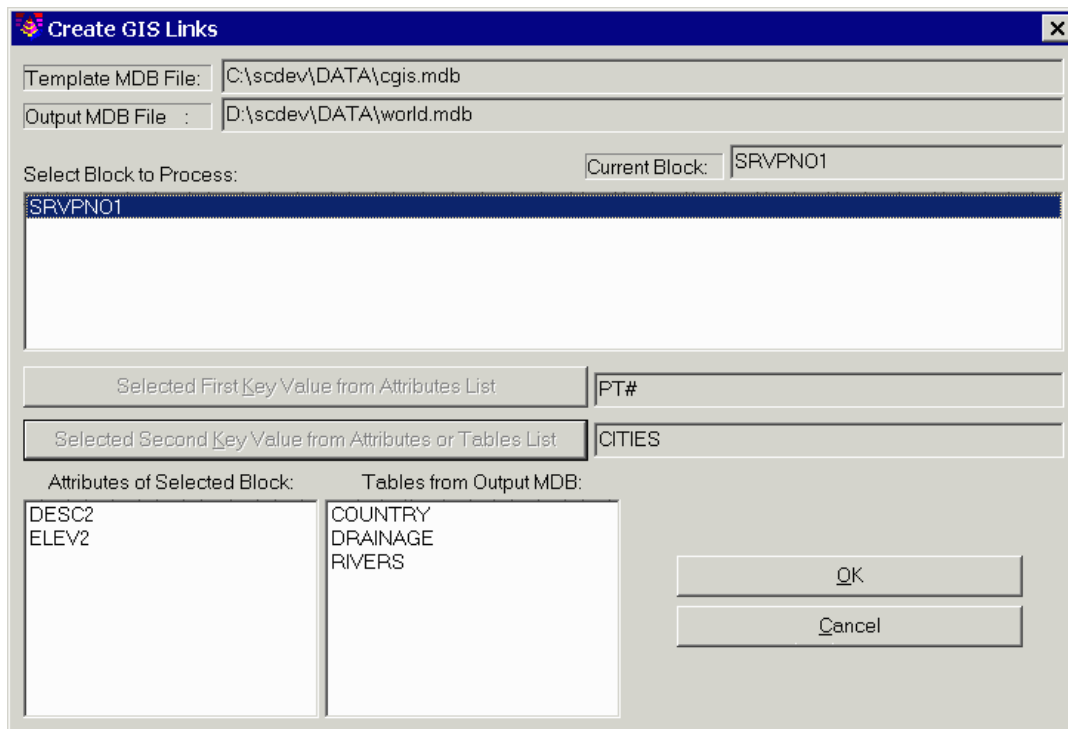
Keyboard Command: link_manager

Prerequisite: Entities with linked GIS information

Create Links

This command makes GIS links between blocks in the drawing and a database table using a key field that is in both the block attributes and the database table. Both the block entities and database records must exist before running this routine.

The routine starts by prompting you to select the block entities to process. Then a dialog appears for choosing the block attribute and table to link. The current template and output database file names are shown at the top of the dialog. Use the GIS Database Settings command to set these file names before running Create GIS Links. The dialog lists all the block names that were found in the entity selection. Choose a block name to process. Then in the lower left of the dialog, there is a list of the attributes for the selected block. Highlight the attribute name that contains the point ID key field for the blocks and then pick the Select First Key Value button. For each block entity, the program will use the value of this attribute to link to the record in database table. This value is matched to the database record using the PT_ID database table field. For example, a block with an attribute value of 402 for the specified attribute name will be linked to the database record with a value of 402 in the PT_ID field.



Next, the database table needs to be specified to either one fixed table name or to table names defined by a block attribute. A list of the available tables in the current output database is displayed. To link all the blocks to one table, highlight the table name from the list and pick the Select Second Key button. Or to link the blocks to various table names based on a block attribute, highlight the attribute name and pick the Select Second Key button. This attribute value for the blocks will then need to contain the database table name. For example, consider a block for electric utility data with two attributes: ID and TABLE. The ID is a number to use as the first key and the TABLE is the table name (i.e. POLE, BOX). Once the key fields are set, pick the OK button to create the links.

Pulldown Menu Location: GIS Data

Keyboard Command: create_links

Prerequisite: Block entities with attribute IDs and a database table with matching IDs.

Erase Links

This command removes all the GIS links from the selected entities (polylines, blocks, etc.).

Pulldown Menu Location: GIS Data

Keyboard Command: erase_links

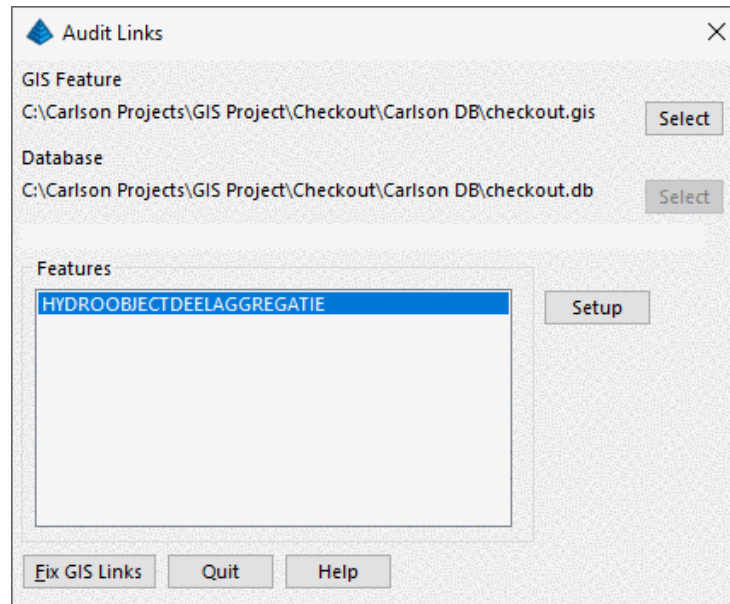
Prerequisite: Entities with GIS links

Audit Links

This command checks the GIS links for the selected entities in the drawing to make sure that the template database, output database and table exist. Any invalid links can be erased from the entities or be fixed by selecting another database or table. For example if a database file (.db) has moved to another directory, then you can use this command to specify the new location.

The routine starts by prompting you to select the entities to check. If no errors are found, then the routine is done. When there are errors, a dialog box appears. Each GIS link is defined by a template database, output database and table. For each combination of these three settings that have an error, this dialog displays the template database, output database and table name from the entities. The number of GIS link combinations with errors is shown in Table Used for Links field (i.e. 1 of 2). The template database is shown at the top. If the template database link is broken, then use the Select New Template DB button to assign another template database file. The output database

also has a Select New Output DB to set the output database file. In the lower left of the dialog is a list of the table names from the output database. You can choose the table to use for the link from this list. The Fix Links for Current Table button will assign the template database, output database and table name from the dialog to all the selected entities. The Erase Links from Table button will remove these broken links from the entities. The Go to Other Table button will process the next GIS link combination with errors.



Pulldown Menu Location: GIS Data

Keyboard Command: audit_links

Prerequisite: Entities with GIS links

Create GIS Polylines by Interior Text

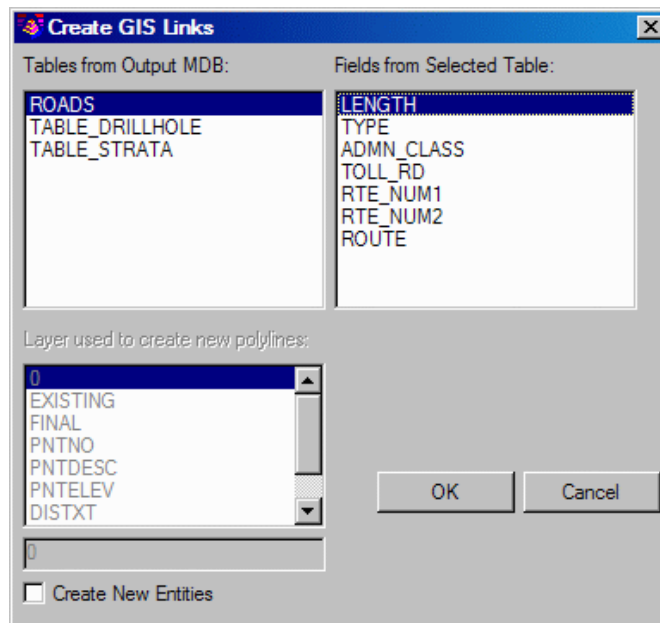
This command works with either Interior Text or Entity Handles. The initial choice of which to work with determines which dialog is displayed. In either case, the dialog box displays Tables from the Output MDB on the left, and the available Fields from that Table on the right. The use of Text gives the option of creating new GIS polylines on a specified layer by taking each text entity and creating a closed polyline around the text from the selected linework, using the text value as a field in the GIS data. If this option is not selected, the GIS data is added to existing polylines. Using Handles does not allow for the option to create new polylines.

Prompts

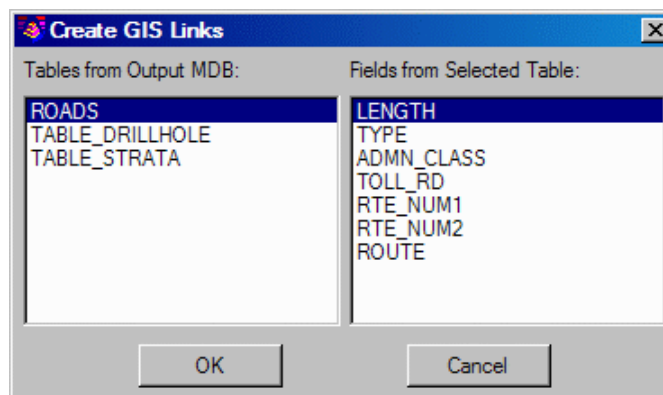
Use as key to match the records from the table Interior Text or Entity Handle[<Text>/Handle]? *press Enter*
Select lines, polylines and text. *select objects*

For the Handle option:

Use as key to match the records from the table Interior Text or Entity Handle[<Text>/Handle]? *press H*
Select lines and polylines *select objects*



Interior Text option dialog



Entity Handle option dialog

Pulldown Menu Location: GIS Data

Keyboard Command: cgis.txt2pl

Prerequisite: Text

Point GIS Editor

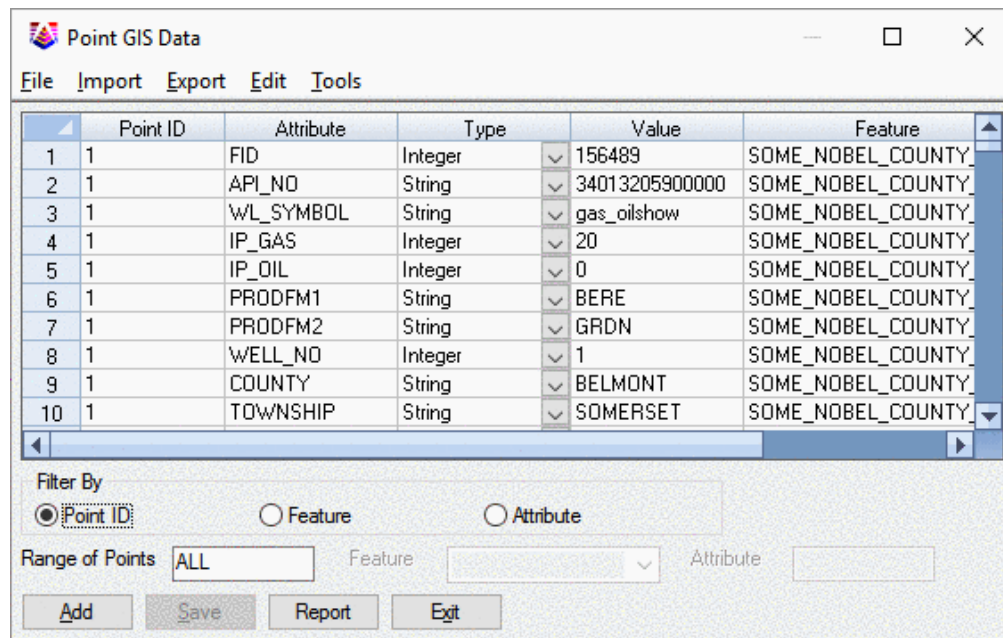
This command can be used to import, export, edit and report GIS data for points. The GIS data is stored in the current coordinate file which must be set to the CRDB format. To set this format, go to Points > Set Coordinate File and enter a file name with the .CRDB extension. To change a current coordinate file into CRDB format, go to Points > Coordinate File Utilities.

The program uses a spreadsheet for the GIS data sorted by point number. Each GIS record contains a point name, attribute name, data type, value and feature. The Point ID references a point name with x, y, z and description from the coordinate file. The Attribute is the name of the GIS data. The Type is the data type of String, Integer for whole numbers and Double for numbers that can have decimals. The Value is the actual GIS data value. The Feature is the name for a set of attributes. For example, Manhole could be a Feature name and the Manhole feature could have Attributes like material, size, condition, etc. The Feature name is optional.

The Filter By settings control the data shown in the spreadsheet. The Point ID method allows displaying data for a specified set of points. The Feature method displays data for the specified feature. The Attribute method displays data for the specified attribute. The Inclusion Perimeter method prompts to select closed polylines and only shows the point data that are inside the selected inclusion polylines.

You can edit directly in the spreadsheet. Use the Add and Delete functions to add or delete records from the spreadsheet. The Save function saves the GIS data to the CRDB. The Report function uses the Report Formatter to make a custom report of the GIS data.

The Import/Export menu has import/export functions to transfer various data with CRDB. These methods are described below. The Edit menu has Find, Replace, Delete Row, Delete Range functions. The Tools menu has a special function to add or subtract a GIS attribute value to the coordinate file elevations such as converting the rim elevations in the coordinate file to inverts by subtracting the depth attribute.



Import from SurvCE GIS Attributes(VTT)

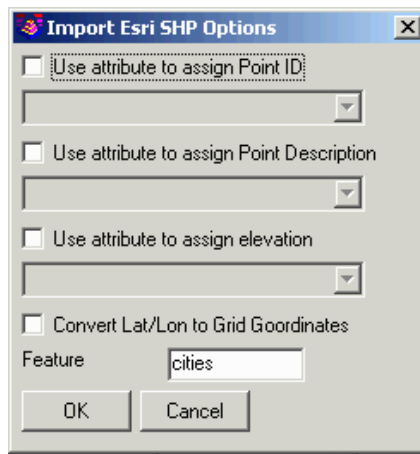
SurvCE stores GIS attribute data in a .VTT file. This file contains point numbers, attribute name, value and feature name which imports directly to the CRDB.

Import from Carlson Link

This method imports GIS data assigned to entities with the GIS Settings > Single File Database.

Import from ESRI SHP File

The SHP method imports from the ESRI SHP file. The import options include selecting which GIS attribute to use for the coordinate file point name, description and elevation. There is also a setting to convert the ESRI SHP lat/lon coordinates to grid coordinates which requires the grid projection to be already defined in the Drawing Setup command.



The MSC method reads the ESRI MSC data that is assigned to drawing entities.

Import from Google Earth File

This function reads GIS data contained in Google Earth KML or KMZ files. The program converts the lat/lon coordinates from the Google file into grid coordinates which requires the grid projection to be already defined in the Drawing Setup command.

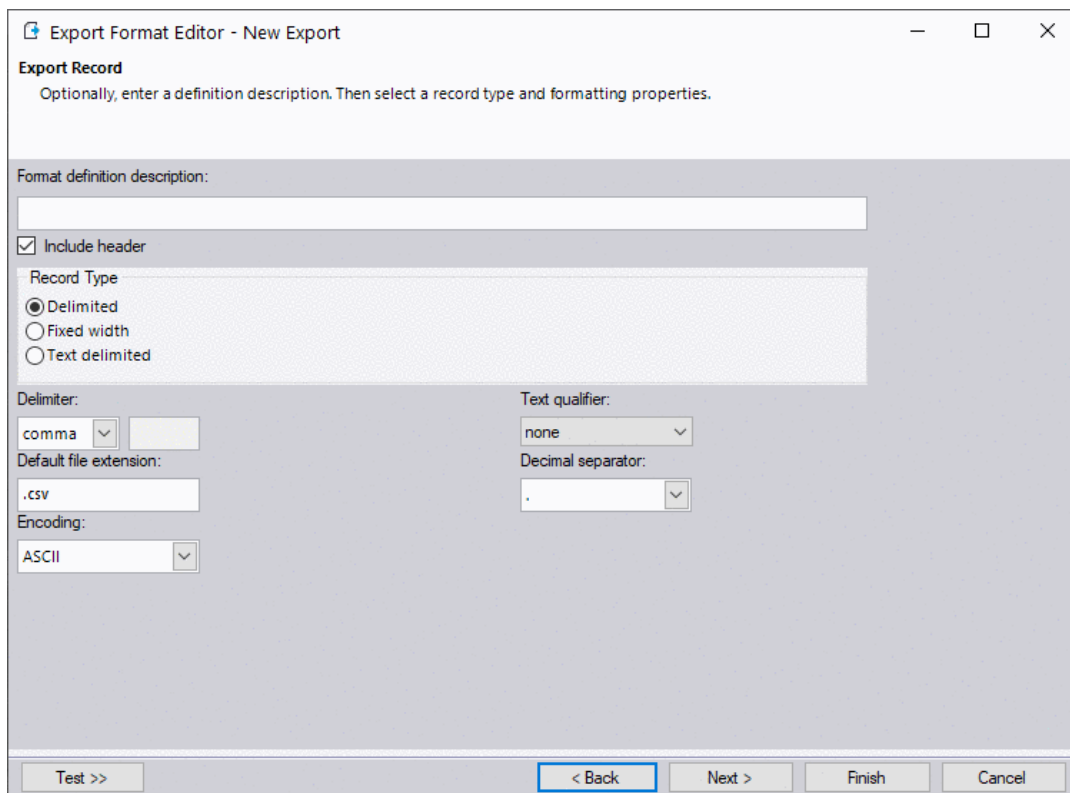
Import from StarNet PST File

This function import a PST file in from StarNet. It is required to define GIS Features first .

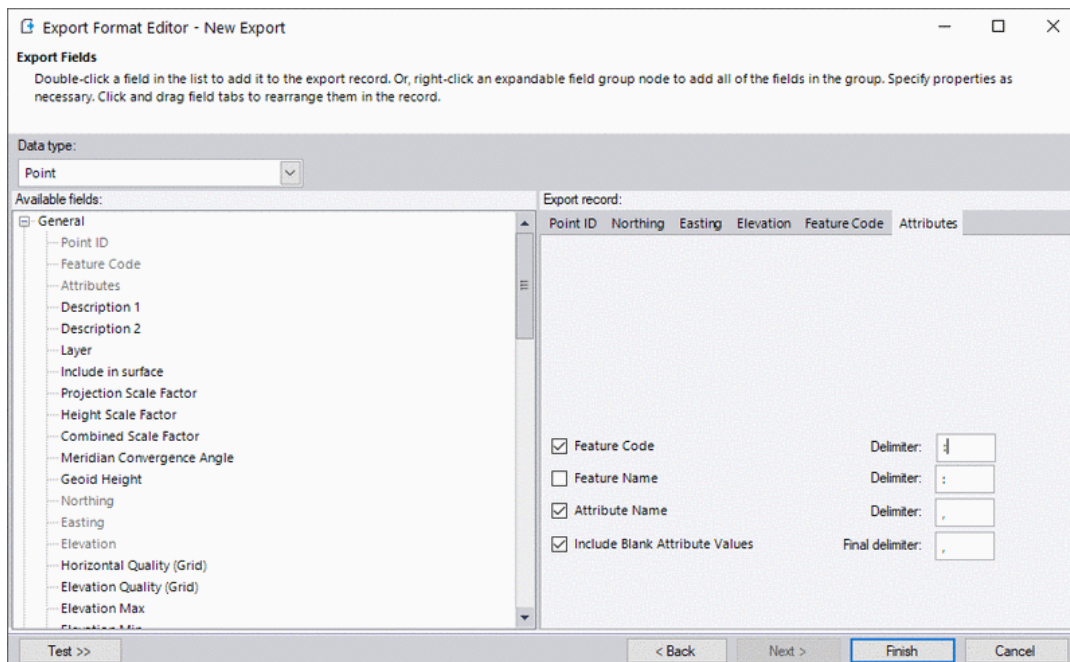
Import from Trimble CSV with Attribute File

This function imports a CSV file in from Trimble Business Center with point name and coordinates followed by attributes. Trimble allows users to modify the format of the CSV file, so it is important to match the format when exporting/importing. The below screenshots show the suggested options when using the Export Format Editor in Trimble Business Center, as well as the suggested import options in Carlson.

It is especially important to note that the encoding of the file should be set to ASCII, and the extension should be set to CSV.

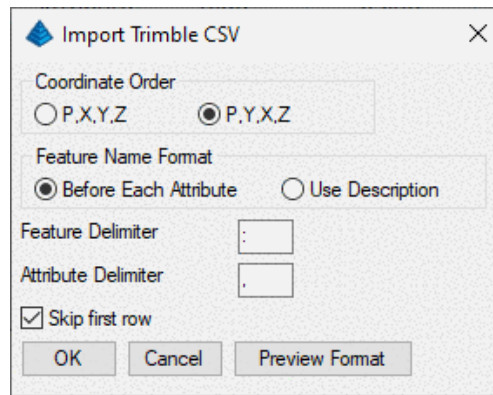


The Point ID, Northing, Easting, Elevation, Feature Code, and Attributes need to be included in the export. The main thing to modify will be the Attributes field, as shown below.

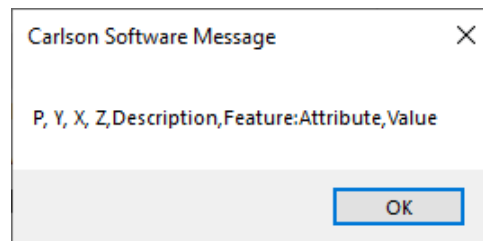


The below image shows the settings needed to import the CSV file exactly as it is configured with the above Export

Settings.



To ensure your import format matches your CSV file, you can click the Preview Format button to see exactly what the program will expect on import. An example is shown below.



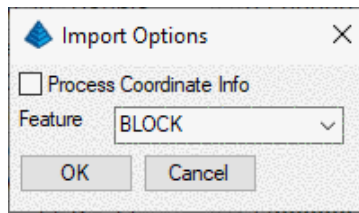
Import from Trimble JXL

This import processes a Trimble JXL file which is an XML based text file.

Import from Text

This function reads the point attributes from a CSV/TXT text file. Before using this command, GIS features should be defined via the Define GIS Features command.

There are two import options on the below dialog - **Process Coordinate Info** and **Feature**. If the points are already defined in the coordinate file (Northing, Easting, Elevation, Description), then the **Process Coordinate Info** option can be disabled to only import the GIS data. If the points are not defined, you can enable this option to import the X,Y,Z,Desc for the points. The **Feature** dropdown list allows you to specify the GIS feature to associate with the newly imported data. Note that only one Feature may be selected, so if your data to import is associated with multiple features, they should be imported separately.



After selecting the appropriate options on the first dialog, the below formatter will appear. You will first need to select the delimiter used in your text file (most commonly a comma if using a CSV file). Each column must be defined used the dropdown list.

If you chose to Process Coordinate Info on the previous dialog, you will see 5 predefined fields: PointID, Northing, Easting, Elevation, and Description. When this option is enabled, you must specify a Northing and Easting for each point. If a PointID is not specified, the program will create one on import.

If you disabled the Process Coordinate Info option, the PointID will be the only predefined option. When this option is disabled, you will only need to specify a PointID for each point.

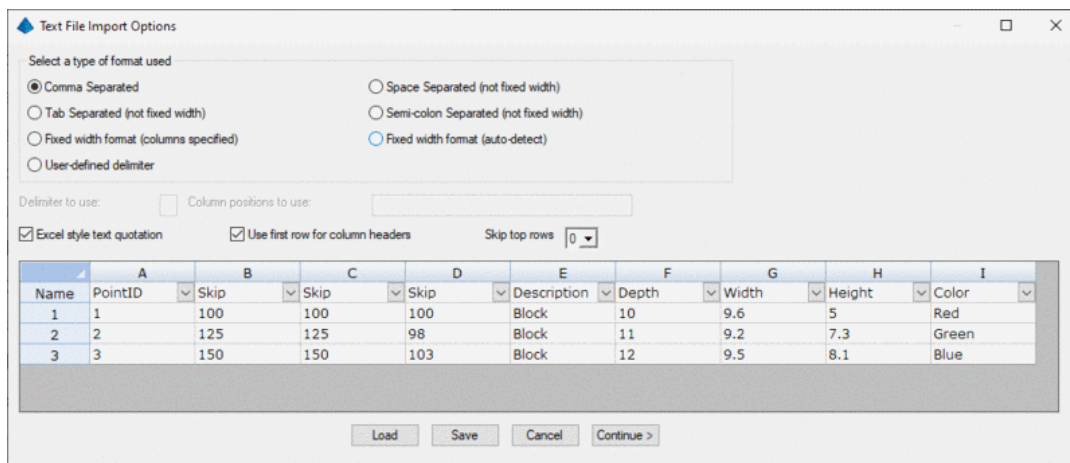
If a column does not need to be imported, you may use the Skip option for the column header to skip that row.

Each column aside from the predefined option is expected to be a GIS attribute. In the below image, four GIS attributes have been defined for Depth, Width, Height, and Color.

Use first row for column headers: When enabled, this option will use the first row of the .csv file to set the column headers. It is important to note when using this option, you should ensure that the column headers for PointID, Northing, Easting, Elevation, and Description are spelled properly, as any misspellings or abbreviations for these columns will prevent proper import.

Skip top rows: This option will skip the specified number of rows when importing. This is useful if you have header information in the .csv file. If you are using the first row for column headers, the row of headers will not be imported even if this value is set to zero.

Excel style text quotation: When enabled, the program will look for quotation marks around text in the .csv file and remove them. For example, if a piece of text is found in the .csv file such as "Red", it will only import Red.



Import from Database

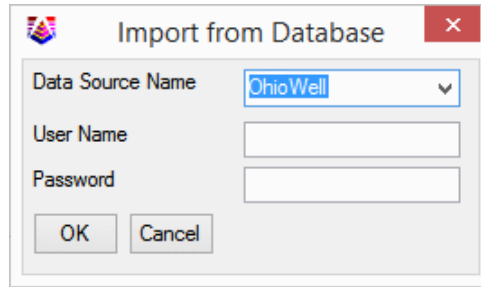
Step 1: Setup DB connection

Data Source Name = Specifies the name of the ODBC data source (DSN). The drop box contains all available DSN in the local host. Please see how to setup DSN at the end of this document.

User Name = Specifies the user name required to login to the RDBMS. (Optional)

Password = Specifies the password required to login to the RDBMS. (Optional)

Usually, DSN is sufficient to connect to the db, but there is some special case that user name and password are also needed. So User name and password are optional.



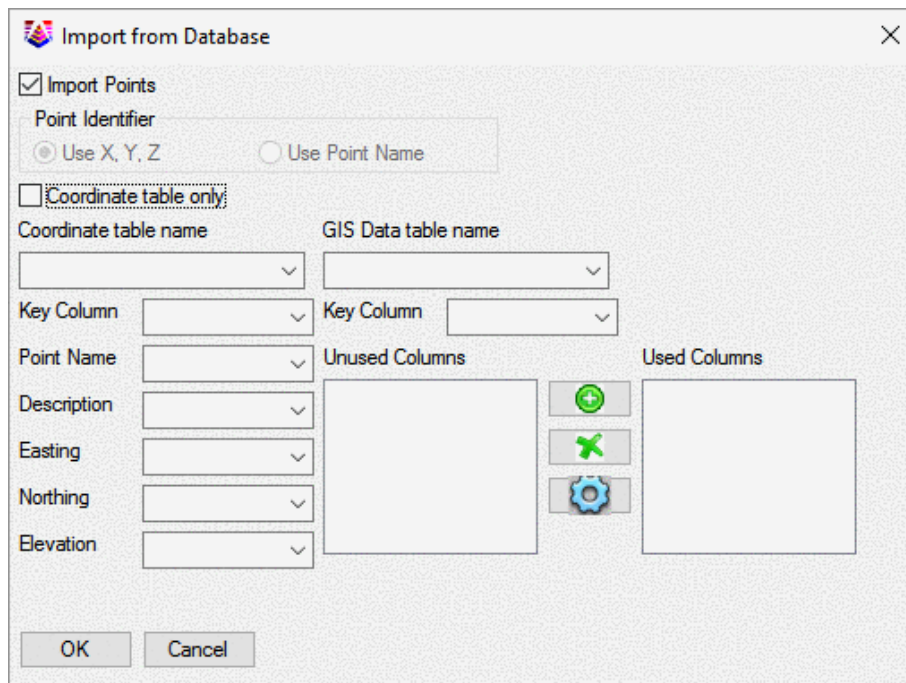
Note: need to hide the password.

Notes: 64 bits application will use 64 bits ODBC drivers. It is possible to connect 64 bits application to 32 bit ODBC driver, but it requires additional setup using ODBC-ODBC Bridge.

Click OK button to continue setup or Cancel button to exit

Step 2: Setup Data matching

Import Points - import points or skip them.



Point Identifier - use for identify points between the user data and CRDB coordinates. If the Import Points toggle box is not selected, then the dialog will prompt user to select the point identifier, either 'Use X, Y, Z' or 'Use Point

Name'.

Use X, Y, Z = use coordinate x, y, z value to identify point.

Use point Name - use point name to identify point.

Coordinate table only - coordinate table contains both coordinate info and GIS Data

Coordinate table name = the table name of the coordinates stored in the user db.

Key Column = a column which will uniquely identify a point.

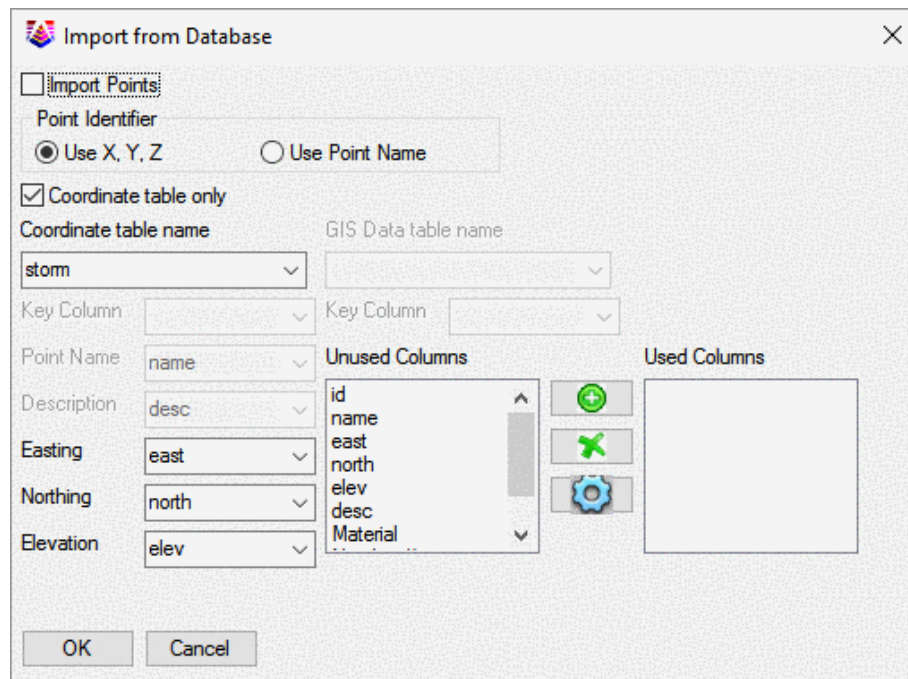
GIS Data table name = the table name of the GIS data stored in the customer's database. This table name will also be used as feature name in the CRDB.

Key Column = a column which will uniquely link to the coordinate table.

Currently, there are 6 fields in CRDB coordinates table, P, N, E, Z, D. P stands for Point Name. N stands for Y-coordinate. E stands for X-coordinate. Z stands for elevation. D stands for description. The Dialog uses Point Name, Description, Easting, Northing, and Elevation to represent the CRDB five columns.

If the 'Import Points' toggle box is selected, then all five fields will be enabled and list the name of the column in the coordinate table. The user needs to select corresponding column name. Column name could be used only once in these five field.

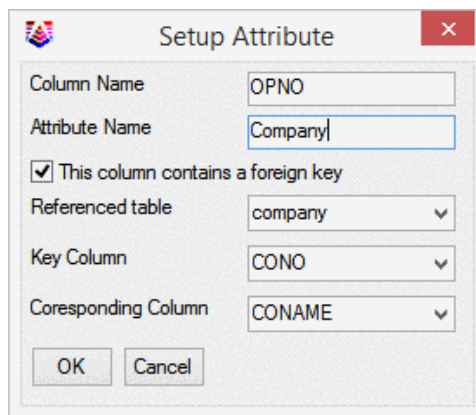
If the toggle box is not selected and if 'Use X, Y, Z' is selected, then 'Easting', 'Northing', and 'Elevation' will be enabled. Otherwise, only 'Point Name' will be enabled.



The 'Unused Columns' and 'Used Columns' are used for selection of the columns in the GIS Data table. The default content of 'Unused Columns' contains a list of columns in the GIS Data table. The user can Add/Delete/Setup each by selected it.

The Setup Button will allow user to setup the association between CRDB GIS Data attributes and the user database.

Step 3: Setup attributes



Column name = name of the column in the GIS Data table

Attribute Name = name of the CRDB GIS Data attribute.

This column contains a foreign key = If the value of the column doesn't contain actual data, but only a reference, then it contains a foreign key. For instance, a DB has a table name 'company' and a table name 'well'. A well belongs to a company, and a company has many wells. 'company' contains an ID and a name. A well will use the company ID to identify a company. In order to import the company name to the CRDB, the user needs to provide the name of the referenced table, the name of the foreign key and the name of the storage column.

Referenced table name = name of the referenced table

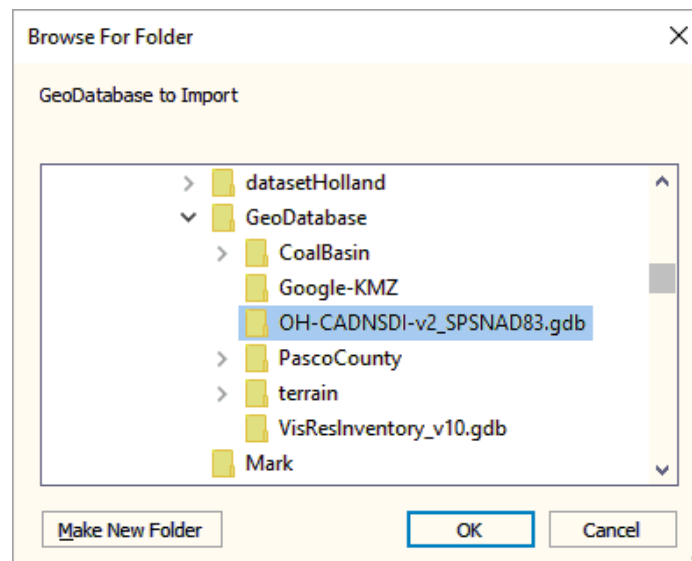
Key Column = name of the referencing key. This combo box contains all columns in the referenced table.

Corresponding Column = name of the column that data actually store. This combo box contains all columns in the referenced table.

If this toggle box is not selected, then actual data is stored in the GIS Data table.

Note: If user selected the same column for the Key Column and Corresponding Column, then just ignore the referenced.

Import ESRI File GeoDatabase to CRDB

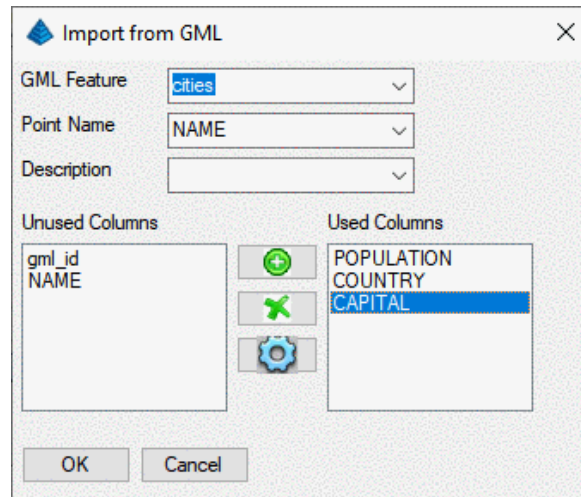


Step 2: Setup Data matching

It has the same interface as import from Database. Please see Import from Database section for detail.

Import OpenGIS GML File to CRDB

Select a feature from the list and select Point Name and Description fields. Also select unused columns and move to the used columns.



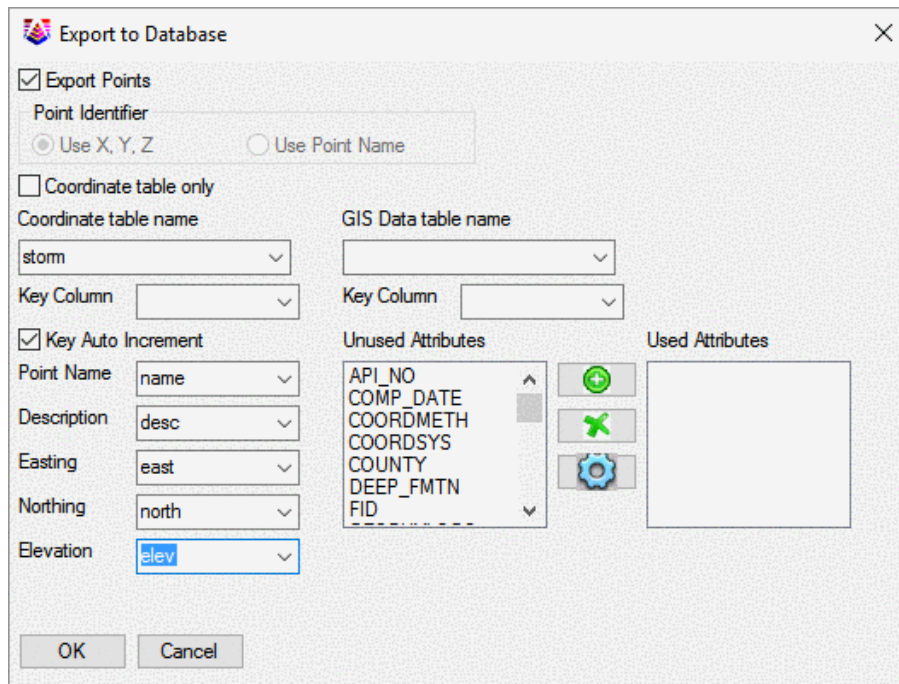
Export CRDB to a database

Step 1: Setup a db connection - this step is the same as import data from a database

Step 2: Setup data matching

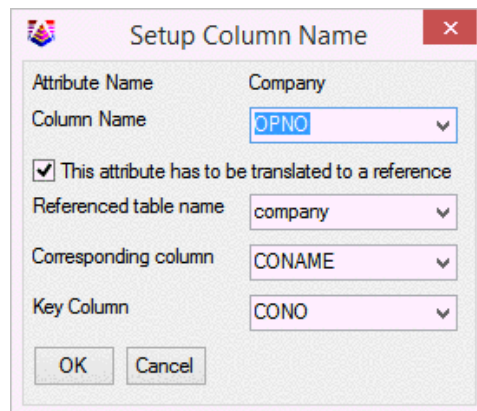
It is similar to import data from a database. Except the following,

Key Auto Increment -the user database will automatically increment the point identifier when a new point is inserted. If this toggle box is not selected, then the CRDB's point name will be used as the primary key. Then the Key Column and Point Name have to be the same column.



Instead of setup column, export function has to setup column. The Attributes list box contains a list of available attribute in CRDB GIS Data table. In order to setup the column, first, the user has selected an attribute name, and then clicks on setup button

Step 3: Setup the column name



Attribute Name = the name of attribute name in the CRDB GIS Data table.

Column Name = the name of column name in the user's GIS Data table.

This attribute has to be translated to a reference = data has to be translated and use the foreign key's value to store in the user's GIS Data table.

Corresponding column = the name of column that will match the value of the attribute

Key column = the name of column that will be the referenced key and stored in the user's GIS Data record.

Click OK button when complete setup. Cancel button to cancel and exit.

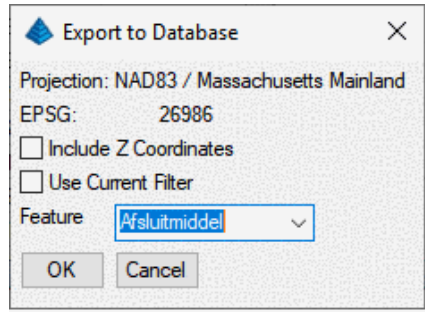
Export CRDB to ESRI File

GeoDatabase Note: make sure the GeoDatabase dataset/table is editable.

It has similar interface as export to a database.

Export CRDB to SQLite/Spatialite DB

After selecting an output file, the Export to Database dialog will display. The program will use the project which is setup in the Drawing Setup dialog. It is optional to include Z coordinates. The user can select any available point feature or select the current feature.



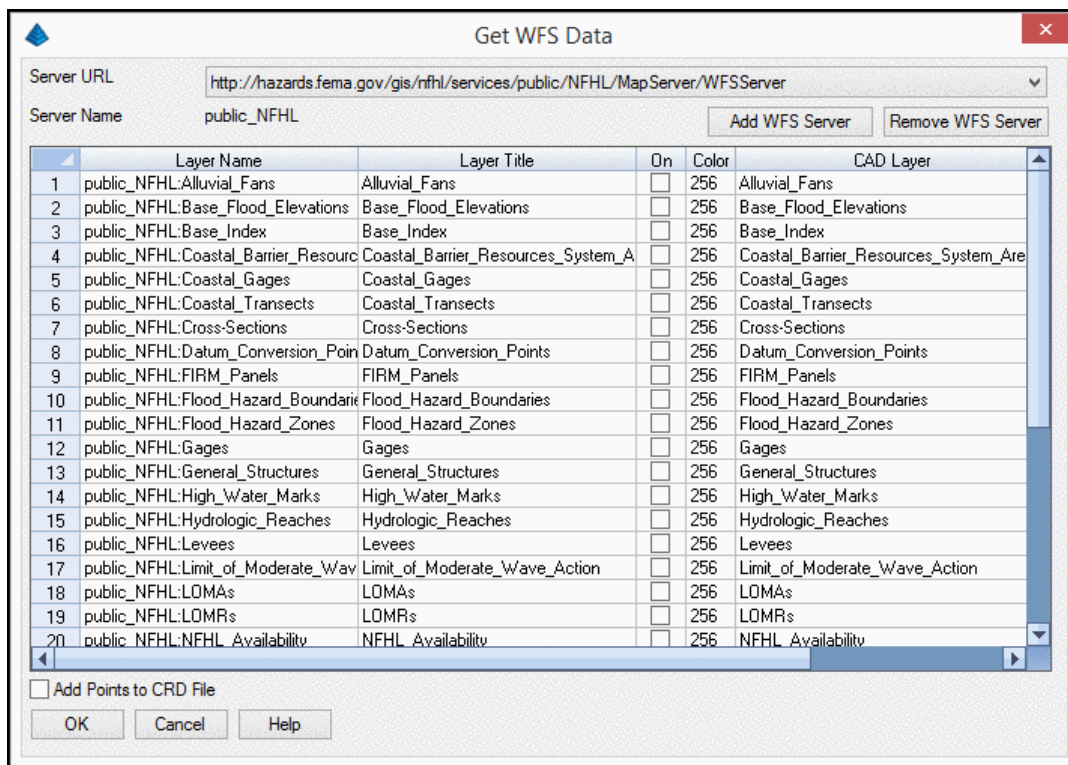
Pulldown Menu Location: GIS Data

Keyboard Command: gis_pt_editor

Prerequisite: Current coordinate file set to CRDB format

Web Feature Service (WFS)

The Web Feature Service (WFS) command provides an ability to fetch GIS data from on-line sources as a means to quickly and easily construct vector map data in and around an area of interest.



Server URL: Select the desired Uniform Resource Locator (URL) from where the geographical data will be obtained. The default URLs include:

- USGS National Map
- USDA NRCS Soil Data
- FEMA

Server Name: A read-only label identifying the name of the WFS server.

Add WFS Server: When clicked, the URL of a WFS server may be input that is subsequently added to the Server URL drop-list.

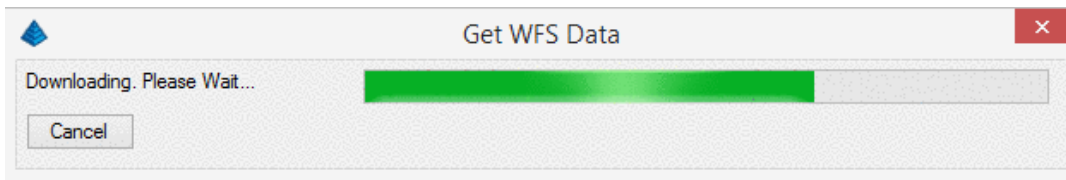
Remove WFS Server: Deletes the currently selected WFS Server from the Server URL drop-list. Prior to the removal of the WFS Server, a confirmation alert box will be displayed asking for confirmation. Any data previously placed into a drawing from the WFS Server will remain in the drawing.

Add Points to CRD File: When enabled and if point data exists for the selected layer, those points will be added to the current coordinate file.

Retrieved Feature(s): As a WFS Server is selected, its list of Features is returned for further manipulation:

- **Layer Name:** The geographical "layer" of data in a read-only format.
- **Layer Title:** The title of the geographical "layer" of data in a read-only format.
- **On:** When enabled, the content of the Feature is eligible to be imported into the drawing based on the bounding box prompts.
- **Color:** Double-click on the desired Feature Color to expose a standard "color-picker" dialog box.
- **CAD Layer:** Key-in the desired Layer name in CAD to house the Feature data and/or double-click on the desired Feature CAD Layer to expose a "layer selector" dialog box.

After layers have been selected, click **OK** and the data for each will be imported.



Note:

- Several freely available WFS addresses can be found at the USGS National Map Viewer.
- Your local governing agency could/should be consulted to see if they publish any data via their own WFS.
- Before starting this command, you might find it useful to position yourself in the desired coordinate location by utilizing the Place Google Earth Image command.
- Data limits fetched by bounding box is based on the following logic:
 1. Point features - only points within the bounding box are returned.
 2. Linear features - only polylines inside or crossing the bounding box are returned.
 3. Polygon features - polygons inside or crossing the bounding box are returned. Additionally, polygons that completely enclose the bounding box are returned (think of the polygon as a solid object upon which the bounding box is etched).

Prompts

Pick first corner of bounding box: *Indicate one corner of a rectangular box that identifies your area of interest.*

Pick second corner of bounding box: *Indicate the other corner of a rectangular box that identifies your area of interest.*

- To import Esri ArcGIS On-line Services data into your drawing, use the Esri ArcGIS Services - Retrieve Map command.
- To import Web Map Service images into your drawing, use the Place Web Map Service (WMS) Image command.

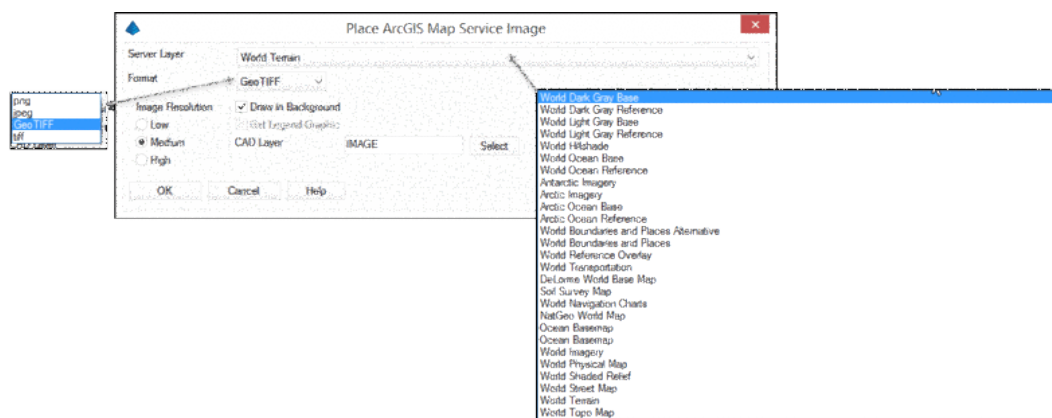
Pulldown Menu Location(s): GIS Data

Keyboard Command: getwfsdata

Prerequisite: Internet connection, Coordinate System Projection set through Drawing Setup

ESRI ArcGIS Services

The ESRI_ArcGIS_Services command provides access to existing ESRI image files.



Server Layer: Contains a list of available image files from the ESRI library

Format Type: Select the image format type from:

- PNG
- JPEG
- GeoTIFF
- tiff

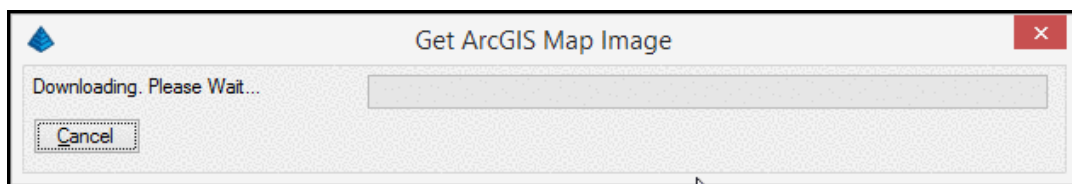
Image Resolution: Sets the resolution for the image as low, medium or high. The resolution may depend on the image selected, display settings and hardware.

Draw in Background: When enabled, sets the draw order to place the image in the background.

Get Legend Graphic: When available, draws a legend graphic.

CAD Layer: Sets the layer for the image file to placed in the drawing.

After selecting the server layer, click **OK** and the data will be imported.



Prompts

Pick first corner of bounding box: *Indicate one corner of a rectangular box that identifies your area of interest.*

Pick second corner of bounding box: *Indicate the other corner of a rectangular box that identifies your area of interest.*

Image File to Save: *Create a name for the image file that will be imported.*

- To import other Web Based Layers into your drawing, use the Web Feature Service (WFS) command.
- To import Web Map Service images into your drawing, use the Place Web Map Service (WMS) Image command.

Pulldown Menu Location(s): GIS Data

Keyboard Command: eagmsExport

Prerequisite: Internet connection, Coordinate System Projection set through Drawing Setup

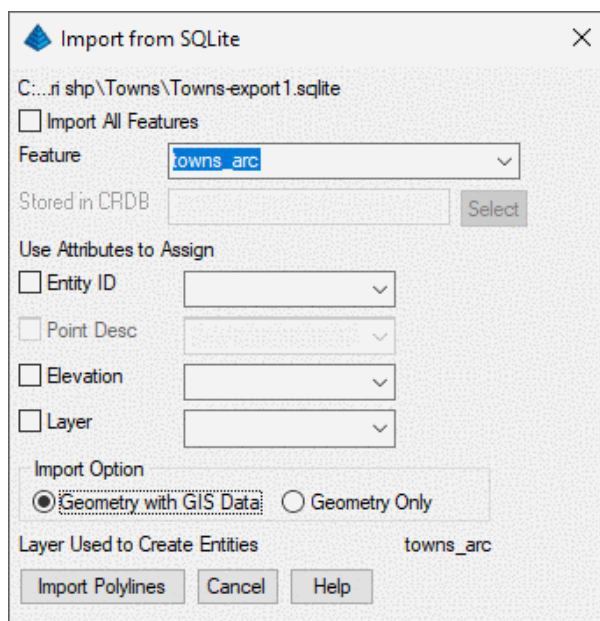
Import SQLite

The import SQLite file commands convert SQLite/Spatialite DB files into Carlson drawing entities. It is optionally write the available attribute data to GIS Data Format which specified in GIS Settings dialog, and the command create GIS links between the drawing entities and the records in the database. If SQLite/Spatial DB is selected as

the GIS Data Format in the Drawing Setup dialog, then the program will only create GIS links between the drawing entities and the records in the database.

There are primarily three types of DB files: Points/ MultiPoint, LineStrings/MultiLineString and Polygons/MultiPolygon. Each will provide different options on Import. Once the GML file is selected, Carlson detects the data contents of the file and sets the dialog options for importing either polygons, linestring or points. LineStrings are brought into Carlson as open polylines and Polygons are as closed polylines in the drawing.

Import from SQLite dialog will list all available tables in the DB File. The user can select import all tables or one of the tables in the list. Then select corresponding fields for the Point ID/Entity ID, Description, Elevation and Layer, etc. The selected fields would be used to fill Carlson internal fields. If no field is selected for Point ID/Entity ID, then the program will create a unique ID for each entity. If no field is selected for Point Desc, then the description will leave blank. If no field is selected for Layer, then the table name will be used as layer name. If the feature type is Point/MultiPoint, then it will allow user to select a name for the CRDB. The program then will also output points geometry and GIS Data to CRDB.



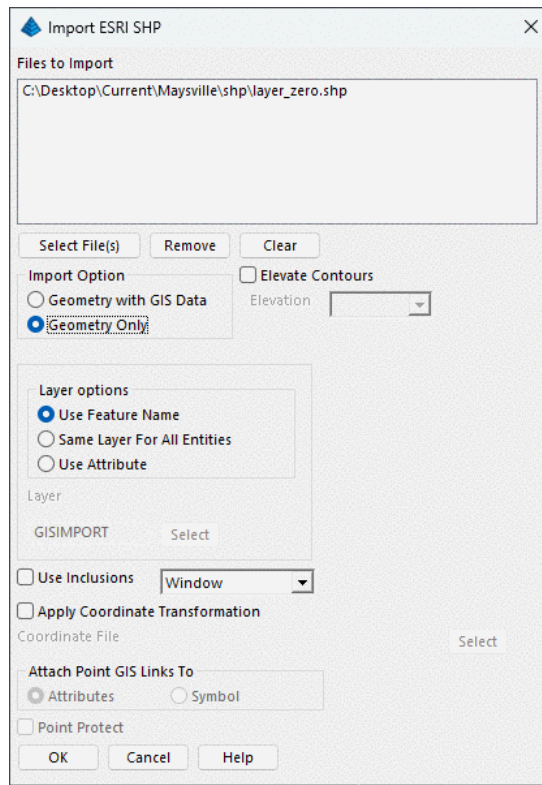
Pulldown Menu Location: GIS Data ->Import GIS Data -> Import from SQLite

Keyboard Command: import_sqlite

Prerequisite:need to specify GIS Feature File, GIS Data Format

Import ESRI SHP File

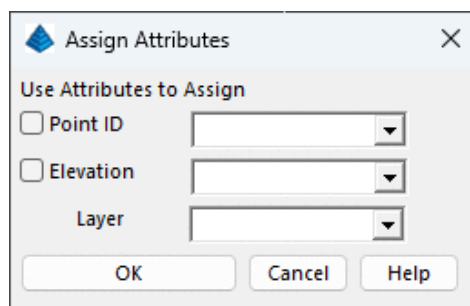
The import function allow one or more SHP to import at once. The Select File(s) can selected more than one file. This command converts ESRI SHP files into Carlson drawing entities and can also optionally write the available attribute data to an external data source which was defined in GIS Database settings. There are two Import Options, Geometry with GIS Data or Geometry only. If the Geometry with GIS Data is selected, then the GIS Data will be imported. Otherwise, if Geometry only is selected, then only the entities will be drawn .



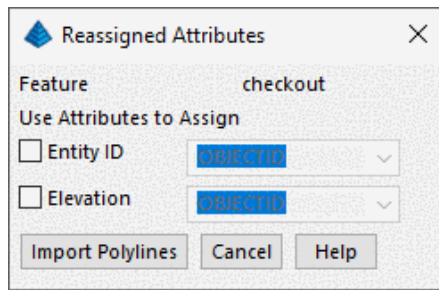
It is required to specify layer name which the entities will be created. There are three options for the layer name, Use Feature Name, Same Layer For All Entities, and Use Attribute Name. If the Same Layer For All Entities is selected, then it is required to enter the Layer Name. If the Use Attribute Name is selected, then it is required to pick a attribute name from the feature's attribute list. This option is only available if there is only one Shapefile selected.

There are primarily three types of ESRI SHP files: Points, Arcs and Polygons. Each will provide different options on Import.

If there is a points SHP in the file list, then the user has the option to setup Attach Point GIS Links to either Attributes or Symbol. Also the user can select Point Protect. Point SHP files are imported in a three step process. The first step uses the Import SHP File command to create a coordinate file (.crd/.crdb) for the points in the SHP file and a corresponding table in the the points database. The second is to use Draw Locate Points to draw the points from the CRD/CRDB file into the drawing. The third step uses Create Links to select the points in the drawing and link the database to these plotted points. Currently, we only allow you to configure attributes for the coordinate file if the Shapefiles have the same schema.



If the SHP file is Arcs or Polygons, then it is possible to reassigned attributes for entity ID or Elevation.



Note: If the SHP file you are Importing is in a different Projection or Units than that specified in the Drawing Setup, then a transformation will occur during Import, as long as the (.PRJ) Projection file is present with the SHP set of files. If there is no (.PRJ) file with the SHP, then no transformations will occur.

Pulldown Menu Location: GIS Data

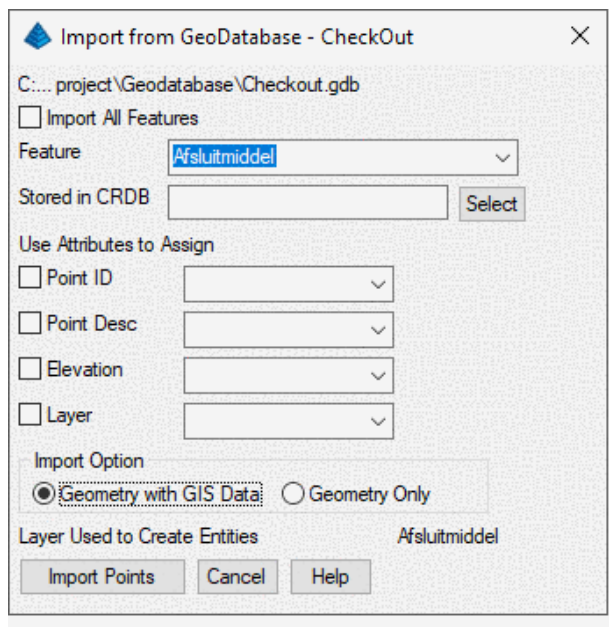
Keyboard Command: import_shp

Import ESRI GeoDatabase

The import File GeoDatabase commands convert File GeoDatabase into Carlson drawing entities. It is optionally write the available attribute data to GIS Data Format which defined in GIS Settings dialog. The command creates GIS links between the drawing entities and the records in the database.

There are primarily three types of GeoDatabase files: Points/MultiPoint, Polylines/MultiLineString and Polygons/MultiPolygon. Each will provide different options on Import. Once the GeoDatabase file is selected, Carlson detects the data contents of the file and sets the dialog options for importing either polygons, polylines or points. Polylines are brought into Carlson as open polylines and Polygons are as closed polylines in the drawing.

Import from GeoDatabase dialog will list all available features in the GeoDB File. The user can select import all features or one of the features in the list. If the user selected one of the feature from the list, then the user can select corresponding fields for the Point ID/Entity ID, Description, Elevation and Layer, etc. The selected fields would be used to fill Carlson internal fields. If no field is selected for Point ID/Entity ID, then the program will create a unique ID for each entity. If no field is selected for Point Desc, then the description will leave blank. If no field is selected for Layer, then the feature name will be used as layer name. If the feature type is Point/MultiPoint and also "Enable Store/Read Point GIS Data to/from SQLite(CRDB) File" is selected in GIS Settings dialog, then the user can select a name for the CRDB. The program will also output points geometry and GIS Data to CRDB.



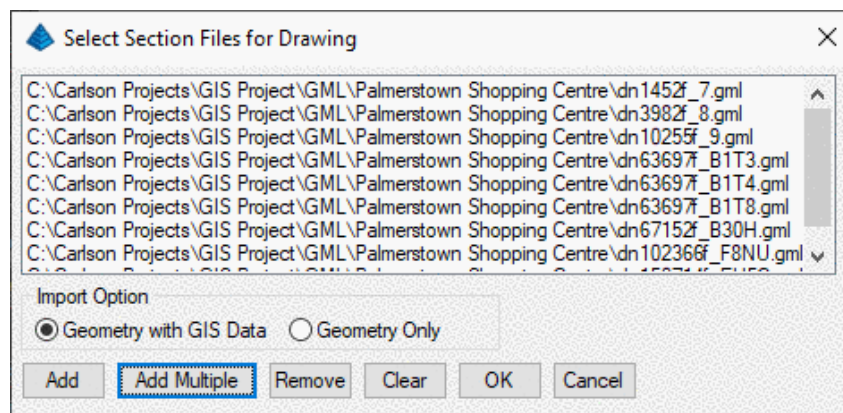
Pulldown Menu Location: GIS Data -> Import GIS Data -> Import from ESRI GeoDatabase

Keyboard Command: import_geodb

Prerequisite: need to specify GIS Feature File, GIS Data Format

Import OpenGIS GML

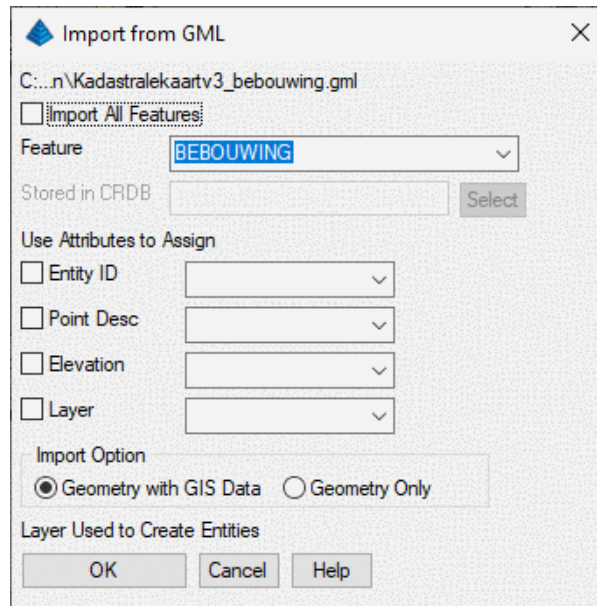
This command converts Open GIS GML files into Carlson drawing entities and can also optionally write the available attribute data to GIS Data Format which defined in GIS Settings. The command creates GIS links between the drawing entities and the records in the data source.



There are primarily three types of GML files: Points/Multipoint, LineStrings/MultiLineString and Polygons/MultiPolygon. Each will provide different options on Import. Once the GML file is selected, Carlson detects the data contents of the file and sets the dialog options for importing either polygons, linestring or points. LineStrings are brought into Carlson as open polylines and Polygons are as closed polylines in the drawing.

If single file has been selected, then it would display Import from GML dialog. This dialog will list all available features in the GML File. The user can select import all features or one of the features in the list. Then select corresponding fields for the Point ID/Entity ID, Description, Elevation and Layer, etc. The program uses the

GML feature name as the table name in the data source. The selected fields would be used to fill Carlson internal fields. If no field is selected for Point ID/Entity ID, then the program will create a unique ID for each entity. If no field is selected for Description, then the description will leave blank. If no field is selected for Layer, then the table name will be used as layer name. If the feature type is Point/MultiPoint, then it will allow user to select a name for the CRDB. The program then will also output points geometry and GIS Data to CRDB.



If multiple files have been selected, then it will not prompt for importing setting. It will automatically create a unique ID for each entity and use the feature name as the default layer name.

Pulldown Menu Location: GIS Data -> Import GIS Data -> OpenGIS GML

Keyboard Command: import_gml

Prerequisite: need to specify GIS Feature File, GIS Data Format.

Import GIS Data from SurvCE

This command reads GIS attribute data collected and imports it into the drawing and embeds it within the point blocks in the drawing as Esri MSC that can be read directly by ArcGIS. There are two import methods for SurvCE and TDS. With SurvCE, the setup is to have a coordinate file (.CRD) set current, the points drawn in the drawing, an applicable attribute definition file (.GIS), and a file that is storing the attribute values (.VTT), that was created by SurvCE. You are prompted to pick the GIS file, and then the data from the VTT is imported and embedded within the matching point blocks in the drawing. With TDS, the process is the same except that the attribute data comes from a TDS .RW5 or .RAW file instead of a .VTT.

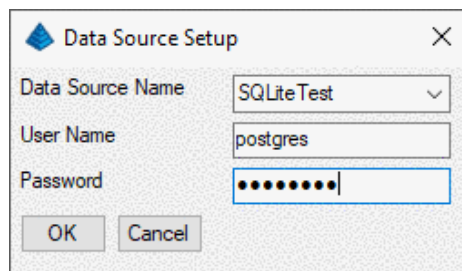
Pulldown Menu Location: GIS Data

Keyboard Command: survce2msd, tds_gis

Prerequisite: GIS file, CRD file, VTT file, or TDS .RW5/.RAW, and points in drawing

Import ODBC data source

This command use ODBC Connection to connect third party database and import database's GIS Data into Carlson drawing entities. It also optionally write the available attribute data to different format which defined in GIS Settings dialog.

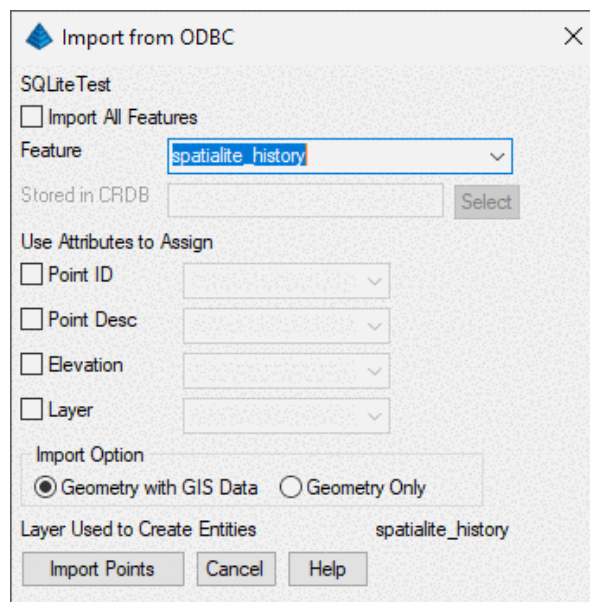


Data Source Name - a data source is a source of data combined with connection information that is required to access that data. Please see How to Setup ODBC Data Source

User Name - RDBMS login name

Password - RDBMS password

After clicked on OK, the program will connect to the data source and display the following setup dialog. This dialog will list all available features in the data source. The user can select import all features or one of the features in the list. Then select corresponding fields for the Point ID/Entity ID, Description, Elevation and Layer, etc when only one of the tables is selected. The selected fields would be used to fill Carlson internal fields. If no Point ID/Entity ID is selected, then it will use the table's primary key as the Point ID/Entity ID. If no field is selected for Description, then the description will leave blank. If no field is selected for Layer, then the table name will be used as layer name. If the table's geometry type is Point/MultiPoint and also "Enable Store/Read Point GIS Data to/from SQLite(CRDB) File" is selected in GIS Settings dialog, then it will allow user to select a name for the CRDB. The program then will also output points geometry and/or GIS Data to CRDB.



Currently, the program only supports the following geometry types, Points, LineStrings, Polygons, MultiPoint, MultiLineString and MultiPolygon. Once the table is selected, Carlson detects the geometry types of the table and sets the dialog options for importing either points, linestrings, or polygons. LineStrings/MultiLineString are brought into Carlson as open polylines and Polygons/ MultiPolygon are as closed polylines in the drawing.

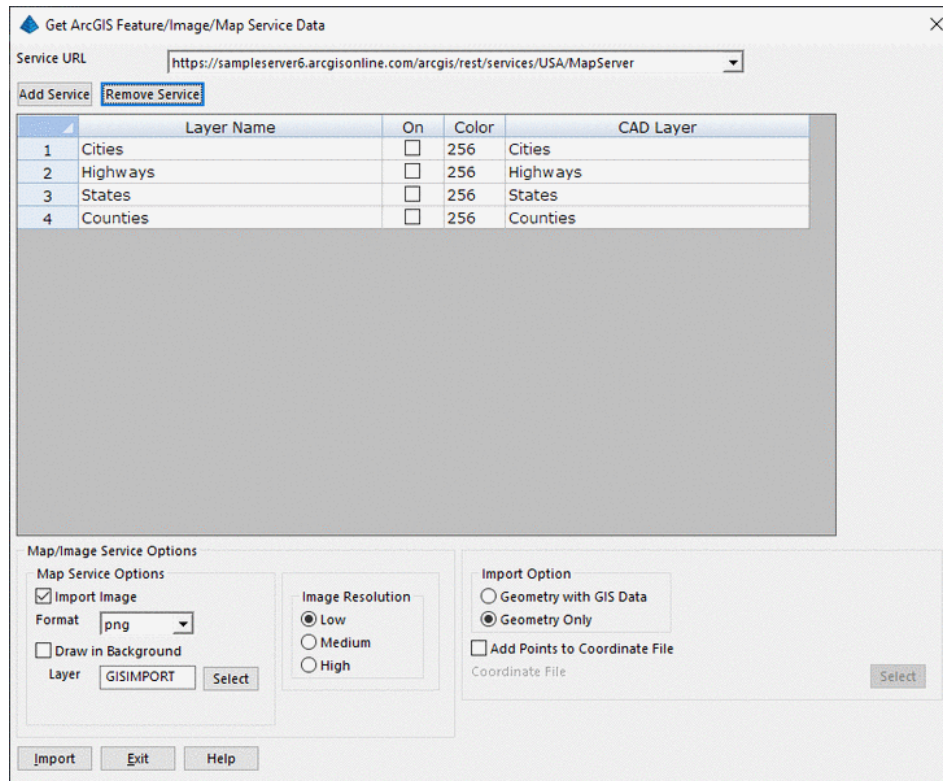
Pulldown Menu Location: GIS Data->Import GIS Data-> Import ODBC

Keyboard Command:import_odbc

Prerequisite: need to specify GIS Feature File, GIS Data Format and ODBC data source.

ArcGIS Feature/Image/Map/Services

This command provides access to ESRI Services which includes Feature Services, Image Services and Map Services. It also provide an ability to fetch GIS Data from on-line Feature/Map Services as a means to quickly and easily construct vector map data in and around an area of interest. Currently, the program only allows to import Image from the Image Services.



Service URL: Contains a list of available services.

Add Service: when clicked, the URL of a Service may be input that is subsequently added to the Server URL drop-list.

Remove Service: Delete the currently selected service from the Service URL drop-list. Prior to the removal of the Service, a confirmation alert box will be displayed asking for confirmation. Any data previously placed into a drawing from the Service will remain in the drawing.



As a Service is selected, its list of Layers is returned for further manipulation:

- Layer Name: The geographical "layer" of data in a read-only format.
- Layer Title: The title of the geographical "layer" of data in a read-only format.
- On: When enabled, the content of the Feature is eligible to be imported into the drawing based on the bounding box prompts.
- Color: Double-click on the desired Feature Color to expose a standard "color-picker"
- CAD Layer: Key-in the desired Layer name in CAD to house the Feature data and/or double-click on the desired Feature CAD Layer to expose a "layer selector" dialog box.

If **Geometry with GIS Data** is selected in Import Option, then Entity ID, Description, Elevation and Layer will be added to the spreadsheet. These options contains a list of fields in the service layer, and the field is going to be used to assign to Entity ID, Elevation and Layer. If the layer's geometry type is a point, then Description will also be included.

After layers have been selected, click **OK** and the data for each will be imported.

If Service type is Map/Image Service, then the following option will be available

Format Type:Select the image format type from:

- PNG
- JPEG
- GeoTIFF
- tiff

Image Resolution: Sets the resolution for the image as low, medium or high. The resolution may depend on the image selected, display settings and hardware.

Draw in Background: When enabled, sets the draw order to place the image in the background.

CAD Layer: Sets the layer for the image file to placed in the drawing.

Add Points to CRD File: When enabled and if point data exists for the selected layer, those points will be added to the current coordinate file.

Prompts

Pick first corner of bounding box: Indicate one corner of a rectangular box that identifies your area of interest.

Pick second corner of bounding box: Indicate the other corner of a rectangular box that identifies your area of interest.

Image File to Save: Create a name for the image file that will be imported.

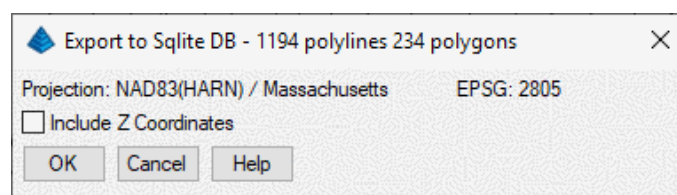
Pulldown Menu Location(s): GIS Data

Keyboard Command: getesrirest

Prerequisite: Internet connection, Coordinate System Projection set through Drawing Setup

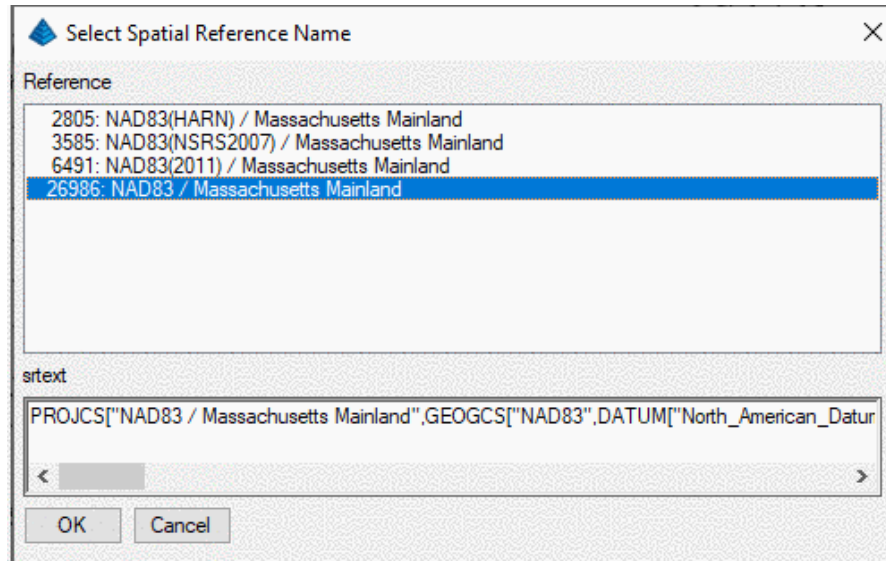
Export SQLite File

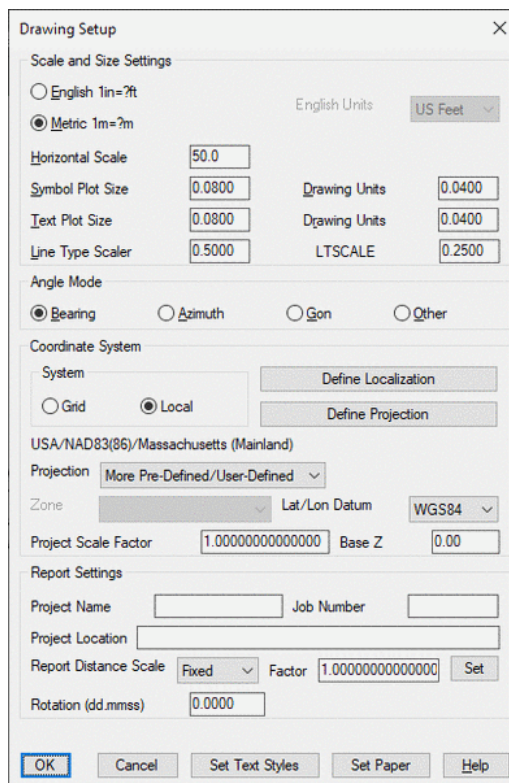
This command create a SQLite file from the entities in the drawing. A dialog shows the drawing's projection and EPSG code and an option to include Z coordinates. The function will export all entities that has linked database information. Any entity that has no linked database information would not be exported.



The command use the Projection and Units that specified in the Drawing Setup to search corresponding EPSG codes in the SQLite/Spatialite reference system. The system can resolved the EPSG code from the Pre-Defined projection and the build-in projection for State Plane 83, State Plane 27 and UTM. If the system can't resolve an unique EPSG code, then it will prompt a dialog and let the user selected the code.

For example, if a predefined project USA/NAD83(86)/Massachusetts (Mainland) is selected in the Drawing Setup, then the following dialog will be display. The dialog list all projection that is related to the predefined projection. To see more detail information about a projection, just move the cursor to the project. The srtext box will display detail information.





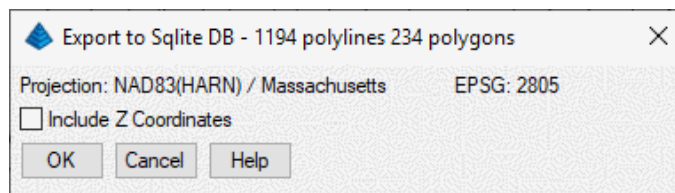
These SQLite files can be imported into QGIS layer.

Prompts

Specify Name for SQLite File dialog select .sqlite name

Select EPSG code: select entry

Export Carlson Entities to SQLite File dialog choose settings, click OK



Pulldown Menu Location: GIS Data -> Export GIS Data -> Export SQLite

Keyboard Command: export_sqlite

Prerequisite: None

Export SHP File

This command creates a SHP file from the selected entities in the drawing. After selecting entities to be converted, a dialog shows the number of Points, Polylines (Arcs) and Closed Polylines (Polygons) found in the drawing selection set. Those Points, Arcs and Polygons with database information linked are displayed with their database

table names. Any Points, Arcs and Polygons without linked database information display as unknown. The Coordinate System setting chooses between creating a SHP file using the northings/eastings from the drawing or using lat/lon by converting the drawing coordinates to lat/lon with the projection parameters from Drawing Setup.

Highlight the Point, Arc and Polygon tables to output or selects Export All to select all entities including the UNKNOWN entities to export into SHP files. The Export SHP File commands outputs all entities selected into SHP files with the same name as their table name into a subdirectory selected. Also Points can be stored in the ESRI Arcview database as 3D X, Y and Z coordinates when Include Z Coordinates is toggled on. SHP files do not have arc entities. So the export routine will convert arcs and polyline arcs into a series of small chords segments. The Offset Cutoff field sets the maximum horizontal shift allowed between the original arc and the chord segments.

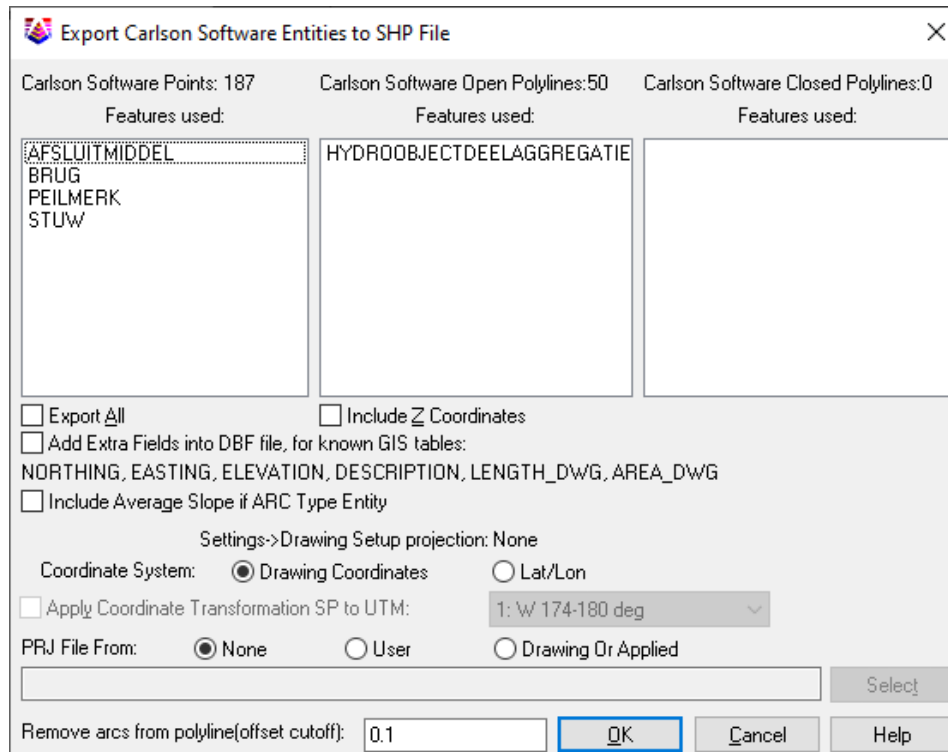
These SHP files can be imported into ESRI's Arcview product. Database GIS links in Carlson are converted to SHP files by storing the GIS database information into DBF files for ESRI's Arcview product to read and link to.

Prompts

Specify Name for SHP File dialog *select .SHP file name*

Select objects *select entities*

Export Carlson Entities to SHP File dialog *choose settings, click OK*



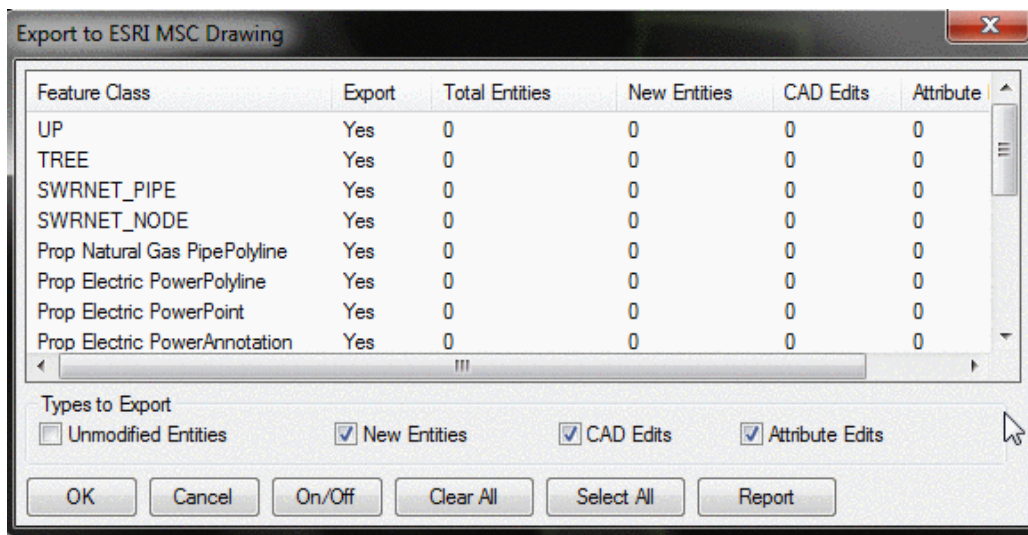
Pulldown Menu Location: GIS Data

Keyboard Command: export_shp

Prerequisite: None

Export DWG File with Esri MSC

This command is used to create a new drawing file that contains Esri MSC Feature data.



The drawing is scanned for MSC data and further, which are new entities with MSC, which are entities with edited geometry, and which are entities with edited attributes. The user specifies which are to be included in the new drawing file. On OK, a new drawing file name is specified.

The drawing is scanned for MSC data and the list is populated with represented Feature Classes. You can select which ones to include in the Export (Export Yes/No column). The check boxes at the bottom allow you to choose whether to Export the unmodified entities for the selected Features, new entities, entities with CAD edits, such as Trim, Extend, Move, etc, and entities with edited attributes. On OK, a new drawing file name is specified. There is also a Report function to review the changes and to make a record of these transactions.

One possible application of this command is to create a DWG from ArcGIS with its Export to CAD tool, open the drawing in Carlson and edit it, and then use this command to send the edits back to ArcGIS as a new DWG with MSC.

Pulldown Menu Location: GIS Data

Keyboard Command: export_msd

Prerequisite: drawing with MSC

Export ESRI Projection File

This command creates an ESRI format Projection File (ESRI PRJ) from the projection definition in Drawing Setup which must be setup before running this routine. The program prompts for the .PRJ file to create.

Pulldown Menu Location: GIS Data

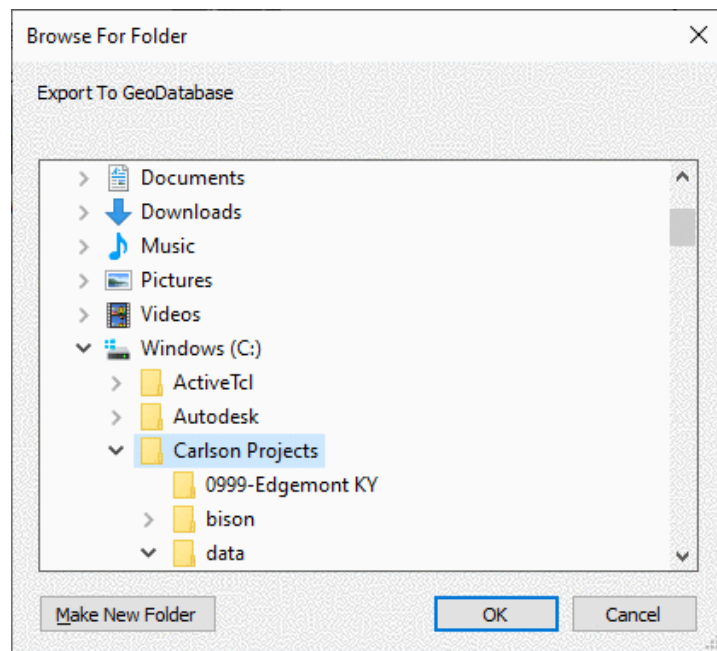
Keyboard Command: write_esri_prj

Prerequisite: projection defined in Drawing Setup

Export ESRI File GeoDatabase

This command creates an ESRI GeoDatabase File from the entities in the drawing, and it exports all that have linked database information. Entities that have no linked data will not be exported

The following dialog will display: and a directory with extension name .gdb has to be selected. If it doesn't exist, then click on "Make New Folder"



Select an existing folder or create a new one with the **Make New Folder** option. The folder must have the extension .GDB.

The ESRI GeoDatabase files will be created from the entities contained in the drawing provided they are linked to a database.

The command will not execute if the original drawing's GIS data format is ESRI File GeoDatabase.

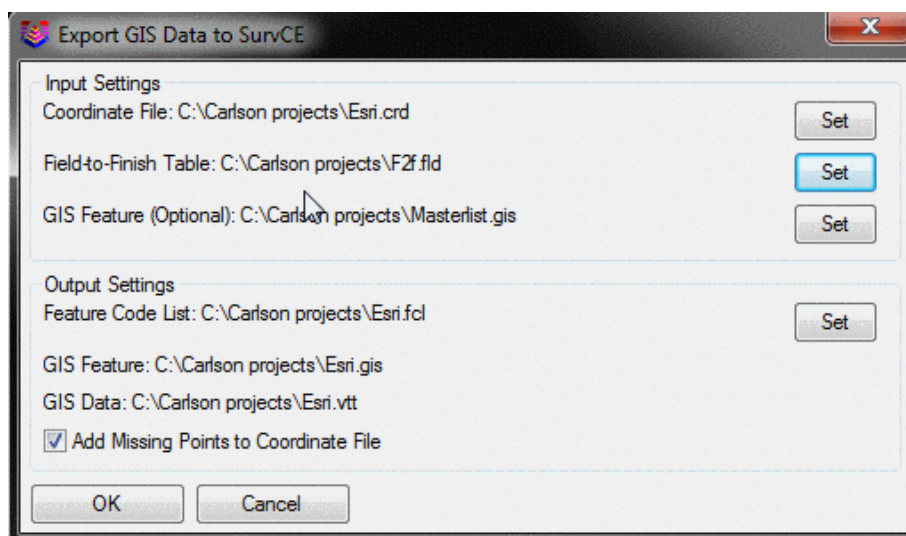
Pulldown Menu Location: GIS Data -> Export GIS Data -> Export ESRI File GeoDatabase

Keyboard Command: geodb_export

Prerequisite: Drawing Projection Set in Drawing Setup

Export GIS Data to SurvCE

This command is used to set up a SurvCE Feature Code Library (.FCL) with attributes from points in a drawing with Esri MSC data.



In this dialog box, the coordinate file is specified, as well as the Field to Finish file being used. The name of the

SurvCE FCL file is then specified. Picking OK prompts the user to select the points in the drawing with Esri MSC data.

Pulldown Menu Location: GIS Data

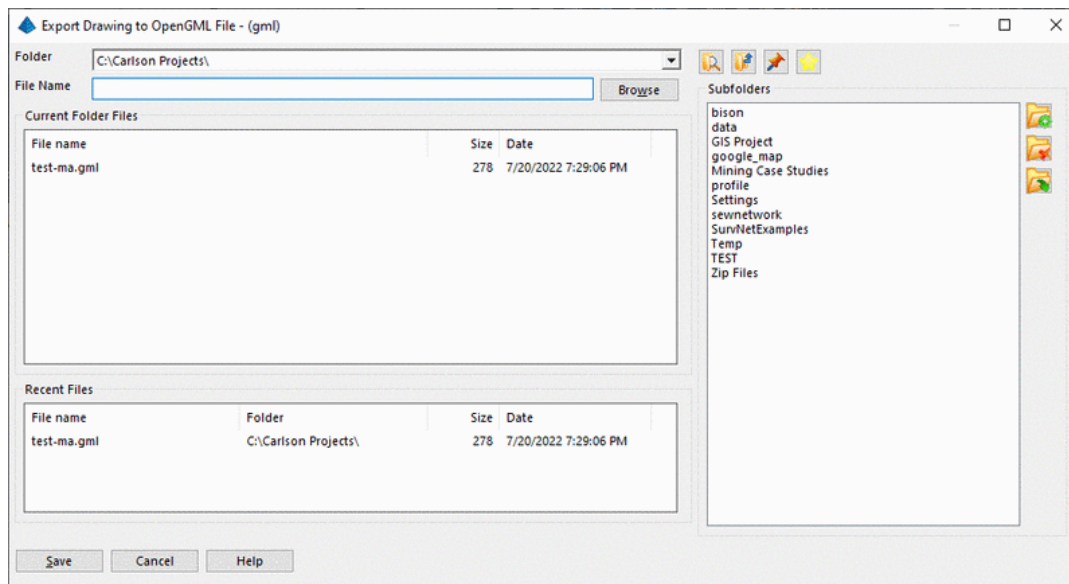
Keyboard Command: msd2survce

Prerequisite: CRD file, FLD file, points in drawing with MSC

Export OpenGIS GML

This command create OpenGIS GML file from the entities in the drawing, and it export all entities that has linked database information. Any entity that has no linked database information would not be exported

The following dialog will display and enter a file name for the GML file



Click on Save button to save, then select the entities to in the drawing to export.

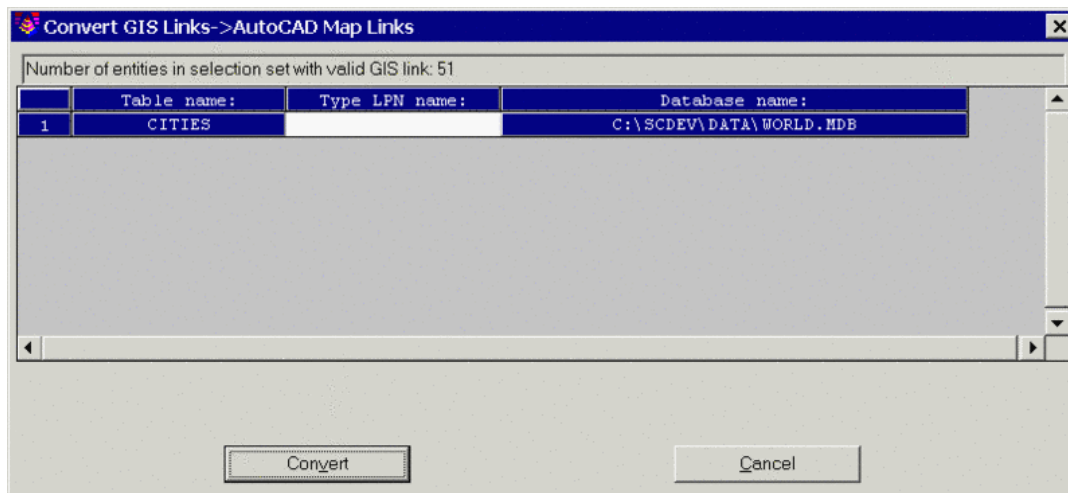
Pulldown Menu Location:GIS Data -> Export GIS Data -> Export OpenGIS GML

Keyboard Command: gml.export

Prerequisite: None

Convert GIS Links to AcadMap

This command converts Carlson GIS links to AutoCAD Map LPN links. The command starts by prompting you to select entities to process. Then the program will read all the Carlson GIS links attached to the selected entities. A dialog is shown with a list of all the pairs of database file and table names from the GIS links. Fill in the LPN name to use for the Carlson GIS link. If the LPN does not exist, it will be created. The LPN will be registered in MapPlus, if MapPlus is available. When the LPN names are filled out, pick the Convert button and the Carlson GIS links will then be removed and replaced by AutoCAD Map LPNs. This routine will run in either plain AutoCAD or AutoCAD Map.



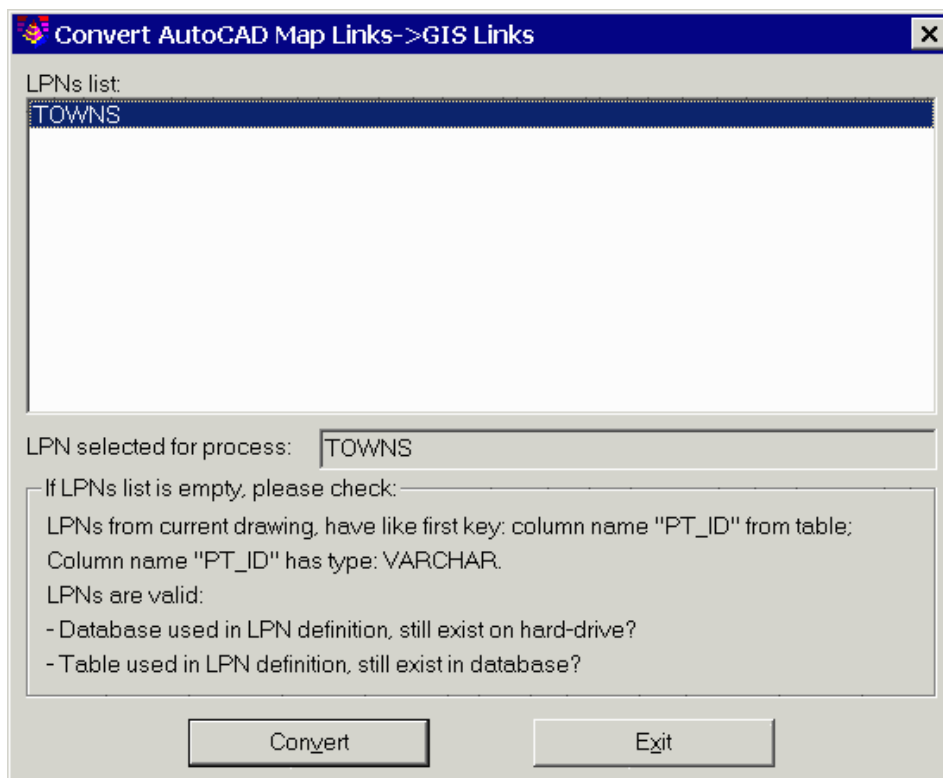
Pulldown Menu Location: GIS Data

Keyboard Command: gis2map

Prerequisite: Entities with linked GIS information

Convert AcadMap to GIS Links

This command converts AutoCAD Map LPN links to Carlson GIS links. The command starts by prompting you to select entities to process. Then the program will read all the Map LPNs attached to the selected entities. A dialog is shown with a list of the LPN names. In order to be able to convert an LPN, the LPN must have a key field called PT_ID. Also, the database file and table referenced by the LPN must exist. In the dialog, highlight the LPN to convert and then pick the Convert button. The Map links are then removed and replaced by Carlson GIS links. This routine will run in either plain AutoCAD or AutoCAD Map.



Pulldown Menu Location: GIS

Keyboard Command: map2gis

Database File Utilities

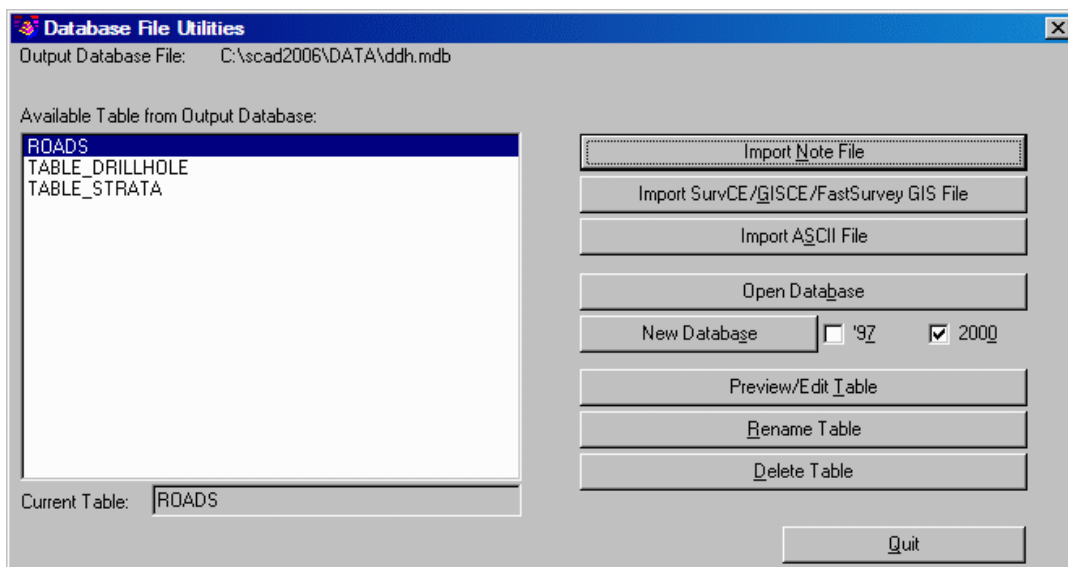
This command is designed to import GIS data from SurvCE, GISCE and FAST Survey files, as well as from user-defined text/ASCII file fields. It also exports data from Carlson Note files (.NOT or .VTT) to Microsoft® Access (.MDB) database tables. The .NOT extension is used when data transfers from desktop. The .VTT extension files are data transfers from data collector.

Note files are associated with Coordinate files (.CRD) and contain additional data for point numbers. For example, the Coordinate file for a manhole point could contain the point number, northing, easting, elevation and 32 character description, while the corresponding note file for that point contains additional data on the manhole such as diameter, depth, condition, etc.. A Carlson Note file for a Coordinate file will have the same name as the Coordinate file, except with a .NOT or .VTT extension instead of the .CRD extension (e.g. PARK.NOT goes with PARK.CRD). The Carlson Note file is a text file which consists of a point number (PT_ID) followed by field names with values. This group of point number and fields can also have a GIS_FILE name, which is used to identify this group of fields. This GIS_FILE name comes from the Note file prompting definition file (.GIS), which defines the field names for the group and is created in the Define Note File Prompts command.

You can select the Note file to process by using the Import Note File button. The program will then list all the GIS_FILE names that were found in the Note file. If a set of data for point number does not have a GIS_FILE name, then this group will appear in the list as UNKNOWN.

The name of the Microsoft® Access database to add the data to is the output database file, listed at the top-left of the Database File Utilities dialog. You can change the output database by using the Open Database button and selecting an existing database, or by clicking New Database to create a new database. The database tables will automatically have the same name as the GIS_FILE. This dialog also allows you to preview and edit a spreadsheet editor, which in turn allows you to modify values in the table. Each set of note file data for a point is displayed on one row with the corresponding record from the database shown on the next row. You can export the Note file data and create a new Access database .MDB file, in Access '97 format or in Access 2000 format, by doing a SAVEAS into .MDB format. You can rename and delete a table as well.

Database File Utilities can be combined with the Create Links command to make GIS links between the point entities in the drawing and the Microsoft® Access database records. The point entities can be drawn with the Draw/Locate Points or Field to Finish commands.



Initial dialog at start of command with primary functions

Available Table from Output Database: Selection list. Pick a table from the Output Database.

Import Note File: Imports a Carlson Note File (.NOT).

Import SurvCE/GISCE/FastSurvey GIS File: Imports a SurvCE/GISCE/FastSurvey Note File.

Import ASCII File: Imports ASCII file.

Open Database: Opens Access database .MDB file.

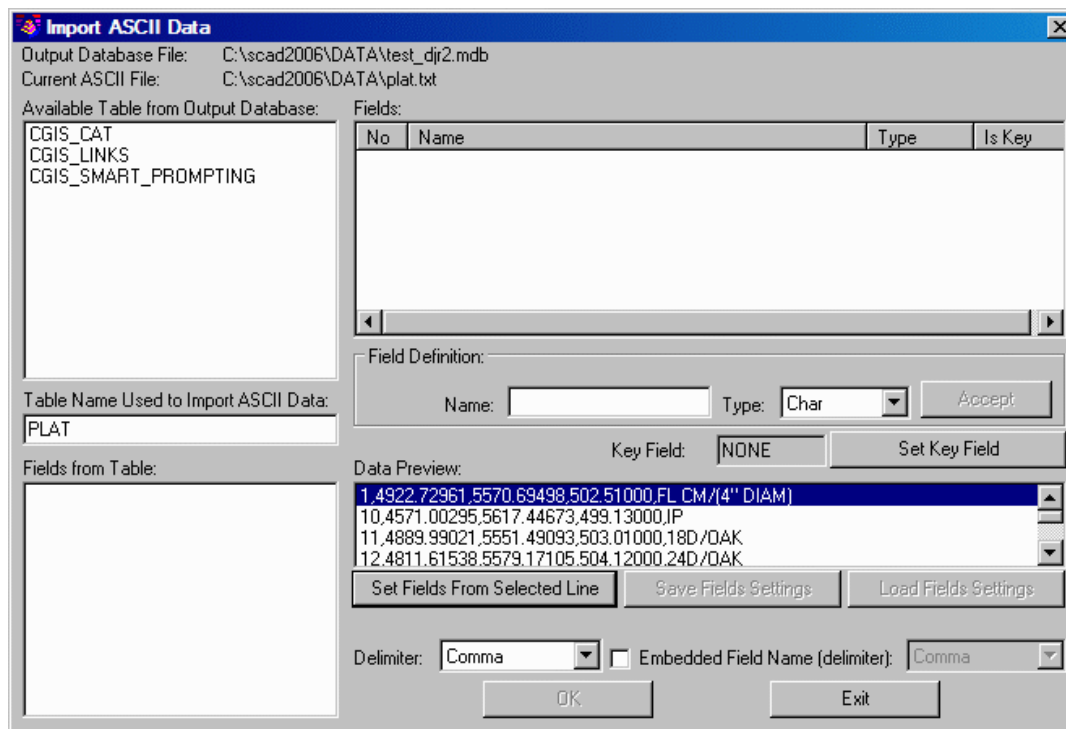
New Database: Creates a new Access database .MDB file in Access '97 format or in Access 2000 format.

Preview/Edit Table: Displays a spreadsheet editor, allowing you to preview/edit values from table.

Rename Table: For renaming a table as needed.

Delete Table: For deleting a table as needed.

Current Table: Displays the selected table from above list.



Dialog seen after choosing Import ASCII File and selecting file name

Pull-down Menu Location: GIS Data

Keyboard Command: noteutil

Prerequisite: A note file (.NOT from desktop or .VTT from data collector)

Define Block Database Links

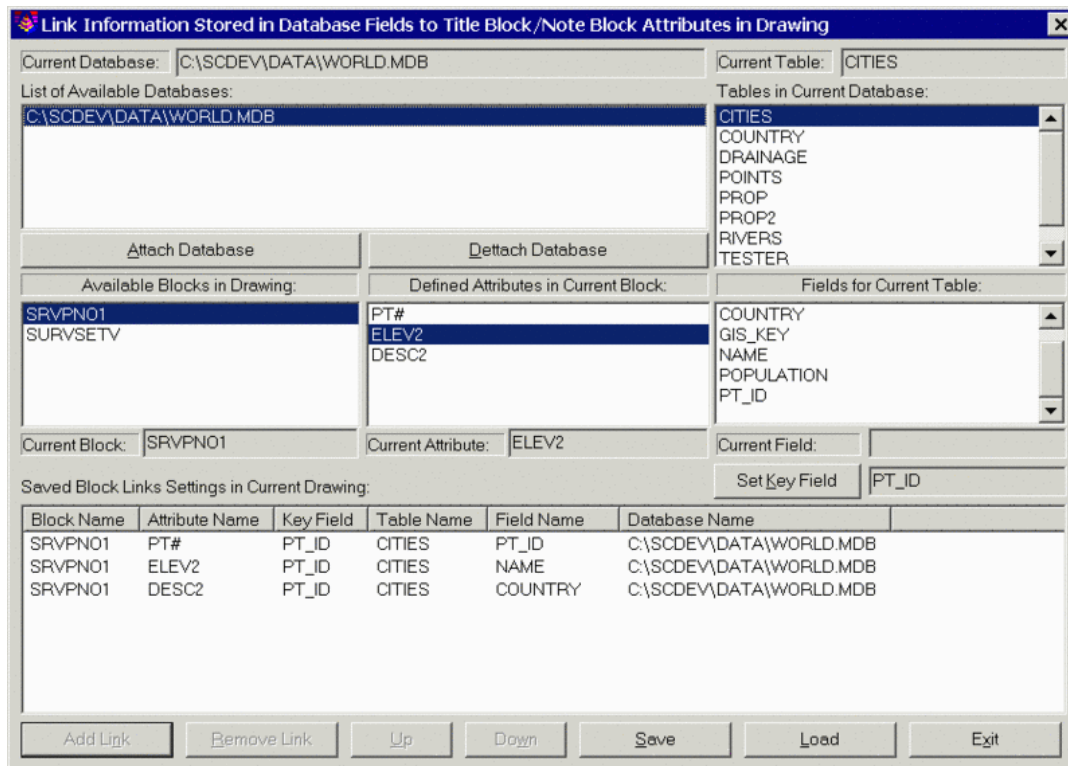
This command creates links between attributes in drawing blocks and fields in database tables for Microsoft® Access, Microsoft® Excel and DBase. The command works through the dialog shown. At the top of the dialog, you specify the database and table to process. Highlight the database and table to process from the lists. To add a database to the available list, use the Attach Database button. This routine brings up a file selection dialog to choose the database file name. Under the "Files of type" field, you can choose between selecting Microsoft® Access, Microsoft® Excel or DBase database files.

After selecting the database table to use, the key field from the table must be specified. Highlight a field name from the Fields list and then pick the Set Key Field button. One of the block attribute links to the database fields must use this field name so that the program can link the block entity with the database record and fill out the other values. For example, consider a block with three attributes: lot_id, owner_name and area. The database table has the following fields: ID, name and area. The database field of ID could be set as the key field and linked to the block attribute called lot_id. Also the database field of name could be linked to attribute owner_name and area linked with area. Programs such as Update Block Data will use the value in the attribute of lot_id to lookup the database record with

the matching value in the ID field. Then the values in the owner_name and area block attributes can be filled out from this database record.

The program lists all the block definitions found in the drawing. Highlight a block name to link to from this list. Then the program will list all the attributes for this block. To create a link, choose an attribute and a field from the database table. Then you can pick the Add Link button to link this block attribute with this database field. The links for a block can be saved and recalled by block name using the Save and Load buttons.

One application of this command is for linking drawing title blocks to a customer database. For example, the drawing title block and the database could contain a customer ID, name, address, etc. The customer ID could be used as the key field. Then you could just fill out the customer ID in the title block and then use the Update Block Data command to fill out the rest of the customer data in the title block.



Pulldown Menu Location: GIS Data

Keyboard Command: define_blk

Prerequisite: Block in drawing and database file

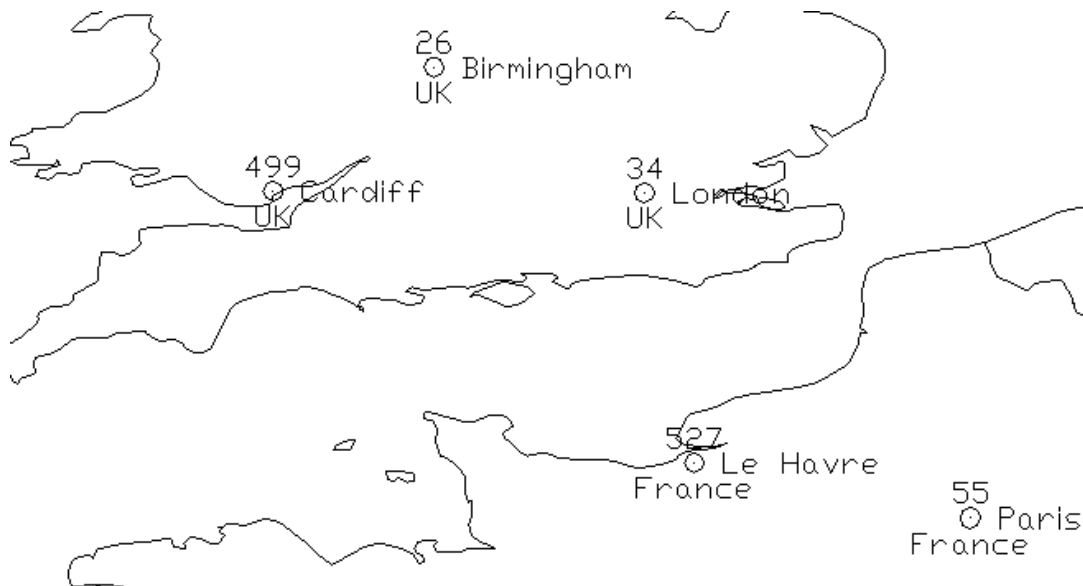
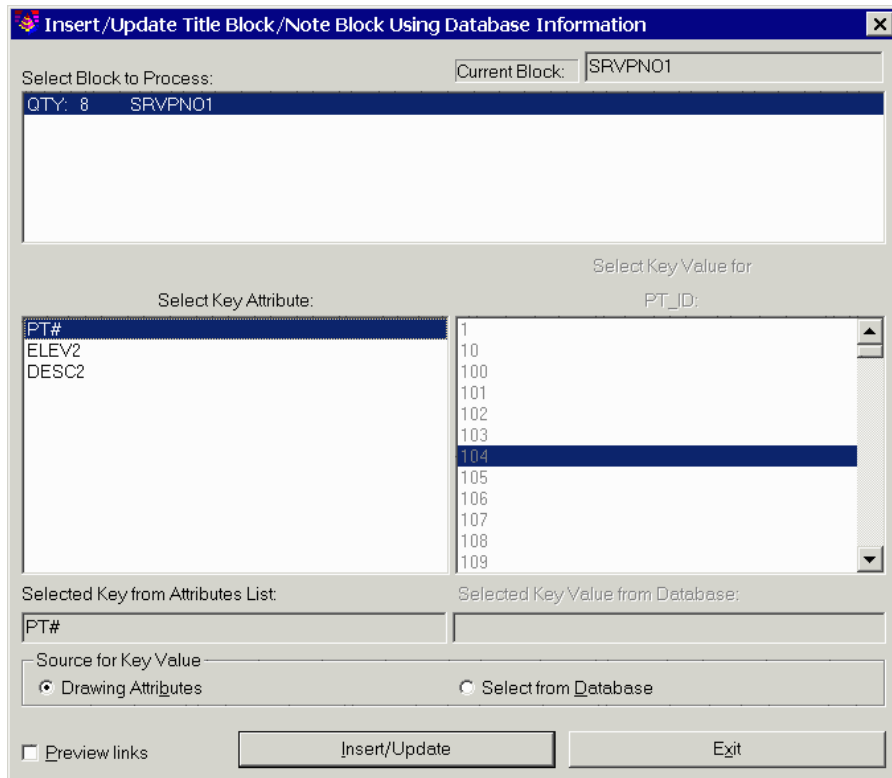
Update Block Data

This command fills out attribute values in drawing blocks using data from database tables. The links between the block attributes and the database fields are defined in the Define Block Database Links command.

The program starts by prompting you to select the block entities to process. Then a dialog is displayed with a list of the block names from the selected entities that also have database links defined. Highlight the block name to process from the list. Then the program lists the attributes for the block. Highlight the attribute that corresponds to the key field from the database table. Then choose the Insert/Update button to update the drawing block attributes.

Under Source for Key Values, the Drawing Attributes option will update each block by reading the value of the key attribute and then looking up the database record with the matching value. The other block attributes and then filled out using the values from this database record. The Select from Database option allows you to choose a database record by highlighting the value from the key field. All the block entities will be filled out using the values from this selected database record.

The Preview Links option will bring up a dialog with a spreadsheet view of the data values from the database for each block. You can edit the values, update the block or skip updating the current block.



Example point blocks updated from CITIES database table

Pulldown Menu Location: GIS Data

Keyboard Command: update_blk

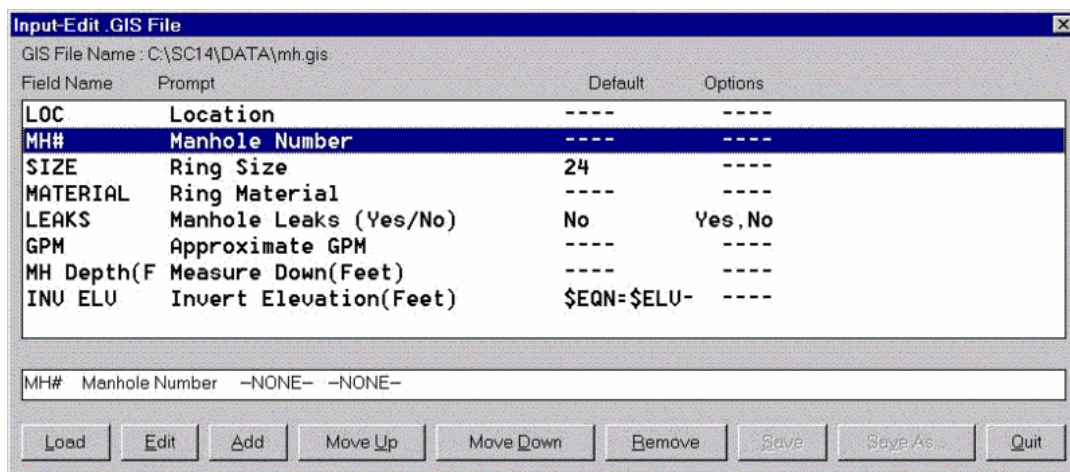
Prerequisite: Blocks, database table and link definition

Define Note File Prompts

This command allows the user to create a .GIS file for use in several other routines in Carlson GIS and other Carlson Software products, such as SurvCE or SurvStar.

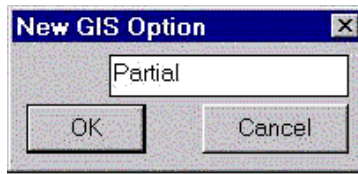
The program starts with the main Define Note File Prompts dialog, as shown below. The Load button allows the user to load an existing GIS file for editing or review. The list box shows the various data capture items in the GIS file, showing the field name, the prompt, the default value and the various options for that field. The Edit button allows the user to edit the highlighted field. The Add button allows the user to add new fields after the highlighted field. The Move Up and Move Down allow the user to change the order in which fields appear in the GIS file, while the Remove button completely removes the highlighted field. The Save button saves the GIS file that is currently being edited, while SaveAs allows the user to save the current GIS file under a different name. The Quit button checks to see if the current GIS file is saved and quits the routine.

When the Edit or Add button is clicked, the dialog box shown here appears, allowing the user to enter and edit data with respect to a particular field in the GIS file. The Field Name is a unique identifier of the field in the GIS file and hence a GIS file cannot have repeated field names. The Prompt is what appears at the command prompt while waits for user input. The Default Value is the value that would be used among various options, if the user presses Enter at the command prompt without typing anything in response to the prompt. The list box, Options for value, contains a list of options that can be selected for the particular field. A new option can be added to the list or removed from the list by clicking the appropriate button. The Add Option button brings up a small dialog and accepts the option to be included in the list. Press OK to accept the values set here. At the minimum, the Field Name and Prompt must be specified.



Define Note File Prompts dialog Load a file, or change variables as required.





Pulldown Menu Location: GIS Data

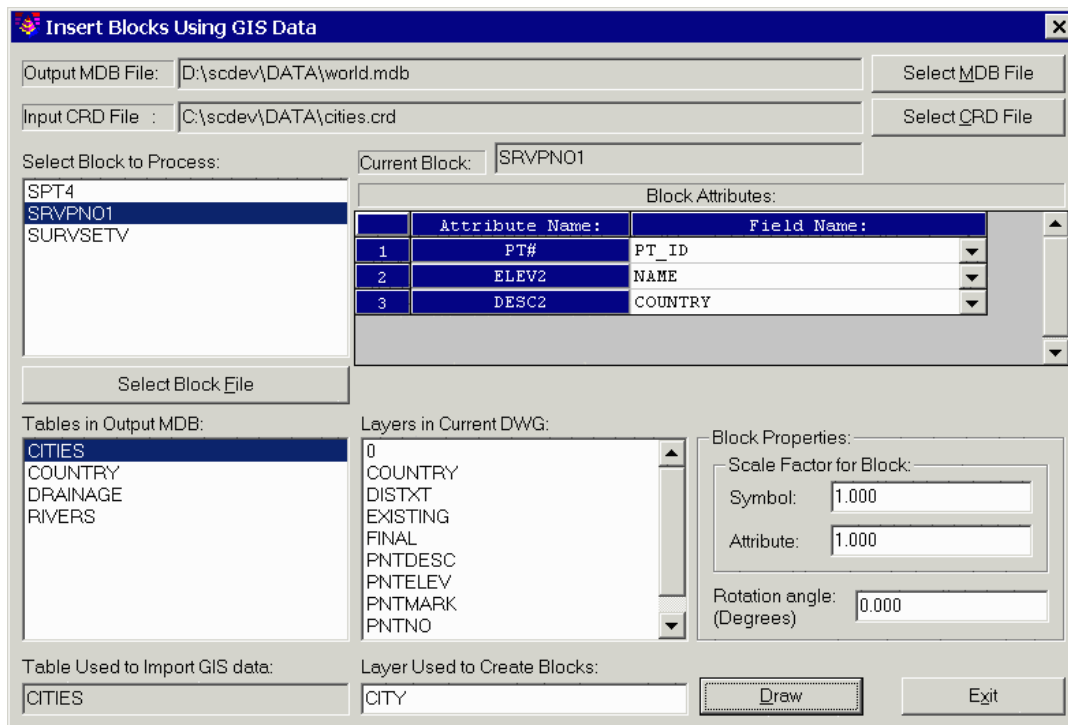
Keyboard Command: defnote

Prerequisite: None

Insert Block with GIS Data

This command draws block entities in the drawing with the attributes filled out using data from a database. The command works through the dialog shown. The MDB File is the database that to be used as the source for the attribute data. The CRD File is the coordinate file that contains point number, northing, easting, elevation, and description. A list of the block definitions found in the drawing is shown on the left. To add a block to this list, pick the Select Block File. Highlight a block name from this list. Then the program will display a list of the block attributes. The tables defined in the current MDB database are listed in the lower left of the dialog. Highlight a table name to use for the data source. Then in the attribute list, for each block attribute you can set the database field to use for the attribute value by selecting the Field Name pulldown arrow. In the lower right of the dialog, you can specify the layer name, scale factors and rotation for the blocks.

Pick the Draw button to create the blocks. For each point number in the coordinate file, the program will insert the specified block in the drawing using the point coordinates from the CRD file. The database record to use is found by matching the point number from the coordinate file with the PT_ID field in the database table.



Pulldown Menu Location: GIS Data

Keyboard Command: insert_blk

Prerequisite: CRD file and MDB database table file

Update Drawing Coordinates

This command update entities' geometry info, and it is used only when the drawing stores GIS Data in SQLite/Spatialite DB, ODBC DB and ESRI GeoDB.

Keyboard Command: update_dwg_coord

How to Setup ODBC Data Sources

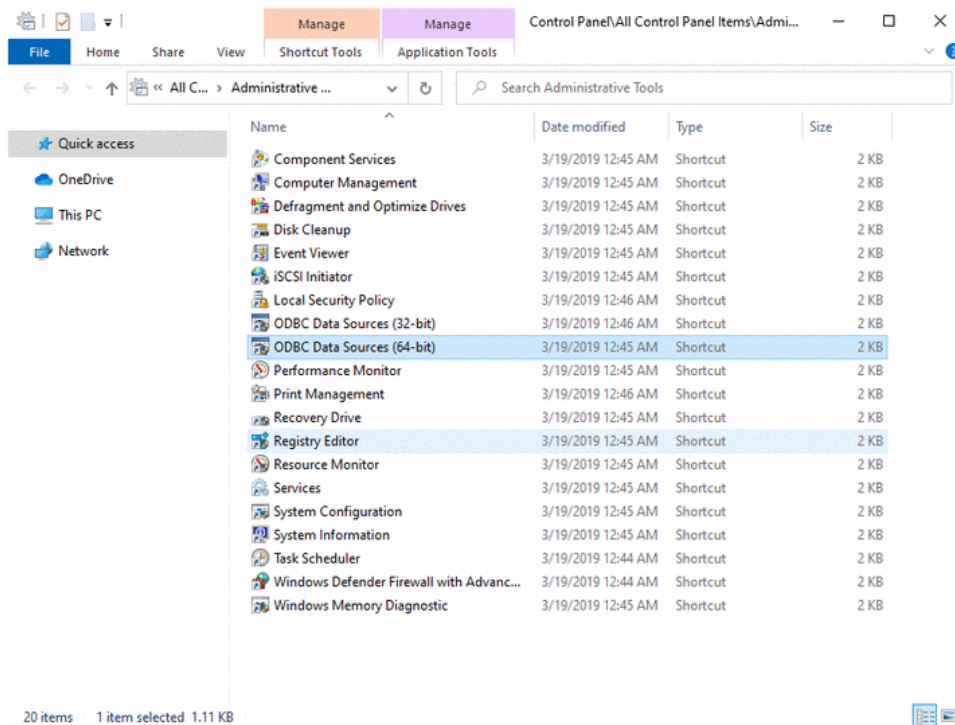
How to setup ODBC Data Source

To add and configure a new ODBC data source in a Windows environment, use the ODBC Data Source Administrator.

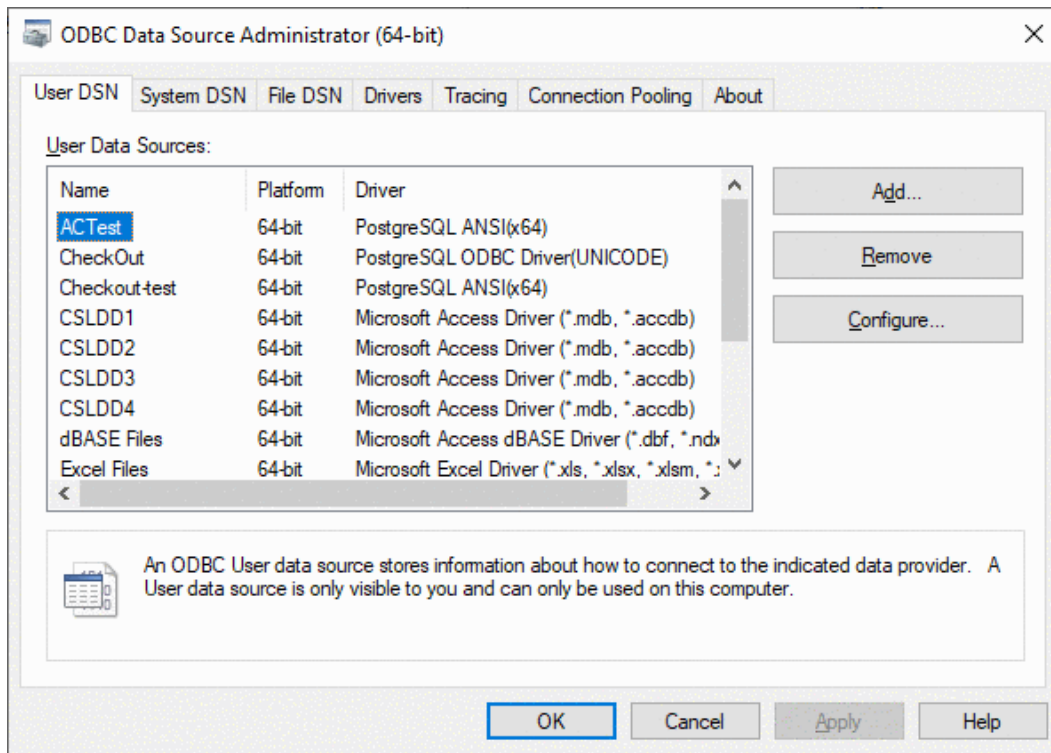
To open the ODBC Administrator from the Control Panel:

Step 1: Right click Start > Control Panel.

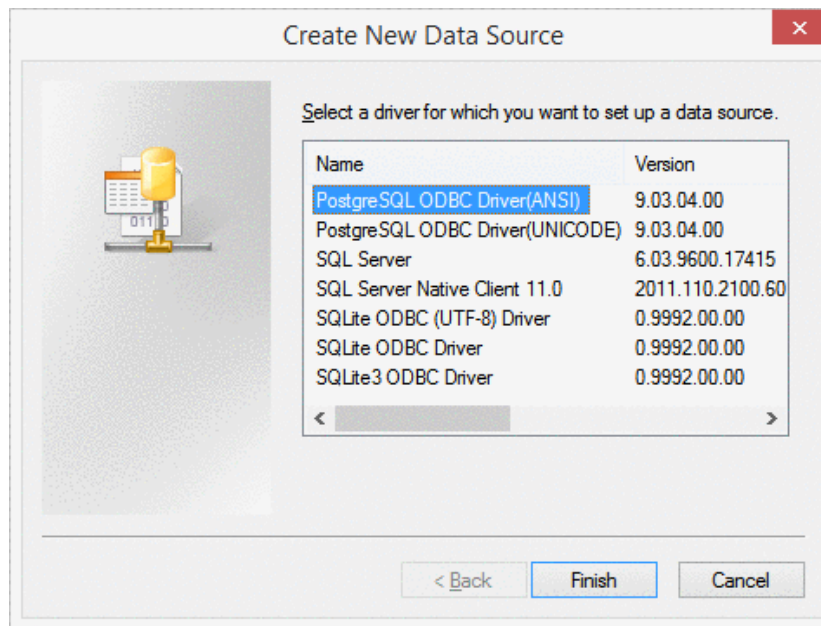
Step 2: Select Administrative Tools, then select ODBC Data Sources (64 bits).



Step 3: In the ODBC Data Source Administrator dialog box, click Add. The Create New Data Source dialog box appears.

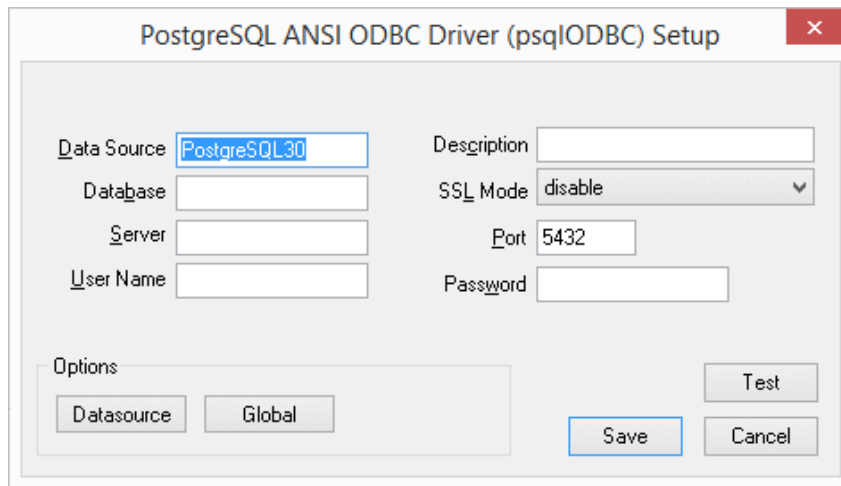


Step 4: Select a driver (e.g., PostgreSQL ODBC Driver) and then click Finish.



The driver configuration dialog box appears.

Step 5: Complete the following field entries:



Data Source Name: enter the name of the data source you want to access. It can be any valid name that you choose.

Description: enter the description needed for the data source.

Host or Server Name (or IP): enter the name of the PostgreSQL server host that you want to access. By default, it is localhost.

Database Name: enter the name of the PostgreSQL database that you want to use as the default database.

User name: enter your PostgreSQL username (your database user ID).

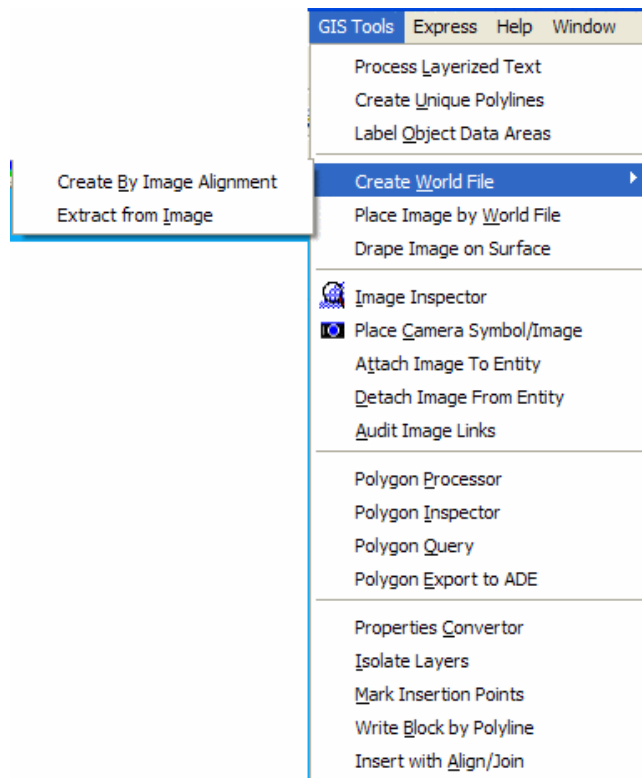
Password: enter your password.

Port: enter the port number if it is not the default value (5432).

Step 6: Click OK to add this data source.

GIS Tools Menu

The GIS Tools menu shown below has commands for processing polyline perimeters, images and other utilities.



Process Layerized Text

This routine is a simplified version of the Polygon Processor command. It uses closed polylines that enclose text. Each text label is used as the data value for the enclosing polyline. The text and polyline must have the same layer name and together they define one layer or topology. This command processes multiple layers by overlaying the layers and finding all the sub-areas. For example, one layer could be used for property boundaries with the property name inside the closed polylines. Another layer could be for soil types. This command could then be used to find all of the different areas broken out by property and soil type (Property 1-Soil 1, Property 1-Soil 2, etc.).

There is an option to draw closed polylines for the resulting sub-areas. You can also label each sub-area with the layer data values for the area (i.e. Property 1/Soil 1). The program also generates a report of the sub-areas, including the data values for each layer, the area and the perimeter. The Report Formatter is used to choose the fields to include in the report and the report layout. There is no explicit link to a database, but the Report Formatter will print and/or save a file of the displayed information, or export the data to Microsoft® Excel and Microsoft® Access.

The function prompts you for a selection set containing the closed polylines and text to be processed. The layer list is formed from the selection set, and you have to specify the sequence in which layers will be processed.

Prompts

Select objects: *choose selection set*

Draw resulting polylines [No/<Yes>]? *press Enter*

Draw name labels [No/<Yes>]? *press Enter*

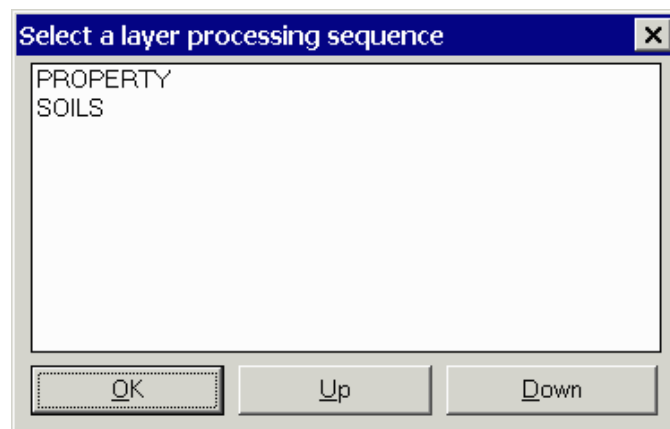
Layer name for resulting polylines : *Property*

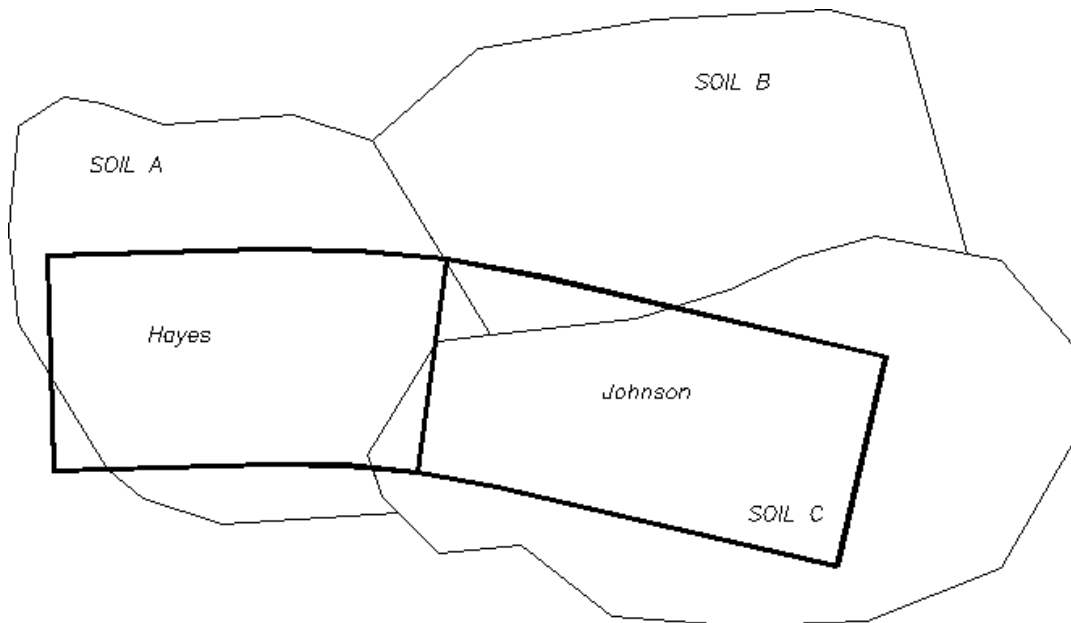
Select a Layer processing sequence dialog *specify the sequence*

Report Formatter Options dialog *customize as needed*

Two-Layer Example:

Two properties (Johnson and Hayes) straddle Soil A, Soil B and Soil C. Note that some of the Hayes property extends beyond the soil polygons. If the properties are "cut" against the Soils, portions of the properties that extend where there are no soil zones will get a blank designation for the soils layer. This command offers a quick method of distinguishing every category of property. In our 2-layer example, the text "Soil A", "Soil B" and "Soil C" are in the same layer as the soil perimeters, and the text "Hayes" and "Johnson" are in the same layer as the property perimeters. Note also that "Soil C" is located inside both the soil and property perimeters. This is OK since it is associated only with the soil polygon because the text and polygon share the same "soil" layer.





```

PROPERTY SOILS Area Perimeter
Hayes 2,713.8 265.12
Hayes SOIL A 80,925.3 1,161.56
Hayes SOIL B 6.9 12.69
Hayes SOIL C 3,960.4 330.70
Johnson SOIL A 2,229.6 229.38
Johnson SOIL B 9,072.0 521.80
Johnson SOIL C 89,665.2 1,276.69
-- Grand Total -----
188,573.1 3,797.95

```

Pulldown Menu Location: GIS Tools

Keyboard Command: layertopo

Prerequisite: None

Create Unique Polylines

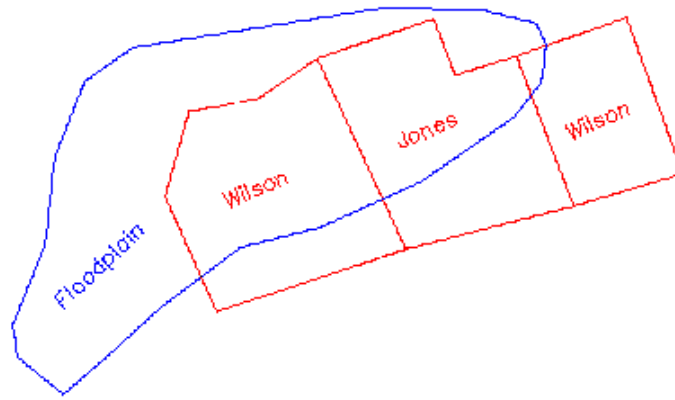
This command takes in a selection set of polylines and lines, and creates a new layer of unique lines, without duplication. They can then be successfully used for topology creation. In the example below, polylines for properties and a floodplain, with several duplicated sides, are used to create a new layer consisting of unique, non-duplicated lines. The text shown is for information only and is not critical to the process.

Prompts

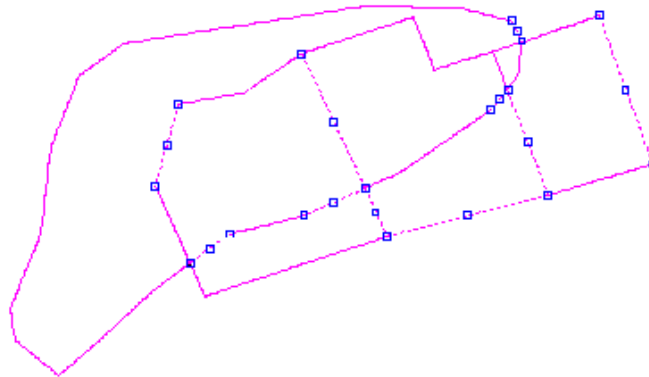
Enter layer name for polylines <CLAYER>: *press Enter*

Select polylines:

Select objects: *select polylines or lines*



Source Drawing



Output drawing isolating to new layer
Certain line segments are highlighted

Pulldown Menu Location: GIS Tools

Keyboard Command: createunipl

Prerequisite: Polylines or lines

Label Object Data Areas

This command labels selected Object Data fields within MPolygon areas with smart rules for different methods to handle any labels that overlap the area perimeter. This command requires AutoCAD Map 2006. The program starts by prompting for the object data table name at the command line. To check the object data table name of an MPolygon entity, double-click on the MPolygon to bring up the Properties dialog. In the dialog, object data is shown in sections labeled OD:Object Table Name.

After selecting the object table name, there is an options dialog. The dialog lists all the field from the object table and you can select which fields to label by adding them to the list on the right. The Annotation Rules are used to position the labels. The rules are applied in order until the labels fit inside the MPolygon. You can choose which rules to use and their precedence:

Inside Area At Centroid: Places the labels at the centroid of the MPolygon

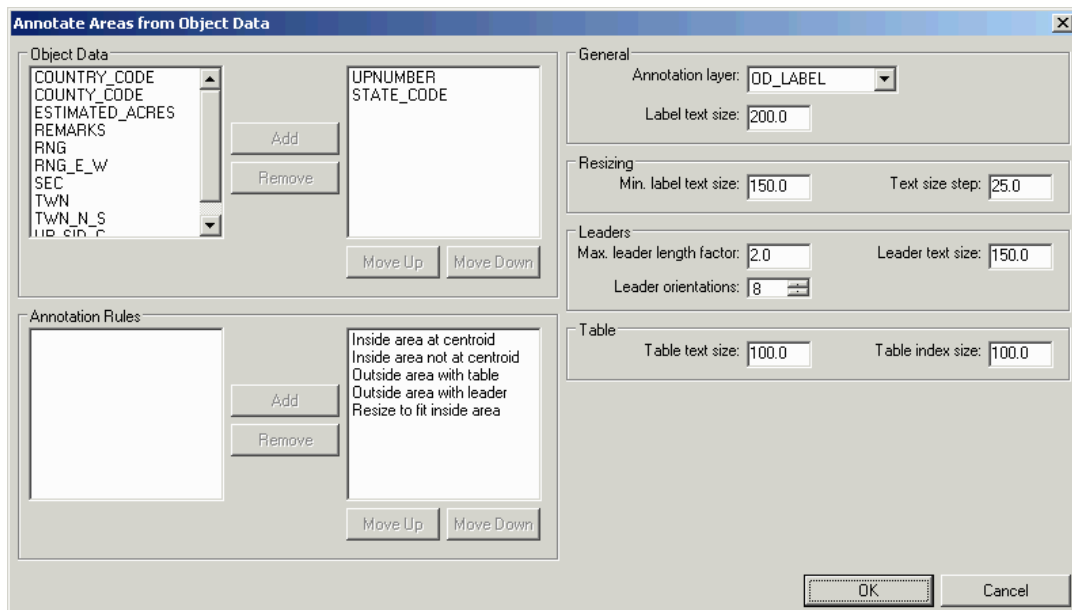
Inside Area Not At Centroid: Searches for any space inside the MPolygon that fits the labels.

Resize To Fit Inside Area: Places the labels at the centroid of the MPolygon at smaller text sizes. The options for this rule control the minimum size to use and the size increments to try starting from the initial size.

Outside Area With Leader: Places the labels outside the MPolygon and draws a leader from the label to the MPolygon center. The options for this rule control the max length of the leader as a factor of the text size, the size of the leader arrowhead, and the number of directions to try as leader positions.

Outside Area With Table: Places the labels in a separate data table and a data id label inside the MPolygon. The

settings for this rule control the size of the table and the id label.



Prompts

Specify map property table: *UP*

Object Data options dialog

Select regions to annotate

Select objects: *select the MPolygons*

Processed 9 of 9 regions.

Pulldown Menu Location: GIS Tools

Keyboard Command: label_area_obj

Prerequisite: MPolygons with object data

Polygon Processor

This routine takes sets of shapes representing land features and generates a set of non-overlapping shapes, where each shape has data attached from all shapes in the original sets it intersects. The input shapes in one layer designate one property like: lease, owner, county and etc.

Properly prepared data should follow these rules:

1. Each property layer should consist of polylines/lines and text.
2. Lines and polylines should form complete loops, with no ends not snapped to the end of the same or other line/polyline.
3. Each formed loop should contain one and only one unique text entity representing the property value for that loop. Multiple entities with the same text are permitted within the loop.
4. Corresponding text and lines should be on the same layer.

Once all the necessary entities are selected, the routine will separate entities by layer and then analyze each layer for compliance to the rules above. If problems are found, the XXXX.ERR layer is created (where XXXX is original layer name) with marks to help locate the problem. The loose ends are marked by placing a circle with center on the endpoint of the line/polyline. The loops with no text value inside are drawn in red color and ones with too much text are drawn in green.

Layers with no problems are processed and the resulting loops are created on the XXXX.RES layer. If all the layers were analyzed successfully with no problems found, these result layers are combined and one final layer called POLYPROC is created with each loop-piece bearing the data from all loops on other layers it belongs to. That data are stored as extended entity data in name-value pairs, where name is a name of the original layer.

Prompts

Select objects: *select shapes and/or polylines*

Pull-down Menu Location: GIS Tools

Keyboard Command: polyproc

Prerequisite: Shapes and/or polylines

Polygon Inspector

This routine allows you to review quickly the results generated by the Polygon Processor. As the cursor moves over a polyline with attached results, the information from that polyline is displayed and the polyline is highlighted.

For more extensive analysis of the data resulting from Polygon Processor, please refer to the Polygon Query command.

Pull-down Menu Location: GIS Tools

Keyboard Command: polyproc_inspect

Prerequisite: None

Polygon Query

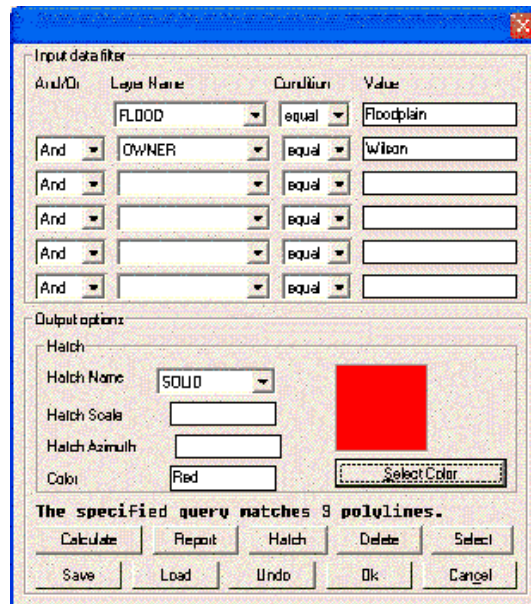
This routine provides the ability to analyze, search and represent data generated by Polygon Processor.

A query is defined based on the parameter name (original layer name) and a condition. Several expressions may be combined using ANDs or ORs to produce a complex query. To verify the query, click on the Calculate button and the number of matching polygons will be reported.

The values stored in the polygon areas are text strings from the original text labels used in Polygon Processor. The expressions are evaluated on these text strings, but the program will compare numbers embedded into strings as numbers:

A12B < A50B
A12B > A5B
A12B < ABC
123 < A123
5-ABC < 13-ABC

A complete report on all polygons matching the specified query may be obtained by clicking on the Report button. All the parameters, along with the areas and polyline handles, are shown in the standard report formatter dialog. The Hatch button applies various hatches to the polygons matching separate queries to create a visual representation of the data. To remove unwanted polygons, i.e. ones which do not contain a combination of required parameters, you may click on the Delete button. This removes processed closed polylines from the POLYPROC layer. To pre-select queried polygons to be used in some other command by means of "previous selection set", use the Select button.



Pulldown Menu Location: GIS Tools
Keyboard Command: polyproc_hatch
Prerequisite: Data generated by Polygon Processor

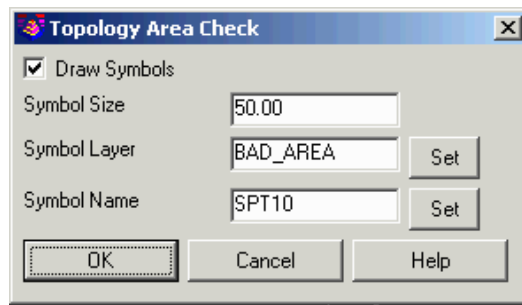
Polygon Export to Map

Polygon data stored in EED form are exported to ADE format (object data), supported by AutoCAD Map. You may then use AutoCAD Map querying functionality to research the data. Since Map's ability to query text is limited, and data coming from Polygon Processor is always stored in text format, for each XXXX parameter in original data two parameters will be present in ADE form: XXXX with text field as is and XXXX_REAL with a number converted from text (if conversion at all is valid).

Pulldown Menu Location: Tools in GIS
Keyboard Command: polyproc_export
Prerequisite: Carlson polygons

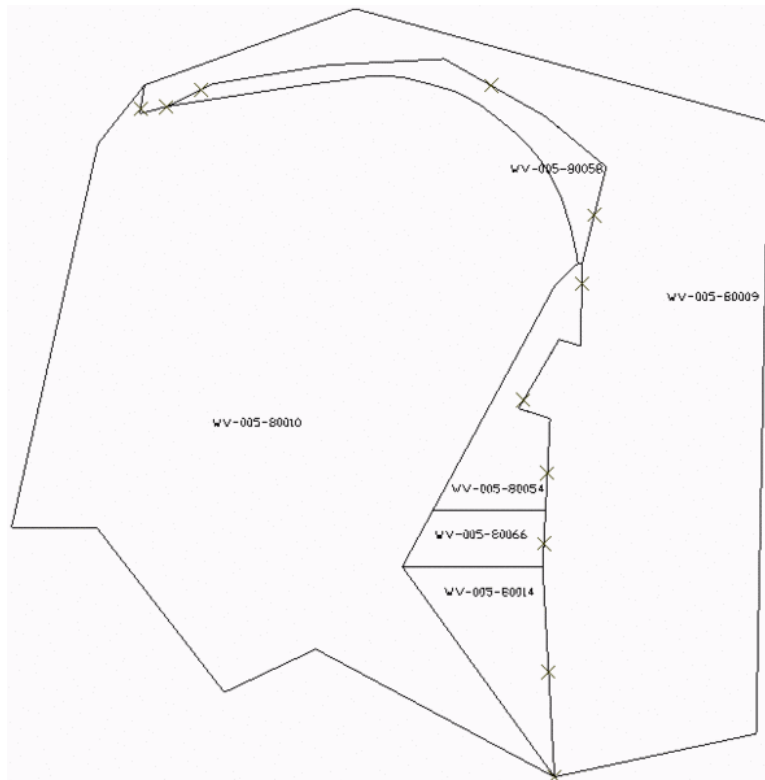
Topology Area Check

This command checks an area coverage made of closed polylines to find any gaps or overlaps. For example, the closed polylines could be for property boundaries. The coordinates at the gaps and overlaps are reported. There is also an option to draw symbols at the gaps and overlaps to highlight these problem areas in the drawing.



Topology Area Check

Gap area at 1800590.727,338038.170
Overlap area at 1800562.557,338071.789
Overlap area at 1800712.729,338371.050



Prompts

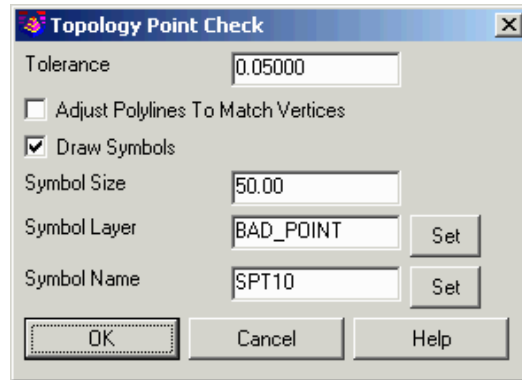
Topology Area Check dialog
Select closed polylines to process.
Select objects: *pick the polylines*

Pulldown Menu Location: GIS Tools
Keyboard Command: area_check
Prerequisite: closed polylines

Topology Point Check

This command checks closed polylines to find vertices that are within the specified tolerance distance of each other and are not identical. The closed polylines could represent an area topology such as for property boundaries. In this case, the program can be used to check that there is a single point for the corner of adjoining properties. The program reports the points that are within the tolerance.

In the dialog, the **Tolerance** is the distance to check. The **Adjust Polylines To Match Vertices** option will move the polyline vertices within the tolerance to the other near point. This option is a way to automatically repair the polylines to share the same coordinates. The **Draw Symbols** option inserts symbols at the points within the tolerance.



Topology Point Check

Tolerance: 0.05000

Point: 1799653.330,338826.420

Point: 1799653.190,338827.230

Checked 97 points

Prompts

Topology Point Check dialog

Select closed polylines to process.

Select objects: *pick the polylines*

Pulldown Menu Location: GIS Tools

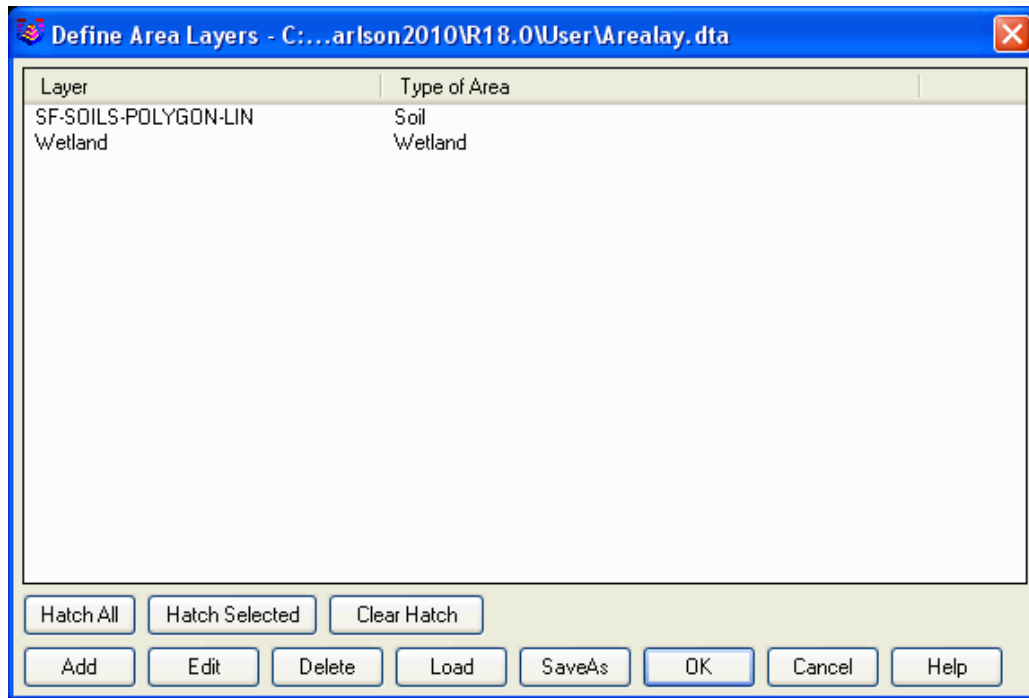
Keyboard Command: pt_check

Prerequisite: closed polylines

Define Area Layers

This command sets up a list of layer names for use with the Report Areas By Layer command. For each layer in the list, an Area Type is assigned which is used for reporting in the Report Areas By Layer command. You can also specify hatch parameters for each layer. These hatch settings are used by the Hatch All and Hatch Selected functions. The Hatch All function hatches all the closed polylines in the drawing on the defined layers. The Hatch

Selected function hatches only the currently highlighted layer. The purpose of these hatch function is to visualize and verify the areas. The Add, Edit and Delete buttons are used to manage the list of layers. The Load and SaveAs functions allow you to save and recall the defined layers to a .ALF file.



Pulldown Menu Location: GIS Tools

Keyboard Command: define_area_layers

Prerequisite: None

Report Areas By Layers

This command reports the area within a closed polyline broken out by sub-areas for areas on defined layers. For example, the main perimeter polyline could be the property boundary and the defined layer polylines could be for wetland areas. Then this routine would report the overall property boundary along with the wetland area.

The layers to process are setup in the Define Area Layers command. The defined layer polylines need to be closed polylines. The program finds all the defined layer areas within the main perimeter polyline and layer polylines that cross the perimeter are automatically trimmed to get the area within the perimeter. The program uses the report formatter to display or export the results.

Pulldown Menu Location: GIS Tools

Keyboard Command: report_area_layers

Prerequisite: Define Area Layers

Properties Converter

This routine facilitates the user in performing massive conversions of entities in the drawing. For example, it can move all blue polylines into a designated layer or change the size of all yellow text.

The user can define a number of different conversion rule sets to work with drawings coming from various sources. Within one set, multiple rules may be selected to be executed at the same time on the set of entities.

In the left upper corner dialog sets of conversion rules may be added by clicking on the Add button. When the set name and file to store a set are specified, the set is added to a list of available ones. To pick a set for editing or to run

it, highlight a particular set in the list and the list of the defined rules will appear in right upper corner of the dialog where users may either add/modify any conversion rules within the set or select rule(s) to be executed on selected entities.

When sets of rules are already defined, the dialog appears as shown below enabling the user to select multiple rules and apply them to the drawing. When a new rule is added, or when the user clicks on the Full Edit button, the dialog expands to show the actual rule definition for the currently highlighted rule. To execute rule(s), select one or more rules (hold Ctrl or Shift keys to highlight multiple items) and click on Execute button. When multiple rules are selected, these rules are applied in a top to bottom order on the entities that the user selects.

The top portion of the rule description defines additional filtering among the selected entities, allowing the user to narrow the user-provided set of entities to a small one containing only entities passing the filter. Filtering can be done by entity types, layer, linetype or color, or by any combinations of these. Filters may be defined easily by clicking on the Select button in either entity type, layer or color boxes and then selecting any entities which bear a desirable feature.

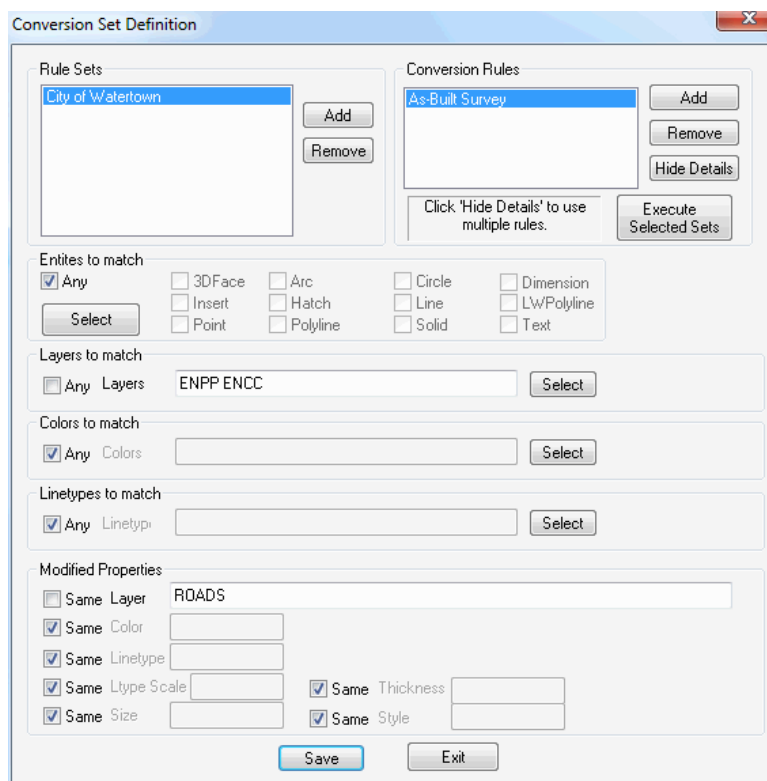
The bottom half of the dialog, Modified Properties, defines what changes will be applied to the entities which are selected and pass through the filter, if any. The following properties may be modified:

Layer: The current layer may be changed to any one on the fixed list of permitted layers. If this option is used, the user has no control over color or linetype, since they come predefined for every layer on the list.

Color: Specify color number 1..255 or BYLAYER

Linetype: Specify a valid linetype name or BYLAYER

More properties that may be modified are **Linetype Scale, Size, Thickness and Text Size.**



Pulldown Menu Location: Tools in GIS

Keyboard Command: convert_set

Prerequisite: None

Isolate Layers

This is a modified Isolate Layers command providing ability to retain entities converted by Property Converter.

This function allows the user to temporarily freeze layers which are not currently being used, to clean up the work area. By default, the function will freeze all layers which do not contain any entities modified by the Property Converter command. You may disable this behavior and specify additional entities, whose layers have to remain visible.

Prompts

Retain layers with converted entities [<Y>/N]? *N*

Select objects on layers to isolate.

Select objects: *select entities to isolate*

Pulldown Menu Location: GIS Tools

Keyboard Command: insertmark

Prerequisite: None

Mark Insertion Points

This routine defines future alignment points within the block to be inserted. It creates a point in the drawing, and that point stores the position and description of the point. When the block containing the point is inserted into the master drawing the description is used to prompt the user for the location of the matching point in the master drawing.

If one such a point is found in the block, it defines the shift. When there are two, they define the shift, rotation and scale of the block being inserted. To define an insertion point, specify the point location and a prompt you want the user to see when the block-to-be is inserted.

Prompts

Select an insertion point for the marker: *pick a point*

Type an insertion prompt <Pick Alignment Point 1>: *press Enter*

Pulldown Menu Location: GIS Tools

Keyboard Command: insertmark

Prerequisite: None

Write Block by Polyline

This command creates a smart block for future insertion into a master drawing. The block contains pre-defined insertion point(s) which define block alignment in the master drawing. This function requires a closed clipping polyline and one or two insertion points. Any linework entities which cross the specified closed polyline are broken at the intersection and only internal portions are inserted.

Prompts

Select a clipping closed polyline:

Select object: *pick a closed polyline*

Select smart insertion markers to include in a block:

Select objects: *select markers*

Pulldown Menu Location: GIS Tools

Keyboard Command: wblockpoly

Prerequisite: Closed clipping polyline

Insert with Join/Align

This command places the previously created smart block into the master drawing. After selecting a block to be inserted, the program finds insertion points and prompts the user to pick the corresponding points in the master drawing.

The next step is cleaning up an area of the master drawing which is now overlaid with the inserted block. The program detects the layers being affected and prompts user with a dialog which lists layers affected, asking the user to specify the layers containing entities of the master drawing to be removed. This list contains only layers which are present in both master drawing and a block default to be removed.

After modification of the layer list is complete, the user is given a chance to review entities to be remove from the master. At first, the entities within the insertion area on the selected layers are highlighted, but the user may unselect or select entities before removal.

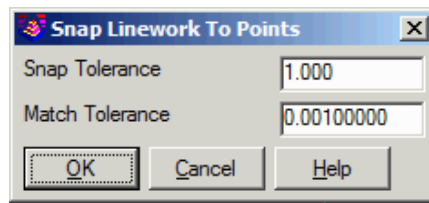
Pulldown Menu Location: GIS Tools

Keyboard Command: smartinsert

Prerequisite: Smart block

Snap Linework To Points

This command changes the vertices of linework (lines and polylines) to match coordinates from the current coordinate file. The purpose is to improve linework for the case of having coordinate data that is more accurate than the linework. The program prompts to select the linework entities to process and then uses all the coordinates from the current coordinate file to snap the linework vertices. Use the Set Coordinate File command to set the current coordinate file.



In the dialog, the Snap Tolerance is the max distance that a linework vertex will move onto a point. The Match Tolerance is the distance between linework vertex and point that the program will consider equal and won't move the linework vertex.

Prompts

Snap Linework To Points Dialog

Select linework to process.

Select objects: *pick lines and polylines*

Pulldown Menu Location: GIS Tools

Keyboard Command: snap_lwork

Prerequisite: Linework in the drawing and points in the current coordinate file

Set Text Layer By Text String

This command sets the layer for each selected text entity to the string of the text entity.

Prompts

Select text to set layer.

Select objects: *pick text*

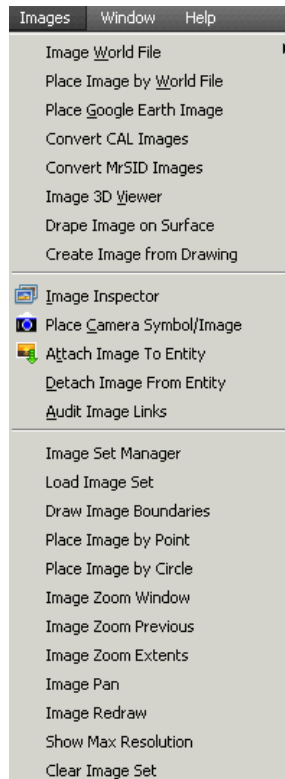
Pulldown Menu Location: GIS Tools

Keyboard Command: layertxt

Prerequisite: text entities

GIS Image Menu

The Images menu has commands for placing aerial images, attaching images to entities and managing an image database.



Create World File from Image Alignment

This command creates an image world file (TFW) from a TIF or JPG image. First the image file to be used is selected, and then two points in the drawing are specified along with two corresponding points on the image. The image is scaled and rotated to match the input data, and a World File is written with the specified data.

Prompts

Image File To Process: *choose .TIF or .JPG file*

Select First Reference Point in the Drawing: *pick a point in the drawing*

Select location of First Reference Point on the Image: *pick a point on the image*

Select Second Reference Point in the Drawing: *pick a second point in the drawing*

Select location of Second Reference Point on the Image: *pick a second point on the image*

Pulldown Menu Location: Images > Image World File

Keyboard Command: make_world_file

Prerequisite: TIF image file

Create World File by Image in Drawing

Once an image has been inserted, scaled and rotated into a drawing, it may be desirable to create a "world file" (see the frequently changing Wikipedia topic at http://en.wikipedia.org/wiki/World_file for additional information) from the image so that the image can be re-inserted into a drawing at the proper placement, scale and rotation angle by using the Place Image by World File command.

A common method for inserting raster images into a drawing that don't have an accompanying "world file" would be through the use of the Place Google Earth Image command.

Prompts

Select images to process.

Select entities: *Select the raster entities whose world file should be created*

Pulldown Menu Location: Images

Keyboard Command: make_geotiff

Prerequisite: Raster image in the DWG

Create Image World File

This command creates an image world file including TFW from a TIF image and JPW from JPEG 2000 image. The image file needs to have georeference data embedded inside the file that contains the image position and pixel size. The world file such as TFW is an externalized georeference file for this data that the Place Image By World File and Image commands use.

Prompts

Image File To Process: *choose image file*

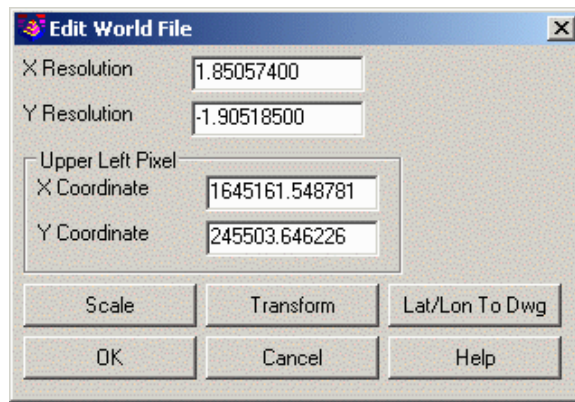
Pulldown Menu Location: Images > Image World File

Keyboard Command: maketfw

Prerequisite: TIF, JP2, J2K, JPF, JPX or JPM image file

Edit World File

This command edits the information in an image world file. After selecting the world file to edit, the parameters are shown in a dialog where you can edit the resolution or coordinates. The resolution defines the size of a pixel in drawing units. The Scale function multiplies the resolution and coordinates by a factor and can be used to convert a world file between English and metric units. The Transform function converts the world file between coordinate systems such as from UTM to State Plane. The Lat/Lon To Dwg function converts a world file in lat/lon units into units for the current drawing using the projection definition from Drawing Setup.



Pulldown Menu Location: Images

Keyboard Command: edit_world_file

Prerequisite: Image world file

Place Image by World File

This routine is intended for users of Carlson products that do not have the AutoCAD Map platform. If you have the Map extension available, it is recommended that you use the tool provided.

This function allows you to insert Geo-Referenced TIF files into the drawings. This process requires the presence of an accompanying TFW file. The TFW file contains information about the location and scaling of the actual raster image TIF file. This eliminates the guesswork in inserting, moving, and rotating raster images to the project area. You begin by selecting the TFW or JGW file to process. If the related TIF file is present in the same directory, the image will be inserted into the proper coordinates.

Prompts

Select World File: *choose existing .TFW or .JGW file*

Pulldown Menu Location: GIS Tools

Keyboard Command: geotiff

Prerequisite: None

Place Image by World File

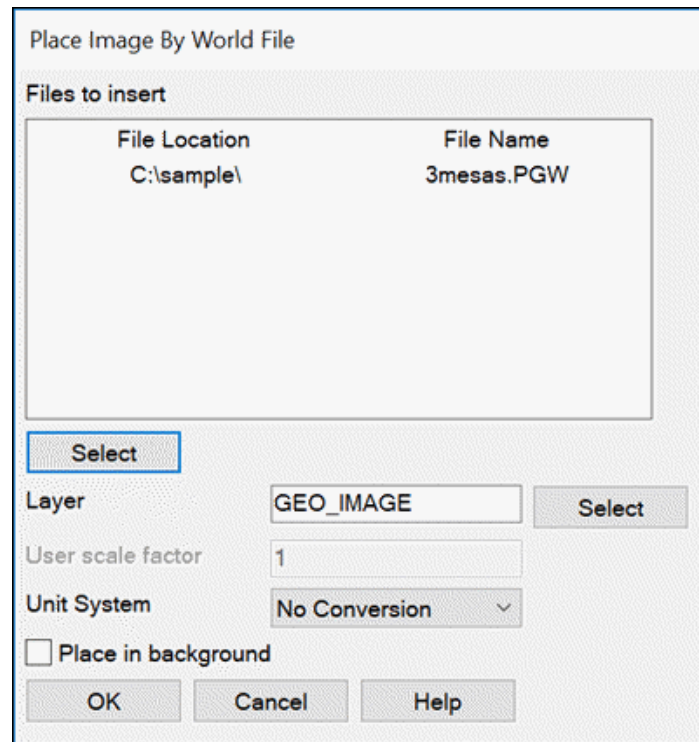
This function inserts a geo-referenced image files into the drawing. The geo-reference data supplies the position for the image so that the image can be placed without prompting for the position, rotation or scale. This eliminates the guesswork in inserting, moving, and rotating raster images to the project area. This process requires the presence of an accompanying World file. The supported World File formats include: .TFW, .JGW, .J2W, .PGW, .SDW, .JP2, .J2K, .JPF, .JPX, .JPM and .ECW.

The World Files are a separate file from the image file. For example, a .JGW is the World File for a .JPG image file. The world file and the image file need to be in the same folder. The program also supports geo-tagged images where the world file data is stored within the image file instead of having a separate file.

In addition to working with World File, this command also supports geo-referenced .TIF file that have the world data embedded in the image file.

You begin by selecting the World file to process by picking the Select button. You can import multiple images. The Layer sets the layer name for the image entity in the drawing. The Unit System allows for scaling the image such as

having an image in metric units and a drawing in feet units. Besides meters-to-feet conversions, you can also apply a custom scale factor. The Place In Background sets the display order of the image to the back.



Pulldown Menu Location: GIS Tools

Keyboard Command: geotiff

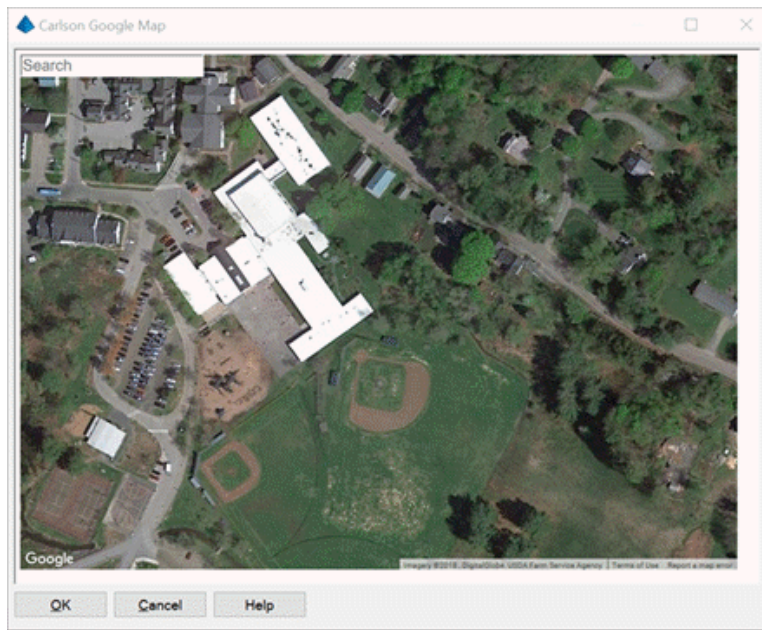
Prerequisite: A georeferenced image file

Place Google Earth Image

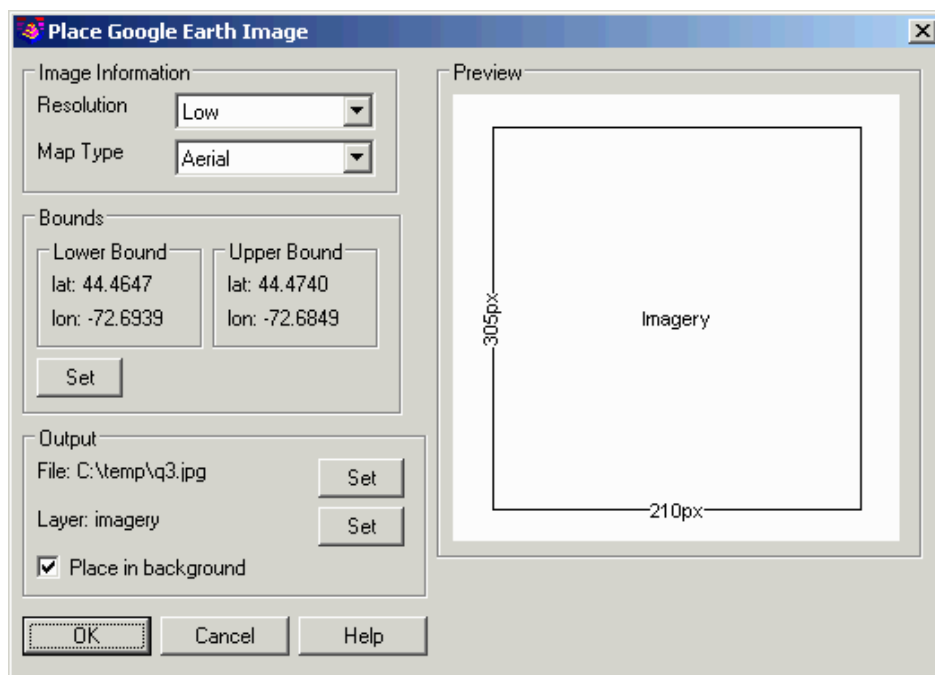
This command inserts a geo-referenced image into the drawing using the Google Earth imagery server. This command requires an internet connection. In order to automatically place the geo-referenced image in the drawing, the drawing must also be geo-referenced by defining the grid projection in the Settings > Drawing Setup command. If the drawing isn't geo-referenced, you still can place the image by manually picking the location.

The command starts by prompting for the image file to create. The Google Earth image will be saved to this file name and then inserted into the drawing.

Next the program prompts to pick the area in the drawing for the image. You can either screen pick two points to window the area for the image, or use the Map option to graphically set the image area. In the Map dialog, use the mouse wheel to zoom in/out and click-n-drag the mouse to pan. You can also enter a place name or address in the Search field to set the view.



Then there is an options dialog.



Resolution: controls the size of the image file. The Preview window shows the number of pixels for the currently selected Resolution.

Map Type: selects the type of image to create.

Bounds: shows the currently selected area in decimal lat/lon. Pick the Set button to pick a new area in the drawing.

File: the image file to create.

Layer: sets the layer for the image in the drawing.

Place in background: sets the display order for the image to back.

Note:

- Once an image has been inserted into the CAD drawing, it may be helpful to create an associated "world" file for the image in case it needs to be re-inserted into the active drawing (or an alternate drawing). Creating a "world" file assists with this task. To generate a world file, please reference the Create by Image in Drawing command.
- Once the image has been inserted into the CAD drawing, it may be helpful to control its "display order" by using the View > Display Order > Order by Layer command or View > Display Order > Send to Back command.
- The Place Google Earth Image routine fetches aerial imagery in real-time from the Google servers and requires an Internet connection to proceed.

Prompts

Select Output File *Choose image file name to create*

Pick from Screen or use Map [**<Screen>/Map**]? *press Enter*

Pick first corner of bounding box: *Identify one corner of a drawing window that should be used to set the Google Earth display*

Pick second corner of bounding box: *Identify the opposite corner of a drawing window that should be used to set the Google Earth display*

Place Google Earth Image dialog

- To import KML or KMZ content into your drawing, use the Import Google Earth File command.
- To export content from your drawing to a KML or KMZ file, use the Export Google Earth File command.

Pulldown Menu Location: Images

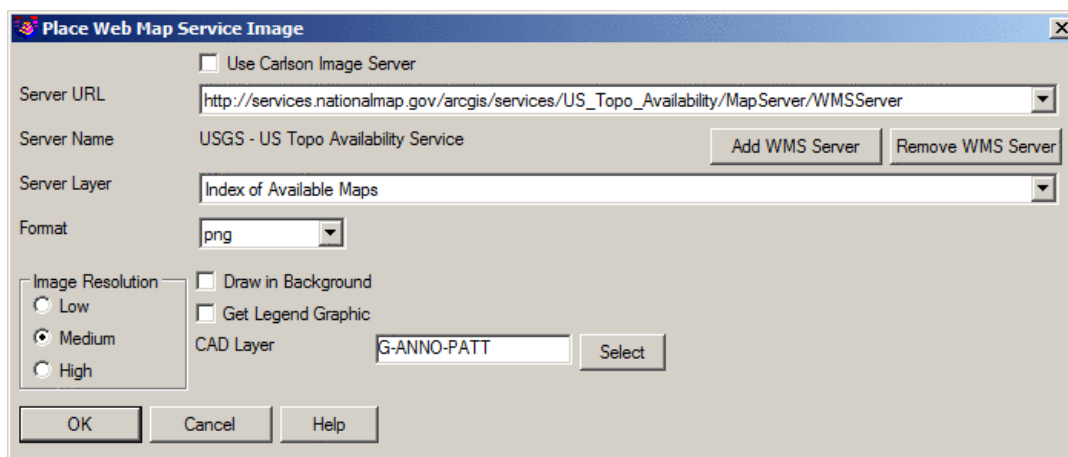
Keyboard Command: google_image

Prerequisite: Coordinate projection system, Internet connection

Place Web Map Service (WMS) Image

The Place Web Map Service (WMS) Image command provides an ability to fetch map data from on-line sources as a means to quickly and easily construct raster map data in and around an area of interest. In addition to producing a raster image, an accompanying "World" file for the image is also generated that permits the image to be properly repositioned for geo-referencing purposes.

For WMS servers that require an API Key such as NearMap, put the API Key into the Server URL at the end of the web address. The API Key needs to be between "?apikey=" and "&". For example, https://wmsus.nearmap.com/wms?apikey=your_api_key&



Use Carlson Image Server: When enabled, images are fetched from a WMS server hosted by Carlson Software.

The Carlson Image Server includes links for:

- USGS EROS Ortho Imagery (an alternative source to Google Earth for aerial images)
- USGS Vegetation
- USGS Topo

- USDA Soil Map
- US Fish and Wildlife Wetlands

- ESRI World Street and Topo
- OpenStreetMap

Server URL: Select the desired Uniform Resource Locator (URL) from where the image data should be obtained.

Server Name: A read-only label identifying the name of the WMS server.

Add WMS Server: When clicked, the URL of a WMS server may be input that is subsequently added to the Server URL drop-list.

Remove WMS Server: Deletes the currently selected WMS Server from the Server URL drop-list. Prior to the removal of the WMS Server, a confirmation alert box will be displayed asking for confirmation. Any images previously placed into a drawing from the WMS Server will remain in the drawing.

Server Layer: Choose the type of image data to retrieve from the selected Server URL.

Format: Indicate the type of image format.

Image Resolution: Controls how many "picture elements" (pixels) will be used to render the image:

- **Low:** The image is divided into a small number of pixels such that each pixel in the overall image covers a relatively large geographical region.
- **Medium:** The image is divided into a larger number of pixels (4-times as many as that produced through the *Low* resolution such that each pixel in the overall image covers a relatively smaller geographical region.
- **High:** The image is divided into the largest number of pixels (4-times as many as that produced through the *Medium* resolution such that each pixel in the overall image covers (relatively) the smallest geographical region.

Draw in Background: When enabled, the resulting image is drawn "underneath" existing entities in the drawing (essentially the same as View – Draw Order – Send to Back). An optional method to help control the visibility of placed images would be through the Order by Layer command.

Get Legend Graphic: When enabled (and for WMS "Layers" that support the generation of an image legend), a secondary image file that helps describe the content of the primary image.

CAD Layer: Key-in the desired Layer name in CAD that is to contain the image being created or use the **Select** button to select the CAD layer from the Select Layer dialog box.

Note:

- Several freely available WMS addresses can be found at the USGS National Map Viewer.
- Your local governing agency could/should be consulted to see if they publish any data via their own WMS.

Prompts

Pick first corner of bounding box: *Indicate one corner of a rectangular box that identifies your area of interest.*

Pick second corner of bounding box: *Indicate the other corner of a rectangular box that identifies your area of interest.*

- To import Esri ArcGIS On-line Services data into your drawing, use the Esri ArcGIS Services - Retrieve Map command.

- To import Web Feature Service information into your drawing, use the Web Feature Service (WFS) command.

Pulldown Menu Location: Images

Keyboard Command: getwmsimage

Prerequisite: Internet connection, Coordinate System Projection set through Drawing Setup

Convert CAL Images

This command converts .CAL format image files into .TIF format.

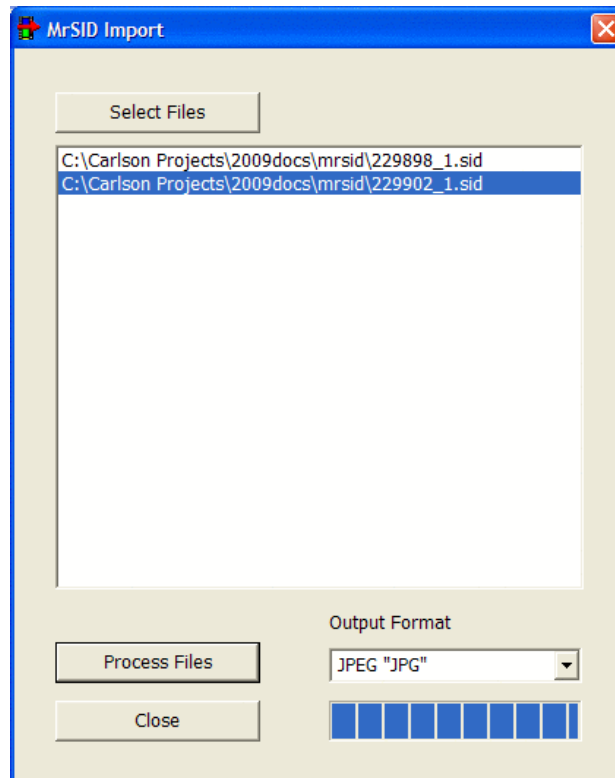
Pulldown Menu Location: Images

Keyboard Command: import_cal

Prerequisite: .CAL image

Import MrSID Images

This command allows you to select one or more image files in the MrSID format (.SID) and have them converted to either a (.TIF) or (.JPG) file format. Corresponding World files are also created. The new image files use the same file name and folder of the original MrSID image files except with the new file extension of .jpg or .tif. This conversion supports up to MG4 (Generation 4) MrSID files.



Pulldown Menu Location: Images

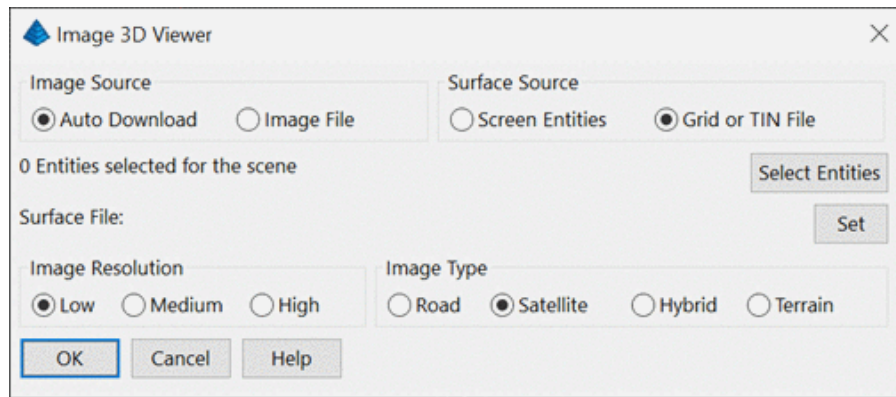
Keyboard Command: import_mrsid

Prerequisite: .SID file

Image 3D Viewer

This command provides a 3D Viewer that can be used to show a georeferenced image draped over a corresponding surface. The image can be either a georeferenced image file or automatically downloaded from Google Earth. For

the download, the dialog has choices for the image resolution and image type. The surface model can be either a surface file or 3D Face entities in a drawing.



Pulldown Menu Location: Images

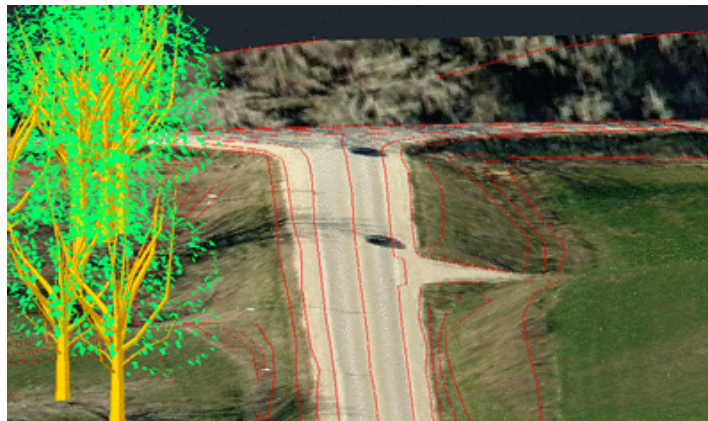
Keyboard Command: view_geotiff

Prerequisite: 3D Faces or TIN or GRD file for surface and corresponding georeferenced image file

Drape Image on Surface

This command drapes (or "overlays") an image onto a surface in the drawing. You are first asked to select a TIF, JPG or SID geo-referenced image file, and then to select the 3D Faces of the surface to drape the image on.

After the image is draped on the 3D faces, you can view the image in the drawing by setting the CAD render mode. For example, use the SHADEMODE command and set it to Realistic.



Note:

- The 3D Faces of a surface can be placed by commands such as the Draw 3D Grid File, Triangulate Tab of Triangulate & Contour or Draw Triangular Mesh.
- Functionality similar to the *Drape Image on Surface* command can be found in the Image 3D Viewer command.

Prompts

Image File To Process: Choose a geo-referenced .TIF or .SID file

Select surface faces.

Select entities: *pick the 3D faces*

Pulldown Menu Location(s): Images

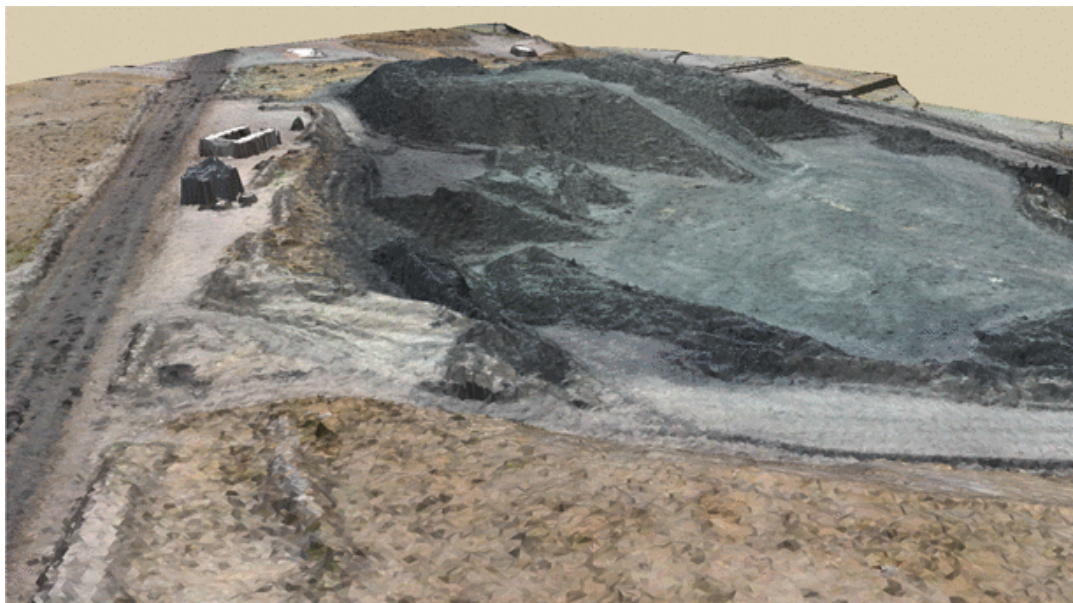
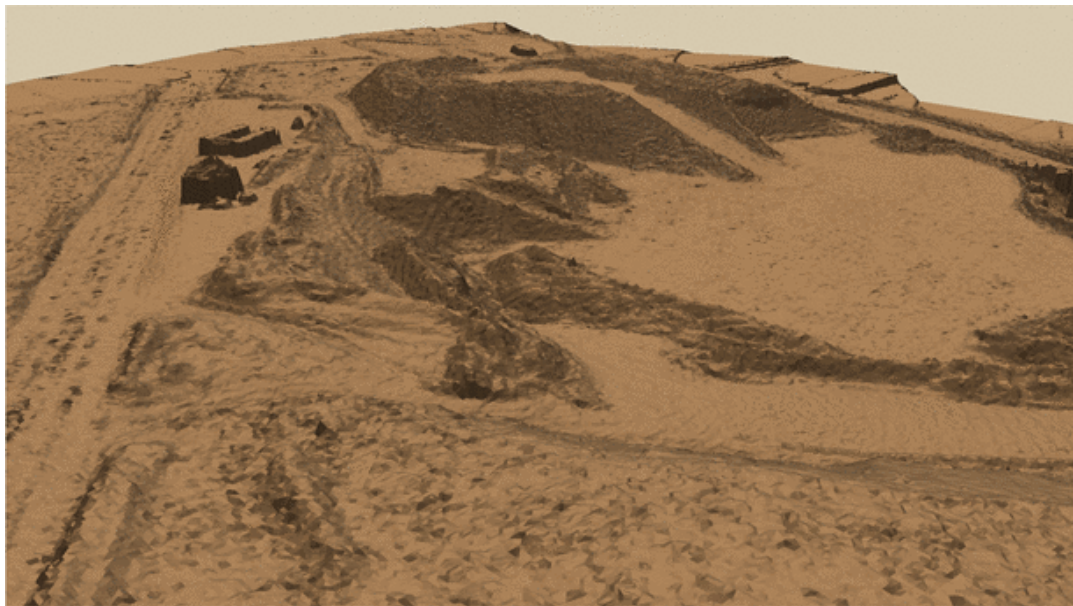
Keyboard Command: `drape_geotiff`

Prerequisite: geo-referenced TIF or SID image file and 3D Faces of a corresponding surface.

Color TIN By Image

This command assigns colors to the TIN faces using colors from a georeferenced image. This command applies to having an aerial image and a ground surface model that you can to colorize. The program prompts for the image file, input TIN file and output TIN file. The georeferenced image can be a TIF, JPF, JP2 or ECW format. The colorized TIN can be viewed in the 3D Viewer image.

Here's an example of the TIN before and after Color TIN By Image.



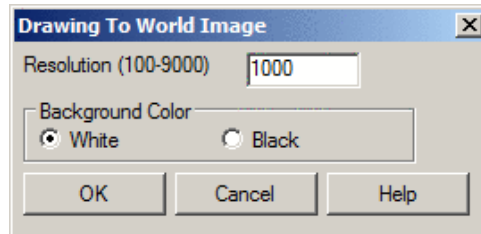
Pulldown Menu Location: Images

Keyboard Command: image_tin

Prerequisite: .TIN file and geo-referenced image

Create Image from Drawing

The *Create Image from Drawing* command allows you to create a raster image of selected entities in your drawing to produce an photographic style image with a corresponding "world" meta-data file. This combination of files (*.TIF, *.TFW) allows the image to be correctly located with routines such as the Place Image by World File and Surface 3D Fly-over commands.



Resolution: Indicate the width of the image about to be created in pixels. Higher values permit the image to carry crisper details but increases the resulting file size at a geometric rate.

Background Color: Indicate the desired color of the background portion of the image.

Note:

- For normal (best) results, a "plan" view angle is suggested and can be accomplished through the *View > Viewpoint 3D* command.
- The "X-direction" limits ("X" is normal to the current angle) of the entities selected for the image will be contained by the image width (or *resolution*) specified above to produce a "Pixel:CAD Units" ratio. This ratio is multiplied by the vertical, "Y-direction" limits ("Y" is normal to the current view angle) of the entities selected to produce the image height.
- The TIF image produced by the command will carry an X and Y "resolution" of 72 dpi (dots per inch). For an image 1000 pixels wide, the "measureable" width of the image would be 13.889 inches and is the quotient of image width divided by the resolution (*e.g.* 1000/72).

Prompts

Image File to Create: *Specify the name of the image that is to be created.*

Select entities for image.

FILter/<Select entities>: *Select the entities that are to display in the image and press Enter when complete.*

Pulldown Menu Location: Images

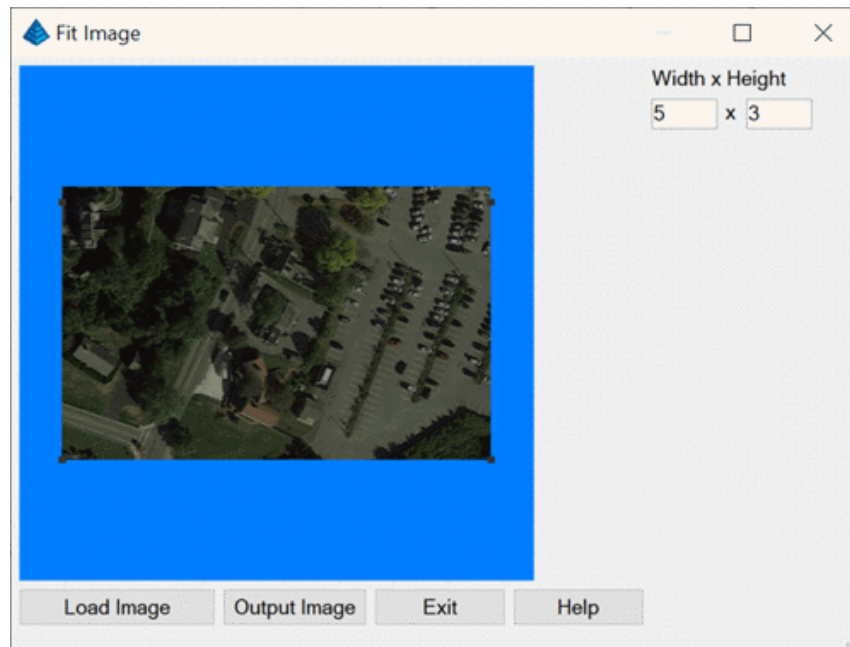
Keyboard Command: dwg2geotiff

Prerequisite: Entities in the drawing

Fit Image To Aspect Ratio

This command crops an image to fit the specified aspect ratio. After selecting an image file, the dialog shows the image with grip squares at the corners for the crop. The aspect ratio is set in the edit boxes as the ratio between width and height. Click and drag a grip square to resize the area. Pick and drag in the middle of the image to change the crop area.

The Load Image button is to select another image to process. Pick the Output Image button to save the cropped image to an image file.



Pulldown Menu Location: Images

Keyboard Command: fit_image

Prerequisite: Image file

Image Inspector

This command views images and documents attached to entities. At the start, there's a prompt for inspecting on the screen or in a dialog. For Screen, the program highlights all entities that have attached images or documents. When you move the cursor over these entities, the attached image or document name is displayed in a window. If you click within the image window, the program will start the image application editor that is setup for your system. For documents, click on the document name to start the document application. This application, such as Microsoft Internet Explorer, depends on your Windows system setup. Also while moving the cursor over drawing entities, you can use the up/down arrows to resize the image. When multiple images or documents are attached to the same entity, use the left/right arrows to cycle through the images.

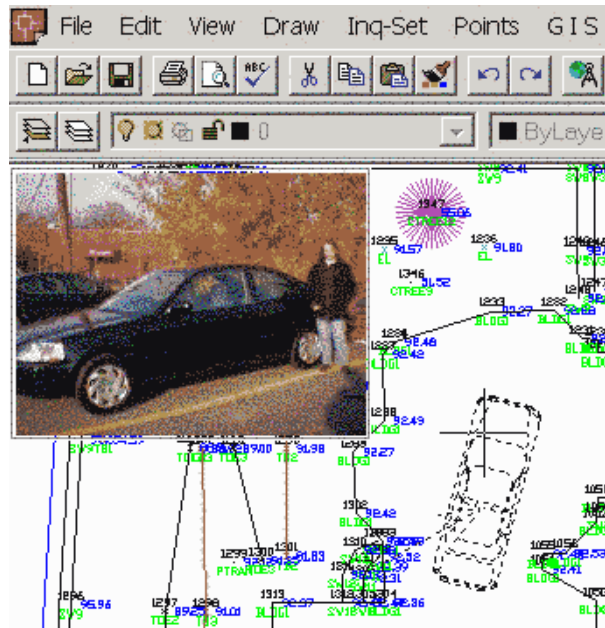
For the Dialog method, the dialog shows a list of all the images/documents found in the drawing. When the image is highlighted in the list, the dialog shows a graphic preview. You can use the Zoom buttons to zoom the drawing to the highlighted image. The Report function makes a report of all the images in the drawing.

Prompts

Inspector view [<Screen>/Dialog]? *press Enter for Screen*

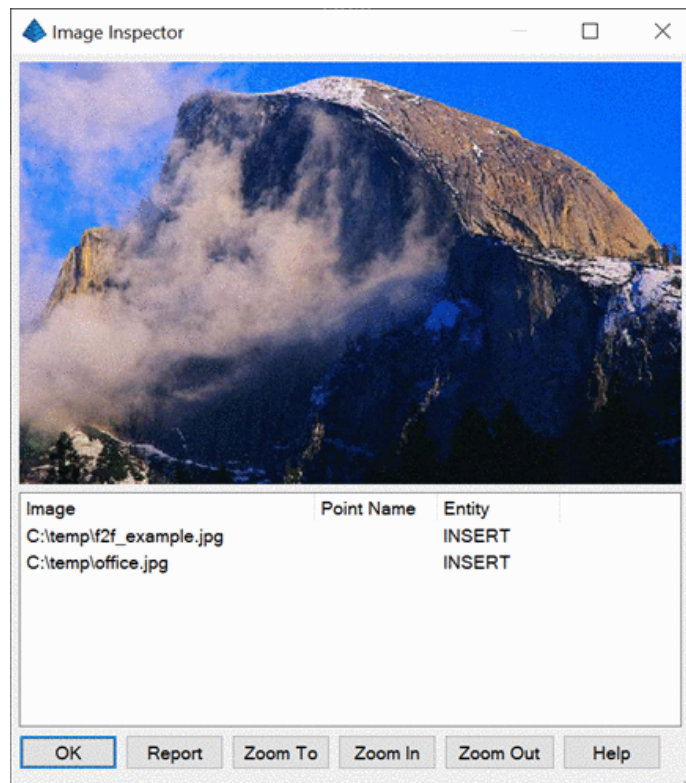
Arrow keys Up/Down=Image Size; Left/Right=Cycle Images; Pick Image=Open Image

Move pointer over entity with image (Enter to End): *press Enter*



Car image displays in upper-left of drawing when cursor is over car symbol

Inspector view [`<Screen>/Dialog`]? *D for Dialog*



Pulldown Menu Location: Images

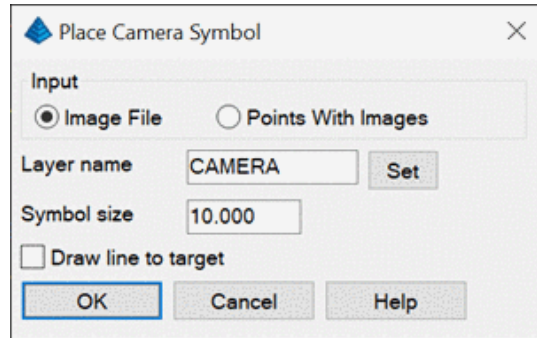
Keyboard Command: view_image

Prerequisite: drawing entity with attached image or document

Place Camera Symbol/Image

This command allows you to place the location of a camera symbol into the drawing, and then attach an image to the camera. The Draw Line To Target option draws a line from the camera symbol to the target position. There are two input methods. For the Image File, the program prompts to select an image file (.bmp, .tif, .jpg). For the Points With Images, the program prompts to select Carlson point entities in the drawing. The program will then use any images assigned to the points.

After you complete this command, use the Image Inspector command to view the attached image.



Prompts

Image File To Process *select image file*

Place Camera dialog

Pick camera position: *pick a point*

Pick target position: *pick a point to rotate camera*

Attach Image to Object dialog box

Pulldown Menu Location: Images

Keyboard Command: place_camera

Prerequisite: Image file (bmp, jpg, tif)

Attach Image to Entity

This command attaches image and document files to a drawing entity. The possible file formats are .pdf, .doc, .bmp, .jpg and .gif. Any type of drawing entity can be used such as polyline, points or symbols. To run the command, first pick an entity on the screen. Then a dialog appears for selecting the image or document. First set the image directory and then highlight the file name. A graphic of the image should appear in the preview window. Then click Attach Selected Image/Doc.

The Capture New Image button can be used to trigger an attached digital camera to take an image. The Pick Camera and Set Camera buttons can be used to configure the camera to use.

Multiple images or documents can be attached to the entity by picking Attach Selected Image/Doc or Capture New Image multiple times. To cycle the images in the preview, use the Next and Prev buttons. Use the Remove Attached Image to remove the image shown in the preview. Use Remove All Attached Images to clear all images from the entity.

The View Attached Image/Doc button will display in the preview window any image already attached to the entity instead of the selected image file. Also any image already attached to the selected entity is displayed in the Current Image field at the top of the dialog when nothing is selected in the file list.

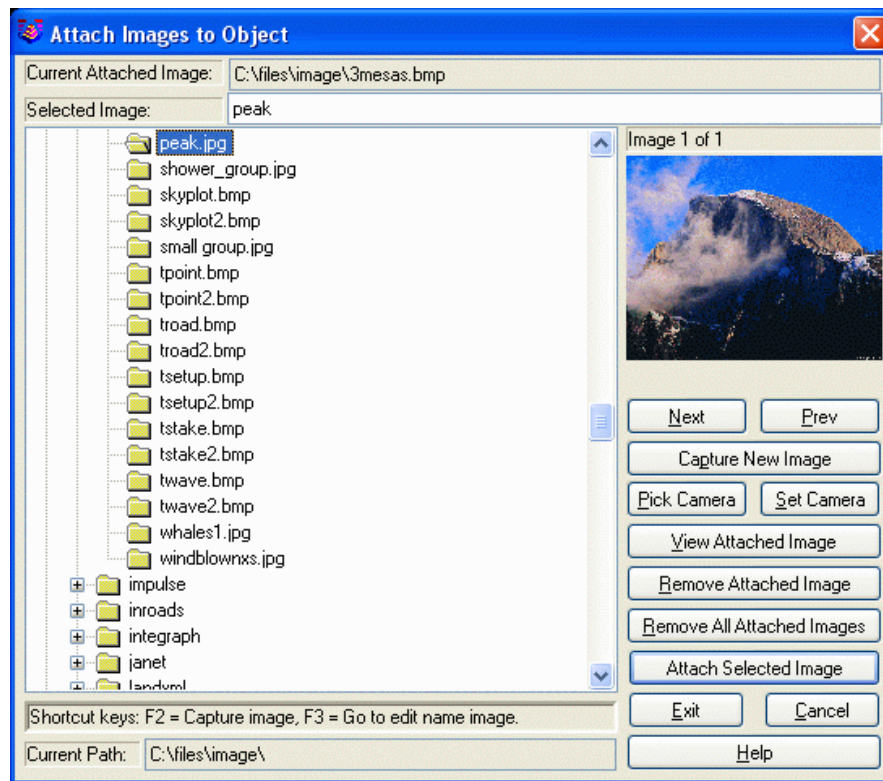
Prompts

Select object to attach symbol to: *pick an entity*

Attach Image to Object Dialog

Done.

Select object to attach symbol to: *press Enter*



Pulldown Menu Location: Images

Keyboard Command: set_image

Prerequisite: A drawing entity and an image or document file

Detach Image From Entities

This command removes image and document file links from the selected entities.

Prompts

Select entities to remove image links from.

Select objects: *select image file links*

Pulldown Menu Location: Images

Keyboard Command: detach_image

Prerequisite: image file links

Audit Image Links

This command checks links between drawing entities and attached images and documents.

Prompts

Select object to attach symbol to: *pick an entity*

Attach Image Dialog

Done.

Select object to attach symbol to (Enter to End): *press Enter*

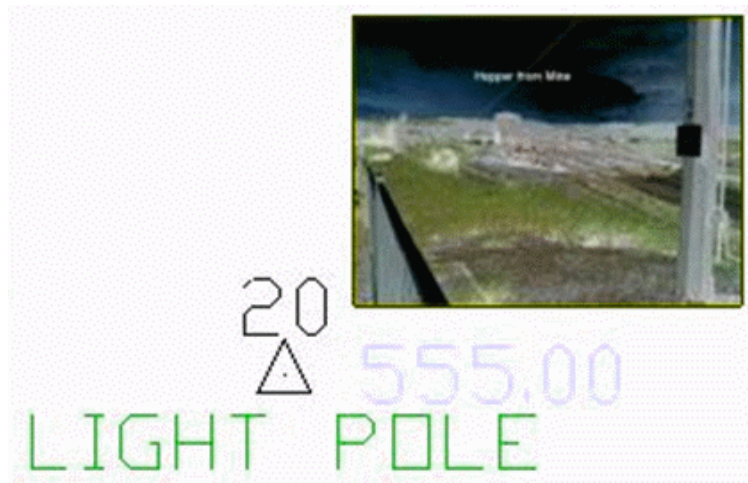
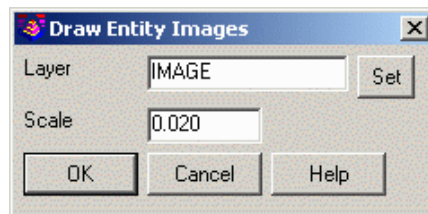
Pulldown Menu Location: Images

Keyboard Command: `audit_image_links`

Prerequisite: a drawing entity and an image file

Draw Entity Images

This command draws images that have been attached to entities by commands such as Attach Image/Doc To Entity. The options dialog sets the layer for the images and a scale factor for sizing the images. After the dialog, the program prompts for the entities to process. The images are located at the coordinates of the entities.



Pulldown Menu Location: Images

Keyboard Command: `draw_image_links`

Prerequisite: Entity with attached image

Image Set Manager

Image Set Manager is a program within Carlson GIS that is used to create image sets for CAD and SurvCE. An image set is a collection of images that cover a geographic area. Image sets can contain thousands of images which are indexed and geo-referenced in the image set database (.IDB). Once an Image Set is created, the rest of the tools in this area of the Images menu become available to view the images, Zoom on the images, Pan through them, etc...

Supported Image Types

Image Set Manager supports GeoTIF, TIF, JPEG, SID and BMP images.

Geo-referenced, (GeoTIF), images are directly supported by reading the embedded tags in the image. TIFF, JPEG and BMP are supported with world files and the World File Editor.

File Menu

Open Image Set

Creates a new Image Set or opens an existing Image Set. The image database and set of images are stored to a project folder.

Import Image Set

Adds an existing Image Set to the current Image Set.

Open Image Files

This command will open and display individual images for processing. Geo-referencing is automatic for images with World Files or supported Geo-TIF. For GeoTIF the image coordinate positions will be displayed at the bottom of the image frame.

When the image is open the following tools become available.

Image Brightness: Generalized adjustment to the tonal range. This tool is useful to lighten the image for background display

World File Editor: Allows the user to import, view, create and edit image geo-reference information.

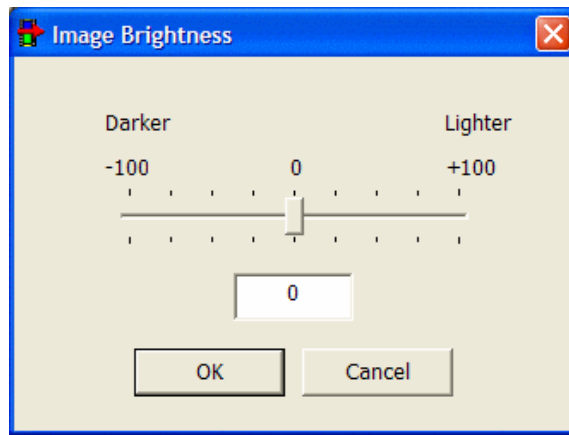
Image Properties: Displays Information about images in the Image Set.

Tools Menu

Image Brightness

The Image Brightness tool is an easy way to make generalized adjustments to the tonal range of the image.

1. Open an Image and choose Image Brightness from the Tools menu. The Image Brightness dialog box appears.
2. Drag the slider to adjust the brightness of the image. Drag the slider to the left to decrease the brightness and to the right to increase it. Values range from -100 to +100.
3. When you have finished making adjustments, click OK.



Convert to Greyscale

This command converts a color image to greyscale.

World File Editor

The World File Editor allows the user to import, view, create and edit image geo-reference information. A World File is an ASCII text file used to geo-reference image files. World files have the following format.

World File Format

```
1.0 <X Resolution>
0.000 <Amount of Translation>*
0.000 <Amount of Rotation>
1.0 <Y Resolution>
424178 <X Coordinate of 1,1 (upper-left pixel)>
4313415 <Y Coordinate of 1,1 (upper-left pixel)>
```

Image Export supports the following World File Formats.

Image Format

TIFF, TIF
JPEG, JPG
Bitmap, BMP

World File

TFW
JGW
BPW

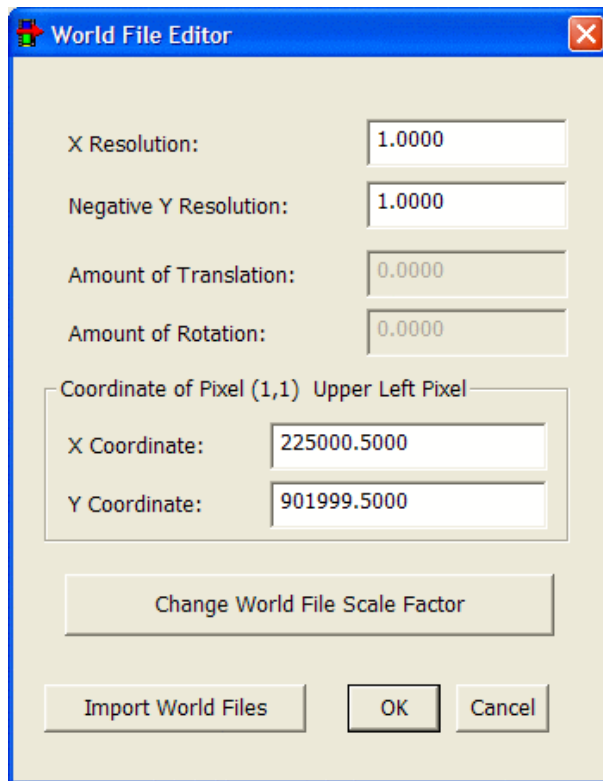
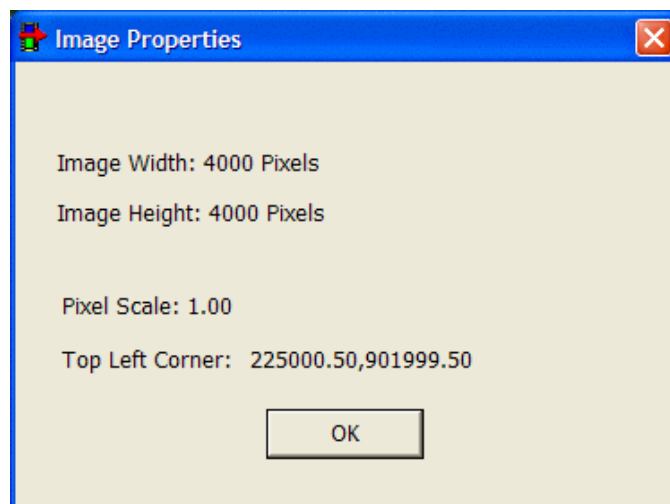


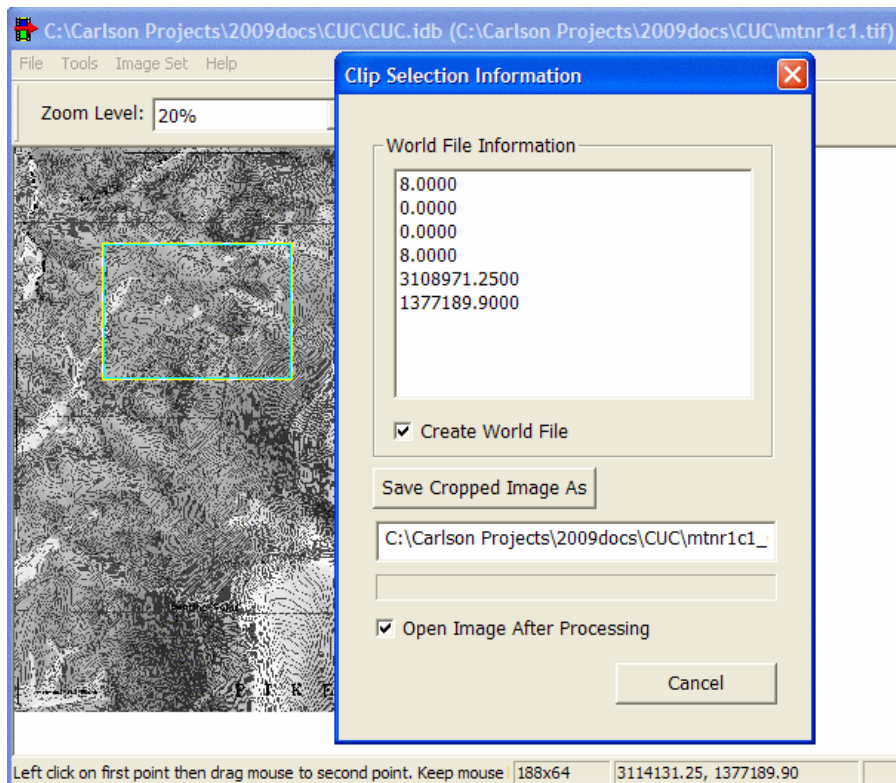
Image Properties

This option is used to display information on the current image.



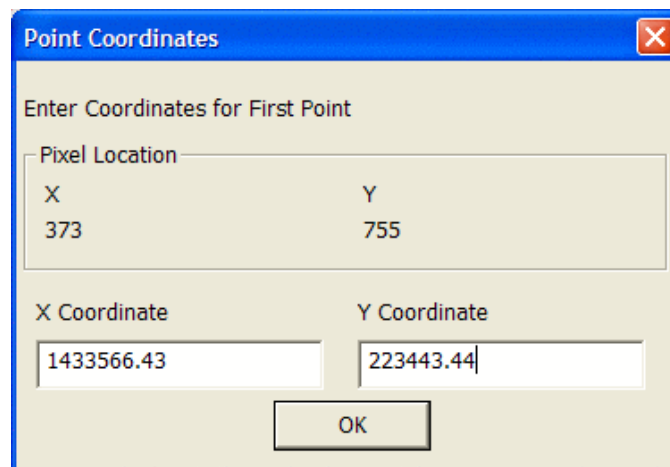
Crop Selection

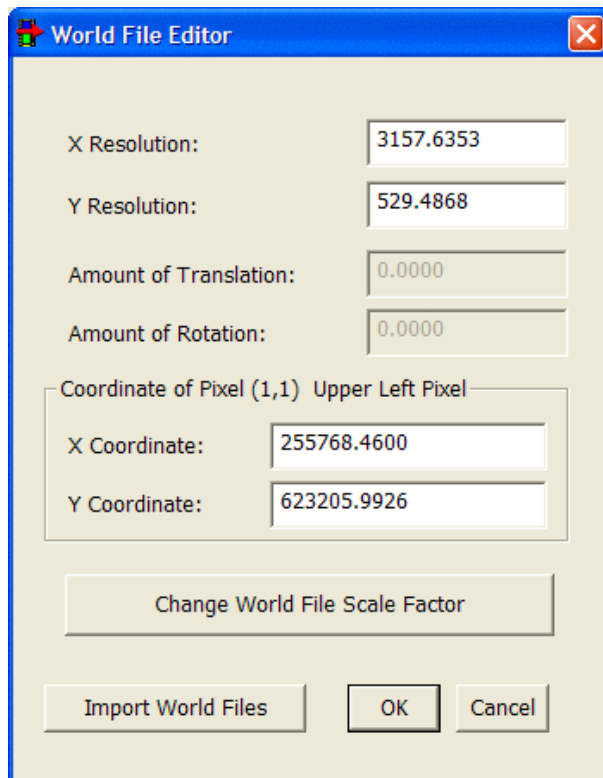
This command allows the user to crop a specific area within an image and save it as a new image.



Geo-reference by User Points

This command allows the user to pick 2 points on the image and assign coordinates to them to geo-reference the image. The user is prompted for coordinates for each point after it is picked, and then the World File Editor is displayed with the resulting data.





Change Resolution

This command allows the user to change the resolution of the image, thereby saving it as a smaller image file size.

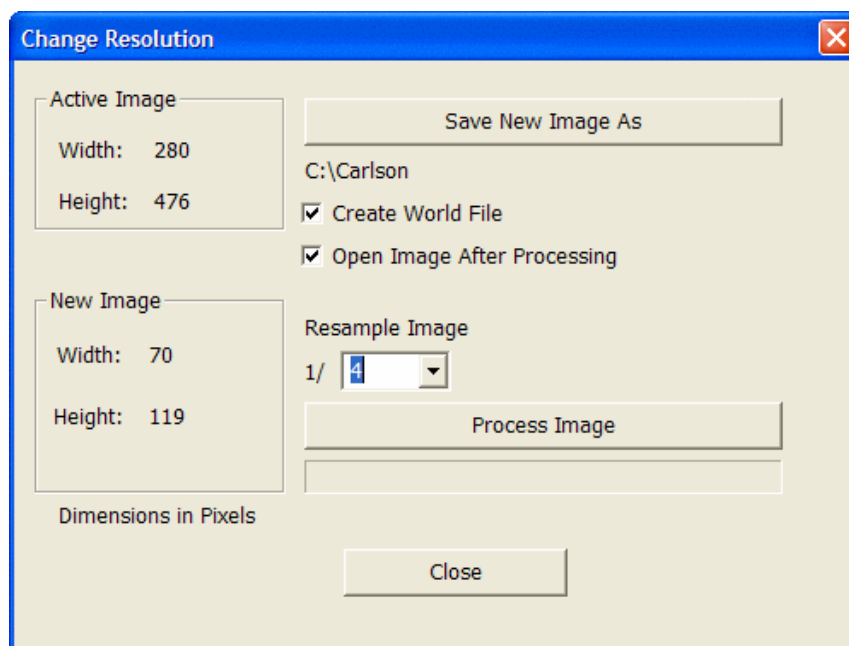


Image Viewer

Image Set Manager has an image viewer to display the current image. Zoom level is controlled by the drop down list at the top left. The X and Y pixel position and geographic coordinates are displayed in the bottom right panel.

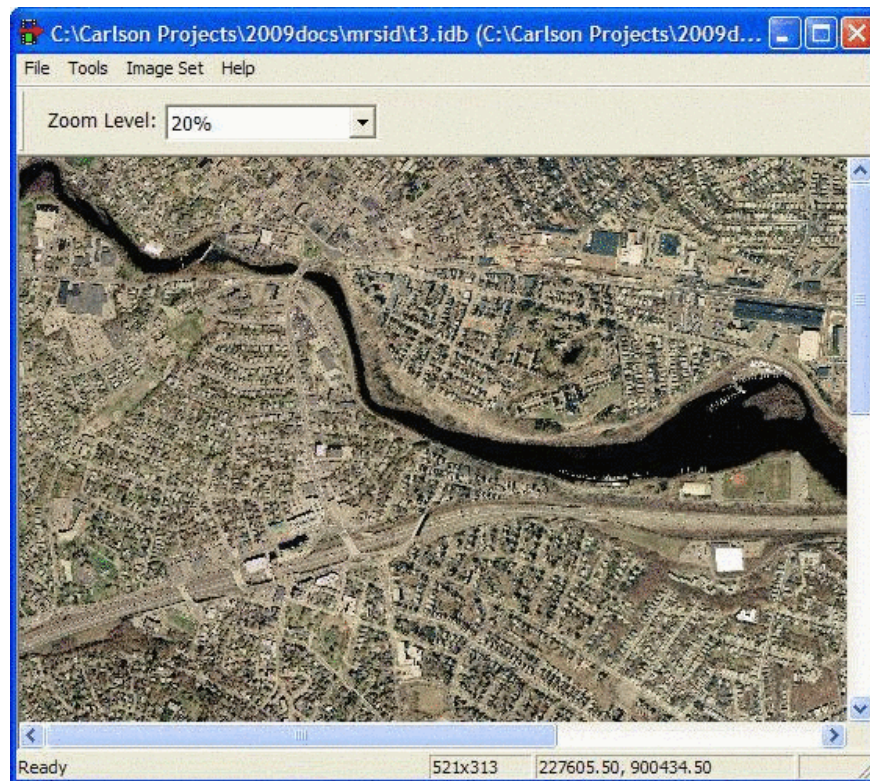
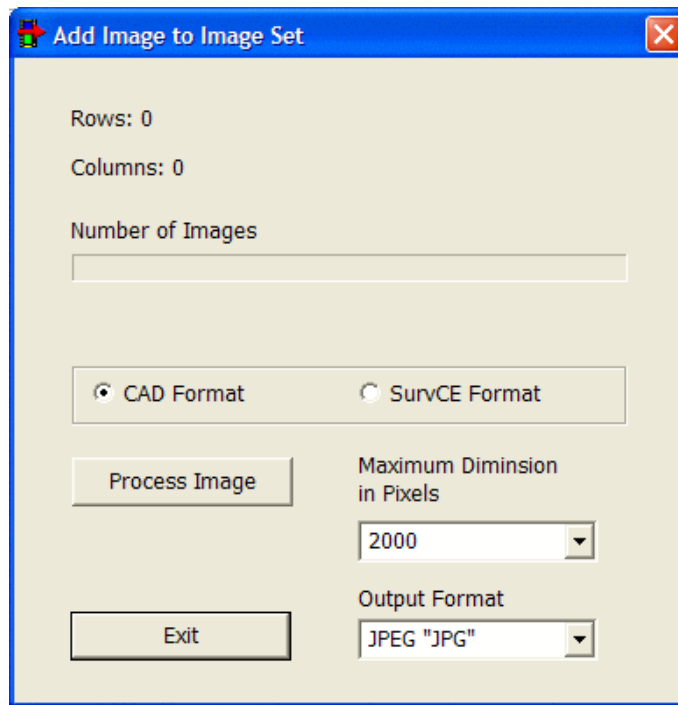


Image Set Menu

Add Image to Set

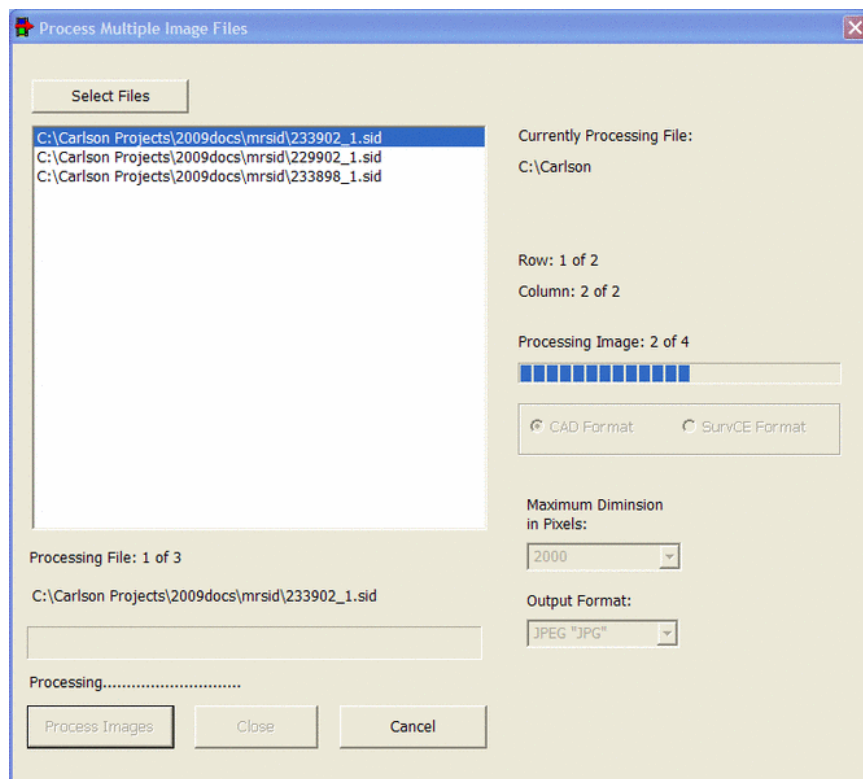
In order to display images they need to be added to the image database.

1. Open an Image using Open Image File in the File Menu.
2. Geo-reference the image. If the image is not a GeoTIF image use the World File Editor to geo-reference the image.
3. Click Process to process and add this image to the image database. This process may take several minutes. The progress bar and dialog text will keep you informed of the progress. Large images are being clipped into smaller pieces for viewing in SurvCE. When complete click OK.
4. Repeat the above steps to add multiple images to the database. Multiple images in the same database should be contiguous. Images from different locations should be placed in separate databases.



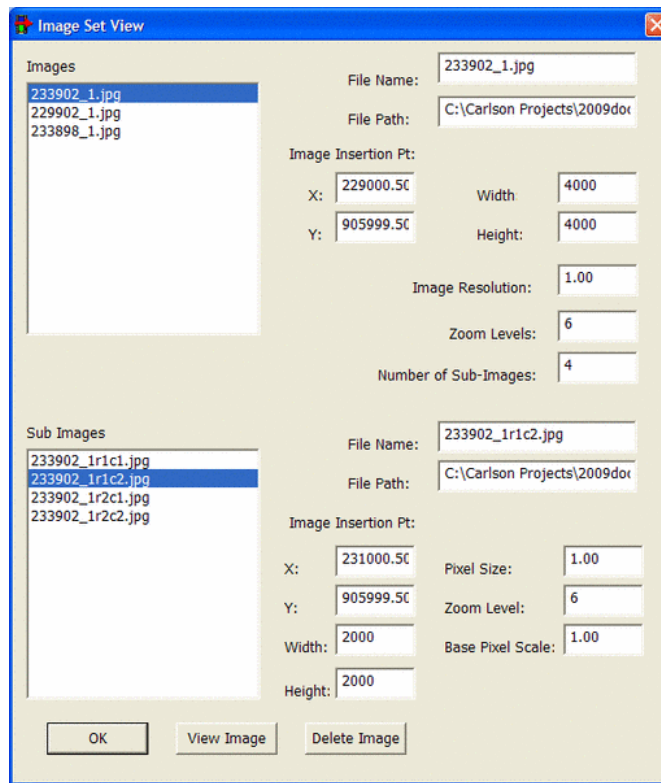
Add Multiple Images to Set

This command allows the user to add multiple images to a single Image Set. It supports the same files as the Open Image command. Images processed with this command must be geo-referenced.



View Image Set

The View Image Set command displays information for the images in the currently selected image database. The images processed are displayed in the top panel, and the sub images of the selected image are displayed in the bottom panel.



Pulldown Menu Location: Images

Keyboard Command: imagemgr

Prerequisite: None

Load Image Set

This command loads an existing Image Set into memory. Once loaded, the rest of the commands in this section of the Images menu can be utilized to view images, etc.

Pulldown Menu Location: Images

Keyboard Command: ireaddb

Prerequisite: A saved Image Set

Draw Image Boundaries

This command draws polylines in the drawing to represent the boundaries of the images in the loaded Image Set.

Pulldown Menu Location: Images

Keyboard Command: ibound

Prerequisite: A loaded Image Set

Place Image By Point

This command allows you to display images within a loaded Image Set by picking on the screen.

Pulldown Menu Location: Images

Keyboard Command: ipoint

Prerequisite: A loaded Image Set

Place Image By Circle

This command allows you display images within an Image Set that fall within the area of a specified circle.

Pulldown Menu Location: Images

Keyboard Command: icircle

Prerequisite: A loaded Image Set

Image Zoom Window

This command Zooms to an area in the loaded Image Set specified with a 2-pick Window. The image with the appropriate resolution is loaded as the Zoom level is changed.

Pulldown Menu Location: Images

Keyboard Command: izw

Prerequisite: A loaded Image Set

Image Zoom Previous

This command Zooms to the previous view of the loaded Image Set.

Pulldown Menu Location: Images

Keyboard Command: izp

Prerequisite: A loaded Image Set

Image Zoom Extents

This command Zooms to the Extents of the images in the Image Set.

Pulldown Menu Location: Images

Keyboard Command: ize

Prerequisite: A loaded Image Set

Image Pan

This command is used to Pan across images in the loaded Image Set. As you Pan, images are loaded as needed.

Pulldown Menu Location: Images

Keyboard Command: ipan

Prerequisite: A loaded Image Set

Image Redraw

This command refreshes the images in the loaded Image Set.

Pulldown Menu Location: Images
Keyboard Command: iredraw
Prerequisite: A loaded Image Set

Clear Image Set

This routine clears the loaded Image Set.

Pulldown Menu Location: Images
Keyboard Command: iclear
Prerequisite: A loaded Image Set.

Construction Module

8

The Construction module has Takeoff commands along with portions of the Survey and Civil modules. The Construction tools cover surface modeling, earthworks, takeoff, roads, machine control data prep, survey as-built mapping and stakeout. The Takeoff commands are covered in this chapter. See the Survey and Civil chapters for descriptions of those commands.

Surface

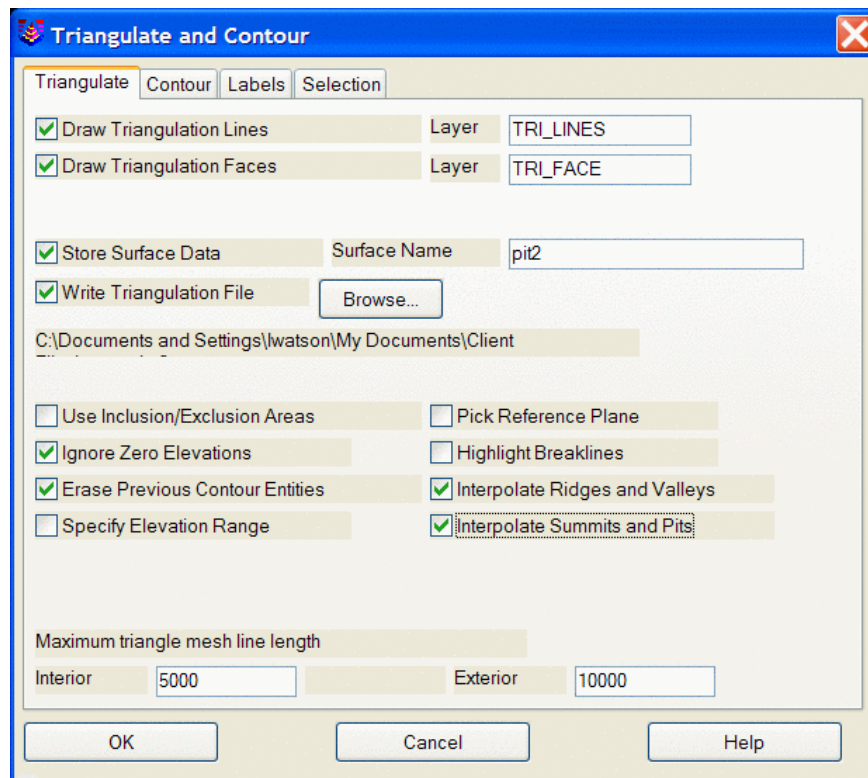
Triangulate & Contour

This command provides all of the functionality related to contouring and creating tin surface models in one routine. Given data entities that represent the surface, this command creates a final contour map with labeled, smoothed, and highlighted contours and/or a surface model that can be saved to a file (to be used in other areas of the program) or drawn on the screen as triangles or faces. Eligible data entities include points, inserts, lines, 2d polylines, 3d polylines, elevation text, 3d faces, and points from ASCII or coordinate (.CRD) files.

Triangulate & Contour has many options which are defined in the exhibits shown in the following pages. With this command, you can do any combination of drawing the triangulation network lines, drawing the contours, drawing triangulation network 3D Faces or lines, writing a triangulation file and storing a surface file.

In order to force *Triangulate & Contour* to correctly interpolate elevations between two points that define a grade break in the surface (such as points on a ridge, wall, or road), a breakline must exist between the points. A breakline line can be specified as a 3D polyline or line. In fact, all 3d polylines and lines with elevation are be treated as breaklines.

Triangulate Tab



Draw Triangulation Lines

When this option is turned on, the program will draw the triangulation as 3D lines. Specify the layer for these lines in the box to the right.

Draw Triangulation Faces

When this option is turned on, the program will draw each triangle in the triangulation network as a 3D Face. These 3D Faces can then be used in AutoCAD's modeling routines such as *HIDE* and *SHADE* or in routines such as *3D Viewer Window*, *3D Surface FlyOver* and *Slope Zone Analysis*. Specify the layer for these 3DFaces in the box to the

right.

Store Surface Data

This option names and creates a surface or surfaces that are associated with the drawing. The creation of a surface is necessary in order for the Surface Tools to function. A Triangulation file must also be specified before using the Store Surface option.

Write Triangulation File

This option stores the triangulation surface model as an .flt or a .tin file. The .flt file format is a text file depicting the edges in the triangulation network. The .tin file is a new binary file format depicting the triangulation network. The .tin file is much faster and more efficient than the previous .flt file format. The triangulation file/s can be used by several commands such as *Volumes By Triangulation*, *Spot Elevations*, and *Profile from FLT File*. Either type in the file name to create or press the Browse button to select a file name.

Use Inclusion/Exclusion Areas

When this box is activated, the program will later prompt you for inclusion and exclusion polylines which are used to trim the contours. The inclusion and exclusion polylines must be closed polylines and must be drawn before starting *Triangulate & Contour*. Only the parts of the contour lines that are within the inclusion polylines will be drawn. For example, an inclusion could be the perimeter of the site. The parts of contour lines that are inside the exclusion polylines are not drawn. Exclusion polylines can be used for areas where you don't want contours such as within buildings.

Ignore Zero Elevations

When activated, this setting will filter out all data points at an elevation of zero from the data set.

Erase Previous Contour Entities

When activated, this setting will erase previously drawn contour entities.

Specify Elevation Range

The program will automatically contour from the lowest elevation in the data set up to the highest at the increment specified in Contour Interval. If you would like to manually set the range over which to contour, select this option.

Pick Reference Plane

The triangulation network is based on the x,y position of the points. This option allows you to contour an overhang or cliff by changing the reference plane to a side view. The reference plane can be specified by first using the *Viewpoint 3D* command and then using the View option, or you can specify three data points on the cliff (two along the bottom and one at the top).

Highlight Breaklines

This option highlights breaklines in the triangulation network by drawing the triangulation lines along breaklines in yellow.

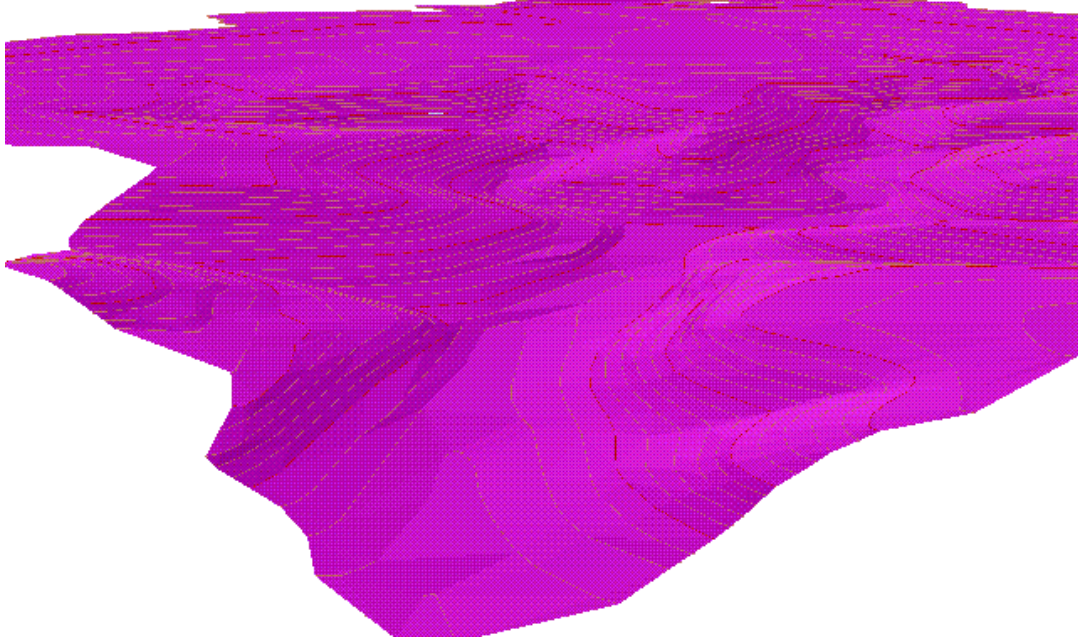
Interpolate Ridges and Valleys

This option creates additional triangulation in a ridge or valley situation to more accurately define the feature during surface modeling operations. This option would commonly be used when creating a surface model from existing contours, since it replaces the need to manually draw 3d polylines along ridges and valleys.

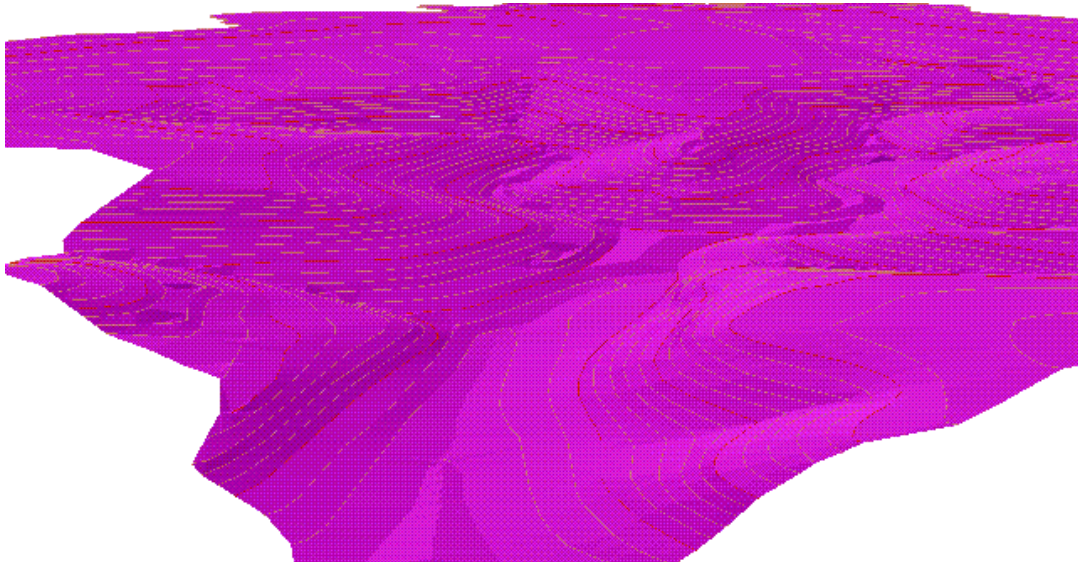
Interpolate Summits and Pits

This option creates additional triangulation in a summit or pit situation to more accurately define the feature during surface modeling operations. This option would commonly be used when creating a surface model from existing contours.

Before: Surface made from an existing contour map. Note the flat spots in the bottom of the valley when Interpolate Ridges and Valleys is disabled.



After: The same surface with Interpolate Ridges and Valleys enabled.

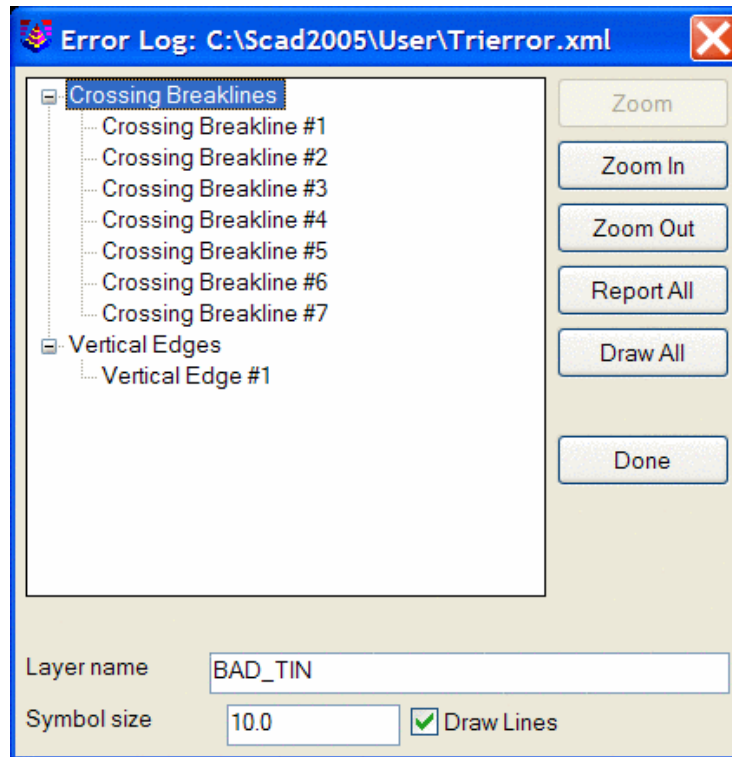


Max Triangle Mesh Line Length

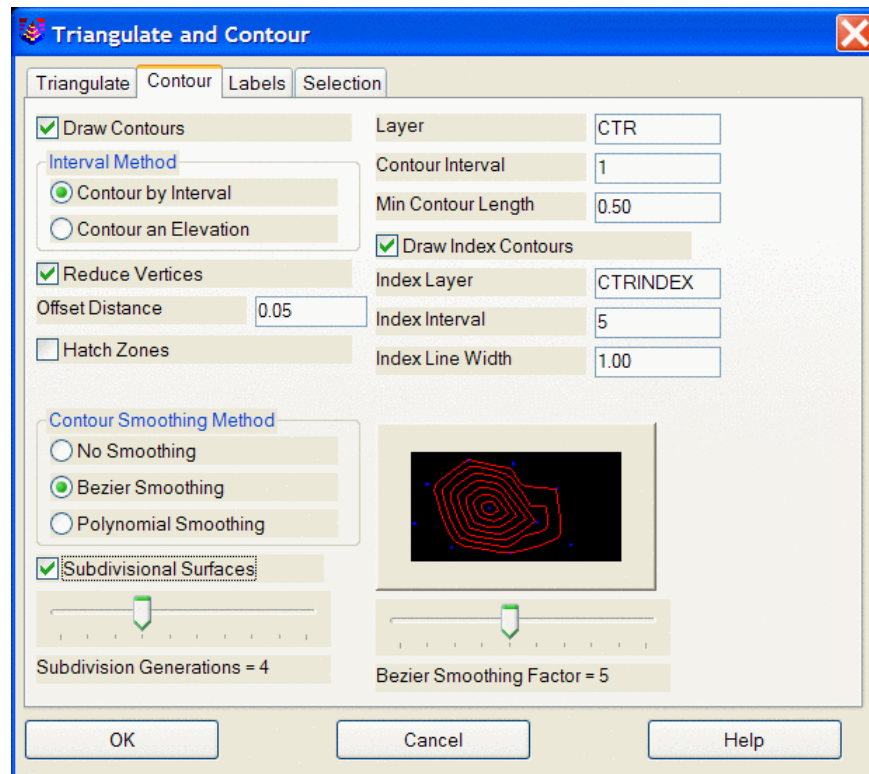
This value limits the length of the triangulation network lines. Any triangulation line that exceeds this limit will not be drawn or included in contouring. This allows you to avoid abnormally long triangulation lines where you have relatively too few data points and on the outskirts of your data points. The **Exterior** value applies to triangulation lines around the perimeter of the triangulation area and the **Interior** value applies all the other triangulation lines. Generally you would have the exterior value larger than the interior.

Error Log

The following dialog box appears when the Triangulate & Contour routine finds a conflict between breaklines or other surface entities. The type of conflict is identified, and when an item is chosen, a highlighted arrow is temporarily placed in the drawing to indicate the exact location of the specific conflict. Crossing Breaklines indicates that the intersection of two entities has two differing elevations. Vertical Edges indicates that two entities or vertexes of differing elevations have the same xy location, thus forming a vertical plane.



Contour Tab



Draw Contours

When this box is checked, the program will draw contour lines after triangulating. Otherwise, only the designated triangulation operations are performed. Specify the layer for contours in the edit box to the right.

Contour by Interval or Contour an Elevation

Select whether to contour by interval (ie: every 10 feet) or to contour a certain elevation. The elevation option allows you to contour specific values. For example, if you want just the 100ft contour, then select elevation and enter 100. The default mode is by interval.

Contour Interval

Specify the interval to contour. Note: If the above option is set to Contour an Elevation, then this field is used to specify the elevation to contour.

Minimum Contour Length

Contour lines whose total length is less than this value will not be drawn.

Reduce Vertices

This option attempts to remove extra vertices from the contour polylines which has the advantages of a faster drawing and smaller drawing size. Default is ON

Offset Distance

When the Reduce Vertices option is enabled, This value is the maximum tolerance for shifting the original contour line in order to reduce vertices. The reduced contour polyline will shift no more than this value, at any point, away from the original contour line. A lower value will decrease the number of vertices removed and keep the contour line closer to the original. A higher value will remove more vertices and allows the contour line to shift more from the original.

Hatch Zones

When activated, this option will allow you to hatch the area between the contours sequentially. A secondary dialog will load allowing the user to specify the hatch type and color.

Draw Index Contours

This option creates highlighted contours at a specified interval. When enabled, the fields for Index Layer, Index Interval and Index Line Width are activated.

Contour Smoothing Method

Select the type of contour smoothing to be performed. Bezier smoothing holds all the contour points calculated from the triangulation and only smooths between the calculated points. Polynomial smoothing applies a fifth degree polynomial for smooth transition between the triangulation faces. The smoothing factor described below affects the smoothing bulge.

Bezier Smoothing Factor

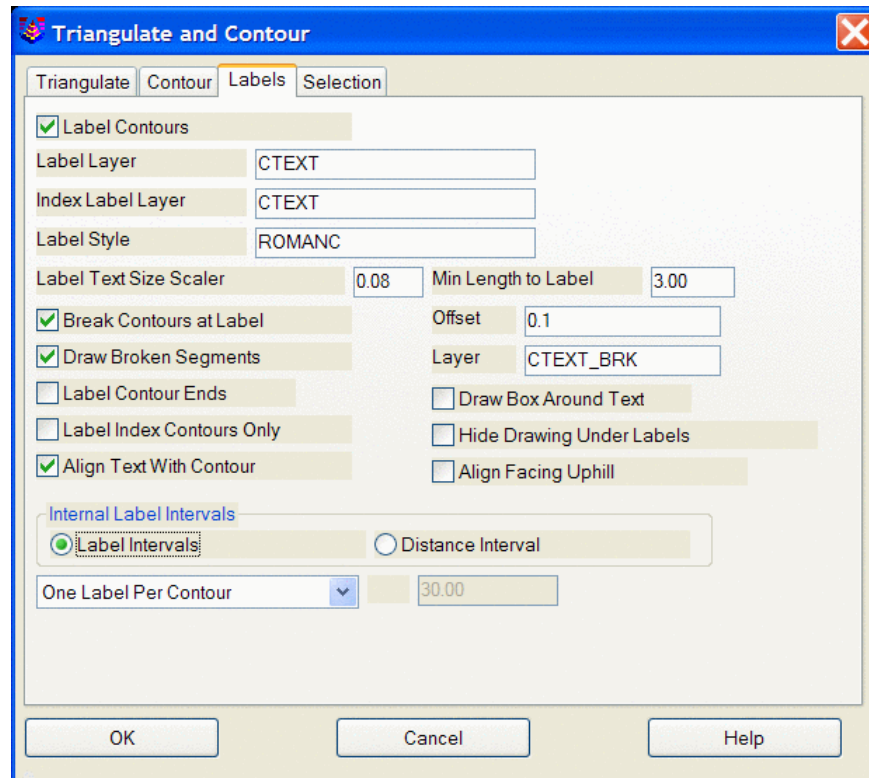
The contour preview window shows you an example of how much smoothing can be expected at each setting. Sliding the bar to the left results in a lower setting which have less looping or less freedom to curve between contour line points. Likewise, moving the slider to the right results in a setting that increases the looping effect.

Subdivisional Surfaces / Subdivisions Generation

This option causes each triangle in the triangulation surface model to be subdivided into an average of three smaller triangles per subdivision generation, with the new temporary vertices raised or lowered to provide smoother contours. More generations increases the smoothness of the algorithm at a cost of increased processing time. If Straight Lines are chosen as the contouring drawing method, then the contours are guaranteed never to cross. The original points of the surface model are always preserved. These modifications to the surface model are only for contouring

purposes and are not written to the triangulation (.FLT) file or inserted into the drawing. If some contour movement is too small for appearance's sake, consider enabling Reduce Vertices.

Labels Tab



Label Contours

When activated, contours will be labeled based on the settings below.

Label Layer

Specifies layer name for intermediate contour labels.

Index Label Layer

Specifies layer name for index contour labels.

Label Style

Specifies the text style that will be used for the contour label text.

Label Text Size Scaler

Specifies the size of the contour labels based on a multiplier of the horizontal scale.

Min Length to Label

Contours whose length is less than this value will not be labeled.

Break Contours at Label

When checked, contour lines will be broken and trimmed at the label location for label visibility. When enabled, the Offset box to the right activates. The Offset determines the gap between the end of the trimmed contour line and the beginning or ending of the text.

Draw Broken Segments

When checked, segments of contours that are broken out for label visibility will be redrawn as independent segments. Specify the layer for these broken segments in the box to the right of this toggle.

Label Contour Ends

When checked, contour ends will be labeled.

Draw Box Around Text

When checked, a rectangle will be drawn around contour elevation labels.

Label Index Contours Only

When checked, only the index contours will be labeled. This option is active only when "Draw Index Contours" has been selected in the Contour tab of the main dialog.

Hide Drawing Under Labels

This option activates a text wipeout feature that will create the appearance of trimmed segments at the contour label, even though the contour is fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels, and the contour lines themselves remain contiguous. This feature will also hide other entities that area in the immediate vicinity of the contour label.

Align Text with Contour

When checked, contour elevation labels will be rotated to align with their respective contour lines. This option also activates the Align Facing Uphill feature explained below.

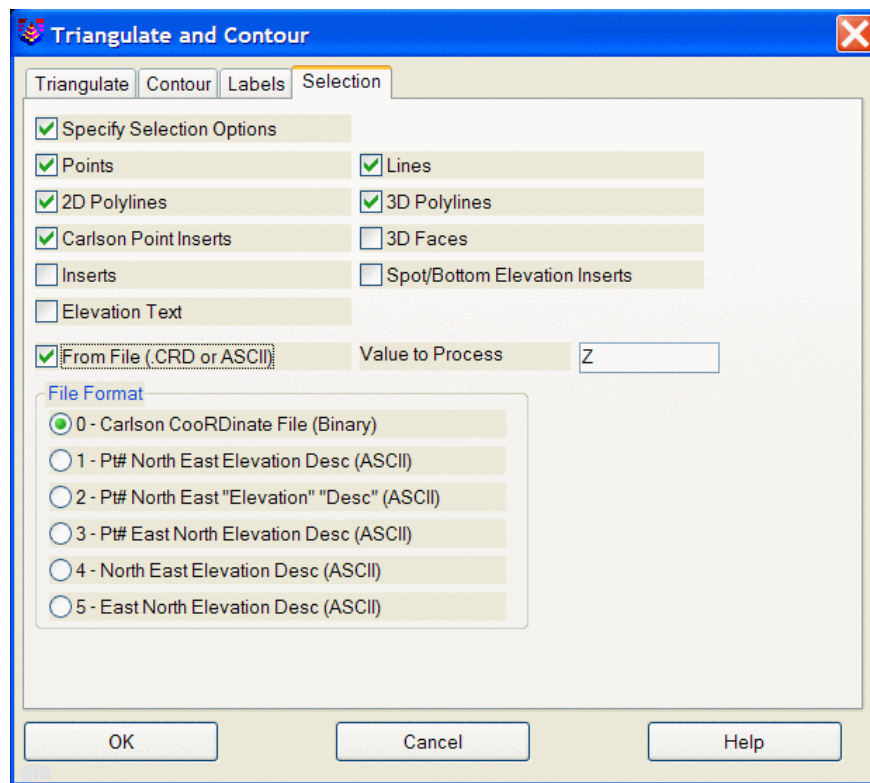
Align Facing Uphill

When checked, contour elevation labels will still be rotated to align with their respective contour lines, but the labels will be flipped in such a manner that the bottom of the text label will always be toward the downhill side of the contours. So as the labels are read right side up, you are always facing uphill.

Internal Label Intervals

Choose between label intervals or distance interval. Label intervals will label each contour with a set number of labels. Distance interval lets you specify a distance between labels.

Selection Tab



Specify Selection Options

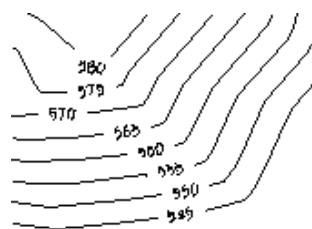
When checked, this allows you to control what type of entities *Triangulate & Contour* uses.

Points, 3D Polylines, 2D Polylines, Lines, Inserts are standard AutoCAD entities types.

Spot/Bottom Elevation Inserts include text entities that start with 'X'.

From File allows you to triangulate from the points in a coordinate (.CRD) or ASCII file.

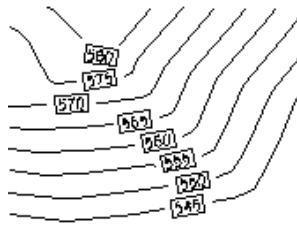
Label Contour Ends



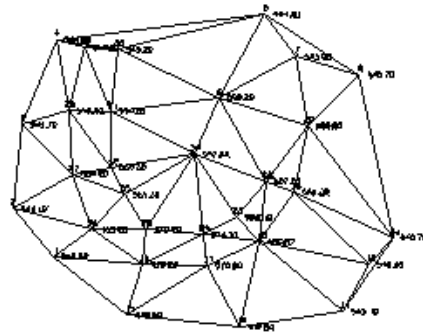
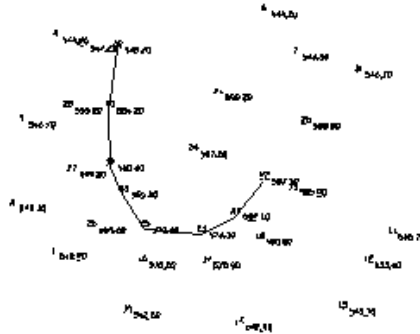
Align Text With Contour ON



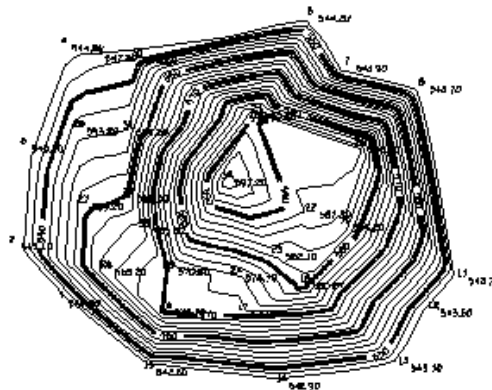
Align Text With Contour OFF



Draw Box Around Text



Triangulation network without contouring
Original data points with one 3D polyline



Contours without triangulation network. The contours are smoothed, reduced, drawn at an interval of 2, and highlighted at an interval of 10 with labeling on the index contours.

Pull-Down Menu Location: Tools-> Surface Tools

Prerequisite: Data points of the surface

Keyboard Command: tri

Triangulation File Utilities

This command allows you to modify TIN surfaces in a variety of different ways, then allows for 3d viewing and shading of the modified surface and finally for saving the file with a choice of output formats. The focus of the routine is to elevate or lower the TIN or selected areas within the TIN, merge TINs with other surfaces, or use data from other TIN files to apply to the current TIN. Operations can be performed on the entire TIN or just on user selected Inclusion and/or Exclusion areas. The routine will automatically rework the TIN network for conformation to a selected boundary, say a building outline. In the case of said building, a value of 10 could be subtracted from the building outline. This will drop all of the triangulation within the outline by 10', thus creating a model of the excavated area for the building. The modified TIN can then be saved to a new file, which could be used to compute an excavation volume with Volumes by Triangulation. This routine does not allow for manual reconfiguration of the TIN network. This is performed under Surface Tools, also in the Contour pulldown menu. This routine also includes conversions to and from TIN files, DXF files and 3D Face entities.

Begin with the dialog shown here. First select a TIN model. You may choose between an .flt or .tin file, a DXF file (that includes 3DFACE entities), or 3DFACE entities in the current drawing. Specify the subject area by choosing inclusion or exclusion perimeters, then press the next button.

Load TIN File: Allows you to specify a triangulation (.flt or .tin) file to load.

Load DXF File: Allows you to specify a DXF file to load. Only loads 3DFACE entities from the selected DXF file.

Select 3D Faces: Allows you to select 3DFACE entities from the current drawing. This also includes rectangular 3d faces from a plotted grid.

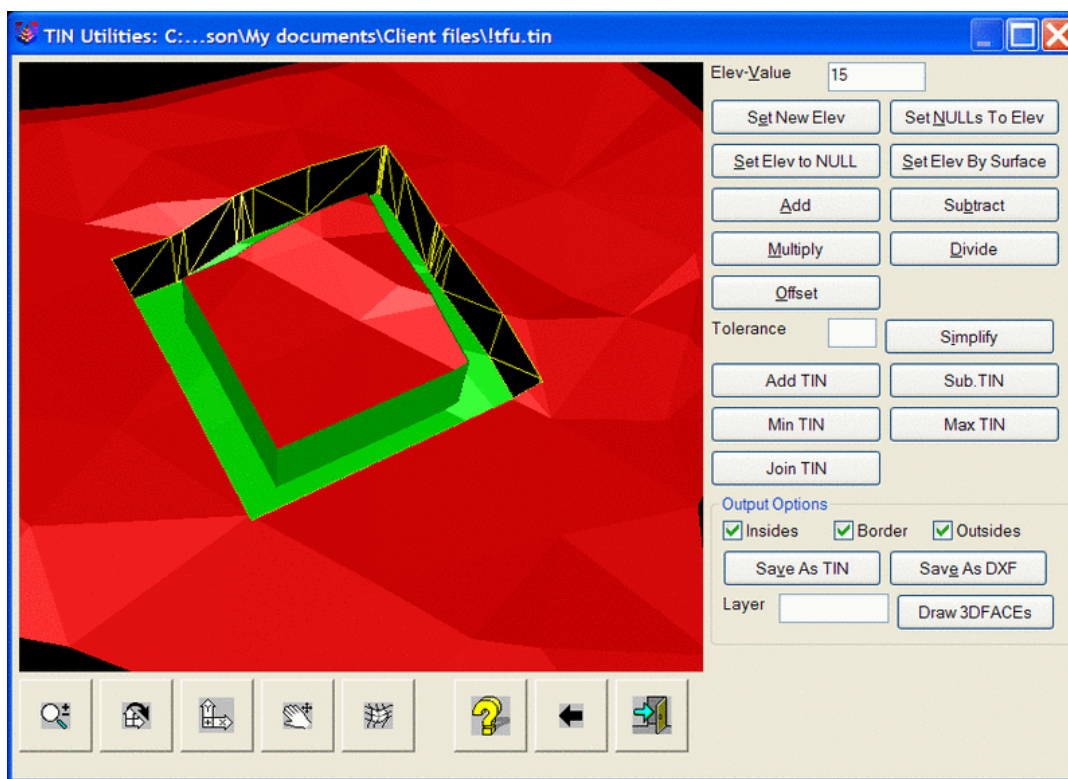
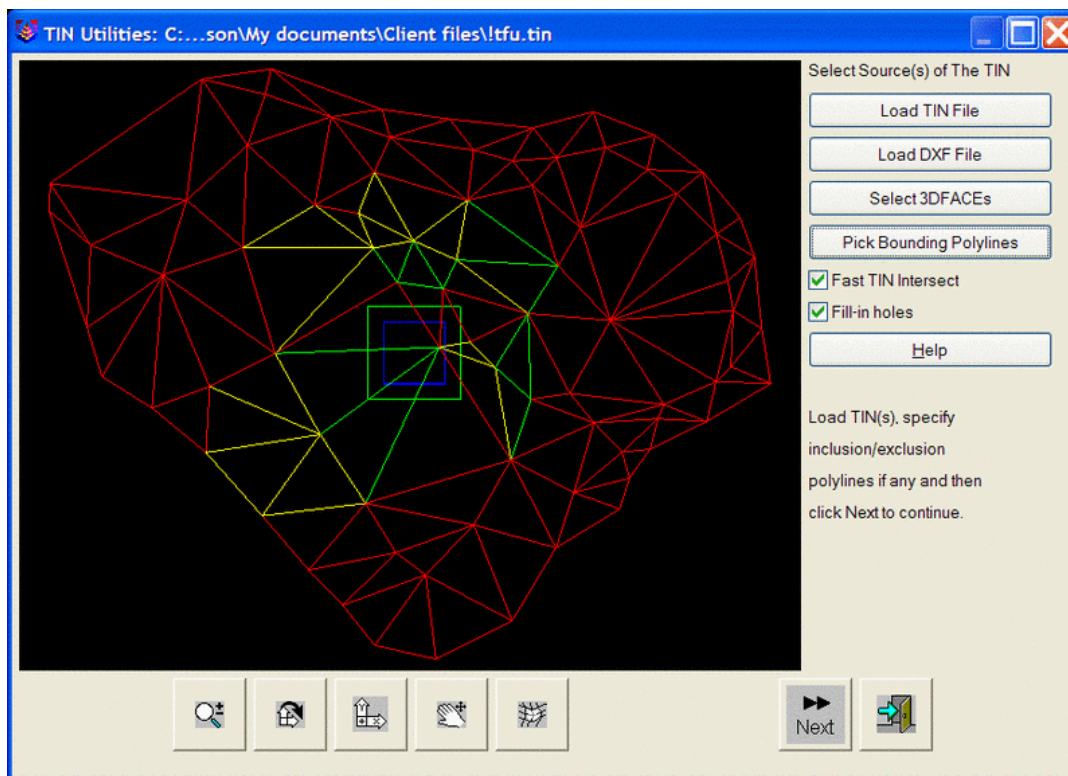
Pick Bounding Polylines: Allows you to select any inclusion/exclusion perimeter(s). When this button is selected, the user is taken back to the drawing and prompted to select the perimeters. Press Enter when the selections are finished to return back to the dialog.

Fast TIN Intersect: When checked, this command will not try and intersect 3DFACE entities.

Fill-in-holes: When checked, any missing triangulation or gap in the surface will be automatically filled in with additional triangles. This option has to set before loading the TIN file to take effect.

Next: Press this button to proceed to the next dialog after all selections have been made .

The next dialog allows you to perform mathematical operation(s) on the loaded TIN. Each operation is described below. Keep in mind that generally these operations are to be performed on an area inside your inclusion perimeter (but excluding anything inside your exclusion perimeters). If you



do not specify any perimeters, the desired operation/s will be performed on the entire TIN.

Elev-Value: Specify either an elevation or value depending on the operation. The value specified will be used for subsequent operations.

Set New Elevation: Sets all TIN faces in the subject area to the elevation specified in the Elev-Value field.

Set NULL's to Elevation: Sets all NULL values in the subject area to the elevation specified in the Elev-Value field.

Set Elevation's to NULL: Sets all of the elevation values in the subject area to NULL.

Set Elevation by Surface: Sets all TIN faces within the subject area to the elevations from a second surface file within the same area. You will be prompted to select a second TIN file or grid file. Only areas common to both surfaces will be applied to the subject TIN.

Add: Adds the value specified in the Elev-Value field to the subject area of the TIN.

Subtract: Subtracts the value specified in the Elev-Value field to the subject area of the TIN.

Multiply: Multiplies by the value specified in the Elev-Value field to the subject area of the TIN.

Divide: Divides by the value specified in the Elev-Value field to the subject area of the TIN.

Offset: Performs a perpendicular offset of the TIN surface by the specified amount.

Tolerance: This setting is used by the Simplify command described below. Specify the maximum average distance that any point can be moved outside of the plane of any triangle that connects to that point. Values might range from .01 to .1 for most purposes.

Simplify: Causes edges within the Tin mesh to be collapsed to reduce the number of triangles, edges, and points within the mesh while having a minimal impact on the overall shape of the mesh.

Add TIN: Raises the subject area of the current TIN by the elevation value from a second user selected TIN file. This function is most applicable to applying a strata thickness TIN.

Subtract TIN: Lowers the subject area of the current TIN by the elevation value from a second user selected TIN file.

Min TIN: This does a comparison between the current TIN and a second user selected TIN file, and applies the lower value of the two TINs to the subject area.

Max TIN: This does a comparison between the current TIN and a second user selected TIN file, and applies the higher value of the two TINs to the subject area.

Join TIN: Merges the existing subject TIN into a second user selected TIN file. The subject TIN file should be the smaller of the two surfaces since the subject file will be joined or merged into the second file.

Insides: If this is the only option checked, only changes made within the inclusion perimeter will be saved. TIN entities outside of the perimeter will not be saved.

Border:







Outsides: If this is the only option checked, TIN entities inside of the inclusion perimeter will not be saved. Everything outside of the perimeter will be saved.

SaveAs TIN: Saves the current TIN as an .flt or .tin file.

SaveAs DXF: Saves the current TIN as a DXF file. This format can be used by many other CAD programs.

Draw As 3DFaces: Draws the current TIN as 3D Faces in the current viewport. The Layer window is used to specify the layer that the faces will be created in.



This icon converts the right mouse button to a zoom function. Hold the button down and move the mouse up or down to zoom in and out.  This icon converts the right mouse button to a rotate function. Hold the button down to rotate the view in any X, Y or Z direction. When the XY appears in the window, the rotation will occur relative to the XY axis. When the mouse is moved toward the outer perimeter of the window, the XY will change to a Z. Holding the button down while the Z is visible will rotate the drawing on the Z axis.  This icon converts the right mouse button to a pan function. Hold down on the button while moving the mouse to pan. Holding down the mouse wheel will also serve as a pan function in any of the above modes.  This icon toggles shading on and off.  This icon restores the graphics to plan view.  This icon reverses the effects of all operations performed on the TIN and reverts it back to its original status.  This icon exits the routine. If the TIN has been modified, you will be prompted to save.

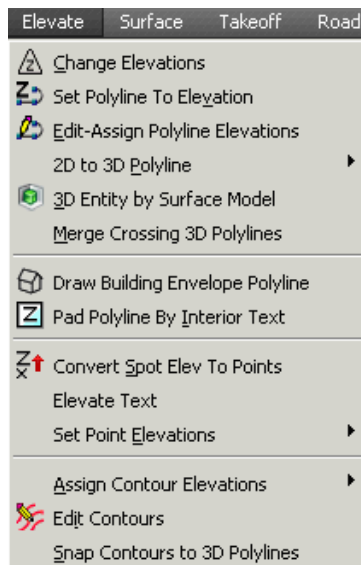
Pull-Down Menu Location: Tools-> Surface Tools

Prerequisite: 3D Faces, a TIN file or a DXF file.

Keyboard Command: TINUTIL

Elevate Menu

The Elevate menu commands convert 2D entities to 3D.



Change Elevations

This command will change the elevation of selected Entities. It can move the entity to a specified elevation from its current elevation (absolute) or do a differential change by adding or subtracting a value from its current elevation. If Carlson Takeoff points are selected, their attribute text and z axis coordinate are changed.

Prompts

Ignore zero elevations (<Yes>/No)? *Press Enter.* If you answer *No*, then entities with elevation 0 will be changed.

[A]bsolute or [D]ifferential Change <A>: A

Select/<Enter Elevation <0.0000>>: 125

Change Layer for changed entities [Yes/<No>]: No

Elevation to change to:

By using the Absolute option all entities selected are changed to the elevation 125.

Select Entities for elevation change.

Select objects: C

First corner: *(pick point)*

Other corner: *(pick point)*

Select objects: *[Enter]*

Keyboard Command: chgelev

Prerequisite: Something to change

Set Polyline to Elevation

This command allows you to assign elevations to one or more polylines. The elevation can be assigned by entering in the value or by picking a text entity that has the elevation.

Prompts

Select/<Enter Elevation <0.0000>>: Select a text entity or type in an elevation. Press enter for the default elevation in brackets.

Select Polyline for elevation change. Pick on the screen a polyline you wish to change such as:

LWPOLYLINE

Done.

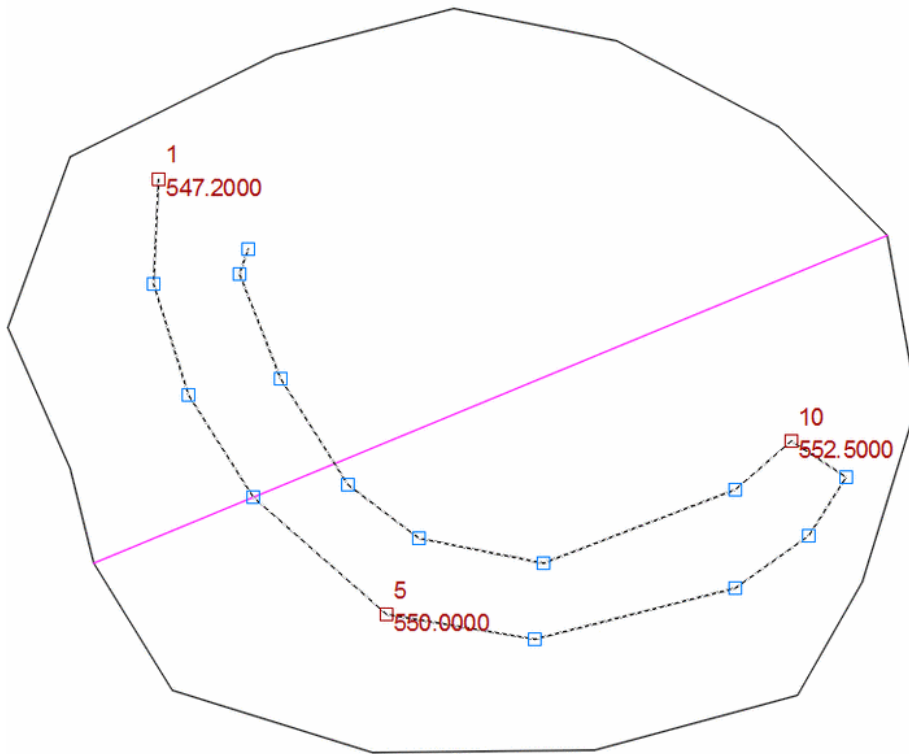
Set another polyline [<Yes>/No]? Press Y to pick another polyline to assign an elevation to. Type in N to finish the command.

Keyboard Command: set_pline_z

Prerequisite: A polyline and an elevation to assign it.

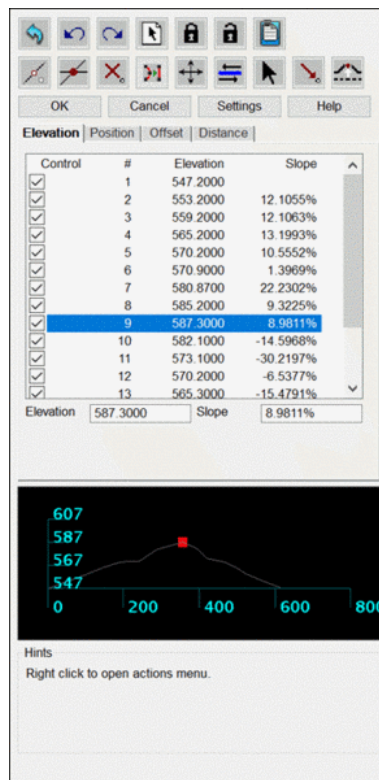
Polyline Elevation Editor

This command allows very precise control of 3D Polylines, specifically in the ability to edit vertex elevations, as well as add, delete, or move vertices. If these polylines are used in the creation of surfaces as breaklines, any editing you do may update the surface automatically. You can also control the location of Polyline vertices as defined by the station and offset of the vertices relative to a Centerline. Polyline vertices are designated as either control or free vertices. The elevation of control vertices are set and held, the elevations of free vertices are interpolated. In the drawing, control vertices are shown by default with red boxes, free vertices with blue boxes, but you may configure these using the Grip Color in the Settings.



Polyline Elevation Editor Dock Dialog:

When you run the Edit Assign Polyline Elevations routine, you are first prompted to select a Polyline to edit. After selecting a Polyline to edit, the following dock dialog appears on the left side of your screen. This dialog is "modeless" meaning that CAD commands and controls are still active while this dialog is active.



Tabs:

The four tabs in the panel provide access to control of Polyline vertex **Elevation, Position, Offset** and **Distance**.

- **Elevation:** This tab displays the vertices of the Polyline, each with a check box to set whether it is a control vertex or a free vertex (Control), its assigned number (#), its elevation (Elevation), and the slope from the previous vertex to that vertex (Slope). Selecting a vertex highlights its grip in the drawing. Once selected, you can enter an "elevation" or "slope" edits for that vertex. If multiple vertices are selected with varying elevation or slopes then "[multiple]" will be displayed in the "Elevation" and or "Slope" edits. When multiple vertices are selected, entering values in the "Elevation" or "Slope" edits will set all selected vertices based on the entered elevation or slope values.
- **Position:** The Position tab displays the Easting and Northing coordinates of each vertex. When one vertex is selected Northing and Easting can be entered in the "Northing" and "Easting" edits. When Multiple vertices are selected average Northing and average Easting are displayed in the "Average Northing" and "Average Easting" edits. When multiple vertices are selected entering values in the "Average Northing" and or the "Average Easting" edits will set all selected vertices Northing and or Easting values to based on the entered values.
- **Offset:** The Offset tab requires the selection of a Centerline to reference. Once a Centerline is designated, the Station and Offset of each vertex relative to the Centerline is displayed and can be edited.
- **Distance:** The Distance tab shows the horizontal distance for each Polyline segment. You can change a distance by highlighting the segment on the list and editing the value in the "Horz Dist" edit. If multiple vertices are selected "[multiple]" will be displayed in the "Horz Dist" edit. The user cannot set multiple segment's horizontal distances at once.

Toolbar Commands:

The toolbar provides access to several commands, some of which are tab specific.



Revert All: Reverts currently selected Polyline vertex positions and control information to vertex positions and control information when Polyline was initially selected. (Note: Selecting a new Polyline resets initial reversion location)



Undo: Undoes the most recent command line action. (Equivalent to ctrl+z)



Redo: Redoes the most recent command line action. (Equivalent to ctrl+y)



Select Polyline: Select Polyline to edit. (Note: if no Polyline is selected a Polyline can be selected using CAD selection)




Control Selected: Set selected vertices controlled, elevation will not be interpolated for control points. (This toolbar command is only available when elevation tab is active)





Free Selected: Set selected vertices free, elevation will be interpolated for free points. (This toolbar command is only available when elevation tab is active)





Output Report: Brings up Report Formatter dialog. Output reports can contain Easting, Northing, and or Elevation information for each vertex as well as whether or not each vertex is controlled or free.


 **Create Vertex:** Creates a vertex at the station nearest the mouse click position. Control/free state depends on the "Create Vertex Mode" setting (see Settings dialog).


 **Create Vertex At Crossing:** Prompts user to select crossing Polyline(s). If Polyline(s) selection is(are) applicable, crossing vertex point(s) are added at the elevation of the selected Polyline(s) at crossing. All created vertices are added as control points.


 **Delete Vertices:** Deletes currently selected vertices.


 **Reduce Vertices:** Prompts user to select an offset cutoff. Removes those vertices from the active Polyline that will not change the position of the Polyline at that vertex by more than offset cutoff. Will not removed controlled vertices.


 **Move Selected:** Move currently selected vertices based on drag offset. (Equivalent of click and drag)


 **Reverse Polyline:** Reverses indices of vertices. This will be reflected by reversing the vertices in the currently active list view.

 **Pick Elevation/Position From Screen:** Elevation- prompts user to pick an entity from screen. Will set elevation based on entity's elevation information. Vertex set controlled. Position-prompts user to pick position from screen sets Northing and Easting based on selected position. (This toolbar command is only available when elevation tab or position tab is active).

 **Pick Elevation From Surface:** Prompts user to select a surface model (tin;ft). Then prompts user to pick a direction point to run the slope from highlighted vertex. Elevation of highlighted point is determined from surface and slope is then applied along the line to the next control point. (This toolbar command is only available when elevation tab is active and one or more vertices is selected).

 **Pick Elevation With Slope:** Prompts user to pick a point or elevation text. Then prompts user to pick slope. An elevation is then calculated based on the run to the point selected at the slope the user input. Elevated vertices will be set controlled. (This toolbarcommand is only available when elevation tab is active).

 **Offset Selected Elevation:** Prompts user to input an offset amount. Adds(subtracts) offset amount from currently selected vertices. Offset vertices will be set controlled. (This toolbar command is only available when elevation tab is active)

 **Select Centerline:** Prompts user to select a Centerline or Polyline to apply Centerline information from screen. (This toolbar command is only available when Offset tab is active)

OK:

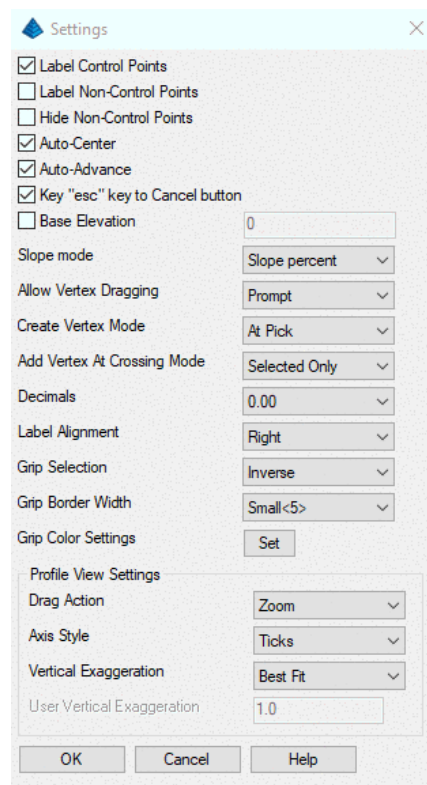
The "OK" button closes the Polyline Elevation Editor dialog, and saves the vertex position and control information of the currently selected Polyline.

Cancel:

The "Cancel" button closes the Polyline Elevation Editor dialog, and reverts the currently selected Polyline's vertex position and control information to the vertex position and control information when the Polyline was first selected.

Settings:

The "Settings" button provides control over various options pertaining to the use of the Polyline Elevation Editor routine.



- **Label Control Points:** Toggle on to display control vertices' numbers and elevations.
- **Label Non-Control Points:** Toggle on to display free vertices' numbers and elevations.
- **Hide Non-Control Points:** Toggle on to remove all free vertices' grips.
- **Auto-Center:** Toggle on to automatically center view such that all currently selected grips when they are selected.
- **Auto-Advance:** Toggle on to automatically advance the currently selected grip to the next available vertex (by vertex number).
- **Key "esc" key to Cancel button:** Toggle on to key "esc" to the Polyline Elevation Editors' dialog "Cancel" button. That is pressing the "esc" key will cancel and exit the Polyline Elevation Editor routine.
- **Base Elevation:** When toggled on the leading significant digits of the Base Elevation value will be added as a prefix to any elevation value entered. Base Elevation values must be whole numbers greater than or equal to zero. Typical use: (Base: 6000)+(User: 100)=(Elevation: 6100 displayed as 100).
 - *Conflicts:* Some user entered elevation values are ambiguous in terms of how to combine the Base Elevation, and the user entered elevation. These conflicts are dealt with as follows:
 - * *Significant Digit Conflicts:* In the case where a user entered value, and the Base Elevation value both contain significant digits in the same column(s), then the user entered value supersedes the Base Elevation value in that(those) column. Ex: (Base: 6500)+(User: 100)=(Elevation: 6100 displayed as 100).
 - * *Negative Value Conflicts:* Negative values are processed as values below the Base Elevation. However, adding a negative value can result in elevations that result in "Significant Digit Conflicts" with the Base Elevation. In this case the logic of the "Significant Digit Conflicts" applies. Ex: (Base: 6000)+(User: -100)=(Elevation: 5900 displayed as 5900).
- **Slope Mode:** Determines how slope at a vertex is displayed and interpreted.

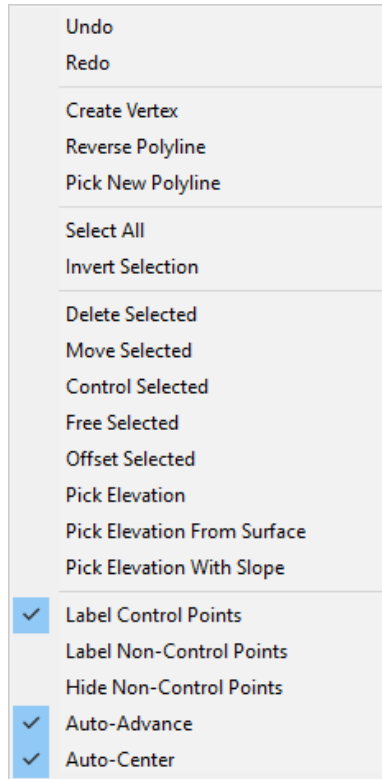
- *Slope Ratio*: Slope as a ratio of line length to line elevation [*run/rise*]
- *Slope Percent*: Slope as a percent [*rise/run*100*]
- *Slope Degree*: Slope as degree offset from 0° from -90° to 90° [*atan(rise/run)*180/PI*]
- **Allow Vertex Dragging**: Determines how dragging selected vertices functions.
 - *Never*: Never allow dragging.
 - *Prompt*: On completion of vertex dragging user is prompted on whether or not to accept the results of the dragging operation.
 - *Always*: Always allow dragging.
- **Create Vertex Mode**: Determines how elevation is calculated for vertices inserted using the Create Vertex toolbar command, double click on line, or right-click Create Vertex command.
 - *On Line*: Creates a vertex at the station nearest the pick point. Elevation is interpolated based on the equation [$elv=(InsertDistance2D/TotalDisatance2D)*(TotalElevationChange)$]. If no control points exist, then the elevation is set to the elevation of all other free vertices on the Polyline. Inserted vertex is a free point.
 - *At Pick*: Creates a vertex at pick point. Vertex is inserted in the Polyline after the vertex preceding the nearest station to the pick point. If the pick point is the result of a snap, the elevation of the new vertex is taken from the elevation of the snapped entity, and the created vertex is added as a control point. Otherwise, the created vertex is added as a free point.
- **Add Vertex Crossing Mode**: When running the function to add vertices at crossings with other linework, this option chooses whether to add intersection vertices on the crossing linework in addition to the current polyline.
- **Decimals**: Determines how many decimal places are displayed in Polyline Elevation Editor dialog. User can choose between 0 and 6 decimal places for display.
- **Label Alignment**: Determines where vertex index, and vertex elevation for all labeled vertices is drawn.
 - Right: Draw label to the "right" of vertex grip. Takes into account screen rotation.
 - Top: Draw label "above" vertex grip. Takes into account screen rotation.
- **Grip Selection**:
 - *Standard*: left-click to drag vertex, shift+left-click to select single grip vertex.
 - *Inverse*: shift+left-click to drag vertex, left-click to select single grip vertex. (legacy control)
- **Grip Border Width**: Allows user to set the relative size of the gripbox for all vertices based on the internal ACAD/ICAD GRIPSIZE variable. (Note: the values associated with each Grip Border Width choice represent the value that the internal ACAD/ICAD variable GRIPSIZE is set to for each Grip Border Width selection. If the current GRIPSIZE ACAD/ICAD variable is not one associated with any of the Grip Border Width choices ["Small," "Medium," "Large," "XLarge"] the Grip Border Width will be set to current ACAD/ICAD GRIPSIZE.)
- **Grip Color Settings**: Set the grip colors for easier editing of the vertices:
 - Hover Vertex
 - Control Vertex
 - Free Vertex
- **Profile View Settings**:
 - Drag Action: Choose between doing Zoom or Pan for click-n-drag in the profile view.
 - Axis Style: Set the profile grid display.
 - Vertical Exaggeration: Control the display for the profile vertical scale.

Help:

Display Carlson documentation for Polyline Elevation Editor.

Right-click Dialog:

There is a right-click menu available when the Polyline Elevation Editor routine is running which gives access to a variety of functions and settings. "Select All" and "Invert Selection" are the only commands unique to the right-click dialog. "Select All" selects all currently visible vertices. "Invert Selection" inverts all currently visible vertices. "Undo," "Redo," "Create Vertex," "Reverse Polyline," "Pick New Polyline," "Delete Selected," "Move Selected," "Control Selected," "Free Selected," "Offset Selected," "Pick Elevation," "Pick Elevation From Surface," "Pick Elevation With Slope," and "Pick Position" are equivalent to the Toolbar Command with the same name. "Label Control Points," "Label Non-Control Points," "Hide Non-Control Points," "Auto-Advance," and "Auto-Center" are equivalent to their settings dialog toggles with the same name.



Pulldown Menu Location: 3D Data

Keyboard Command: EDIT_PLINE.Z

Prerequisite: 3D Polylines with vertices

2D to 3D By Surface Model

This command converts a 2D polyline into a 3D polyline by calculating 3D polyline vertices at all the intersects of the 2D polyline with surface entities (contour polylines, triangulation lines) and by interpolating elevations from these intersections at the original vertices locations. An application for this command is to create breaklines. For example, a ridge breakline could be generated from contour lines by drawing a 2D polyline along the ridge and across the contours. Then this command could grab the contour line elevations along the polyline to make a ridge breakline.

In addition to using entities in the drawing, the 2D polyline can be converted to 3D using a surface model stored in triangulation (.flt or .tin) file. If you use a file, then you can also use the polyline's current elevation as a vertical offset from surface.

Prompts

By Screen Entities:

Source of surface model [File/<Screen>]? Type S for Screen

Select polylines to convert.

Select objects: *select the polyline(s) to convert*

Select surface 3DFaces, lines and polylines.

Select objects: *select the surface entities (contour polylines, breaklines, triangulation lines, etc)*

Reading points ... 692

Keep existing polylines [Yes/<No>]? Press Enter

This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.

Set layer name for converted polylines [Yes/<No>]? Press Enter

This allows you to assign the new polyline to a layer.

Converting polylines ...

Converted 1 polylines.

By a .flt or .tin File:

Source of surface model [<File>/Screen]? Type F for File

Select polylines to convert.

Select objects: *select the surface entities (contour polylines, breaklines, triangulation lines, etc)*

Use current polyline elevations as vertical offset from surface [Yes/<No>]? Press Enter

This will offset the new polyline by its current elevation. That is, if a polyline has an elevation of -4 and the surface you are converting it to has an elevation of 800, then saying Yes will drape the polyline at an elevation of 796.

Keep existing polylines [Yes/<No>]? Press Enter

This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.

Set layer name for converted polylines [Yes/<No>]? Press Enter

This allows you to assign the new polyline to a layer.

Keyboard Command: 2dto3dp

Prerequisite: A polyline and surface lines or grid file or triangulation file.

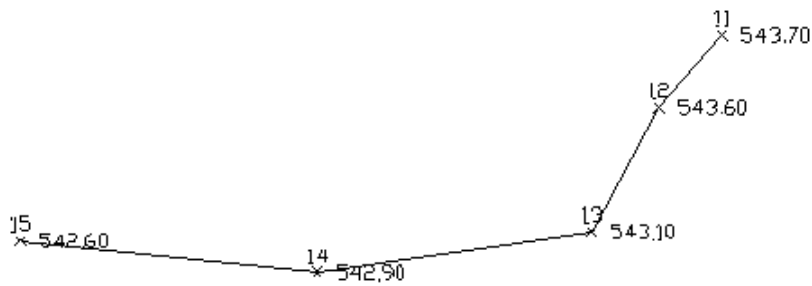
2D to 3D Polyline by Points

This command adds 3d data to polylines by using the elevations of points. At each vertex of the polylines, the program looks for a point with elevation at the same x,y location. The points can be Carlson point blocks or AutoCAD POINT entities. This routine can be useful if the linework is created in 2D at zero elevation, and points with elevation are located along the linework. It can also be used in conjunction with other 2D to 3D commands to elevate polylines by more than one method. The linework can be converted into 3D polylines with this command. For example, a centerline polyline with arcs may need to be created in 2D for stationing because AutoCAD does not allow arcs on 3D polylines. To use this polyline as a breakline in surface modeling, this command can convert the polyline into a 3D polyline.

Prompts

Select points and polylines.

Select objects: select polylines to convert and the points with elevation



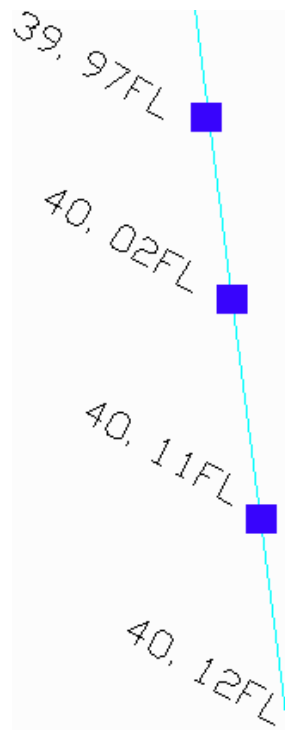
Keyboard Command: 2dto3dpt

Prerequisite: A polyline and points

2D to 3D Polyline-By Text

This command adds 3d data to polylines by elevation labels. This command will prompt you for samples of the elevation labels and the polylines to convert. The program uses these samples to know the layer names for the labels and linework to process. Then select all the polylines with their labels you want to convert.

You will then be prompted to enter in an elevation to add to label values. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 539.97, 540.02, 540.11 sometimes, like in the example on the side, they are listed as 39.97, 40.02, 40.11. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing. This command will assign elevations from the labels to nearby vertices. If vertices do not have a close elevation label than they will be interpolated from vertices that are nearby elevation labels. The vertices elevated in this command will appear as control vertices in the command Polyline Elevation Editor. It can also be used in conjunction with other 2D to 3D commands to elevate polylines by more than one method.



Prompts

Select sample of elevation text: Pick a text label

Select sample of a polyline to convert: Pick a polyline

Select polylines to convert and elevation labels.

Select objects: Select all the entities to process
19 found, 19 total

Enter elevation to add to label values <0.00>: 500

Pre-processing entity #19 of 19

Filtering text entities

Processing elevation text #18

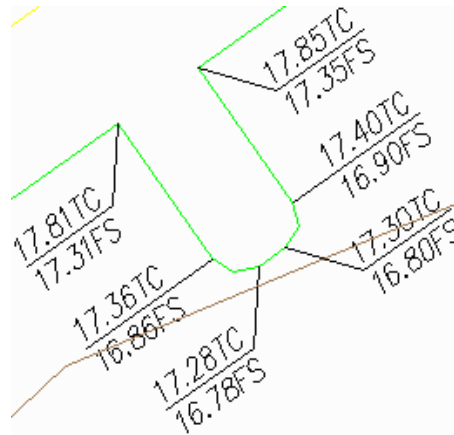
Remaking polyline #1

Keyboard Command: elevfb

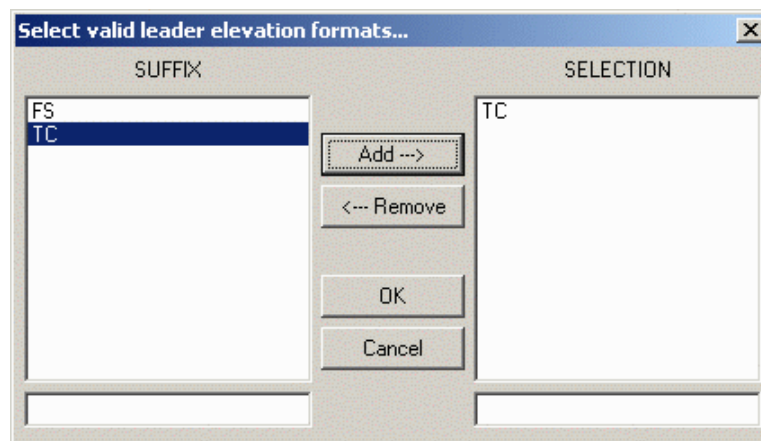
Prerequisite: 2D polyline and elevation labels

2D to 3D By Text With Leader

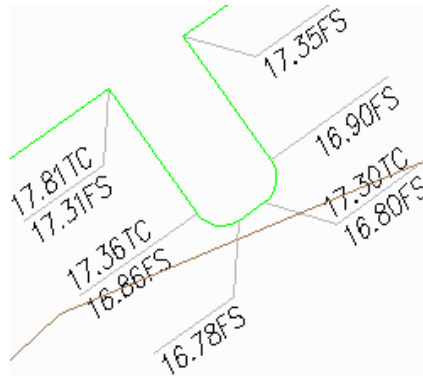
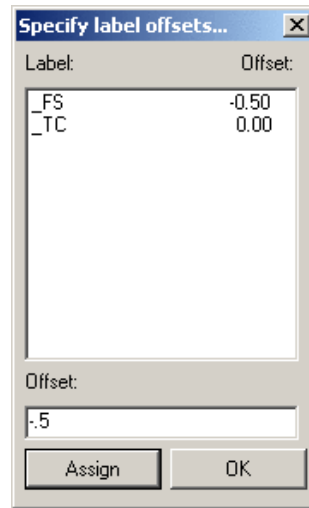
This command will assign elevations from the labels to the polylines by following the label leaders to their corresponding vertices on the polyline.



This command will prompt you for samples of the elevation labels, the leaders, and the polylines to convert. The program uses these samples to know the layer names for the labels and linework to process. Then select all the labels and leaders for the polylines you want to convert. You will then be prompted to enter in an elevation to add to label values. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 800s instead of labeling every elevation 817.85, 817.40, 817.30 sometimes, like in the above example, they are listed 17.85, 17.40, 17.30. This command allows you to add a given amount, such as 800, to every label elevation to produce the correct elevation in the drawing.



Carlson Takeoff searches for all leaders and gathers their associated text. If the program finds different labels in the elevation text, then this dialog box allows you to select the text you want to create 3D polylines. In this example you might want to use elevations followed by TC. This dialog box allows you to select that text and exclude the other text which is not to be used in the elevations of the polyline, such as FS.



If you are creating 3D polylines from multiple elevation labels than this dialog box will allow to offset certain labels by a given amount. In the above example you can offset an elevation labeled FS by .50 so that it matches vertices set by TC labeled elevations. The vertices elevated in this command will appear as control vertices in the command Polyline Elevation Editor. It can also be used in conjunction with other 2D to 3D commands to elevate polylines by more than one method.

Prompts

Select sample of elevation text: Pick a text label

Select sample of an annotation leader: Pick an annotation leader

Select sample of a polyline to convert: Pick a polyline

Select polylines to convert, leaders and elevation labels to process.

Select objects: Select the desired entities

22 found

3 were filtered out.

Select objects:

Enter elevation to add to label values <0.00>: 800

Pre-processing entity #19 of 19

Filtering text entities
Processing leader #6
Remaking polyline #1

Keyboard Command: elevfl

Prerequisite: 2D polyline, elevation labels, and leaders

2D to 3D Polyline by Start-End Elevations

This command allows you to convert a 2D polyline to a 3D polyline by specifying the starting and ending elevations of the polyline. All intermediate polyline vertex elevations are linearly interpolated from these end point elevations.

Prompts

Select polyline to assign elevations:

Enter starting elevation: 109.85

Percent/Ratio/<Enter ending elevation>: 112.16

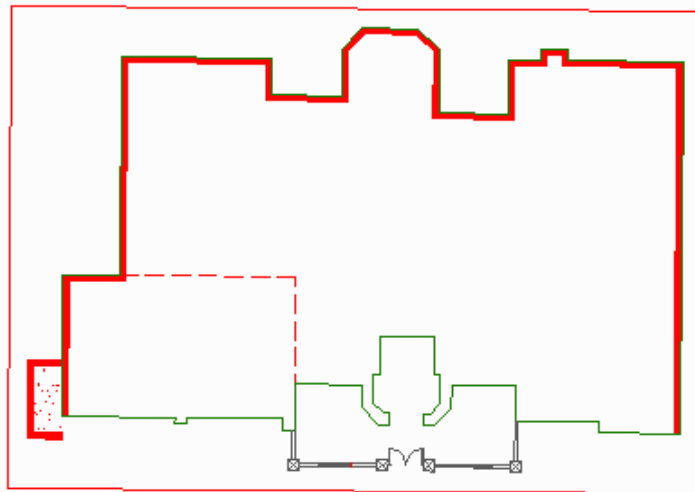
Select polyline to assign elevations (Enter to End): *press enter to end*

Keyboard Command: 2dto3dpl

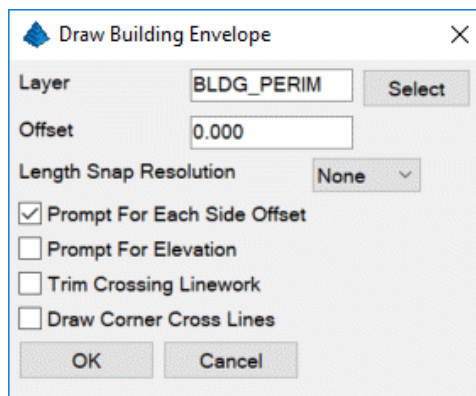
Prerequisite: A polyline

Draw Building Envelope Polyline

This command creates a rectangular polyline around selected linework. This can be used to give a building all one elevation.



Select the entities that make up the building. Next you will be prompted to name the layer and In the dialog, you can set the layer name for the rectangle and other settings.



Offset: This is the amount the building envelope will be offset out from the source linework.

Prompt For Each Side Offset: This option allows you have different offsets for each side of the rectangle.

Length Snap Resolution: Will round the dimensions of the created Building Envelope by a certain tolerance. For example, if you select None you may get a Building Envelope of 37.4 x 25.2. However, if you set the Length Snap Resolution to 0.5, you will get a Building Envelope of 37.5 x 25.0.

Prompt For Elevation: This option allows you to set the elevation for the rectangle. Otherwise the rectangle is drawn at zero.

Trim Crossing Linework: This option trims any linework that crosses the building envelope which applies to linework like contours to ensure that you have a flat pad when building the surface model.

Draw Corner Cross Lines: This option draws two lines between the corners of the rectangle.

Prompts

Select building lines.

Select objects: *pick the linework that makes up the perimeter of the building*

Enter the segment horizontal offset <0.000>: 10

Enter the segment horizontal offset <10.000>: Enter

Enter the segment horizontal offset <10.000>: 5

Enter the segment horizontal offset <5.000>: Enter

Select/<Enter Elevation <0.0000>: 400

Draw another building envelope [<Yes>/No]? No

Pulldown Menu Location: Civil > 3D Data, Survey > COGO, Takeoff > Elevate

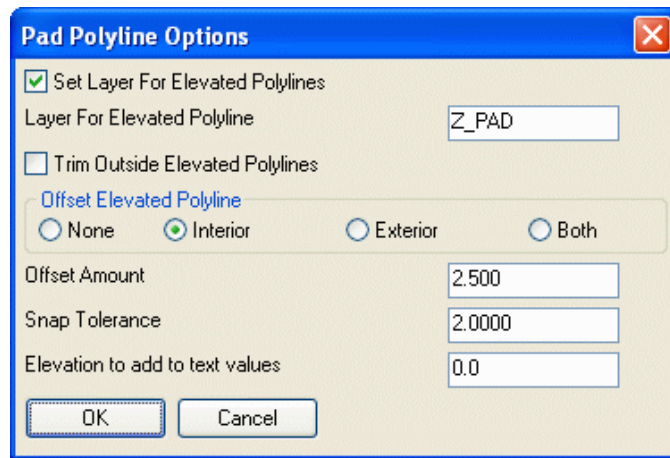
Keyboard Command: bldg_perim

Prerequisite: Building linework

Pad Polyline By Interior Text

This command allows you to set one or more pad elevations using interior text labels.

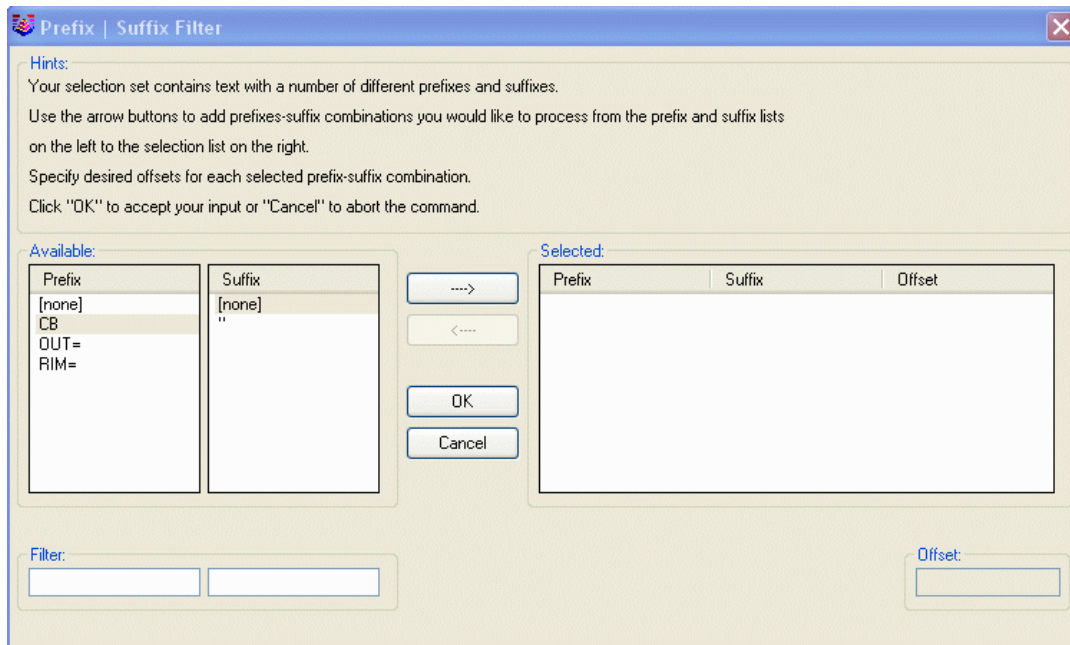
After running the command you will be prompted to select the layers you want to use for the pad elevation and for the boundary of the pad. Sometimes pads are drawn with linework from two different layers and Carlson Takeoff allows you to pick all the correct linework.



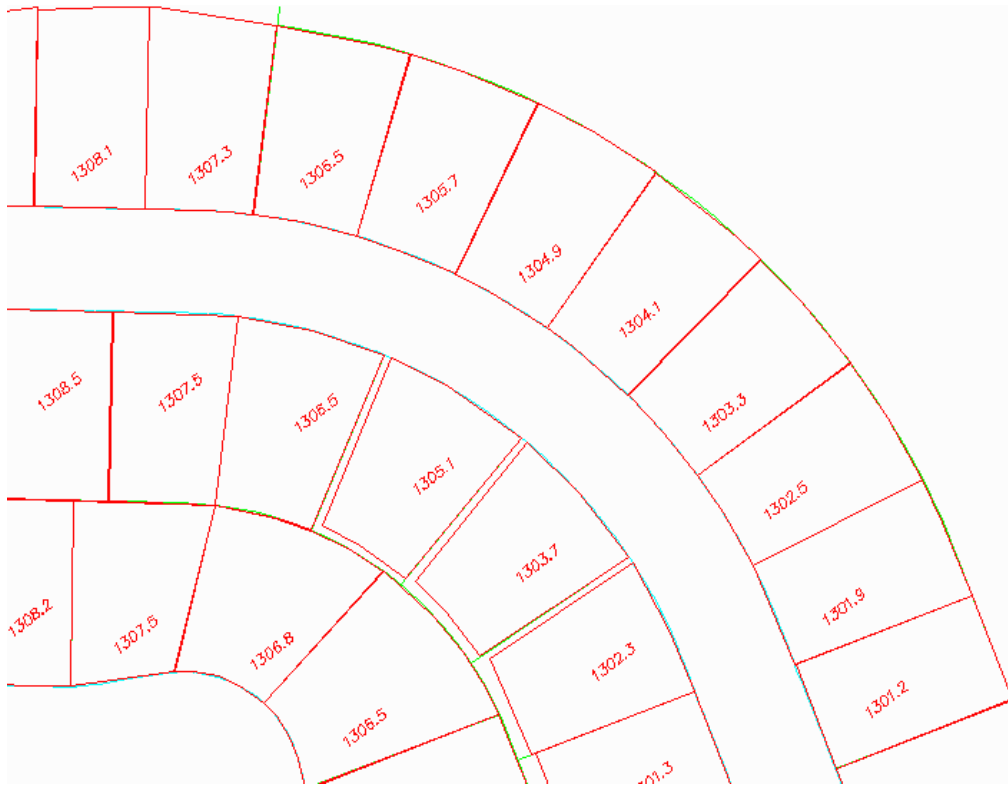
This dialog box allows you to create a new layer with the correct x,y coordinates and elevations. If the pad shares the same coordinates with other linework with different elevations than this dialog box allows you to offset the new polyline to avoid the problem of shared occupied points with different elevations. You can choose to have an interior offset or an exterior offset and also decide how much to offset the new polyline. Selecting Both will give both the interior pad elevation and the exterior contour elevations. This helps the transition from you pad elevation to the design contouring. The Snap Tolerance field joins linework which falls within the range you set to create a pad. Trim Outside Elevated Polylines will trim out contour elevations that go through your pad that you are not using elevations from within the pad.

Elevation to add to text values adds to the values from the elevation labels. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 523.5, 543.3, 537.2 sometimes they are listed as simply 23.5, 43.3, 37.2. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing.

After running the command you will be prompted to select the layers you want to use for the pad elevation and for the boundary of the pad. Sometimes pads are drawn with linework from two different layers and Carlson Takeoff allows you to pick all the correct linework. In addition, if your text has multiple Prefixes and Suffixes you will be prompted to select the ones you want to use the elevation from.



After clicking <OK> select all the pads and their elevation labels that you wish to change, press <Enter>, and the new layer with elevations will be created and placed in the Design target.



Prompts

Select layer sample of elevation text: Pick a label text

Selected text layer —TX07

Select layer sample of boundary linework:

Selected linework layer PAD

Select another layer sample of boundary linework (Enter to continue):

Select text and linework to process.

Select objects: 1 found

Select objects: 1 found, 2 total

Select objects:

Analyzing entire selection...

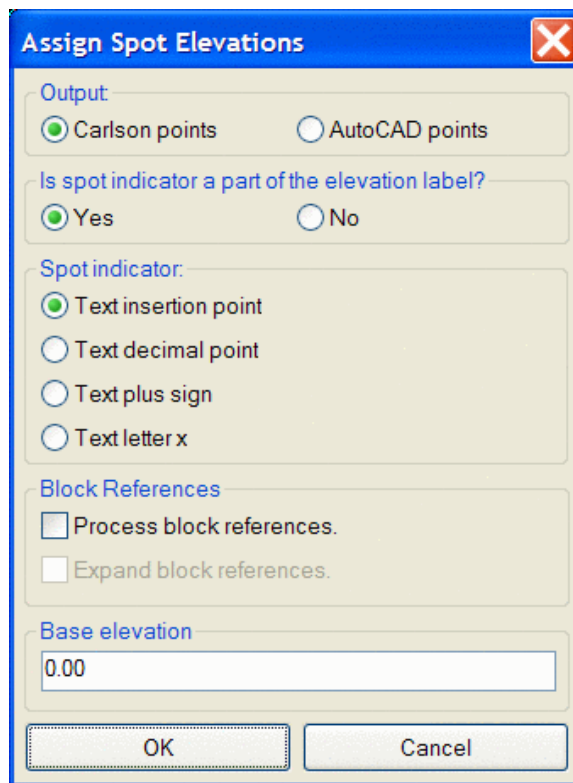
Set elevation for 1 polylines.

Keyboard Command: pad_by_text

Prerequisite: Pad polylines and elevations

Convert Spot Elev To Points

This command takes spot elevation entities with zero elevations and assigns them elevations according to corresponding elevation labels. This dialog box allows you to choose the format of the spot elevations entities that you want to convert.



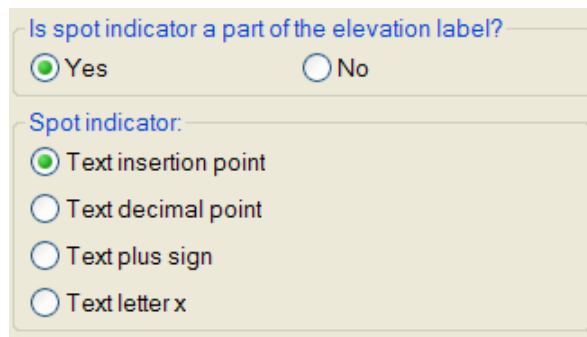
Output:

Carlson points: creates Carlson points at elevation of spot and stores them in coordinate file

AutoCAD points: creates AutoCAD point objects at elevation of spot

Is spot indicator a part of the elevation label?

If set to "Yes", four choices for **Spot indicator** are available to select from:



Text insertion point: uses the insertion point of the text for the location of the new point

Text decimal point: uses the decimal point in the text for the location of the new point

Text plus sign: uses the plus sign in the text for the location of the new point

Text letter x: uses the letter x in the text for the location of the new point

If set to "No", five choices for **Spot indicator** are available to select from:

Is spot indicator a part of the elevation label?

Yes No

Spot indicator:

Linework leader

Linework cross

Text plus sign

Text letter x

AutoCAD point

Linework leader: creates a data point at the end of a leader



Linework cross: creates a data point at the intersection of a linework cross

Text plus sign: creates a data point at the insertion point of a text plus sign

Text letter x: creates a data point at the middle of a text letter x

AutoCAD point: creates a data point at the node of an AutoCAD point

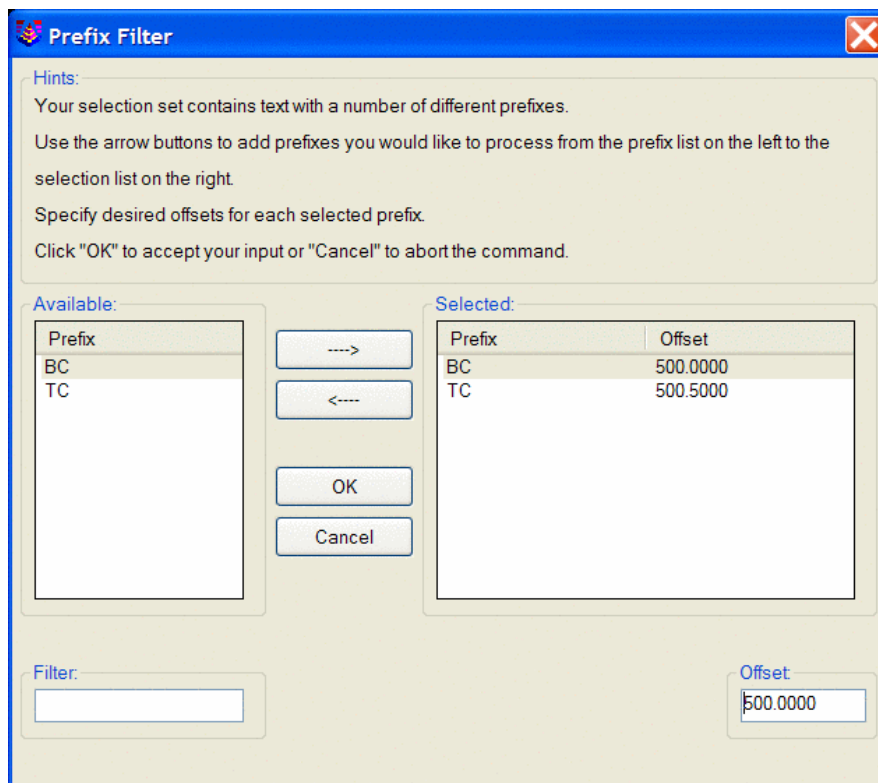
Block References:

Process Block References: If check box is cleared, Carlson Civil searches only text entities for elevations, but if checked, Carlson Civil will search block references for elevations that are stored as attributes of a block. Use this option if the elevation is an attribute and the symbol designating the location of the spot elevation are both part of the block definition.

Expand Block References: Use this option to search block references when the elevation is stored as an attribute of a block, but the symbol designating the location of the spot elevation is a different block or even other geometry that is not defined within a block.

Base elevation: The value entered here is added to the existing spot elevations for all newly created points. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 523.5, 543.3, 537.2 sometimes they are listed as simply 23.5, 43.3, 37.2. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing. Note: The base elevation will not be added to any elevations that are closer to the base elevation value than they are to 0; e.g. if a base elevation of 500 is specified, 500 will be added to elevations like 23.4, 45.5, etc, but will not be added to elevations like 456.4 or 468.9.

Prefix Filter: Carlson Civil examines all selected spot elevations for prefixes or suffixes. If they are all the same, the command proceeds, but if there are different prefixes and/or suffixes found, the Prefix Filter dialog box is invoked. This dialog box allows you to select which prefixes and/or suffixes to use to create spot elevations, and also allows you to use different offset values for each. .



Prompts

Starting point number <1>: *press Enter*

Select TEXT, MTEXT spot elevations to process and any associated leader lines:

Select objects: *pick entities to process*

Pre-processing entity #40 of 40...

Filtering text entities

Processing elevation #40...

Converted 40 spot elevations.

Keyboard Command: spotelv2

Prerequisite: Spot elevations

Assign Contour Elevation - Multiple in Series

This command can be used to quickly and accurately assign the elevation of series of AutoCAD polylines that have been converted from raster or digitized without correct elevations. The routine will automatically assign elevations to the polylines crossing the fence line selected by two points. At the same time the elevations are changed, the program can assign it a new layer, color, linetype, and polyline width. This process usually works best if contours are in a temporary (white) layer to start. When they are processed, they will take on the color of the new layers making it easy to distinguish which polylines have been processed.

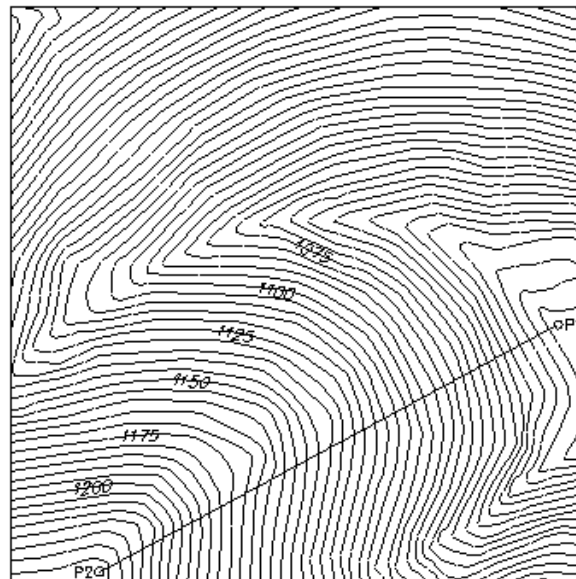
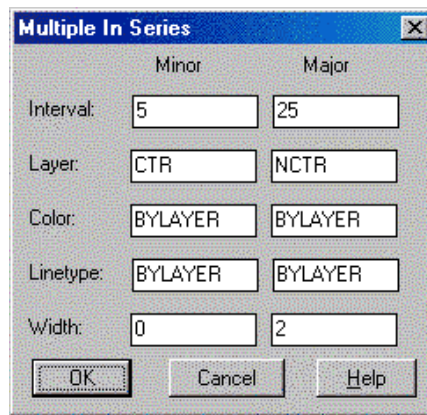
Prompts

Settings/First Point: *(Press S to change settings or pick first point.)*

Second Point: *(Pick second point)*

Beginning Elevation <0.00>: 1020

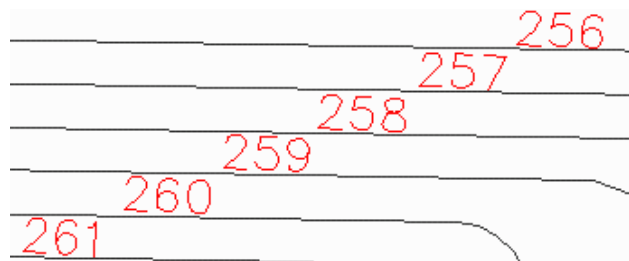
Increment Direction U/D <U>: *(enter)*



Keyboard Command: grpcelev
Prerequisite: digitized polylines

Assign Contour Elevation - From Contour Labels

This command allows you to set elevations to contours from elevation labels.



Select a sample of the elevation text to be used on the contouring. Next, select a sample of the contouring that you want to add the elevations to. In case you have contours on different layers like for major and minor contours, there is a prompt to pick a second contour line example. The program uses the Layers of the selected entities to know which entities to process as the contour labels and lines. Now select all the contours and their corresponding elevation labels. The program will then add elevations to all the contours. You may be prompted to distinguish what contour goes with what elevation label. You can either press <Enter> to accept the contour that the program has selected or you can Press <N> to choose another contour.

Prompts

Select sample of elevation text: *pick a contour label*

Select sample of a contour line: *pick a contour line*

Select second sample of a contour line (Enter for none): *press Enter*

Select contour lines and elevation text to process.

Select objects: *pick lines and labels*

Joining adjacent polylines...

Reading the selection set ...

Joining ...

Pre-processing entity #1008 of 1008

Filtering text entities

Processing elevation text #518

Conflict detected: pick contour corresponding to current elevation text

Press N for next selection or Enter to accept current:

Remaking polyline #311

Pulldown Menu Location: Elevate in Takeoff, 3D Data in Civil

Keyboard Command: TXTCELEV

Prerequisite: contours and contours labels

Assign Contour Elevation - Single Elevation Group

This command changes the elevations of polylines and can be used to set the elevations of contour polylines. The routine begins at a specified elevation and prompts for a selection set of polylines to set to the elevation. Then the routine repeats using the last elevation plus the elevation increment. Enter an empty selection set to exit the routine.

Prompts

Starting elevation <0.0>: *500.0*

Contour interval (negative for down) <1.0>: *5.0*

Select polylines to set to elevation 500.0.

Select objects: *pick the polylines*

Select polylines to set to elevation 505.0.

Select objects: *pick the polylines*

Select polylines to set to elevation 510.0.

Select objects: *Press Enter*

Keyboard Command: setcelev

Prerequisite: polylines

Drape 3D Polyline On Surface

This command converts a 2D polyline into a 3D polyline by calculating 3D polyline vertices at all the intersects of the 2D polyline with surface entities (contour polylines, triangulation lines) and by interpolating elevations from these intersections at the original vertices locations. An application for this command is to create breaklines. For example, a ridge breakline could be generated from contour lines by drawing a 2D polyline along the ridge and across the contours. Then this command could grab the contour line elevations along the polyline to make a ridge breakline.

In addition to using entities in the drawing, the 2D polyline can be converted to 3D using a surface model stored in triangulation (.flt or .tin) file. If you use a file, then you can also use the polyline's current elevation as a vertical

offset from surface.

Prompts

By Screen Entities:

Source of surface model [File/<Screen>]? Type S for Screen

Select polylines to convert.

Select objects: *select the polyline(s) to convert*

Select surface 3DFaces, lines and polylines.

Select objects: *select the surface entities (contour polylines, breaklines, triangulation lines, etc)*

Reading points ... 692

Keep existing polylines [Yes/<No>]? Press Enter

This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.

Set layer name for converted polylines [Yes/<No>]? Press Enter

This allows you to assign the new polyline to a layer.

Converting polylines ...

Converted 1 polylines.

By a .flt or .tin File:

Source of surface model [<File>/Screen]? Type F for File

Select polylines to convert.

Select objects: *select the surface entities (contour polylines, breaklines, triangulation lines, etc)*

Use current polyline elevations as vertical offset from surface [Yes/<No>]? Press Enter

This will offset the new polyline by its current elevation. That is, if a polyline has an elevation of -4 and the surface you are converting it to has an elevation of 800, then saying Yes will drape the polyline at an elevation of 796.

Keep existing polylines [Yes/<No>]? Press Enter

This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.

Set layer name for converted polylines [Yes/<No>]? Press Enter

This allows you to assign the new polyline to a layer.

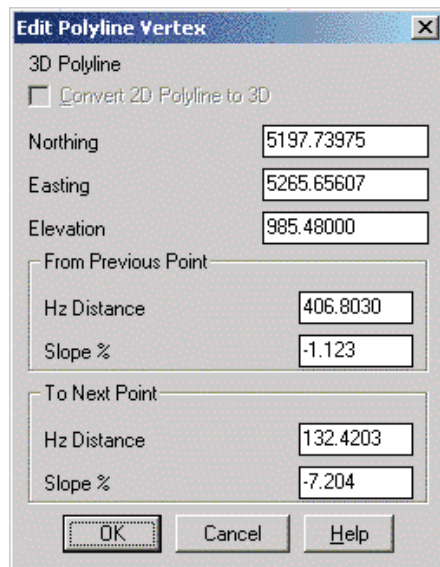
Keyboard Command: 2dto3dp

Prerequisite: A polyline and surface lines or grid file or triangulation file.

Edit Polyline Vertex

This tool allows you to make changes in the coordinates of vertices on all polyline types. Upon execution you will be asked to select a polyline to edit. Upon selection a temporary marker will be placed at all of the vertices of the polyline, making them easy to distinguish. Then pick near the vertex you wish to edit, and the following dialog appears.

At the top of the dialog it identifies the type of polyline, being 2D or 3D. In the case of 2D polylines it allows you convert the polyline. You have the ability to type in new northing, easting or elevation values. You can also determine the 3D coordinate position by using distances and slope to/from adjacent points. As you change the values in the dialog, new values for derivatives are being calculated. For example if you change the horizontal distances, the coordinates will change.



Prompts

Select polyline vertex to edit: *pick a polyline at the point to be modified*

Pick or enter position <5264.23,5048.21>: *pick a point*

Enter elevation <0.00>: *Press Enter*

Select polyline vertex to edit: *Press Enter to end*

Keyboard Command: editpl

Prerequisite: A polyline.

Edit Contours

This command revises a segment of a contour polyline. Begin by picking a point on the contour where you want to start editing. Then pick new points for the polyline. When finished picking new points, press Enter and then pick a point on the contour to connect with the new points. The polyline segment between the start and end points is then replaced with the new points.

Prompts

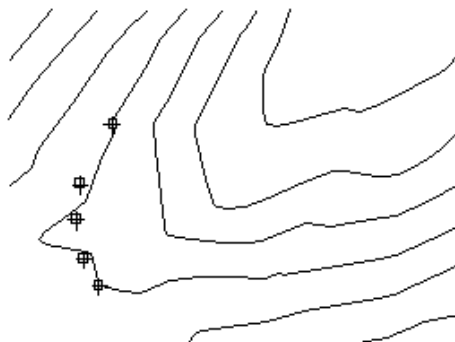
Select contour to edit: *pick the contour polyline at the place to start editing*

Pick intermediate point (Enter to End): *pick a point*

Pick intermediate point ('U' to Undo, Enter to End): *pick a point*

Pick intermediate point ('U' to Undo, Enter to End): *Press Enter*

Pick reconnection point on contour: *pick the contour polyline at the place to join*





Edit this contour by picking new points
Contour with segment replaced with new points

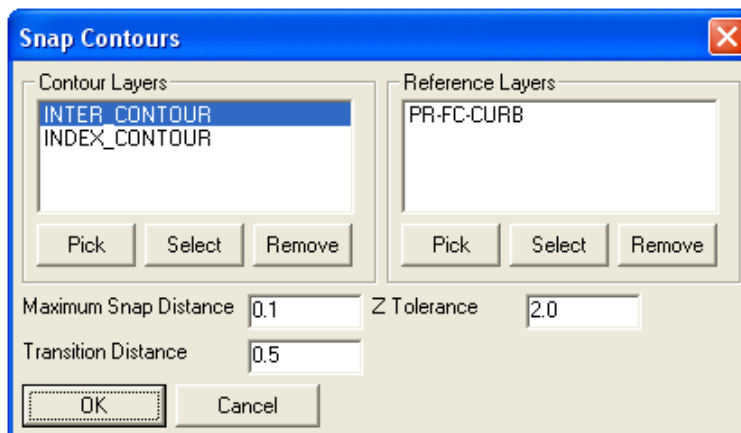
Keyboard Command: editctr

Prerequisite: polylines with elevation (contour polylines)

Snap Contours to 3D Polylines

Snap Contours to 3D Polylines can be used to align contour polylines to match elevation with intersecting of a 3D polylines. Doing so will fix spikes in a surface model. The program will ask for the Contours to be adjusted. Pick will allow you to grab the contours from the plain view, Select allows you to identify the layer(s) from a list. The layers under Contour Layers will be adjusted to match the Reference Layers at the point of intersection. A Reference layer can be identified by Pick or Select as well.

The Maximum Snap Distance is the furthest distance along the Reference line the Contour polyline will move in order to match elevations. Z Tolerance sets the minimum elevation difference between the Reference line and the Contour polyline for the program to process. Anything less than this number will not be modified. Transition Distance is the length over which the positioning change will be applied to the Contour polylines.



After selecting OK, you will be prompted for the entities to process. Pick or Window Select the linework you want to process. You can also type in "all" to select everything. Here is a standard report that is displayed on the command line:

Entities in set: 282

Select entities:

Contour polylines: 125 Processed, 12 Adjusted

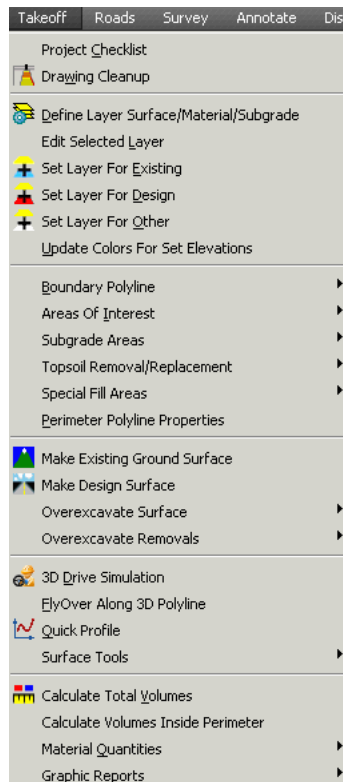
Pulldown Menu Location: Elevate (in Takeoff), 3D Data (in Civil)

Prerequisite: 3D linework

Keyboard Command: snap_cntrs

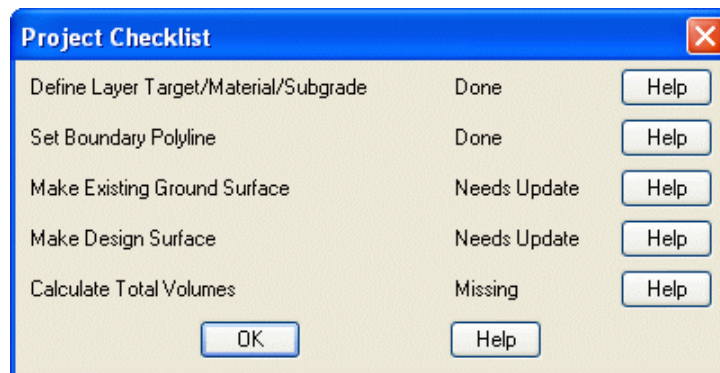
Takeoff Menu

The Takeoff menu commands are for earthwork and material quantity takeoff.



Project Checklist

This command checks the status of steps needed to calculate total volumes.



Prerequisite: None

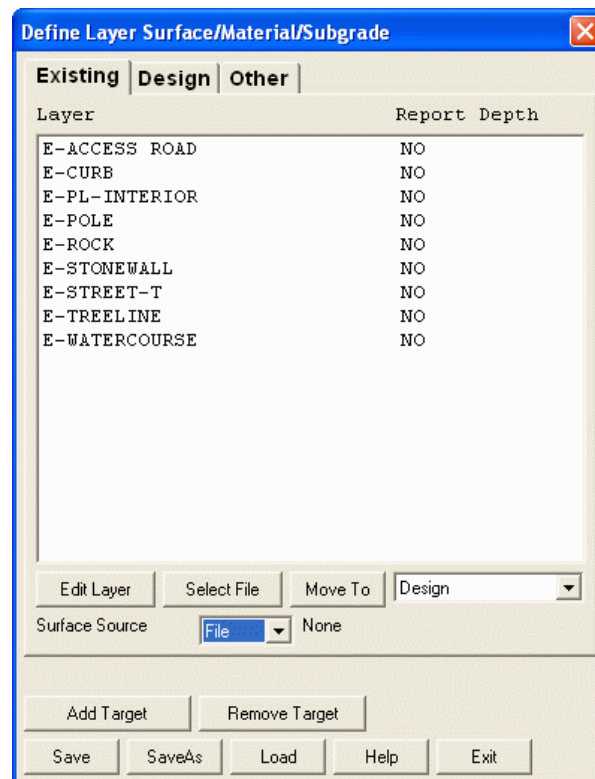
Keyboard Command: tk_checklist

Define Layer Target/Material/Subgrade

The Define Layer Surfaces dialog box (shown here) offers many functions that will ultimately make up the surface models used in volume and material calculations. Every entity (line, polyline, point, etc) in a drawing is assigned a layer name. Carlson Takeoff uses the entity layer names to define which entities represent the existing ground surface, the design surface or no surface. These surfaces are referred to as the "Target" surfaces. Any

previously created triangulation file (.tin) can be set to the design or existing Target with the Surface Source drop-down set to File. In this mode, the Select File button will allow you to pick the .tin file you want to use for the Target.

When the Surface Source drop-down is set to Layers, drawing entities are assigned to target surface by their layer name. For example, if polylines representing design contours are on the layer "Final", then "Final" will be set as a layer for the design surface. For layers of entities that are for neither existing nor design surfaces (such as text labels for street names), the layer target is set to Other. The Define Layer Surfaces dialog has three lists for layer targets: Existing, Design and Other. To switch between lists, pick the tabs at the top of the dialog. To move a layer to a target destination, highlight the desired layer, choose the target from the Move To list and pick the "Move To" button. All layers populate the "Other" target before being assigned to "Existing" or "Design".

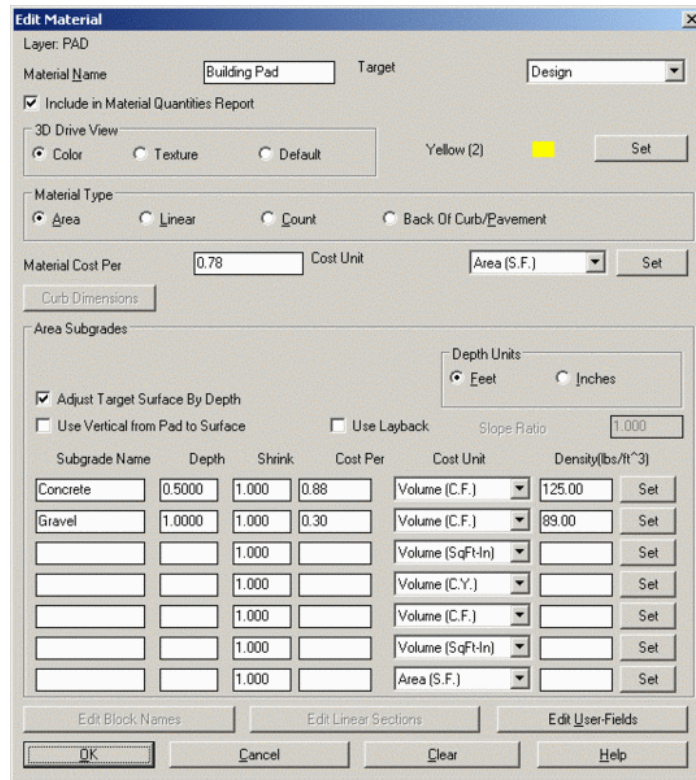


Besides the basic three layer targets (Existing, Design and Other), you can add more target surfaces with the Add Target button. When another target is defined, there will be another tab along the top of the Define Layer Surfaces dialog. Then layers can be assigned to this additional target surface. The only pre-defined additional surface is Overexcavate. The layers that are assigned to the Overexcavate target can be modeled into the Overexcavate surface using the Make Overexcavate Surface command. Besides Overexcavate, the other additional targets are user-defined. The layer targets can be modeled using the Make User-Defined Surface command. Then these surfaces can be used in Takeoff commands by assigning them to a Takeoff existing or design surface using the Set Active Surfaces command.

Edit Materials

The "Edit" button activates the Edit Material dialog box (shown here) and allows you to define the Material name and Subgrade depths and names. Besides assigning target surfaces by layer, layers are also used to define material names and subgrades depths. By assigning a material name, Subgrade names and depths to layers, the volume, area, length and count for entities on these layers can be reported. Also the depth is used to vertically adjust the design surface, or tie into the design surface by a Slope Ratio if "Use Layback" is checked on. For Area and Back Of Curb/Pavement material types, the polylines on the layer used for a Material must be closed polylines. Carlson

Takeoff supports nested Subgrade polylines for exclusion areas such as islands by counting how many Subgrade polylines surround an area. If the number is odd, then the area is included in the Subgrade. The even count regions in the area are not part of the Subgrade. To activate the Edit Material, select a layer from the list and then choose "Edit".



Include in Material Quantities Report

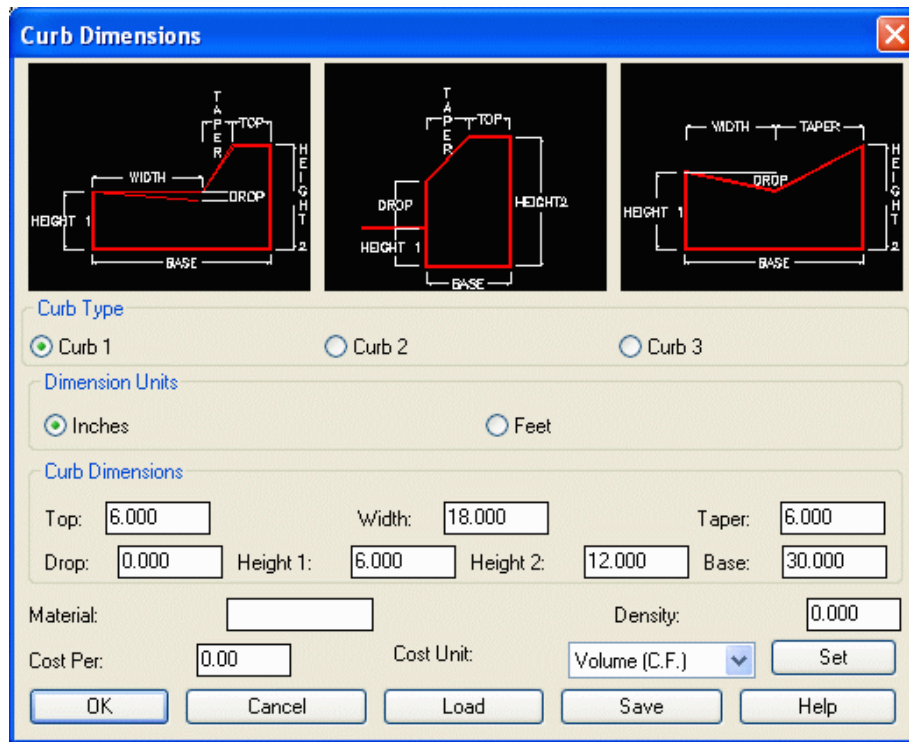
With this option checked on, the material that is named will appear in the Material Quantities Report. The report will include either the area of the material, the linear length of the material, or the number of items counted on the layer defining the material. This is accomplished by choosing "Area", "Linear", or "Count" for the Material Type.

3D Drive View

This option allows you to assign a color or texture for this particular material for display purposes during the 3D view/drive simulator. The color is assigned to the design surface TIN file.

Material Type

This will report the subgrade by area, linear length, count, or as curb/pavement area. If you choose Back of Curb/Pavement then you can pick on the Curb Dimensions button and bring you to the below dialog:



With the Back of Curb/Pavement, the 3D polylines represent the back of curb elevations. When using this method, the curb polylines alone define the pavement areas and no other design entities (ie. design contours or spot elevations) should be in the pavement area. The program will adjust the design surface for the height of the curb above ground to get the elevations to the top of pavement. Also with Curb/Pavement, the program will calculate your curb volume as well as act as the limit of the pavement. For the pavement areas bounded by the curb polylines, the program will apply the subgrade depths defined separately from the Curb Dimensions. These pavement subgrade depths are defined in the Area Subgrades section of the Edit Material dialog. The pavement limit will be from the Back of Curb polyline offset by the length of the Curb base. In the above case the base is 30 inches wide. Therefore, the pavement area will stop 30 inches before the Back of Curb polyline.

Material Cost Per Cost Unit

Use this field to add the value of the multiplier for the unit cost of your material. If the material type is an area that has multiple subgrades, use the available fields below to add each individual subgrade name, depth and cost value per unit type. If a linear or count type material type option is selected, use the "length in feet", or the "count" unit options.

Adjust Design Surface by Depth

This determines whether the subgrade depths are incorporated in the design surface or not.

Use Vertical from Pad to Surface

This will interpolate the surface model out to your layer and then vertically adjust the model to tie into the layer. With this checked off, the program will directly interpolate a surface model between your layer and the elevated entities around it.

Output Solid

This creates a solid model for area type materials using the depth for the thickness of the solid. The solid

model is saved in an .MDL file that is named automatically using the drawing name.

Area Subgrades

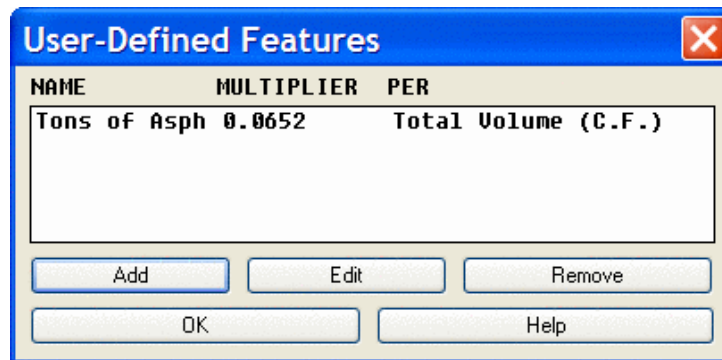
Depth Units

Select the "feet" or "inches" as the unit value desired for depth of subgrades.

Subgrade Name Depth Shrink Cost Per Cost Unit Density

Use these options for areas that are represented with a single/multiple closed polygon/polygons in the drawing, but have multiple material types defining the surface. Simply name each "lift" in the area, issue a depth value and add a cost unit if desired, or click on select and choose a material from the Materials Library (see Define Materials Library for more). Carlson Takeoff will report each subgrade material value in the material quantities report. The Shrink factor is multiplied by the subgrade volume in the material quantities report and represents the fill shrinkage. A Density factor can be entered in when using Cost Per Tons.

If user entered values are needed in the report use the "Edit User-Fields" button to activate the "User Defined Features" dialog box shown here. Choose the "Add" button to define needed fields such as TONS of material or BAGS OF GRASS SEED for reporting options.



Once all of the material subgrades, depths and cost units or user defined units have been defined, select save to preserve the settings in a .trg file, the "save as" function allows the user to name the file to load later.

Prerequisite: None

Keyboard Command: define_tk_layers

Edit Selected Layer

Use this command to click on any layer and advance to the Edit Materials dialog from the Define Layer Target/Material/Subgrade command.

Prerequisite: none

Keyboard Command: edit_tk_layer

Set Layer For Existing

Set Layer For Existing allows the user to pick the layers from objects on the screen and assign them to the Existing Layer.

Prerequisite: none

Keyboard Command: set_existing_layer

Set Layer For Design

Set Layer For Design allows the user to pick the layers from objects on the screen and assign them to the Design Layer.

Prerequisite: none

Keyboard Command: set_design_layer

Set Layer For Other

Set Layer For Other allows the user to pick the layers from objects on the screen and assign them to the Other Layer.

Prerequisite: none

Keyboard Command: set_other_layer

Update Colors For Set Elevations

This command refreshes the color of entities depending on their elevation and layer target. For entities assigned to the Existing or Design layer targets, if the entities are at zero elevation then their color is set to grey. Otherwise the entities have their true, original color. If the Automatic Update Colors command under Settings->Configure->Takeoff Module is toggled off, then this command is the way to update the entity colors after editing elevations.

Prerequisites: none

Keyboard Command: update_tk_colors

Boundary Polyline

The Boundary Polyline options allow the user to Set the Boundary Polyline, Set the Exclusion Polylines, Clear Exclusion Polylines, Hatch the Boundary Area, Erase the Boundary Hatched area.

Set Boundary Polyline

Use this command to select the "CLOSED" polyline that defines the outer most limit of the disturbed area. This boundary should dissect the site at the point where the design contours meet the existing contours, or where the limit of work will occur. If your site contains separated areas (such as different phases or isolated sections of work), then multiple Boundary Polylines can be used. Volume calculation will take place inside this boundary.

Prerequisite: a closed polyline

Keyboard Command: tag_inclu

Set Exclusion Polylines

Use this command to select the "CLOSED" polylines that define the areas inside the Boundary Polyline that will not be disturbed. These boundaries should also be at the intersection of the proposed and existing surface. A pond or wetland that will not be removed during construction is a good example of an Exclusion Area.

Prerequisite: a closed polyline

Keyboard Command: tag_exclu

Clear Exclusion Polylines

Use this command to select polylines that were previously defined as exclusion polylines but are no longer needed as exclusion areas.

Prerequisite: exclusion polylines

Keyboard Command: untag_exclu

Highlight Boundary Polylines

This command highlights the polyline you set as the Boundary Polyline.

Prerequisite: a boundary polyline

Keyboard Command: highlight_boundary

Hatch Boundary Area

Use this command to confirm the boundary polylines that have been selected are correct. This hatched area can also be utilized in exhibits of the drawing.

Prerequisite: a boundary polyline

Keyboard Command: hatch_boundary

Erase Boundary Hatch

This command erases the hatch drawn in the plan view.

Prerequisite: a boundary hatch

Keyboard Command: erase_boundary

Areas Of Interest

Areas of Interest can be used to calculate volumes and material quantities within a specified area. The Area Of Interest perimeters are defined by user-selected closed polylines and each area is assigned a name. The Area Of Interest polylines can be assigned either as inclusion or exclusion perimeters for the area. You can have any number of exclusion perimeters within an inclusion but inclusion perimeters cannot be inside exclusions. The same area name can be used with multiple inclusion polylines to combine the quantities from those polylines in the report.

The Areas Of Interest (AOI) commands allow you to Tag/Untag Areas of Interest, Identify/Report Areas of Interest and Hatch/Label Areas of Interest.

Tag Area Of Interest

This command allows the user to select polylines and exclusion perimeters that define phases of a project. Carlson Takeoff will separate each area of interest in the volume and material reports.

Prerequisite: a closed polyline

Keyboard Command: tag_aoi

Area Of Interest by Interior Text

This command allows the user to select text from the screen to name AOIs and linework to determine the area.

Prerequisite: area linework and text

Keyboard Command: txt2aoi

Untag Area Of Interest

This command allows the user to remove previously tagged areas.

Prerequisite: an area of interest

Keyboard Command: untag_aoi

Identify Area Of Interest

This command allows users to identify AOI by either picking on a polyline(s) or by searching the entire drawing. The command will then report the AOI name, layer, type, starting point, and highlight the polyline in the plan view.

Prerequisite: an area of interest

Keyboard Command: id_aoi

Report Area Of Interest Areas

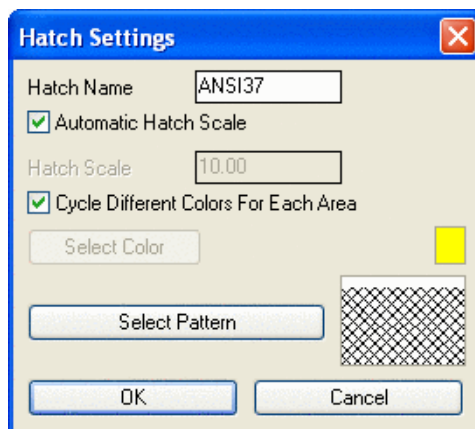
Use this command to report the Inclusion or Exclusion area (SF), the name, the layer, and the starting point.

Prerequisite: an area of interest

Keyboard Command: report_aoi

Hatch Area Of Interest Areas

This command allows the user to visually see AOIs in the plain view.



This command draws a hatch with a specified color and pattern for the Areas of Interest. The purpose is to allow you to visually review AOIs to make sure that the area coverage is correct.

The command displays a dialog for the hatch pattern, color and scale. The scale determines how spread out the pattern is within the hatch. The Automatic Hatch Scale option checks the size of the subgrade areas and sets the scale to make the pattern fit. Cycle Different Colors For Each Area will give each AOI it's own color so that you can distinguish between different AOIs.

The resulting hatch areas show where the AOI is applied. Exclusion Areas of AOIs will not be hatched.

Prerequisite: an area of interest

Keyboard Command: hatch_aoi

Erase Area Of Interest Hatch

This command erases AOI hatching.

Prerequisite: hatched area of interest

Keyboard Command: erase_aoi_hatch

Label Area Of Interest Areas

This command labels the AOI name and area in the plain view.

Prerequisite: an area of interest

Keyboard Command: label_aoi

Erase Area Of Interest Labels

This command erases AOI labeling.

Keyboard Command: erase_aoi_labels

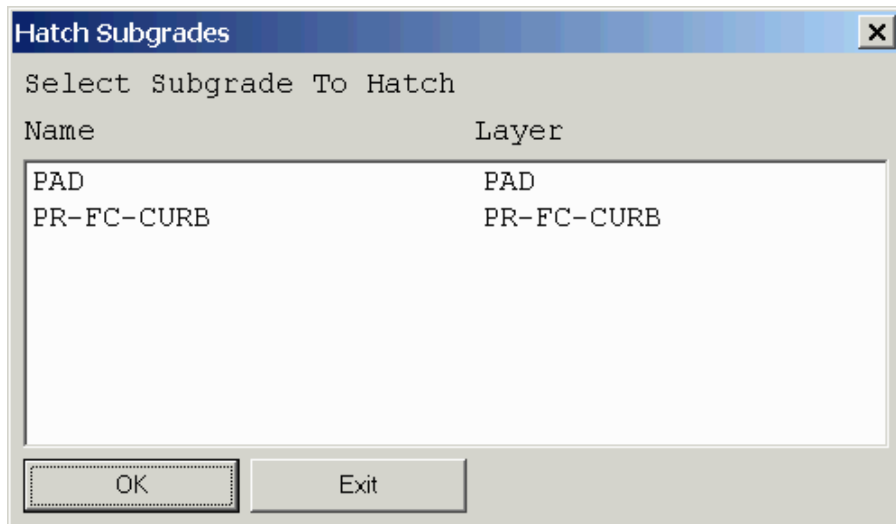
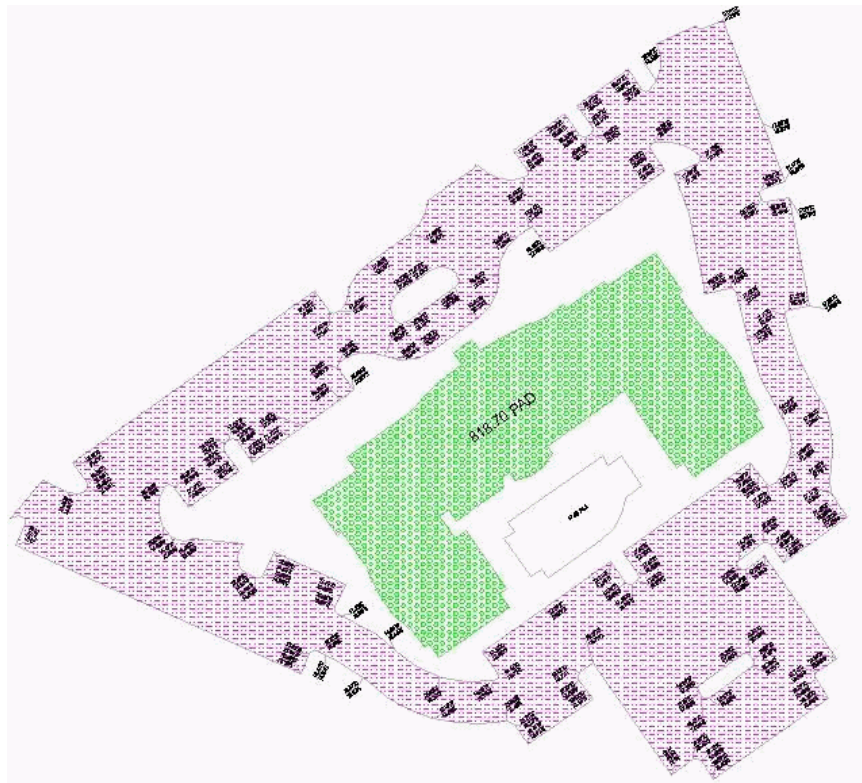
Prerequisite: hatched area of interest

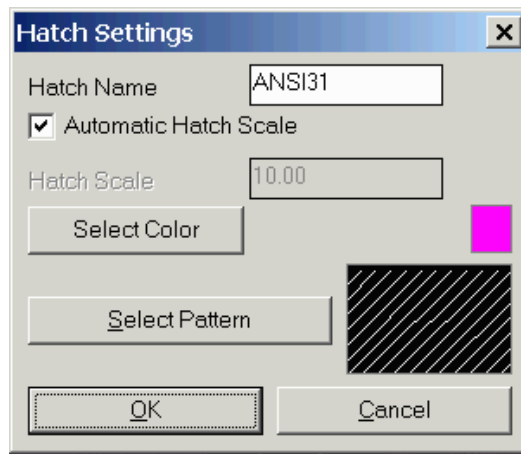
Hatch Subgrade Areas

This command draws a hatch with a specified color and pattern for the area that the selected subgrade area applies to. The purpose is to allow you to visually review a subgrade area to make sure that the area coverage is correct.

The command displays a dialog to select which subgrade to hatch. The list of available subgrades comes from the layers with subgrade depths as set in the Define Layer Target/Material/Subgrade command. Then there is a dialog for the hatch pattern, color and scale. The scale determines how spread out the pattern is within the hatch. The Automatic Hatch Scale option checks the size of the subgrade areas and sets the scale to make the pattern fit. The hatch settings are stored separately for each subgrade along with the Define Layer Target/Material/Subgrade command settings in the drawing TRG file.

The resulting hatch areas show where the subgrade is applied. In the example below, notice how the islands are not hatched because they are curb polylines that are already inside another curb polyline. Also note that the smaller pad area is not hatched because this polyline layer is different than the bigger pad polyline.





Pulldown Menu Location: Takeoff > Subgrade Areas

Keyboard Command: hatch_subgrade

Prerequisite: subgrades

Erase Subgrade Hatches

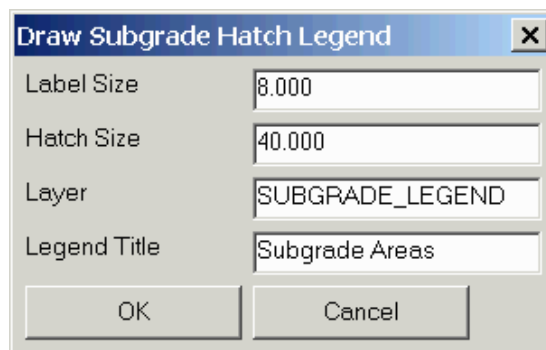
This command removes from the screen the subgrade hatches created by the command Hatch Subgrade Area.

Keyboard Command: erase_subgrade

Prerequisite: hatch subgrade areas

Draw Subgrade Hatch Legend

This command draws a legend for the subgrade areas currently in the drawing. The legend includes the subgrade names and squares of the hatch patterns. The size of the labels, size of the hatch squares, layer for the legend entities and the legend title are set in the dialog shown below. The subgrade hatches to include in the legend are automatically selected from all the subgrade hatches currently in the drawing that were created by the Hatch Subgrade Areas command.



Pulldown Menu Location: Takeoff > Subgrade Areas

Keyboard Command: draw_subgrade_legend

Prerequisite: hatched subgrade areas

Report Subgrade Areas

This command reports all the subgrade areas in the drawing. For each subgrade polyline, the report includes the layer name, subgrade depth, area and polyline starting point.

Sample Report:

Layer Depth Area Starting Point

PAD 1.500 21979.7 6135018.84,2190093.71

CURB 1.000 50420.2 6134994.81,2190125.80

CURB 1.000 114507.3 6135191.33,2190335.27

Pull-Down Menu Location: Takeoff > Subgrade Areas

Keyboard Command: report_subgrade

Prerequisite: Subgrade Areas

Label Subgrade Areas

This command lets you label the subgrade depth and area (in sq. ft. or meters). The label is placed at the center of the subgrade area, but can be moved with the Move command under Edit.

Pull-Down Menu Location: Takeoff > Subgrade Areas

Keyboard Command: label_subgrade

Prerequisite: Subgrade Areas

Erase Subgrade Labels

This command erases subgrade labels.

Pull-Down Menu Location: Inquiry-> Subgrade Areas

Keyboard Command: erase_subgrade_labels

Prerequisite: subgrade labels

Topsoil Removal and Replacement

The Topsoil Removal and Replacement options (shown here) allow the user to Define Topsoil removal and replacement depths, Set topsoil removal and replacement areas by selecting closed polylines, Clear the selected boundary polylines if needed, Hatch the topsoil removal and replacement areas and Erase the hatched areas.



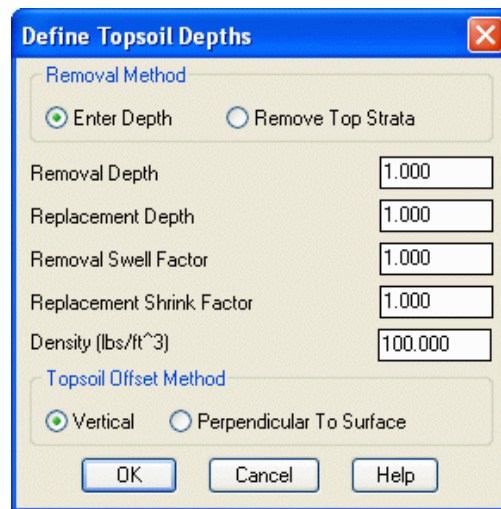
Define Topsoil Depths

This command requires user input to define the depth, or strata, of topsoil removal and replacement. Fill in the options available in the Define Topsoil Depths dialog (shown here). Carlson Takeoff will perform four functions with these values. First, the value set for the Removal Depth, or the Top Strata if selected, will be the "defined" removal amount from the Existing Ground Surface. Second, the calculated volume of topsoil removed will be included in the reporting options. Third, the value set for the Replacement Depth will be added "BELOW" the Finished Ground Surface model. Fourth, the amount of topsoil replaced will be included in the reporting options.

When topsoil depths are defined, the volume report routines will include the topsoil quantities. These topsoil quantities are in addition to the cut/fill for the existing to design surfaces for the site.

The Removal Swell Factor and Replacement Shrink Factor are multiplied by the topsoil removal and replacement quantities respectively in the volume report routines. The Density is used to report topsoil tons when the volume report option for tons is active.

The Topsoil Offset Method choose between offsetting the topsoil depth vertically or perpendicular to the surface. The perpendicular method will result in more topsoil quantities since it represents applying the topsoil depth to the slope area of the surface whereas the vertical method represents applying the topsoil depth to the horizontal area.



Prerequisite: topsoil depths

Keyboard Command: define_topsoil

Identify Topsoil Polylines

This command allows users to identify topsoil polylines by either picking on a polyline(s) or by searching the entire drawing. The command will then report the layer name and starting point for both removal and replacement polylines. These polylines are also highlighted in the plain view.

Prerequisite: topsoil polylines

Keyboard Command: id_topsoil

Report Topsoil Areas

Use this command to report the Inclusion or Exclusion area (SF), the type, the depth, the layer, and the starting point.

Prerequisite: topsoil areas

Keyboard Command: report_topsoil

Label Topsoil Areas

This command labels the topsoil type and area in the plain view.

Prerequisite: topsoil area

Keyboard Command: label_topsoil

Erase Topsoil Labels

This command erases topsoil labeling.

Prerequisite: hatched topsoil

Keyboard Command: erase_topsoil_labels

Set Topsoil Removal Polygons

Use this command to select the "CLOSED" polyline boundary defining the extents of topsoil removal and any "CLOSED" interior polylines that define the topsoil removal area. The layer names for these boundaries is irrelevant. You will be prompted to use the Removal Depth defined in the Define Topsoil Depths command or to customize your depth.

Prerequisite: polylines for removal

Keyboard Command: tag_topsoil_remove

Clear Topsoil Removal Polygons

This command allows the user to remove and previously selected Topsoil Removal Polygon boundaries.

Prerequisite: topsoil polygons

Keyboard Command: untag_topsoil_remove

Hatch Topsoil Removal Area

Use this command to display a hatch pattern over the entire area designated for topsoil removal.

Prerequisite: topsoil areas

Keyboard Command: hatch_topsoil_remove

Erase Topsoil Removal Hatch

Use this command to remove the hatch pattern that defined the topsoil removal area.

Prerequisite: hatched topsoil

Keyboard Command: erase_topsoil_remove

Set Topsoil Replacement Polylines

Use this command to select the "CLOSED" polyline boundary defining the extents of topsoil replacement, and any "CLOSED" interior polylines that define the topsoil replacement. The layer names for these boundaries is irrelevant. You will be prompted to use the Topsoil Replacement amount defined in the Define Topsoil Depths command or to customize your amount.

Prerequisite: polylines for replacement

Keyboard Command: tag_topsoil_replace

Clear Topsoil Replacement Polylines

This command allows the user to remove and previously selected Topsoil Replacement Polyline boundaries.

Prerequisite: topsoil polylines

Keyboard Command: untag_topsoil_replace

Hatch Topsoil Replacement Area

Use this command to display a hatch pattern over the entire area designated for topsoil replacement.

Prerequisite: topsoil areas

Keyboard Command: hatch_topsoil_replace

Erase Topsoil Replacement Hatch

Use this command to remove the hatch pattern that defined the topsoil replacement area.

Prerequisite: hatched topsoil

Keyboard Command: erase_topsoil_replace

Special Fill Areas

Special Fill Areas can be used to identify areas to report fill separately. This can be used for areas where a different type of fill is needed. Such as under a building pad. The Special Fill Areas perimeters are defined by user-selected closed polylines. Carlson Takeoff will separate the special fill volume within the Calculate Total Volume Report.

Tag Special Fill Area

This command allows the user to select perimeter polylines that define special fill areas. Note: The inclusion and exclusion polylines are selected at the same time. The polyline to the inside will be used as an exclusion polyline.

Untag Special Fill Area

This command allows the user to remove previously tagged Special Fill areas.

Identify Special Fill Area

This command allows the user to identify Special Fill Areas by either picking on a polyline(s) or by searching the entire drawing. The command line report the layer, starting point, and highlight the polyline(s) in the plan view.

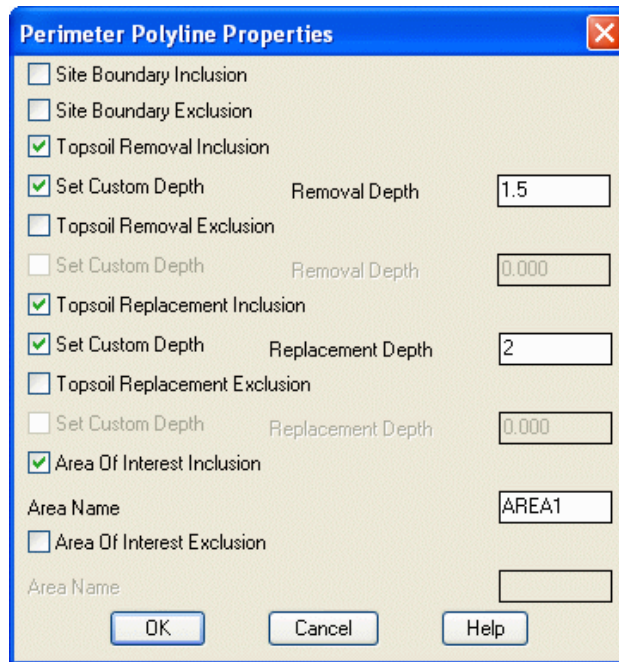
Pulldown Menu Location: Takeoff

Prerequisite: Closed polylines that represent Special Fill Areas

Keyboard Command: tag_special_area, untag_special_area, id_special_area

Perimeter Polylines Properties

This command allows you to control the properties of any perimeter polyline (Note: Perimeter polylines also have to be closed polylines). Select a polyline and the following dialog appears. Here you can define the functionality of the polyline in regards to, the Site Boundary, Areas of Interest, and Topsoil Removal/Replacement. These properties can also be set separately using the Boundary Polyline, Areas of Interest, and Topsoil Removal/Replacement commands found under the Tools menu of Carlson Takeoff.

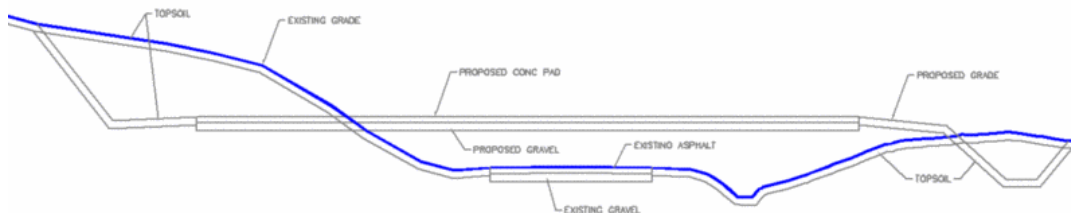


Prerequisite: a polyline

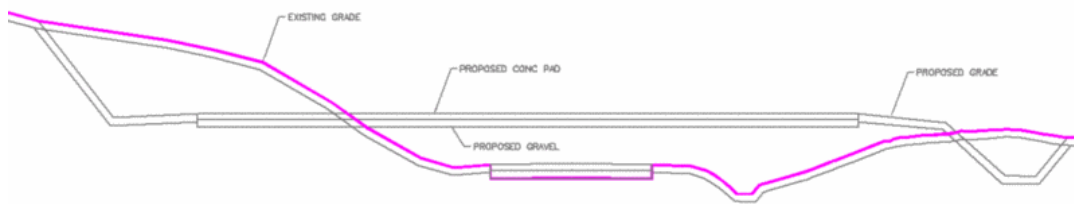
Keyboard Command: perim_prop

Make Existing Ground Surface

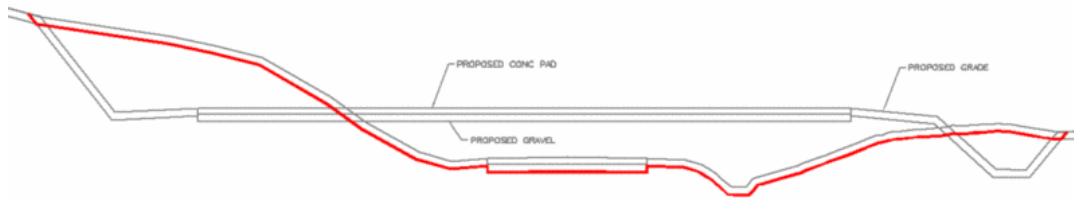
This command makes the triangulation models for the existing ground surface. There are three surfaces that are created: initial original ground (og), original ground after applying subgrade zones (ze), and original ground after subgrade zones and topsoil removal (ex). These surface files are automatically named as "filename-og.tin", "filename-ze.tin" and "filename-ex.tin" respectively. The "filename" is set to the name of the current drawing (dwg) file. Also, the file extension will be .tin for the binary format triangulation and .flt for the ASCII format triangulation. This file format is set in Configure->Takeoff.



Original Ground (OG Surface)



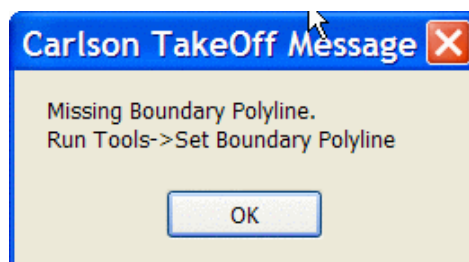
Existing Ground after subgrade zone removal (ZE Surface)



Existing Ground after subgrade and topsoil removal (EX Surface)

The surface is built using 3D entities in the drawing on the layers define in Define Layer Target/Material/Subgrade command. Also, the surface elevation for any drillholes are used for the model. The subgrade zones are defined in the Define Layer Target/Material/Subgrade command. If there aren't any subgrade zones for the Existing surface, then the original ground after subgrades surface with be the same as the initial original ground surface. The topsoil removal depths and areas are set with the commands in the Topsoil Removal/Replacement sub-menu. The topsoil removal areas will lower the ground surface by the topsoil depth. If there aren't any topsoil removal areas, then the original ground after subgrade and topsoil surface will be the same as the original ground after subgrade surface.

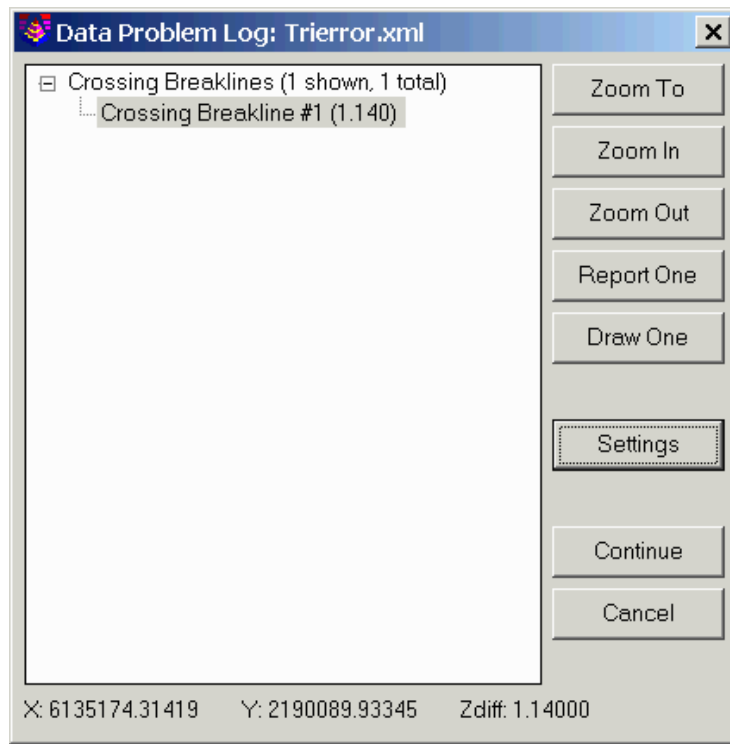
Before running this command, the layer names for the entities on the Existing layer target must be set in the Define Layer Target/Material/Subgrade command. Also these entities must be at their proper elevations. The entity elevations can be reviewed using commands from the Inquiry menu and the elevations can be assigned if needed using command from the Elevate menu. Another prerequisite is that the Boundary Polyline must be set for the site. If the boundary has not been set, the following error message will appear.



If this error message appears, run the "Set Boundary Polyline" command and pick the CLOSED polyline representing the boundary of the site.

When the program finds errors in the existing entities, a Data Error Log dialog reports these errors. Three types of conflicts are reported: Crossing Breaklines, Vertical Edges, and Breakline T-Intersections. Crossing Breaklines indicates that the intersection of two entities does not have a common elevation. Vertical Edges indicates that two entities or vertices of differing elevations have the same x-y location, thus forming a vertical plane. Breakline T-Intersections indicates that a 3d entity is abutting another entity, but the second entity doesn't have a vertex at the point of intersection. Each type of conflict is listed in its own category.

The Data Error Log shows the amount of elevation difference at each error. You can use the Data Error Log to review, report and draw markers at these error locations. Then you can exit the Data Error Log and fix the data errors with the commands in the Elevate menu or other drafting tools. After these errors are fixed, you can run Make Existing Ground Surface again.



Clicking to the "plus" sign beside a category will display the individual conflicts within that category. When a line item error is selected, a highlighted arrow is temporarily placed in the drawing to indicate the exact location of the specific conflict. Zoom functionality allows the user to more closely inspect the specific problem area, and if needed a marker can be drawn or a report generated for an individual conflict or conflicts.

Zoom To pans the drawing to move the selected conflict to the center of the screen. The zoom functions are only active when a single line item is selected.

Zoom In zooms in on the highlighted area for closer inspection. Multiple picks on the zoom button will increase the magnification.

Zoom Out zooms out away from the highlighted area.

Report All/One toggles between One and All depending whether a single line item conflict or a category is selected from the error log. An error report is generated listing the x-y position and the elevation difference of the entities in conflict.

Draw All/One toggles between One and All depending whether a single conflict or a category is selected from the list. This option draws an "X" symbol at each selected conflict. The layer and size of the symbol is controlled in the fields below.

Continue closes the Error Log and proceeds with the contouring operation.

Settings has controls for the tolerances for error reporting and for the Layer Name and Symbol Size to use with the Draw function.

Keyboard Command: mk_exist_tin

Prerequisite: a boundary polyline and elevated entities on the Existing layer target

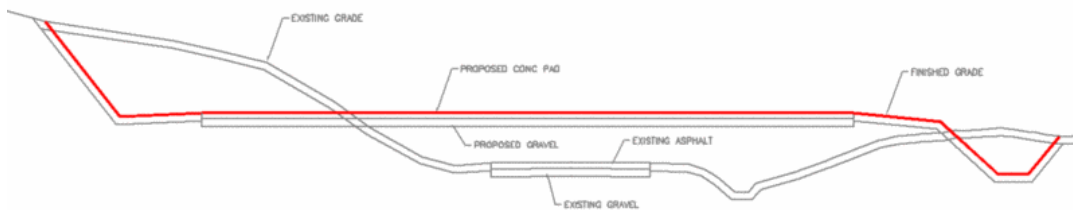
Make Design Surface

This command makes the triangulation models for the design surface. There are three surfaces that are created: initial unadjusted design/finished grade (fg), design after applying subgrade zones (zn), and design after subgrade zones and topsoil replacement (sg). These surface files are automatically named as "filename-fg.tin", "filename-zn.tin" and "filename-sg.tin" respectively. The "filename" is set to the name of the current drawing (dwg) file. Also,

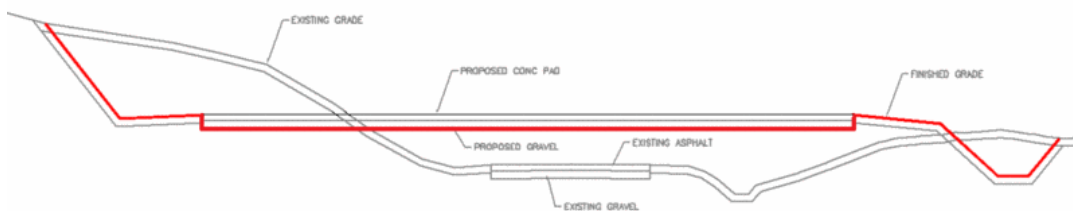
the file extension will be .tin for the binary format triangulation and .flt for the ASCII format triangulation. This file format is set in Configure->Takeoff. The subgrade zones are defined in the Define Layer Target/Material/Subgrade command. If there aren't any subgrade zones for the Design surface, then the design after subgrades surface will be the same as the initial design surface. The topsoil replacement depths and areas are set with the commands in the Topsoil Removal/Replacement sub-menu. The topsoil replacement areas will lower the design surface by the topsoil depth to leave room for the topsoil replacement. If there aren't any topsoil replacement areas, then the design after subgrade and topsoil surface will be the same as the design after subgrade surface.

Before running this command, the layer names for the entities on the Design layer target must be set in the Define Layer Target/Material/Subgrade command. Also these entities must be at their proper elevations. The entity elevations can be reviewed using commands from the Inquiry menu and the elevations can be assigned if needed using command from the Elevate menu. Another prerequisite is that the Boundary Polyline must be set for the site.

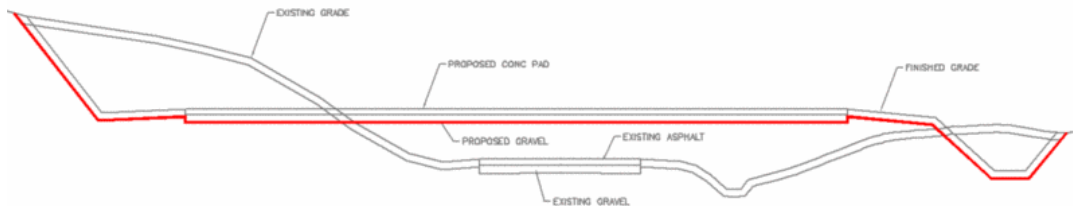
When the program finds errors in the existing entities, a Data Error Log dialog reports these errors. Refer to the Make Existing Surface command for more information on the Data Error Log dialog.



Unadjusted Design/Finished Grade (FG surface)



Design after subgrades (ZN surface)



Final Design after subgrades and topsoil (SG surface)

Keyboard Command: mk_final.tin

Prerequisite: a boundary polyline and elevated entities on the Existing layer target

View Overexcavate Surface

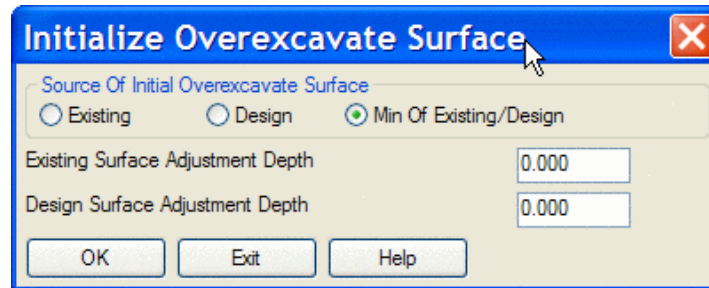
Use this command to view the current overexcavate surface. The Takeoff 3D Viewer will display the 3D faces of the adjusted surface. Shade the 3D model and adjust its perspective to view a rendered display. The surface that is displayed will depend on the latest surface created using the make and adjust routines.

Prerequisite: an overexcavate surface

Keyboard Command: cube_overx

Make Overexcavate Surface From Existing/Design Surfaces

The Initialize Overexcavation Surface dialog box shown here allows the user to select which surface model to overexcavate and to enter in the depth value for the desired adjustment. Use the Min Existing/Design option to set the overexcavate as the minimum of the existing and design surfaces. If a single surface is selected the value entered will be applied to that surface only. The resulting surface of Make Overexcavate Surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.



Prerequisite: Existing and/or Design surfaces

Keyboard Command: set_overx

Make Overexcavate Surface From Surface File

This command sets the overexcavate surface to a selected TIN surface file by copying the TIN file to the overexcavate surface. The overexcavate surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.

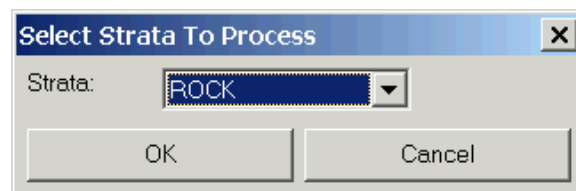
Pulldown Menu Location: Takeoff > Overexcavate Surface

Keyboard Command: overx_by_file

Prerequisite: .tin file

Make Overexcavate Surface From Strata

This command sets the Overexcavate surface to a selected strata surface. Before running this command, the strata surface must be created with the Make Strata Surfaces command in the Drillhole menu. The resulting overexcavate surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.



Prerequisite: Strata surfaces

Keyboard Command: overx_by_strata

Make Overexcavate Surface From Screen Entities

This command makes the overexcavate surface from entities on the layers defined as Overexcavate in the Define Layer Target/Material/Subgrade command. The resulting surface of Make Overexcavate Surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.

Prerequisite: overexcavate entities
Keyboard Command: mk_overx_tin

Adjust Overexcavate Surface

This command adjusts the overexcavate surface vertically within the selected perimeter polylines. This command allows the site to be overexcavated at a variety of depths in specified areas represented with CLOSED polyline boundaries. Select the desired areas to be adjusted when prompted at the command line.

Keyboard Command: adjust_overx
Prerequisite: an overexcavate surface

Draw Overexcavate Surface 3D Faces

Use this command to draw the 3D faces of the overexcavated surface model on the screen. The 3D faces will be drawn in the TK_OVERX_SURFACE layer and will depend on the latest surface created using the make and adjust routines.

Prerequisite: An overexcavate surface
Keyboard Command: draw_overx

Erase Overexcavate Surface 3D Faces

Use this command to remove the previously drawn 3D Faces from the screen.

Prerequisite: 3D Faces
Keyboard Command: erase_overx

Draw Overexcavate Cut Color Map

Use this command to display a cut color map on the screen that shows the areas of overexcavate cut. The colors will graduate from white to red based on zero cut depth to maximum cut depth. This command also offers the user to place a legend of the cut depths on the screen. Pick the desired location and type the desired scale of the legend when prompted at the command line.

Prerequisite: An overexcavate surface
Keyboard Command: overx_cfmap

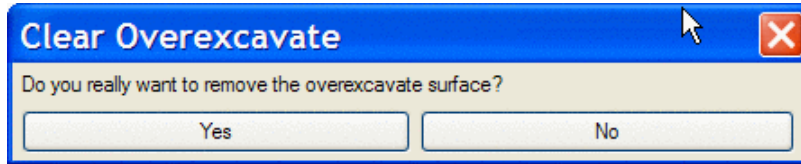
Erase Overexcavate Cut Color Map

Use this command to remove the previously drawn Cut Color Map and Legend from the screen.

Prerequisite: An overexcavate cut color map
Keyboard Command: overx_cfmap2

Clear Overexcavate Surface

Use this command to remove the overexcavate surface. When the overexcavate surface is removed, the rest of the Takeoff commands will not calculate overexcavate volumes. You will be prompted to confirm before the remove is done.



Pulldown Menu Location: Takeoff > Overexcavate Surface

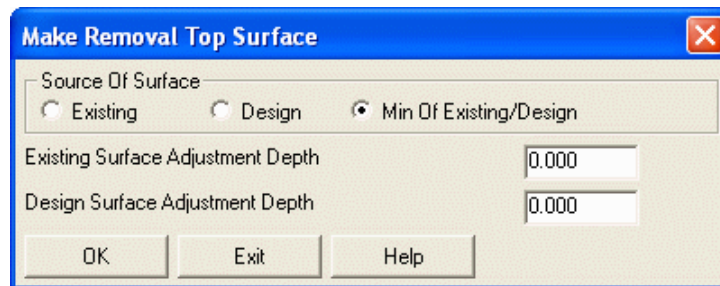
Prerequisite: An overexcavate surface

Keyboard Command: clear_overx

Make Top Surface

From Existing/Design Surfaces

This command sets the top Overexcavate surface (dwgname-rm.tin) that will be compared to a bottom Overexcavate surface for removal volumes. In the below dialog, Existing and Design surfaces created in Takeoff can be used as the top Overexcavate surface. Min Of Existing/Design is the minimum, or lowest grade, between the Existing and Design surfaces. Adjustment Depth allows you to drop either the Existing or Design surface by a specified amount.



Prerequisite: an existing or design surface

Keyboard Command: set_rm_top

From Triangulation Surface File

This routine allows a previously created surface .tin or .flt file to be loaded as the top Overexcavate surface.

Prerequisite: a previously created surface .tin or .flt file

Keyboard Command: rm_top_file

From Screen Entities

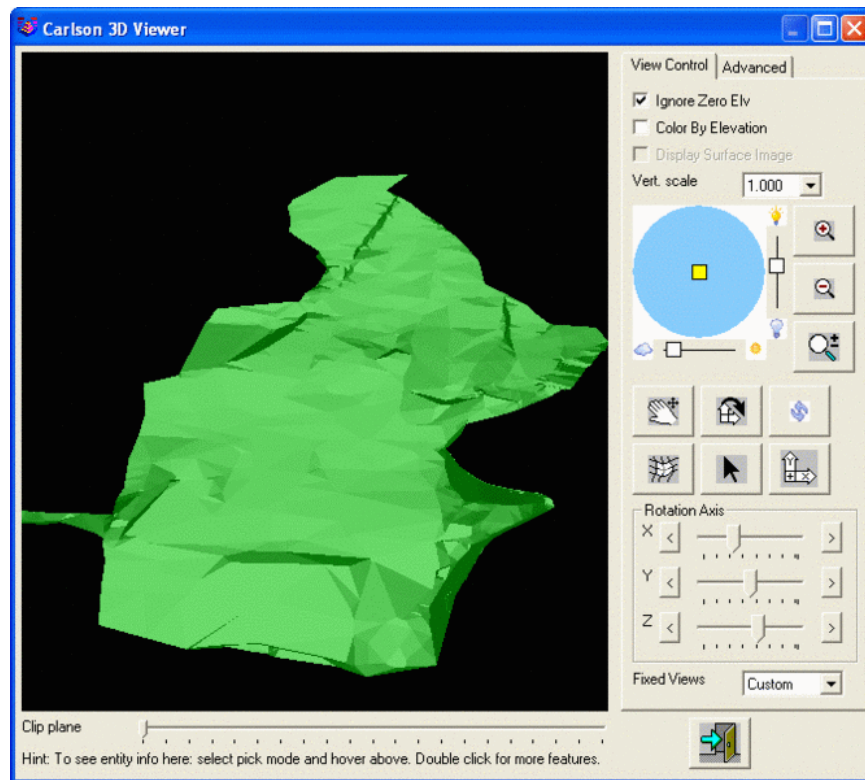
This command will create the top Overexcavate surface from entities in the plan view. Entities will need to have elevation such as contours, 3D faces, or elevated polylines.

Prerequisite: screen entities with elevation

Keyboard Command: mk_rm_top


View Top Surface


This command allows you to view the top Overexcavate surface in 3D mode.






In the top right of the control bar you can check to Ignore Zero Elev and Color By Elevation and change the Vertical Scale. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Color By Elevation shows elevation change with the change of colors. Note: Color By Elevation is used in the above example. To adjust the color use the color circle on the right.

The magnify glass icons can be used to zoom in and out. Click on the plus magnify glass to zoom in and

the minus magnify glass to zoom out. With the  icon click and drag up to zoom in and drag down to zoom out. The hand icon below the color circle allows you to pan around the viewer. Click and drag the direction you

want to move. The  icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point

according to the z axis. The  icon toggles on and off the shading of the surface (the shading is shown in the above drawing). The arrow icon reports the elevations at the bottom of the screen as you move around the surface.

The  icon restores the surface viewpoint to flat. The  icon exits 3D Driver Simulation.

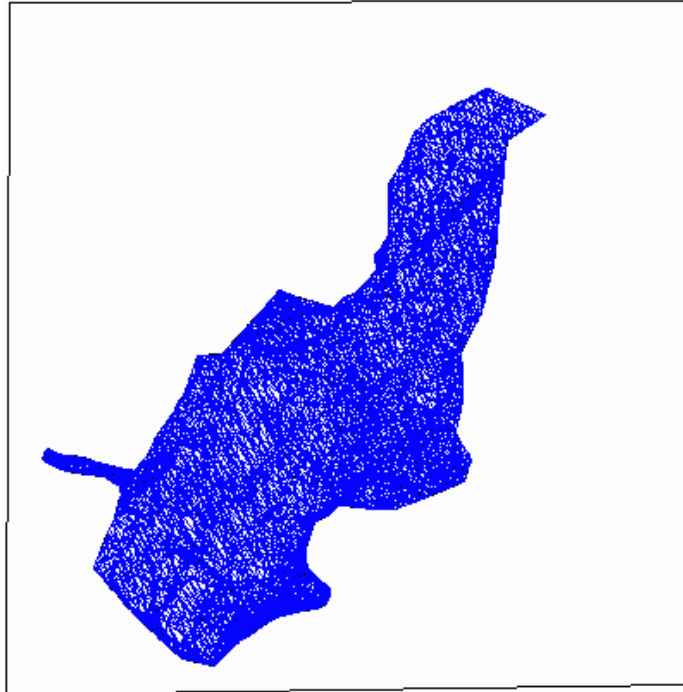
Rotation Axis: These three control bars rotate the surface around the x, y, and z axis. Clip plane trims the size of the surface shown in the viewer.

Prerequisite: a top Overexcavate surface

Keyboard Command: cube_rm_top

Draw Top Surface 3D Faces

This command will display the top Overexcavate surface as 3D faces in the plan view.



Keyboard Command: draw_rm_top

Prerequisite: Make Top Surface

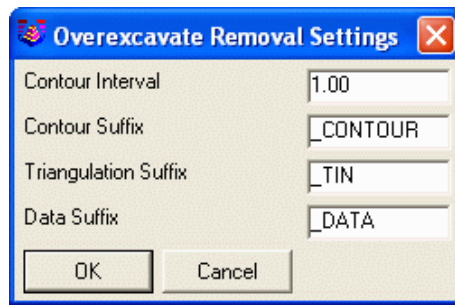
This command will erase the plan view entities created in Draw Top Surface 3D Faces.

Keyboard Command: erase_rm_top

Prerequisite: Draw Top Surface 3D Faces

Removal Settings

This command sets the layers suffixes for the entities created in the commands Draw Removal Surface and Draw Removal Contours. These Surfaces are added to the Removal Area names. For example, if Topo2 is set to Current in the Removal Manager, Draw Removal Surface will create 3D faces on the layer Topo2.TIN. Likewise, Draw Removal Contour will create contours on the layer Topo2.CONTOUR. The interval that the contours are drawn are also set here.

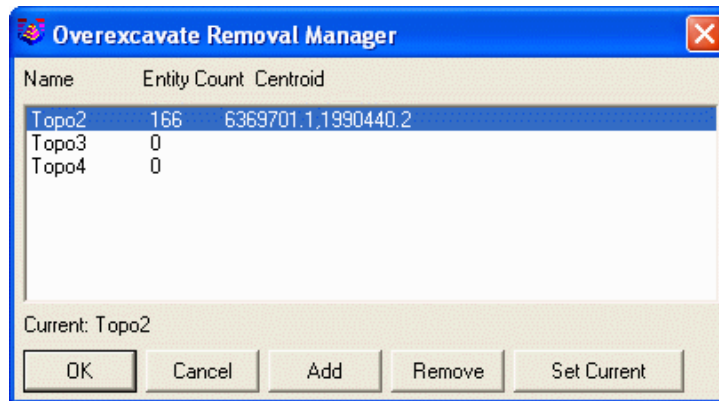


Prerequisite: none

Keyboard Command: rm_overx_setup

Removal Manager

In Removal Manager command every Removal Area in a project and the entities that define them is displayed as well as the Centroid (center coordinate) for that Area. Add allows you to name and create a new Removal Area. Remove will delete the Removal Area. When a Removal Area is set to Current, it will be used by other Removal commands when processing.

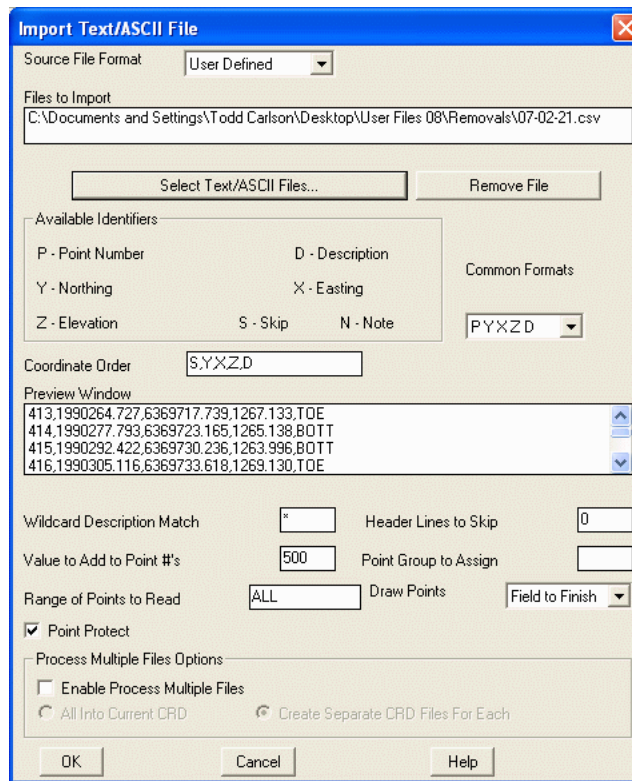


Prerequisite: none

Keyboard Command: rm_overx_mgr

Import Removal Text ASCII File

This command converts point data from an ASCII text file into the current Carlson coordinate (.CRD) file. The points brought in with this command will be assigned to the Current Removal Area if Draw Points is set to Points or Field-to-Finish. Each line of the text file can contain any combination of point number, northing, easting, elevation and description. All point information should be on one line with the values separated by a comma, space or other delimiter. Under the Source File Format setting you can choose from some specific formats or User-Defined. For User-Defined, the format of the text file is specified in the Coordinate Order field where the value identifiers are listed with the appropriate delimiters.



Common formats can be selected from the Common Format List. All the lines in the text file should contain only point data and any header lines should be removed. To read the text file, pick the Select Text/ASCII File button and choose the file to read. Then the selected file is displayed in the Preview Window to help with filling out the Coordinate Order. When the Coordinate Order is set, click OK to read the text file. The Wild Card Descriptions Match allows for only point with matching descriptions to be imported. With Point Protect active, the program will check if a point number already exists in the CRD before importing the point. If a point conflict is found, you can either assign a new point number or overwrite the old point. The Value to Add to Point Numbers allows you to renumber the points as they are imported. The Header Lines to Skip value is the number of lines not to be processed at the start of the text file. The Point Group To Assign option will create a point group with the specified name for the coordinate file containing the point numbers imported with Import Text/ASCII File. Special formats can be directly imported by choosing that File Format at the top of the dialog.

Prerequisite: Text/ASCII File and a Removal Area created and set to Current

Keyboard Command: rm_overx_mgr

Draw Removal Field to Finish

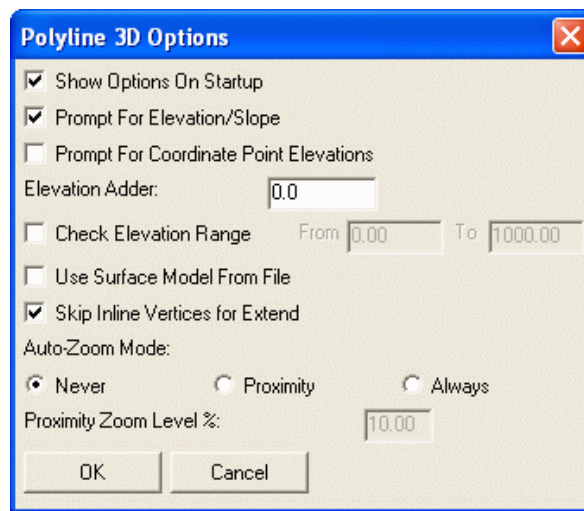
This command turns data collector field notes into Removal Area points and linework by matching the descriptions of the field points with user-defined codes. Two files are used in Field-to-Finish - a coordinate file and a field code definition file. For more on these files and their settings see Draw Field-to-Finish under Survey.

Keyboard Command: rm_overx_f2f

Prerequisite: A data file of points with descriptions and a Removal Area created and set to Current

Draw Removal Breakline

This command allows you to draw 3D linework for the Current Removal Area.



The Show Options on Startup dialog will appear every time the command is run, unless this is turned off. If it is off, then the last settings will apply. To get the box back, choose O for Options.

Prompt for Elevation/Slope controls whether the elevation of each picked point will be entered in, or hit S for slope to draw a slope line.

Use Surface Model from File will use a grid or triangulation file as a surface model. Wherever the points are picked on the surface, the elevation of the surface will be assigned to the polyline.

There are 3 options under Auto-Zoom Mode. Never will not zoom to the last point picked. Proximity will zoom to the percent proximity set below. Always will always zoom center on every point.

If the Proximity Auto-Zoom mode is checked, the percent of the proximity is set in the Proximity Zoom Level% box.

Keyboard Command: rm_overx_3dp

Prerequisite: a Removal Area set to Current

Removal Entities

Tag Removal Entities

This command allows the user to select polylines and points that define the Current Removal Area. Carlson Takeoff will separate each Removal Area in the Calculate Removals Volumes Report.

Prerequisite: linework and/or points intended for the Current Removal Area

Keyboard Command: tag_rm_overx

ID Removal Entities

This command allows users to identify Removal Entities by either picking on a polyline(s) or by searching the entire drawing. The command will then highlight the polyline in the plan view.

Prerequisite: Tag Removal Entities

Keyboard Command: id_rm_overx

Untag Removal

This command allows the user to remove previously tagged Removal Entities.

Prerequisite: Tag Removal Entities

Keyboard Command: untag_rm_overx

Set Removal Boundary

Use this command to select the "CLOSED" polyline that defines the outer most limit of the Current Removal Area. This boundary should dissect the site at the point where the Current Removal Entities end. Volume calculation will take place inside this boundary.

Prerequisite: a closed polyline

Keyboard Command: rm_overx_perim

Make Removal Surface

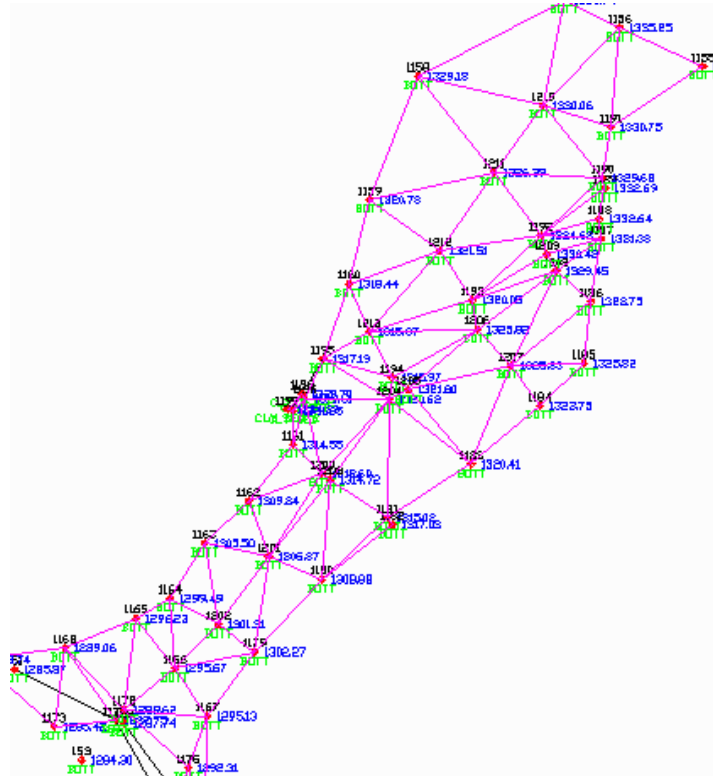
This command makes the triangulation models for the Removal Surfaces. The surface is automatically named as "filename-removalname.tin". The "filename" is set to the name of the current drawing (dwg) file. The "removalname" is determined by the Removal Area current in the Removal Manager. Before running this command, you must have a current Removal Area with elevated entities. Another prerequisite is that the Removal Boundary must be set for the site.

Keyboard Command: mk_rm_overx

Prerequisite: Removal Entities and a Removal Boundary

Draw Removal Surface

This command draws the current Removal Surface as 3D faces in the plan view.

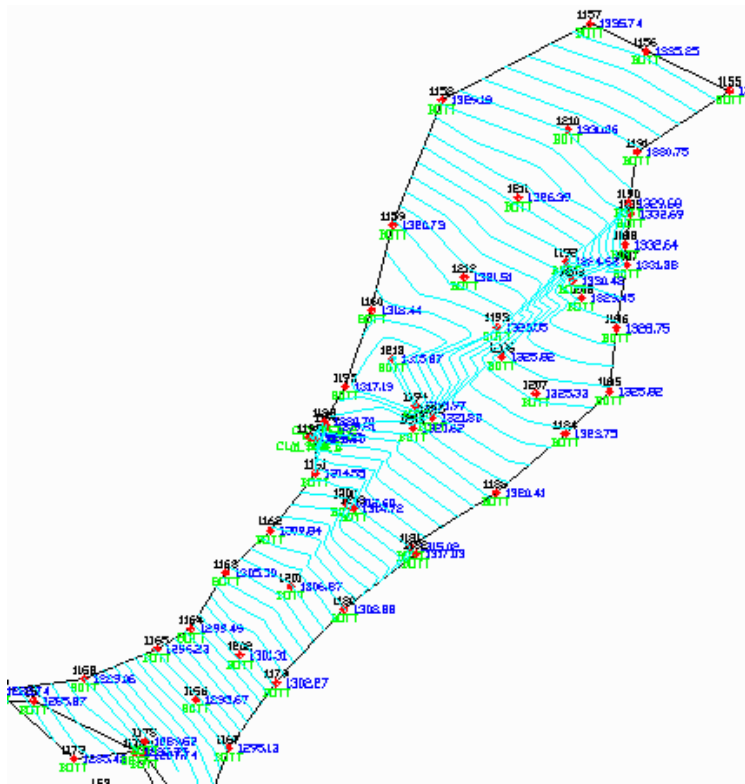


Keyboard Command: draw_rm_overx_tin

Prerequisite: Make Removal Surface

Draw Removal Contours

This command displays all the contours that represent current Removal Surface. They are created off of the Removal Area .tin model. For contour interval, see Removal Settings.



Keyboard Command: draw_rm_overx_ctr

Prerequisite: Make Removal Surface

Make Combined Removal Surface

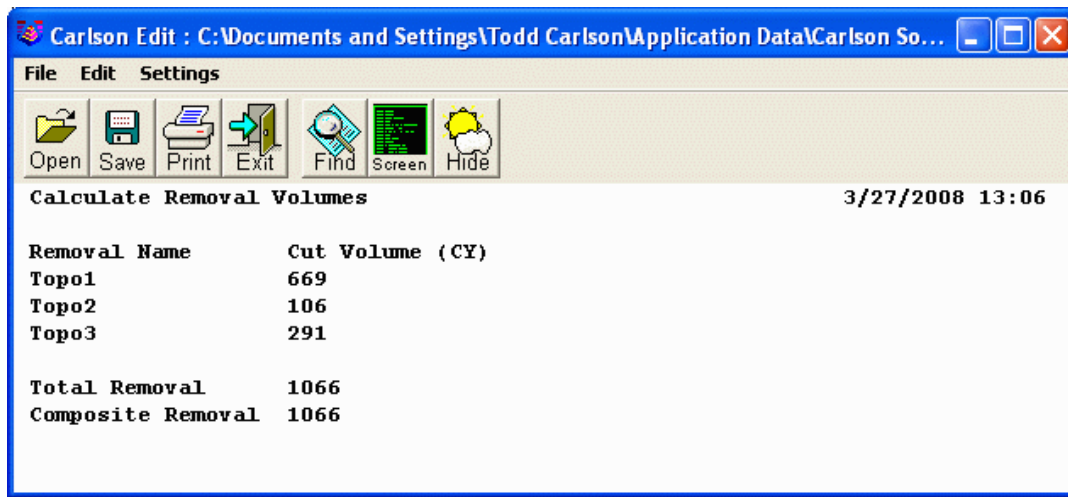
This command makes a TIN surface from all the removal surfaces from the Removal Manager. The command reports all the removal surfaces from the current drawing and then prompts for the TIN file to create.

Keyboard Command: overx_combine

Prerequisite: Overexcavate Removal Surfaces

Calculate Removals Volumes

This command reports the volumes in cubic yards for each Removal Area against the Top Removal Surface. The volumes are given for each area as well as a total for all the areas. Calculate Removal Volumes then creates and reports a Composite Surface against the Top Removal Surface taking the lowest grade in overlapping Removal Areas.



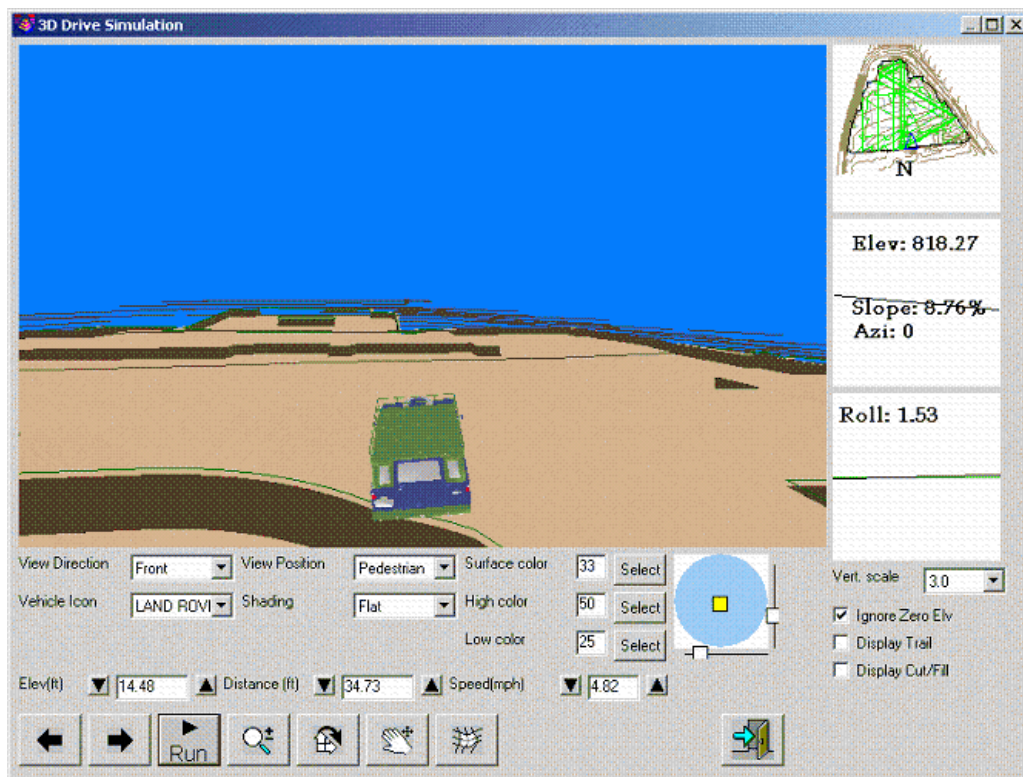
From the Standard Report Viewer, you can Save, Print, or place on the Screen the volume numbers. You can also type your own text into the report.

Keyboard Command: calc_rm_overx

Prerequisite: Make Top Surface and Make Removal Surface


3D Drive Simulation


This command allows you to view and move around the design surface in 3D mode.




Use the arrows on your keypad to move around the drawing.

At the very bottom of the window you will find the basic commands: Run will start to drive your vehicle around the surface, once your vehicle is moving the Run button turns into the Stop button. The arrows moves your vehicle left and right. The magnify glass zooms in and out. Click and drag up to zoom in and click and drag down

to zoom out. When your vehicle is stopped the  icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point according to the z axis.

The hand icon allows you to pan around the viewer. Click and drag the direction you want to move. The 

icon toggles the shading of the surfaces. The  icon exits 3D Driver Simulation.

Above the basic command buttons you can change the Elevation and Distance away from your vehicle. Also, you can set the speed at which your vehicle travels. For a smaller drawing you may want to move around slower, for a larger drawing faster. Note: Unrealistic speeds such as 500 mph in a dozer may cause 3D Drive Simulation to freeze.

View Direction: Sets the direction of the view from the Front, Back, Left, or Right.

Vehicle Icon: You can select which Vehicle you want to use whether: Dozer, Hummer, School Bus or none at all.

View Position: Sets the elevation and distance to either that of the driver, a pedestrian, or bird.

Shading: Here you can set the shading of the surface to either Flat, Smooth, Elevation, Cut/Fill, or none. Flat just shades the contours as they are. Smooth smooths contours to look for realistic. Elevation colors different elevations in different colors so differences can visual be seen. Cut/Fill colors areas of cut differently than areas of fill so they can be visually seen. None merely shows the triangulation and does not shade in a surface.

You can select the Surface, High, and Low color by enter in an AutoCAD defined color number or you can choose Select to pick a color. The circle on the right determines the shade of the color.

In the top right of the 3D viewer is an aerial map of your surface. Below that the Elevation, Slope percentage, Azimuth, and Roll are updated as your vehicle moves around the surface. Slope and Roll are shown visually here as well.

On the bottom right you can set the Vertical Scale and check to Ignore Zero Elev, Display Trail, and Display Cut/Fill. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Display Trail draws a line where your vehicle has driven. Display Cut/Fill displays the cut and the fill.

Prerequisite: a design surface

Keyboard Command: tk_flyby

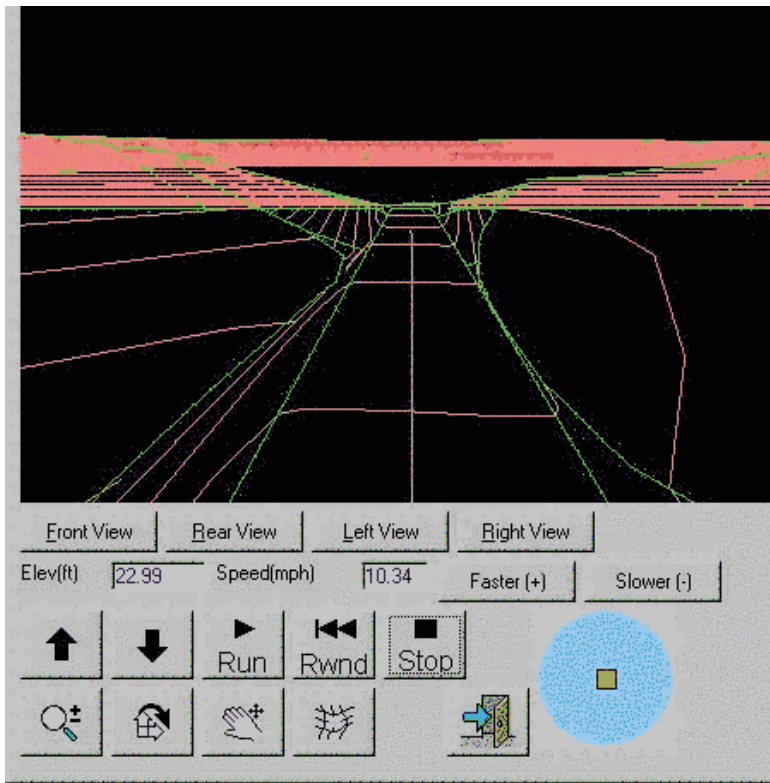
FlyOver Along 3D Polyline













This command allows you to view a self guided animation of following a path through a 3D surface model. There are two variations to this command. When the command is started, you must specify whether you want to use a surface model from file or screen entities.

Surface model from file: Using this method, you can select either a triangulation (.TIN) file or a grid (.GRD) file, then you have the option of following a polyline or following a "free" path. If you choose the polyline method, then the animation is limited to following the polyline. If you choose the "free" path method, you first specify two points to obtain a starting direction, the while inside the viewer you can point the animation in any direction.

Screen entities: Using this method, you must select a 3D polyline to follow. The animation is limited to following the polyline.

After making the above selections, the 3D graphics window is opened. The main window is for the animation, the smaller upper right window shows you the overall plan view, and the smaller window located at middle right shows you the current elevation, slope and azimuth. While following a "free" path, you will have a 3rd small window located at lower right which shows you the amount of roll at your current position.

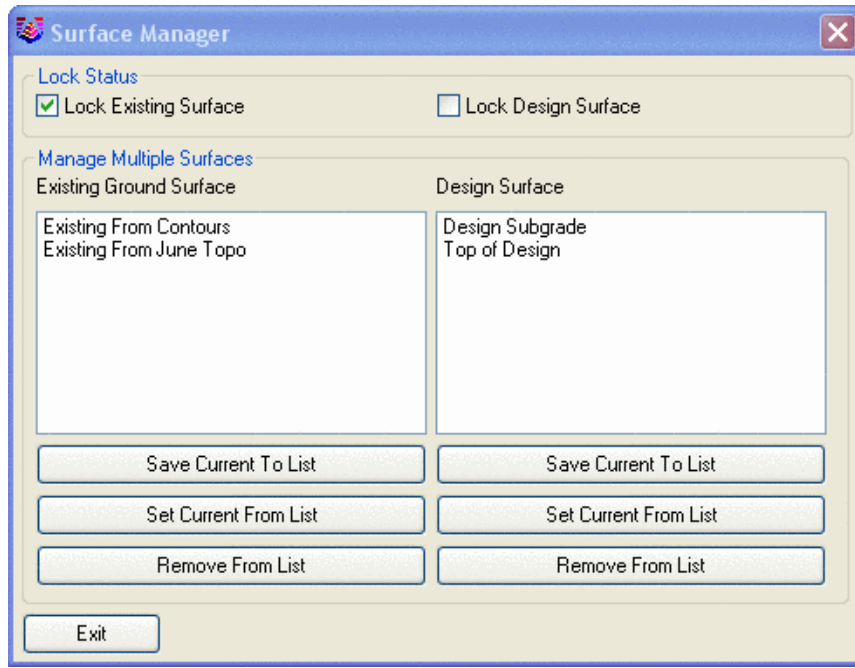


 This button raises the elevation of your viewing position.
  This button lowers the elevation of your viewing position.
  This button turns your viewing position to the left.
  This button turns your viewing position to the right.
  This button allows you to zoom in and out.
  This button allows you to rotate the main animation window in any X, Y or Z direction.
  This button allows you to pan.
  This button toggles shading on and off.
  This button starts the animation in the main window.
  This button stops the animation.
  This button exits the 3D Surface FlyOver command.
  Control for position of the light source, viewed from above.

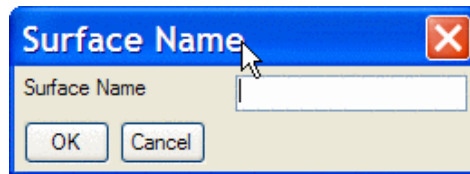
Prerequisite: Surface Model and optionally a 3D Polyline
Keyboard Command: flyby

Surface Manager

This command allows the user to name and manage multiple surface models. The Surface Manager dialog shown here has options to name and save the current "existing and design" surface models. The "current" surface is dictated by the layers that populate a target and the Make Surface command. If layers are removed from a target, and others assigned, multiple surfaces can be created and stored. When the Lock Status is check, the Current Surface will remain current even if you run Takeoff > Make Existing Ground Surface or Make Design Surface. If the Lock Status is uncheck, then Making the Existing or Design Surface will become the current surface, overriding the current surface selected in the Surface Manager. Highlight a named surface and select the Set Current From List option to make that model active. Use the Remove From List option to remove a named surface model from the list.



Selecting the Save Current To List options brings up the Surface Name dialog box shown here. Type the desired name that describes a particular surface model and select OK.



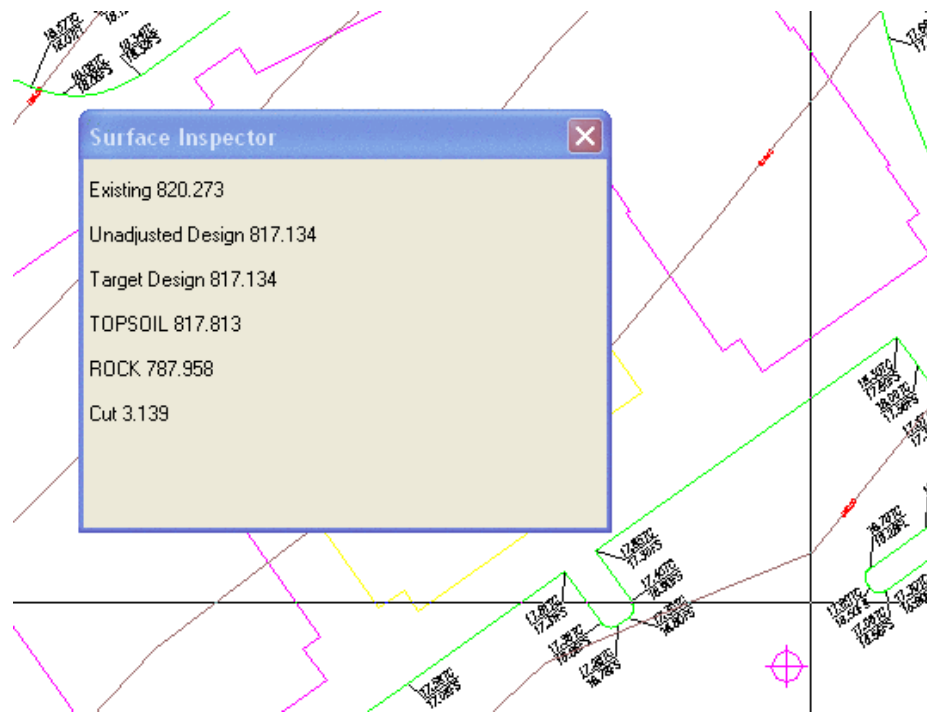
Pulldown Menu Location: Takeoff > Surface Tools

Prerequisite: none

Keyboard Command: surf_mgr

Surface Inspector

This command allows you to report and optionally label elevations from your drawing. You can analyze all of your different surface files at one time. After running the command, Surface Inspector will begin showing you real-time elevations for each surface as you move the cursor on the screen. If you pick a point or enter coordinates, the elevation will be labeled along with the surface name.



Surface inspector shows you real-time elevations as you move the cursor over your surface.

Prerequisite: Surface Model (s)

Keyboard Command: surfvals

Surface Report

This command reports a variety of information on each of your different surfaces. This is useful for checking for bad data and the file names of your surfaces. An example is below.

Surface Report

Max Cut: 18.327 at 409269.984,207196.674

Max Fill: 1.943 at 409389.586,207248.866

Original Ground After Topsoil Removal

File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-ex.tin

Date Modified: Thu Feb 10 10:02:05 2005

File Size: 64,028

Points: 259, **Edges:** 744, **Triangles:** 486

Min Z: 184.000 at 409299.790,206879.287

Max Z: 210.000 at 409571.562,207177.240

Design With Subgrade and Topsoil Replacement

File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-sg.tin

Date Modified: Thu Feb 10 10:02:08 2005

File Size: 153,038

Points: 609, **Edges:** 1,779, **Triangles:** 1,171

Min Z: 176.000 at 409357.096,206821.604

Max Z: 206.000 at 409551.532,207185.124

Original Ground Before Topsoil Removal

File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-og.tin

Date Modified: Thu Feb 10 10:02:05 2005

File Size: 64,028

Points: 259, Edges: 744, Triangles: 486

Min Z: 184.000 at 409299.790,206879.287

Max Z: 210.000 at 409571.562,207177.240

Design Without Subgrade or Topsoil Replacement

File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-fg.tin

Date Modified: Thu Feb 10 10:02:08 2005

File Size: 153,038

Points: 609, Edges: 1,779, Triangles: 1,171

Min Z: 176.000 at 409357.096,206821.604

Max Z: 206.000 at 409551.532,207185.124

Design With Subgrade

File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-zn.tin

Date Modified: Thu Feb 10 10:02:08 2005

File Size: 153,038

Points: 609, Edges: 1,779, Triangles: 1,171

Min Z: 176.000 at 409357.096,206821.604

Max Z: 206.000 at 409551.532,207185.124

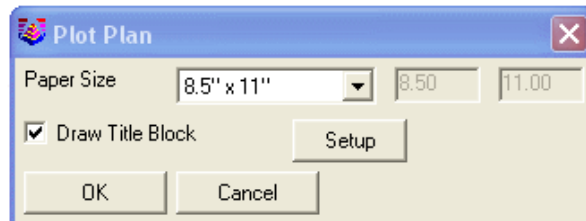
Prerequisites: A Surface

Keyboard Command: SURF_STATS

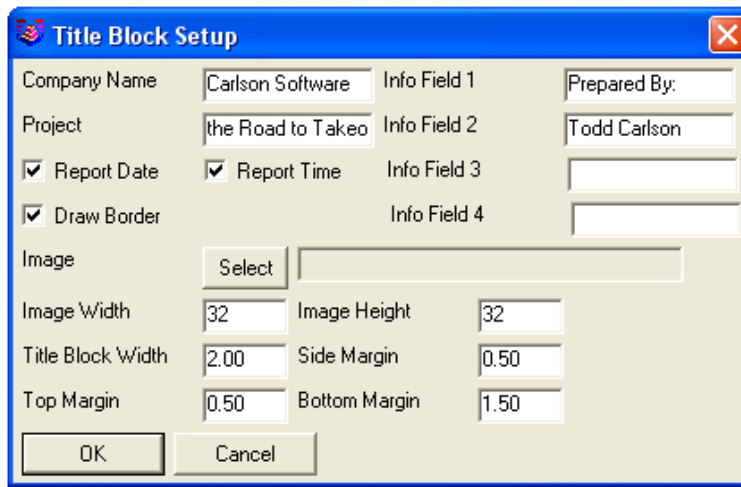
Graphic Reports

The Graphic Report commands can be used to create .PDF files for different Profiles, Cross Sections, 3D Surfaces, and of the Plan View. Adobe Reader or Pro is required for these routines to function.

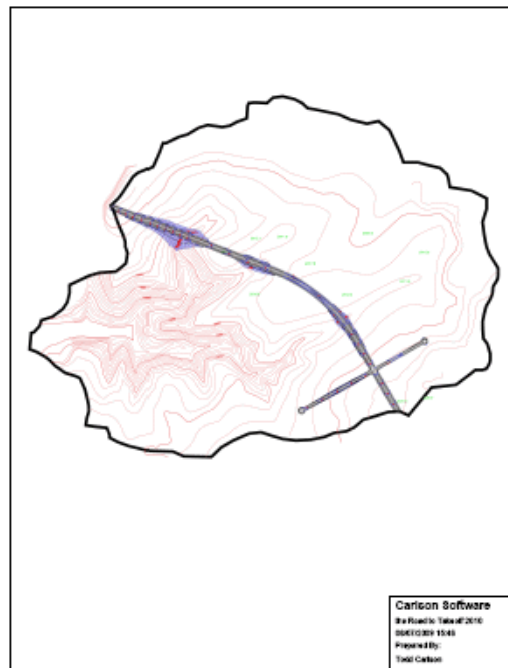
The Plan Graphic Report command creates a plan view report of the site. Isolate the layers you want to show up in the report before you run this command. The routine will zoom extents to the limits of the layers you select and fit them to the Paper Size you select.



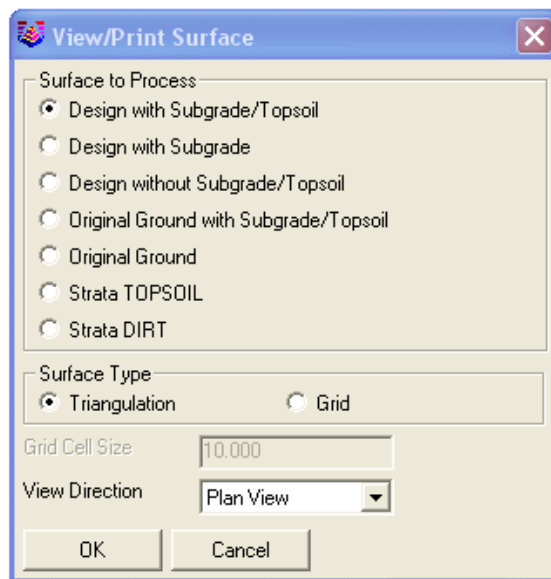
The Title Block Setup dialog draws a border and title block for the selected sheet size. Here you can enter in text you want to displayed in the report.



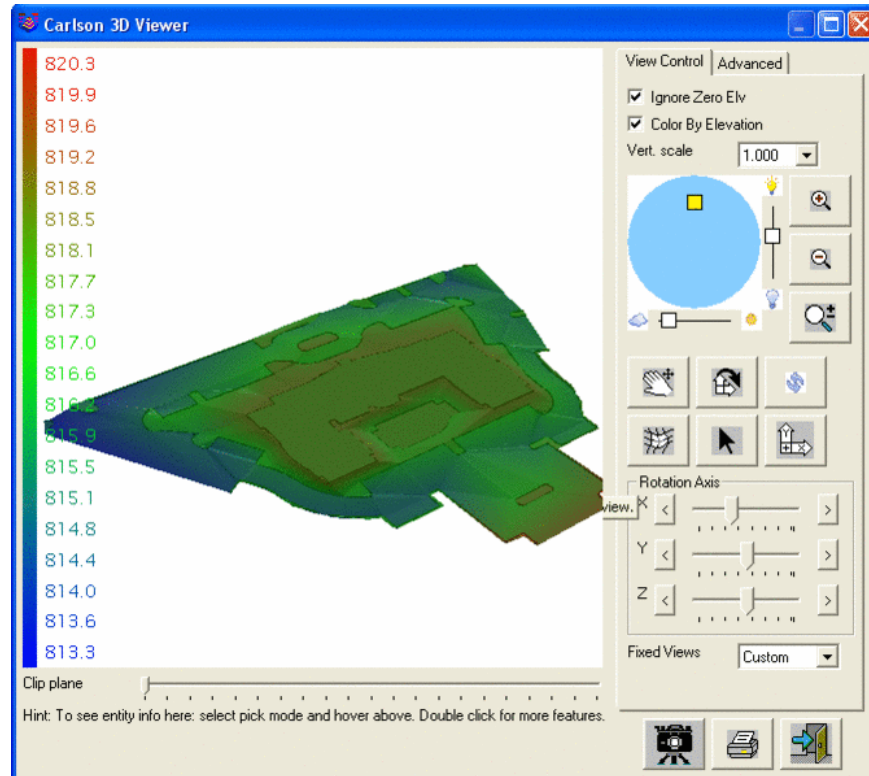
Here is an example of the Plan Graphic Report.



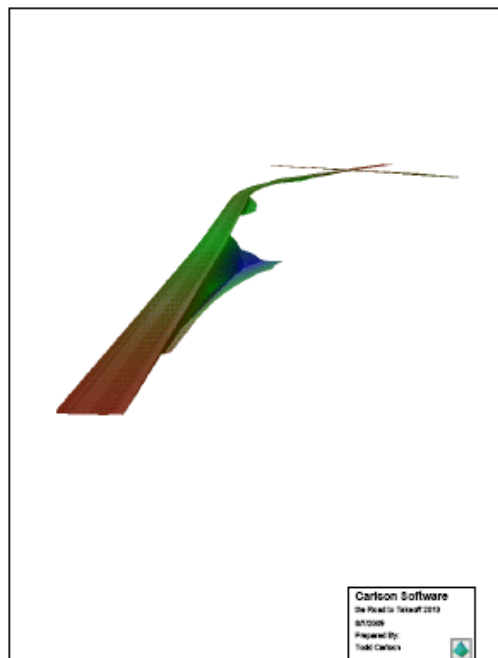
The Surface Graphic Report command creates a 3D view PDF report of any surface (.tin or .flt files) created in the drawing, including existing, design, and strata surfaces. In the first dialog, select the surface you wish to see in the report. This dialog also gives you the option to view the surface as a triangulation mesh or grid. View Direction determines the orientation of the surface in the next dialog.



The Carlson 3D Viewer gives you the ability to zoom in and out, pan, rotate around the X,Y,Z axis and shade the lighting. When you have orientated the model to your liking, click on the Printer icon in the lower right corner of the dialog.

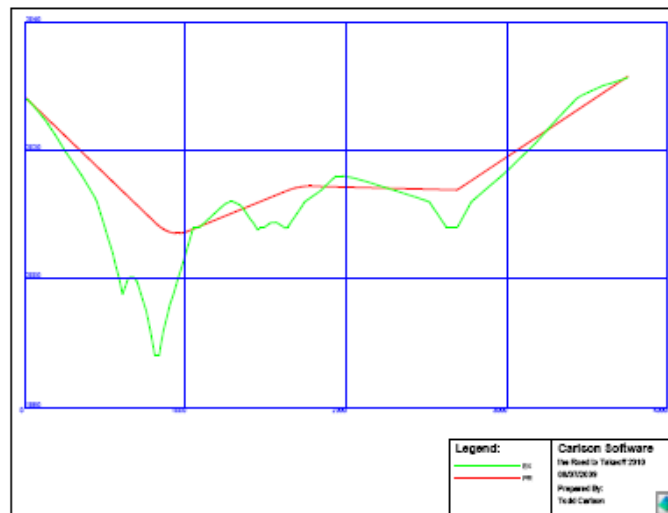


Here is an example of the Surface Graphic Report.

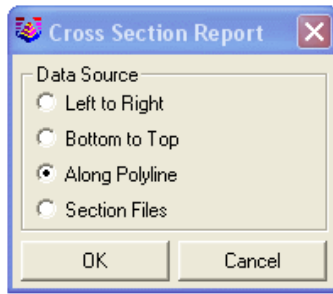


The Profile Graphic Report creates a 3D view profile of the site existing or design surface. You will first be prompted for how to create the Profile:

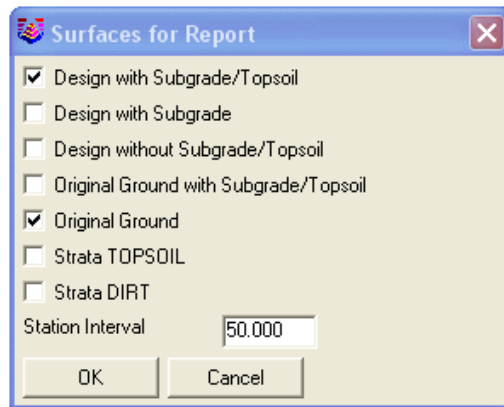
"Pick starting point (CL-Centerline,P-Polyline)". CL prompts you to load a previously created Centerline (.cl) file, P allows you to pick any polyline in the plan view, or you can pick two points on the screen that you want the program to create a profile through. Next, you will be prompted for Title Block settings similar in other Graphic Reports. Here is an example of the Profile Graphic Report.



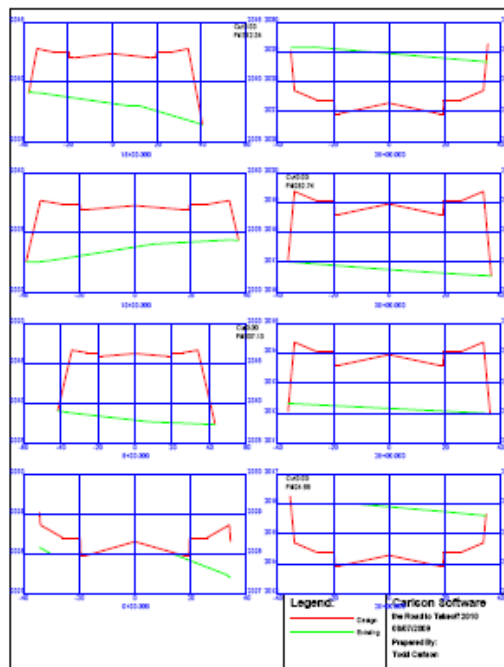
The Section Graphic Report creates cross section in a single step with option to include the plan view showing the section alignment. Cross Sections can be created within the routine from Left to Right, Bottom to Top, or along any polyline in the drawing. For more control over the orientation of the cross sections, Select Files allows you to select a previously created section file (.sct).



Like many other Graphic Reports, several of the surface models (.tin or .flt) created in the job can be used in the report. Simply select the surfaces you want to see in the report (there is no limit) and at what Station Interval (in feet) you want to see them. Note: this is not an opinion when using a .sct file.



Here is an example of the Section Graphic Report.



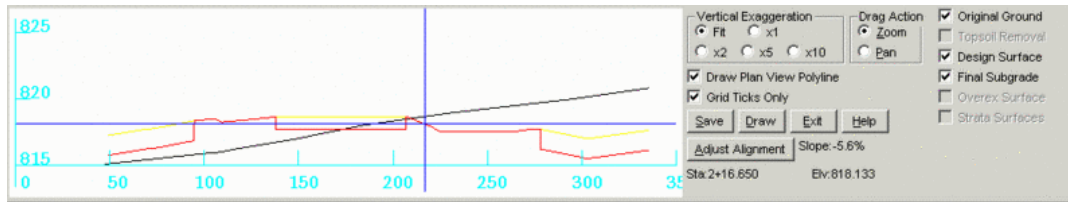
Pulldown Menu Location: Tools > Graphic Reports

Prerequisite: Existing and design surfaces

Keyboard Command: plotplan, plotsurf, tk_quickpro, plotsct

Quick Profile

This command allows you to create, view, edit, and report profiles from the Takeoff surfaces.



Pick starting point (CL-Centerline,P-Polyline): To make a profile you need to define the alignment by: 1) picking points on the screen; 2) typing in CL in the command prompt, and selecting a centerline file; or 3) typing in P and choosing a polyline from the screen. After doing so, the above profile viewer is created.

The far right dialog box allows you to toggle on and off different Surfaces to view in the profile viewer including: Original Ground, Topsoil Removal, Design Surface, Final Subgrade, Overex Surface, Strata Surfaces. If a surface is not defined in the current Takeoff project, like Topsoil Removal in this example, than you will not have the option to display it. In this example, the three Surfaces that can be displayed, Original Ground, Design Surface, and Final Subgrade, are displayed in the profile viewer.

When you move the cursor around the profile viewer a crosshair follows along the surface and reports the Station, Slope %, and Elevation at each point. It is displayed towards the bottom-right side of the screen next to Adjust Alignment. In this example the station is 2+16.650, the Slope is -5.6%, and the Elevation is 818.133. A crosshair can be seen in the profile drawing and along the alignment in the main drawing as well.

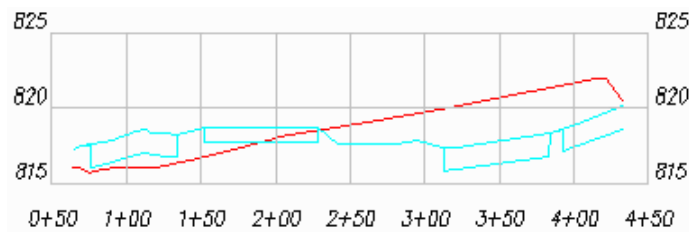
Vertical Exaggeration: x1 is the actual appearance of the surface(s). Depending on the flatness of the surface(s), you can select x2, x5, x10 vertical exaggerations to better see the elevation differentiation and different surfaces. The option Fit automatically exaggerates the vertical to best fit the profile viewer.

Drag Action: This dialog allows you to zoom in and out, and pan around the profile. To zoom in click and drag up, to zoom out click and drag down. To Pan, click and drag the direction you want to move.

The Adjust Alignment icon allows to pick the polyline or centerline that you used and move it to your liking. If you selected an endpoint vertex, you can pivot that vertex around 360 degrees and the profiles will update in real time. This is helpful when checking for spikes. If you select the middle vertex then you can shift the entire centerline around.

If you created a profile alignment by picking points and you want to save that polyline you created then toggle on Draw Plan View Polyline. If you do not choose Draw Plan View Polyline than the polyline will be lost when you exit out of the Quick Profile command. Grid Ticks Only marks elevations and distances but does not draw them into grids.

The Save icon allows you to save the profile as a (.pro) file by whatever name you give it. The Draw icon allows you to draw the profile right on your drawing. Set the layer name, vertical and horizontal scale as desired, pick a starting point to draw, and the profile is created. Note: the below example has a vertical scale of 5 feet per grid and a horizontal scale of 50 feet per grid.



Prompts

Pick starting point (CL-Centerline,P-Polyline): p

Polyline should have been drawn in direction of increasing stations.

CL File/<Select polyline that represents centerline>:

Loading edges...

Loaded 5057 points and 14923 edges

Created 9866 triangles

Prerequisite: a surface

Keyboard Command: TK_QUICKPRO, QUICKPRO

Cut/Fill Centroids

This command calculates all the areas of cut and fill between two triangulation surfaces. The center of mass or centroid for each area is calculated. The Minimum Region Volume is an optional filter that will skip reporting areas with volumes less than the specified amount. The Generate Labels option draws a symbol at the centroid and text for the region name and volume. The Generate Boundaries option draws closed polylines for the perimeters for each area. The Hatch Regions option is used to visually shows cut/fill areas in your drawing. Separate hatch patterns can be used for cut and fill areas.

The Use Inclusion/Exclusion Areas option will make the program prompt for polyline perimeters for the inclusion and exclusion areas on the site. For example, use an inclusion perimeter to calculate within an area of interest. When this option is off, the program uses the full extent of the surfaces.

A report is generated with the volumes and centroids for all the cut and fill areas. When the Report Optimized Earth Movement option is active, the report includes a list of the earthwork movements between cut and fill areas that minimizes the overall earthwork movement (volume * distance) where the distance is distance between the centroids.

Cut & Fill Centroids

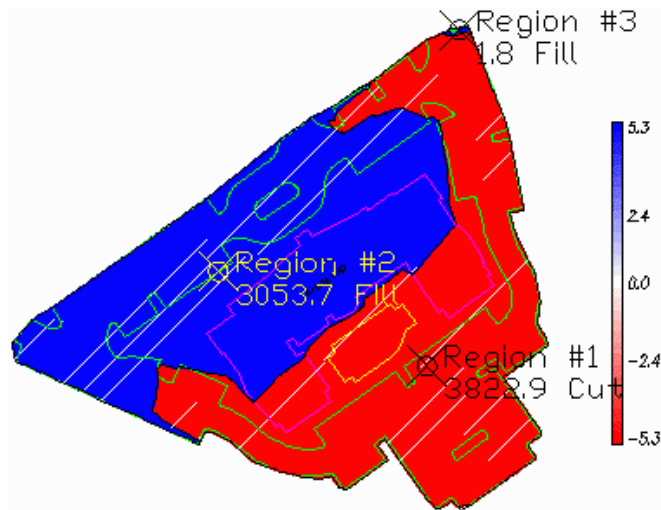
General
 Min. region volume: 1.0
 Cut/Swell factor: 1.000
 Fill/Shrink factor: 1.000

Labels
 Generate labels
 Text size scaler: 0.100
 Layer: CF_CENTROID

Boundaries
 Generate boundaries
 Layer: CF_BOUNDARY

Hatches
 Hatch regions
 Hatch scale: 1.0
 Automatic Hatch Scale
 Layer: CF_HATCH
 Fill style: ANS137
 Cut style: ANS136

Use Inclusion/Exclusion Areas
 Report optimized earth movement



Cut & Fill Centroid Report

Original Ground: G:\oem4\src2\work\demo1-ex.flt
Design Surface: G:\oem4\src2\work\demo1-fn.flt

Region	Volume(C.Y.)		Northing	Easting
1	3822.9	Cut	2190032.48	6135170.29
2	3053.7	Fill	2190117.40	6134982.48
3	1.8	Fill	2190334.67	6135199.41

Earth Movement Report:

From Region	To Region	Volume(C.Y.)	Distance
1	2	3053.7	206.12
1	3	1.8	303.59
1	External	767.4	
Total Internal Volume * Distance:			629972.20
Total External Volume:			767.44

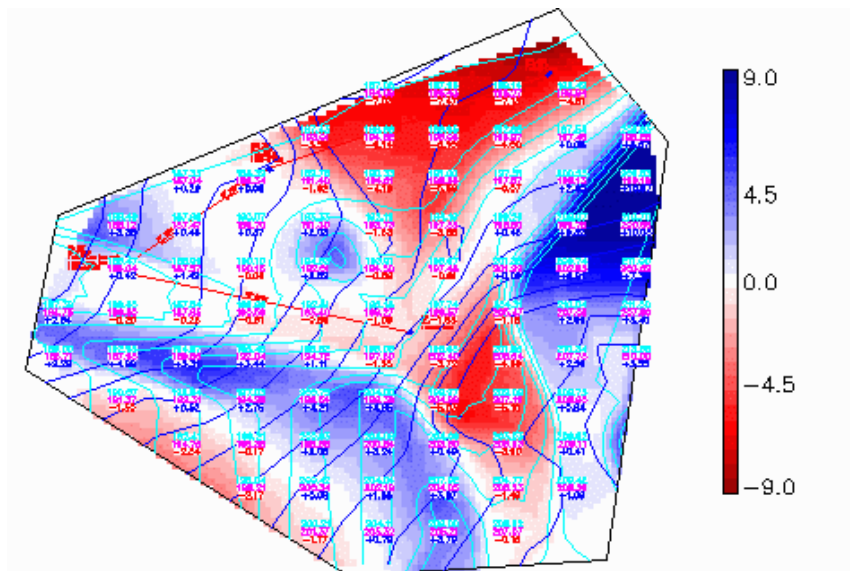
Here is the Cut and Fill Centroid Report for the above example. It shows the volumes, the coordinates of the centroids, and the Earth Movement Report. The Earth Movement Report shows the minimal distances for moving Cut to Fill areas.

Prerequisite: Existing and Design surfaces

Keyboard Command: tk_cutfillc

Cut/Fill Map Legend

This command will draw a Cut/Fill Map Legend on your drawing. It will display the cut/fill amount, color, and range, as seen below.



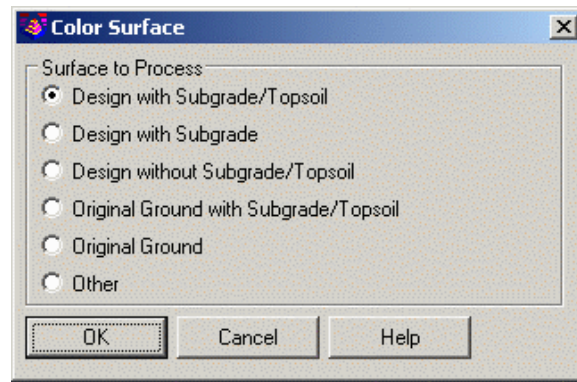
Prerequisite: Cut/Fill amounts

Keyboard Command: CF_MAP_LEGEND

Color Surface By Layer

This command sets the color of the triangles in a triangulation file for the areas within closed polylines using the layers and color assignments from Define Layer Surface/Material/Subgrade. The colored areas of the triangulation surface can then be viewed in commands like Surface 3D Viewer. For example, this command can set separate colors on the triangulation for pavement areas, building pads, ponds, etc.

The color assignments should be set before running this command by using Define Layer Surface/Material Subgrade. This command only prompts for which triangulation file to colorize. Then use a viewer command like Surface 3D Viewer to see the colored surface.



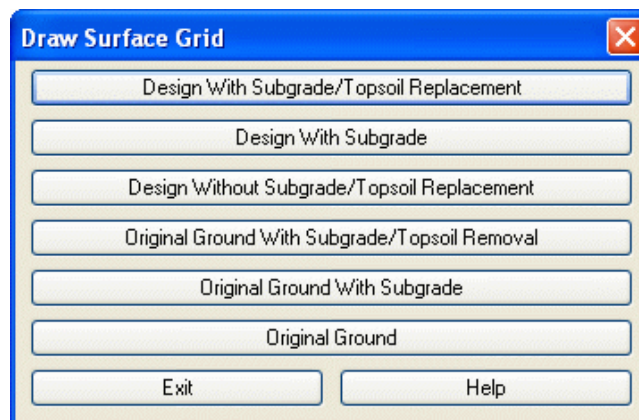
Pulldown Menu Location: Takeoff > Surface Tools

Prerequisite: A surface and colors set in Define Layer Surface/Material/Subgrade

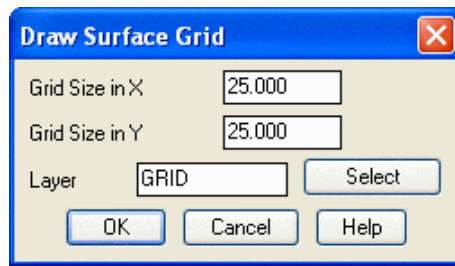
Keyboard Command: color_tin

Draw Surface As Grid

This command writes a grid file (.grd) from an existing triangulation file (.flt or .tin) in the current drawing.



After selecting the triangulation file to convert, you are prompted for the X, Y grid interval and the Layer name.

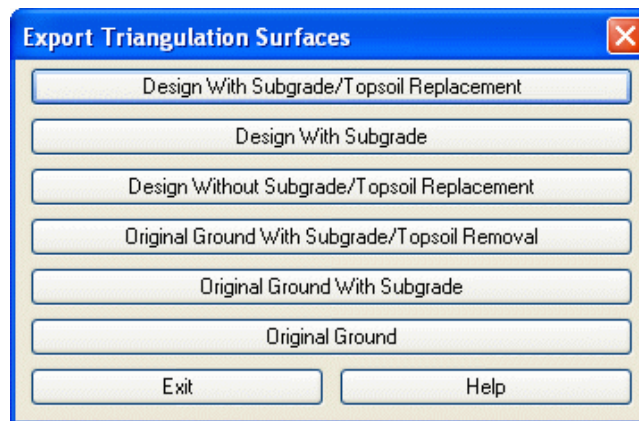


Prerequisite: an existing or design surface

Keyboard Command: draw_surface_grd

Import/Export Carlson Triangulation Files

Import Carlson Triangulation Files allows you to import an external surface file into Takeoff to use as a named surface. Export Carlson Triangulation Files allows you to take a Surface Triangulation file and save it independent of the drawing.



Prerequisite: .TIN or .FLT files

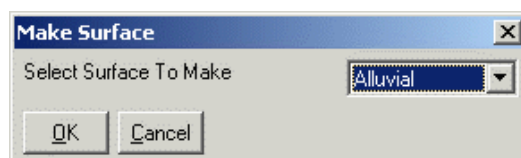
Keyboard Command: import_tin, export_tin

Make User Defined Surface

This command makes a surface from the entities on the layers defined as user-defined targets in the Define Layer Target/Material/Subgrade command. The purpose of user-defined surfaces is for modeling surfaces besides existing ground and design. The drawing needs to contain entities that represent the elevations of the user-defined surface. For example, the user-defined surface could be for alluvial soil and the drawing has contour polylines for this surface.

There is a dialog to select which surface to make. The surface is stored in a triangulation file that is named after the current drawing name with the user-defined surface name appended.

This user-defined surface can be applied to Takeoff routines by running the Set Active Surfaces command.



Pulldown Menu Location: Takeoff > Surface Tools
Prerequisite: Define Layer Target/Material/Subgrade command
Keyboard Command: mk_user_tin

Volumes By Triangulation

Volumes By Triangulation is an alternative volume method that compares two triangulation networks. This method is different from the grid based volume routines (*Stockpile Volumes, etc.*) and the cross section volume routine (*Calculate Section Volume*). Volumes by Triangulation calculates faster in most cases than the other methods, and it is the most accurate because it uses true TIN to TIN prismatic volumes. This added accuracy in general is very small. The grid resolution is usually sufficient to model the surface for the grid based volumes. The Volume By Triangulation accuracy applies well when there is a feature like a 5 foot wide ditch. Then the grid resolution would need to be less than 5 foot to model the ditch which might be difficult on a large site.

The disadvantage to this routine is that it lacks the output options that help the analysis of the volume such as Difference Contours. Also Volumes by Triangulation does no extrapolation and stops calculating volume at the perimeter of the smaller of the two triangulation networks. Volumes By Triangulation is better when used with point data instead of contour data because contour data requires triangulating all the contour polylines as breaklines which creates a large triangulation network and is slower.

The triangulation networks to compare are defined in .flt files that are created by *Triangulate & Contour* with the Write Triangulation File option. Before using this command, run *Triangulate & Contour* twice to create an triangulation (.TIN) file for each surface. The volume calculation is limited by either the extent of the triangulation networks or by an inclusion/exclusion perimeter(s). These perimeters must be closed polylines.

Output data includes area, tons by density, average thickness, shrink and swell, ratio, and total volume.

Prompts

Select EXISTING surface Tmesh File Choose a .tin file
Select FINAL surface Tmesh File Choose another .tin file
Pick inclusion perimeter polyline (ENTER for none): *pick a closed polyline perimeter*
Calculating ...
Write report to file (Yes/<No>)? *Press Enter*
Write report to printer (Yes/<No>)? *Press Enter*
Comparing Triangulation files: C:\SCADXML\DATA\TRI1.FLT
and C:\SCADXML\DATA\TRI2.FLT
Cut volume: 66891.35 C.F., 2477.46 C.Y.
Fill volume: 43458.01 C.F., 1609.56 C.Y.

Pull-Down Menu Location: Tools-> Surface Tools
Prerequisite: Two .tin files.
Keyboard Command: trivol

Calculate Stockpile Volume

This command is a customized and simplified method for calculating volumes in a situation in which the entire volume to be calculated is above the perimeter elevation, such as in the case of a stockpile of material. The complimentary command, *Calculate Pond/Pit Volume*, is for the opposite situation, in which the entire volume to be calculated is below the elevation of the perimeter.

The program internally computes base and final grid surfaces from drawing geometry. The base surface is calculated from a 3D polyline representing the perimeter of the area being analyzed. If that 3D polyline is drawn on the PERIMETER layer, the command will automatically detect and use it. If no 3D polyline is found on that layer, you have an opportunity to manually select another 3D polyline to use. The 3D polyline perimeter can be drawn with

the Draw 3D Polyline Perimeter command before using this routine.

The 3D polyline perimeter is also used as the inclusion perimeter for the volume calculation.

Additional 3D polylines can also be specified to more precisely define the base surface. These must be on the BASE_BREAKLINE layer to be used for this purpose. These can be generated by the *Draw 3DPoly Base Breakline* routine.

The final surface is calculated from all of the other selected drawing entities such as points, line, inserts, and polylines, along with the perimeter polyline, but not including the BASE_BREAKLINE polylines.

You have the option of setting the resolution of the grids.

Prompts

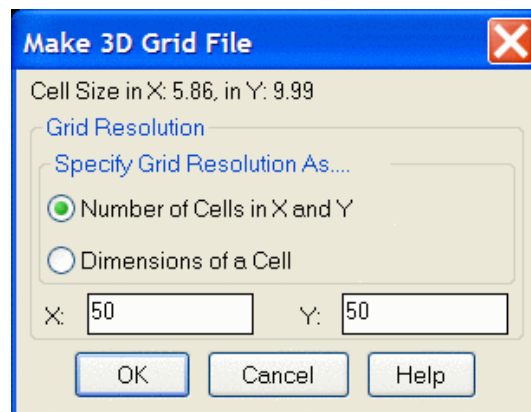
Material density lbs/ft³ (Enter for none): *enter a material density in lbs per cubic foot, or press Enter for none*

Ignore Zero Elevations [<Yes>/No**]:**

Select stockpile entities and perimeter.

Select objects: *pick the objects that define the stockpile and the 3D polyline perimeter*

Select stockpile perimeter polyline:



Make Grid File dialog Set the resolution and then click OK.

Sample volume report

Volume report

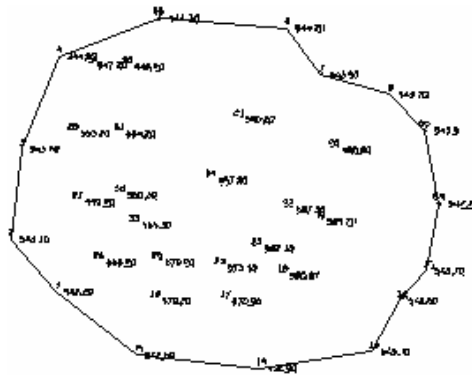
Lower left grid corner: 15965.45,12657.05

Upper right grid corner: 16269.40,12906.29

X grid resolution: 50, Y grid resolution: 50

X grid cell size: 6.08, Y grid cell size: 4.98

Stockpile volume: 1191674.87825 cubic ft, 44136.107 cubic yards



Stockpile defined by points and a 3D polyline perimeter
Window these objects to obtain the volume report

Keyboard Command: stockvol

Prerequisite: Data representing the stockpile surface and a 3D polyline representing the perimeter of the stockpile.

Calculate Pond/Pit Volume

This command is a customized and simplified method for calculating volumes in a situation in which the entire volume to be calculated is below the perimeter elevation, such as in the case of a pond or pit. The complimentary command, *Calculate Stockpile Volume*, is for the opposite situation, in which the entire volume to be calculated is above the elevation of the perimeter.

The program internally computes base and final grid surfaces from drawing geometry. The base surface is calculated from a 3D polyline representing the perimeter of the area being analyzed. If that 3D polyline is drawn on the PERIMETER layer, the command will automatically detect and use it. If no 3D polyline is found on that layer, you have an opportunity to manually select another 3D polyline to use. The 3D polyline perimeter can be drawn with the Draw 3D Polyline Perimeter command before using this routine.

The 3D polyline perimeter is also used as the inclusion perimeter for the volume calculation.

Additional 3D polylines can also be specified to more precisely define the base surface. These must be on the BASE_BREAKLINE layer to be used for this purpose. These can be generated by the *Draw 3DPoly Base Breakline* routine.

The final surface is calculated from all of the other selected drawing entities such as points, line, inserts, and polylines, along with the perimeter polyline, but not including the BASE_BREAKLINE polylines.

You have the option of setting the resolution of the grids.

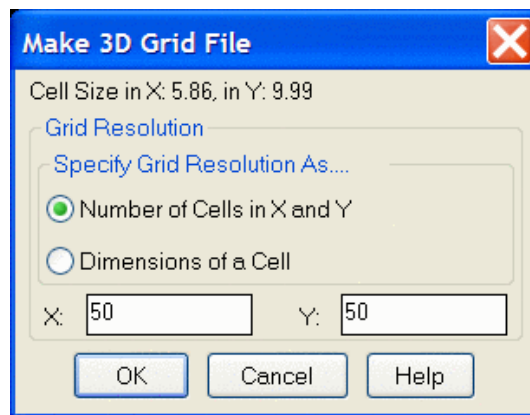
Prompts

Ignore Zero Elevations [<Yes>/No]?

Select Pond/Pit entities and perimeter.

Select objects: *pick the objects that define the surface and the 3D polyline perimeter*

Select Pond/Pit perimeter polyline:



Make Grid File dialog Set the resolution and then click OK.

Keyboard Command: pitvol

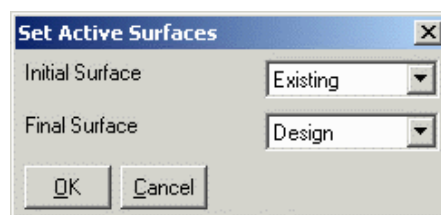
Prerequisite: Data representing the pond/pit surface and a 3D polyline representing the perimeter of the pond/pit.

Set Active Surfaces

This command assigns which surfaces to use for initial and final. These surfaces are used by all the Takeoff routine that compare surfaces including:

- Calculate Total Volumes
- Calculate Volumes Inside Perimeter
- 3D Drive Simulation
- Cut/Fill Contours/Labels/Color Map
- Surface Inspector
- Quick Profile
- etc.

The surface created by the Make Existing Ground Surface command is called "Existing" and is the default for the Initial Surface. The surface created by the Make Design Surface command is called "Design" and is the default for the Final Surface.



The purpose of this routine is for selecting user-defined surfaces to use in place of the existing ground or the design surface. For example, there could be a user-defined surface for alluvial soil that is set as the initial surface while design is set to the final surface. Then the calculate volume routines will report the quantities between alluvial soil and design. Also the Display->Cut/Fill Color Map routine will make the map for the difference between the alluvial soil and design surfaces.

These user-defined surfaces can be created using the Add Target function in the Define Layer Target/Material/Subgrade command combined with the Make User-Defined Surface command.

Prerequisite: a surface model

Keyboard Command: set_active_tins

Design Surface Vertical Offset

This command can be used to lower or raise the design surface within a defined perimeter or by the entire surface.

Prerequisite: a design surface

Keyboard Command: adjust_final

Existing Surface Vertical Offset

This command can be used to lower or raise the existing surface within a defined perimeter or by the entire surface.

Prerequisite: an existing surface

Keyboard Command: adjust_exist

Merge Existing With Design

This command allows you to merge the existing surface with design surface within perimeter polylines. The resulting merged surface can be saved to update either the Existing or Design surfaces. The program prompts for inclusion and exclusion perimeter polylines. These polylines must be closed. The merge will be applied inside the inclusion perimeters and not inside the exclusion perimeters. The exclusion perimeters are optional.

For example, if a portion of the site is completed, you can update the existing surface to match the design for the completed area. First, draw a closed polyline around the completed area. Then run Merge Existing With Design and choose the merge results target as Existing. Then select the perimeter polyline.



Pulldown Menu Location: Takeoff > Surface Tools

Prerequisite: existing and design surfaces and an inclusion perimeter polyline

Keyboard Command: merge_final

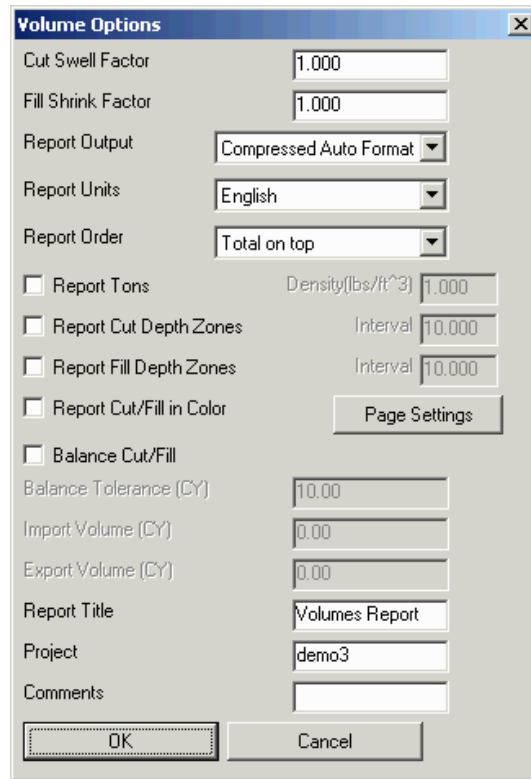
Calculate Total Volumes

Use this command to report total volume calculations within the site boundary polyline. The report includes the cut and fill quantities, slope and horizontal area, average and max strata cut depth and max cut/fill depths and locations. Also in the report, strata and topsoil quantities if the site has strata and topsoil defined. Besides reporting the total quantities for the site boundary, Area Of Interest polylines can be used to report quantities within named perimeters.

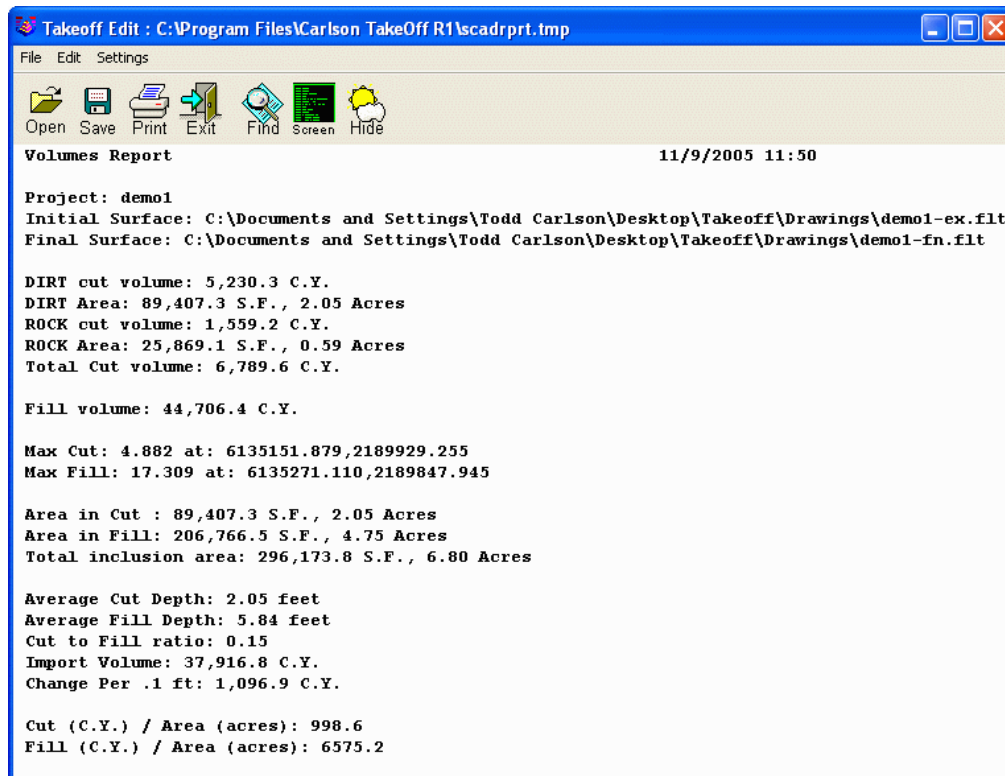
Before running this command, the existing and design surfaces must be created and the boundary polyline must be assigned. Also, the strata surfaces, topsoil and Area Of Interest polylines need to be set before this command if those features are to be reported.

The Volume Options dialog box shown here offers options for the final report. Here you can select four different types of reports: Standard Report Viewer, Custom Report Formatter, Expanded Auto Format, and Compressed Auto Format. The Cut Swell Factor is multiplied by the cut volume and the Fill Shrink Factor is multiplied by the fill volume. Report Cut/Fill Depth Zones breaks the Cut/Fill volumes up by user-defined depth intervals. The Report Units setting chooses between English and Metric quantities for the report. In Drawing Setup in Takeoff, you set the drawing units as either English or Metric. The Report Units will default to match the drawing units but you can change the Report Units to the other mode and the program will apply the conversion between English and Metric for the report. So you can have a drawing in English units and create a report with Metric quantities.

Note: As the quantities are calculated within each area, the area is hatched with a solid fill as a visual verification that the right area is being processed.

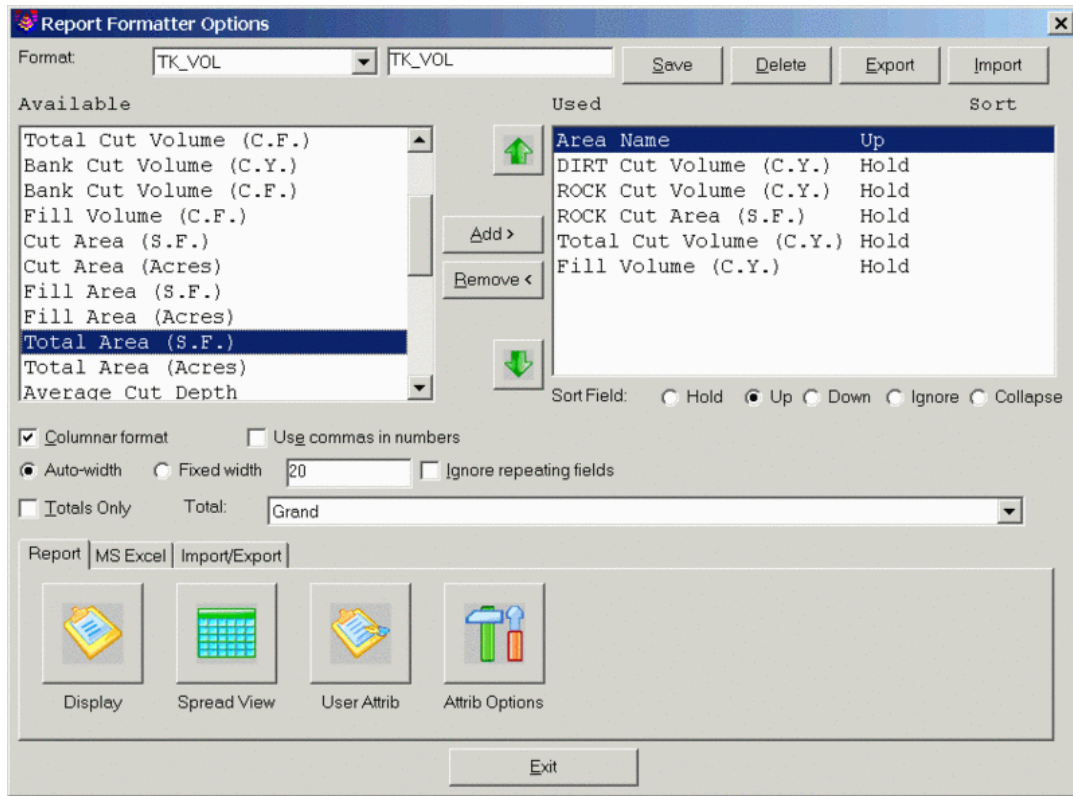


Shown here is an example of a Standard Report Viewer.

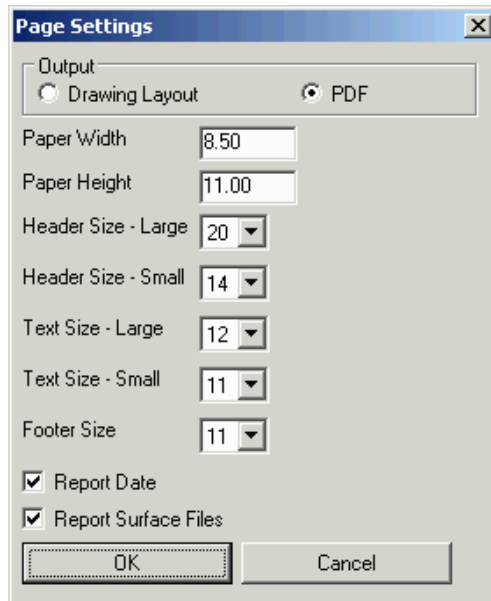


Use Customs Report Formatter to customize or "user define" the reporting options. The Report Formatter Options

dialog box shown here offers a variety of output options including Excel. You can choose the fields to report from the Available list and set their report order under the Used list.



For the Compress and the Expanded Auto Formats, the Report Cut/Fill in Color option will color the cut values in red and the fill values in blue in the report. Also, the Page Settings button has more controls for these auto formats including whether to output the report to PDF or to a drawing layout.



The Expanded Auto Format is shown below.

Volumes Report

Project: Takeoffdemo2

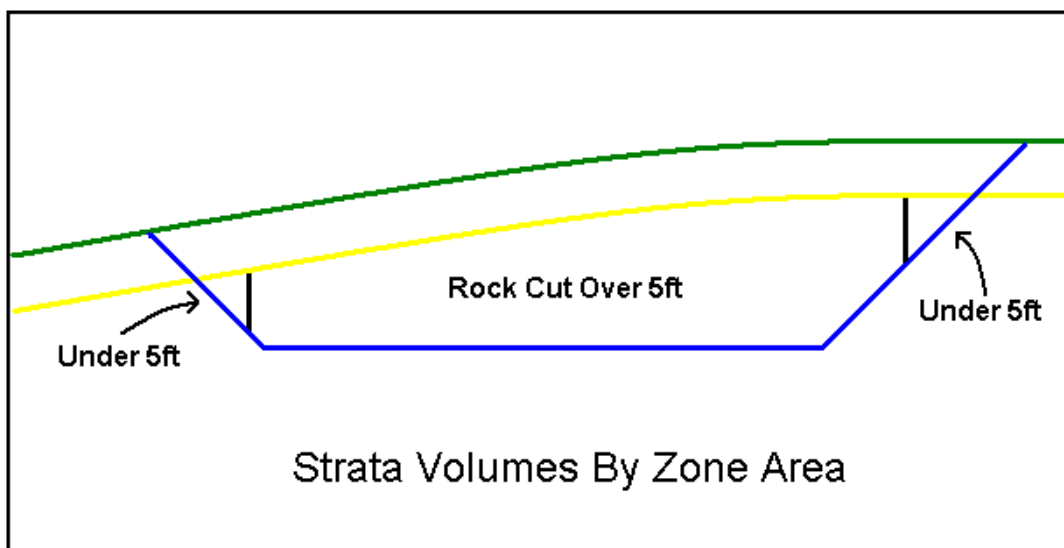
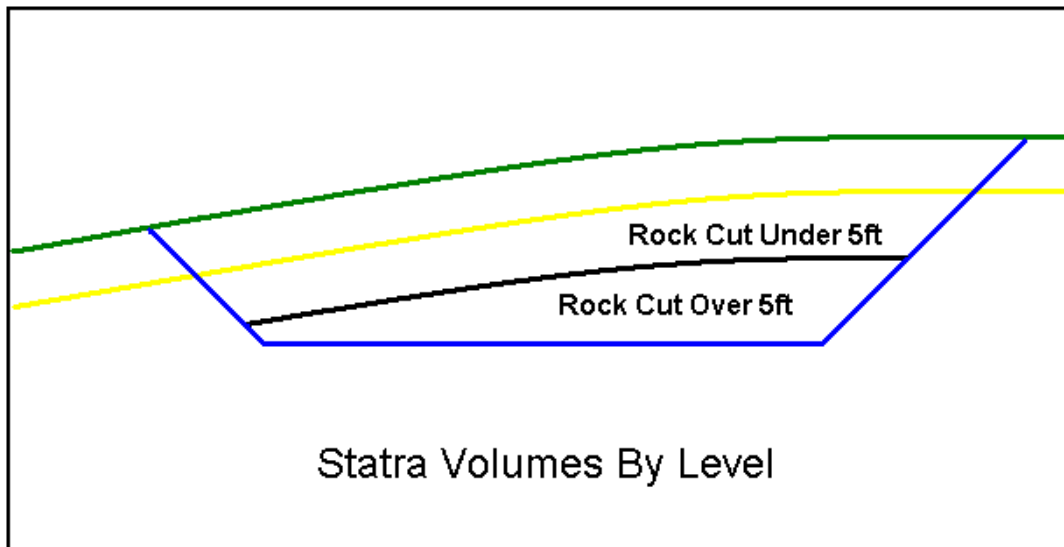
Date: 03/24/2008 09:59

Initial Surface: C:\Carlson Projects\Takeoffdemo2-ex.tin

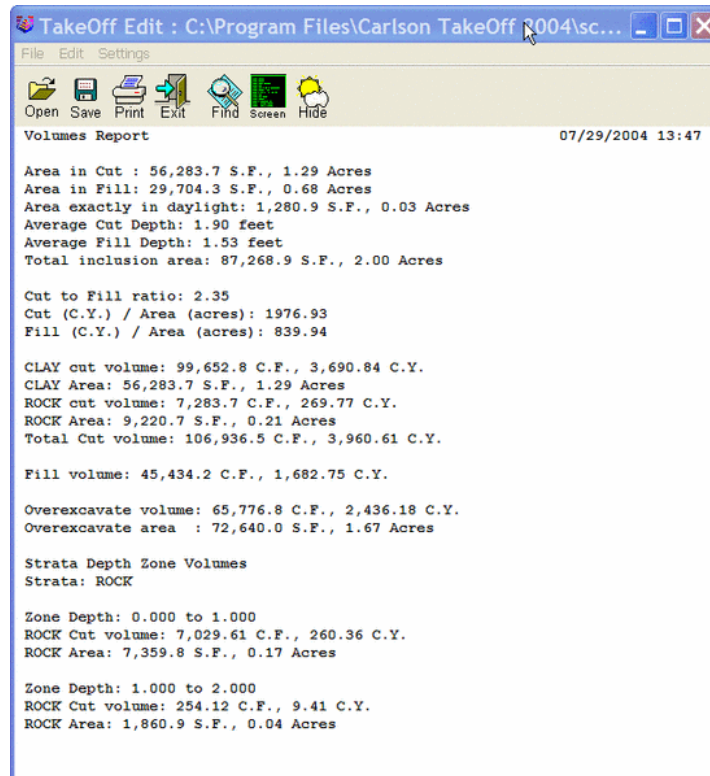
Final Surface: C:\Carlson Projects\Takeoffdemo2-fn.tin

	Bank Volume (CY)	Swell/ Shrink	Adjusted Volume (CY)	Area (SqFt)	Average Depth	Change Per .1 FT	Import Volume (CY)
Cut	5451.7	1.000	5451.7	51661.6	2.85	403.6	
Fill	5839.4	1.000	5839.4	57304.2	2.75	403.6	387.7

If drillholes have been located on the drawing and strata types and depths have been defined, a calculate Strata Depth Zones Volume option becomes available. Here strata volumes are broken down by user-specified depth intervals. The depths are either determined horizontally (By Level) or by the area of the deepest cut (By Zone Area).



Shown here is an example of the report if strata depth intervals have been defined.



The Balance Cut/Fill option vertically raises or lowers the design surface to balance the cut and fill. The report states how much the design surface was adjusted for the balance. When the drawing has grade entities, these entities can be adjusted for the balance. For example, pad polylines created by the Elevate Pads By Grade Rules can be raised or lowered for the balance. After the volumes report, the program asks whether to save the changes for the balance. This will update the grade entities and the design surface files for Design Surface After Zones/Topsoil (-sg) and Design Surface After Subgrade Zones (-zn).

With the balance routine, you can set import or export volumes. Use these options if extra material is available or needed elsewhere.

Pulldown Menu Location: Takeoff

Prerequisite: Existing and design surfaces and a boundary polyline

Keyboard Command: tin_volume

Calculate Volumes Inside Perimeter

Use this command to create volume reports inside the selected closed perimeter polyline. The same reporting options are available for this command as are for the Calculate Total Volumes command.

Keyboard Command: tin_volume2

Prerequisite: Existing and Design surfaces and a closed perimeter polyline

Draw 3DPoly Perimeter

This command draws a 3D polyline in the PERIMETER layer. This polyline is required by the *Calculate Stockpile Volume* and *Calculate Pond/Pit Volume* routines. In these routines, this polyline is used as the inclusion perimeter

for volumes. You may want to set your AutoCAD Object Snap prior to running this routine so that you obtain the elevations of existing points while creating the 3D polyline.

Prompts

Select the 3Dpolyline options you want and press enter.

Command: 3dperim

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: (Pick)

Interpolate/screen Pick/<Elevation> <0.00>: 818

Z: 818.00

[Arc/Close/Direction/Follow/Undo/<Pick point or point numbers>]: (Pick)

Slope/Ratio/Interpolate/Degree/screen Pick/<Elevation> <0.00>: 814

Z: 814.00, Hz dist: 60.64, Slope dist: 60.77, Slope: -6.6% Ratio: -15.2:1

[Arc/Close/Direction/Extend/Follow/Undo/<Pick point or point numbers>]: (Pick)

Slope/Ratio/Interpolate/Degree/screen Pick/<Elevation> <0.00>: 815

Z: 815.00, Hz dist: 26.32, Slope dist: 26.33, Slope: 3.8% Ratio: 26.3:1

[Arc/Close/Direction/Extend/Follow/Undo/<Pick point or point numbers>]: c for close

Pull-Down Menu Location: Tools-> Surface Tools

Keyboard Command: 3dperim

Prerequisite: None

Draw 3DPoly Base Breakline

This command draws a 3D polyline in the BASE_BREAKLINE layer. This polyline is used by the *Calculate Stockpile Volume* and *Calculate Pond/Pit Volume* routines to model the base surface. You may want to set your AutoCAD Object Snap prior to running this routine so that you obtain the elevations of existing points while creating the 3D polyline.

Prompts

Select the 3Dpolyline options you want and press enter.

command: 3DBASE

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: (Pick)

Interpolate/screen Pick/<Elevation> <0.00>: 818

Z: 818.00

[Arc/Close/Direction/Follow/Undo/<Pick point or point numbers>]: (Pick)

Slope/Ratio/Interpolate/Degree/screen Pick/<Elevation> <0.00>: 814

Z: 814.00, Hz dist: 60.64, Slope dist: 60.77, Slope: -6.6% Ratio: -15.2:1

[Arc/Close/Direction/Extend/Follow/Undo/<Pick point or point numbers>]: (Pick)

Slope/Ratio/Interpolate/Degree/screen Pick/<Elevation> <0.00>: 815

Z: 815.00, Hz dist: 26.32, Slope dist: 26.33, Slope: 3.8% Ratio: 26.3:1

[Arc/Close/Direction/Extend/Follow/Undo/<Pick point or point numbers>]: (Enter)

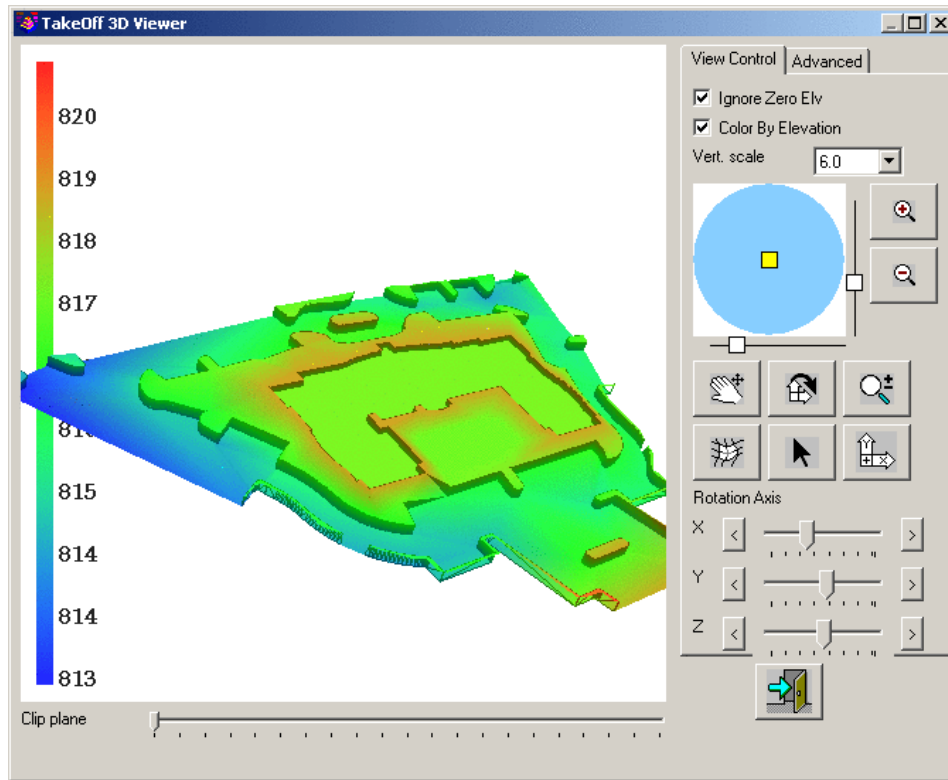
Pull-Down Menu Location: Tools->Surface Tools

Prerequisite: None

Keyboard Command: 3dbase


Design Surface 3D Viewer


This command allows you to view the design surface in 3D mode.






In the top right of the control bar you can check to Ignore Zero Elev and Color By Elevation and change the Vertical Scale. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Color By Elevation shows elevation change with the change of colors. Note: Color By Elevation is used in the above example. To adjust the color use the color circle on the right.

The magnify glass icons can be used to zoom in and out. Click on the plus magnify glass to zoom in and

the minus magnify glass to zoom out. With the  icon click and drag up to zoom in and drag down to zoom out. The hand icon below the color circle allows you to pan around the viewer. Click and drag the direction you

want to move. The  icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point

according to the z axis. The  icon toggles on and off the shading of the surface (the shading is shown in the above drawing). The arrow icon reports the elevations at the bottom of the screen as you move around the surface.

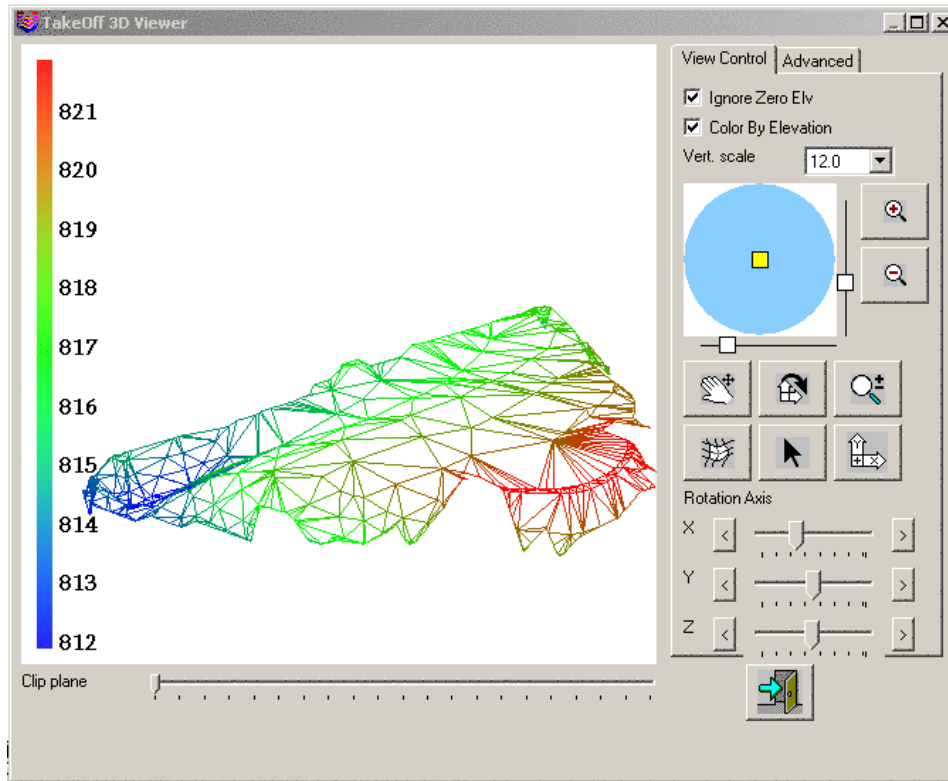
The  icon restores the surface viewpoint to flat. The  icon exits 3D Driver Simulation.

Rotation Axis: These three control bars rotate the surface around the x, y, and z axis. Clip plane trims the size of the surface shown in the viewer.

Prerequisite: a design surface
Keyboard Command: cube_design


Existing Surface 3D Viewer


This command allows you to view the existing surface in 3D mode.





In the top right of the control bar you can check to Ignore Zero Elev and Color By Elevation and change the Vertical Scale. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Color By Elevation shows elevation change with the change of colors. Note: Color By Elevation is used in the above example. To adjust the color use the color circle on the right.

The magnify glass icons can be used to zoom in and out. Click on the plus magnify glass to zoom in and

the minus magnify glass to zoom out. With the  icon click and drag up to zoom in and drag down to zoom out. The hand icon below the color circle allows you to pan around the viewer. Click and drag the direction you want

to move. The  icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point according

to the z axis. The  icon toggles on and off the shading of the surface. The arrow icon reports the elevations

at the bottom of the screen as you move around the surface. The  icon restores the surface viewpoint to flat.

The  icon exits 3D Driver Simulation.

Rotation Axis: These three control bars rotate the surface around the x, y, and z axis. Clip plane trims the size of the surface shown in the viewer.

Prerequisite: an existing surface

Keyboard Command: cube_exist

Import MicroStation DGN

This command takes a MicroStation DGN file and imports the MicroStation elements into your current DWG drawing. After selecting the DGN file you want to import, you will receive a conversion report like the one below:

```
Element
Type Name Encountered Converted
3 Line3d 273 273
4 LineString3d 48 48
6 Shape3d 1 1
12 ComplexString 321 321
17 Text3d 131 131
-1 Model 1 1
775 total 775 converted elements in 1 seconds.
```

After the command has imported the DGN file, run View > Zoom > Extents to see the converted entities.

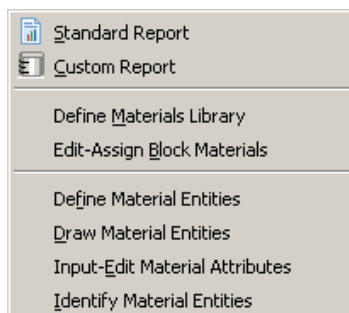
Pulldown Menu Location: Tools > Import/Export

Prerequisite: a MicroStation DGN file

Keyboard Command: dgn_in

Material Quantities

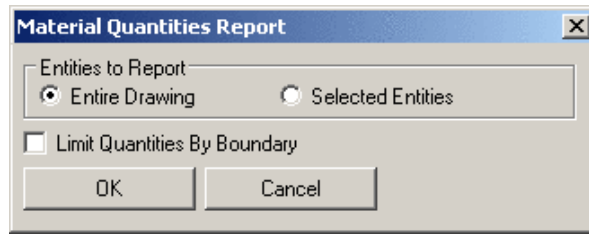
The Material Quantities flyout shown here offers many options for quantity reporting including standard and custom report options. User-defined attributes can also be drawn, edited, and identified from the different command options.



Material Quantities draws heavily from the command Define Layer Target/Material/Subgrade found in the Takeoff and SiteNET pull-down menus. Material Quantities are calculated from the entities in the drawing. Several entity properties can be reported including entity count, length, area, and volume. The type of material for each entity is determined by the layer for the entity. Use Define Layer Target/Material/Subgrade, to assign the material types by layer before running the Material Quantities Standard or Custom Report.

Standard Report

Use this command to display all or a selected set of material quantities in present in the drawing. Quantities can be limited by the established boundary polyline(s) or can look outside your boundary area for materials that might be outside of your Cut/Fill limits.



The command will then report out the materials found in the drawing based off of settings made in Define Layer Target/Material/Subgrade. This text report can be edited, printed, or put onto the screen.

Takeoff Edit : C:\Users\Todd Carlson\AppData\Roaming\Carlson Software\Carlson Takeoff R7\R2014\enu\U... Wed Jun 04 10:47:01 2014

File Edit Settings

Open Save Print Exit Find Screen Hide

Material Quantities

Breakout Quantities

Material	Count	Length(Ft)	Area(S.F.)	Volume(C.Y.)	Shrink	Section /Depth	Cost
Paving	5	6686.0	67262.2			1.92ft	
Asphalt				1245.6	1.000	0.50ft	\$121,912.65
Sand				1669.1	1.000	0.67ft	\$21,631.51
Road Agg				1868.4	1.000	0.75ft	\$49,437.68
Z_PAD	21	7479.6	145445.5			2.42ft	
Concrete				3609.2	1.000	0.67ft	\$288,736.21
Sand				4040.2	1.000	0.75ft	\$52,360.37
Building Agg				5386.9	1.000	1.00ft	\$142,536.57

Sub-Total Quantities

Material	Count	Length(Ft)	Area(S.F.)	Volume(C.Y.)	Tons	Cost
Asphalt	5	6686.0	67262.2	1245.6	2438.3	\$121,912.65
Building Agg	21	7479.6	145445.5	5386.9	10181.2	\$142,536.57
Concrete	21	7479.6	145445.5	3609.2		\$288,736.21
Road Agg	5	6686.0	67262.2	1868.4	3531.3	\$49,437.68
Sand	26	14165.6	212707.6	5709.3	9249.0	\$73,991.88

Total Material Cost: \$676,614.99

Material Parameters

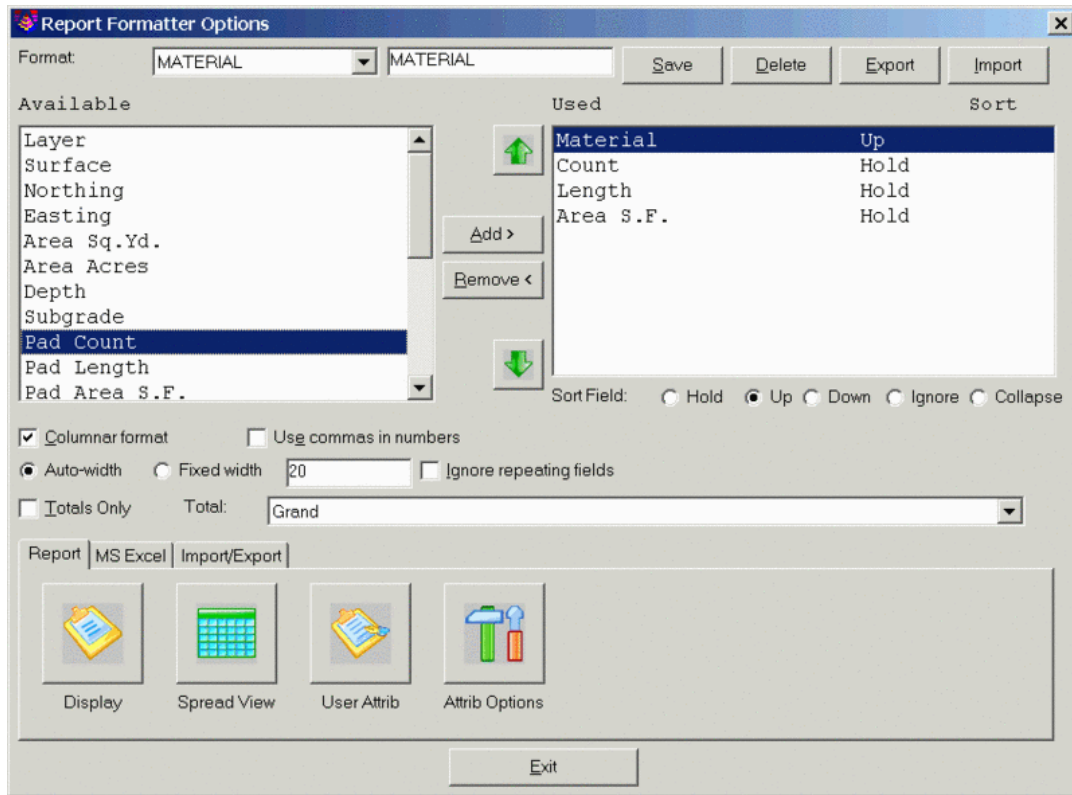
Material	Unit Cost	Density (lbs/ft^3)
Asphalt	\$50.00/Ton	145.00
Building Agg	\$14.00/Ton	140.00
Concrete	\$80.00/Volume(CY)	
Road Agg	\$14.00/Ton	140.00
Sand	\$8.00/Ton	120.00

Prerequisite: Defined materials

Keyboard Command: materials_report2

Custom Report

Use this command to customize or "user define" the reporting options. This command first prompts whether to report quantities for all the entities in the drawing or selected entities. Then if the drawing contains Areas Of Interest polylines, there is an option report quantities by these areas which adds the area name to the available report fields to allow sorting and grouping by area name. The Report Formatter Options dialog box shown here offers a variety of output options. You can choose the fields to report from the Available list and put them in report order under the Used list.



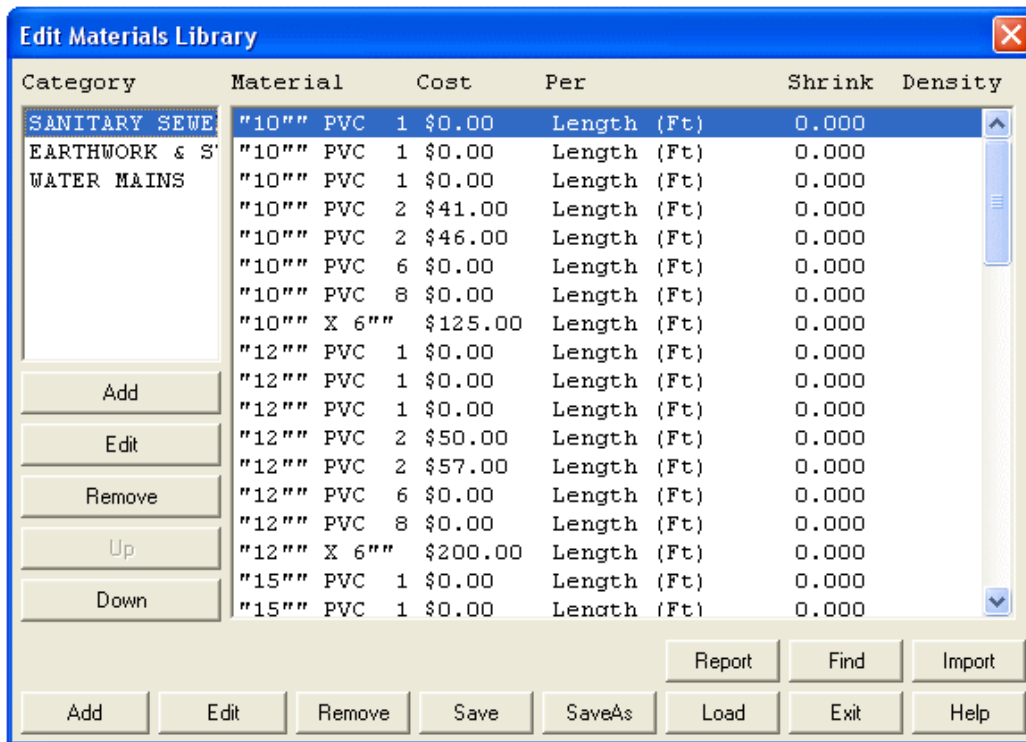
Selecting the Display option shows the report in the standard Carlson Takeoff report viewer. Reports can be exported to an Excel spreadsheet as well.

Prerequisite: Defined materials

Keyboard Command: materials_report

Define Materials Library

Define Materials Library allows you to Add, Remove, Load, Save, and Report a list of material costs. Costs can be set per area, count, volume, ton, or length by using the Edit function at the bottom of the dialog. The left side of the dialog can be used to set categories for different material costs. Material costs can also be Imported from user-specified text files (.txt, .dat, or .csv). Once this library is defined, the items can be used in Define Layer Target/Material/Subgrade and reported out with the Material Quantities standard and custom reports.

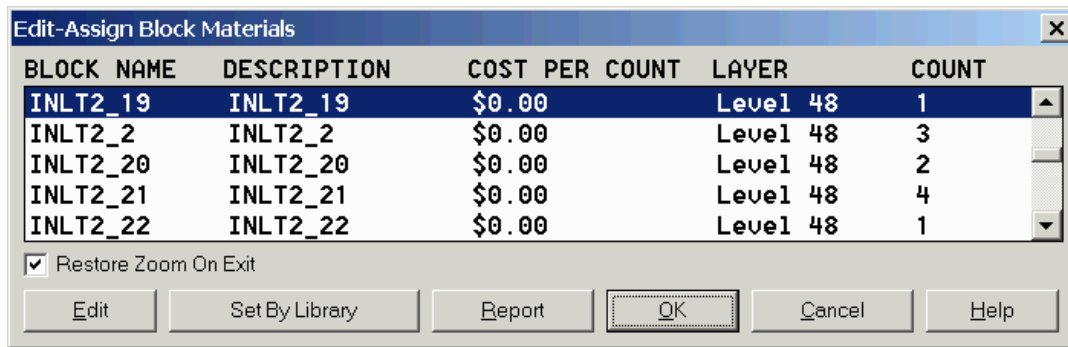


Prerequisite: pricing for materials

Keyboard Command: define_tk_materials

Block Materials Editor

This command scans the current drawing to find and report block/symbol names and their count. For example, when the drawing contains different symbols for different types of utilities, this command identifies each type of symbol and the number.



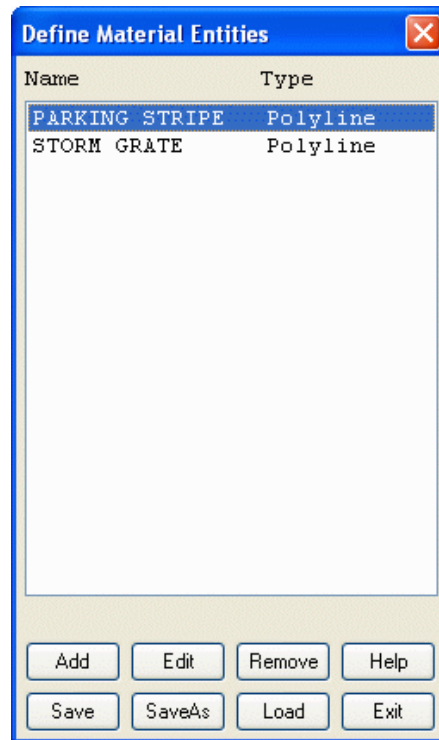
From this command, you can set the Description and Cost of the block by using the Edit button. You can also set the Description and Cost by predefined Materials by using the Set By Library button. When a block name is highlighted from the list, the drawing is zoomed to the location of one of those blocks so that you can see what it looks like. To Report these materials as part of the Standard Report, check on Include Materials Quantities Report in the Edit Materials dialog of the block layer found in the Define Layer Target/Material/Subgrade command. You can also just click on the Report button for a simple report.

Prerequisite: Blocks

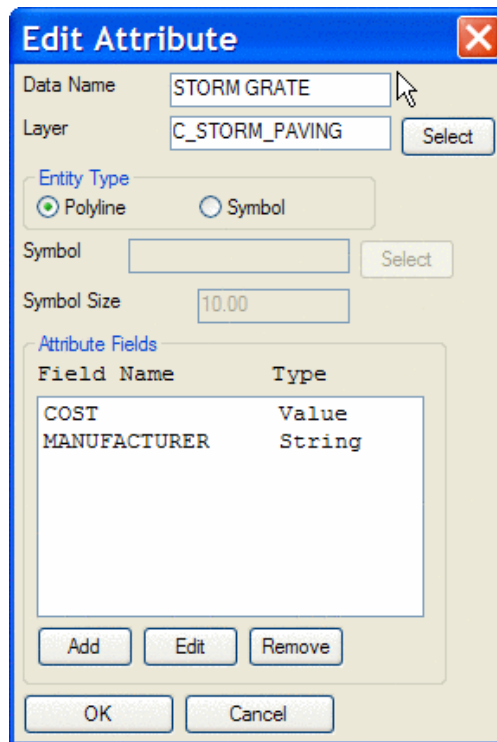
Keyboard Command: edit_all_blocks

Define Material Attributes

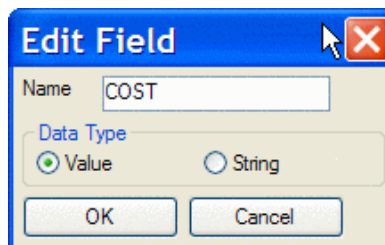
Use this command to create user-defined attributes that can be assigned to objects in the drawing. Once attributes have been defined, use Material Entities Editor to add the attributes to existing linework or Draw Material Entities to create new linework. For reporting, the material attributes defined in this command will be shown as Available items in the Material Quantities Custom Report command.



The Define Attribute dialog box shown here allows the user to "Add", "Edit", or "Remove" attributes and save the definitions for later use. Simply "Load" a saved attribute definition file with the "tkd" extension for future use.



Selecting the Add or Edit options produce the edit attribute dialog box shown here. Use this command to define the Data name and the layer the objects currently reside on and the layer that future objects will be drawn on. Two entity types can be used, polyline data or point data. If the symbol option is selected the user has the option of which symbol will represent the object. Attribute fields must be defined for material reporting.



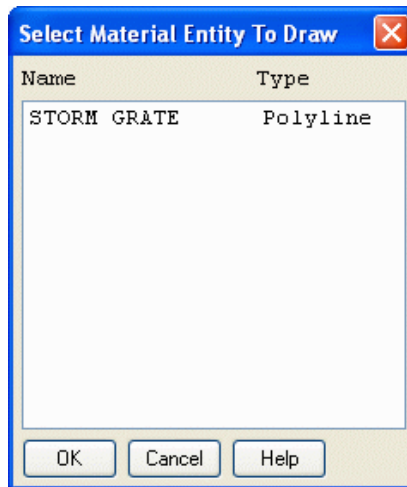
Selecting the Add or Edit button on the Edit Attribute dialog box brings up the Edit Field dialog box shown here. Use this dialog to define the field name and type. If the Value option is selected, only numeric values will be allowed when prompted. If the String option is selected, the user will have the ability to type in a text message when prompted.

Prerequisite: attributes

Keyboard Command: define_tk_data

Draw Materials Entities

Use this command to apply attribute data to objects as you draw or digitize them. Select the predefined attribute type to draw from the list available in the Select Attribute to Draw dialog box shown here.



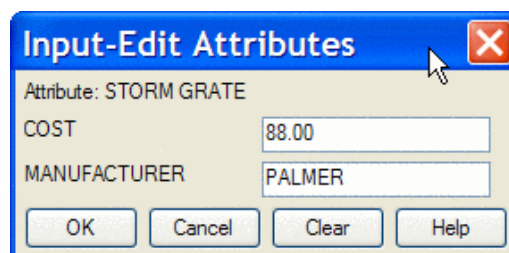
The command line will prompt the user to pick the points of the desired location of the object and allow the attribute data fields to be filled out upon completion or each "enter".

Prerequisite: defined attributes

Keyboard Command: draw_tk_data

Material Attributes Editor

Use this command to assign predefined attribute information to an object already existing in the drawing. The command line prompt will require the user to select the object that attribute information is to be applied, and offer the Attribute dialog box shown here. This dialog box will display all predefined fields for that particular attribute type.



Prerequisite: predefined attribute information

Keyboard Command: edit_tk_data

Identify Materials Entities

Use this command to display all the objects that have attribute data assignments. The user will have the options of selecting the objects by picking them individually or by searching the entire drawing database. The objects that have attribute information assignments will "highlight" on the screen and the command line will display the attribute information.

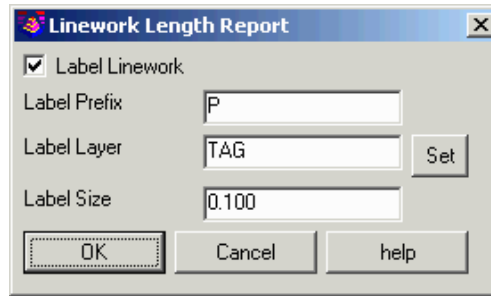
Prerequisite: attributes

Keyboard Command: id_tk_data

Linework Length Report

This command reports the length and properties of selected linework. When the layer for the linework is defined with a linear cost in the Define Layer Surface/Material Subgrade command, then the report will report costs. The Material name for the report also is defined in Define Layer Surface/Material Subgrade.

This command has an option to draw labels next to the linework. The dialog has settings for the label layer, prefix and size.



Length Report

Tag	Material	Length	Cost	Layer
P1	Edge	235.1	\$775.79	PRLCST
P2	Building Base	920.3	\$2300.63	PAD
P3	Parking Curb	1273.0	\$2864.20	PR-FC-CURB
P4	Parking Curb	1817.6	\$4089.61	PR-FC-CURB
P5	Parking Curb	84.7	\$190.59	PR-FC-CURB
P6	Parking Curb	107.6	\$242.00	PR-FC-CURB
Total		4438.2	\$10462.81	

Pulldown Menu Location: Takeoff > Material Quantities

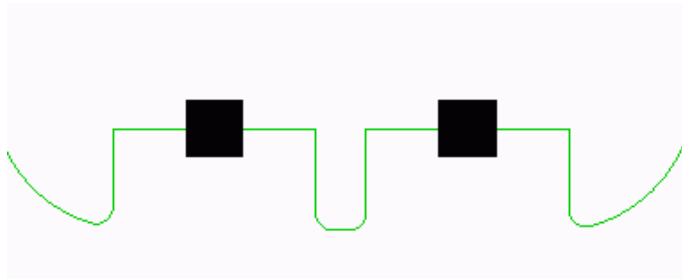
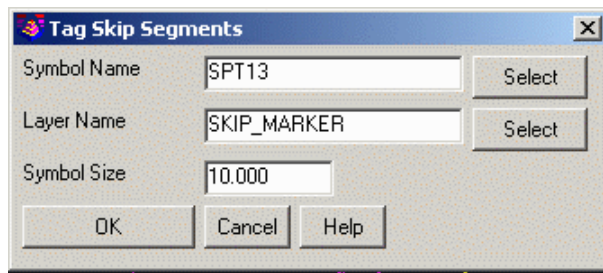
Keyboard Command: length_report

Prerequisite: linework

Tag Segments To Skip Linear

This command tags segments of polylines to skip for material linear quantities. This command applies to materials that are both used for volume/area and linear quantities. In this case, the polyline needs to be closed for the volume/area quantities but when there is a portion of the polyline to exclude from the linear quantities, then this command can be used. For example, curb quantities can use a closed polyline for reporting the pavement volume/area and the length of the polyline for reporting the length of curbing. When there is an entrance for pavement area, there is a gap in the curb but the closed polyline goes across any gaps. So the segments across gaps can be tagged to exclude for the linear curb length report.

The program draws a symbol on these segments to skip. The program prompts for the symbol name, layer and size to use for the tags. Then the Material Quantities report commands will automatically pick up these tags.



Two segments tagged to exclude from linear quantities for curb polyline at entrances to parking area

Prompts

Tag Skip Segments

Pick segment to skip: *pick a point on the polyline*

Pick segment to skip (Enter to end): *press Enter*

Pulldown Menu Location: Takeoff > Material Quantities

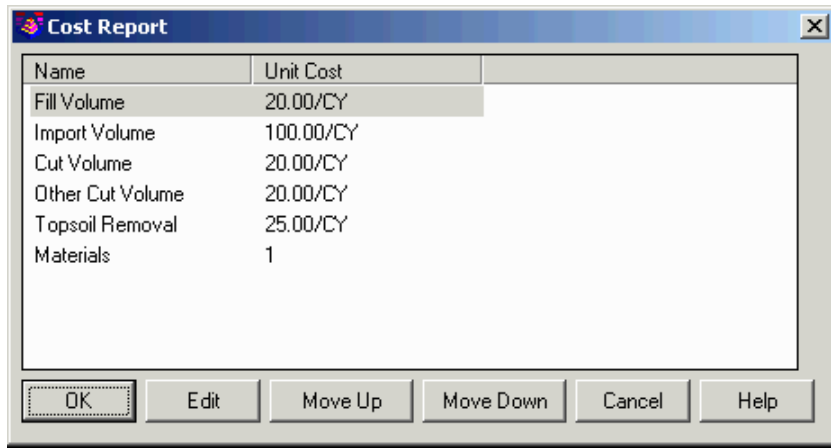
Prerequisite: a closed polyline

Keyboard Command: tag_skips

Cost Summary Report

This command reports the combined costs from the quantities from the Calculate Total Volumes and Material Quantities routines. For the volume quantities, there are separate costs for the different types of volumes. For the material quantities, the report just has the total material cost. For detail material costs, use the Material Quantities routines.

In the dialog, use the Edit button to set the unit cost and the report name. Use the Move Up and Down functions to change the report order. After picking OK, the report data is shown in the Report Formatter for setting the report style and output method such as Excel.



Cost Summary

Name	Cost	Amount	Rate
Fill Volume	\$88,173.85	4,408.7	\$20.00/CY
Import Volume	\$103,218.02	1,032.2	\$100.00/CY
Cut Volume	\$67,530.25	3,376.5	\$20.00/CY
Other Cut Volume	\$141,351.69	7,067.6	\$20.00/CY
Topsoil Removal	\$106,025.27	4,241.0	\$25.00/CY
Materials	\$70,144.21		
Total	\$576,443.29		

Pulldown Menu Location: Takeoff

Keyboard Command: tk_cost

Prerequisite: Calculate Total Volumes or Material Quantities

Make Layout

This command creates a layout with a title block and optionally creates a model view. In the dialog, choose the **Printer/Plotter** and **Paper Size** that the program will use to set the print area on the layout.

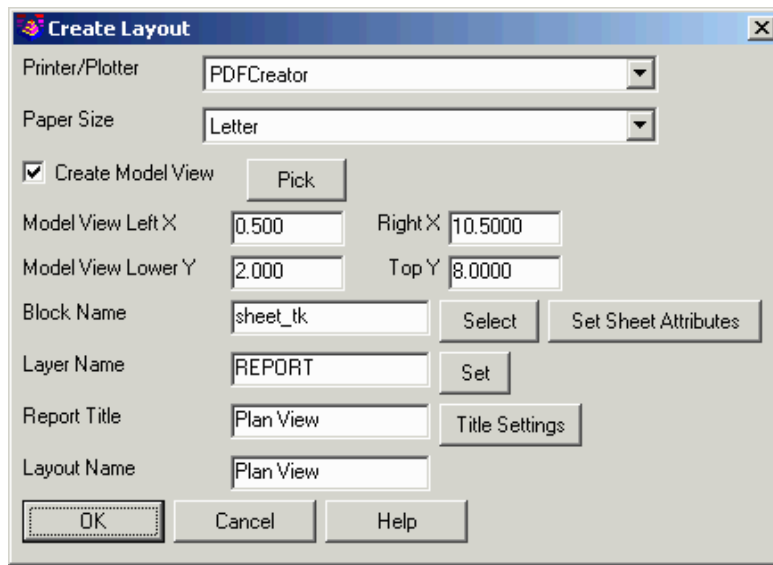
The **Create Model View** option makes a viewport on the layout. The Model View X/Y settings are in paper units and define the borders of the model view window. For the model view, the program will automatically size the drawing to fit in the viewport by using the boundary polyline from the Set Boundary Polyline command.

The **Block Name** sets the title block to use. You can use a custom drawing that has your company logo and title block. The **Set Sheet Attributes** defines how to fill out any block attributes defined in the block drawing.

The **Layer Name** is used for the layer of the block and report title label in the layout.

The **Report Title** is the label to draw in the layout at the position defined under Title Settings.

The **Layout Name** is the name for the new layout to create.



Pulldown Menu Location: Takeoff > Graphic Reports

Keyboard Command: make_layout

Prerequisite: None

Place Image On Layout

This command inserts an image in the drawing to fit within a view window. The image is cropped to fit the window. This command is a way to handle an image with a different aspect ratio than the window area for a report. The image is sized to fill the whole window area. For example, you can use the screen shot function in the 3D Viewer Window to make an image of your surface model and then use this command to fit this image onto a report.

After selecting the image file, the program prompts to pick the corners of the window for the image in the drawing. Then the dialog shows the image and four white boxes for the corner points for cropping. Click and drag the corner points to move and resize these corner points. When you click in the middle, all the corners move together. When you click near a corner that corner point can be moved in or out to resize.

The program creates a new image file for the adjusted image. This file is named after the original file name with ".fit" added. The original image file is not changed.



Prompts

Select Image File

Pick 1st corner for image: *pick a point*

Pick other corner for image: *pick a point*

Fit Image dialog

Pulldown Menu Location: Takeoff > Graphic Reports

Keyboard Command: image_layout

Prerequisite: Image file

Save Layout Layers

This command saves the current freeze/thaw status of the layers for the current layout. When the Output Layouts To PDF command generates the PDF for the layout, the saved layout layer settings are used for the PDF output. The Restore Layout Layers command can also be used to reset the freeze/thaw status for the layout.

This command is useful when making a layout using certain layers and then later making new layers in the drawing and you want the layout to use only the original layers.

Pulldown Menu Location: Takeoff > Graphic Reports

Keyboard Command: save_layout

Prerequisite: None

Restore Layout Layers

This command sets the freeze/thaw status of the layers for the current layout using the settings stored by the Save Layout Layers command.

Pulldown Menu Location: Takeoff > Graphic Reports

Keyboard Command: restore_layout

Prerequisite: Save Layout Layers

Convert Dashes To Polylines

This command connects sequential dashes into polylines. There are two processing methods. The Individual method creates one polyline at a time. The Selection method processes a selection set of dashes and creates multiple polylines. The program will prompt you to select a sample and adjacent dash line. This identifies the layer and approximate gap between dash lines. It is helpful to isolate this layer before running the command. Next, select the dash lines to process and press enter. Below are typical results of the conversion.



Before Conversion



After Conversion

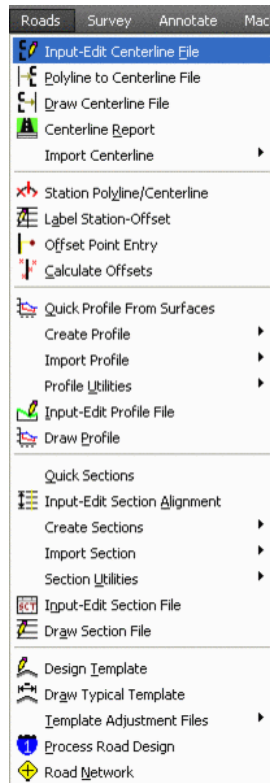
Pulldown Menu Location: Edit > Polyline Utilities

Keyboard Command: dash2pl

Prerequisite: Dashed linework to convert into continuous polylines.

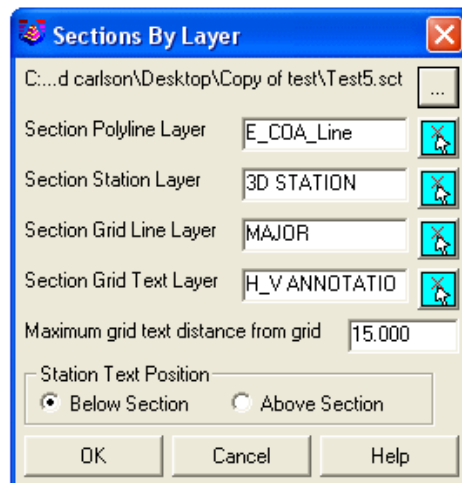
Roads Menu

The Roads pull-down is a composite of commands taken from Carlson Civil. The pull-down includes centerline, vertical profile, cross-sectioning, and Road Network functions for estimating, plan view representation, and the 3D modeling of roads and highways. All of the commands for Roads are described in the manual under the Civil chapter.

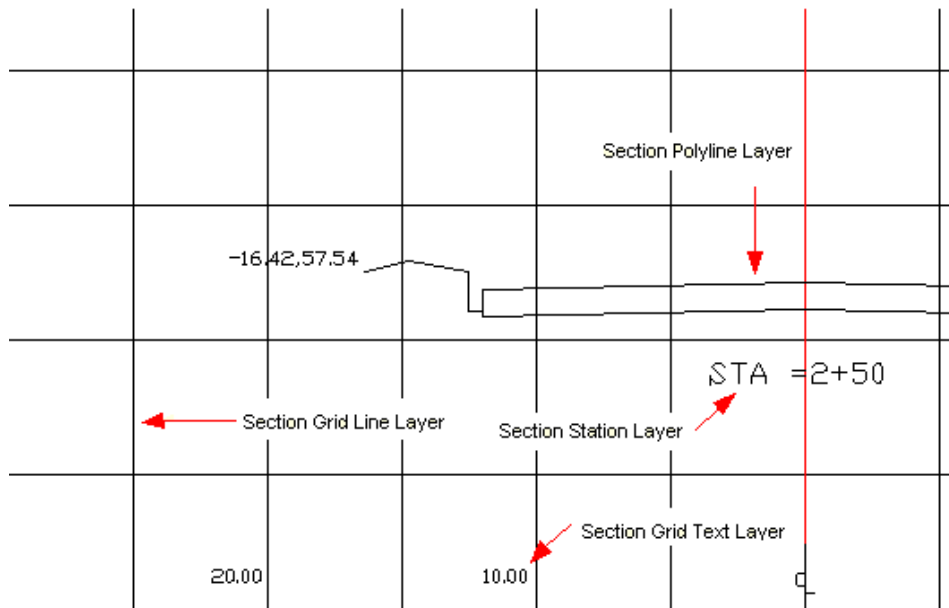


Sections From Polylines On Section Grids

Section From Polylines By Layer command can be used to automatically import section data from drawing entities by selecting samples of the different section elements. When running the command, you will first be prompted to name and save the section file you are about to create. The next dialog shown below will ask you to select the layers that represent different elements of the section. Use the mouse pick on the right side to pick the layer in the plan view, or type in the layer name in the middle column.



Here is an example of the linework needed for Section From Polylines.



The "Maximum grid text distance from grid" allows you to determine how far from the grid linework the routine searches in order to find text. The program uses the grid linework and the text associated with the linework to calculate the scale of the grid. Station Text Position tells the routine to assign stationing from text either Above or Below the section linework. After saying okay to this dialog, the routine will prompt you to select the entities to process. You can pick on linework, window around a group of linework or type in All.

Pulldown Menu Location: Roads > Create Sections

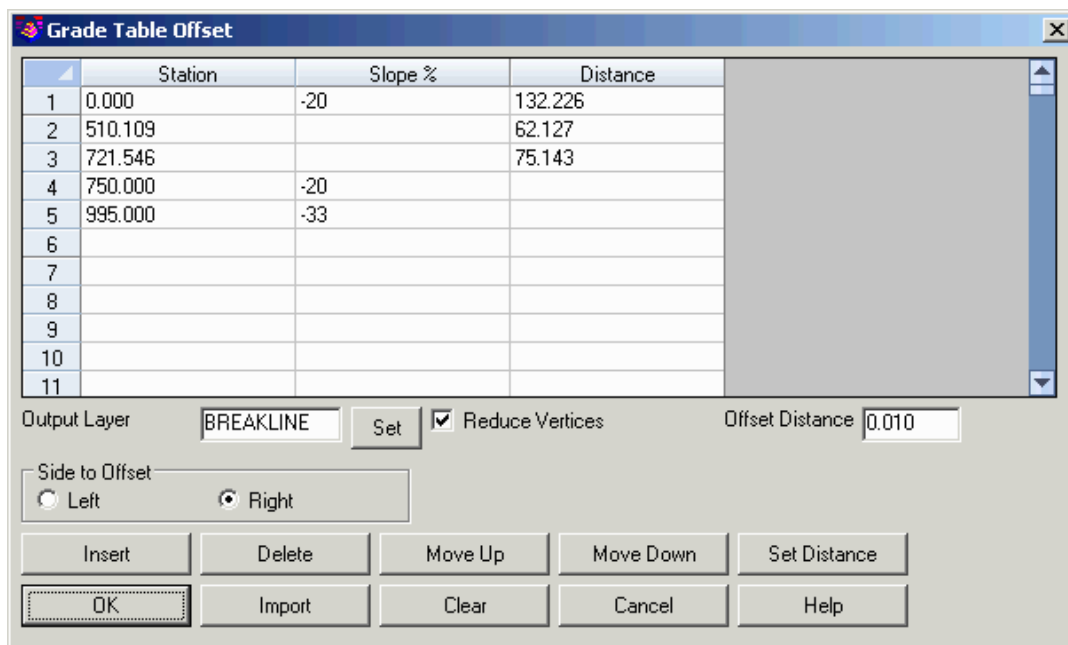
Prerequisite: dwg with section grids

Keyboard Command: scfsct

Offset 3D Polyline By Grade Table

This command creates a 3D polyline that is offset from a reference 3D polyline using the distances and slopes from a Grade Table file. This command provides a simple and easy way to handle complex offsets for features like bottom of ditch polylines offset from the road edge-of-pavement in a road design with transitions.

The Grade Table defines the slopes and offset distances at different stations along the reference polyline. Slopes and offsets for polyline segments between defined stations in the list are interpolated. For example, if the distance at station 0 is 10 and the distance at station 100 is 14, then the offset distance for a polyline vertice at station 50 would be 12. Then slopes and distances are interpolated separately. So if there is a slope change at a station but the distance doesn't change at that station, then you only need to make an entry in the table with the station and slope and leave the distance blank.



The **Output Layer** is used for the offset 3D polyline that is created. The **Side To Offset** controls the offset direction from the reference polyline. The **Reduce Vertices** is used to reduce the number of vertices in the offset polyline. The **Offset Distance** is the maximum amount that the polyline can move horizontally when removing a point. To not reduce vertices, set this Offset Distance to zero.

The **Set Distance** function has several methods for filling out the distance in the table.

Polyline method: prompts to pick the reference and target polylines and then figures the different distances between them. This method applies when there is a 2D polyline for the target location of the 3D offset.

Profile method: prompts for the reference 3D polyline and a target profile. The slope must already be defined in the Grade Table. This method uses the elevation difference between the reference 3D polyline and a profile plus the slope to fill out the distances.

Surface method: prompt for the reference 3D polyline and the target surface. The slope must already be defined in the Grade Table. This method finds the intersection of the surface from the reference 3D polyline at the slope to fill out the distances.

Section method: prompt for the reference 3D polyline and a cross section file. The reference polyline should be at offset zero of the cross sections. The slope must already be defined in the Grade Table. This method finds the intersection of the cross section from the reference 3D polyline at the slope to fill out the distances.

The **Import** function reads in station, slope and distance data into the table. The data can come from either a text file, drawing graphics or superelevation file. For the text file, the format should have station, slope% and distance separated by a delimiter such as a comma. For the drawing graphics, the import reads a polyline on a superelevation diagram grid to set the transition slopes. For superelevation file method, the import reads the transition slopes from a .sup file.

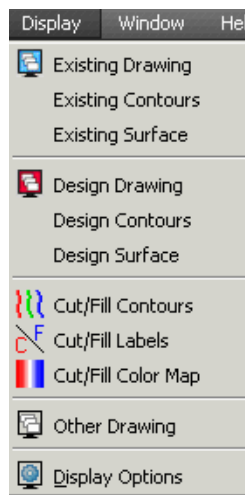
Pulldown Menu Location: Roads

Prerequisite: 3D Polyline

Keyboard Command: tgotffset

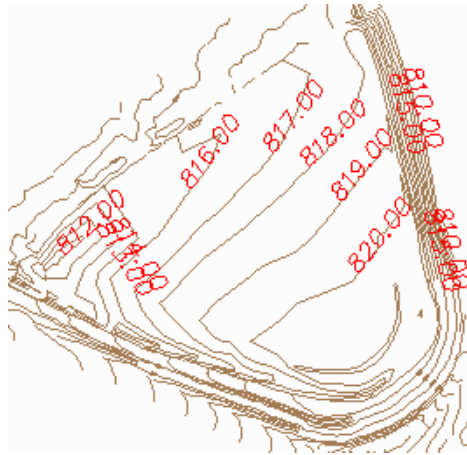
Display Menu

The Display menu commands are quick controls for the drawing view.



Existing Drawing

This command allows you to display all the entities on the layers that are grouped as part of the Existing Drawing.



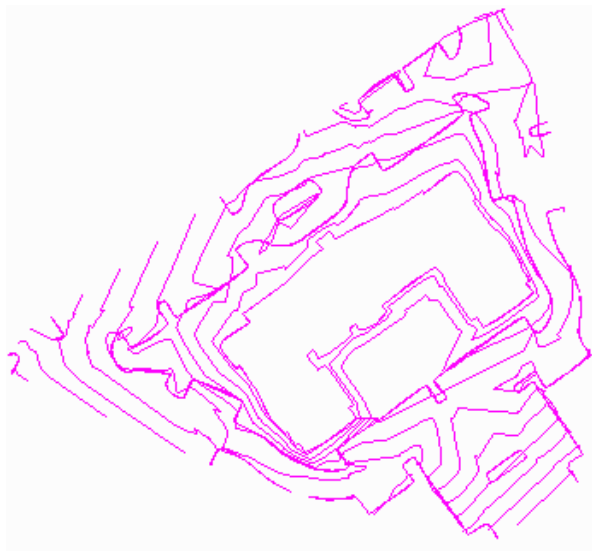
Carlson Takeoff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. When Existing Drawing is checked then the existing drawing will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the existing drawing in isolation or in accordance to the other drawings.

Keyboard Command: set_display_exist_dwg

Prerequisite: Define Layer Target/Material/Subgrade

Existing Contours

This command displays all the contours that represent the existing surface (For contouring options see Display Options). Clicking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.



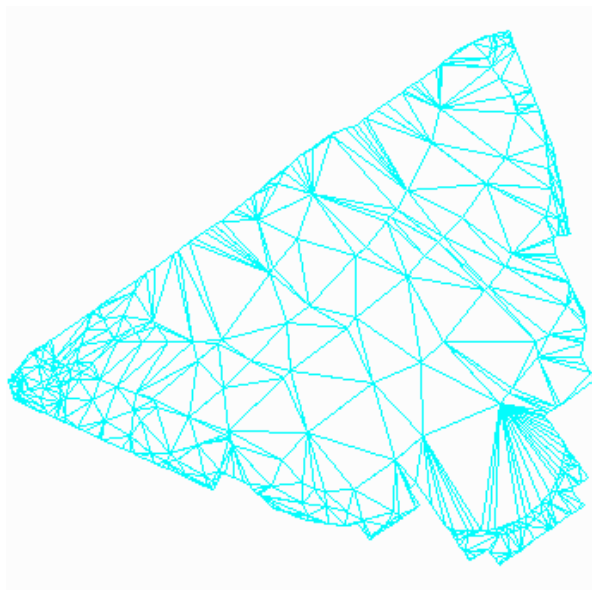
When Existing Contours is checked than all the contours for the existing surface will be displayed. If it is not checked they will not be displayed.

Prerequisite: existing surface

Keyboard Command: set_display_exist_ctr

Existing Surface

This command allows you to display the surface triangulation for the existing drawing.



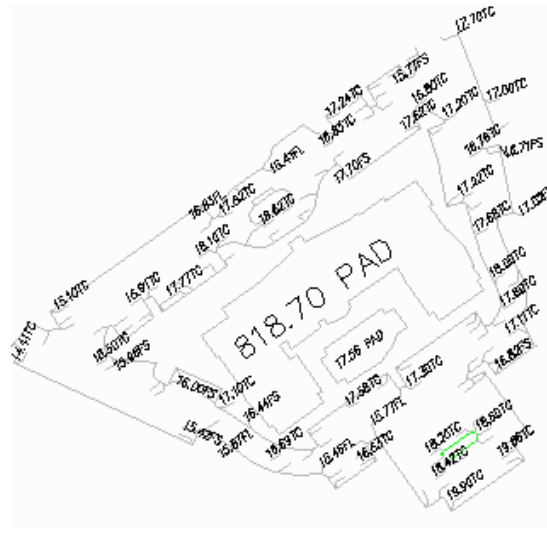
When Existing Surface is checked than all the triangulation for the existing will be displayed. If it is not checked, they will not be displayed.

Keyboard Command: set_display_exist_grd

Prerequisite: an existing surface

Design Drawing

This command allows you to display all the entitles on the layers that are grouped as part of the Design Drawing.



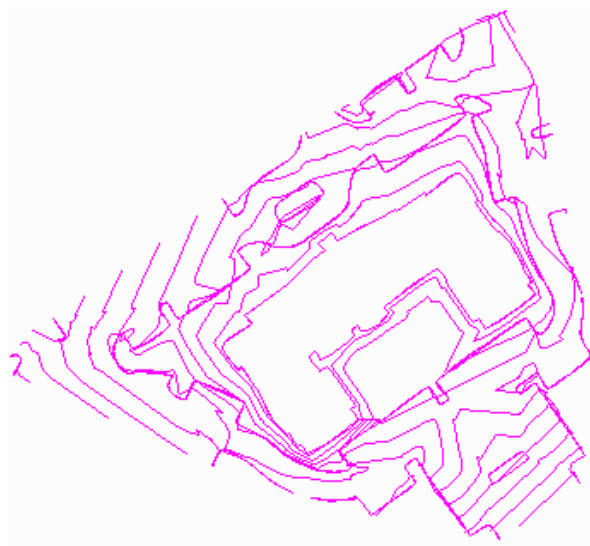
Carlson Takeoff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. When Design Drawing is checked than the design drawing will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the Design drawing in isolation or in accordance to the other drawings.

Keyboard Command: set_display_final_dwg

Prerequisite: Define Layer Target/Material/Subgrade

Design Contours

This command displays all the contours that represents the design surface (For contouring options see Display Options). Clicking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.



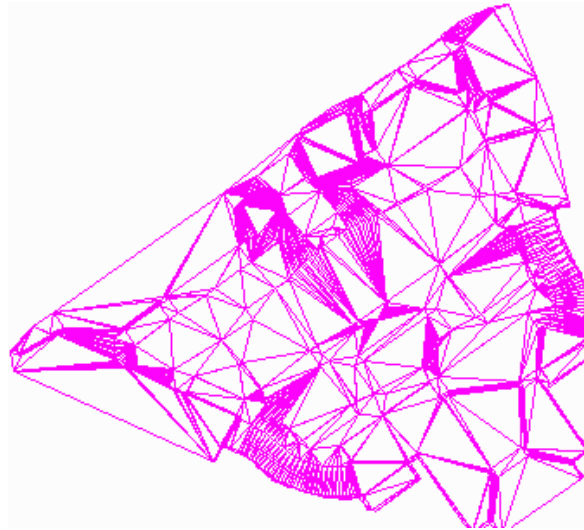
When Design Contours is checked than all the contours for the design will be displayed. If it is not checked they will not be displayed.

Keyboard Command: set_display_final_ctr

Prerequisite: design surface

Design Surface

This command allows you to display the surface triangulation for the design drawing.



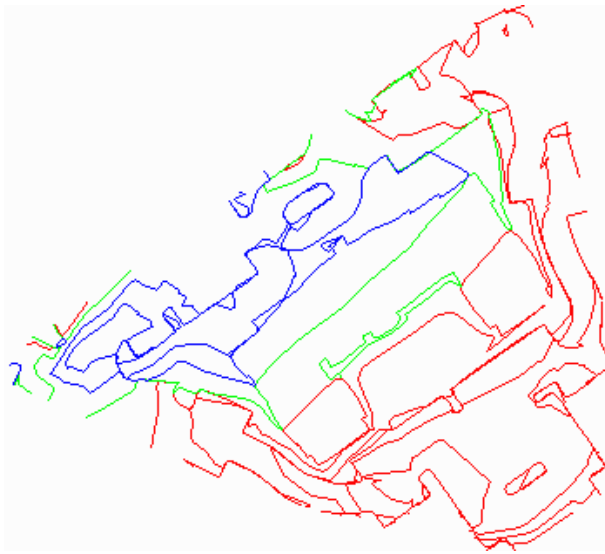
When Design Surface is checked than all the triangulation for the design will be displayed. If it is not checked, they will not be displayed.

Keyboard Command: set_display_final_grd

Prerequisite: a design surface

Cut/Fill Contours

This command compares the existing and design surfaces and shows the cut/fill contours in blue for fill and red for cut. There is a Draw Only Cut/Fill Daylight option and Draw Labels option as part of the Display Options command (See Display Options for more information). Clicking on Cut/Fill Contours from the menu runs the command and puts a check mark on the menu. Picking again turns it off.

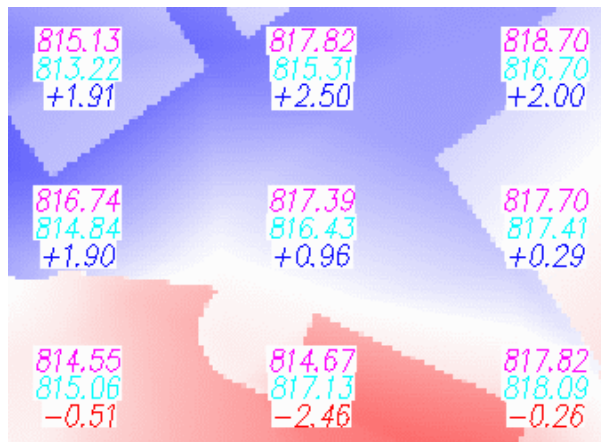


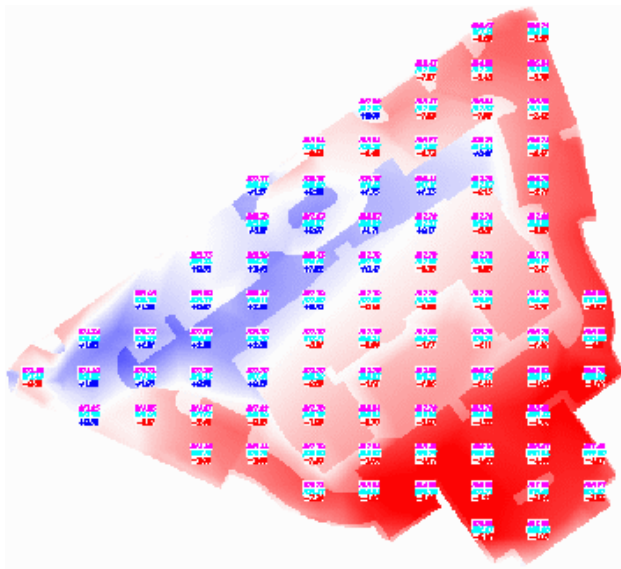
Keyboard Command: set_display_cf_ctr

Prerequisite: elevation differences between existing and design

Cut/Fill Labels

This command displays the design elevation, the existing elevation, and the amount to either cut or fill right on the screen (See Display Options for information about labeling options). Picking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.

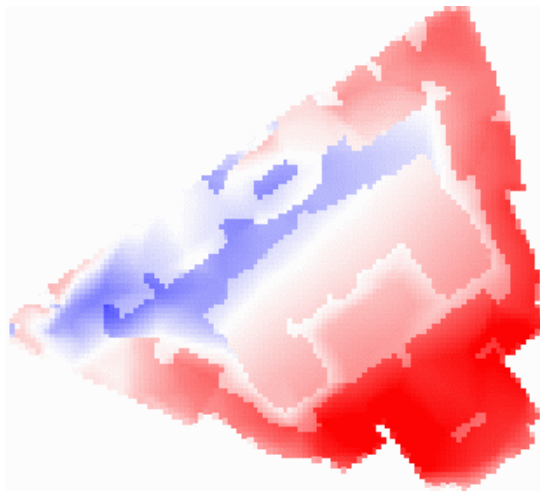




Keyboard Command: set_display_cf_txt
Prerequisite: existing and design surfaces

Cut/Fill Color Map

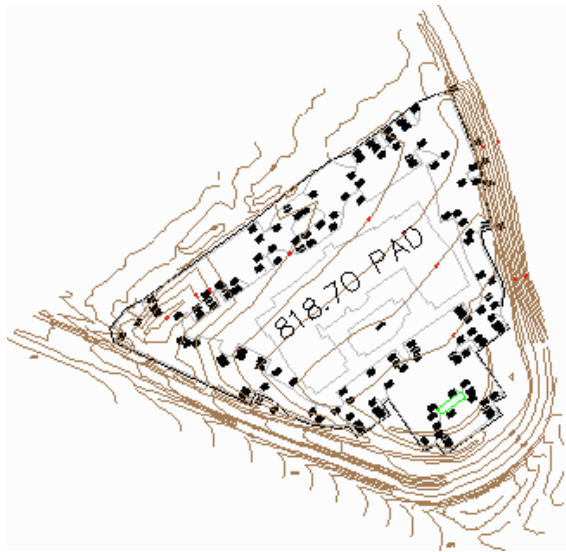
This command compares the existing and design surfaces and shows the cut/fill regions in blue for fill and red for cut (See Display Options for information on pixel resolution). Clicking on Cut/Fill Color Map from the menu runs the command and puts a check mark on the menu. Picking again turns it off.



Keyboard Command: set_display_cf_map
Prerequisite: existing and design surfaces

Other Drawing

This command allows you to display all the entities on the layers that are grouped as part of the Other drawing.



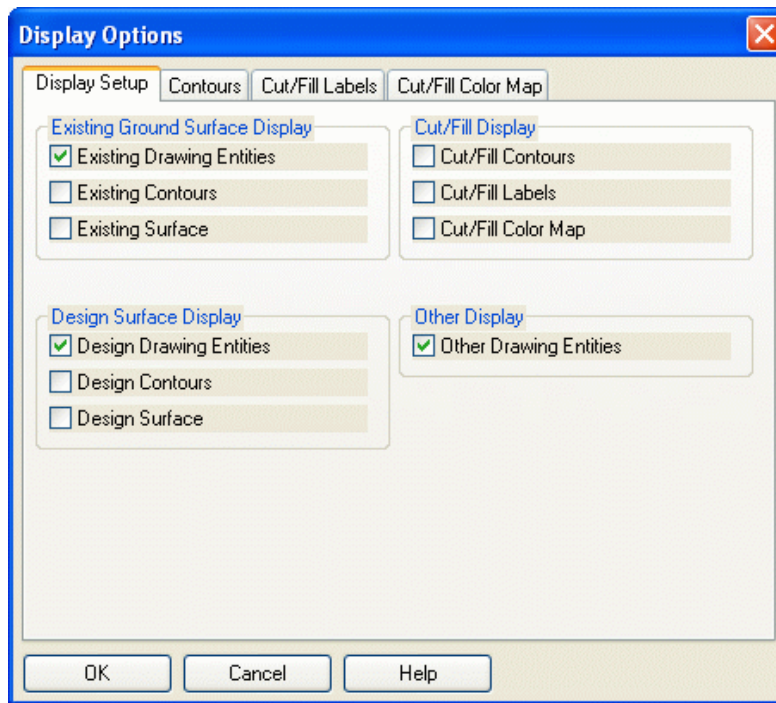
Carlson Takeoff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. Typically, most layers are listed under Other before they are assigned to Existing or Design. Some layers, like perimeter, are neither apart of the Existing or the Design drawing so they remain under Other. When Other Drawing is checked than the entities grouped under Other will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the Other surface in isolation or in accordance to the other surfaces. In this example, Existing, Design, and Other are all shown.

Keyboard Command: `set_display_other_dwg`

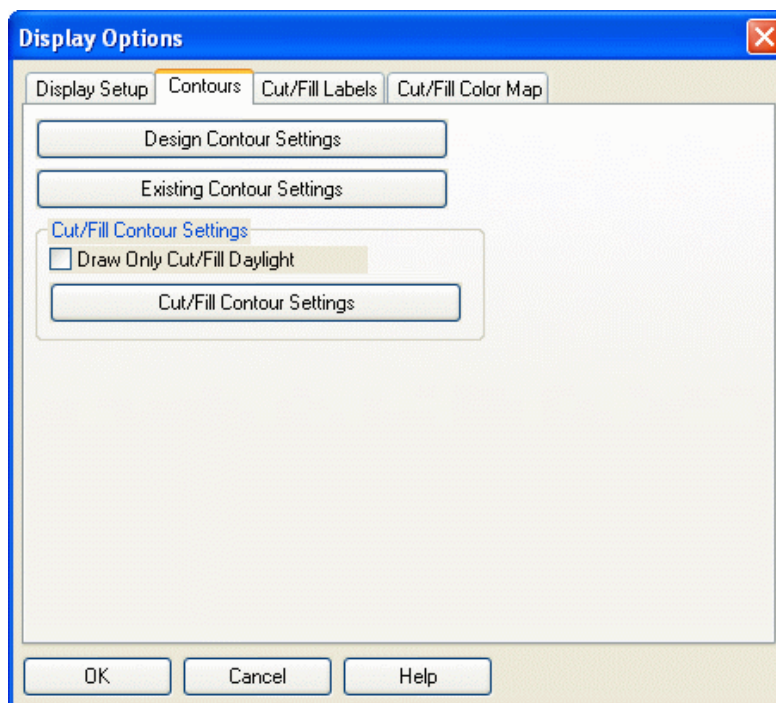
Prerequisite: Define Layer Target/Material/Subgrade

Display Options

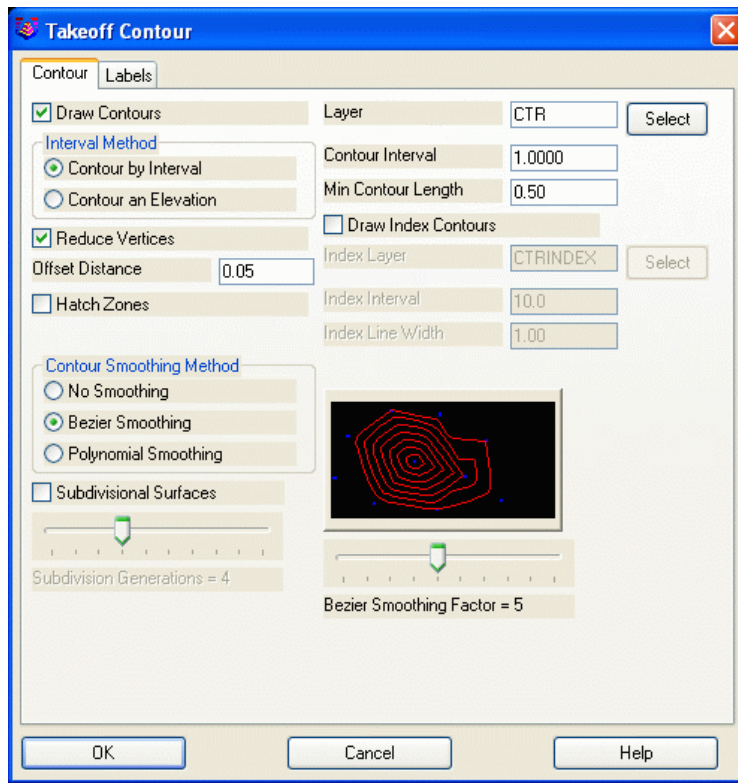
This command allows you to change the features of the different display commands. Note: You can toggle on/off the Existing, Design, and Other surfaces by right clicking with your mouse. To activate this feature type in "short-cutmenu" in the command line and then <1>. To turn off the feature type in <0>.



Display Setup: Here is the master list for the major things you can display, including: the Entities, Contours, and Surface for both the Existing and Design, Cut/Fill Displays, and Other Drawing Entities.



Contour Options: Here you can set the interval, the elevation difference between each contour, for the Existing, Design and Cut/Fill by clicking on their Contour Settings. You can also choose to draw only the daylight line between Existing and Design instead of the Cut/Fill contours at an interval.



Draw Contours

When this box is checked, the program will draw contour lines after triangulating. Otherwise, only the designated triangulation operations are performed. Specify the layer for contours in the edit box to the right.

Contour by Interval or Contour an Elevation

Select whether to contour by interval (ie: every 10 feet) or to contour a certain elevation. The elevation option allows you to contour specific values. For example, if you want just the 100ft contour, then select elevation and enter 100. The default mode is by interval.

Contour Interval

Specify the interval to contour. Note: If the above option is set to Contour an Elevation, then this field is used to specify the elevation to contour.

Minimum Contour Length

Contour lines whose total length is less than this value will not be drawn.

Reduce Vertices

This option attempts to remove extra vertices from the contour polylines which has the advantages of a faster drawing and smaller drawing size. Default is ON

Offset Distance

When the Reduce Vertices option is enabled, This value is the maximum tolerance for shifting the original contour line in order to reduce vertices. The reduced contour polyline will shift no more than this value, at any point, away from the original contour line. A lower value will decrease the number of vertices removed and keep the contour line closer to the original. A higher value will remove more vertices and allows the contour line to shift more from the original.

Hatch Zones

When activated, this option will allow you to hatch the area between the contours sequentially. A secondary dialog will load allowing the user to specify the hatch type and color.

Draw Index Contours

This option creates highlighted contours at a specified interval. When enabled, the fields for Index Layer, Index Interval and Index Line Width are activated.

Contour Smoothing Method

Select the type of contour smoothing to be performed. Bezier smoothing holds all the contour points calculated from the triangulation and only smooths between the calculated points. Polynomial smoothing applies a fifth degree polynomial for smooth transition between the triangulation faces. The smoothing factor described below affects the smoothing bulge.

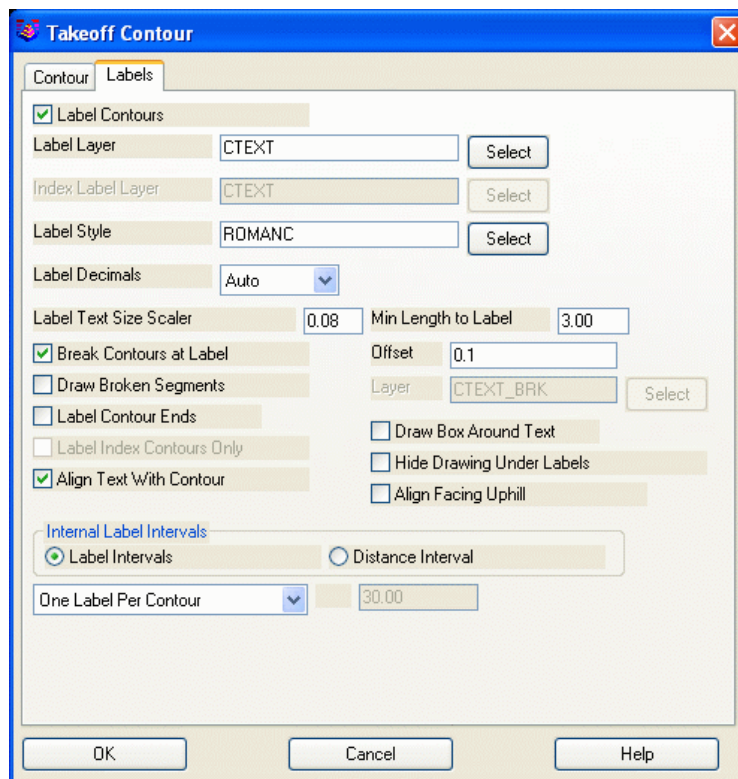
Bezier Smoothing Factor

The contour preview window shows you an example of how much smoothing can be expected at each setting. Sliding the bar to the left results in a lower setting which have less looping or less freedom to curve between contour line points. Likewise, moving the slider to the right results in a setting that increases the looping effect.

Subdivisional Surfaces / Subdivisions Generation

This option causes each triangle in the triangulation surface model to be subdivided into an average of three smaller triangles per subdivision generation, with the new temporary vertices raised or lowered to provide smoother contours. More generations increases the smoothness of the algorithm at a cost of increased processing time. If Straight Lines are chosen as the contouring drawing method, then the contours are guaranteed never to cross. The original points of the surface model are always preserved. These modifications to the surface model are only for contouring purposes and are not written to the triangulation (.FLT) file or inserted into the drawing. If some contour movement is too small for appearance's sake, consider enabling Reduce Vertices.

Label Tab



Label Contours

When activated, contours will be labeled based on the settings below.

Label Layer

Specifies layer name for intermediate contour labels.

Index Label Layer

Specifies layer name for index contour labels.

Label Style

Specifies the text style that will be used for the contour label text.

Label Text Size Scaler

Specifies the size of the contour labels based on a multiplier of the horizontal scale.

Min Length to Label

Contours whose length is less than this value will not be labeled.

Break Contours at Label

When checked, contour lines will be broken and trimmed at the label location for label visibility. When enabled, the Offset box to the right activates. The Offset determines the gap between the end of the trimmed contour line and the beginning or ending of the text.

Draw Broken Segments

When checked, segments of contours that are broken out for label visibility will be redrawn as independent segments. Specify the layer for these broken segments in the box to the right of this toggle.

Label Contour Ends

When checked, contour ends will be labeled.

Draw Box Around Text

When checked, a rectangle will be drawn around contour elevation labels.

Label Index Contours Only

When checked, only the index contours will be labeled. This option is active only when "Draw Index Contours" has been selected in the Contour tab of the main dialog.

Hide Drawing Under Labels

This option activates a text wipeout feature that will create the appearance of trimmed segments at the contour label, even though the contour is fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels, and the contour lines themselves remain contiguous. This feature will also hide other entities that area in the immediate vicinity of the contour label.

Align Text with Contour

When checked, contour elevation labels will be rotated to align with their respective contour lines. This option also activates the Align Facing Uphill feature explained below.

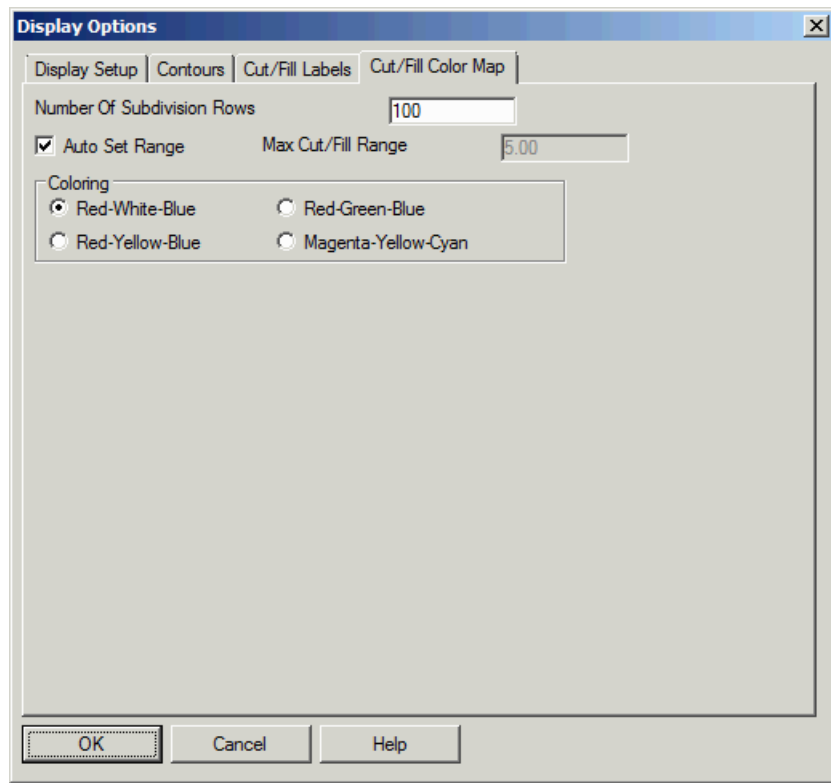
Align Facing Uphill

When checked, contour elevation labels will still be rotated to align with their respective contour lines, but the labels will be flipped in such a manner that the bottom of the text label will always be toward the downhill side of the contours. So as the labels are read right side up, you are always facing uphill.

Internal Label Intervals

Choose between label intervals or distance interval. Label intervals will label each contour with a set number of labels. Distance interval lets you specify a distance between labels.

Cut/Fill Label Options: Here you can customize the Cut/Fill labels. Text can be added either before or after the Cut/Fill amount, the Existing elevation, and the Design elevation with the Prefix and Suffix fields. You can also choose whether or not to display the Existing Surface elevations, the Design Surface elevations and Strata Cut Thickness. The colors for the Cut, Fill, Existing Elevation, and Design Elevation text are all customizable. Carlson Takeoff gives you the option to draw a marker symbol for where each label represents. You can also hide the drawing under the labels so that you can read the labels clearly. Text Size chooses the text size for each line of the label. Text Style allows you change the Font Style displayed in the labels. Decimal Places sets to how many decimal places the labels will report. The Spacing of the labels can be determined by intervals or by a selected number of spaces. The size of each space is determined by the Text Size.



Cut/Fill Color Map Options: Number Of Subdivision Rows is the number of blocks both horizontally and vertically in the Color Map. If the box reads 100 that means 100 blocks left to right and 100 blocks up and down or 10,000 total pixels. A higher the Number Of Subdivision Rows will make the Color Map sharper, however too high number can cause Carlson Takeoff to run slower. Auto Set Range will automatically set the red to blue scale for your cut/fill levels. However, if you desire greater contrast, then use Max Cut/Fill Range to manual set the range. Use lower numbers for greater contrast. There are several coloring schemes with different Cut-Daylight-Fill colors. For example, the Red-White-Blue scheme means red for cut, white for daylight and blue for fill.

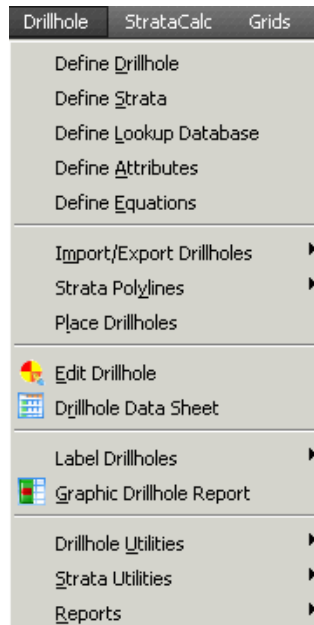
Keyboard Command: tk_display_options

Prerequisite: a drawing

GeoTech Module

9

The GeoTech module is a combination of drillhole routines from the Geology module, surface routines from the Civil module and mapping routines from the Survey module. The GeoTech functions focus on geotechnical tools for civil and environmental firms. The GeoTech menus have commands for drillholes, strata modeling and mapping. Commands that are not covered in the GeoTech chapter are found in the Geology, Survey or Civil chapters.



Drillhole Menu

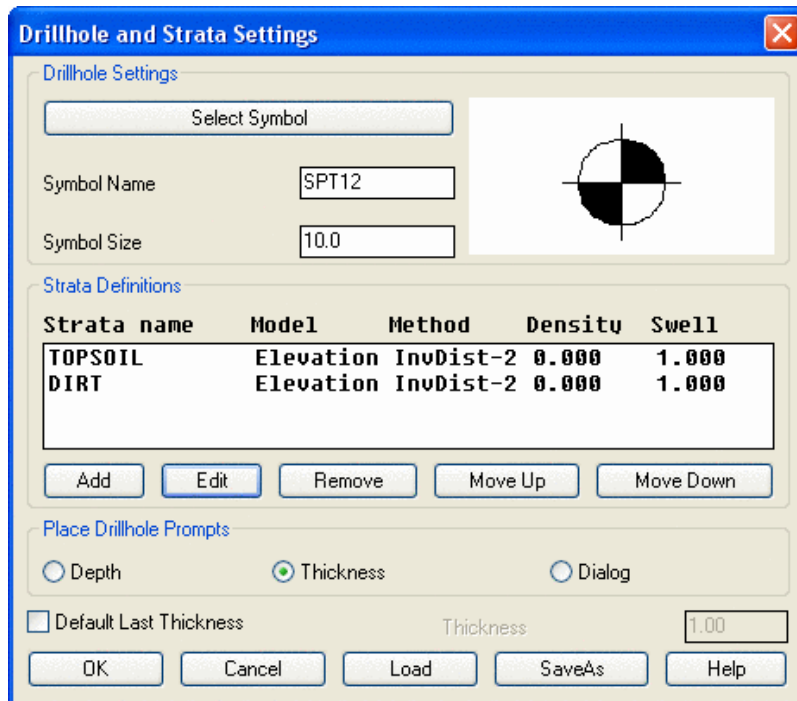
This chapter provides information on using the commands from the Drillhole menu to produce, import and edit drillhole strata settings, place drillholes and report drillholes.

Drillhole Strata Settings

This command selects drillhole symbols, defines strata, and determines how you place drillholes.

Note: The order in which the Strata are defined in the Strata Definitions list will be the default order for the strata when you create new drillholes through **Place Drillhole**.

The dialog box below shows the layout of the Drillhole and Strata Settings.

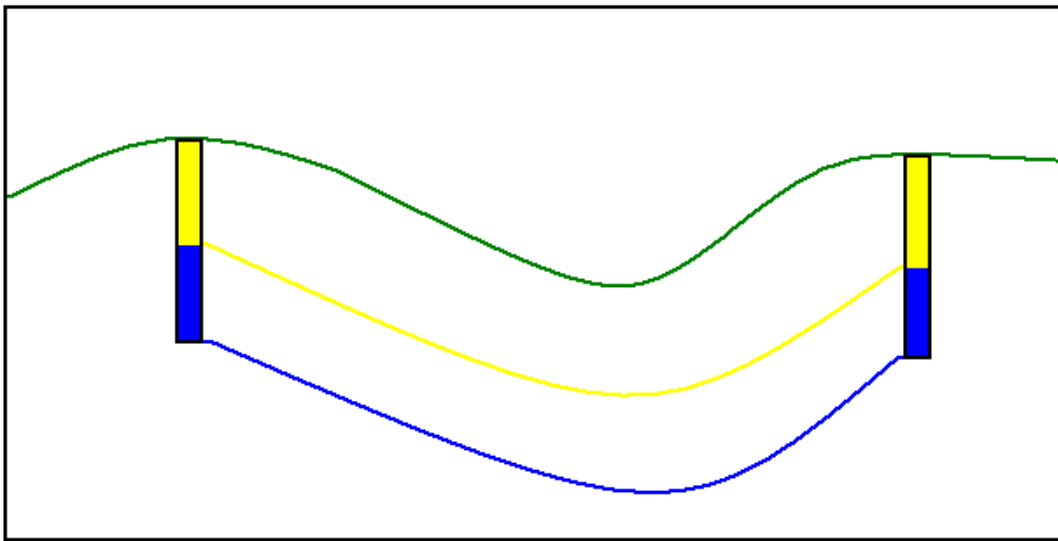


- **Select Symbol:** Select a symbol to represent the drillhole location on the screen.
- **Symbol Name:** This name corresponds to the symbol selected.
- **Symbol Size:** This field can be edited to adjust the symbols size displayed on the screen.
- **Strata Definitions:** This is not directly editable. Select the Strata you are interested by highlighting it, then select the Edit button.

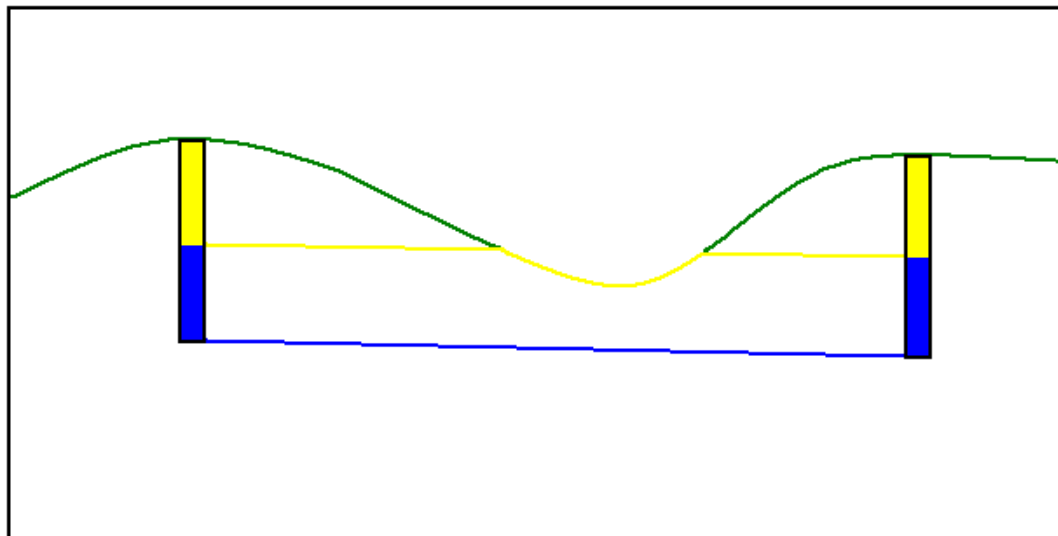
- **Add:** This option adds additional strata to the available Strata name list. See Edit Strata dialog box below.
- **Edit:** Similar to Add, this option is available to make changes to the Strata, including adding a swell factor.
- **Strata Name:** The name of the strata.

Density (lbs/ft³): The Strata Density field is the default density used to calculate strata tons. Density is strata-specific.

- **By Depth:** This option will generate a strata surface by modeling the strata depth values in the drillholes. This strata surface will follow the existing ground surface at the model depth.



- **By Strata Elevation:** This option will generate a strata model that connects strata irrespective of the upper surface elevation changes.



- **Strata Modeling Method:** There are three ways to model strata by Inverse Distance to the 2nd power, 3rd power, or by linear least squares.

Inverse Distance (Power 2 and Power 3) Modeling Method

Inverse distance calculates the strata model by assigning weights to the drillholes. The strata model calculated by inverse distance are a weighted average of the drillhole data. Inverse distance will not carry trends and the calculated surface model will never be higher than the highest drillhole elevation. Likewise the calculated strata model will never be lower than the lowest drillhole elevation. The weights are proportional to the inverse of the distance between the point to be estimated and a drillhole. Closer drillholes are weighted more than points farther away. The inverse distance can be calculated to the second or third power which are $(1/d^2)$, and $(1/d^3)$ respectively. Inverse Distance - Power 3 will weigh drillholes less that are further away.

Linear Least Squares Modeling Method

The linear least squares method finds the least squares best fit plane across the surface model. The least squares routine weights each drillhole by inverse distance so that closer points are weighted more than points farther away.

So the best fit plane varies at different points on the surface. The linear least squares method extrapolates trends very well. Least squares will trend and allows for data points that are new highs and lows, that don't appear in the original drillhole data.

- **Remove:** This will remove a strata name from the available strata.
- **Move Up:** This option will move the selected strata name up one place in the strata name list.
- **Move Down:** This option will move the selected strata name down one place in the strata name list.
- **By Strata Elev:** This method will generate a strata surface by modeling the strata elevation values from the drillholes. This strata surface is independent of the existing ground surface.
- **Place Drillhole Prompts:** If Depth is selected, then when you run Place Drillholes you will be prompted for the depth of each strata in your drillhole. If Thickness is selected, you will be prompted for the thickness of each strata. If Dialog is selected, you will go straight into the Place Drillhole dialog when you create a drillhole.
- **Default Last Thickness:** Will set the thickness of your bottom strata to the same amount for all your drillholes.

Keyboard Command: tk_chdef

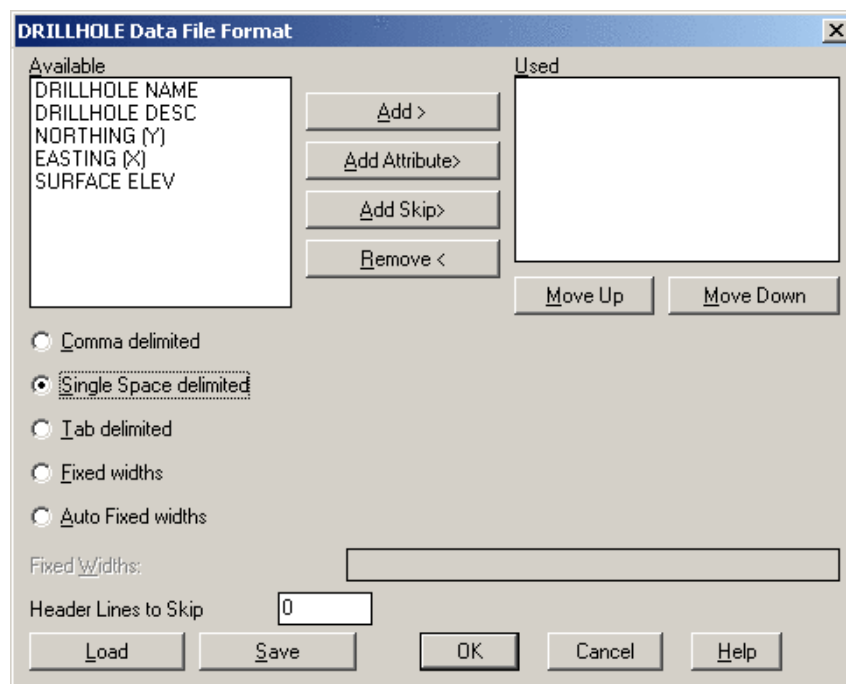
Prerequisite: strata information

Drillhole Import

This command allows you import existing drillhole files. When you select Drillhole Import from the Drillhole menu, a command prompt shows:

"Use separate drillhole and strata files [Yes/<No>]?" If you have two separate files, one with strata info, and the other file has drillhole locations, select Yes. If you enter Yes, the dialog box below appears.

This command creates drillholes from the data contained in text files. Currently there are many company-specific formats. A Drillhole Data File Formatter that is flexible to handle almost any drillhole text file format is below. The format to use is chosen in the dialog shown here.



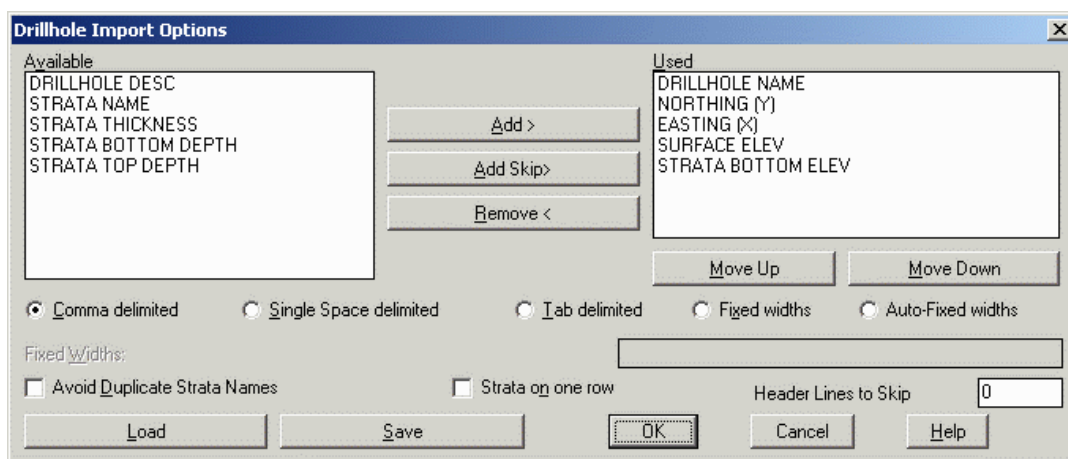
The import text can have comma delimited, space delimited or fixed width columns of data. All the data for a record should be on one row. For the fixed width format, choose the Fixed Width toggle and then enter the column numbers separated by spaces in the edit box. For example, "8 15 24 32".

The Custom format can import all the drillhole and strata data from one text file or the drillhole data from one file and the strata data from another file. The method to use is set at the Use separate drillhole and strata files prompt.

Use the following commands to prepare a file format that will match the *.imp imported file.

- **Add:** Moves the selected entry from Available to Used.
- **Add Attribute:** Allows user input attributes into the Used section.
- **Add Skip:** Adds a "Skip" place holder in the Used List
- **Remove:** Moves a selected item from Used to Available list.
- **Move Up:** Moves the selected item up one place in the list.
- **Move Down:** Moves the selected item down one place in the list.
- **Comma Delimited:** Select this if your *.imp file has commas separating each field.
- **Single space delimited:** Select this if your *.imp has a space separating each field.
- **Tab delimited:** Select this if your *.imp file has tabs separating each field.
- **Fixed widths:** Select this if your *.imp has a defined width of space separating each field.
- **Auto Fixed widths:** Select this to automatically determine the fixed widths that separate each field in the *.imp file.
- **Header Lines to Skip:** If your *.imp file has header lines, enter the number of header lines here.
- **Load:** Takes you to select/brows for your *.imp file.
- **Save:** This command will save your imported file as a *.imp file.

The dialog box below details the drillhole import options.



In addition to the previously listed import commands above, this dialog box also has the following prompts:

- **Avoid Duplicate Strata Names:** Select this to prevent have more than one strata with the same name.

- **Strata on one row:** Select this option if all of your strata info is on one row.

Keyboard Command: tk_chimport

Prerequisite: drillhole files

Place Drillhole

This command allows you to screen pick, enter coordinates or, locate by station-offset the placement of a drillhole.

Go to **Drillhole/Strata Settings, Place Drillhole Prompts**, to determine how you would like to be prompted. When you select **Place Drillhole** from the **Drillhole menu**, the command line prompt shows:

"Station/<Pick Drillhole Location>:" Type in x-y coordinates or move your pointer around the screen to pick the placement of the drillhole. To load a centerline file, press S, select the centerline file to reference, then type in the desired Station and Offset amount. If you are in Dialog Mode defined in **Drillhole/Strata Settings**, once a location has been selected, the following dialog box appears:

STRATA NAME	DEPTH	BOTTOM ELU	THICKNESS
LOOSE LOAM	0.00	0.00	0.00
GLATIAL TILL	0.00	0.00	0.00
SHALE	0.00	0.00	0.00
SEDIMENT	0.00	0.00	0.00

Place Drillholes generates drillholes in the drawing that are required to run strata surface application routines. Each drillhole consists of a surface elevation, strata, and optional description(s). Every strata has a name, bottom elevation, thickness. Within a drillhole, the strata names must be unique, but each real-world strata should have the same strata name across all the drillholes. This is because strata surface applications connects together the strata with the same name.

The drillhole data can be entered in the dialog shown here, or if Depth or Thickness Mode is selected under **Drillhole/Strata Settings**, then the data can be entered in on the command line when you place each drillhole. Make sure to specify the surface elevation and drillhole description. While in Dialog Mode or to change data, use the Edit and Insert/append buttons to enter strata data. The symbols are defined in DrillHole/Strata Settings and drillhole may be changed in Edit DrillHole. Pick Save when done and a drillhole symbol is drawn.

- **Edit:** Make changes to the highlighted strata name. Thickness, Bottom Elevation, Depth.

When placing drillholes, every strata must be assigned a bottom elevation and a thickness. The bottom elevation is the elevation of the bottom of the strata. There are different methods for entering this information.

- **Insert Above:** To add a Strata above the highlighted strata name.
- **Append to Bottom:** To add a strata to the bottom of the available strata name list.
- **Remove:** Removes a strata from the available Strata Name list.
- **Surface Elevation:** This field can be set by you to establish the surface elevation of the drillhole.
- **Drillhole Name:** The name of the drillhole
- **Description:** Drillhole descriptions are intended for storing of drillhole specific information in the drillhole. One general drillhole description is predefined and user may define any number of specific drillhole descriptions. Typical additional descriptions are DRILLER, DATE, TOWNSHIP, and etc. You will be prompted for values of these descriptions in Place DrillHole.
- **Adjust Bottom Elevations:** Will make adjustments to the bottom elevation based on thickness changes.
- **Adjust Next Thickness:** Will adjust the next thickness to hold the bottom elevation unchanged.
- **Save:** This command saves this drillhole as listed.
- **Zoom In:** This increases the magnification of the black view window, cross-section view of the drillhole.
- **Zoom Out:** This decreases the magnification of the black view window, cross-section view of the drillhole.
- **Cancel:** Ends Drillhole placement routine without making changes.

Keyboard Command: tk_chplace

Prerequisite: drillhole information

Edit Drillhole

This command allows you to screen pick an existing drillhole and edit its properties. When you select **Edit Drillhole** from the **Drillhole menu**, a command prompt shows:

"Select Drillhole to edit:" Move your pointer around the screen to pick the drillhole you want to edit. Once a drillhole is picked on the screen, the following dialog box appears:

STRATA NAME	DEPTH	BOTTOM ELV	THICKNESS
DIRT	2.00	815.70	2.00
ROCK	27.00	790.70	25.00

- **Edit:** Make changes to the highlighted strata name. Thickness, Bottom Elevation, Depth.

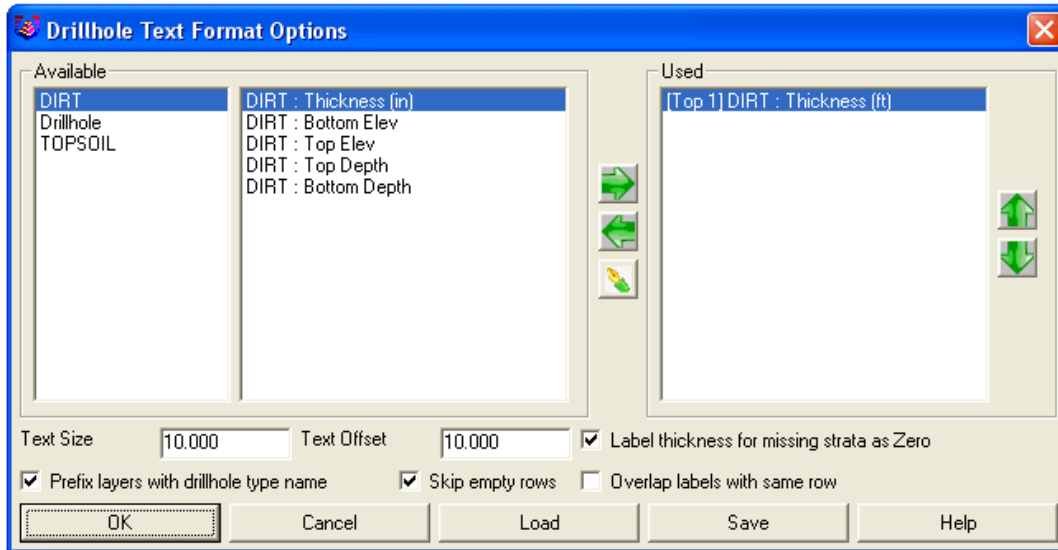
- **Insert Above:** To add a Strata above the highlighted strata name.
- **Append to Bottom:** To add a strata to the bottom of the available strata name list.
- **Remove:** Removes a strata from the available Strata Name list.
- **Surface Elevation:** This field can be set by you to establish the surface elevation of the drillhole.
- **Water Table Elevation:** This elevation is used for modelling the water table surface.
- **Drillhole Name:** The name of the drillhole
- **Description:** The screen display description of the drillhole
- **Adjust Bottom Elevations:** Will make adjustments to the bottom elevation based on thickness changes.
- **Adjust Next Thickness:** Will adjust the next thickness to hold the bottom elevation unchanged.
- **Save:** This command saves this drillhole as listed.
- **Zoom In:** This increases the magnification of the black view window, cross-section view of the drillhole.
- **Zoom Out:** This decreases the magnification of the black view window, cross-section view of the drillhole.
- **Cancel:** Ends Drillhole placement routine without making changes.

Keyboard Command: tk_chedit

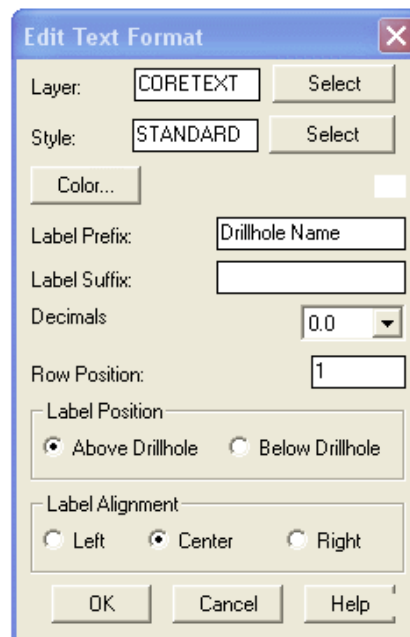
Prerequisite: drillhole information

Label Drillhole

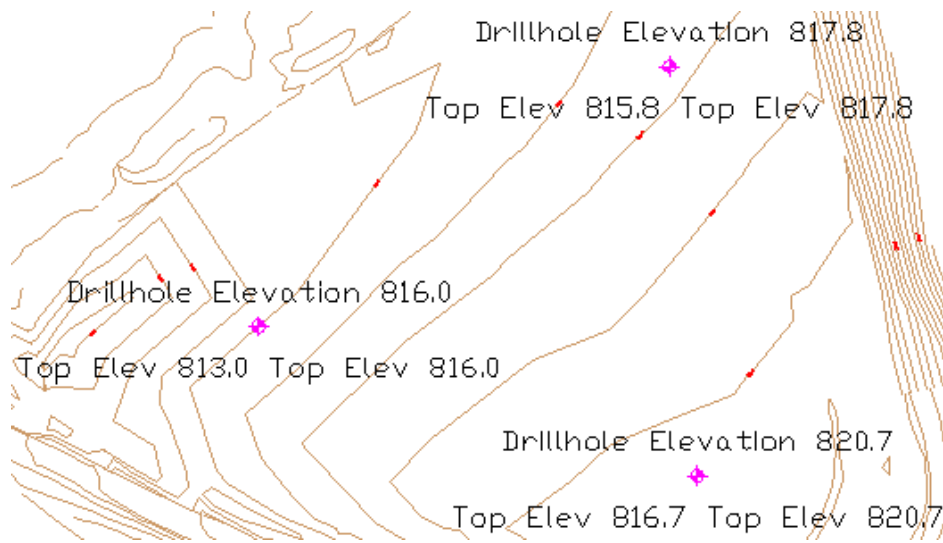
Label Drillhole can be used to label selected properties from drillholes in the current drawing. The first prompt will ask you to select the drillholes you wish to label. To do this, either window around the drillholes you wish to label or type in "all" enter to select all the drillholes in the drawing. Next, a dialog will appear that gives you control on what is displayed in the Labels.



On the left side of this dialog are the Available label options for the drillholes. You have control what is shown for each strata surface as well as general drillhole information such as the name and coordinates. Use the arrows in the middle of the dialog to move an Available label option into the Used column on the right. When you do this, the below dialog will appear with more options.



In the Edit Text Format dialog, you can control the Layer, Style, Color, Prefix/Suffix, Decimals, Row Position, and Alignment of text. These items when changed become default the next time the dialog is opened. Here is an example of Drillhole Labels.



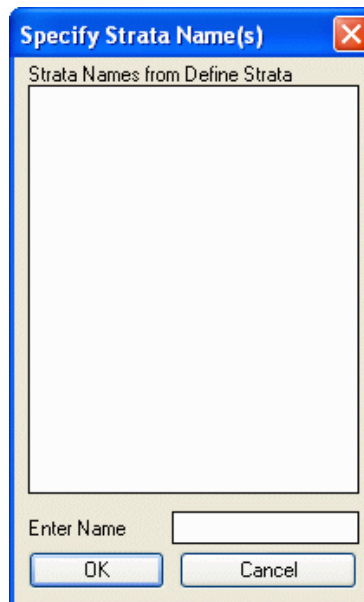
Pulldown Menu Location: Drillhole
Prerequisite: Place Drillhole
Keyboard Command: chtext2

Strata Polylines

Strata Polylines define strata elevation or thickness along linework instead of a single point like Place Drillhole. Linework defined as Strata Polylines are incorporated with Drillhole Data to create surface models. Note: Strata surface models can not be made exclusively from Strata polylines, some drillholes need to be placed as well.

Tag Strata Polylines

This command allows the user to select polylines that define a Strata. Pick the Strata from the list or type in the name in the Enter Name field. Any Strata you enter in must match a strata defined in Drillhole/Strata Settings in order for the surface to be created.



After selecting a Strata and pressing enter you will be prompted for the type of polyline.

Type of strata polyline [<Elevation>/Thickness]?

Elevation signifies that the Z value for the polyline(s) you are about to select represent the bottom elevation of the previously selected strata. Thickness means that the Z value represents thickness of the strata. Choose one of these options and select the polylines.

Prerequisite: Drillhole/Strata Settings, desired polylines

Keyboard Command: stratatag

Highlight Strata Polylines

This command allows users to identify Strata Polylines by either picking on a polyline(s) or by searching the entire drawing. The command will then dash the polyline in the plan view.

Prerequisite: Tag Strata Polylines

Keyboard Command: highlight_strata_pl

Identify Strata Polylines

This command allows users to identify topsoil polylines by picking on a polyline. The command will then report the Strata name and Type (either Elevation or Thickness).

Prerequisite: Tag Strata Polylines

Keyboard Command: strataid

Untag Strata Polylines

This command allows the user to remove previously tagged Strata Polylines so that they are not included in the strata model.

Prerequisite: Tag Strata Polylines

Keyboard Command: stratauntag

Drillhole Reports

This command allows you to generate a report of selected drillholes. When you select **Reports** from the **Drillhole** menu, a sub-menu choice of **Standard Drillhole Report** or **Custom Drillhole Report**, is displayed.

Standard Drillhole Report

If this is selected, several prompts are asked at the command line. They are as follows:

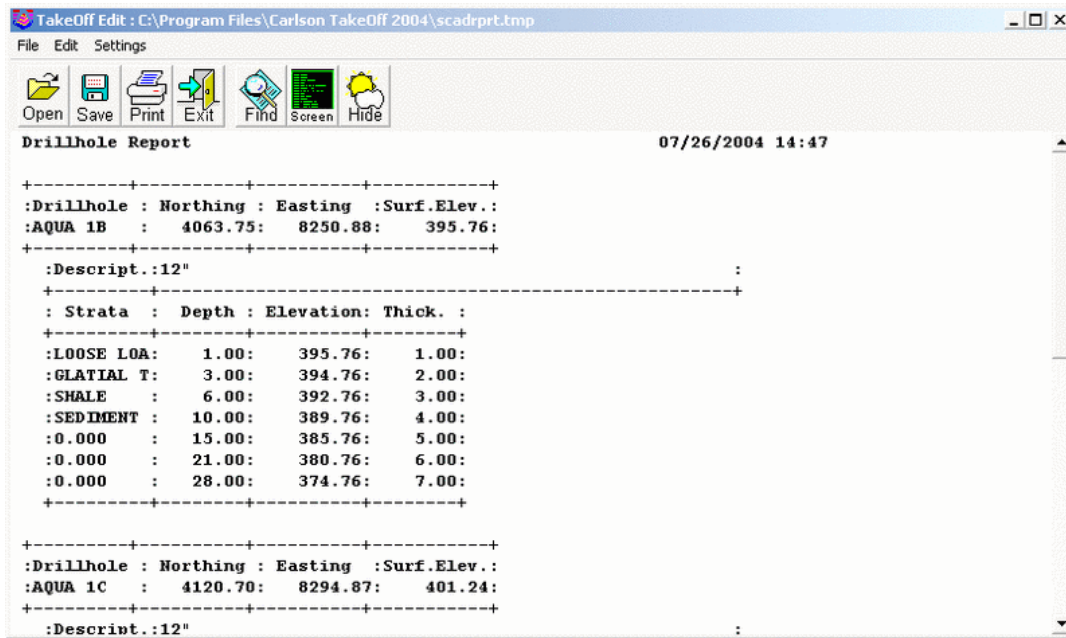
Select objects:

Add Page break between drillholes [Yes/<No>]?

Report Strata depth to [Top/<Bottom>]?

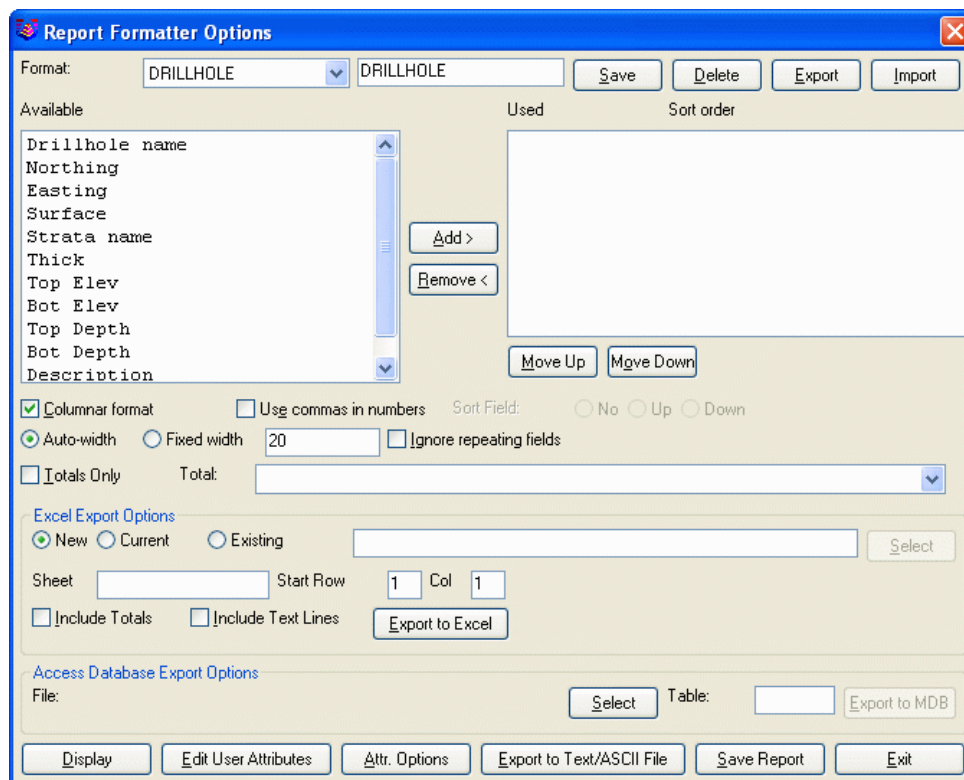
Report Strata elevation of [Top/<Bottom>]?

The report is then displayed accordingly.



Custom Drillhole Report

This function allows you to customize your report format.



Prompts:

Command: tk_chreport2

Select the Drillholes for report.

Select objects: Specify opposite corner: 271 found

262 were filtered out.

Keyboard Command: tk_chreport, tk_chreport2

Prerequisite: drillholes

Make Strata Surface

This command generates multiple strata surfaces based on strata definitions and placements of drillholes. Strata surfaces are generated at the bottom of each strata. These strata surfaces can then be used in other Takeoff commands like **Calculate Total Volumes**. They can be viewed on screen, through the command **Draw Strata Surface**.

This command also builds the water table surface when the drillholes have water table elevations specified. The water table surface is independent of the strata surfaces. When you have a water table surface, you can use it in other commands such as **Draw Strata Surface**. The water table surface is also used in **Calculate Total Volumes** to report cut quantities below the water table.

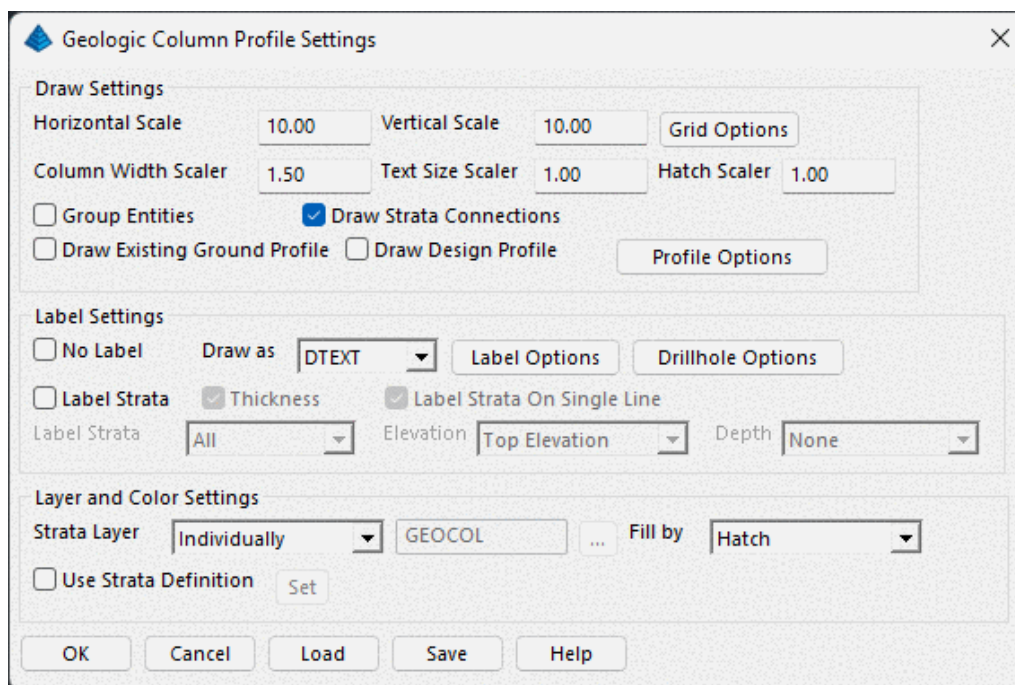
Note: By observing the command line, one can see the status of each strata surface generation.

Keyboard Command: tk_chgrid

Prerequisite: Define Drillhole/Strata Settings, Place Drillhole

Geologic Column Profile

This command will draw drillholes in a profile view, projected along a centerline. Prior to running this command, you must draw a polyline through (or close to) the drillhole symbols. Note that all of these settings may be saved to a geologic column settings file (.gcl).

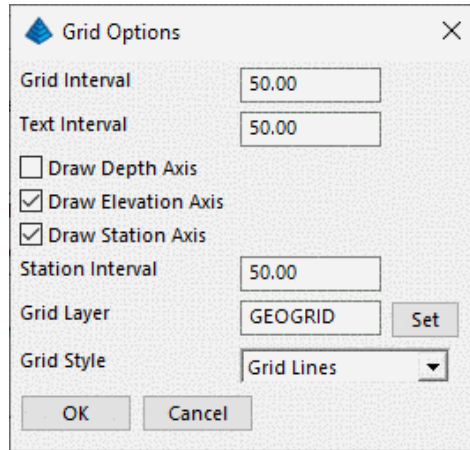


Horizontal Scale: This value sets the overall drawing scale that many other options in the dialog reference

for scaling. This allows you to quickly change the size the geologic columns with a single value.

Vertical Scale: This value, relative to the Horizontal Scale, determines the vertical exaggeration. For example, a Horizontal Scale of 50 and a Vertical Scale of 10 will produce a 5X vertical exaggeration.

Grid Options: This button will open the below dialog. These settings adjust the axis lines drawn behind the drillhole itself.



Grid Interval: This value sets the grid line spacing for the Elevation and Depth axes. For example, a Interval of 25 will draw a tick/grid line every 25 ft/m of elevation change.

Text Interval: This value sets the text intervals for the Elevation and Depth axes. For example, a Text Interval of 100 will draw an elevation/depth label every 100 ft/m of elevation change.

Draw Elevation Axis: This option will draw an elevation axis for the background grid.

Draw Depth Axis: This option will draw a depth axis (vertical line) through the individual drillholes.

Station Interval: This value sets the grid line spacing for Station axis. For example, a Interval of 25 will draw a tick/grid line every 25 ft/m along the length of the fence polyline.

Grid Layer: This option sets the layer for the background grid.

Grid Style: This option controls how the grid will be drawn.

The *Grid Lines* option will draw a true grid with lines spanning the appropriate axes.

The *Ticks Only* option will draw the grid axes, but instead of full lines spanning the axes, small tick marks will be placed along each axis.

The *Ticks and Dots* option will draw tick marks along each axis, as well as points where true grid lines would intersect.

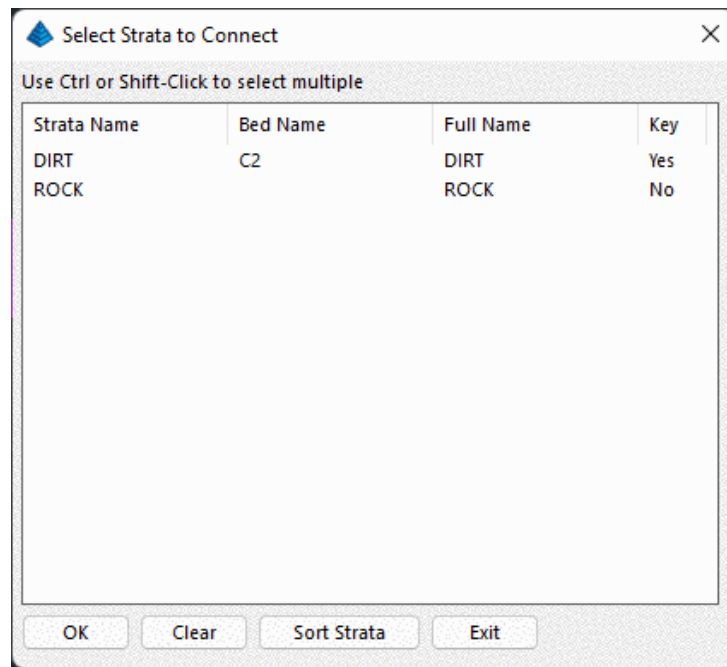
Column Width Scaler: This value, relative to the Horizontal Scale, determines the width of the geologic column. For example, a Column Width Scaler of 0.15 and a Horizontal Scale of 100 will produce a column with a width of 15 drawing units. **Text Size Scaler:** This value, relative to the Horizontal Scale, set the text size labels for strata labels.

Hatch Scaler: This value is only used when the *Fill by Hatch* option is selected. This value, relative to the hatch scales in the current Strata Definition File, determines the hatch scale of each strata's hatch pattern. For example, a Hatch Scaler of 5 and a strata with a defined hatch scale of 2 will produce a hatch pattern with a scale of

10. If a strata is not found in the Strata Definition File, the Hatch Scaler will be multiplied by a value of 5 to set the hatch scale of the strata.

Group Entities: This option will draw each column as a group of entities (not to be confused with a block entity).

Draw Strata Connections: This option will draw polylines connecting similar strata between drillholes. Note that if a strata pinches out between columns, this pinchout will not be drawn. After clicking OK on the dialog, you will be prompted for the strata to connect, as shown below. Note that the bed names are not usable by the Geotech module and may be ignored.

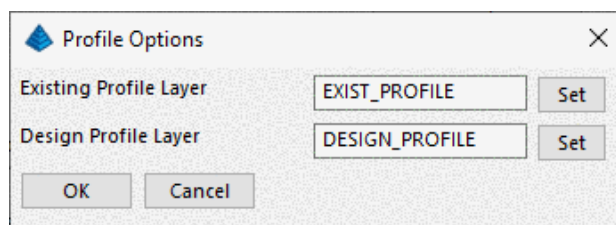


Sort Strata: This button will sort the list alphabetically by Strata Name.

Draw Existing Ground Profile: This option will draw a profile of the existing ground.

Draw Design Profile: This option will draw a profile of the design surface.

Profile Options: This button will display the below dialog to set the layers for the Existing Ground and Design Profiles.



No Labels: This option will turn off labeling for the drillholes and will also ghost out all labeling options in the

dialog.

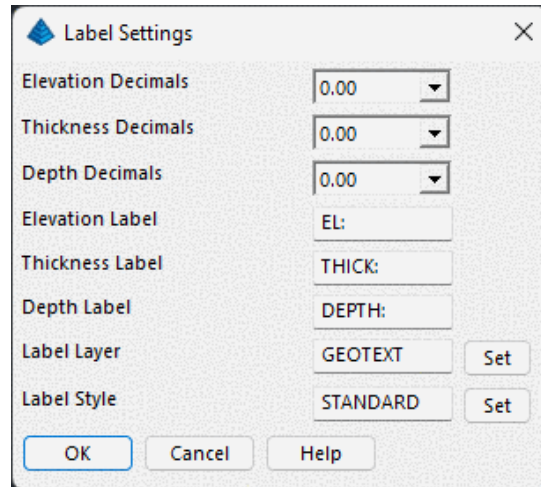
Draw Labels as: This option controls the type of text that is used for labels.

The *DText* option will use a standard text entity for labels.

The *MText* option will use MText (multi-line text) entities for labels.

The *MLeader* option will use Multileader entities for labels, which are simply MText entities attached to a leader line.

Label Options: This button will open the below dialog to control labels specific to each strata layer.



Elevation/Thickness/Depth Decimals: These options control the number of decimals to display for elevation, thickness, and depth labels, respectively.

Elevation/Thickness/Depth Prefix: These text strings set the prefix for elevation/thickness/depth labels, respectively.

Label Layer: This option sets the layer for the strata labels. Clicking the **Set** button will display the list of existing layers in the drawing.

Label Style: This text string sets the text style for the labels. The **Set** button may be used to select a pre-existing text style. Text styles may be edited via the STYLE command.

Drillhole Options: This button will display the below dialog to control labels about the drillhole itself.

Label Northing and Easting: This option will draw a label of the Northing and Easting coordinates for each column.

Label Surface Elevation: This option will draw a label of the Surface Elevation for each column.

Label Drillhole Name: This option will draw a label of the Drillhole Name for each column.

Label Drillhole Descriptions: This option will draw a label of the Drillhole Description for each column.

Label Drillhole Depth: This option will draw a label of the maximum depth for each column.

Label Offset Distance for Projected Method: This option is only applied when the Horizontal Alignment is set to the *Projected* option. This option will draw a label indicating the perpendicular distance from the column to the alignment polyline.

Label Water Level: This option will draw a label of the water table elevation in the column. When this option is selected, the **Water Level Attribute** must match the name of the water level attribute defined in the drillhole. Note that the water level is an attribute of the drillhole itself; it should not be listed as an attribute of a particular strata.

Northing Prefix/Suffix: These two text strings will set the prefix and suffix for the label of the column Northing coordinate.

Easting Prefix/Suffix: These two text strings will set the prefix and suffix for the label of the column Easting coordinate.

Elevation Prefix/Suffix: These two text strings will set the prefix and suffix for the label of the column Surface Elevation.

Offset Prefix/Suffix: This option will only be applied when the Horizontal Alignment is set to the *Projected* option. These two text strings will set the prefix and suffix for the label of the column Offset from the fence alignment polyline.

Label Position: This option determines where the drillhole label is drawn.
If the *Above Drillhole* option is selected, the label will be center justified above the column.
If the *On Side* option is selected, the label will be drawn to the right of the column.

Drillhole Text Layer: This option sets the layer for the drillhole labels. The **Set** button may be used to select a pre-existing layers.

Drillhole Text Style: This text string will set the text style for the drillhole label. The **Set** button may be used to select a pre-existing text style. Text styles may be edited via the STYLE command.

Drillhole Text Size Scaler: This value, relative to the Horizontal Scale, set the text size labels for the drillhole labels. For example, a Text Size Scaler of 0.15 and a Horizontal Scale of 100 will produce a label with a height of 15 drawing units.

Label Strata: This option will draw labels for each strata found in the drillhole. All strata label options will be ghosted when this option is not selected.

Strata to Label: This option will determine which strata are labeled.
The *All* option will label all strata.
The *Selected* option will prompt you to select the strata to label.
The *Fit-Only* option will only draw a label if the strata is thick enough to place a label without having to adjust the label position to avoid overlaps.

Elevation Label: This option determines which elevation will be labeled for each strata.
If the *None* option is selected, no elevation label will be drawn.
If the *Top Elevation* option is selected, the top elevation of each strata will be labeled.
If the *Bottom Elevation* option is selected, the bottom elevation of each strata will be labeled.

Depth Label: This option determines which depth will be labeled for each strata.
If the *None* option is selected, no depth label will be drawn.
If the *Top Depth* option is selected, the depth to the top of each strata will be labeled.
If the *Bottom Depth* option is selected, the depth to the bottom of each strata will be labeled.

Strata Layers: This option will determine which CAD Layers are used for each strata in the column.
If the *Individually* option is used, each strata will be drawn on the CAD Layer specified in the Strata Definition File.
If the strata is not found in the Strata Definition File, each strata will be drawn on a CAD Layer matching the name of the strata.
If the *All Same* option is selected, all strata will be drawn on the CAD Layer specified just to the right of this option.
The ellipsis button may be used to select a pre-existing CAD Layer.

Fill by: This option determines the hatch pattern for each strata.
If the *Outline Only* option is selection, no hatch pattern will be drawn.
If the *Hatch* option is selected, the strata will be filled with the hatch specified in the Strata Definition File. If a strata is not found in the Strata Definition File, the strata will be hatched with the ANSI31 pattern.

If the *Solid Fill* option is selected, all strata will be hatch with the SOLID hatch pattern.

Use Strata Def: This option allow the use of a different Strata Definition File (which controls layering, colors, hatching, etc.) other than the one associated with the drawing. Note that if you leave this option unchecked, the program will use the settings defined in the Define Drillhole and Strata Settings command. The ellipsis button will prompt you for this Strata Definition File.

Pull-Down Menu Location: StrataCalc

Keyboard Command: geocol_profile

Clear Strata Surface

This command clears the strata surfaces previously generated with **Make Strata Surface**. This removes the strata surfaces from processing in other takeoff commands.

Note: This command will not remove the surface from the screen view. You must use the command **Erase Strata Surface** to remove them from view.

Keyboard Command: tk_chclear

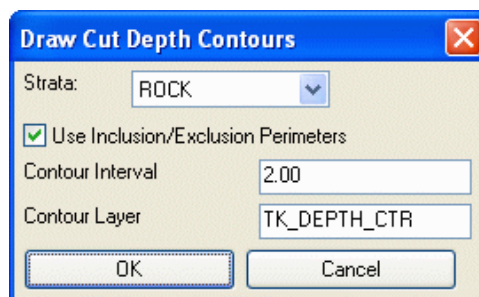
Prerequisite: Make Strata Surface

Draw Strata Cut Depth Contours

This command will draw the **Strata Cut Depth Contours**. This command creates contours for the cut depth between the design surface and strata.

You must have created Strata Surfaces through the **Make Strata Surface** command.

Then select **Draw Strata Cut Depth Contours** from the **Drillhole menu**. You will be prompted to select the Strata from the dialog box below.



You can assign a contour interval and contour layer for the contours to be drawn. If **Use Inclusion/Exclusion Perimeters** is checked on you will be prompted for an Inclusion polyline and an Exclusion polyline if needed, otherwise the drawing's Boundary linework will be used.

Keyboard Command: tk_chdepth

Prerequisite: Make Strata Surface

Erase Strata Cut Depth Contours

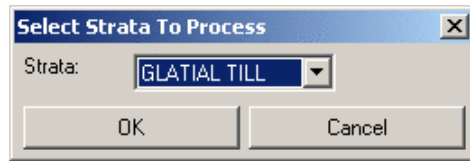
This command will erase the Strata Cut Depth Contours from the screen display.

Keyboard Command: tk_chdepth2

Prerequisite: Strata Cut Depth Contours

Draw Strata Cut Color Map

This command will generate a map of areas where the design surface cuts into the selected strata.



Prompts

Select point for color legend: - Use your pointing device to select the top left corner of where you want the cut color legend to be displayed.

Legend size <10.0>: Screen display size.

Label all zones or summary [All/<Summary>]? This pertains to the number of elevation labels on the legend.

Keyboard Command: tk_chmap

Prerequisite: Make Strata Surface

Erase Strata Cut Color Map

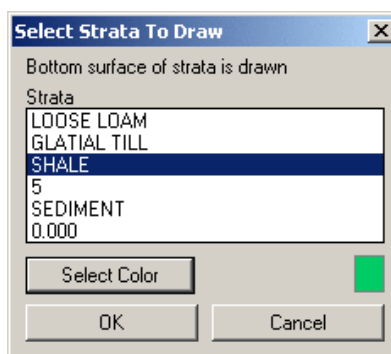
This command will erase all Strata Cut Color information from the screen display.

Keyboard Command: tk_chmap2

Prerequisite: Draw Strata Cut Color Map

Draw Strata Surface

This command will display the selected strata surfaces as 3D faces. The bottom elevation of the strata is drawn.



A color can be selected to distinguish each strata.

Keyboard Command: tk_chplot

Prerequisite: Make Strata Surface

Erase Strata Surface

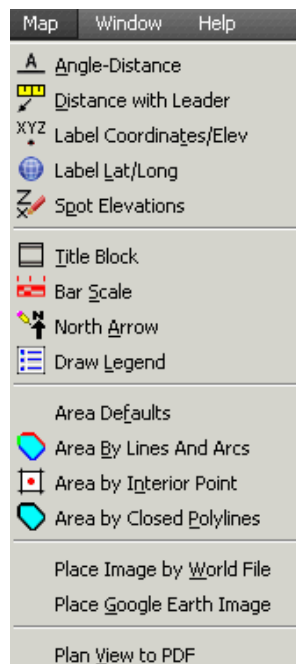
This command will erase all strata surface 3D faces from the screen display.

Keyboard Command: tk_chplot2

Prerequisite: Draw Strata Surface

Map Menu

The Map menu commands are for annotating the plan view. Most of these commands are described in the Annotate section of the Survey module.



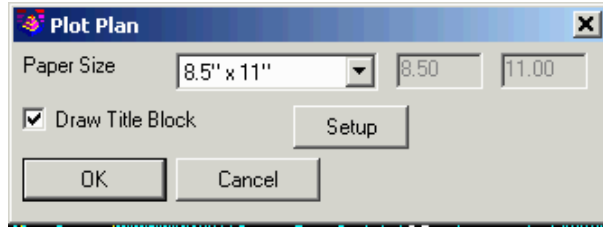
Plan View To PDF

This command is a quick and simple way to create a PDF of drawing. The command prompts for the page size and two points to window the area from the drawing to output. Then the program generates and displays the PDF which can be printed as saved to a file. On the dialog for the page size, there is an option to draw a title block which has settings for the margins and contents for the title block. For more control and advanced output to PDF, create the title block in the drawing and then use the File > Plot command.

Prompts

Plot Plan dialog

Pick Lower Left corner: *pick a point*
Pick Upper Right corner: *pick a point*



Pulldown Menu Location: Map
Keyboard Command: plotwindow
Prerequisite: None

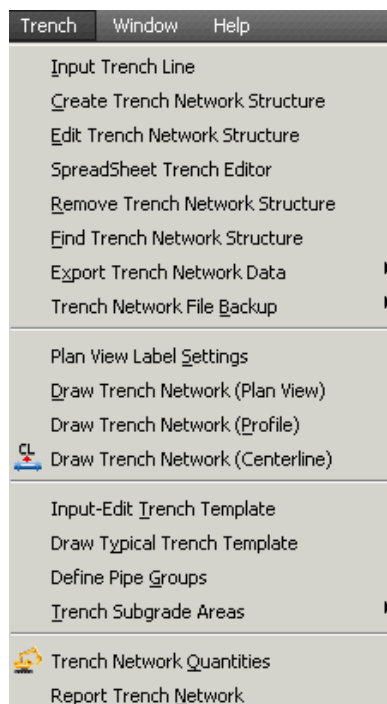
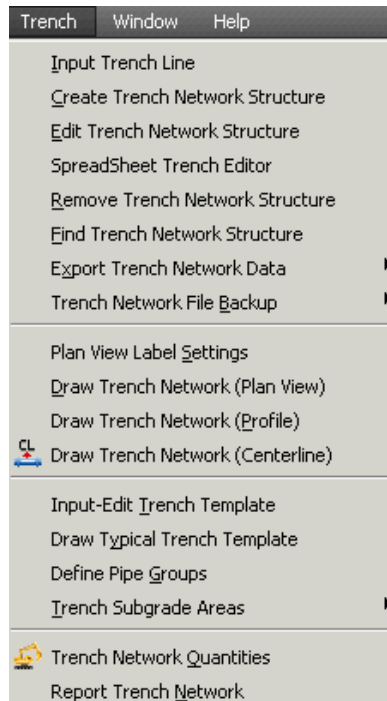


Trench Module

10

Trench Menu

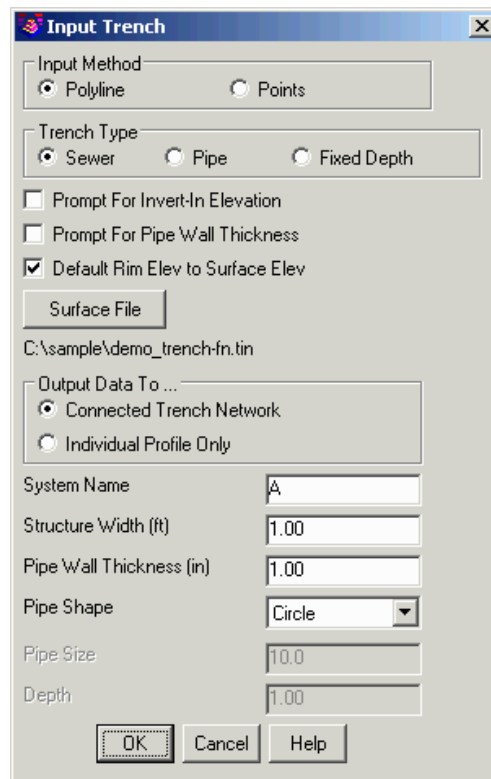
The Trench menu commands layout trench networks and calculate trench quantities.



Input Trench Line

This command allows you to input a trench sewer network structure from polylines or points. It first prompts you the **Input Trench Line Dialog** where you specify the Trench Type, Trench System, and the System Name. The Fixed Depth method automatically assigned the trench elevations at a specified depth from the surface. The Individual

Profile option lets you input one trench reach at a time and save its information to a profile (.pro). The Connected Network option lets you input all the trench polylines on the drawing, merge them into a trench network structure and save the whole structure to a .sew file. For trenching or utilities without Invert-Ins, uncheck Prompt For Invert-In Elevation. If you want to set the Rim Elevation to any surface elevations, check on Default Rim Elev to Surface Elev and then use the Surface Button to select the desired .tin or .flt surface file. Prompt For Pipe Wall Thickness allows you to enter in the pipe thickness that will be used in calculating backfill quantities in two prompts: 1) the interior pipe size and, 2) the thickness of the pipe wall. If this is check off, the value in Pipe Wall Thickness will be automatically added to the Pipe Size for backfill volumes. You can also enter in a Structure Width to be considered in the Cut volumes. Pipe Shape determines the prompting so that you can create Circular, Elliptical, or Rectangular pipe. Click OK to start inputting trench structures.



There are two different types of Input Methods in this command: Polyline or Points. Points allows you to pick freely the location of each sewer structure. With the polyline method, Takeoff extracts the coordinates of all the vertices of the polyline to determine the location of the structures. With both methods you are prompted for the starting station number. Takeoff computes the station values based on the starting station number. Next, you are prompted to enter the Manhole ID (Sewer Trench) or Station ID (Pipe Trench), Invert Elevation, Manhole Elevation (Sewer Trench), and Pipe Size or Pipe Group for every station. You can either enter the values manually or select the texts that represent these values on the drawing. When you finish inputting a polyline, the command would ask you for a profile name to store the profile data if you are doing Individual Profile; otherwise the command would ask you to pick next polyline that is in the same trench network.

Prompts:

Pick a polyline that represents a trench reach: *pick a polyline on your drawing*

Starting Station of trench reach <0.0>: *press Enter to accept 0.0 as the starting station or enter a value*

For station 0.00 ...

Enter/<Select text of Manhole ID>: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*

Enter/⟨Select text of invert elevation⟩: *select the invert elevation text on the drawing or enter Enter on the keyboard to enter the invert elevation value manually*

Enter/⟨Select text of manhole elevation⟩: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manhole elevation value manually*

For station 270.22 ...

Enter/⟨Select text of Manhole ID⟩: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*

Enter/⟨Select text of invert elevation⟩: *select the invert elevation text on the drawing or enter Enter on the keyboard to enter the invert elevation value manually*

Enter/⟨Select text of manhole elevation⟩: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manhole elevation value manually*

Undo/Select/Group/⟨Enter Pipe Size <0.0000⟩: *select the pipe size text on the drawing or enter Enter on the keyboard to enter the pipe size value manually or select Group to enter in a Pipe Group*

For station 425.02 ...

Enter/⟨Select text of Manhole ID⟩: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*

Enter/⟨Select text of invert elevation⟩: *select the invert elevation text on the drawing or enter Enter on the keyboard to enter the invert elevation value manually*

Enter/⟨Select text of manhole elevation⟩: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manhole elevation value manually*

Undo/Select/Group/⟨Enter Pipe Size <0.0000⟩: *select the pipe size text on the drawing or enter Enter on the keyboard to enter the pipe size value manually or select Group to enter in a Pipe Group*

For station 649.73 ...

Enter/⟨Select text of Manhole ID⟩: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*

Enter/⟨Select text of invert elevation⟩: *select the invert elevation text on the drawing or enter Enter on the keyboard to enter the invert elevation value manually*

Enter/⟨Select text of manhole elevation⟩: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manhole elevation value manually*

Undo/Select/Group/⟨Enter Pipe Size <0.0000⟩: *select the pipe size text on the drawing or enter Enter on the keyboard to enter the pipe size value manually or select Group to enter in a Pipe Group*

Another Polyline [⟨Yes⟩/⟨No⟩?] *enter Yes to input another trench reach from a polyline or enter No to finish*

At the end of the command, a file opening dialog would be prompted to you to specify a .sew file name to store the trench network structure.

Prerequisite: A drawing with one or more polylines that represent the trench structure.

Keyboard Command: pline_trench

Create Trench Network Structure

This command allows you to create or modify a trench network structure on a drawing. Before you are able to locate the trench structure, the drawing has to be open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. You can locate the trench structure by one of three methods: picking points on the drawing, entering the point number, or specifying the station and offset of a centerline. If you use centerline method, you need to specify a centerline file. After you locate a point on the drawing, you are prompted the **Sewer Structure Data Dialog** for entering the sewer structure information, such as Structure Name, System Name, Symbol Name, and Elevations. Take a look at the list of the trench points that have been defined. If there is any point that is connected upstream to the current point, you add it to the Upstream Connections list. The Invert Elevation and the Pipe Size fields will be filled with the information of the upstream point. Use Pipe Group allows you to set multiple pipes for the trench run by using an existing or new Pipe Group. Click OK to finish entering the trench structure data. The command will repeatedly ask you to pick a structure point until you hit **Enter** to finish. The trench network structure data is saved in a .sew file.

Trench Structure Data

Structure Data

N: 207183.8 E: 409363.6 Z: 190.8

Structure Name: Symbol Name:

System Name: No Structure (Junction Only) Structure Type:

Structure Template:

Rim Elevation (ft): Structure Width (ft):

Invert Elev OUT (ft): Depth (ft):

Pipe Data

Upstream Connections:

602	A
-----	---

 Available:

35	A
600	A
601	A
602	A

Invert Elev IN (ft): Slope Percent:

Use Pipe Group

Pipe Interior Size (in): Pipe Width (in): Pipe Shape:

Pipe Wall Thickness (in): Trench Type: Min Cover: 1.92

Prompts

By Pick:

Locate by pick point, point number or station-offset [**<Pick>/Number/CL**]? *press Enter to do Pick point*

Loading edges...
 Loaded 4 points and 5 edges
 Created 2 triangles

Pick structure location: *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*
Pick structure location (Enter to end): *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*
Pick structure location (Enter to end): *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*
Pick structure location (Enter to end): *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*
Pick structure location (Enter to end): *press Enter to finish*

By station-offset of CL:

Locate by pick point, point number or station-offset [**<Pick>/Number/CL**]? **CL** (enter CL to do locating trench structure by station-offset of a centerline)
 Specify a centerline file.

Loading edges...
Loaded 4 points and 5 edges
Created 2 triangles

Structure Station: 0 (*enter the station number on the centerline*)
Structure Offset: 200 (*enter the offset from the centerline*)
Sewer Structure Data Dialog: *enter trench structure information*
Structure Station (Enter to end): 100 (*enter the station number on the centerline*)
Structure Offset: 200 (*enter the offset from the centerline*)
Sewer Structure Data Dialog: *enter trench structure information*
Structure Station (Enter to end): *press Enter to finish*

Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface.

Keyboard Command: locate_trench

Edit Trench Network Structure

This command edits the existing trench structure data on the drawing. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name. The command first prompts you to pick a sewer structure on the drawing. If there is no such structure in the .sew file, you would get an error message like this: "Error: unable to locate structure in file C:\temp\takeoff\SANI1x.sew, otherwise this command will restore the trench structure data from the corresponding .sew file and display it on the Sewer Structure Data Dialog for editing. Click OK to confirm your modification. You are prompted to edit another structure point until you press Enter to finish. All modifications are saved in the .sew file.

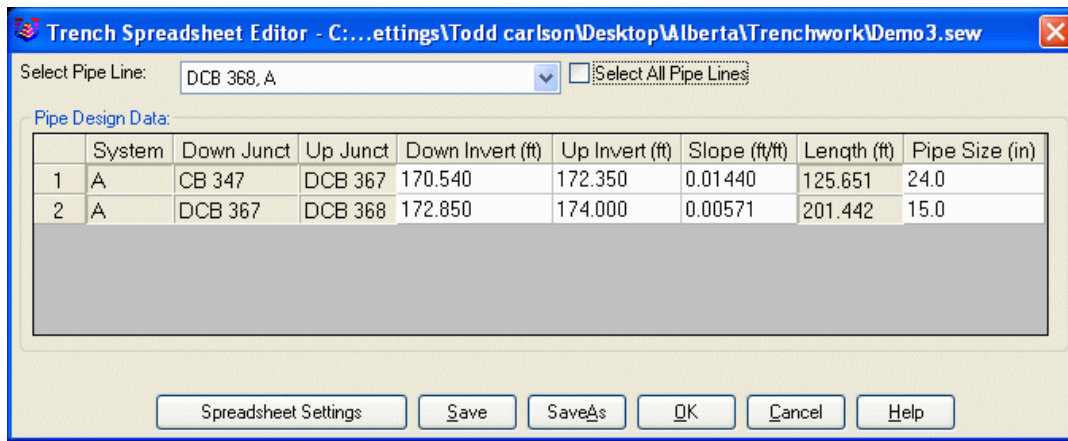
Set Location will return you to the plain view and prompt you for a new location for the structure by either typing in the coordinates or picking on the screen. In the dialog you can change the Structure Name, Symbol, Width, Depth, and Type. Setting a Structure Template will allow you to set the dimensions of the Structure with a .tch file. See Trench Template Editor for details on creating a .tch file. Here you can also manage how the Structure is connect to other Structures. Under Upstream Connections you will see the Structure(s) currently connected to upstream and a list of available Structures on the right. Pick Add to connect to a Structure you have selected under Available, and Remove to disconnect to any selected Structures. Other options are to edit the Rim Elevation, Invert In and Out, as well as Pipe information between your Structure and the highlighted Upstream Connection. Use Pipe Group allows you to set multiple pipes for the trench run by using an existing or new Pipe Group. Min Cover shows you the depth between the Design Surface and top of pipe. Set Min Cover will adjust your Invert In and Out elevations so that you have at least the value you enter as the Min Cover.

Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: edit_trench

Trench Spreadsheet Editor

This command allows you to view and edit existing trench network data in spreadsheet form. Upon running the command, the program will open the .sew file associated with the drawing, or if one has not been established, you will be prompted to select one.



The Trench Spreadsheet Editor allows you to select the Pipe Line you want to edit, or view all the Pipe Lines at once by checking on "Select All Pipe Lines". After selecting a Pipe Line, each segment of the Pipe Line will be displayed as: the downstream connection (Down Junct), upstream connection (Up Junct), the invert in of the downstream manhole (Down Invert), the invert out of the upstream manhole (Up Invert), and the Slope, Length and Pipe Size between the two. Any value between two manholes can be edited except for the Length. Spreadsheet Settings allows you to choose what elements of a segment are displayed.

Click OK to confirm your modification. All modifications are saved in the .sew file.

Prerequisite: Sewer Network File

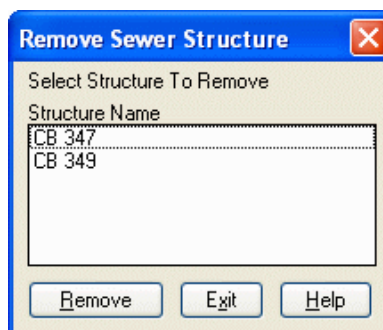
Keyboard Command: edit_trench2

Remove Trench Network Structure

This command removes the existing trench structure data. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name. The command first prompts you to pick a sewer structure on the drawing or to select from a List of your Sewer Structures. If there is no such structure in the file, you will get an error message like this: "Error: unable to locate structure in file C:\temp\takeoff\SANI1x.sew, otherwise this command removes the structure from both the drawing and the .sew file immediately. You are prompted to remove another structure point until you press **Enter** to finish. The removed trench structure points would no longer be found in the .sew file.

Prompts

Select structures to erase by screen pick or name list [<Pick>/List]? Pick to choose from the screen, or List to choose from the below dialog.



Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has

been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: remove_trench

Find Trench Network Structure

This command will center the screen and draw an arrow to the structure you specify.

Prerequisite: a Trench Network

Keyboard Command: findswr

Import Trench Network Data - Utility Network

This command imports a sewer network file (.sew) into the current trench network model.

Pulldown Menu Location: Trench

Keyboard Command: import_swrnet

Prerequisite: .sew file

Import Trench Network Data - Utility Network

This command imports a utility network file (.util) into the current trench network model.

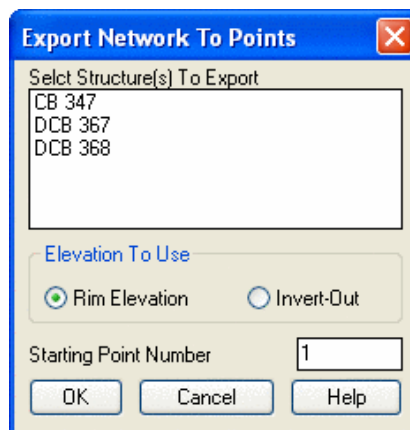
Pulldown Menu Location: Trench

Keyboard Command: import_utilnet

Prerequisite: .util file

Export Trench Network Data

Export to Points



This command will add points at your trench structures and add them into your coordinate file by either the Rim Elevation or the Invert-Out.

Export to Profiles



This command will create a profile file (.pro) of your trench either going Upstream or Downstream. The (.pro) file can then be drawn under Roads->Draw Profile.

Prerequisite: a Trench Network

Keyboard Command: swr2pts, swr2pro

Trench Network File Backup

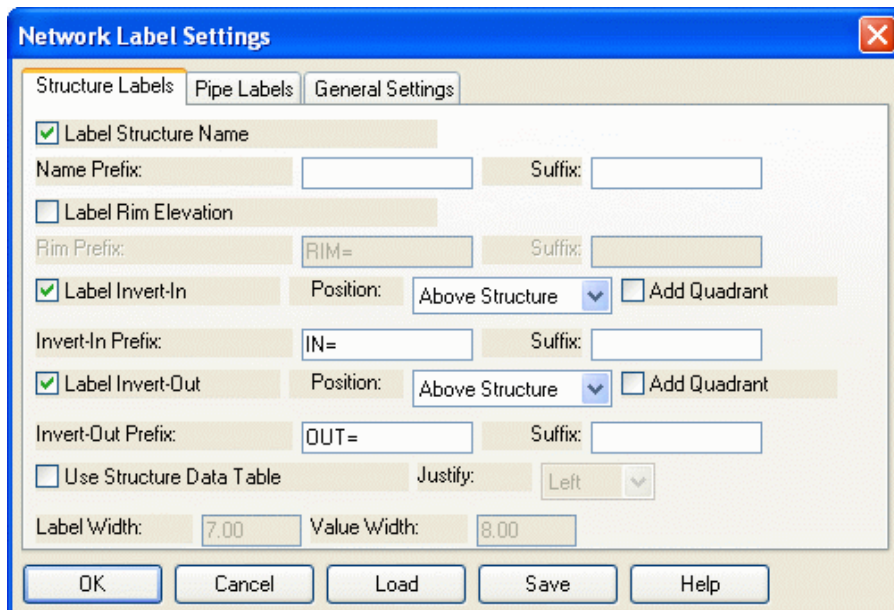
Save Trench Network File saves your trench network as a (.sew) file. *Load Trench Network File* loads a previously saved (.sew) file.

Prerequisite: none

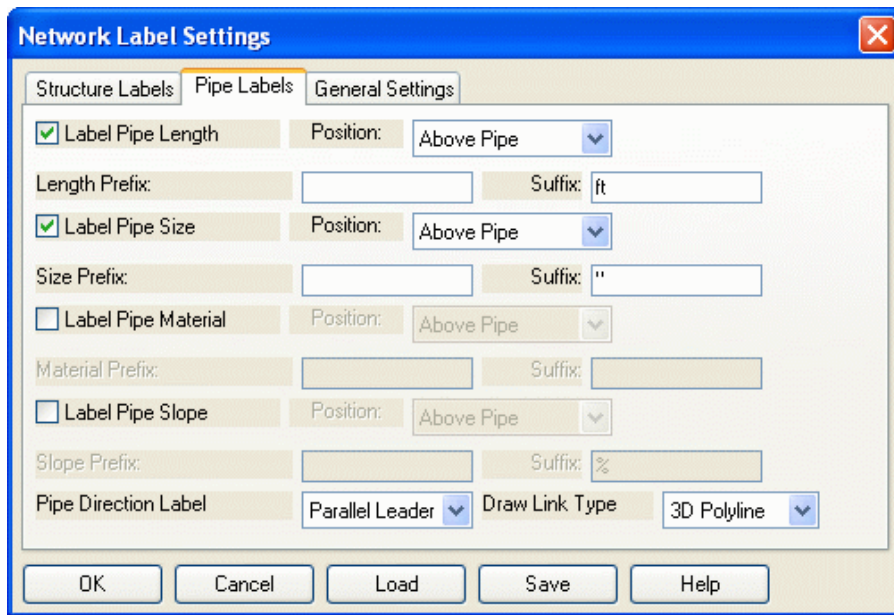
Keyboard Command: save_trench, load_trench

Plain View Label Settings

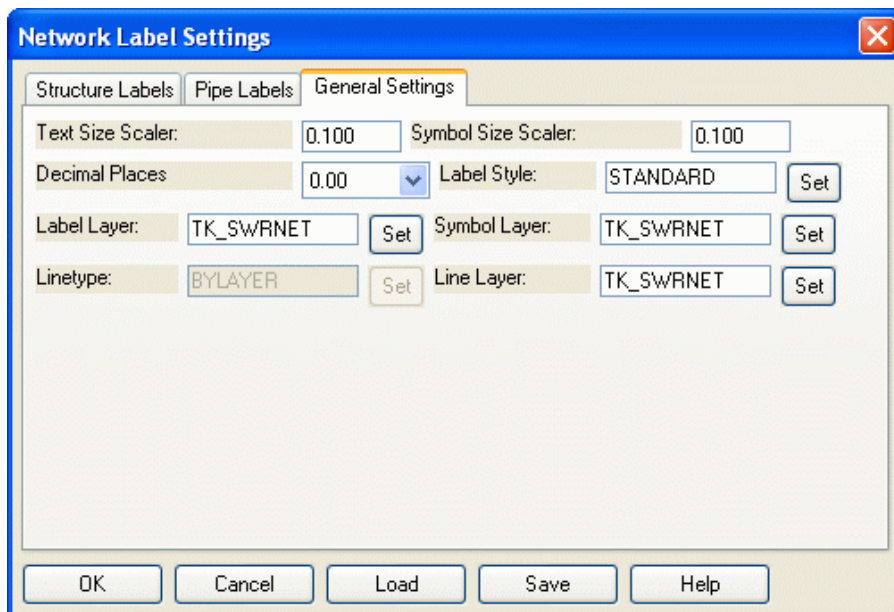
This command allows you to set the labeling for your structures and piping. The below dialog box gives you the option to display the Structure Name, the Rim Elevation, the Invert-In, and Invert-Out. In addition, you can set the Prefixes, Suffixes and labeling location as you so desire. The Use Structure Data Table will create linework around each Structure's labeling.



This below dialog box gives you the option to display the Length, Size, Material, and Slope for you Piping. In addition, you can set the Prefixes, Suffixes and labeling location as you so desire. To specify to which structure the label is meant for, select Arrow On Pipe, Parallel Leader, or None. You can also set the type of linework to draw.



In this dialog you can set the properties for your Symbol and Linework as well as the decimal places to report.



Prerequisite: a trench network

Keyboard Command: swrsetup

Draw Trench Network - Plan

This command draws trench network structures, pipes, and labels on the screen, based on the Plan View Label Settings command and the trench network structure data in the .sew file whose name is as same as the drawing name. Pipe linework will automatically be displayed as 3D faces in the command 3D Viewer Window. If Takeoff can't find a .sew file in the same directory where the drawing is located, nothing would be drawn on the screen.

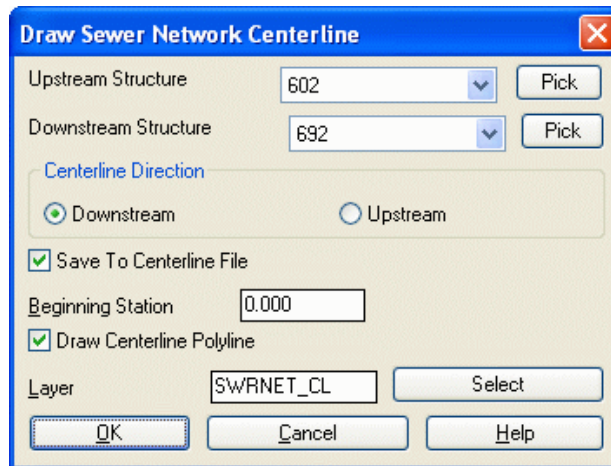
Prerequisite: A open drawing

Keyboard Command: plan_trench

Draw Trench Network Centerline

This command allows you to draw a branch of the trench network structure as a centerline. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the

drawing name, otherwise you would get an error message like "Error: no data in sewer network file". The command first prompts you the **Draw Sewer Network Centerline Dialog**. Select the Upstream and Downstream Structure for the centerline you are about to create. The Centerline Direction determines from which structure the polyline is drawn. You can also choose to save the centerline data to a .cl file with the option of entering in the Beginning Station. In this dialog is the ability to set the Layer name as well. Click OK to draw.



Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: drwswrcl

Draw Trench Network - Profile

This command allows you to draw a branch of the trench network structure as a sewer/pipe profile. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name, otherwise you would get an error message like "Error: no data in sewer network file". The command first prompts you the Draw Sewer Network Dialog. Select the Upstream and Downstream Struct that you want to draw. If you want to draw the existing and final design surface, as well as Strata Surfaces, toggle on Draw Existing Ground Surface, Draw Final Design Surface, and Draw Strata Surfaces options. If your profile is from upstream to downstream, then select the Profile Direction as Downstream, otherwise Upstream. You can also choose to save the profile data to a profile file. Click OK to draw.



Initializing Draw Profile command ...

Draw Sewer Profile Dialog Enter drawing parameters such as Grid scale, text scaler, starting and ending stations

etc. for drawing the sewer profile.

Draw Sewer Profile

Draw Grid Draw Sheet
 Draw Horiz Axis Elev Draw Horiz Label Box
 Draw Break Point Sta Draw Break Point Elev
 Draw Break Point Desc

Starting Station: 0.000
Ending Station: 150.000
Label Scaler: 0.100
 Match Line Elevations
Elevation Range: 30.000

Grid Scale and Interval
Horizontal Scale: 50 Vertical Scale: 10
Horiz Grid Interval: 50 Vert Grid Interval: 10
Horiz Text Interval: 50 Vert Text Interval: 10

Layers Colors Text Styles Linetypes
OK Cancel Load Settings Save Settings Help

Enter general sewer profile settings such as elevations (Rim, Invert-In, Invert-Out) to draw and label.

Draw Sewer Options

General Manhole Pipe

Draw Horiz Axis Annotations
 Draw Annotations At Manholes
 Tick Mark for Station
 Label Invert Elev w/ Vert. Line
 Station Manholes By Another Centerline

Rim Label Position ...
 None
 Label with Leader
 Above Manhole

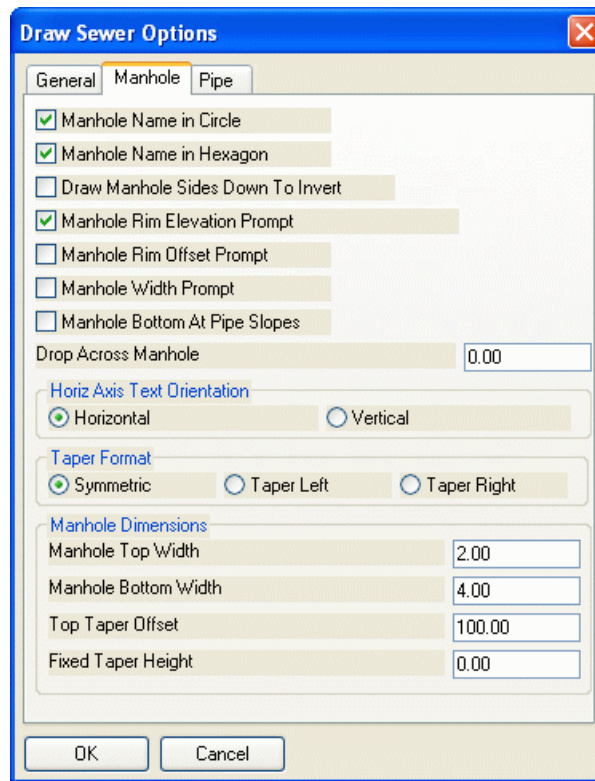
Draw Sump Sump Height: 2.00
 Draw Base Base Height: 1.00
Label Precision

Sewer Label Position
 Pick Row Above Manhole

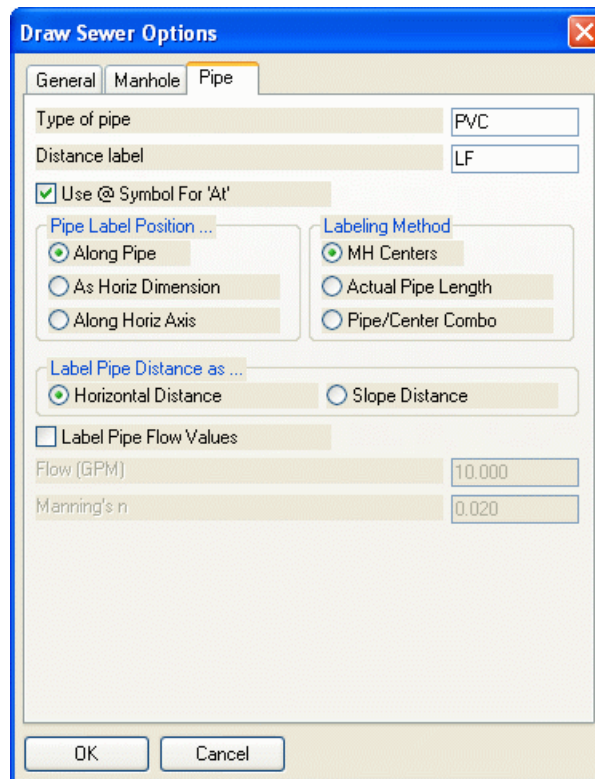
Draw Station Label: STATION
 Draw Reference CL Offset Label: OFFSET
 Draw Surface Label: SURFACE
 Draw Invert In Label: INVERT IN
 Draw Invert Out Label: INVERT OUT
 Draw Description Label: DESC

OK Cancel

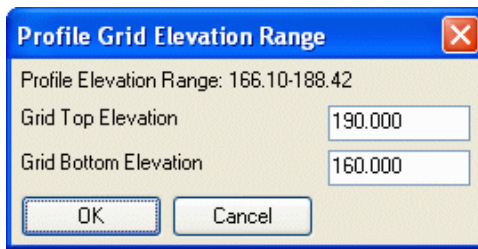
Use the Manhole tab to define what manhole information is labeled in your trench profile.



Use the Pipe tab to define what piping information is labeled in your trench profile.



The command will find the elevation range of your profile and display it at the top of this dialog. Here you can set the elevation top and bottom of the profile's grid.

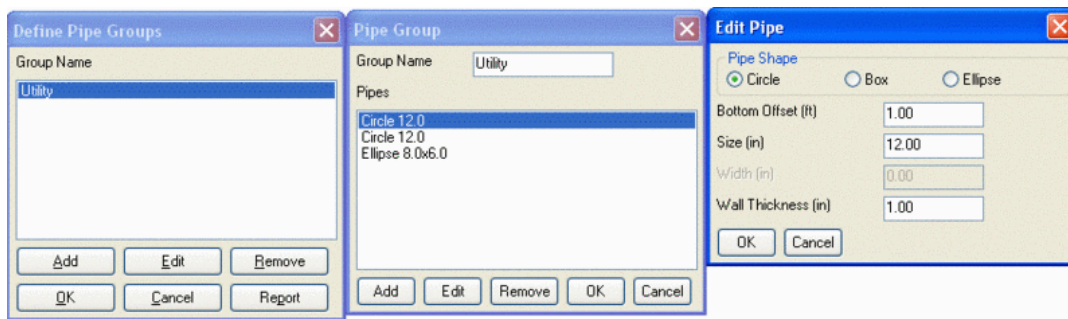


Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: profile_trench

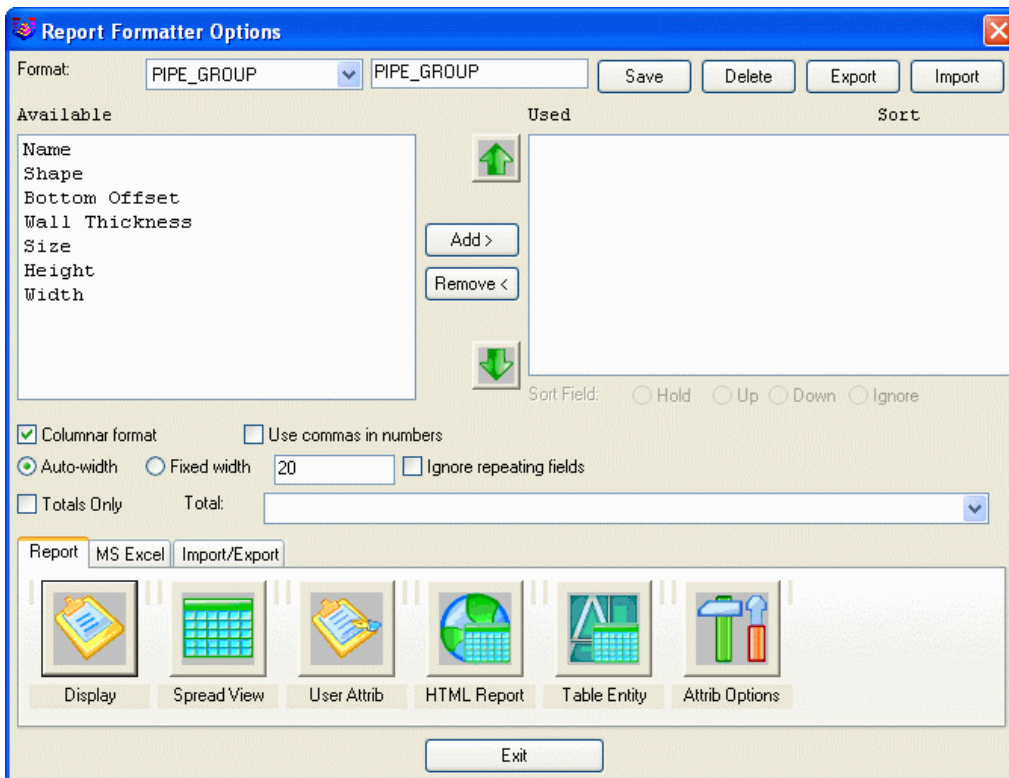
Define Pipe Groups

Pipe Groups allow users to define multiple pipes inside a single trench. Pipe Groups can be applied to a Trench Run during the creation of a Trench Network or after the fact with Edit Trench Structure. In the Define Pipe Groups dialog you can Add, Remove, Edit, or Report different Pipe Groups.



Click Add and a blank Pipe Group dialog will appear. Here you can enter in a Name for the Group and Add different types of pipes into the Group. Clicking Edit or Add in the Point Group dialog will bring up the Edit Pipe dialog. Here you can specify the Pipe Shape, Bottom Offset (from the bottom of the Trench), the Size, Width (when needed), and the Wall Thickness of the pipe.

Report in the Define Pipe Groups dialog will bring up the below dialog:



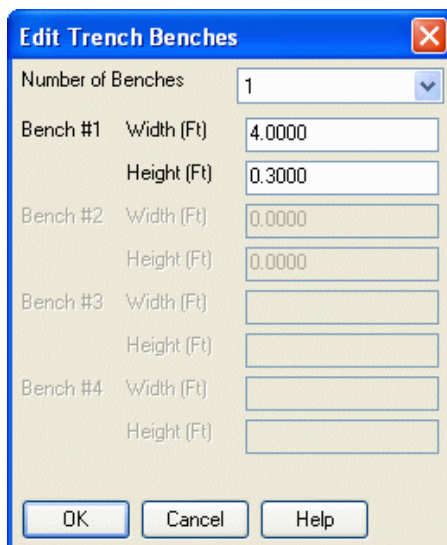
From this dialog you can pick on what you want to report and view it or export it into Excel.

Prerequisite: none

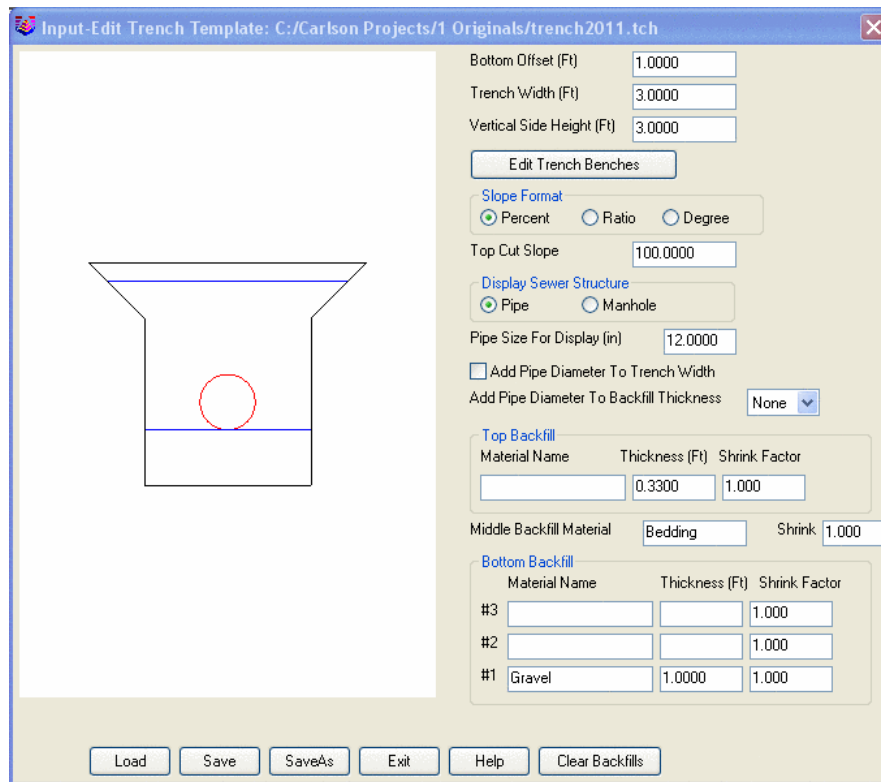
Keyboard Command: define_pipe_grp

Trench Template Editor

This command lets you create a new trench template or modify an existing trench template. It prompts you the **Trench Template Editor Dialog**. If you are modifying a trench template, click the **Load** button on the dialog to open a trench template file and display the template data on the dialog. Enter the dimensions of the trench: bottom offset, trench width and vertical side height. The Edit Trench Benches button will bring up the below dialog, and allows you to enter in up to four benches into your trench.



There are three methods for entering the cut slope, Percent, Ratio and Degree. Choose one of the methods and enter the slope value. Display Sewer Structure allows you to see your pipe or manhole as part of the trench. This is for display purposes only, calculations will be drawn from the pipe size you set in the Trench Network Structure commands. Add Pipe Diameter To Trench Width will increase the size of your trench by the diameter of your different pipe sizes. The Add Pipe Diameter To Backfill Thickness will increase the thickness of backfill material either by half the pipe diameter or the full pipe diameter. For example, when set to Half #1, the backfill thickness for #1 material is increased by half the pipe diameter.



If there is a Top Backfill, enter the Material Name, Thickness and if needed Shrink Factor. The thickness of the Top Backfill is assessed from the top of the trench down. There are three trench Bottom Backfill layers that can be defined. Enter the layer label in the material name field, the depth of the layer in the thickness field. The thickness of these values are assessed from the bottom of the trench up. The Middle Backfill Material is any volume between the Top and Bottom Backfills and can fluctuate depending on the depth of the trench. No set thickness can be applied to the Middle Backfill. Click Save or SaveAs to save the template information in a .tch file, and Click Exit to quit this command.

Prompts:

Trench Template Editor Dialog

Enter the dimensions of the trench template, save the information to a template file (.tch).

Pulldown Menu Location: Trench

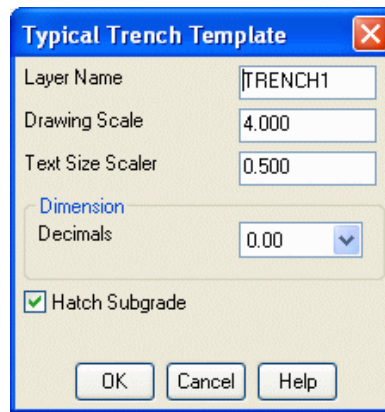
Prerequisite: None

Keyboard Command: make_trench_tpl

Draw Typical Trench Template

This command draws a trench template on the screen. After you select a trench template file (.tch) to draw, a **Typical Trench Template Dialog** is prompted for entering the layer name, drawing scale, text size scaler and selecting how many decimal points you want. You can also hatch the backfill on the drawing. Click OK to draw the

template at the position that you pick on the screen.



Prompts

Pick position to draw template: *pick a position on the screen*

Prerequisite: None.

Keyboard Command: draw_trench_tpl

Trench Subgrade Areas

The purposes of Trench Subgrades is to assign a different type of trench template when a trench passes under a road, building pad, etc.

Set Trench Subgrade Polylines

Choose a closed polyline that defines the area that you want a different trench template for, ie a building pad polyline.

Prerequisite: None.

Keyboard Command: tag_trench_subgrade

Clear Trench Subgrade Polylines

This command untags selected polylines for trench subgrade use.

Prerequisite: Trench Subgrade Polylines

Keyboard Command: untag_trench_subgrade

Hatch Trench Subgrade Area

This command hatches trench subgrades for easy viewing.

Prerequisite: Trench Subgrade Polylines

Keyboard Command: hatch_trench_subgrade

Erase Trench Subgrade Hatch

This removes previous made trench hatching.

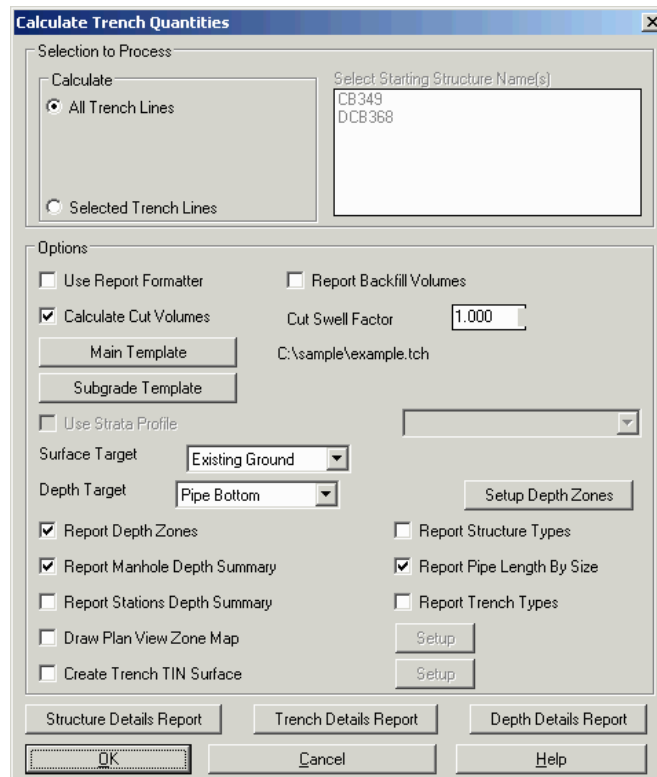
Prerequisite: Trench Subgrade Hatching

Keyboard Command: erase_trench_subgrade

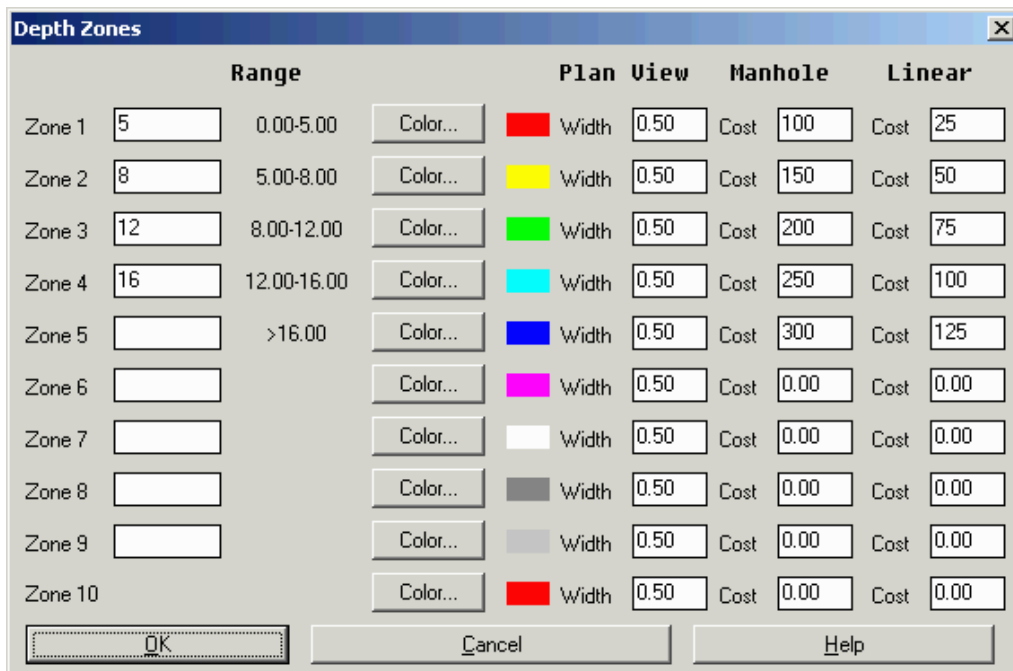
Trench Network Quantities

This command calculates the trench volumes. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name, otherwise you would get an error message like "Error: no data in sewer network file".

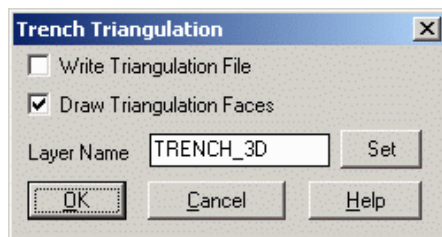
The command loads the trench network data and splits them into individual trench lines and displays them on the **Calculate Trench Quantities Dialog**. You can choose to calculate the trench volume of one trench line or several trench lines at a time. You need to set a **Main Template** in order to calculate volumes. To create a template, run Trench> **Trench Template Editor**. The trench cut volume is multiplied by the Cut Swell Factor. Surface Target determines the Surface that the bottom of the trench is compared to, either: the Existing Surface, the Design, the Existing and Design to minimize cut, or simply to the Rim Elevations (no surface required). Trench Depths can be reported by either the bottom of the trench or bottom of the pipe by using the Depth Target pull-down. If you have Strata Surfaces defined then the program can calculate cut volumes for a strata you select. For more comprehensive reports you can customize, click on the Structure, Trench, and Depth Details Reports buttons. For the Standard Report, click the OK button.

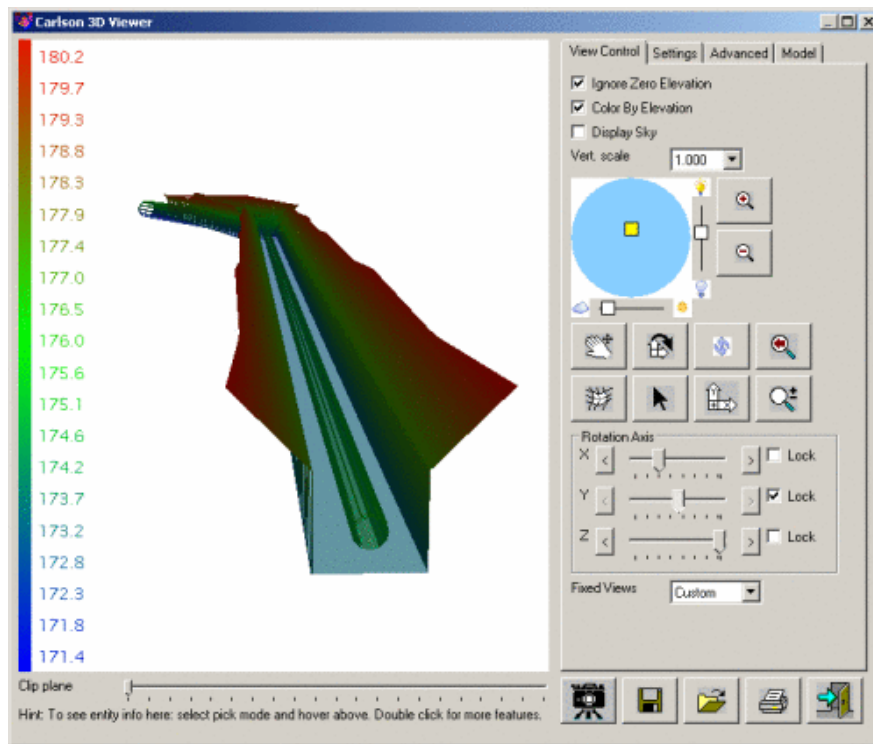


Setup Depth Zones will breakout your trench cut volumes according to user-defined "depth zones" of the trench. You can also color the trench in the drawing by these defined zones. The Width value is used with the Draw Plan View Zone Map for the width of the trench lines. The Manhole Cost is for reporting the structure cost by zone depth. The Linear Cost is for reporting the linear cost for the trench lines by zone depth.



Create Trench TIN Surface will create a triangulation surface of the trench. Under the Setup dialog, the Write Triangulation File option will have the program prompt for a TIN file to create for the trench surface. The Draw Triangulation Faces option creates 3D faces in the drawing on the Layer Name from this dialog. Use "View > 3D Viewer Window" to display the new 3D faces in 3D. Note: Pipes are automatically displayed in 3D from the linework created in "Draw Trench Network (Plan View)". To turn 3D pipes off in the viewer, freeze the layer created with "Draw Trench Network (Plan View)". By default this layer is "SWRNET".





Click OK to compute the template volumes. Backfill quantities take into account pipe size. A report would be shown after the calculation and any depth zone linework and 3D faces will be drawn in the plan view.

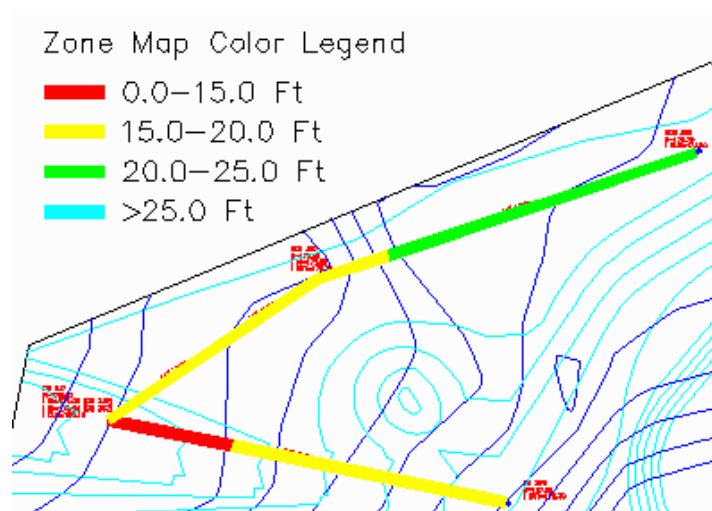
Prompts

Trench Quantities Report Window

Draw zone map color legend on the screen [Yes/No]: *y for Yes*

Pick a point for color legend: *pick a point away from site*

Legend size <10.00>: *Press <Enter> for the default*



Pulldown Menu Location: Trench

Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: calc_trench

Trench Quantities By Profiles

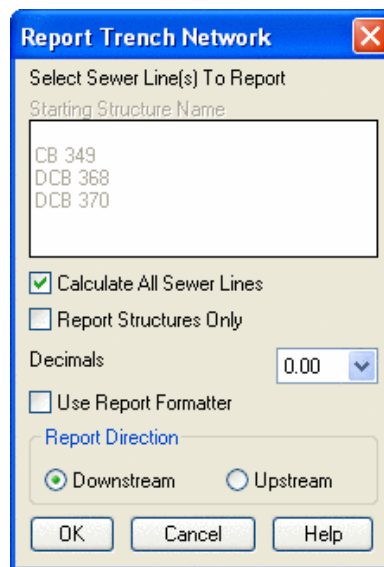
This command calculates trench quantities using two profiles that define the ground surface and the pipeline. See the Pipe Depth Summary command in the Profiles chapter for more information.

Pulldown Menu Location: Trench

Prerequisite: Two profiles

Report Trench Network

This command will a report the Name, Station Distance, Invert-In Slope, Invert-Out Width, the Rim Elevation, Trench Type, Manhole Depth, and the Area Direction for the selected Trench. You may also choose to report the Trench Network from Downstream or Upstream, or just the Structures.



Takeoff Edit : C:\Program Files\Carlson TakeOff R1\scadrprt.tmp

File Edit Settings

Open Save Print Exit Find Screen Hide

Trench Network Report 3/22/2005 09:54

Trench Network File: C:\Program Files\Carlson TakeOff R1\WORK\demo3.sew
 Ground Surface File: C:\Program Files\Carlson TakeOff R1\WORK\demo3-fn.tin

System A
 Sewer Line From CB 349 To CB 347

Name	Station	Invert-In	Invert-Out	Rim Elev	Depth	Type
	Distance	Slope	Width(in)	Min Cover	Direction	
CB 349	0+00.00		178.30	187.80	9.50	
	205.39	3.55	24.00	9.21	S 24°46'49" W	
CB 347	2+05.39	171.00	166.10	176.50	10.40	

System A
 Sewer Line From DCB 368 To CB 347

Name	Station	Invert-In	Invert-Out	Rim Elev	Depth	Type
	Distance	Slope	Width(in)	Min Cover	Direction	
DCB 368	0+00.00		174.00	178.75	4.75	
	201.44	0.82	15.00	4.95	S 05°26'03" E	
DCB 367	2+01.44	172.35	172.35	178.50	6.15	
	125.65	4.97	15.00	5.40	S 21°15'30" E	
CB 347	3+27.09	166.10	166.10	176.50	10.40	

System B
 Sewer Line From DCB 370 To DCB 369

Name	Station	Invert-In	Invert-Out	Rim Elev	Depth	Type
	Distance	Slope	Width(in)	Min Cover	Direction	
DCB 370	0+00.00		166.10	176.50	10.40	
	125.65	132.19	15.00	9.96	N 21°15'30" W	
DCB 369	1+25.65	0.00	0.00	98765432.	98765432.10	

Prerequisite: a sewer line

Keyboard Command: reportswr

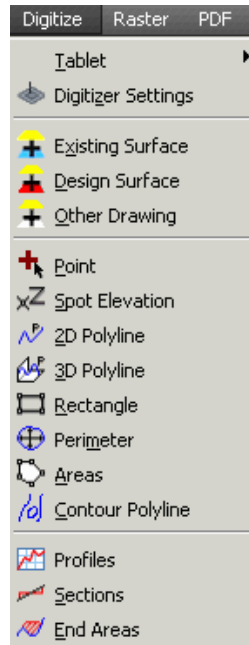


CADnet Module

11

Digitize Menu

The Takeoff menu commands create drawing entities by digitizing on paper plans or on-screen raster images.



Tablet On

Executes AutoCad's TABLET command to set the tablet on. Refer to the AutoCad Reference manual for further information.

Note: Function key [F4] can toggle on/off tablet.

Keyboard Command: tablet

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Tablet Off

Executes AutoCad's TABLET command to set the tablet on. Refer to the AutoCad Reference manual for further information.

Note: Function key [F4] can toggle on/off tablet.

Keyboard Command: tablet

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

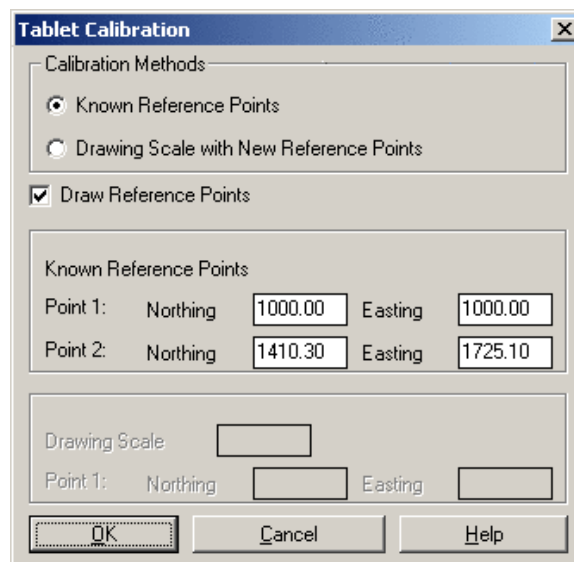
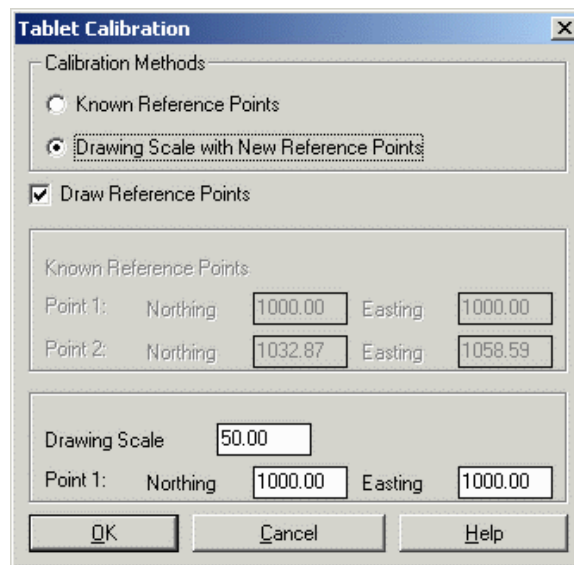
Tablet Calibrate

You can calibrate the tablet/digitizer in one of two ways: **Known Reference Points** or **Drawing Scale with New Reference Points**. Reference points are the foundations of whatever data you digitize into the computer. Takeoff bases everything from drawing location to drawing scale on the reference points you digitize.

Drawing Scale with New Reference Points method is very convenient when you don't know the precise coordinates of the entities on your drawing. As long as you can obtain the drawing scale from your plan, this method can establish a coordinate system relative to the position of the plan on the digitizer board. In addition to the drawing scale, you are required to enter a random coordinate for the first reference point, the default coordinate is (1000,1000). Takeoff would computer the coordinate of the second reference point that you pick based on the first point. The coordinates of these two reference points would be saved and will be display on the **Tablet Calibration Dialog** next time when you calibrate the tablet, so you can digitize back to the previous coordinates using **Know Reference Points** method if you are working on the same drawing, though you might have moved or rotated your drawing on the digitize board..

If you know the precise coordinates of two points, you can select **Known Reference Points** method, which establishes a coordinate system that is exactly match the coordinates in the field or on your drawing. Furthermore, Takeoff saves the coordinates of the two reference points from previous calibration and displays them on the **Tablet Calibration Dialog** next time when you calibrate the tablet. If you want to continue to work on the same drawing, you can use the **Know Reference Points** method with the saved coordinates to digitize back to your previous coordinates although you might have moved or rotated your drawing on the digitizer board.

For accurate takeoff calculations, choose two points that can be easily found in the field and are farther apart rather than closer together.



Prompts

Tablet Calibration Dialog

Specify the Calibration Methods. If you select Drawing Scale method, enter the drawing scale and the coordinate of the first reference point. Otherwise enter the exact coordinates of the first and second reference points.

Pick first reference point: *pick a point on the drawing*

Pick second reference point: *pick another point on the drawing*

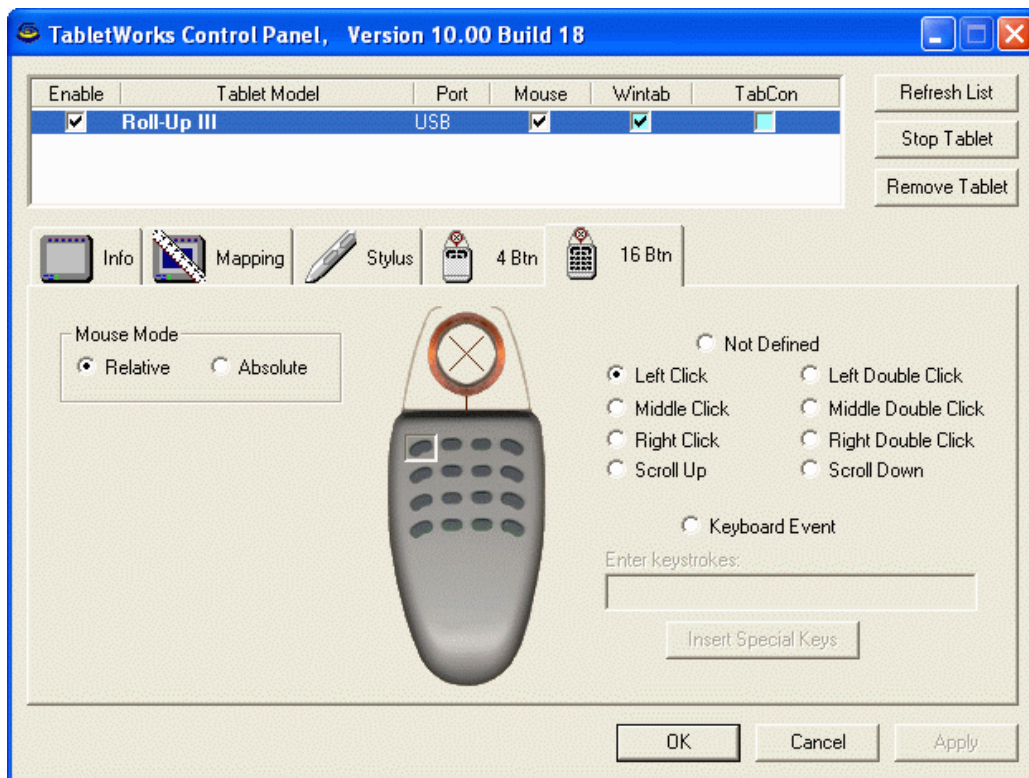
Keyboard Command: digsetup

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Digitizer Setup

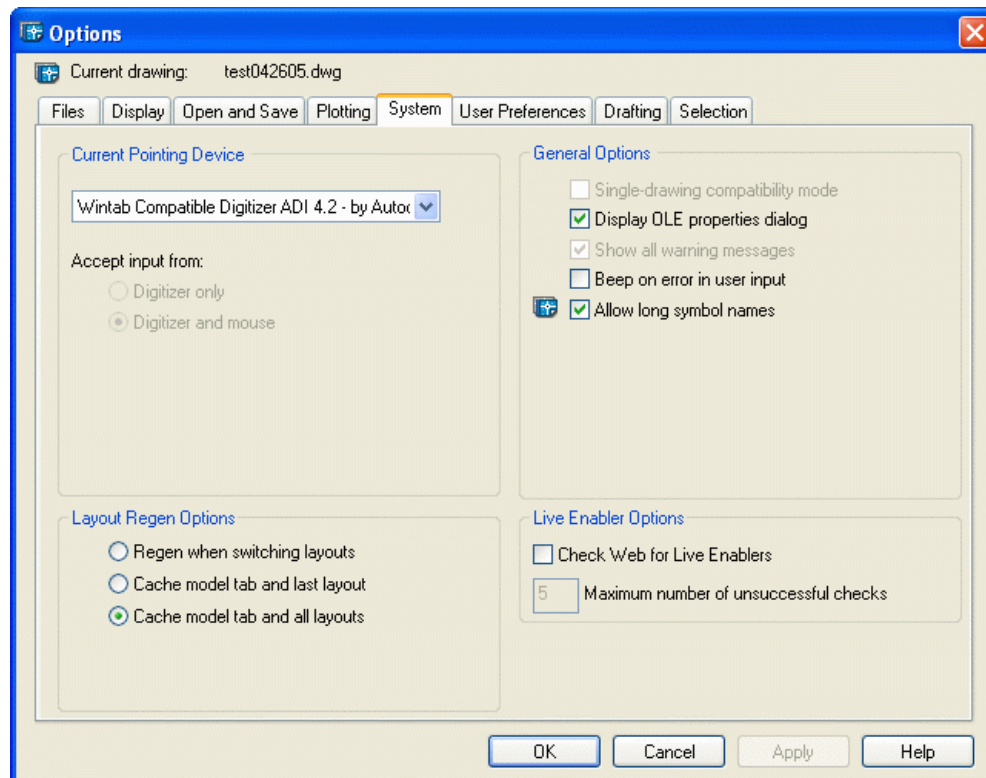
Digitizing is the act of inputting data into the computer by tracing the data from a plan sheet. You need to have a digitizer board, puck, Carlson Takeoff, your computer and your plan to do digitizing. Wintab is a digitizer driver that lets you to use the digitizer cursor as both a digitizer cursor and a mouse. You need to install Wintab when you install Carlson Takeoff. Wintab can be downloaded from GTCO web site: <http://www.gtcocalcomp.com/supportgtcosoftware.htm>. Select the driver version that suits the type of your digitizer board well.

After you installed Wintab driver on your computer, you set up you digitizer to the correct point mode. In Windows 2000/XP, go to **Start->Settings->Control Panel->TabletWorks**, high light the **16-Btn Cursor**, and select **Mouse** as the **Pointing Mode**, which lets the digitizer cursor moves relatively to the screen coordinates. This step is indicated in the following **TabletWorks Control Panel** dialog.



The next is to set up the pointing device in Carlson Takeoff. Open up Takeoff and go to pull-down **Settings-**

>**Preferences**, click tab **System**, select **Wintab Compatible Digitizer** as **Current Pointing Device**, and set the **Accept input from** to **Digitize and mouse**. Please refer to the following **Options** dialog.



Now, you are ready to use your digitizer. On the bottom of the screen, there is a tray icon **TABLET** on the right side of **MODEL**. You can use accelerator key **F4** to toggle on/off the tablet.

Save Tablet Calibration

This command saves current tablet calibration to a file. You are prompted to enter a file name.

Keyboard Command: tablet1

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Load Tablet Calibration

This command restores the tablet calibration parameters from a file and load it into the current drawing. You are prompted to specify a file name.

Keyboard Command: tablet2

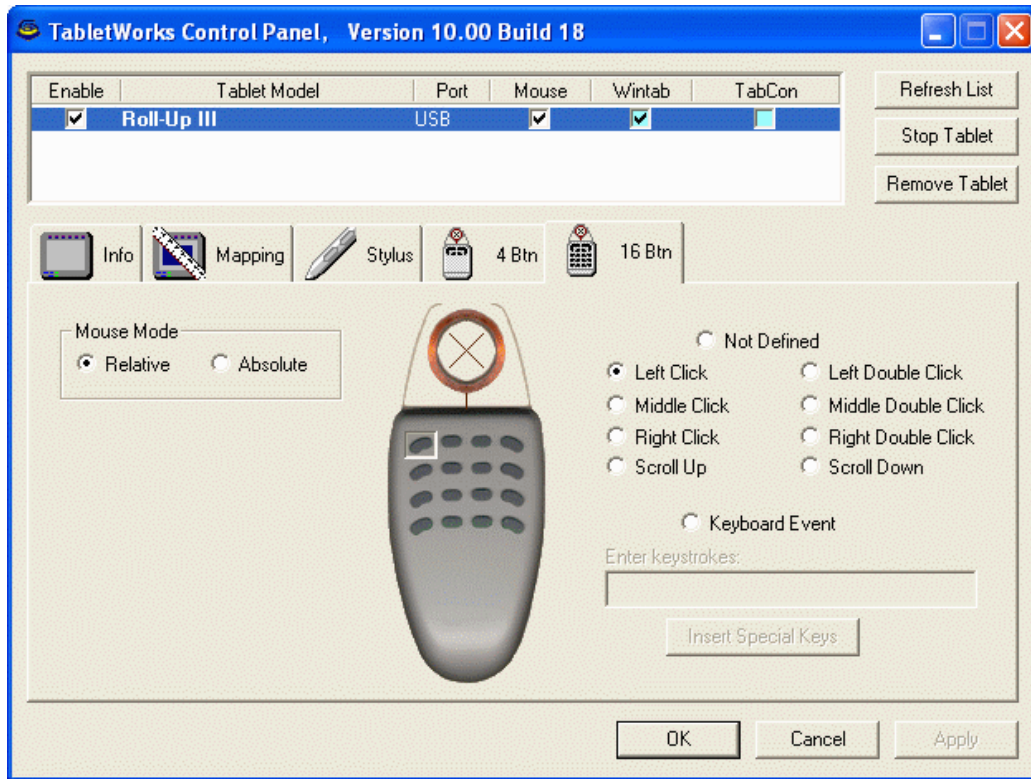
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. The calibration file should be associated to the current drawing, and the current drawing shouldn't have been moved on the digitizer board since last calibration.

Digitize

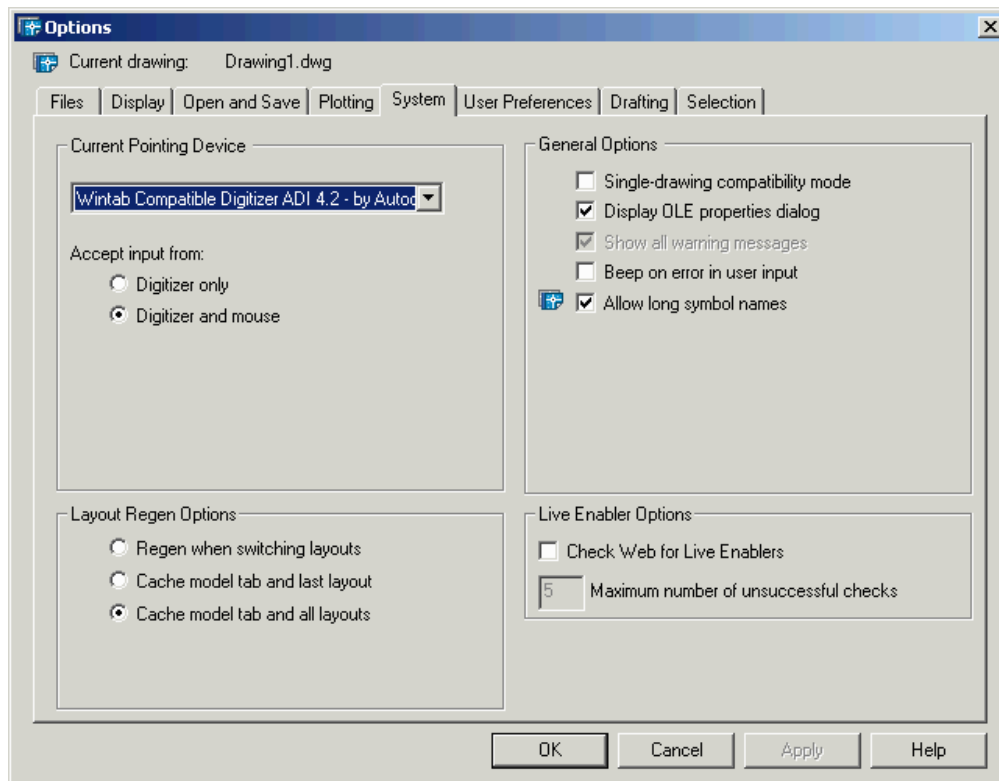
Digitizing is the act of inputting data into the computer by tracing the data from a plan sheet. You need to have a digitizer board, puck, Carlson Takeoff, your computer and your plan to do digitizing. Wintab is a digitizer driver that lets you to use the digitizer cursor as both a digitizer cursor and a mouse. You

need to install Wintab when you install Carlson Takeoff. Wintab can be downloaded from GTCO web site: <http://www.gtcocalcomp.com/supportgtcosoftware.htm>. Select the driver version that suits the type of your digitizer board well.

After you installed Wintab driver on your computer, you set up you digitizer to the correct point mode. In Windows 2000/XP, go to **Start->Settings->Control Panel->TabletWorks**, high light the **16-Btn Cursor**, and select **Mouse** as the **Pointing Mode**, which lets the digitizer cursor moves relatively to the screen coordinates. This step is indicated in the following **TabletWorks Control Panel** dialog.



The next is to set up the pointing device in Carlson Takeoff. Open up Takeoff and go to pull-down **Settings->Preferences**, click tab **System**, select **Wintab Compatible Digitizer as Current Pointing Device**, and set the **Accept input from** to **Digitize and mouse**. Please refer to the following **Options** dialog.



Now, you are ready to use your digitizer. On the bottom of the screen, there is a tray icon TABLET on the right side of MODEL. You can use accelerator key **F4** to toggle on/off the tablet.

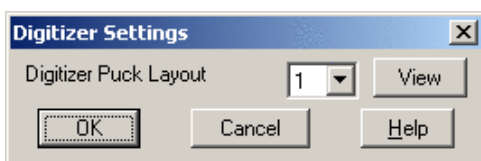
The 16-button puck can be used as either a mouse or a digitizer. It's very important to understand how the 16 buttons are mapped in both modes.

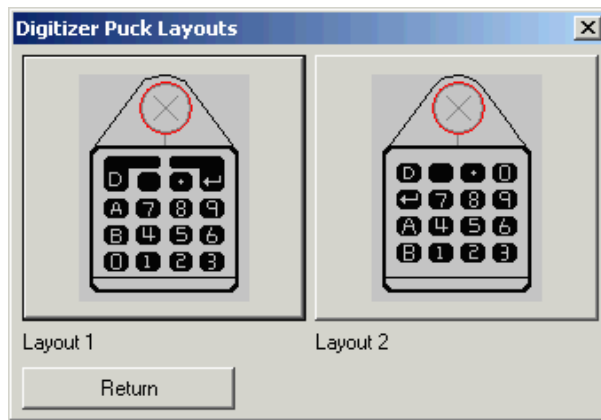
Mouse Mode:

When the tablet is off, the puck is in Mouse Mode. The top-left button is the left mouse click, and the top-right button is the right mouse click. The labels on the other buttons do not mean anything. All buttons are mapped as same as the buttons of the default pointing device in AutoCad . Please refer to AutoCad Reference manual for further information.

Digitize Mode:

When tablet has been calibrated and is on, the puck is in digitize mode. It is mapped as a small keyboard, which enables you to enter numerous values such as elevation, thickness and offset etc., and also provide you some functionality to digitize various entities. Currently there are two puck layouts in Takeoff, shown in the figure below. After you install Carlson Takeoff and finish setting up the digitizer, you go to the pull-down menu **Digitize->Puck Layout** to select a 16-button puck layout. A button mapping would be created and Takeoff would recognize the buttons as represented.





Layout 1 is Carlson Puck Layout, which is the most common layout used in Carlson Takeoff. Layout 2 is for users who don't have a Carlson Puck. If your puck is different than these two layouts, please contact Technical Support for help setting the mapping for your 16 button puck.

Prompts:

Digitizer Settings Dialog

Specify the Digitizer Puck Layout to layout 1 or 2

Keyboard Command:

dig_config

Pull-Down Menu Location:

Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Function:

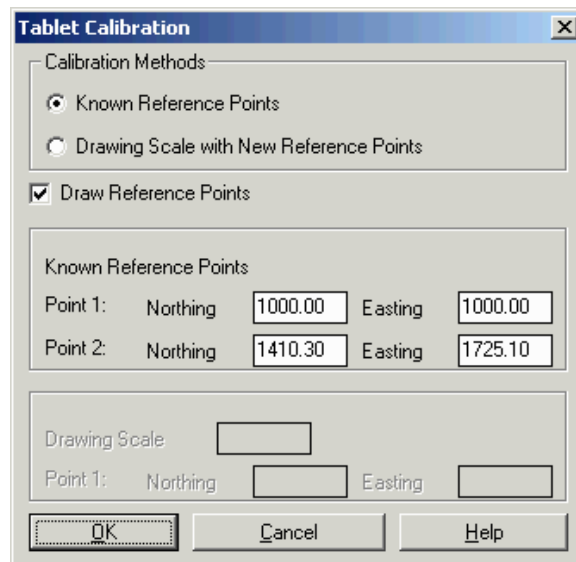
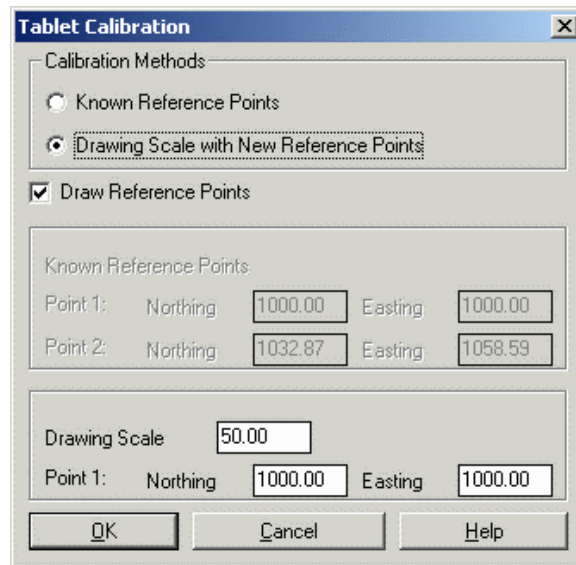
You can calibrate the tablet/digitizer in one of two ways: **Known Reference Points** or **Drawing Scale with New Reference Points**. Reference points are the foundations of whatever data you digitize into the computer. Takeoff bases everything from drawing location to drawing scale on the reference points you digitize.

Drawing Scale with New Reference Points method is very convenient when you don't know the precise coordinates of the entities on your drawing. As long as you can obtain the drawing scale from your plan, this method can establish a coordinate system relative to the position of the plan on the digitizer board. In addition to the drawing scale, you are required to enter a random coordinate for the first reference point, the default coordinate is (1000,1000). Takeoff would computer the coordinate of the second reference point that you pick based on the first

point. The coordinates of these two reference points would be saved and will be display on the **Tablet Calibration Dialog** next time when you calibrate the tablet, so you can digitize back to the previous coordinates using **Know Reference Points** method if you are working on the same drawing, though you might have moved or rotated your drawing on the digitize board..

If you know the precise coordinates of two points, you can select **Known Reference Points** method, which establishes a coordinate system that is exactly match the coordinates in the field or on your drawing. Furthermore, Takeoff saves the coordinates of the two reference points from previous calibration and displays them on the **Tablet Calibration Dialog** next time when you calibrate the tablet. If you want to continue to work on the same drawing, you can use the **Know Reference Points** method with the saved coordinates to digitize back to your previous coordinates although you might have moved or rotated your drawing on the digitizer board.

For accurate takeoff calculations, choose two points that can be easily found in the field and are farther apart rather than closer together.



Prompts:

Tablet Calibration Dialog

Specify the Calibration Methods. If you select Drawing Scale method, enter the drawing scale and the coordinate of the first reference point. Otherwise enter the exact coordinates of the first and second reference points.

Pick first reference point:*pick a point on the drawing*

Pick second reference point:*pick another point on the drawing*

Keyboard Command:

digsetup

Pull-Down Menu Location:

Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Function:

Executes AutoCad's TABLET command to set the tablet on. Refer to the AutoCad Reference manual for further information.

P.S. Function key [F4] can toggle on/off tablet.

Keyboard Command:

tablet

Pull-Down Menu Location:

Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Function:

Executes AutoCad's TABLET command to set the tablet on. Refer to the AutoCad Reference manual for further information.

P.S. Function key [F4] can toggle on/off tablet.

Keyboard Command:

tablet

Pull-Down Menu Location:

Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Function:

This command saves current tablet calibration to a file. You are prompted to enter a file name.

Keyboard Command:

tablet1

Pull-Down Menu Location:

Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Function:

This command restores the tablet calibration parameters from a file and load it into the current drawing. You are prompted to specify a file name.

Keyboard Command:

Pull-Down Menu Location:

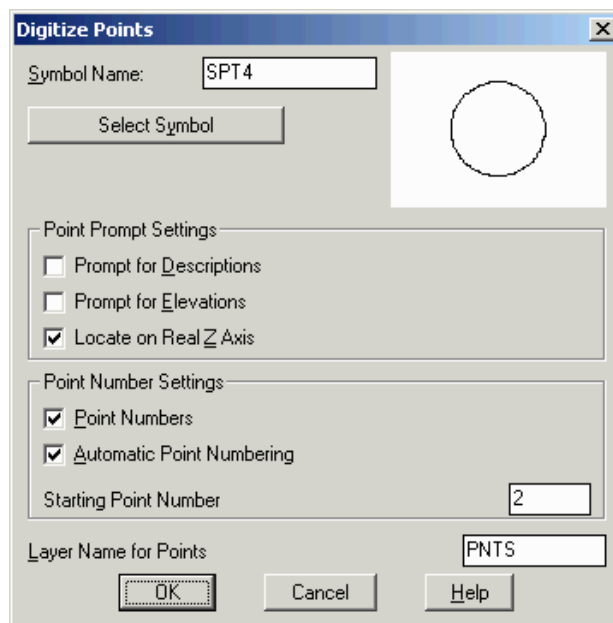
Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. The calibration file should be associated to the current drawing, and the current drawing shouldn't have been moved on the digitizer board since last calibration.

Function:

This command allows you to digitize individual points one at a time. The first time it prompts you the **Digitize Points Dialog** for entering point symbol styles, point prompt settings and number settings, starting point number and layer name. If you want to enter the elevation and description for each point, select **Prompt for Descriptions** and **Prompt for Elevations**. After having digitized a point, you can continue to digitize next point by picking the point on the drawing. The command defaults to the last layer name, point symbol, elevation, description and the last point number plus 1. If you have finished digitizing points, press **Enter** to finish.



Prompts:

Digitize Points Dialog

Specify a layer name and select the point symbol, point prompt settings and number settings.

Pick point to create (Enter to end): *pick a point on the drawing*

Select/<Enter Point Elevation <>>: *enter the elevation or type <Select> to select the elevation text on the screen*

Enter Point Description <>: *enter the point description*

Result like "N: 1231.16 E: 1099.17 Z: 30.00" would be display on the command line, and a point would be drawn on the screen with the text of its number, elevation and description.

Pick point to create (Enter to end):*pick next point or press Enter to finish digitizing points*

Keyboard Command:

dig_pt

Pull-Down Menu Location:

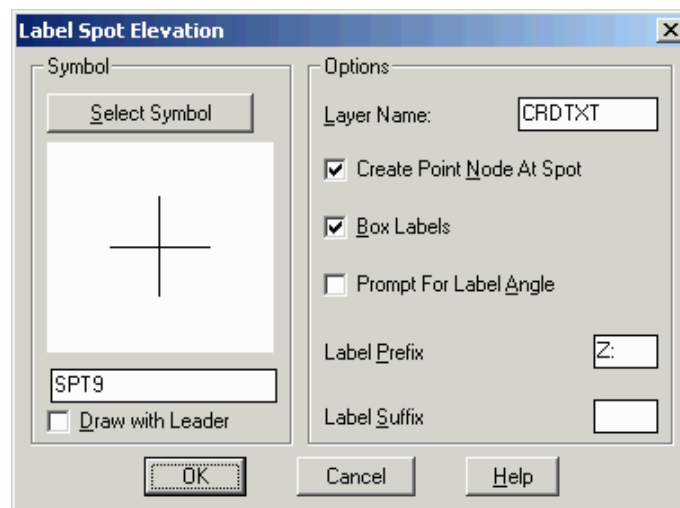
Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Function:

This command allows you to label points with their elevation. The point can either be digitized from a drawing, picked on a screen or specified by a point number. The command first prompts you the **Label Spot Elevation Dialog** for entering layer name, label prefix and suffix and symbol types etc. Click OK to start. After specifying the point, the command prompts you to enter the elevation if its elevation is unknown and then pick an angle from the location of the point to label the elevation. You can repeat labeling points until you press **Enter** to finish.



Prompts:

Label Spot Elevation Dialog

Specify a layer name, label prefix and suffix and select the spot symbol.

Point to Label ?

Pick point or point number: 2 (enter a point number)

PointNo. Northing(Y) Easting(X) Elev(Z) Description

2 1231.16 1099.17 30.00 bb

Note: if the point number you entered is not in the drawing, you will be prompted again to pick point or enter a point number.

Elevation <30.000>: *press enter*

Pick angle for label: *pick an angle from the spot*

Point to Label (ENTER to End)?

Pick point or point number: *pick a point on the drawing*

Elevation <0.000>: *enter elevation*

Pick angle for label: *pick an angle from the spot*

Point to Label (ENTER to End)?

Pick point or point number: *press enter to finish*

Keyboard Command:

labspot

Pull-Down Menu Location:

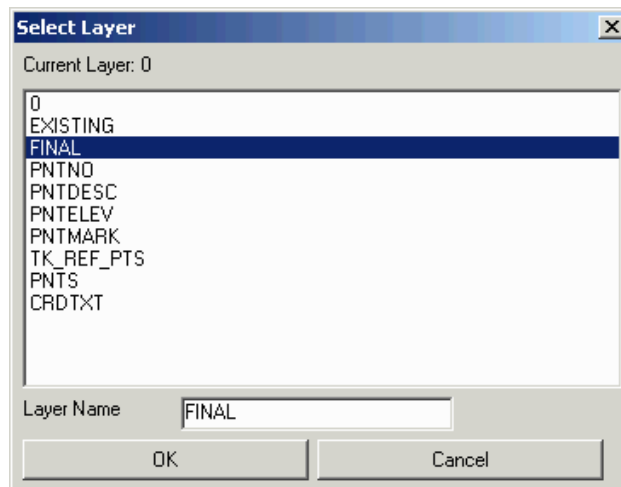
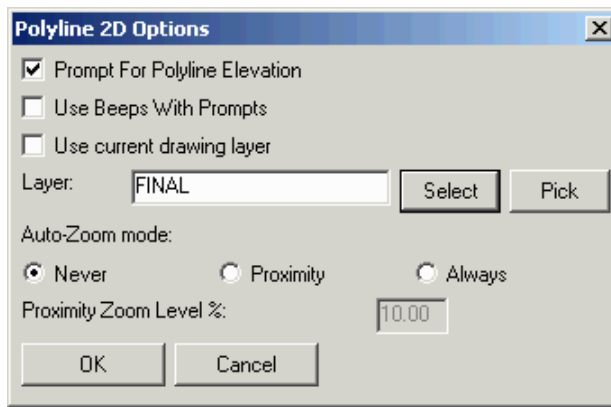
Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Function:

2D polyline is a line of connected points that have the same elevation. This command lets you digitize a 2D polyline by picking points along the lines on the drawing. It prompts you first the **Polyline 2D Options Dialog** for entering the layer name. **Prompt For Polyline Elevation** option allows you to enter the elevation for each polyline, otherwise all 2D polylines have 0.0 elevation. **Auto-Zoom** mode would automatically zoom the display to center around the last point when you get near the edge of the screen while picking points. There are three ways to enter a layer name, **Use current drawing layer**, **Select** from a list of layer name, or **Pick** an entity on the screen to get its layer name. While digitizing a polyline, the command keeps prompting you to pick next point until your **press Enter** to finish digitizing, or **press A** on the puck or **enter Close** on the keyboard to close the polyline on itself. If you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a polyline, the command prompts your to digitize another polyline until you **press B** or **enter No**.



Prompts:

Polyline 2D Options Dialog

Enter the layer name and select the options of Prompt For Polyline Elevation and Auto-Zoom mode etc.

Enter default elevation <0.00>: 100

First point: *pick a point on the drawing using puck*

Segment length: 0.00, Total length: 0.00

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Segment length: 119.03, Total length: 119.03

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Segment length: 121.76, Total length: 240.80

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Segment length: 115.23, Total length: 356.03

Close[A]/Undo[B]/Pick next point (Enter to end): *press enter to finish digitizing or press A to close the polyline*

Digitize Another FINAL Polyline [Yes(A)/<No(B)>]?: *press A on the puck or enter Yes on the keyboard to digitize next 2D polyline, press B on the puck or enter No on the keyboard to finish digitizing 2D polyline.*

Keyboard Command:

dig_2dp

Pull-Down Menu Location:

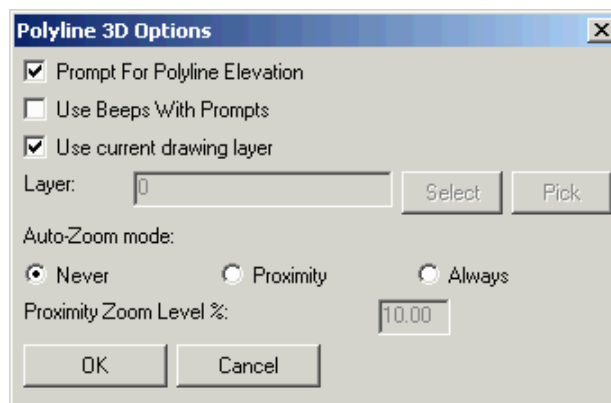
Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Function:

3D polyline is a line of connected points that have various elevations, and the slope between points is constant. It can be used in defining pads, excavations, drainage ditched and slopes from proposed design features to meet existing site conditions. This command lets you digitize a 3D polyline by picking points along the lines on the drawing. It prompts you first the **Polyline 3D Options Dialog** for entering the layer name. **Prompt For Polyline Elevation** option allows you to enter the elevation for each point you pick. If you don't select this option, all points have 0.0 elevation and you get a 2D polyline. There are four ways to enter elevations: known elevation of the point, slope from previous point, ratio from previous point and degree from previous point. You can choose one of the methods between picking points. **Auto-Zoom** mode would automatically zoom the display to center around the last point when you get near the edge of the screen while picking points. While digitizing a polyline, the command keeps prompting you to pick next point until your **press Enter** to finish digitizing, or **press A** on the puck or **enter Close** on the keyboard to close the polyline on itself. If you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a polyline, the command prompts your to digitize another polyline until you **press B** or **enter No**.



Prompts:

Polyline 3D Options Dialog

Enter the layer name, toggle on Prompt For Polyline Elevation and select the Auto-Zoom mode.

First point: *pick a point on the drawing*

Elevation <0.00>: *12 (enter the elevation)*

Z: 12.00, Hz dist: 0.00, Slope dist: 0.00, Slope: 0.0% Ratio: 0.0:1

Close[A]/Undo[B]/Pick next point (Enter to end): *pick a point on the drawing*

Slope/Ratio/Point/Degree/<Elevation> <0.00>: *S (switch to entering Slope)*

Ratio/Point/Elevation/Degree/<Slope> <-8.24>: *12 (enter the slope)*

Z: 29.47, Hz dist: 145.56, Slope dist: 146.61, Slope: 12.0% Ratio: 8.3:1

Close[A]/Undo[B]/Pick next point (Enter to end): *pick a point on the drawing*

Ratio/Point/Elevation/Degree/<Slope> <-15.31>: *R (switch to entering Ratio)*

Slope/Point/Elevation/Degree/<Ratio> <-6.53>: *23 (enter the ratio)*

Z: 37.83, Hz dist: 192.45, Slope dist: 192.63, Slope: 4.3% Ratio: 23.0:1

Close[A]/Undo[B]/Pick next point (Enter to end): *pick a point on the drawing*

Slope/Point/Elevation/Degree/<Ratio> <-5.44>: *D (switch to entering Degree)*

Slope/Ratio/Elevation/Point<Degree(dd.dddd)> <0.0000>: 22 (enter the degree)

Z: 121.04, Hz dist: 205.94, Slope dist: 222.11, Slope: 40.4% Ratio: 2.5:1

Close[A]/Undo[B]/Pick next point (Enter to end): pick a point on the drawing

Slope/Ratio/Elevation/Point<Degree(dd.dddd)> <0.0000>: P (switch to entering Point)

Slope/Ratio/Elevation/Degree/<Point> <0.00>: 34 (enter Z value of the point)

Z: 34.00, Hz dist: 206.05, Slope dist: 223.68, Slope: -42.2% Ratio: -2.4:1

Close[A]/Undo[B]/Pick next point (Enter to end): press Enter to finish digitizing current polyline, or enter A to close the polyline

Digitize Another 0 Polyline [Yes(A)/<No(B)>]? press A on the puck or enter Yes on the keyboard to digitize next 3D polyline, press B on the puck or enter No on the keyboard to finish.

Keyboard Command:

dig_3dp

Pull-Down Menu Location:

Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Function:

Perimeter is a 2D polyline that all points on it have the same elevation. It can be used as boundary polyline of your targets on your drawing. This command allows you to digitize a perimeter by picking points on the drawing. While digitizing a polyline, the command keeps prompting you to pick next point until you **press Enter** to finish digitizing, or **press A** on the puck or **enter Close** on the keyboard to close the polyline on itself. If you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a perimeter, the command prompts your to digitize another polyline until you **press B** or **enter No**.

Prompts:

First point: pick a point on the drawing using puck

Segment length: 0.00, Total length: 0.00

Close[A]/Undo[B]/Pick next point (Enter to end): pick next point

Segment length: 104.27, Total length: 104.27

Close[A]/Undo[B]/Pick next point (Enter to end): pick next point

Segment length: 153.14, Total length: 257.41

Close[A]/Undo[B]/Pick next point (Enter to end): pick next point

Segment length: 104.89, Total length: 362.30

Close[A]/Undo[B]/Pick next point (Enter to end): press Enter to finish the perimeter, or press A to close the perimeter

Digitize Another PERIMETER Polyline [Yes(A)/<No(B)>]? press A or enter Yes to continue digitizing another perimeter, press B or enter No to finish digitizing perimeters.

Keyboard Command:

dig_perim

Pull-Down Menu Location:

Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

A contour is a line of points with a constant elevation, representing the natural contour of the site. In Takeoff, there are two layer targets: **Existing Ground Surface** and **Design Surface**. Contour Polyline has two sub-command to digitize contour lines into Existing Contour and Final Contour layers directly for assigning them easily into Existing Ground Surface and Design Surface in the future analysis.

There are two ways to digitize contour lines: sketch mode or point mode.

Sketch Mode:

In sketch mode, you press the digitize button on the puck and release it to make the digitizer ready to sketch the contour lines. Then you move the digitizer cursor along the lines on your drawing. If you make a mistake, you first release the digitizer by press the **Enter** button on the puck if you are using a Carlson Puck Layout, otherwise you need to press the top-right button. Next you press B on the puck or enter Undo on the keyboard to remove your mistake, keep doing this step until you remove all you mistake. After that, you move your cursor to the position that you want to continue digitizing on the same line, press and release the digitize button, and trace the contour line by moving the cursor. You can always stop digitizing by pressing **Enter** button of a Carlson Puck (or top-right button for all other pucks) or keyboard, and then press the digitize button again to start tracing. When you finish digitizing, your first put the pen down by pressing **Enter** on the Carlson Puck (or top-right button for all other pucks) or keyboard, then press **Enter** on the puck to finish the contour line, or **press A** on the puck or **enter Close** on the keyboard to close the contour on itself.

Usually sketch mode uses more points then point mode. But you can reduce the number of points by specifying larger increment in sketch mode. Choosing smaller increment if your contour lines have more curved parts, otherwise choosing larger increment. Also Takeoff will automatically reduce the number of points on the straight parts of lines. Sketch mode digitizing is fast and accurate. You might need a little practice in order to get familiar with the command line prompts.

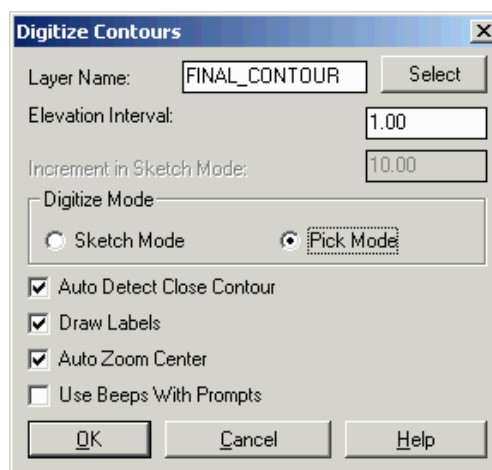
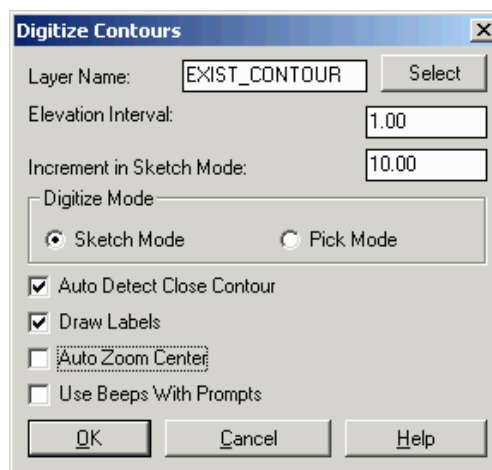
Pick Mode:

Pick mode digitizing allows your to digitize a contour line one point at a time. This method creates a new data point each time you press and release the digitize button. If you make a mistake, you can **press B** on the puck or **enter Undo** on the keyboard to remove the mistake, and then continue to digitize by picking point along the line on the drawing. In general, we recommend using pick mode to digitize because it reduces the number of points and speeds up Takeoff's calculations.

Function:

This command lets you digitize contours as polylines one at a time. The first time it prompts you the **Digitize Contours Dialog**. Enter the layer name or select it from a list of existing layer. Look at your plans and determine an elevation interval that is between most of the contours and enter it in the Elevation Interval field. You are able to modify both the value and the direction of the elevation interval between digitizing contour lines, using the buttons on the puck. If you decide to do Sketch mode digitizing, you need to enter the increment field. To have Takeoff automatically close contours whose beginning and ending points are within a specified range, check the Auto Detect Close Contour. Draw Labels would draw the elevation at the starting point of the contour. In Pick mode, if you want the Takeoff to automatically zoom the display to center around the last point when you get near the edge of the screen while picking points, check the Auto Zoom Center. Click OK to start digitizing.

Place your cursor at one end of the contour line and begin digitizing the line. While digitizing a line, you can force a contour to close on itself by **pressing A** on the puck to end the contour and connect the last point to the first point, or remove a mistake by **pressing B** on the puck. At the end of the contour line, **press Enter** on your puck or keyboard. The contour is completed, and the elevation for the next contour is automatically incremented. You would be asked to digitize next contour. If you **press A** on the puck or **enter Yes** on the keyboard, you can digitize another contour, or **press B** on the puck or **enter No** on the keyboard to finish digitizing contours.



Digitize Contour in Sketch Mode Digitize Contour in Pick Mode

Prompts:

Sketch Mode:

Digitize Contours Dialog

Enter Layer Name, Elevation Interval, Increment in Sketch Mode if select Sketch Mode, and toggle on/off Auto Detect Close Contour etc.

Increment(1.00)[A]/Direction(+)[B]/Elevation <573.00>: 450 (*enter elevation or press Enter to accept current value*)

Pick and drag to digitize (Enter to finish): <Pen down> <Pen up> (*press Enter on the puck or keyboard to finish sketching*)

1 polyline with 243 edges recorded.

Note:

If you are using puck layout 2, Takeoff would prompt you differently here.

Pick and drag to digitize (Press [0]/Enter to finish): <Pen down> <Pen up> (*press 0 on the puck or press Enter on the keyboard to finish sketching*)

1 polyline with 243 edges recorded.

Close[A]/Undo[B]/Pick to continue (Enter to finish): *press Enter on the puck or keyboard to finish the contour line*

Digitize Another Contour [<Yes(A)>/No(B)]? A (*press A to digitize next contour line*)

Increment(1.00)[A]/Direction(+)[B]/Elevation <451.00>: A

(Elevation is automatically incremented. Press A on the puck or keyboard to increase it by the Increment again.)

Increment(1.00)[A]/Direction(+)[B]/Elevation <452.00>: *press Enter on the puck or keyboard to start digitizing*

Pick and drag to digitize (Enter to finish): <Pen down> <Pen up> (*press Enter on the puck or keyboard to finish sketching*)

1 polyline with 234 edges recorded.

Note:

If you are using puck layout 2, then Takeoff would prompt you differently here.

Pick and drag to digitize (Press [0]/Enter to finish): <Pen down> <Pen up> (*press 0 on the puck or press Enter on the keyboard to finish sketching*)

1 polyline with 234 edges recorded.

Close[A]/Undo[B]/Pick to continue (Enter to finish):

Digitize Another Contour [<Yes(A)>/No(B)]? A (*press A to digitize next contour line*)

Increment(1.00)[A]/Direction(+)[B]/Elevation <453.00>: B

(Elevation is automatically incremented. Press B on the puck or keyboard to change the increment from positive to negative.)

Increment(1.00)[A]/Direction(-)[B]/Elevation <452.00>: A (*Press A on the puck or keyboard to increase it by the Increment again.*)

Increment(1.00)[A]/Direction(-)[B]/Elevation <451.00>: A (*Press A on the puck or keyboard to increase it by the Increment again.*)

Increment(1.00)[A]/Direction(-)[B]/Elevation <450.00>: A (*Press A on the puck or keyboard to increase it by the Increment again.*)

Increment(1.00)[A]/Direction(-)[B]/Elevation <449.00>: A (*Press A on the puck or keyboard to increase it by the Increment again.*)

Increment(1.00)[A]/Direction(-)[B]/Elevation <448.00>: *press Enter on the puck or keyboard to start digitizing*

Pick and drag to digitize (Enter to finish): <Pen down> <Pen up> (*press Enter on the puck or keyboard to finish sketching*)

1 polyline with 109 edges recorded.

Note:

If you are using puck layout 2, then Takeoff would prompt you differently here.

Pick and drag to digitize (Press [0]/Enter to finish): <Pen down> <Pen up> (press 0 on the puck or press Enter on the keyboard to finish sketching)

1 polyline with 109 edges recorded.

Close[A]/Undo[B]/Pick to continue (Enter to finish): *press Enter on the puck or keyboard to finish the contour line*

Digitize Another Contour [<Yes(A)>/No(B)]? B *(press B to finish digitizing contour line in sketch mode)*

Pick Mode:

Digitize Contours Dialog

Enter Layer Name, Elevation Interval, Increment in Sketch Mode if select Sketch Mode, and toggle on/off Auto Detect Close Contour etc.

Increment(1.00)[A]/Direction(+)[B]/Elevation <458.00>: *press Enter on the puck or keyboard to start digitizing*

First point:

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

(You can force a contour to close on itself by pressing A or remove mistake by pressing B.)

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Close[A]/Undo[B]/Pick next point (Enter to end): *press Enter to finish picking*

Digitize Another Contour [<Yes(A)>/No(B)]? A *(press A to digitize next contour line)*

Increment(1.00)[A]/Direction(+)[B]/Elevation <459.00>: *press Enter on the puck or keyboard to start digitizing*

First point:

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Close[A]/Undo[B]/Pick next point (Enter to end): *press Enter to finish picking*

Digitize Another Contour [<Yes(A)>/No(B)]? B *(press B on the puck or enter No on the keyboard to finish digitizing contour line in pick mode)*

Keyboard Command:

digcont_exist, digcont_final

Pull-Down Menu Location:

Digitize

Prerequisite:

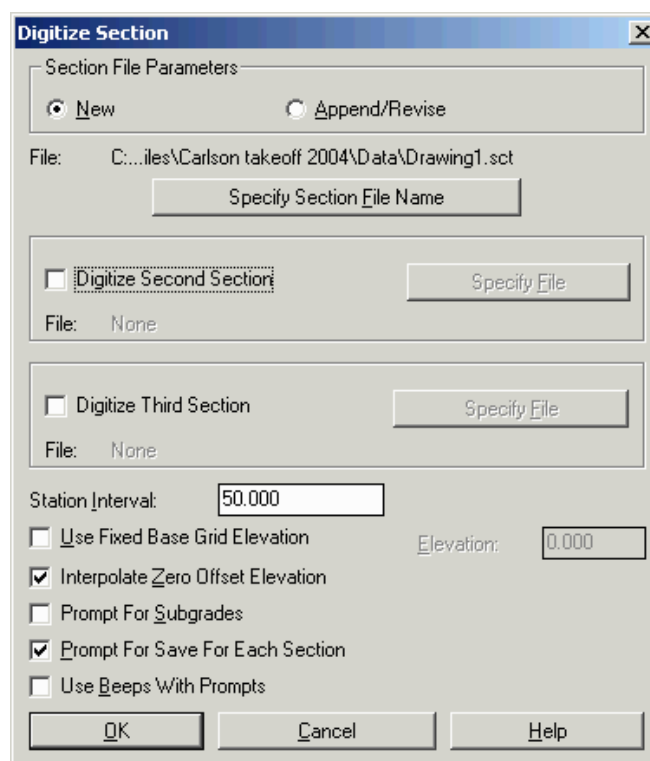
Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Functions:

This command allows you to digitize section lines and store the section data in the section file you have specified. The command first prompts you the **Digitize Section Dialog**. Enter the section file name and determine if you want to digitize second and third sections at the same station. Look at your plans and determines the station interval,

which is used to automatically default to the next station value when digitizing a series of stations. If the grids at all the stations have the same base elevation, toggle on Use Fixed Base Grid Elevation. You can also toggle on Interpolate Zero Offset Elevation, Prompt for Subgrades, Prompt for Save for Each Section and Use Beeps with Prompts. Click OK to start digitizing.

Takeoff prompts you to calibrate the section sheet before you digitize the section lines. You pick three points and specify their offsets to the centerline and elevations in order to determine the horizontal and vertical intervals. Corners on the section grid are preferred reference points. Place your cursor at one end of the section line and begin digitizing the line. While digitizing a line, you can remove a mistake by **pressing A** on the puck or **entering Undo** on the keyboard. At the end of the section line, **press Enter** on your puck or keyboard. The station is completed, and the station value is automatically incremented. The command would prompts to digitize next section. You can **press A** on the puck or **enter Exit** on the keyboard to finish digitizing. If you want to continue to digitize next section, **press Enter** or enter the new station number. For every station after the first one, you can calibrate the grid sheet by picking one reference point and specify its offset and elevation. After you digitize the section lines on your drawing, all the section data would be saved in a section file (.sct).



Prompts:

Digitize Section Dialog

Enter Section File Name, Station Interval, and toggle on/off Use Fixed Base Grid Elevation etc.

Section station to digitize <0.000>: *press Enter to start with station 0.0 or enter a station number*

Calibrate section sheet

Pick First section sheet reference point: *pick a grid point of this station on your drawing*

Enter offset <0.0>: *press Enter to accept the offset or enter the offset of the point to the centerline*

Enter elevation: 1030 *(enter the Elevation of the reference point)*

Pick Second section reference point: *pick the second grid point*

Enter offset: 0 *(enter the offset of the point to the centerline)*

Enter elevation: 1040 *(enter the Elevation of the reference point)*

Pick Third section reference point: *pick the third grid point*
Enter offset: 50 (*enter the offset of the point to the centerline*)
Enter elevation: 1040 (*enter the Elevation of the reference point*)

3 calibration points

Transformation type: Orthogonal Affine Projective

Outcome of fit: Success Exact Impossible

RMS Error: 11.49

Standard deviation: 2.38

Largest residual: 14.08

At point: 2

Second-largest residual: 14.08

At point: 1

Digitize break point for DRAWING1 section 0.000 (Enter to end): *pick a point on the section line*

Offset: -39.81 Elev: 1028.80

Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *pick a point on the section line*

Offset: -9.94 Elev: 1030.03

Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *pick a point on the section line*

Offset: 49.44 Elev: 1034.93

Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *press Enter to finish*

Save changes to DRAWING1 section 0.000 [<Yes(A)>/No(B)]? A (*press A or B*)

Exit[A]/Section station to digitize <50.000>: 200 (*enter next station number*)

Calibrate next section

Pick section reference point: *pick a grid point of the station on your drawing*

Enter offset <0.00>: *press Enter to accept the offset or enter the offset of the point to the centerline*

Enter elevation <1030.00>: 1020 (*enter the Elevation of the reference point*)

Digitize break point for DRAWING1 section 200.000 (Enter to end): *pick a point on the section line*

Offset: -40.40 Elev: 1008.07

Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*

Offset: -5.38 Elev: 1019.98

Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*

Offset: 27.86 Elev: 1030.02

Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*

Offset: 50.33 Elev: 1035.80

Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *press Enter to finish*

Save changes to DRAWING1 section 200.000 [<Yes(A)>/No(B)]? A (*press A or B*)

Exit[A]/Section station to digitize <250.000>: A (*press A to finish or enter the station number to continue*)

Keyboard Command:

digxsec

Pull-Down Menu Location:

Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Functions:

There are two types of end areas: cut area and fill area. This command allows you to digitize both cut area and fill area on the drawing and writes data to a .ew file. The command first prompts you to calibrate the section sheet by picking three points and specify their offsets to the centerline and elevations in order to determine the horizontal and vertical intervals. Corners on the section grid are preferred reference points. Then it prompts you to digitize the cut area and fill area respectively. Place your cursor at one end of the end area and begin digitizing the outline of the area. At the end of the section line, **press Enter** on your puck or keyboard. The end area is completed, and its area is printed on the command line, and you are prompted to digitize next end area. After you finish all the end area at one station, accumulated cut area and fill area are computed and printed out on the screen. All data of cut area and fill area at every station would be saved in the area file (.ew) that you have specified.

Prompts:

Calibrate section sheet

Pick First section sheet reference point: *pick a point on the drawing*

Enter offset <0.0>: *press Enter to accept the offset (or enter the offset of the point to the centerline)*

Enter elevation: 1020 *(enter the Elevation of the reference point)*

Pick Second section reference point: *pick a point*

Enter offset: 0 *(enter the offset of the point to the centerline)*

Enter elevation: 1030 *(enter the Elevation of the reference point)*

Pick Third section reference point: *pick a point*

Enter offset: 50 *(enter the offset of the point to the centerline)*

Enter elevation: 1030 *(enter the Elevation of the reference point)*

3 calibration points

Transformation type: Orthogonal Affine Projective

Outcome of fit: Success Exact Impossible

RMS Error: 11.69

Standard deviation: 2.40

Largest residual: 14.29

At point: 2

Second-largest residual: 14.29

At point: 3

Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area, 0*(0.211129 1030.76)*

Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area, 1*(11.5804 1030.49)*

Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area, 2*(17.8643 1030.73)*

Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area, 3*(19.0216 1032.35)*

Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area, 4*(-0.777246 1030.75)*

Digitize cut area (Enter to end): *press Enter to finish*

End area: 17.2312

Accumulated Cut Area: 17.2312

More Cut Areas [Yes(A)]/<No>(B)]? *press A to digitize more Cut Areas, or press B to finish digitizing Cut Areas.*

Accumulated Cut Area: 17.2312

Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area, 0*(-18.9614 1029.65)*

Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area, 1*(-18.1315 1030.75)*

Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area, 2*(-11.9592 1030.49)*

Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area, 3*(-2.06761 1030.72)*

Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area, 4*(-10.0082 1030.01)*

Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area, 5*(-18.531 1029.67)*

Digitize fill area (Enter to end): *press enter to finish*

End area: 8.64646

Accumulated Cut Area: 8.64646

More Fill Areas [Yes(A)/<No>(B)]? *press A to digitize more Fill Areas, or press B to finish digitizing Fill Areas.*

Accumulated Cut Area: 8.64646

Total Cut Area: 17.2312

Total Fill Area: 8.64646

Store data to file [<Yes>(A)/No(B)]? *press A or B*

Opened file: C:\Program Files\Carlson Takeoff 2004\DATA\Drawing1.ew

Station Number: 1 *(enter Station Number)*

Data Stored in file: C:\Program Files\Carlson Takeoff 2004\DATA\Drawing1.ew

Digitize another station [<Yes>(A)/No(B)]? **B** *(press A or B)*

Keyboard Command:

digendar

Pull-Down Menu Location:

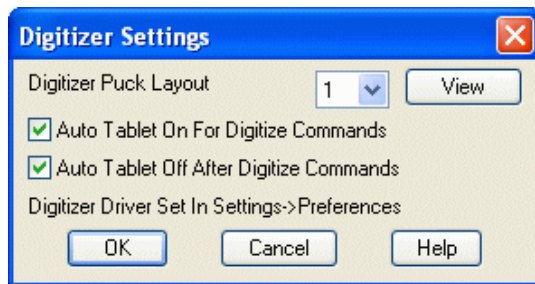
Digitize

Prerequisite:

Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitizer Settings

This command allows you to select the puck layout and set Auto On/Off features.



Auto Tablet On For Digitize Commands means after you select a digitize command your puck will automatically be put in Digitize Mode. If this is toggle off, then you will need to turn Tablet on separately from running a digitize command.

Auto Tablet Off After Digitize Commands means you will return to Mouse Mode after running a digitize command. Read below for more on Mouse and Digitize Mode.

Puck Layout

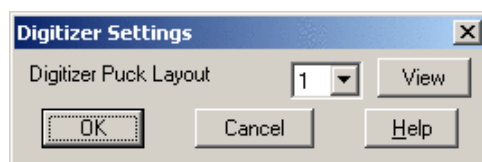
The 16-button puck can be used as either a mouse or a digitizer. It's very important to understand how the 16 buttons are mapped in both modes.

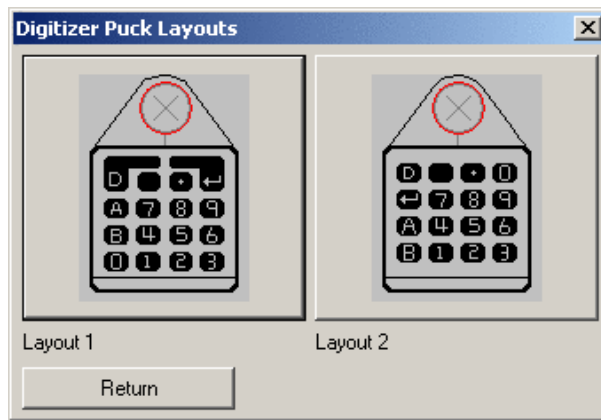
Mouse Mode

When the tablet is off, the puck is in Mouse Mode. The top-left button is the left mouse click, and the top-right button is the right mouse click. The labels on the other buttons do not mean anything. All buttons are mapped as same as the buttons of the default pointing device in AutoCad . Please refer to AutoCad Reference manual for further information.

Digitize Mode

When tablet has been calibrated and is on, the puck is in digitize mode. It is mapped as a small keyboard, which enables you to enter numerous values such as elevation, thickness and offset etc., and also provide you some functionality to digitize various entities. Currently there are two puck layouts in Takeoff, shown in the figure below. After you install Carlson Takeoff and finish setting up the digitizer, you go to the pull-down menu **Digitize->Puck Layout** to select a 16-button puck layout. A button mapping would be created and Takeoff would recognize the buttons as represented.





Layout 1 is Carlson Puck Layout, which is the most common layout used in Carlson Takeoff. Layout 2 is for users who don't have a Carlson Puck. If your puck is different than these two layouts, please contact Technical Support for help setting the mapping for your 16 button puck.

Prompts

Digitizer Settings Dialog

Specify the Digitizer Puck Layout to layout 1 or 2

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed.

Keyboard Command: dig_config

Digitize Existing

This sets the layer target to existing. Set this prior to running any digitizing command and anything you digitize will be assigned for your existing surface. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

Keyboard Command: set_digit_exist

Prerequisite: none

Digitize Design

This sets the layer target to design. Set this prior to running any digitizing command and anything you digitize will be assigned for your design surface. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

Prerequisite: none

Keyboard Command: set_digit_final

Digitize Other

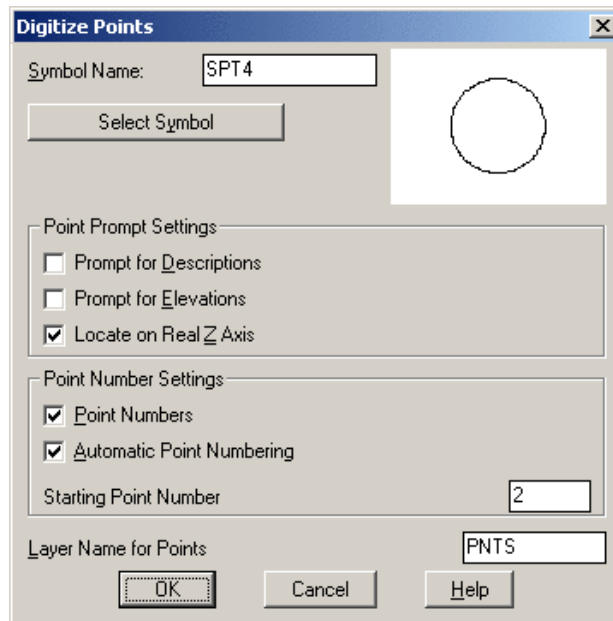
This sets the layer target to other. Set this prior to running any digitizing command and anything you digitize will be assigned to the Other target. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

Keyboard Command: set_digit_other

Prerequisite: none

Digitize Point

This command allows you to digitize individual points one at a time. The first time it prompts you the **Digitize Points Dialog** for entering point symbol styles, point prompt settings and number settings, starting point number and layer name. If you want to enter the elevation and description for each point, select **Prompt for Descriptions** and **Prompt for Elevations**. After having digitized a point, you can continue to digitize next point by picking the point on the drawing. The command defaults to the last layer name, point symbol, elevation, description and the last point number plus 1. If you have finished digitizing points, press **Enter** to finish.



Prompts

Digitize Points Dialog

Specify a layer name and select the point symbol, point prompt settings and number settings.

Pick point to create (Enter to end): *pick a point on the drawing*

Select/<Enter Point Elevation <>>: *enter the elevation or type <Select> to select the elevation text on the screen*

Enter Point Description <>: *enter the point description*

Result like "N: 1231.16 E: 1099.17 Z: 30.00" would be display on the command line, and a point would be drawn on the screen with the text of its number, elevation and description.

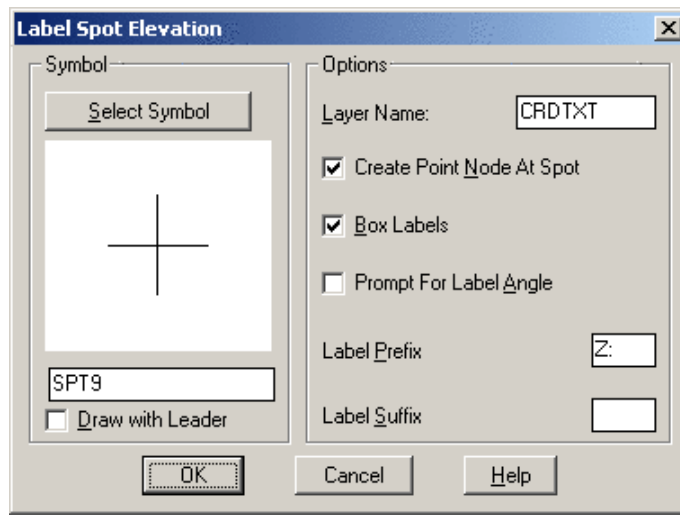
Pick point to create (Enter to end): *pick next point or press Enter to finish digitizing points*

Keyboard Command: dig_pt

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize Spot Elevation

This command allows you to label points with their elevation. The point can either be digitized from a drawing, picked on a screen or specified by a point number. The command first prompts you the **Label Spot Elevation Dialog** for entering layer name, label prefix and suffix and symbol types etc. Click OK to start. After specifying the point, the command prompts you to enter the elevation if its elevation is unknown and then pick an angle from the location of the point to label the elevation. You can repeat labeling points until you press **Enter** to finish.



Prompts

Label Spot Elevation Dialog

Specify a layer name, label prefix and suffix and select the spot symbol.

Point to Label ?

Pick point or point number: 2 (*enter a point number*)

PointNo. Northing(Y) Easting(X) Elev(Z) Description

2 1231.16 1099.17 30.00 bb

Note: if the point number you entered is not in the drawing, you will be prompted again to pick point or enter a point number.

Elevation <30.000>: *press enter*

Pick angle for label: *pick an angle from the spot*

Point to Label (ENTER to End)?

Pick point or point number: *pick a point on the drawing*

Elevation <0.000>: *enter elevation*

Pick angle for label: *pick an angle from the spot*

Point to Label (ENTER to End)?

Pick point or point number: *press enter to finish*

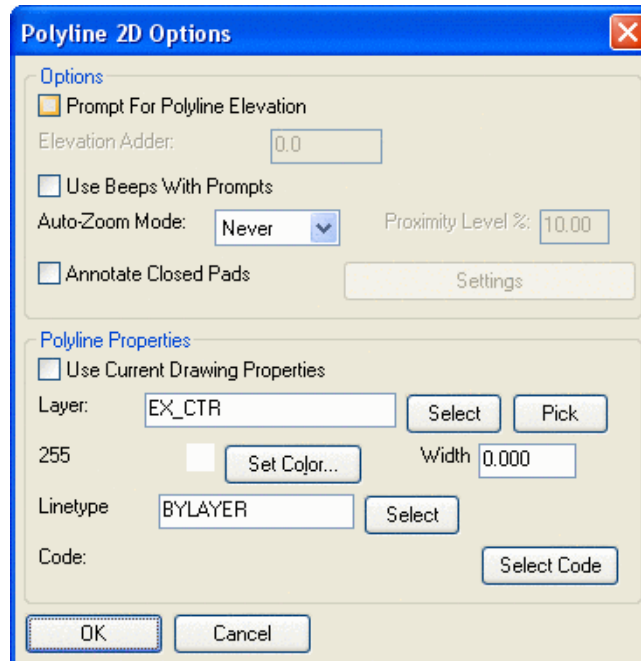
Keyboard Command: labspot

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize 2D Polyline

A 2D polyline is a line of connected points that have the same elevation. This command lets you digitize a 2D polyline by picking points along the lines on the drawing. It prompts you first the **Polyline 2D Options Dialog** for entering the layer name. **Prompt For Polyline Elevation** option allows you to enter the elevation for each polyline, otherwise all 2D polylines have 0.0 elevation. **Auto-Zoom** mode would automatically zoom the display to center around the last point when you get near the edge of the screen while picking points. There are three ways to enter a layer name, **Use current drawing layer**, **Select** from a list of layer name, or **Pick** an entity on the screen to get its layer name. While digitizing a polyline, the command keeps prompting you to either pick the next point or **press 0** to create an Arc until your **press Enter** to finish digitizing. **Press A**

on the puck or **enter Close** on the keyboard to close the polyline on itself. You can define an **Arc** by Radius, Arc length, Chord length, Delta angle, or by simply picking 3 points along the arc. If at any point you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a polyline, the command prompts you to digitize another polyline until you **press B** or **enter No**.



Prompts

Polyline 2D Options Dialog

Enter the layer name and select the options of Prompt For Polyline Elevation and Auto-Zoom mode etc.

Enter default elevation <0.00>: 100

First point: *pick a point on the drawing using puck*

Segment length: 0.00, Total length: 0.00

Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *pick next point*

Segment length: 119.03, Total length: 119.03

Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *pick next point*

Segment length: 115.23, Total length: 234.26

Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *press 0*

[Radius[0]/Second pt[A]/Undo[B]/<Pick Endpoint>]: *press A*

Second point or point number: *pick a point along the arc*

Endpoint or point number: *pick the last point along the arc*

Segment length: 500.82, Total length: 735.08

Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *pick next point*

Segment length: 115.23, Total length: 850.31

Close[A]/Undo[B]/Pick next point (Enter to end): *press enter to finish digitizing or press A to close the polyline*

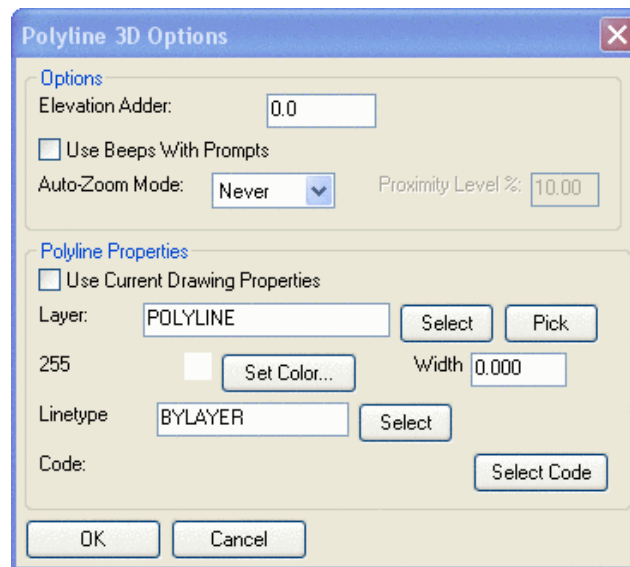
Digitize Another FINAL Polyline [Yes(A)/<No(B)>]? *press A on the puck or enter Yes on the keyboard to digitize next 2D polyline, press B on the puck or enter No on the keyboard to finish digitizing 2D polyline.*

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Keyboard Command: dig_2dp

Digitize 3D Polyline

A 3D polyline is a line of connected points that have various elevations, and the slope between points is constant. It can be used in defining pads, excavations, drainage ditched and slopes from proposed design features to meet existing site conditions. This command lets you digitize a 3D polyline by picking points along the lines on the drawing. It prompts you first the **Polyline 3D Options Dialog** for entering the layer name. **Elevation Adder** allows you to truncate the elevations you have to enter in by add a given amount to them. There are five ways to enter elevations: known elevation of the point, interpolate, slope from previous point, ratio from previous point and degree from previous point. You can choose one of the methods between picking points. **Auto-Zoom** mode would automatically zoom the display to center around the last point when you get near the edge of the screen while picking points. While digitizing a polyline, press **A** to interpolate the elevation or **B** to enter it in. The command keeps prompting you to either pick the next point or **press 0** to create Arc cords until you **press Enter** to finish digitizing. **Press A** on the puck or **enter Close** on the keyboard to close the polyline on itself. You can define **Arc cords** by Radius, Arc length, Chord length, Delta angle, or by simply picking 3 points along the arc. You can also use the OSNAP command to pick points by pressing the decimal [.] button on the digitizer puck. If you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a polyline, the command prompts your to digitize another polyline until you **press B** or **enter No**.



Prompts

First point:

Interpolate[A]/screen Pick/<Elevation[B]> <0.00>: 256

Z: 256.00

Arc[0]Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): Pick point

Slope/Ratio/Interpolate[A]/Degree/screen Pick/<Elevation[B]> <256.00>: A

Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: Pick point

This point elevation will be interpolated upon completion.

Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: 279

Z: 279.00, Hz dist: 30.01, Slope dist: 37.81, Slope: 76.6% Ratio: 1.3:1

Arc[0]Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): Pick point

Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: Press Enter

Z: 279.00, Hz dist: 24.18, Slope dist: 24.18, Slope: 0.0% Ratio: 0.0:1

Arc[0]Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): A

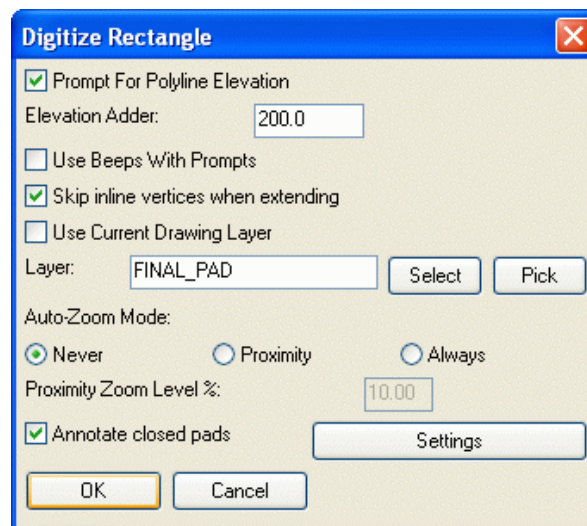
Digitize Another EXIST_PLINE Polyline [Yes(A)/<No(B)>]? B
<Tablet Off>

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

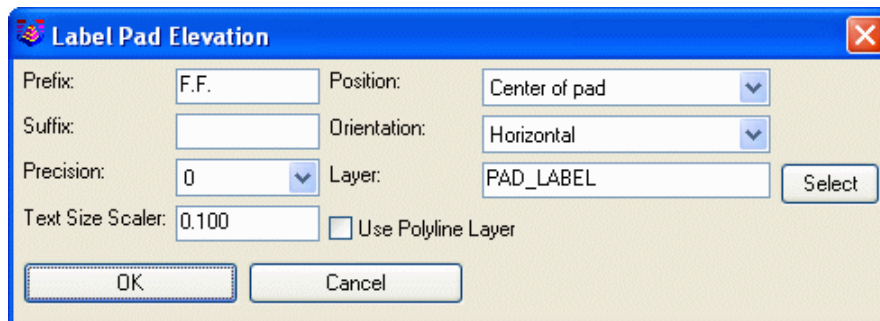
Keyboard Command: dig_3dp

Digitize Rectangle

This command enables you to quickly create rectangles while digitizing. In the dialog, you can pick to set elevations to the rectangles, otherwise all rectangles will have 0.0 elevation. The Elevation Adder will be added to the value you enter in for the prompt "Enter polyline elevation <0.00>:". For example, if you know all the rectangles you are creating are in the 200s for elevation, you can put in this value for the Elevation Adder and simply put 46, 54, 57, etc. when prompted, and your rectangles will end up with the elevations of 246, 254, 257 etc. There are three ways to enter a layer name, Use current drawing layer, Select from a list of layer name, or Pick an entity on the screen to get its layer name. Auto-Zoom mode would automatically zoom the display to center around the last point when you get near the edge of the screen while picking points.



Annotate closed pads will label your rectangles according to the Settings button/dialog shown below:



In this dialog, you can enter in a Prefix or a Suffix to the elevation, and determine the labels position, orientation, precision out to 5 decimal places, its layer, and text size.

Prompts

Target surface: Design

Digitize Rectangle Dialog Make any changes you desire in the above dialogs.

Enter polyline elevation <0.00>: 200

First point: *pick a point on the drawing using puck*

Segment length: 0.00, Total length: 0.00

Close[A]/Undo[B]/Osnap[.]/Pick next point: *pick next point*

Segment length: 1105.96, Total length: 1105.96

Close[A]/Undo[B]/Osnap[.]/Pick next point: *pick next point*

Segment length: 426.83, Total length: 1532.79, Area: 236021.59

Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *After 3 points you can press (A) for Close to create a rectangle*

Digitize Another FINAL_PAD Polyline [Yes(A)/<No(B)>]? *B for No*

Prerequisite: a digitizer

Keyboard Command: DIG_RECT

Digitize Perimeter

Perimeter is a 2D polyline that all points on it have the same elevation. It can be used as boundary polyline of your targets on your drawing. This command allows you to digitize a perimeter by picking points on the drawing. While digitizing a polyline, the command keeps prompting you to pick next point until you **press Enter** to finish digitizing, or **press A** on the puck or **enter Close** on the keyboard to close the polyline on itself. If you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a perimeter, the command prompts you to digitize another polyline until you **press B** or **enter No**.

Prompts

First point: *pick a point on the drawing using puck*

Segment length: 0.00, Total length: 0.00

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Segment length: 104.27, Total length: 104.27

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Segment length: 153.14, Total length: 257.41

Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Segment length: 104.89, Total length: 362.30

Close[A]/Undo[B]/Pick next point (Enter to end): *press Enter to finish the perimeter, or press A to close the perimeter*

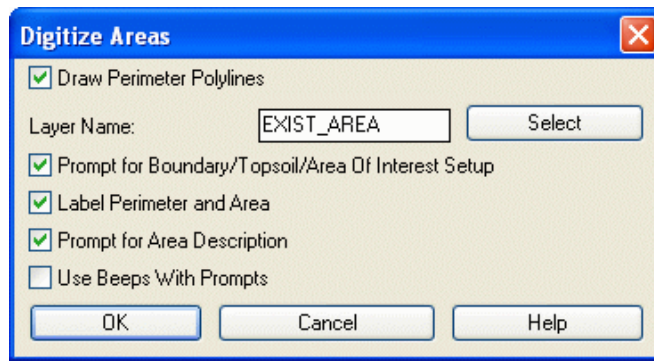
Digitize Another PERIMETER Polyline [Yes(A)/<No(B)>]? *press A or enter Yes to continue digitizing another perimeter, press B or enter No to finish digitizing perimeters.*

Keyboard Command: dig_perim

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize Areas

This command allows you to find an area in digitize mode. With the puck, pick around the area you wish to calculate. If Draw Perimeter Polyline is toggled on then the linework of your perimeter will be displayed. You can then set the Layer Name and choose to label the Perimeter and Area and enter in an Area Description. You can also set the area you created as a Boundary, Topsoil, or Area of Interest.



Prerequisite: a digitizer

Keyboard Command: dig_area

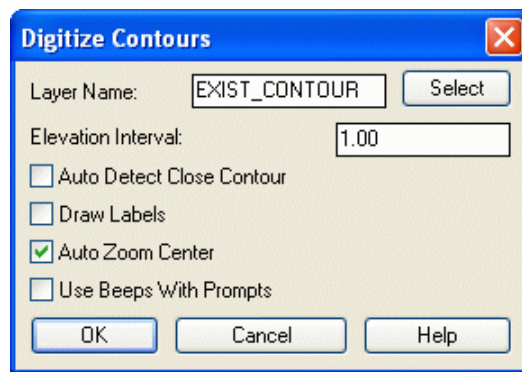
Digitize Contour Polyline

A contour is a line of points with a constant elevation, representing the natural contour of the site. In Takeoff, there are two layer targets: Existing Ground Surface and Design Surface. Contour Polyline has two sub-command to digitize contour lines into Existing Contour and Final Contour layers directly for assigning them easily into Existing Ground Surface and Design Surface in the future analysis.

There are two ways to digitize contour lines: sketch mode or point mode. You can start digitizing a contour with one mode and switch to the other during digitizing the contour. Sketch mode uses more points than pick mode. In general, we recommend using pick mode to digitize the straight parts of lines because it reduces the number of points and speeds up Takeoff's calculations, but using sketch mode to digitize the curved parts because it is fast and accurate.

This command lets you digitize contours as polylines one at a time. The first time it prompts you the Digitize Contours Dialog. Enter the layer name or select it from a list of existing layer. Look at your plans and determine an elevation interval that is between most of the contours and enter it in the Elevation Interval field. You are able to modify both the value and the direction of the elevation interval between digitizing contour lines, using the buttons on the puck. To have Takeoff automatically close contours whose beginning and ending points are within a specified range, check the Auto Detect Close Contour. Draw Labels would draw the elevation at the starting point of the contour. In Pick mode, if you want the Takeoff to automatically zoom the display to center around the last point when you get near the edge of the screen while picking points, check the Auto Zoom Center. Click OK to start digitizing.

If this is your first time digitizing a contour, you are defaulted to the Pick Mode digitizing, otherwise you would be defaulted to the last digitize mode. If you want to use the other digitize mode, press 0 on the puck or enter 0 from the keyboard. Place your cursor at one end of the contour line and begin digitizing the line. While digitizing a line, you can force a contour to close on itself by pressing A on the puck to end the contour and connect the last point to the first point, remove a mistake by pressing B on the puck, or switch to the other digitize mode by pressing 0. During Sketch Mode digitizing, you can stop digitizing by pressing Pick or Enter button on the puck, take some rest or changes, and start sketching again. At the end of the contour line, **press Enter** on your puck or keyboard. The contour is completed, and the elevation for the next contour is automatically incremented. You would be asked to digitize next contour. If you press A on the puck or enter Yes on the keyboard, you can digitize another contour, or press B on the puck or enter No on the keyboard to finish digitizing contours.



Prompts

Digitize Contours Dialog

Enter Layer Name, Elevation Interval, and toggle on/off Auto Detect Close Contour etc.

Increment(1.00)[A]/Direction(+)[B]/Elevation <573.00>: 450 (*enter elevation or press Enter to accept current value*)

Start Digitizing...

Sketch[0]/Pick the first point: *pick a point to start Pick Mode digitizing (press 0 to switch to Sketch Mode)*

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): 0 (*press 0 on the puck or enter 0 on the keyboard to use Sketch Mode*)

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): pick and drag

Drag to digitize (Pick or press Enter to stop sketching)... pick or press Enter to stop sketching

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): B (undo the last point)

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): B (undo the last point)

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): pick and drag again

Drag to digitize (Pick or press Enter to stop sketching)... pick or press Enter to stop sketching

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): 0 (*press 0 on the puck or enter 0 on the keyboard to use Pick Mode*)

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): press Enter to finish digitizing

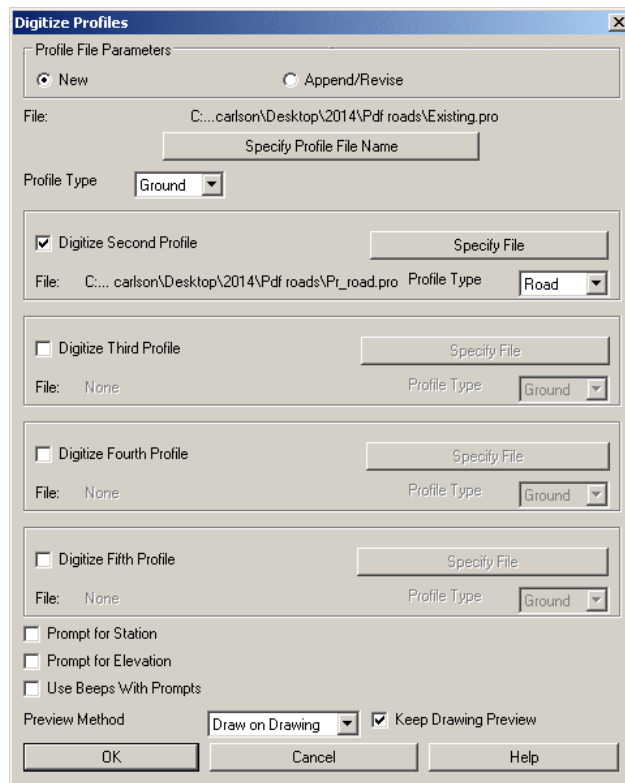
Digitize Another Contour [<Yes(A)>/No(B)]? B (*press B to finish digitizing*)

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Keyboard Command: digcont_exist, digcont_final

Digitize Profiles

This command allows you to digitize profile lines and store the profile data into the profile file (.pro) you have specified. The command first prompts you the **Digitize Profile Dialog**. Enter the profile file name and determine if you want to digitize up to five profiles from the same grid. Next, determine the **Profile Type**: Ground, Road, or Pipe. Selecting Road will prompt you for a "Vertical Curve" to be applied between the current point and previous profile point. If no Vertical Curve exists, type "0"; entering without specifying any Vertical Curve will turn off the VC prompt on subsequent profile points. Selecting Pipe will prompt you for a "Step up for the pipe" and "Enter the Pipe size" at each profile point. With Ground, there are no additional prompts.



Towards the bottom half of the dialog, there are toggles for **Prompt for Station** and **Prompt for Elevation**. These will show you a preview value for the Station and Elevation based on the point that you picked. Press Enter to except or type in a new value in manually. This is most useful for accurate import of Road and Pipe data. Also at the bottom, **Preview Method** offers 2 ways to view the profiles as you digitize. "Graphic Dialog" displays the profile data in a grid dialog and is best when digitizing from paper plans. "Draw on Drawing" draws 2D polylines in your CAD drawing and is best when digitizing over an Image in your drawing. "Keep Drawing Preview" will leave the 2D polylines in your CAD drawing (having this checked off will erase the 2D polylines after each station).

After clicking OK in the main dialog, Takeoff prompts you to **calibrate the profile sheet** before you digitize the profile lines. You will pick three points and specify their station and elevations in order to determine the horizontal and vertical scales. Corners on the profile grid are preferred reference points. Once calibrated, place your cursor at one end of the profile line and begin digitizing the line. While digitizing a line, you can remove a mistake by **pressing A** on the puck or **entering Undo** on the keyboard. At the end of the profile line, **press Enter** on your puck or keyboard. The command then prompts you to digitize the next profile. You can **press A** on the puck or **enter Exit** on the keyboard to finish digitizing. After you digitize the profile lines on your drawing, all the profile data would be saved in a profile file (.pro).

Prompts

Calibrate profile sheet

- Pick First profile sheet reference point:** *pick a grid point of a station on your drawing*
- Enter station <0.0>: 1000** *press Enter to accept the station or enter a new station value*
- Enter elevation: 95** *(enter the Elevation of the reference point)*
- Pick Second profile reference point:** *pick a second grid point*
- Enter station: 1200** *(enter the grid's station)*
- Enter elevation: 110** *(enter the Elevation of the reference point)*
- Pick Third profile reference point:** *pick a third grid point*
- Enter station: 1500** *(enter the grid's station)*

Enter elevation: 105 (*enter the Elevation of the reference point*)

3 calibration points

Transformation type: Orthogonal Affine Projective

Outcome of fit: Success Exact Impossible

RMS Error: 11.49

Standard deviation: 2.38

Largest residual: 14.08

At point: 2

Second-largest residual: 14.08

At point: 1

Digitize break point/ (Enter to end): *pick a point on the profile line*

Station: 1000 Elev: 106.77

Digitize break point (Undo[A],Enter to end): *pick a point on the profile line*

Station: 1145 Elev: 101.18

Digitize break point (Undo[A],Enter to end): *pick a point on the profile line*

Station: 1440 Elev: 100.49

Digitize break point (Undo[A],Enter to end): *press Enter to finish*

Keyboard Command: digprof

Prerequisite: For "on-screen" digitizing, have an image loaded and Digitizer Settings set to "Use Mouse". For paper plans, have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize Sections

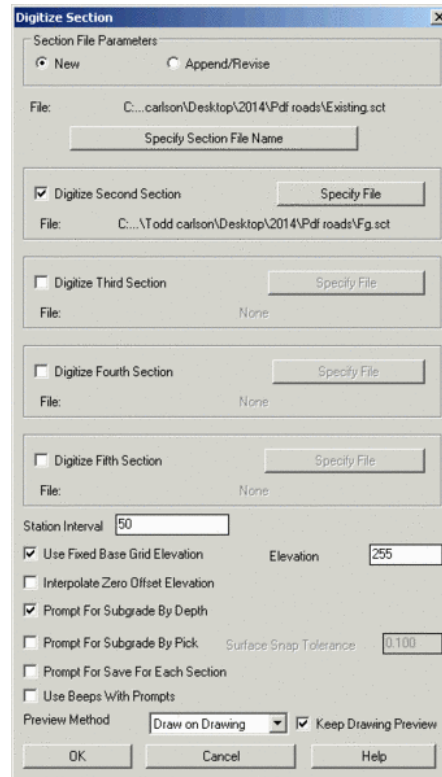
This command allows you to digitize section lines and store the section data in the section file you have specified. The command first prompts you the **Digitize Section Dialog**. Enter the section file name and determine if you want to digitize second and third sections at the same station (i.e., for existing, finish grade, unsuitable, etc.). Look at your plans and determines the station interval, which is used to automatically default to the next station value when digitizing a series of stations. If the grids at all the stations have the same base elevation, toggle on **Use Fixed Base Grid Elevation**. The **Interpolate Zero Offset Elevation** option calculates a section point at the zero offset when the zero offset point isn't digitized. The **Prompt for Save for Each Section** option can be used to save the SCT file after completing each station. The **Use Beeps with Prompts** option gives audio feedback to help keep your eyes on the sheet. **Surface Snap Tolerance** sets the maximum distance that the program will automatically snap the tie back point between the subgrade and design surface. The **Prompt For Descriptions** option will prompt for a description for each digitized section point.

There are two ways to account for Subgrades. **Prompt For Subgrade By Depth** will ask you for a subgrade depth while you are digitizing a section file and apply that subgrade depth below each point you pick for that section. If no subgrade linework is shown on the plans, but the depth is known, this is a good option. **Prompt For Subgrade By Pick** allows you to digitize in the subgrade linework after initial section has been digitized. This option is best when your Subgrade Depth is not uniform.

Preview Method offers 2 ways to view the sections as you digitize. "**Graphic Dialog**" displays the section data in a grid dialog and is best when digitizing from paper plans. "**Draw on Drawing**" draws 2D polylines in your CAD drawing and is best when digitizing over an Image in your drawing. "Keep Drawing Preview" will leave the 2D polylines in your CAD drawing (having this checked off will erase the 2D polylines after each station). Click OK to start digitizing.

Takeoff prompts you to calibrate the section sheet before you digitize the section lines. You pick three points and specify their offsets to the centerline and elevations in order to determine the horizontal and vertical intervals. Corners on the section grid are preferred reference points. Place your cursor at one end of the section

line and begin digitizing the line. While digitizing a line, you can remove a mistake by **pressing A** on the puck or **entering Undo** on the keyboard. At the end of the section line, **press Enter** on your puck or keyboard. The station is completed, and the station value is automatically incremented. The command would prompts to digitize next section. You can **press A** on the puck or **enter Exit** on the keyboard to finish digitizing. If you want to continue to digitize next section, **press Enter** or enter the new station number. For every station after the first one, you can calibrate the grid sheet by picking one reference point and specify its offset and elevation. After you digitize the section lines on your drawing, all the section data would be saved in a section file (.sct).



Prompts

Digitize Section Dialog

Enter Section File Name, Station Interval, and toggle on/off Use Fixed Base Grid Elevation etc.

Section station to digitize <0.000>: *press Enter to start with station 0.0 or enter a station number*

Calibrate section sheet

Pick First section sheet reference point: *pick a grid point of this station on your drawing*

Enter offset <0.0>: *press Enter to accept the offset or enter the offset of the point to the centerline*

Enter elevation: 1030 *(enter the Elevation of the reference point)*

Pick Second section reference point: *pick the second grid point*

Enter offset: 0 *(enter the offset of the point to the centerline)*

Enter elevation: 1040 *(enter the Elevation of the reference point)*

Pick Third section reference point: *pick the third grid point*

Enter offset: 50 *(enter the offset of the point to the centerline)*

Enter elevation: 1040 *(enter the Elevation of the reference point)*

3 calibration points

Transformation type: Orthogonal Affine Projective

Outcome of fit: Success Exact Impossible

RMS Error: 11.49

Standard deviation: 2.38

Largest residual: 14.08

At point: 2

Second-largest residual: 14.08

At point: 1

Digitize break point for DRAWING1 section 0.000 (Enter to end): *pick a point on the section line*

Offset: -39.81 Elev: 1028.80

Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *pick a point on the section line*

Offset: -9.94 Elev: 1030.03

Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *pick a point on the section line*

Offset: 49.44 Elev: 1034.93

Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *press Enter to finish*

Save changes to DRAWING1 section 0.000 [<Yes(A)>/No(B)>]? A** *(press A or B)***

Exit[A]/Section station to digitize <50.000>: 200 *(enter next station number)*

Calibrate next section

Pick section reference point: *pick a grid point of the station on your drawing*

Enter offset <0.00>: *press Enter to accept the offset or enter the offset of the point to the centerline*

Enter elevation <1030.00>: 1020 *(enter the Elevation of the reference point)*

Digitize break point for DRAWING1 section 200.000 (Enter to end): *pick a point on the section line*

Offset: -40.40 Elev: 1008.07

Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*

Offset: -5.38 Elev: 1019.98

Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*

Offset: 27.86 Elev: 1030.02

Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*

Offset: 50.33 Elev: 1035.80

Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *press Enter to finish*

Save changes to DRAWING1 section 200.000 [<Yes(A)>/No(B)>]? A** *(press A or B)***

Exit[A]/Section station to digitize <250.000>: A *(press A to finish or enter the station number to continue)*

Keyboard Command: digxsec

Prerequisite: For "on-screen" digitizing, have an image loaded and Digitizer Settings set to "Use Mouse". For paper plan digitizing, have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize End Areas

There are two types of end areas: cut area and fill area. This command allows you to digitize both cut area and fill area on the drawing and writes data to a .ew file. The command first prompts you to calibrate the section sheet by picking three points and specify their offsets to the centerline and elevations in order to determine the horizontal and vertical intervals. Corners on the section grid are preferred reference points. Then it prompts you to digitize the cut area and fill area respectively. Place your cursor at one end of the end area and begin digitizing the outline of the area. At the end of the section line, **press Enter** on your puck or keyboard. The end area is completed, and its area is printed on the command line, and you are prompted to digitize next end area. After you finish all the end area at one station, accumulated cut area and fill area are computed and printed out on the screen. All data of cut area and fill area at every station would be saved in the area file (.ew) that you have specified.

Prompts

Calibrate section sheet

Pick First section sheet reference point: *pick a point on the drawing*

Enter offset <0.0>: *press Enter to accept the offset (or enter the offset of the point to the centerline)*

Enter elevation: 1020 *(enter the Elevation of the reference point)*

Pick Second section reference point: *pick a point*

Enter offset: 0 (enter the offset of the point to the centerline)
Enter elevation: 1030 (enter the Elevation of the reference point)
Pick Third section reference point: pick a point
Enter offset: 50 (enter the offset of the point to the centerline)
Enter elevation: 1030 (enter the Elevation of the reference point)
3 calibration points
Transformation type: Orthogonal Affine Projective

Outcome of fit: Success Exact Impossible

RMS Error: 11.69

Standard deviation: 2.40

Largest residual: 14.29

At point: 2

Second-largest residual: 14.29

At point: 3

Digitize cut area (Enter to end): pick a point that is on the outline of the cut area, 0*(0.211129 1030.76)

Digitize cut area (Enter to end): pick a point that is on the outline of the cut area, 1*(11.5804 1030.49)

Digitize cut area (Enter to end): pick a point that is on the outline of the cut area, 2*(17.8643 1030.73)

Digitize cut area (Enter to end): pick a point that is on the outline of the cut area, 3*(19.0216 1032.35)

Digitize cut area (Enter to end): pick a point that is on the outline of the cut area, 4*(-0.777246 1030.75)

Digitize cut area (Enter to end): press Enter to finish

End area: 17.2312

Accumulated Cut Area: 17.2312

More Cut Areas [Yes(A)/<No>(B)]? press A to digitize more Cut Areas, or press B to finish digitizing Cut Areas.

Accumulated Cut Area: 17.2312

Digitize fill area (Enter to end): pick a point that is on the outline of the fill area, 0*(-18.9614 1029.65)

Digitize fill area (Enter to end): pick a point that is on the outline of the fill area, 1*(-18.1315 1030.75)

Digitize fill area (Enter to end): pick a point that is on the outline of the fill area, 2*(-11.9592 1030.49)

Digitize fill area (Enter to end): pick a point that is on the outline of the fill area, 3*(-2.06761 1030.72)

Digitize fill area (Enter to end): pick a point that is on the outline of the fill area, 4*(-10.0082 1030.01)

Digitize fill area (Enter to end): pick a point that is on the outline of the fill area, 5*(-18.531 1029.67)

Digitize fill area (Enter to end): press enter to finish

End area: 8.64646

Accumulated Cut Area: 8.64646

More Fill Areas [Yes(A)/<No>(B)]? press A to digitize more Fill Areas, or press B to finish digitizing Fill Areas.

Accumulated Cut Area: 8.64646

Total Cut Area: 17.2312

Total Fill Area: 8.64646

Store data to file [<Yes>(A)/No(B)]? press A or B

Opened file: C:\Program Files\Carlson Takeoff 2004\DATA\Drawing1.ew

Station Number: 1 (enter Station Number)

Data Stored in file: C:\Program Files\Carlson Takeoff 2004\DATA\Drawing1.ew

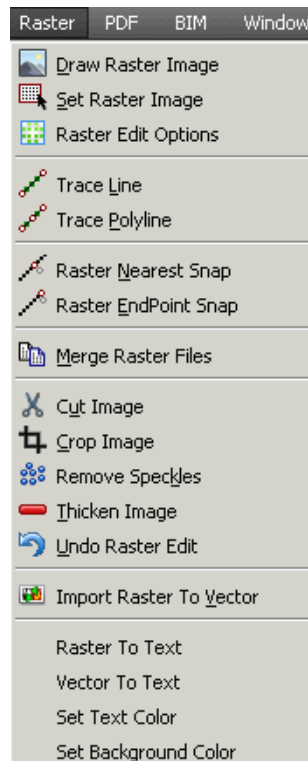
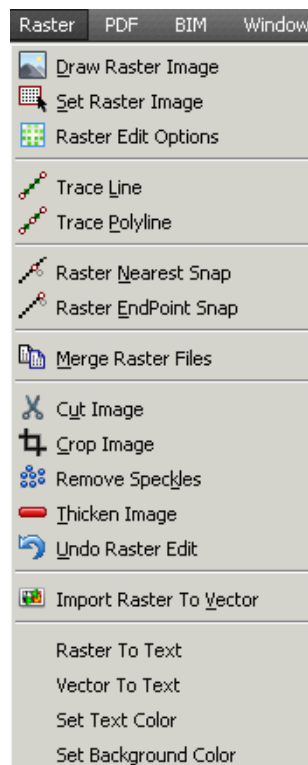
Digitize another station [<Yes>(A)/No(B)]? B (press A or B)

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Keyboard Command: digendar

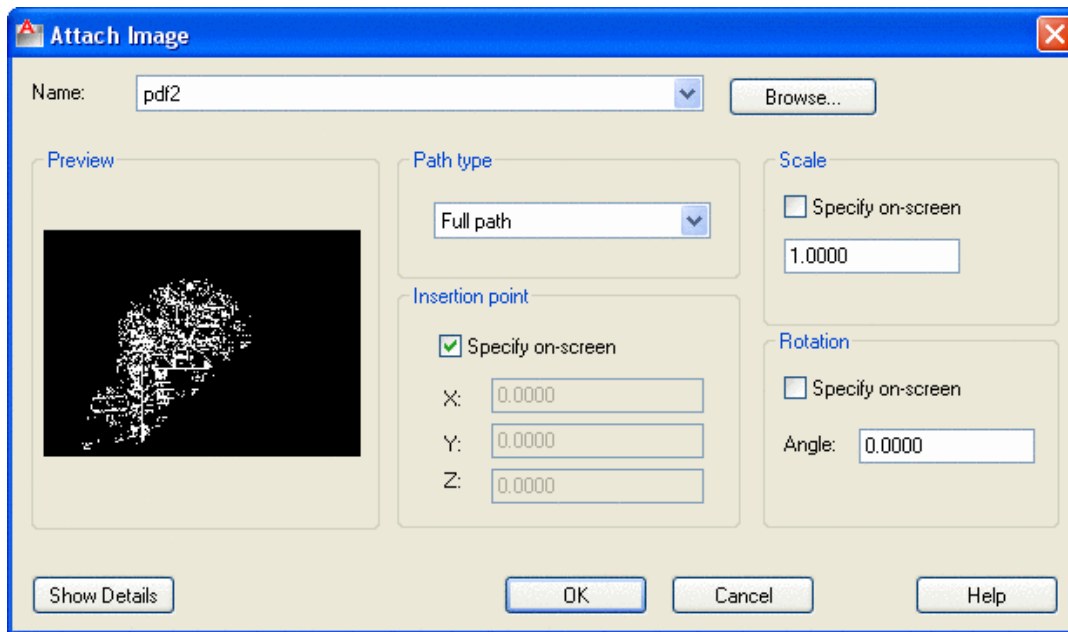
Raster Menu

The Raster menu commands convert raster images into drawing entities.



Draw Raster Image

This command inserts an Image file into your current drawing. After selecting the file you wish to draw, the following dialog is shown:



The name of the file is shown at the top with a preview of the file shown below. To select a different file, click on the Browse button. Path Type can be set to the full (absolute) path, the relative path to the image file, or No Path, the name of the image file (the image file must be located in the same folder as the current drawing file). If the scale factor is known, you can enter it under Scale.

If the scale factor is unknown, it is recommended to use the default scale factor of 1 and adjust the Scale with Edit > 2D Scale once the Image is inserted and a scale factor can be determined. Specify On-Screen allows you to input the scale at the Command prompt. Insertion Point specifies the insertion point for the selected image file. Specify On-Screen is the default. The default insertion point is 0,0,0.

Specify On-Screen Directs input at the Command prompt or the pointing device. If Specify On-Screen is unchecked, enter the insertion point as X, Y, and Z coordinate values. Rotation specifies the rotation angle of the selected image. If Specify On-Screen is selected, you may wait until you exit the dialog box to rotate the object with your pointing device or enter a rotation angle value at the Command prompt. If Specify On-Screen is unchecked, enter the rotation angle value in the dialog box. The default rotation angle is 0.

Below is list of Images that can be inserted into the drawing. For PDF files, use the Import PDF File (loadpdf) command available in Carlson Takeoff.



Prompts

Specify the insertion point: pick on the screen or typing in a coordinate (Example: 1000,1000).

Specify rotation angle: To accept the default value displayed, press Enter, or enter the rotation angle (Example: 90).

Specify scale <1.0>: To accept the default value displayed, press Enter, or enter a scale factor. If the scale factor is not known, which is typical, accept the defaults to this prompt. The proper scale factor can be determined by running Inquiry>Standard Distance on a known distance on the site (ie, the side of a building or the distance across the road). If the side of a building is labeled as 60' and Standard Distance reports it is at 120', then the Scale factor is 0.5 (60/120). Run Edit>2D Scale, select the imported objects, specify a base point of 0,0 and use the Scale Factor you determined with Standard Distance to scale the entities correctly.

After the command has imported the Image file, run View > Zoom > Extents to see the converted entities.

Pulldown Menu Location: Raster > Draw Raster Image

Keyboard Command: imageattach

Prerequisite: None

Set Raster Image

The Raster pull-down has several commands for manipulating images. These commands work with one image at a time. Set Raster Image determines which image in a drawing is "current" to edit. Simply run the command and select the image.

Prompts

Select image: Pick on the image (often you will need to pick on the boundary of the image to select it)

Image selected. Image file: pdf1.bmp

Pulldown Menu Location: Raster

Keyboard Command: rassel

Prerequisite: an image in the drawing

Save As Raster Image

This command saves the current image to a file. This command is a way to make a backup of the image before doing image adjustment commands. Also this command is a way to change the format of the image file such as .bmp to .jpg.

Pulldown Menu Location: Raster > Set Raster Image

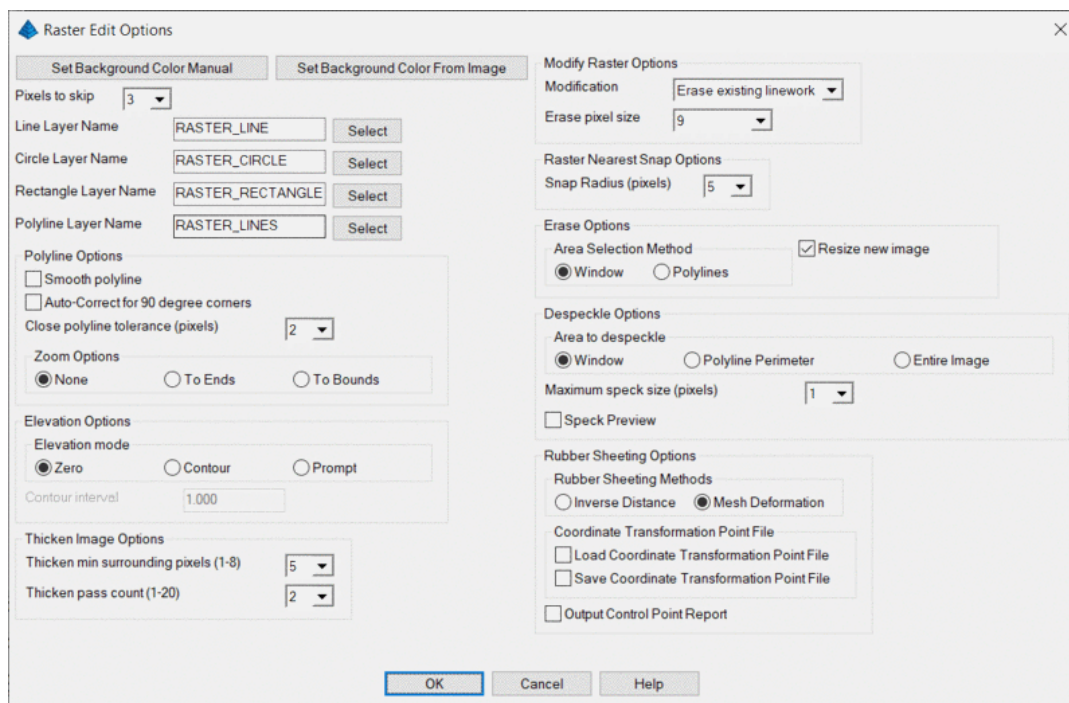
Keyboard Command: rassaveas

Prerequisite: Set Raster Image

Raster Edit Options

General Settings for working with images can be found in the command Raster Edit Options. Defaults are shown below.

Raster Edit Options Menu:



Set Background Color Manual/From Image: These settings apply to the Trace Line, Trace Rectangle, Trace Circle, and Trace Polyline routines. Methods to specify "background" color in image. Pixels in image similar to the "background" color will be considered part of the "background" of an image, as opposed to part of the linework of an image.

Pixels to skip: This setting applies to the *Trace Line* and *Trace Polyline* routines. As the program determines where to draw the new linework, it can "skip" or pass over a given amount of pixels who's color does not match the rest of the linework that is being processed. This allows for longer length polylines to be created on poor quality images. A larger amount of pixels to skip will typically create longer length new linework.

Line Layer Name: determines layer on which to draw lines when using the *Trace Line* routine.

Rectangle Layer Name: determines layer on which to draw rectangles when using the *Trace Rectangle* routine.

Circle Layer Name: determines layer on which to draw circle entities when using the *Trace Circle* routine.

Polyline Layer Name: determines the Layers for new linework in various *Trace Polyline* routine.

Polyline Options: Options specific to the *Trace Polyline* routine.

- **Smooth Polyline:** Interpolates between nodes of the polyline currently being traced to create interstitial nodes in the current polyline that give the current polyline a smoother
- **Auto-Correct for 90 degree corners:** Attempts to determine if a section of line work represents a 90 degree corner. If so, creates a 90 degree corner in the current polyline traced over line work.
- **Close polyline tolerance (pixels):** Sets the close polyline distance tolerance (in pixels, 0-25) for *Trace Polyline*. If the beginning and end of the current polyline are closer than the close polyline distance tolerance then *Trace Polyline* will determine the current polyline to be closed, geometry will be automatically added to the poyline to connect the beginning and end of the polyline. Any redundant nodes created by connecting the beginning and end of the polyline will be removed.
- **Zoom Options:** Specifies how to update view after new geometry is added to the polyline being traced.
 - *To Ends:* Zooms view to the end nearest the most recently added polyline geometry. This end is set as the currently active end)
 - *To Bounds:* Zooms view such that the entirety of the geometry of the currently traced line work is visible.
 - *None:* View does not change after the addition of new geometry.

Elevation Options: Options specifying how polyline geometry is elevated in the *Trace Line*, *Trace Circle*, *Trace Rectangle*, and *Trace Polyline* routines.

- **Elevation mode:**
 - *Zero:* All polyline geometry is drawn with its z value set to 0.0.
 - *Contour:* Sets polyline geometry z values based on elevation set by the *contour prompt* When enter is pressed, or the currently active polyline is closed, the user is prompted to update contour information (again, using the
 - *Prompt:* Prompts user to enter z value of traced polyline geometry. If enter is pressed or polyline is closed, user is again prompted to enter z value of traced polyline geometry.
- **Contour Interval:** When in contour mode, specifies amount to increment elevation after a polyline is ended or closed in *Trace Polyline* routine.

Thicken Image Options: Options used to specify how *RASTHICKEN* routine (command line only, not in pull-down menu) will update modify selected imagery.

- **Thicken min surrounding pixels (1-8):** When running the *RASTHICKEN* command, the routine looks at the surrounding pixels of an individual pixel and will change the color of that individual pixel to the surrounding pixels if minimum amount is met. The lower the number, the more "thickening" or "densifying" will occur.
- **Thicken pass count (1-20):** This is the number of times the *RASTHICKEN* command will run on a selected area. The greater number of passes, the more "thickening" or "densifying" will occur.

1. *No modification, do not modify the "current" image.*

2. *Erase existing linework,* Sets pixels on "current" image to the current background color up to a distance of 'Pixel size' from the current polyline geometry.

3. *Draw new linework,* First erases existing linework (See: Erase existing linework), then draws a 1 pixel wide line

using the color of the current linework on the "current" image.

Modify Raster Options: Options for modifying selected imagery when running any of the Trace routines (*Trace Line*, *Trace Circle*, *Trace Rectangle*, or *Trace Polyline*)

- **No modification:** do not modify the "current" image.
- **Erase existing linework:** Sets pixels on "current" image to the current background color up to a distance of 'Pixel size' from the current polyline geometry.
 - **Pixel Size:** Specifies the number of pixels to set to background color around current polyline geometry.
- **Draw new linework:** First erases existing linework (See: Erase existing linework), then draws a 1 pixel wide line using the color of the current linework on the "current" image.

Raster Nearest Snap Options: Snap Radius, tolerance factor for selecting line work. If line work is found at a distance less than 'Snap Radius' away from most recent user selected pick point, that line work will be added to the geometry of the current polyline.

Erase Options: Options specifying how manipulate imagery using the *Cut Image* and *Crop Image* routines.

- **Area Selection Mode:** Specify method for selecting area of image for cut or crop
- **Resize new Image:** Specifies whether or not to resize image in drawing after cut or crop.

Despeckle Options: Options specific to the *Despeckle* routine.

- **Area To Despeckle:** Specify method for selecting area of image to despeckle.
- **Maximum Spec Size:** Maximum size of a cluster of pixels to consider a speck.
- **Speck Preview:** Specifies whether or not to generate a preview for approval before despeckling image.

Rubber Sheeting Options: Options specific to the *Rubber Sheet* routine.

- **Rubber Sheeting Methods:** Methods for deforming image. *Inverse Distance* method weights the delta x/y from each control point as the inverse of the distance to the control point. This way the closer control points have a higher weight. The *Mesh Deformation* method uses an affine transformation.
- **Coordinate Transformation Point File:** If *Load Coordinate Transformation From Point File* is toggled, control points are specified based on crd file selected by user, otherwise user will be prompted to specify control points. If *Save Coordinate Transformation To Point File* is toggled, user will be prompted to specify crd file to save control points to after transformation.
- **Output Control Point Report:** Toggle on to generate control point report after selecting control points.

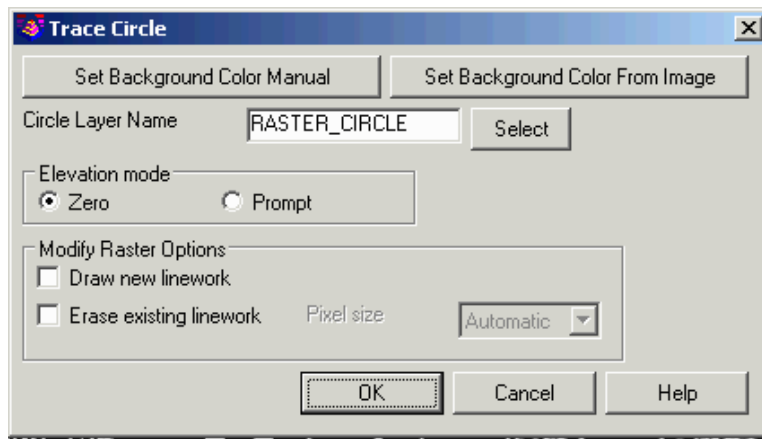
Pulldown Menu Location: Raster > Raster Edit Options

Keyboard Command: rasopts

Prerequisite: none

Trace Circle

This command draws a circle by following a circle in the image. The Draw New Linework option draws the circle in the image as well as the drawing. The Erase Existing Linework option removes the circle in the image by setting the circle pixels to the background color.



Prompts

Trace Circle dialog

Select a point on the circle [Options/Undo] (Enter to exit): *pick a point on the image circle*

Select a point on the circle [Options/Undo] (Enter to exit): *press Enter*

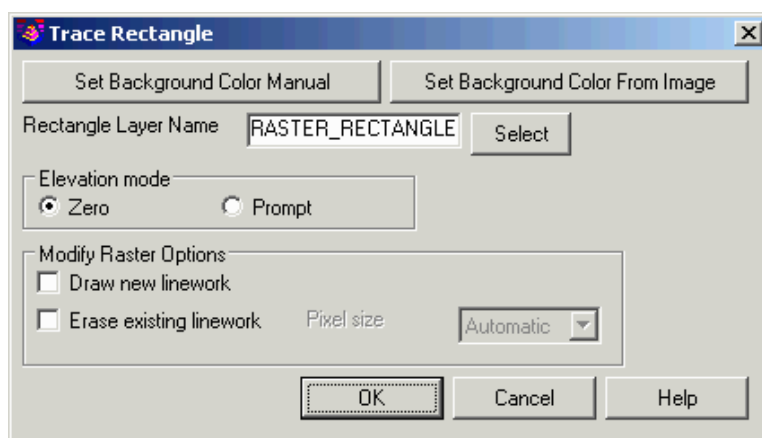
Pulldown Menu Location: Raster

Keyboard Command: rascircle

Prerequisite: an Image

Trace Rectangle

This command draws a rectangle by following a rectangle in the image. The Elevation Mode to Prompt is useful for cases like building pads and you want to set the pad elevations. The Draw New Linework option draws the rectangle in the image as well as the drawing. The Erase Existing Linework option removes the rectangle in the image by setting the rectangle pixels to the background color.



Prompts

Trace Rectangle dialog

Select a point on the rectangle [Options/Undo] (Enter to exit): *pick a point on the image rectangle*

Select a point on the rectangle [Options/Undo] (Enter to exit): *press Enter*

Pulldown Menu Location: Raster

Keyboard Command: rasrect

Prerequisite: an Image

Trace Line

Trace Line will convert a line in an image into a single CAD polyline. If an image is "current" through the **Set Raster Image** command, simply run **Trace Line** and click on a line in the image. In the command line, you will see the **Line Length**, **Angle**, and **Thickness** reported for the new polyline. The command line will also prompt you to change the **Angle**, **Length** or **Reverse** the direction the new polyline if desired. To accept the new line press Enter. To cancel, type C or press Esc.

Different images have different resolutions or quality to them. To account for this, you can adjust the parameters that the program tries to recognize linework in an image with **Edit Raster Options**. Here is also where the default **Layer** for the new polyline is determined.

Prompts

Pick a point for line (Enter to end): Select a line in the image

Line Length=780.00 Angle=269°32'47'' Thickness=5

Enter to accept line or [Angle/Reverse/Length/Cancel]: Type in the first letter in a word to adjust that element of the new polyline. For example, typing in "A" would allow you to adjust the Angle.

Pulldown Menu Location: Raster > Trace Line

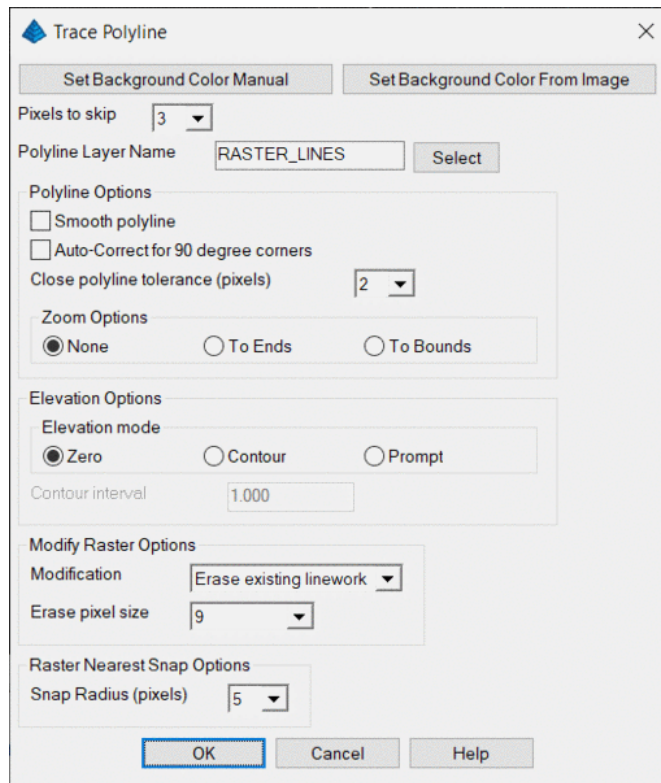
Keyboard Command: rasline

Prerequisite: Set Raster Image

Trace Polyline

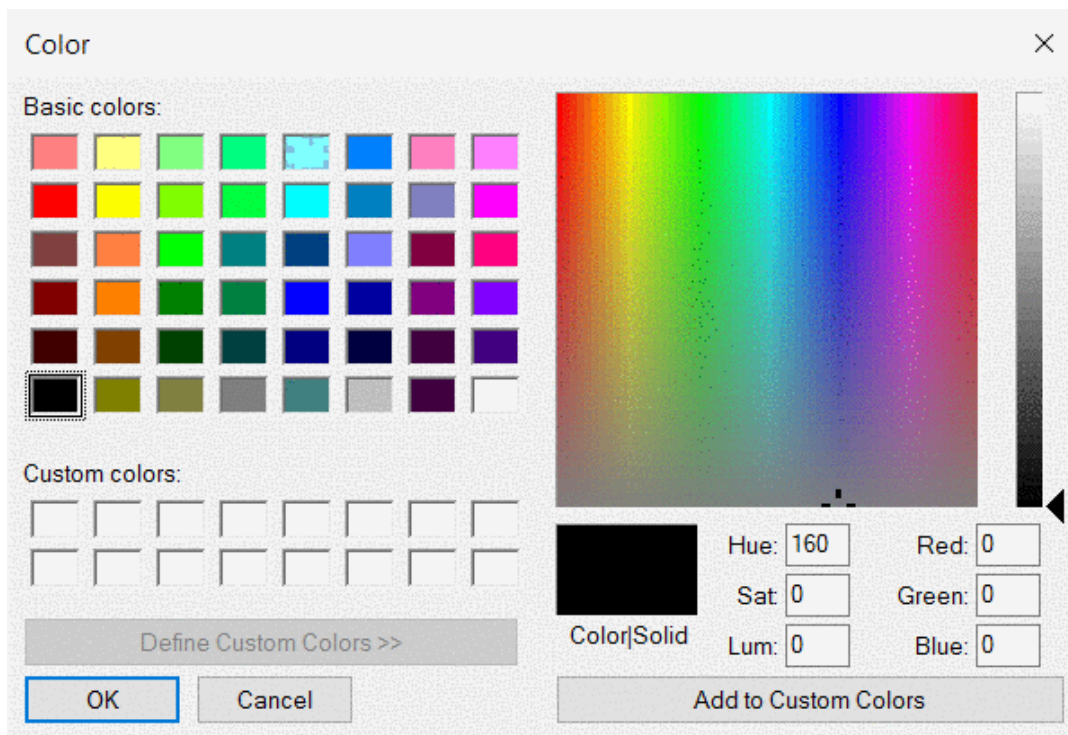
Trace Polyline will convert linework in an image into a CAD polyline with multiple vertices. Trace Polyline requires that an image is "current" this can be accomplished either by using the **Set Raster Image** command or if only one image exists in the drawing, running **Trace Polyline** will automatically set the lone image in the drawing as the "current" image.

Trace Polyline options menu:



Set Background Color Manual/From Image: These settings apply to the Trace Line, Trace Rectangle, Trace Circle, and Trace Polyline routines. Methods to specify "background" color in image. Pixels in image similar to the "background" color will be considered part of the "background" of an image, as opposed to part of the linework of an image.

Trace Polyline currently works best if the image being processed has a distinct background color that pervades most of the image and is easily distinguished from the color of the line work of the image. Set Background Color Manual allows the user to set the color that **Trace Polyline** interprets as the background color manually, by selecting a color from the Color dialog (shown below). Alternately The user can select the background color From Image.



Pixels to skip:

Sets the number of pixels (0-25) that **Trace Polyline** will "skip" (pass over) in order to connect two groups of pixels of the same color. This allows for longer length polylines to be created on poor quality images that contain gaps in between traceable line work. A larger number of pixels to skip will typically create longer length CAD line work. However, if the image line work contains few gaps, then the *Pixels to skip* should be set lower to avoid incorrectly connecting traced line work between unrelated traceable line work.

Smooth Polyline:

Interpolates between nodes of the polyline currently being traced to create interstitial nodes in the current polyline that give the current polyline a smoother appearance.

Auto-correct for 90 degree corners:

Attempts to determine if a section of line work represents a 90 degree corner. If so, creates a 90 degree corner in the current polyline traced over line work.

Close polyline tolerance (pixels):

Sets the close polyline distance tolerance (in pixels, 0-25) for **Trace Polyline**. If the beginning and end of the current polyline are closer than the close polyline distance tolerance then **Trace Polyline** will determine the current polyline to be closed, geometry will be automatically added to the polyline to connect the beginning and end of the polyline. Any redundant nodes created by connecting the beginning and end of the polyline will be removed.

Zoom Options:

1. *To Ends*, zooms view to the end nearest the most recently added polyline geometry. This end is set as the currently active end (see *switch ends*).
2. *To Bounds*, zooms view such that the entirety of the geometry of the currently traced line work is visible.
3. *None*, view does not change after the addition of new geometry.

Elevation Mode:

1. *Contour*, sets polyline geometry z values based on elevation set by the *contour prompt*. When enter is pressed, or the currently active polyline is closed, the user is prompted to update contour information (again, using the *contour*

prompt).

2. *Prompt*, prompts user to enter z value of traced polyline geometry. If enter is pressed or polyline is closed, user is again prompted to enter z value of traced polyline geometry.
3. *Zero*, all polyline geometry is drawn with its z value set to 0.0.

Contour Interval:

When in contour mode, specifies amount to increment elevation after a polyline is ended or closed in Trace Polyline routine.

Modify Raster Options:

1. *No modification, do not modify the "current" image.*
2. *Erase existing linework*, Sets pixels on "current" image to the current background color up to a distance of 'Pixel size' from the current polyline geometry.
3. *Draw new linework*, First erases existing linework (See: Erase existing linework), then draws a 1 pixel wide line using the color of the current linework on the "current" image.

Raster Nearest Snap Options:

Snap Radius, tolerance factor for selecting line work. If line work is found at a distance less than 'Snap Radius' away from most recent user selected pick point, that line work will be added to the geometry of the current polyline.

Prompts: Pick segment or [Options/Manual point/cross line Pick/Undo/Close/Switch active end/Exit] (Enter to end polyline):

(Select pick point to continue tracing polyline.)

(O)-Options, opens the Trace Polyline options dialog.

(M)-Manual point, allows user to contribute additional geometry to current polyline manually, ie not automatically traced.

(P)-cross line Pick, allows user to select pick point by crossing line work to be added with a cross line.

(U)-Undo, undoes previously drawn geometry.

(S)-Switch active end, switches the end to which geometry will be added (given new geometry begins near active end).

(E)-Exit, exits **Trace Polyline**.

*(C)-Continue, if no geometry has been traced, continue allows user to select on screen polyline geometry and use that as the basis for new line work.

Pick manual point or [enable snap Nearest/Switch active end/Exit] (Enter to end):

(Pick manual point to add to polyline geometry.)

(N)-enable snap Nearest, see **raster Nearest Snap Options**.

(S)-Switch active end, switches the end to which geometry will be added.

(E)-Exit, exits **Trace Polyline**.

Enter elevation or [Increment 1.00/Direction +/-/Exit] Elevation <0.00>:

(Enter elevation in contour mode.)

(I)-Increment, increment the elevation according to "direction" (either + or -).

(D)-Direction, toggles incremental direction from + to - or vice versa.

(E)-Exit, exits **Trace Polyline**.

Enter elevation or [Exit] Elevation <0.0>:

(Enter elevation in Prompt mode.)

(E)-Exit, exits **Trace Polyline**.

Pulldown Menu Location: Raster > Trace Polyline

Keyboard Command: raspline

Prerequisite: Set Raster Image

Raster Nearest Snap

Similar to the standard Object or Entity Nearest Snap with CAD entities, **Raster Nearest Snap** will snap to the nearest point on a linework segment or point in an Image. This command can be used with Draw commands such as Draw > 2D Polyline.

Pulldown Menu Location: Raster > Raster EndPoint Snap

Keyboard Command: rnea

Prerequisite: most Draw commands

Raster EndPoint Snap

Similar to the standard Object or Entity EndPoint Snap with CAD entities, **Raster EndPoint Snap** will snap to the closest endpoint of a point or linework segment in an Image. This command can be used with Draw commands such as Draw > 2D Polyline.

Pulldown Menu Location: Raster > Raster EndPoint Snap

Keyboard Command: rend

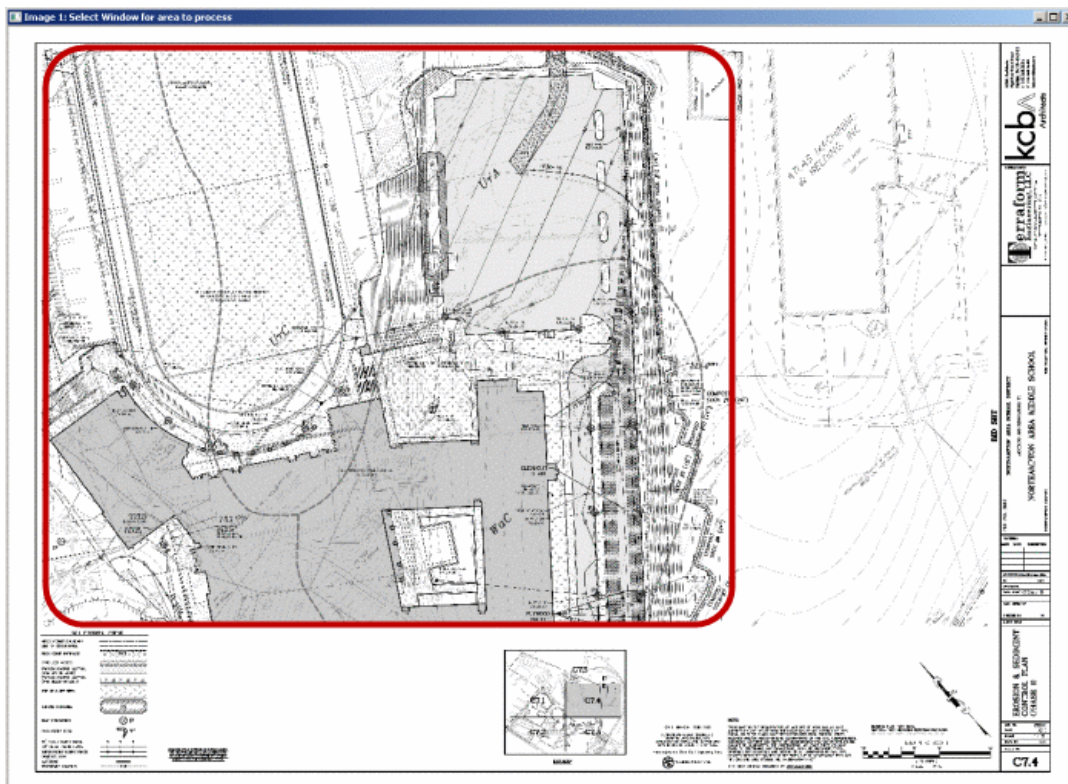
Prerequisite: most Draw command

Merge Raster Files

This commands merges bitmaps or other images (not pdf-based, unless the pdf has been turned into an image). There are two different methods for merging raster files: Automatic and Manual.

Automatic Merge

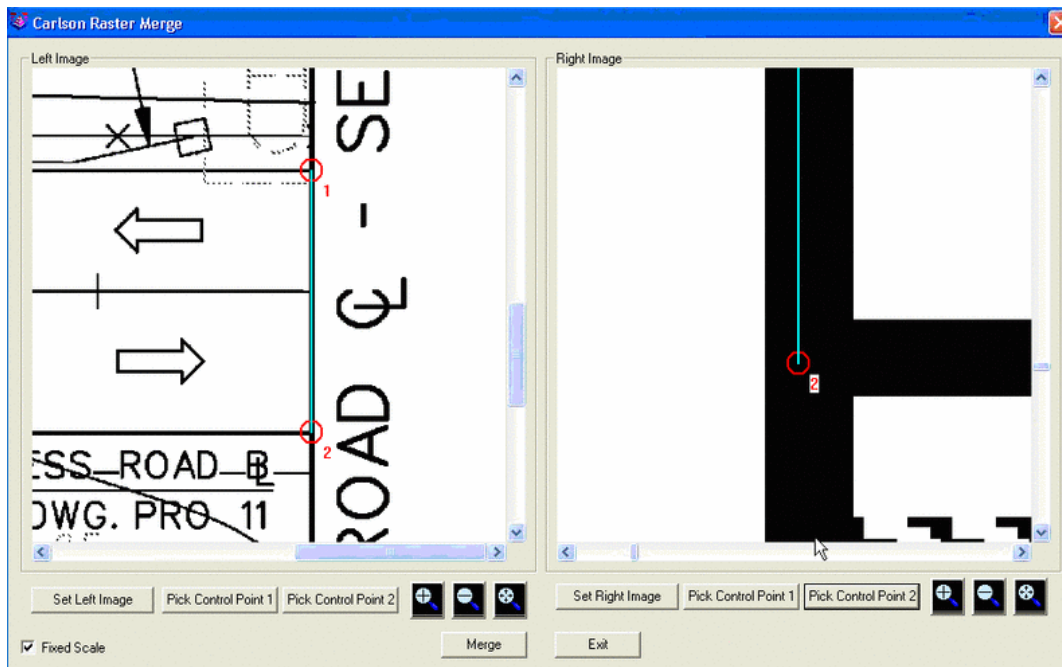
The program will first prompt you for the two existing files to merge and then for a new file name to create of the results. After picking and naming these files, the program will bring up a preview of each file to merge. Here, you will be prompted to "Select Window for area to process":



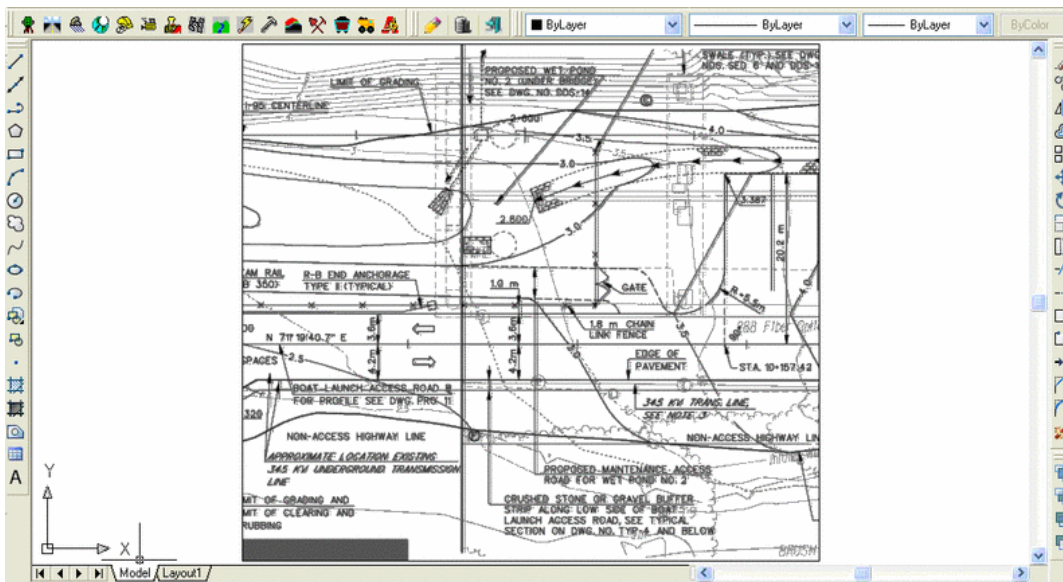
Create a Window along the given or approximate match lines (shown above in red), excluding elements like the title box. After doing this on both images, the program will perform a best-fit match and crop out any unneeded linework. The results will be shown in a preview window with a prompt: "Select Window for area to output". This allows you to select the whole or a smaller area of the preview to create the final image from. After Windowing the desired area, the new image will be created and the command will exit out.

Manual Merge

First, run "Set Left Image" and "Set Right Image" to determine the files to merge. Next, pick identical control points on the left side and right side. Zoom in and pick the best that you can. This establishes the scale, rotation, and alignment for the merge. Note: the preview windows are labeled "Left Image" and "Right Image", but the program will merge images "Top" to "Bottom" if the control points are aligned in that orientation.



After establishing control points, click the Merge button and it merges the left with the right side. When you click Save Image, you can save it in a number of distinct forms (typically as a .bmp is sufficient). Notice the program automatically removed the match line text, with no overlap. The key is that your two reference points for scale and rotation, which match, must be at the linear overlap line, because everything to either side is removed automatically. Originally, the two images overlapped, but now that has been removed.



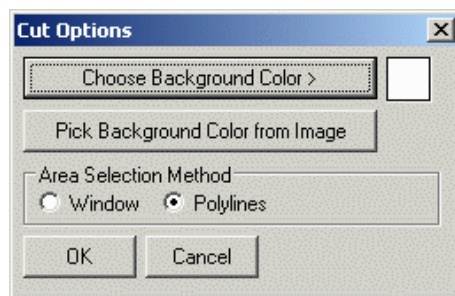
Pulldown Menu Location: Raster > Merge Raster Files

Keyboard Command: rasmerge

Prerequisite: Two or more Images you'd like to combine

Cut Image

This command wipes out an area of an image using the specified Background Color and is used to clean up an image. The area to cut can be selected either by picking a window or selecting closed inclusion polylines. Everything in the image to the inside of the area is removed. Warning: this command will update the image seen on the screen as well as the source Image.



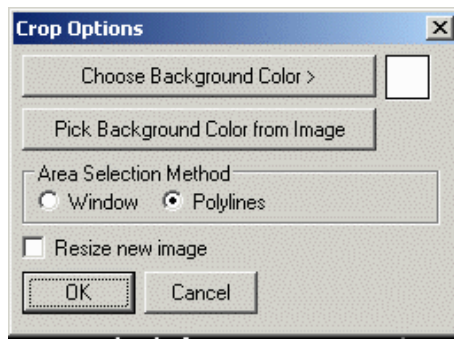
Pulldown Menu Location: Raster

Keyboard Command: rascut

Prerequisite: an Image and a closed polyline

Crop Image

This command is used to clean up an image. Everything in the image to the outside of crop area is wiped out using the specified Background Color. The area is defined either by picking a window or selecting closed inclusion polylines. The Resize New Image option will make the image file smaller to remove the cropped out area. Warning: this command will update the image seen on the screen as well as the source Image.



Pulldown Menu Location: Raster

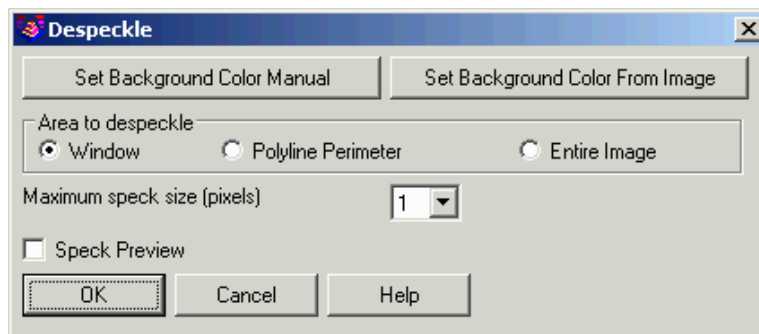
Keyboard Command: rascrop

Prerequisite: an Image and a closed polyline

Remove Speckles

Images that have been scanned in from paper plans often have unwanted black dots or "speckles" that can be removed with this command. Depending on the image, there is a max speck size that can be set. The larger the size indicated, the more specks will be removed. You can specify the area to remove specks from based on the entire image, a window area or by a closed inclusion polyline. The Speck Preview option will highlight the specs to be removed for confirmation before the removal.

Warning: this command will update the Image seen on the screen as well as the source Image.



Pulldown Menu Location: Raster > Adjust Image

Keyboard Command: rasdespeckle

Prerequisite: an Image

Mirror Image

This command flips an image either top to bottom or left to right.

Prompts

Mirror direction [**<Left-right>/Up-down**]? *press Enter*

Pulldown Menu Location: Raster

Keyboard Command: rasmirror

Prerequisite: Set Raster Image

Decompress Image

This command changes the format of a TIF image from the compressed TIF format to the uncompressed TIF format. First the program prompts for the input image file to read and then the output image file to create. The program handles LZW compression.

Pulldown Menu Location: Raster

Keyboard Command: rascompress

Prerequisite: Image File

Deskew Image

This command rotates an image. To do the rotation, this command defines a line by picking two points on the image and the image is rotated to make this line horizontal.

Prompts

Pick point 1 of the horizontal: *pick point 1*

Pick point 2 of the horizontal: *pick point 2*

Crop deskewed image [Yes/<No>]? *press Enter*

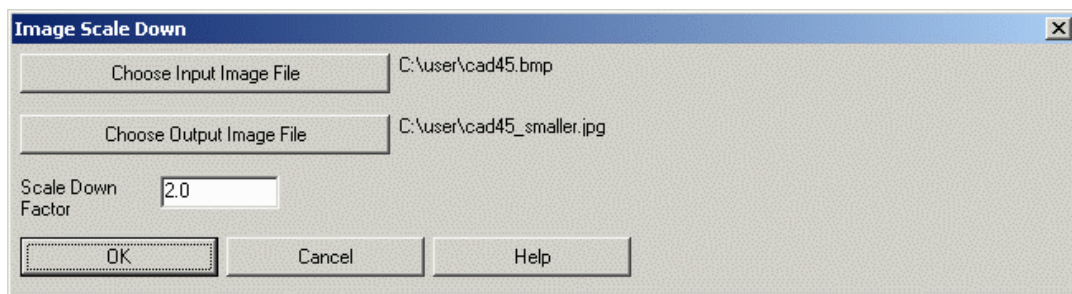
Pulldown Menu Location: Raster

Keyboard Command: rasdeskew

Prerequisite: Set Raster Image

Shrink Resolution

This command decreases the resolution of an image file. This feature applies when an image resolution is too dense to process in some other routines or you simply want to make the file size smaller to save space. This command prompts for the input image file to read and the output image file to write. The Scale Down Factor sets how many times smaller to make the output file.



Pulldown Menu Location: Raster

Keyboard Command: rasres

Prerequisite: Set Raster Image

Black/White By Threshold

This command makes the current image binary by assigning grays and colors to either black or white based on a brightness threshold. A binary image is easier to process in commands like Raster To Text and Raster To Vector. The Calculate Threshold function analyzes the image using the specified Threshold Method to estimate a good threshold value.



Pulldown Menu Location: Raster

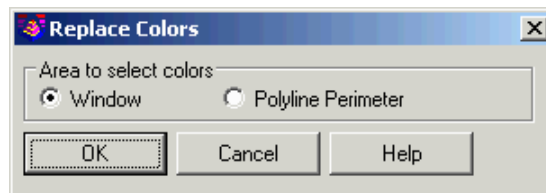
Keyboard Command: rastreshold

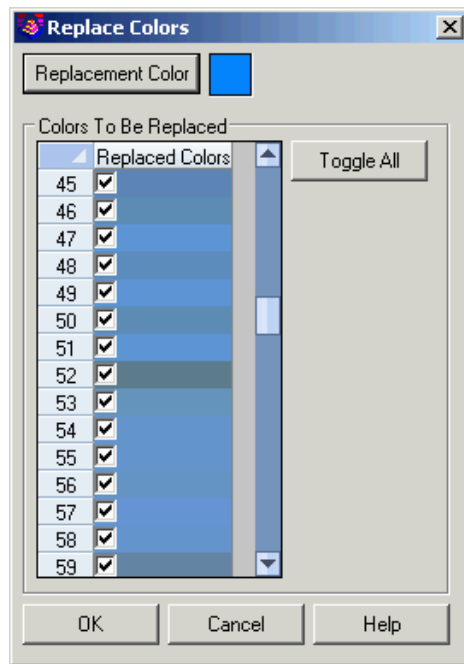
Prerequisite: Set Raster Image

Replace Color

This command replaces colors in the image with a specified new color. You can specify the area to get colors by window area or by a closed inclusion polyline. The program makes a list of all the colors from the selected area. From this list, toggle which colors that you want to be replaced and pick the Replacement Color button to select the new color.

Warning: this command will update the Image seen on the screen as well as the source Image.





Pulldown Menu Location: Raster > Adjust Image

Keyboard Command: rascolorsub

Prerequisite: an Image

Brighten/Darken Image

This command changes the brightness of the current image and applies to color images. Enter a positive number to brighten and negative to darken.

Prompts

Enter brightness change from -255 to +255: 64

Pulldown Menu Location: Raster

Keyboard Command: rasbright

Prerequisite: Set Raster Image

Negative Image

This command inverts the colors of an image. For example, black pixels are changed to white.

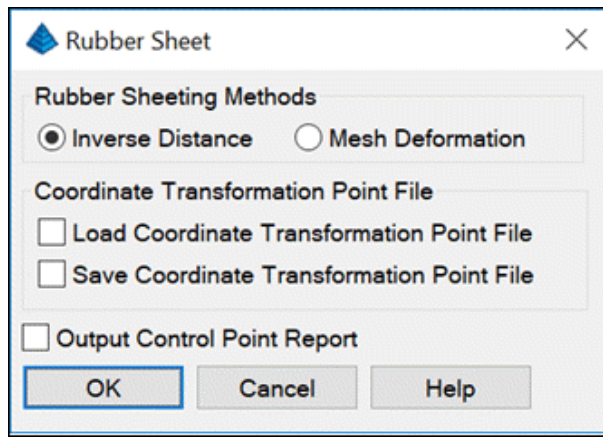
Pulldown Menu Location: Raster

Keyboard Command: rasnegative

Prerequisite: Set Raster Image

Rubber Sheet

This command adjusts an image to fit control points. This image adjustment applies when the image distortion varies and requires more than a single translate, rotate and scale transformation. This command changes the image file. The original image file is backed up to a separate file with a .bak extension.



There are two methods to model the adjustment from the control points. The Inverse Distance method weights the delta x/y from each control point as the inverse of the distance to the control point. This way the closer control points have a higher weight. The Mesh Deformation method uses an affine transformation. The control points can be either loaded from a file stored previously by Rubber Sheet, or by screen picking. With picking, the control points are defined by screen picking a point on the image and then the point to move this point on the image to. In the example shown here, the roads in the image don't line up with the road linework. After picking the road points on the image and the corresponding points on the linework, the image is adjusted to fit the road linework. The Output Control Point Report generates a report of the control point coordinates.

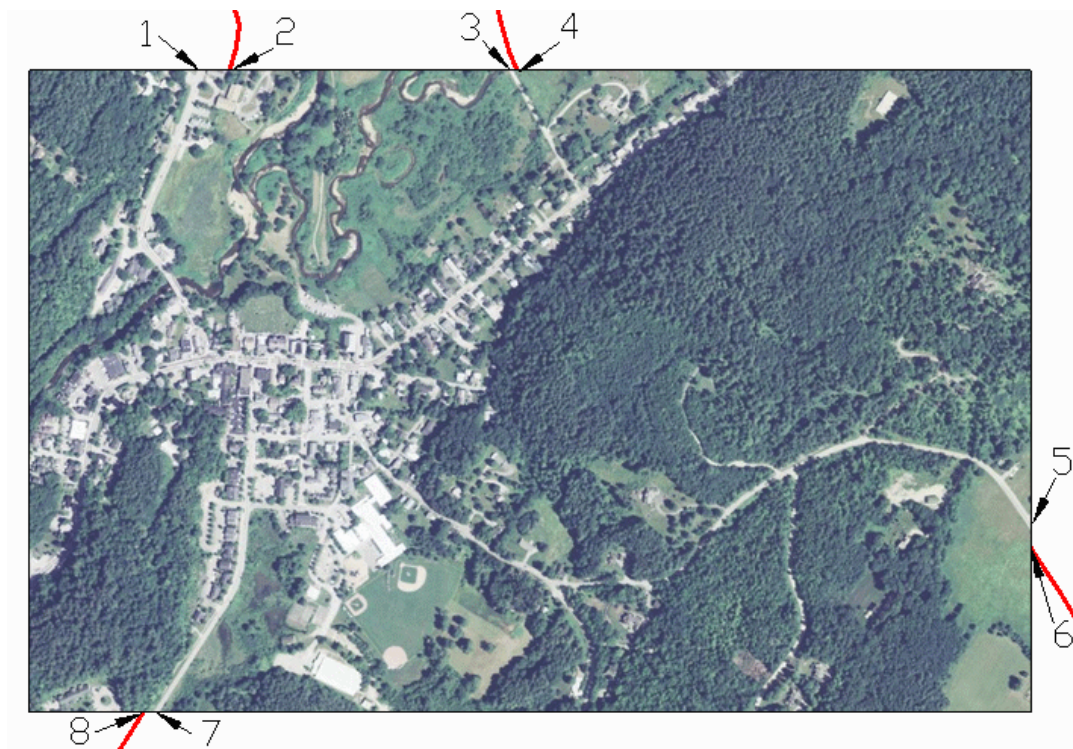


Image before Rubber Sheet showing control points



Image after Rubber Sheet

Prompts

- Pick origin of control point 1:** *pick point 1*
- Pick destination of control point 1:** *pick point 2*
- Pick origin of control point 2:** *pick point 3*
- Pick destination of control point 2:** *pick point 4*
- Pick origin of control point 3 (Enter to end):** *pick point 5*
- Pick destination of control point 3:** *pick point 6*
- Pick origin of control point 4 (Enter to end):** *pick point 7*
- Pick destination of control point 4:** *pick point 8*
- Pick origin of control point 5 (Enter to end):** *press Enter*

Pulldown Menu Location: Raster

Keyboard Command: rasrubber

Prerequisite: an Image

Undo Raster Edit

Undo Raster Edit will revert an Image back to its original form (both on-screen and source) from changes made by: Image Cut, Image Crop, Remove Speckles, Trace Line or Trace Polyline. It will not remove any polylines created by Trace Line or Trace Polyline.

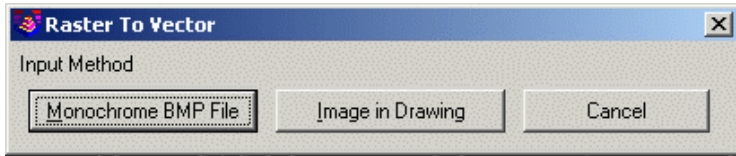
Pulldown Menu Location: Raster > Undo Raster Edit

Keyboard Command: rasundo

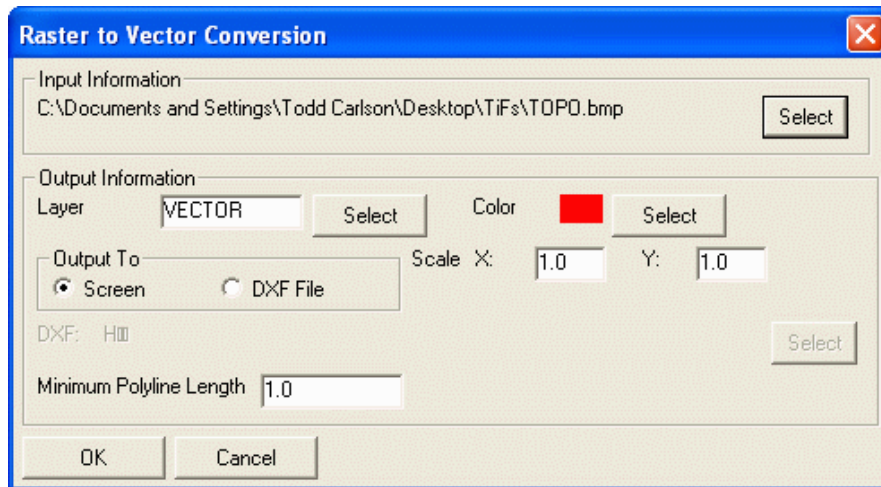
Prerequisite: an Image that has been edited

Import Raster To Vector

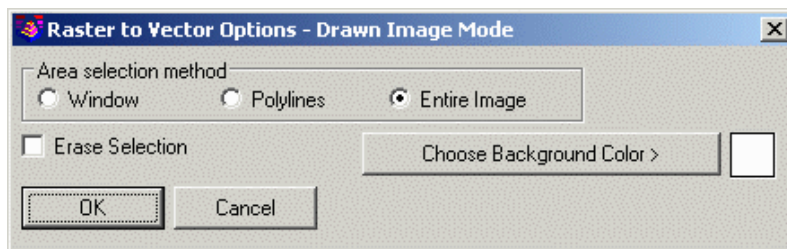
This command creates polylines from a raster image. The image can be either an external .bmp file or an image inserted in the drawing.



For the external file to process other file types, such as .tif or .jpeg, open the image up in Microsoft Paint and Save As the file as a monochrome .bmp. In the dialog below, you can specify the Layer, Color, Scale, and whether to draw the entities on the screen or write a .dxf file. Minimum Polyline Length will reduce the amount of line segments created from the conversion.



For an image in the drawing, you can process the entire image or limit the area to process by window or inclusion polylines. The Erase Selection option wipes out the image with the specified Background Color in the area that is processed.



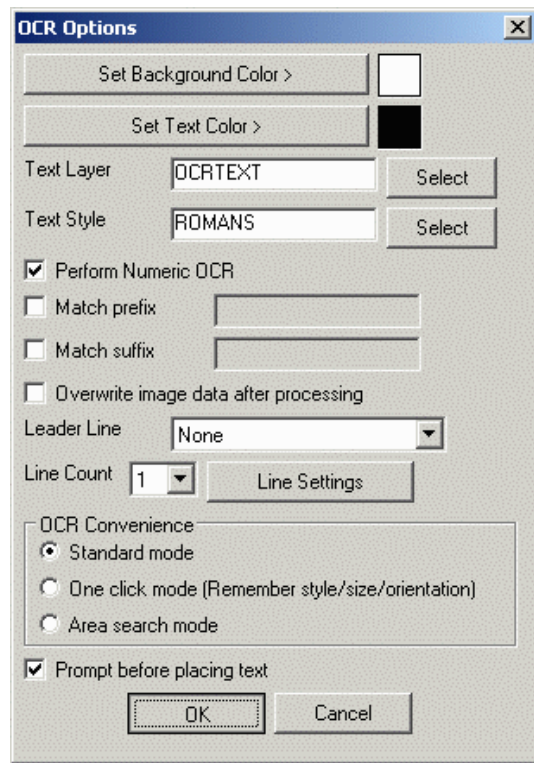
Pulldown Menu Location: Raster

Keyboard Command: ras2vec

Prerequisite: a monochrome .bmp file

Raster To Text

This command will create text from an image in your drawing generated from commands such as PDF Import, Raster Import, Attach Image, etc. one line at a time. From the OCR (**Optical Character Recognition**) Options dialog, you have the ability to identify the Background and Text Color of the image being used. The Text Layer and the font Style the text will appear on once converted can also be set here.



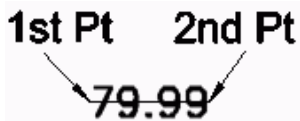
Perform Numeric OCR will limit the text recognition to only numeric values. By default, this command attempts to convert to alphabetic characters as well. When dealing with only numeric text, turn this feature on so that no alphabetic characters are accidentally used during the conversion. Use the Match Prefix or Match Suffix to help the recognition when the numbers have an alpha prefix or suffix like "TC 123.4".

The **Overwrite Image** option will erase the raster text from the image after it has been converted to CAD text.

The **Leader Line** option draws a leader along with converting the text.

For the OCR Mode, the **Standard Mode** prompts to pick two points across the raster text for each text to convert. These points define the area of the image to process. The **One Click Mode** converts the first text like Standard Mode and gets the text size and angle properties from this first text. Then for the additional text to convert, you only need to pick a single point on the text instead of two points. For the **Area Search Mode**, the first text is again processed the same as Standard Mode and then you window an area and the program looks for other raster text with the same size and angle as the first text.

The **Prompt Before Placing Text** shows the conversion results in a dialog with a chance to review, edit or cancel.





After clicking OK to the OCR Options dialog, you will be prompted for a "start of strike-line" then "end of strike-line". Pick left to right through the text you want to convert. A **Replacement Text dialog** will appear. Here you can review the values that the program has converted and change them if needed before accepting them by clicking OK. You will then be prompted to "strike-through" another line of text. Press Enter to end.

Prompts

OCR dialog

Pick start of stike-line or [Options] (Enter to end): *pick 1st point*

Pick end of stike-line (Enter to end): *pick 2nd point*

Pick start of stike-line or [Options] (Enter to end): *press Enter*

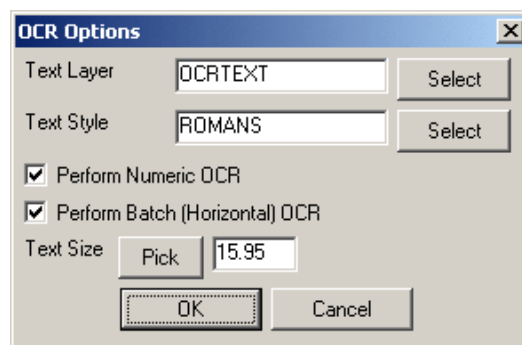
Pulldown Menu Location: Raster

Keyboard Command: RASOCR

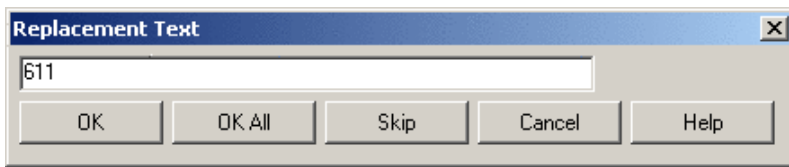
Prerequisite: An image in the drawing

Vector To Text

This command will create text from linework in a drawing. From the OCR (**Optical Character Recognition**) Options dialog, set the Text Layer and the font Style the text will appear on in drawing once converted. **Perform Numeric OCR** will limit the text recognition to only numeric values. By default, this command attempts to convert to alphabetic characters as well. When dealing with only numeric text, turn this feature on so that no alphabetic characters are accidentally used during the conversion.



By default, this command converts one line at a time. **Perform Batch (Horizontal) OCR** allows you to convert multiple line at once as long as the text is aligned horizontally. Choosing a **Text Size** helps the program to distinguish text linework vs. standard linework. "Pick" allows you to pick two points in the plan view to set the text height to be used.



After clicking OK to the "OCR Options" dialog (with "Perform Batch (Horizontal) OCR" checked off), you will be prompted for a "start of stike-line" then "end of stike-line". Pick left to right through the linework you want to convert. With "Perform Batch (Horizontal) OCR" checked on, you will be prompted to "Select objects", which can be individually picked linework or windowing around a group of linework. After making your selection, a **Replacement Text dialog** will appear. Here you can review the values that the program has converted and change them if needed before accepting them by clicking OK. You may also Skip the conversion altogether or OK All the conversions at once in this dialog.

Pulldown Menu Location: Raster

Keyboard Command: vecocr

Prerequisite: Linework to convert to text

Set Text Color

This command sets which color to use as the text color for other Raster commands such as Raster To Text. For example, one image could have a black background and white text, and another image could have the opposite with a white background and black text. The Raster commands need to know which type of text color the image has.

This command uses the current image. If the current image isn't already set, then the program prompts to select the image. Then the program reports the current text color and prompts to pick a point on the image. The program reads the color from the image at the picked point to set the text color.

Prompts

Select image: Pick on the image (often you will need to pick on the boundary of the image to select it)

Current text color: red=0 green=0 blue=0

Pick a point in the image to set the text color: *pick point*

Text color changed to: red=201 green=234 blue=157

Pulldown Menu Location: Raster

Keyboard Command: ocrtextcolor

Prerequisite: an image in the drawing

Set Background Color

This command sets which color to use as the background color for other Raster commands such as Trace Line. For example, one image could have a black background and white lines, and another image could have the opposite with a white background and black lines. The Raster commands need to know which type of background the image has.

This command uses the current image. If the current image isn't already set, then the program prompts to select the image. Then the program reports the current background color and prompts to pick a point on the image. The program reads the color from the image at the picked point to set the background color.

Prompts

Select image: Pick on the image (often you will need to pick on the boundary of the image to select it)

Current background color: red=255 green=255 blue=255

Pick a point in the image to set the background color: *pick point*
Background color changed to: red=232 green=232 blue=232

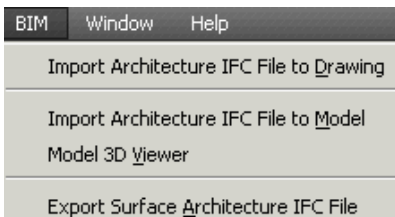
Pulldown Menu Location: Raster

Keyboard Command: ocrbgcolor

Prerequisite: an image in the drawing

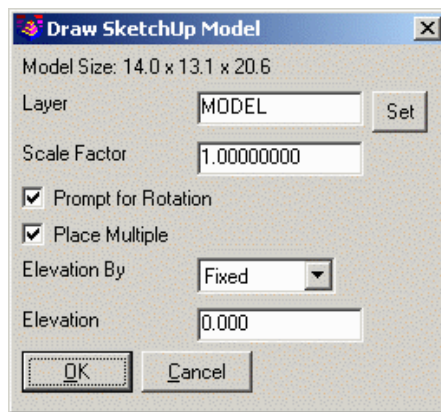
BIM Menu

The BIM menu commands are for exchanging Carlson project data with BIM.



Import SketchUp File to Drawing

This command creates a CAD block from a SketchUp model and inserts this block into the drawing.



After selecting the .skp file to import, the options dialog shows the dimensions of the model at the top of the dialog.

Layer: Sets the layer for the block in the drawing.

Scale Factor: Allows resizing the model. For example, when the SketchUp model is in inches and the drawing in feet, use a scale factor of 0.0833333.

Prompt For Rotation: This option prompts to pick a second point for rotation of the block in the drawing.

Place Multiple: Sets whether to place a single block in the drawing or allow for drawing multiple copies.

Elevation By: The Pick method uses the elevation of the picked points. The Fixed method uses the **Elevation** from the dialog. The Surface method prompts for a grid or triangulation file to use for the elevation.

See <http://www.sketchup.com/> for information on SketchUp and see <https://3dwarehouse.sketchup.com/> for a collection of available SketchUp models.

Prompts

Select SketchUp File

Draw SketchUp Model dialog

Pick position: *pick a point*

Pick rotation: *pick a point*

Pick position (Enter to end): *press Enter*

Pulldown Menu Location: BIM

Keyboard Command: skp2dwg

Prerequisite: .skp file

Import Architectural IFC File to Drawing

The **Import Architectural IFC File to Drawing** permits architectural information stored in an "open IFC" format (see <http://www.buildingsmart.org/standards/ifc> for additional information regarding Industry Foundation Classes) to be imported into the current drawing. Common uses of this routine are to permit the graphical representation of a building (or its component entities) within the Carlson environment for construction staking.

Note:

- In the event the data represented by the IFC file needs to be scaled to better align with the units found within Drawing Setup, consider using the Scale Wizard command.
- In the event the data represented by the IFC file needs to be aligned to better fit the coordinates of the project, consider using the 2D Align command.
- In the event the data represented by the IFC file needs to be elevated for better visualization within the project, consider using the Change Elevations command.
- Quick visualizations of the imported IFC file can be viewed via the 3D Viewer Window command (experiment with the **Clip Plane** control along the bottom of the Viewer window to slice through the model).
- As entities are imported from the IFC file, they are assigned to a layering convention based on the IFC entity type. A template drawing (Carlson_ifc.dwt, set-able via Carlson Configure is available that contains an introductory layer scheme for many of the classifications found within IFC files.
- Various sample IFC files abound on the Internet and can be found at locations such as:
 - <http://www.ifcwiki.org/index.php/Examples>
 - <http://download.bimserver.org/ifc-examples/>
 - <http://openifcmodel.cs.auckland.ac.nz/Model>

Prompts

IFC File to Read: *Navigate to and select an IFC 2x or later file and click Open.*

Pulldown Menu Location(s): BIM

Keyboard Command: ifc2dwg

Prerequisite: IFC 2x or later file

Import Architectural IFC File to Model

Similar to the procedure found within the Import Architectural IFC File to DWG command, the **Import Architectural IFC File to Model** command imports and converts an architectural IFC file into:

- *.OBJ - An "object" geometry file
- *.MLT - A "material" file

This command is useful for larger IFC data sets that might otherwise tax the performance of the CAD engine or clutter a drawing file.

Note:

- The OBJ and MLT file(s) are created with the same file name as that of the IFC file.

- The representation of the OBJ file can be viewed with the Model 3D Viewer command.

Prompts

IFC File to Read: *Navigate to and select an IFC 2x or later file and click Open.*

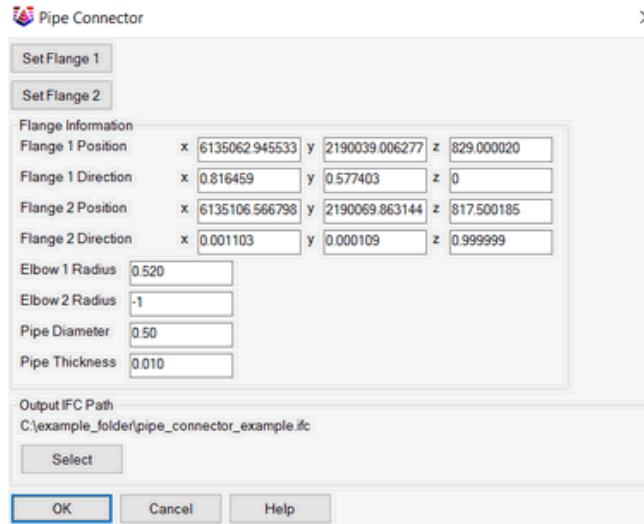
Pulldown Menu Location(s): BIM

Keyboard Command: ifc2obj

Prerequisite: IFC 2x or later file

Pipe Connector

This command draws the geometry of the connector pipe between two points in the current drawing, and outputs the connector pipe geometry to an IFC file.



Set Flange 1 & 2:

Allows you to select groups of points representing the first and second (start and end) points to be connected. Groups of points are expected to be laid out in a pattern resembling a circle. The best estimate of the center of this group of points will be the first or second connection point respectively¹. Using Set Flange 1 or 2 will also set flange direction 1 and 2 such that the first point is "facing" the second point and the second point is "facing" the first point.

Flange 1 & 2 Position:

Position of the first and second point that the connector pipe will join.

Flange 1 & 2 Direction:

Direction from which the beginning and end of the connector pipe will emanate.

Elbow 1 Radius:

Radius of the 1st elbow in the connector pipe geometry. This will be -1 if no pipe geometry is feasible².

Elbow 2 Radius:

Radius of the 2nd elbow in the connector pipe geometry. This will be -1 if only 1 elbow is needed to connect the start and end points of the connector pipe, or if no pipe geometry is feasible².

Pipe Diameter:

Diameter of connector pipe².

Pipe Thickness:

Thickness of the connector pipe².

Output IFC Path:

Output file path for the IFC file. Can be set using the "Select" button.

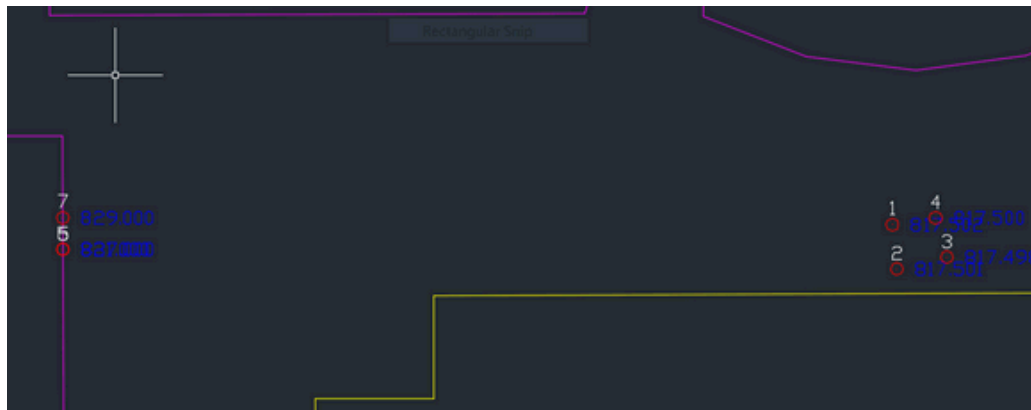
1. Pipe Connector performs various calculations when determining the geometry of the connector pipe. Because of this, the actual point that is determined by Set Flange 1 & 2 may not be the center of the group of points selected by the user. For example, if the two points can be joined by a pipe with only one elbow if one of the points is altered "slightly" Pipe Connector will alter the two points to simplify the pipe connector geometry in this way.

2. The feasibility of the connector pipe geometry depends on many variables. The two most common:

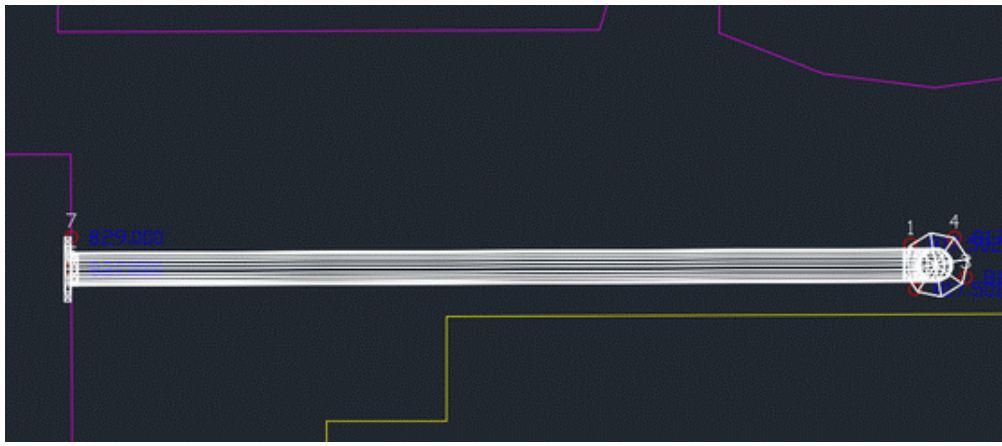
A) set flange 1 and 2 will attempt to set the the direction of the first and second point such that they face each other. However as the direction is editable, it is possible to set the point and direction of the first and second points of the connector geometry that cannot be connected.

B) Given a pipe Diameter and thickness it may be impossible to curve a pipe such that it connects the first point of the connector pipe to the second point of the connector point. If a larger radius is feasible, Pipe Connector will automatically compute this value, otherwise radius 1 and 2 will be set to -1.

Example



In this drawing the points (1, 2, 3, 4) and (5, 6, 7) will be used to specify the start and end point of the connector pipe. After being selected by Set Flange 1 and Set Flange 2 respectively and pressing okay, an ifc file will be written to the output IFC path and the connector geometry will be drawn in the drawing "demo3." (see below).



Pulldown Menu Location: BIM

Keyboard Command: pipe_connector

Prerequisite: Points on connections

Model 3D Viewer

Similar to the functionality offered by the Surface 3D Viewer command, the **Model 3D Viewer** permits the viewing of 3D models (such as those produced by the Import Architectural IFC File to Model command).

Note:

- Please reference the Surface 3D Viewer command for the functionality regarding the various controls available in the **Model 3D Viewer** command.

Prompts

Select Model to View: *Navigate to and select the desired model file and click Open.*

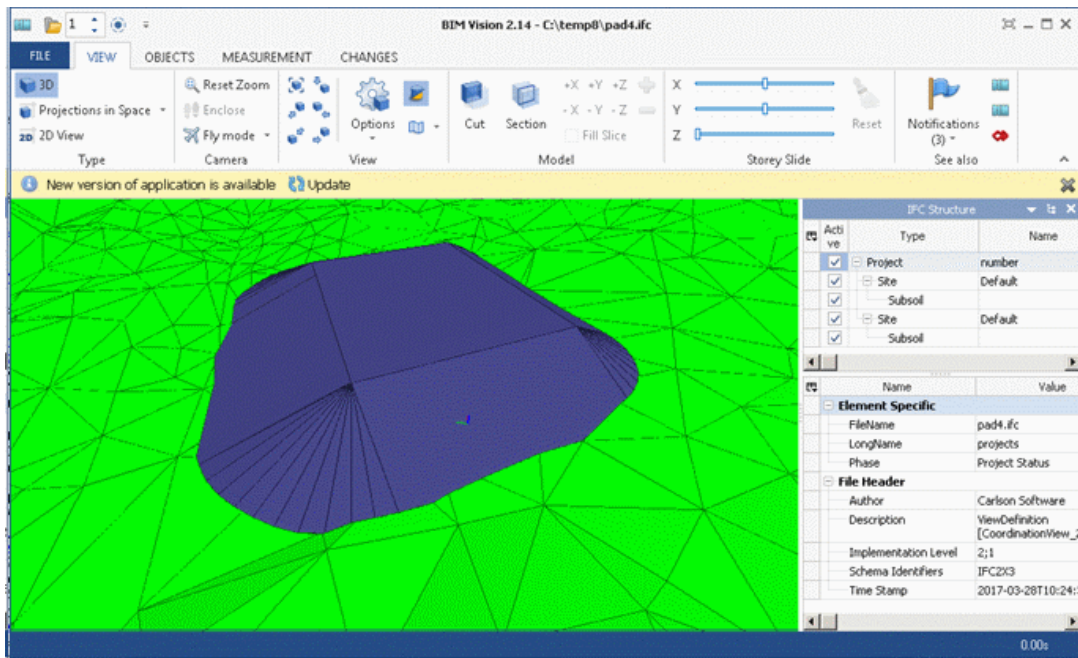
Pulldown Menu Location(s): BIM

Keyboard Command: viewmdl

Prerequisite: A valid 3-D MDL, DXF or OBJ file

Export Surface Architectural IFC File

This command outputs a Carlson triangulation surface model (.tin) to an "open IFC" format (see <http://www.buildingsmart.org/standards/ifc>) and subsequently shared with other applications that make use of IFC-structured files (typically, architectural-oriented products). When the Carlson triangulation surface uses different colors, the IFC file is created with separate models for each color which is a way to create separate models for each different type of surface area.



Prompts

Select Triangulation File to Read: *Navigate to and select the desired Carlson surface model file and click Open.*

IFC File to Write: *Navigate to the desired folder location and indicate the desired IFC file name to write and click Save.*

Pulldown Menu Location(s): BIM

Keyboard Command: tin2ifc

Prerequisite: A Carlson surface model (TIN or FLT) file

Export Solid Model IFC File

This command creates an IFC model file from a Carlson Solid model file (.MDL). For example, you can use Takeoff to create solid models of the pavement subgrades and then create IFC models to load into a site model. First the program prompts for the MDL file to read and then prompts for the IFC file to create.

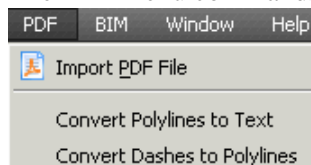
Pulldown Menu Location: BIM

Keyboard Command: mdl2ifc

Prerequisite: A Carlson solid model (.MDL)

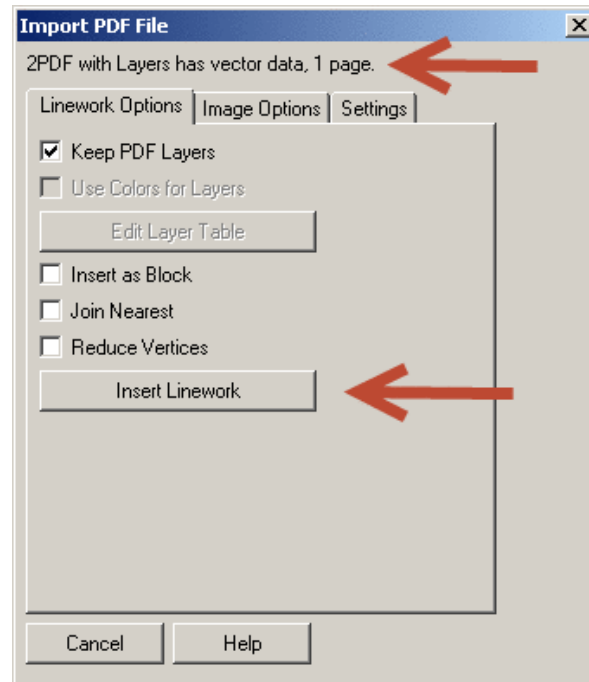
PDF Menu

The PDF menu commands are for importing from PDF.



Import PDF File

This command allows you to bring in an Adobe .PDF file and convert it into CAD linework or an image file. First the program prompts to select the file you want to convert. Next, the below dialog will open. The first line in the dialog will show important information about the PDF, including the file name, data type (Vector or Raster), and the number of pages. It will also display "Layers" or "3D" if those elements are found in the file. Vector vs Raster data is an important distinction. Vector PDFs are typically of higher quality and can be brought in with the "Linework Options". Raster PDFs are lower quality and it is recommended to bring them in with the "Image Options".

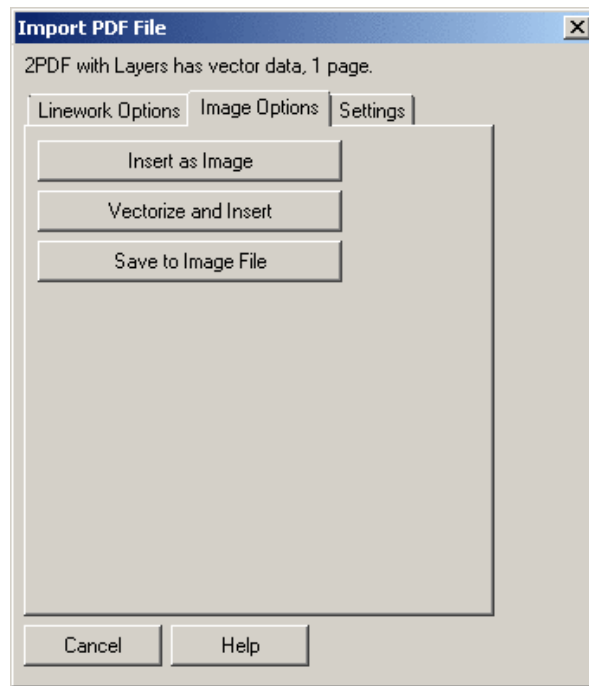


Linework Options

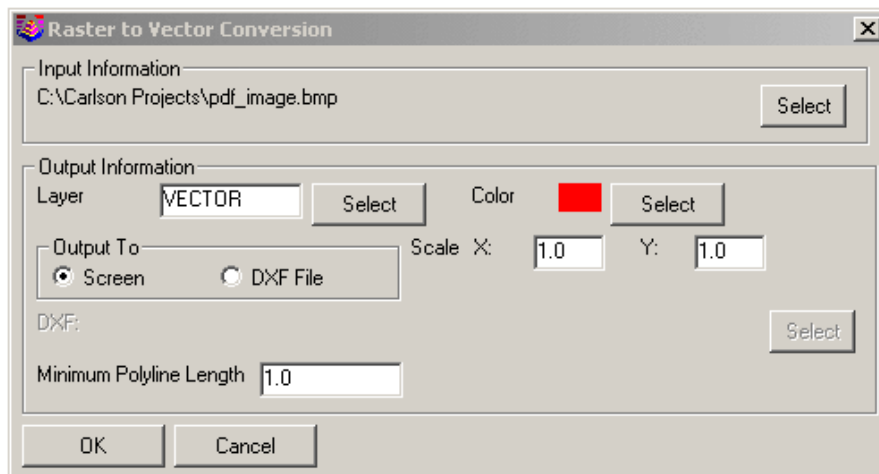
The "Insert Linework" button shown above will convert the PDF file into usable CAD polylines. If layers are present in the PDF file, "Keep PDF Layers" will import the PDF layers into CAD layers. If there are not layers in the PDF file, "Use Colors for Layers" will parcel out the converted polylines onto layers based on the linework's color, gray scale, width, horizontal and vertical orientation. "Insert as Block" will create one block entity for all the linework. "Join Nearest" can be used to clean up the converted linework by combining adjacent linework but requires longer processing time. The "Reduce Vertices" simplifies the linework by removing extra vertices. The "Use Zero Linewidth" option creates linework with the line width set to zero. Otherwise the linewidth will be set from the linework properties in the PDF. The "Skip Tiny Lines" option doesn't draw any super short lines which is helpful for handling a PDF that has a dense hatch pattern made of tiny lines.

Image Options

In some cases, it may be preferred to bring in the PDF as an Image. "Image Options" shown below has three different methods for working with the converted Image. "Insert as Image" will convert the PDF into a Image file that is then inserted into the drawing. When inserted, the Image is visible on the screen, but unusable as CAD linework. "Save to Image File" simply saves the Image file without inserting it into the drawing.



"Vectorize and Insert" can be used to convert raster PDFs into CAD linework. If the program recognizes "vector data", indicated in the status section of the dialog, then it is recommended to use the "Linework Options" to produce CAD linework. Using "Vectorize and Insert" on "raster data", you will receive this additional dialog:



In this dialog, you can specify the Layer, Color, Scale, and whether to draw the entities on the screen or write a .dxf file. Minimum Polyline Length will reduce the amount of line segments created from the conversion.

Inserting the Converted PDF

When you are inserting the converted PDF into the drawing, will you receive the following prompts:

Pick point to insert PDF: Specify the insertion point for the PDF converted linework by either picking on the screen or typing in a coordinate (Example: 1000,1000).

Specify rotation angle: To accept the default value displayed, press Enter, or enter the rotation angle (Example: 90).

Specify scale <1.0>: To accept the default value displayed, press Enter, or enter a scale factor. If the scale factor is not known, which is typical, accept the defaults to this prompt.

Note: The proper scale factor can be determined by running Edit>Scale Wizard after inserting the PDF.

After the command has imported the PDF file, run View > Zoom > Extents to see the converted entities.

Settings

The program has two different conversion engines available, "Visual Integrity" and "Ghostscript". "Visual Integrity" is recommended and the default. In order for Ghostscript to work you need to download and install it from the website <http://sourceforge.net/projects/ghostscript/>. Click on the "View all files" button to the right of the "Download Now!" button. A list of Ghostscript versions will appear below (i.e., 9.23, 9.22). Choose the 9.23 or newer version. After downloading the .exe to your computer double click on it to install. Ghostscript is a free software and should only take a few minutes to install. Once it is installed, Carlson Takeoff will automatically utilize this software when Ghostscript is selected as the PDF Import Engine.

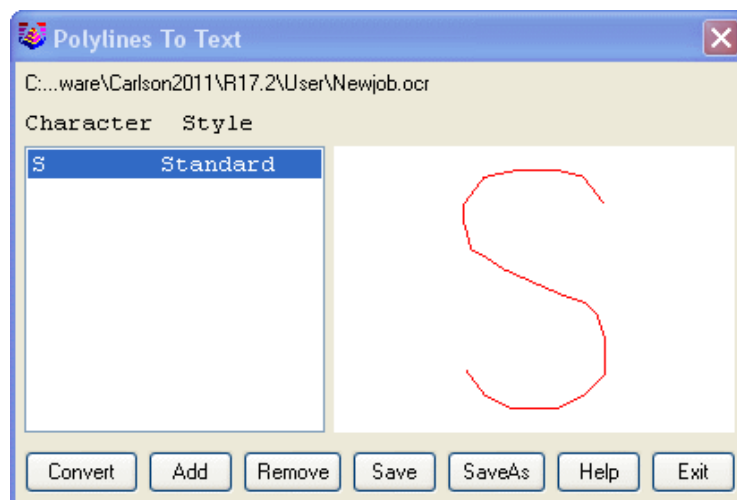
Pulldown Menu Location: Tools > Import/Export

Prerequisite: a PDF file, Ghostscript installed

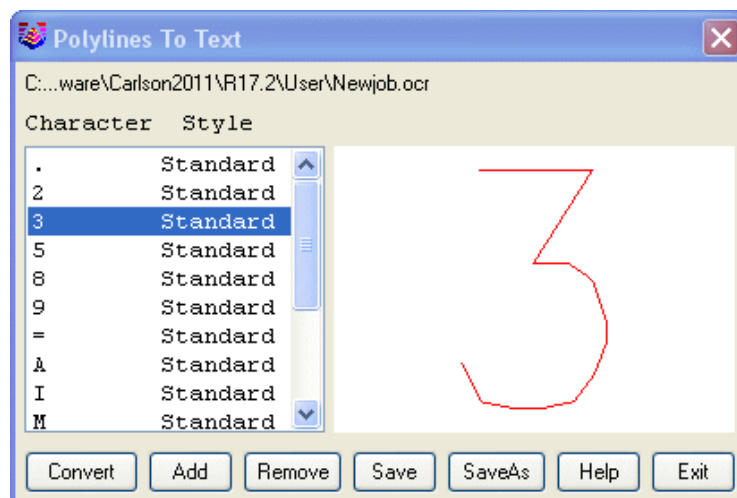
Keyboard Command: loadpdf

Convert Polylines To Text

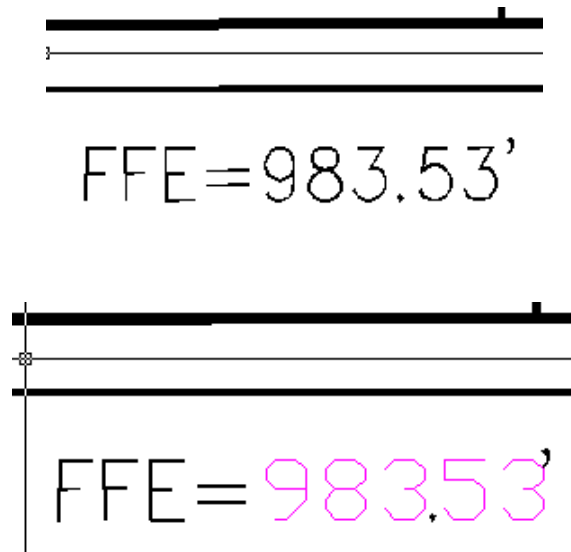
This command will create text from polylines generated by a PDF or raster import. When you run the command you will be prompted to load or create a Character file (.ocr). This file acts as a library that associates polylines in your drawing with a particular character (A, B, C, 1, 2, 3, etc.) to convert into text. From the Polyline To Text dialog, you have the ability to modify the Character library and to Convert selected linework in the drawing.



To build the Character library, pick on the Add button. Here you type in the character you'd like to add, choose the font style the text will appear on, and then select the polyline(s) that comprise the character.



After the desired library has been compiled, Convert will prompt you to select the linework you wish to process from the library into text. Below is an example of PDF linework convert from polylines into text.



Run Save or Save As to use the Character file (.ocr) on future projects.

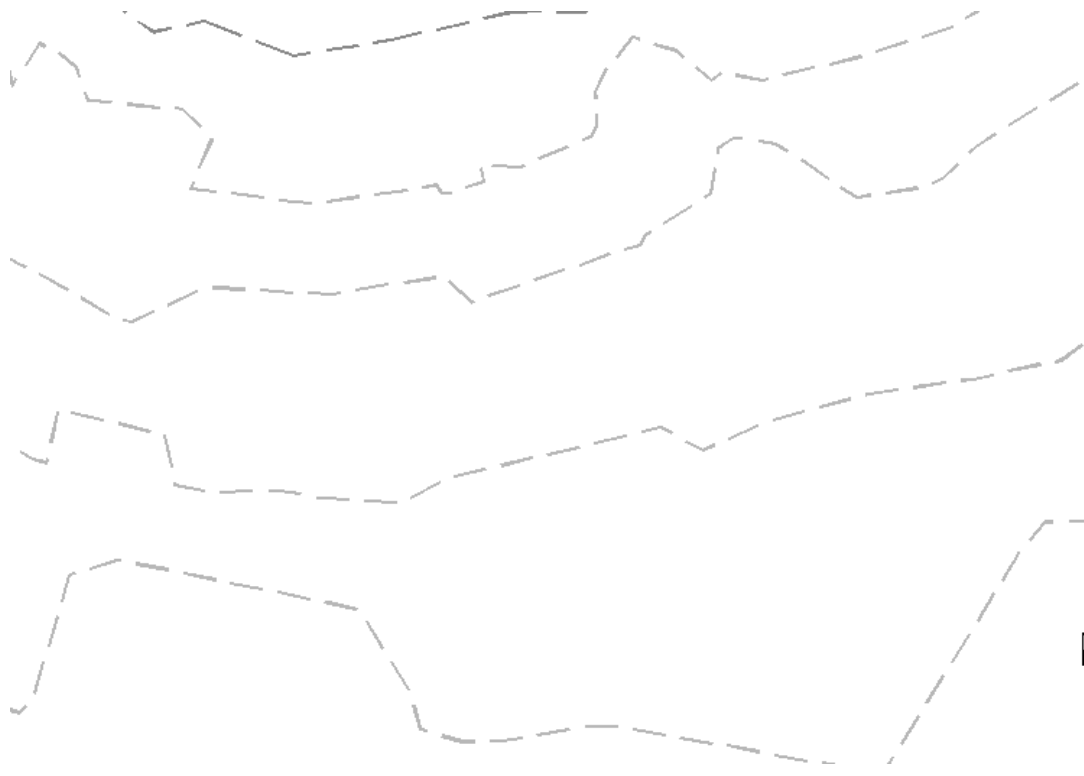
Pulldown Menu Location: PDF

Keyboard Command: pl2txt

Prerequisite: Polylines to convert

Convert Dashes To Polylines

This command connects sequential dashes into polylines. There are two processing methods. The Individual method creates one polyline at a time. The Selection method processes a selection set of dashes and creates multiple polylines. The program will prompt you to select a sample and adjacent dash line. This identifies the layer and approximate gap between dash lines. It is helpful to isolate this layer before running the command. Next, select the dash lines to process and press enter. Below are typical results of the conversion.



Before Conversion



After Conversion

Pulldown Menu Location: PDF

Keyboard Command: dash2pl

Prerequisite: Dashed linework to convert into continuous polylines.

Drawing to 3D PDF

This command creates a PDF file from 3D entities in the drawing. The elevations of the entities are used to make the PDF a "3D PDF" that can then be opened and oriented in 3D with a standard PDF viewer such as Adobe Acrobat Reader.

Note:

- See the 3D Viewer to 3D PDF command for a complete description of the 3D PDF Options controls.
- Additional controls can be exposed in the 3D PDF via a "right mouse click" while the 3D PDF is open.

Prompts

Select entities for 3DPDF.

<Select entities>: *Select the entities that should be written to the 3D PDF file.*

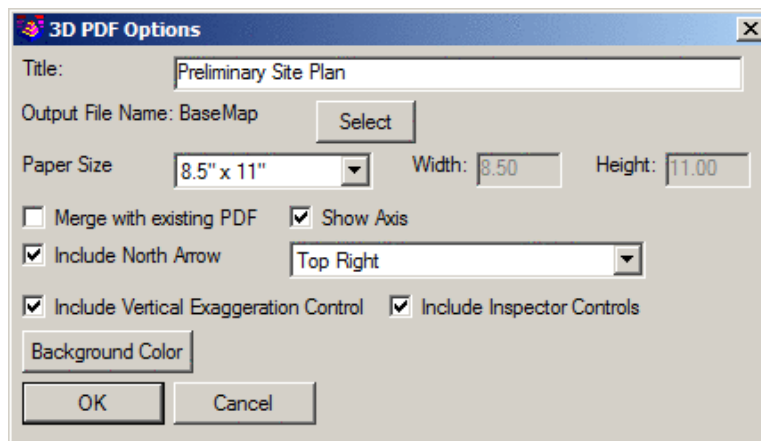
Pulldown Menu Location: PDF

Keyboard Command: dwg2pdf3d

Prerequisite: Entities in a drawing

3D Viewer to 3D PDF

This command creates a PDF file from 3D viewer files (.3DX) which are created in commands such as 3D Viewer Window and Surface 3D Viewer. The PDF file is a "3D PDF" that can then be opened and oriented in 3D with a standard PDF viewer such as Adobe Acrobat Reader.



Title: Indicate a title (if any) that should appear at the top of the PDF file you are about to create.

Output File Name: Use the **Select** button to supply the desired PDF file name to which the drawing content shall be written.

Merge with existing PDF: Use this option with the Template File Name to add the 3D PDF onto an existing PDF. This option is a way to have your own title block as a template PDF and then insert in the 3D PDF.

Paper Size: Indicate the desired PDF paper size.

Append with existing PDF: When enabled, the new PDF output will be appended as a new page to the **Output File Name** if the file already exists.

Plan View Margins: Controls the margins around the 3D view window and the border of the page.

Background Color: Controls the background color for the PDF.

Show Axis: When enabled, the X,Y,Z axis directions will be displayed in the PDF.

Include North Arrow: When enabled, a North Arrow will be displayed in the PDF.

Include Vertical Exaggeration Control: When enabled, a Vertical Exaggeration control will be available in the PDF.

Include Inspector Controls: When enabled, an inspection (Signature) control is provided in the PDF.

Background Color: Indicate the desired background color of the PDF.

Note:

- To create a 3D PDF from entities directly selected from a drawing, see the Drawing to 3D PDF command.
- Additional controls can be exposed in the 3D PDF via a "right mouse click" while the 3D PDF is open.

Prompts

3DX File to Create PDF: *Select the *.3DX file that should be written to the 3D PDF file.*

Pulldown Menu Location: PDF

Keyboard Command: 3dx2pdf

Prerequisite: A *.3DX file

Convert LDD Contours

This command allows you to convert LandDesktop contours (known as AECC_CONTOUR objects) into polylines. You must have the AEC Object Enabler installed before using this command. If you do not have the object enabler installed, download the latest version from www.autodesk.com.

You can use the *List* command to determine if contours are polylines or AECC_Contour objects. Here is an example listing:

AECC_CONTOUR Layer: "CONT-MJR"

Space: Model space

Handle = 429

Major Contour Interval

Elevation: 1005.00

Smoothing: None

Number of Vertices: 48

Open

Length: 560.25

Constant width: 0.00

Style Name: Standard

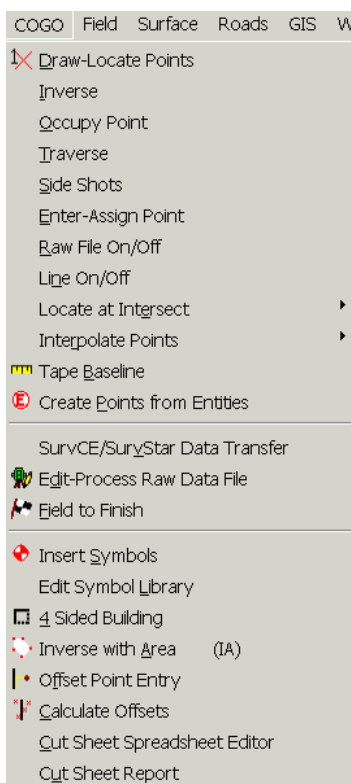


Field Module

12

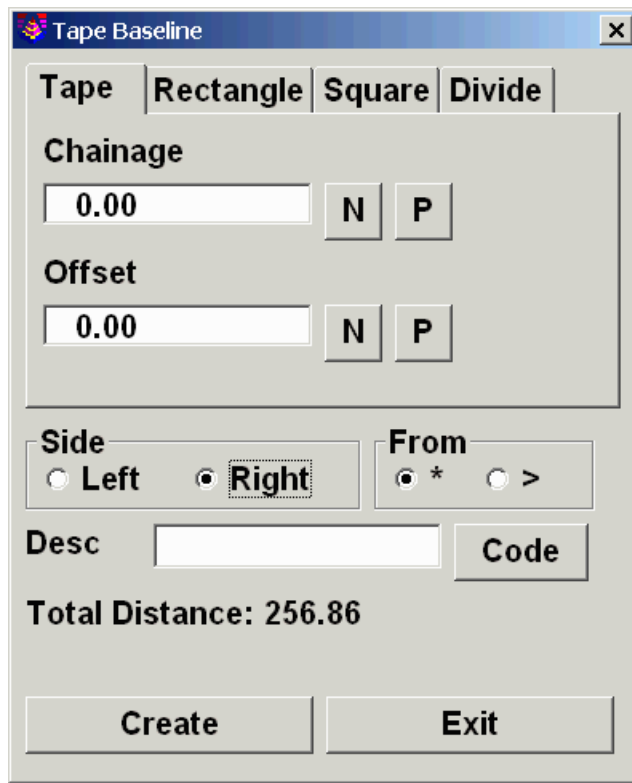
COGO Menu

The most of the commands in the COGO pull-down menu are described in the Survey manual. Only the few commands that are specific to Field are described here.



Tape Baseline

This command creates points or linework along a baseline that is defined by two points. After specifying the baseline start and end points by either entering point numbers from the coordinate file or screen picking points, the program has a dialog with different methods for creating the points. The Tape method creates points at the specified chainage (distance) and offset from the baseline. reports the cut or fill between your current position and a design surface. The design surface can be one flat elevation, a grid file, a triangulation file, a road design file, or a section file. The Rectangle method draws a rectangle as a closed polyline using two points specified by chainage and offset from the baseline. The Square method draws a square as a closed polyline with a starting point at a baseline endpoint and the other corner specified by a distance along the baseline. The Divide method creates points at an interval between the baseline endpoints.



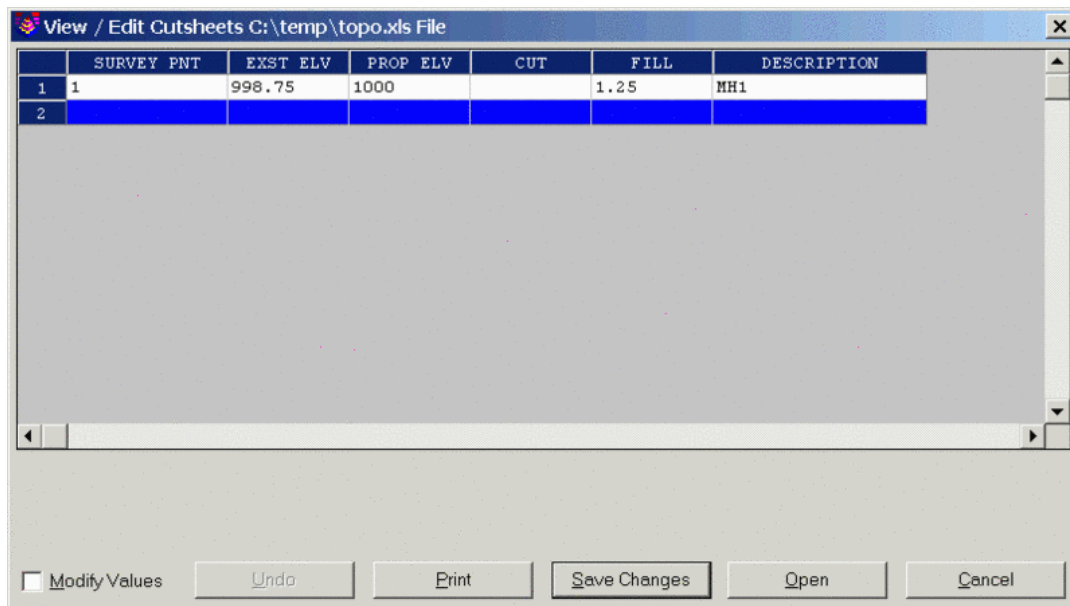
Pulldown Menu Location: COGO

Prerequisite: None

Keyboard Command: tape.blinc

Cutsheet Spreadsheet Editor

This command edits and reports cut sheet data that is stored in an Excel (.xls) file. To create this data with Field stakeout routines, the option to Store Cutsheet Data In Spreadsheet must be set active in Configure Field->Stakeout Settings.



Pulldown Menu Location: COGO

Prerequisite: None

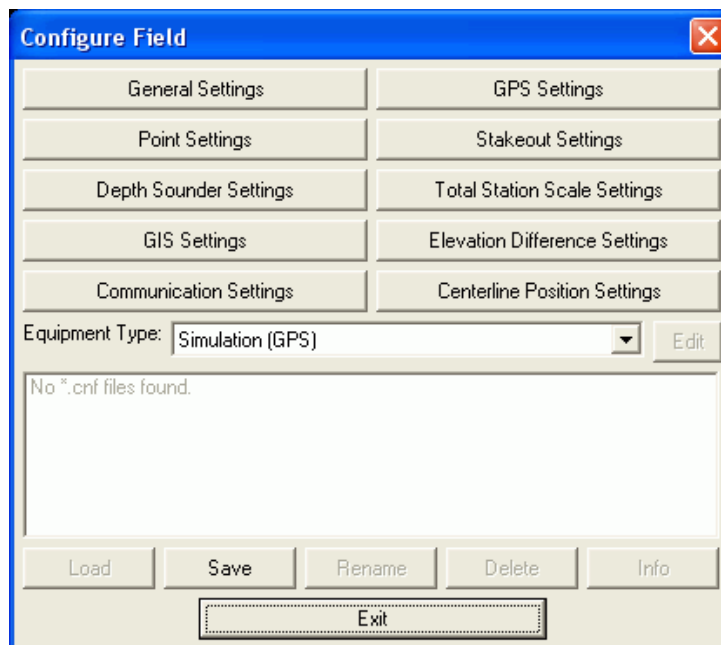
Field Menu

The Field pull-down menu has the main functions for Field including equipment setup, storing points and stakeout.



Configure Field

This command sets the equipment type, communication parameters and other Carlson Field options. Make sure the *Equipment Type* box shows the correct GPS or Total Station equipment that you'll be using. The down triangle button to the right of this box brings up a list of the equipment types to choose from. There are ten *Settings* buttons to bring up the dialog boxes which are used to change Carlson Field's default settings. Explanations for each are shown below.



General Settings

If you are using a total station, *Rod Height* is the distance from the prism to the ground. For GPS, *Rod Height* is the distance from the center of the GPS antenna to the ground.

The *Show Carlson Field Startup Icon* controls whether the Carlson Field Startup Icon is displayed in the lower right of the screen. This startup icon brings up the Carlson Field function menu for launching Carlson Field commands without having to pick them from the pull-down menu.

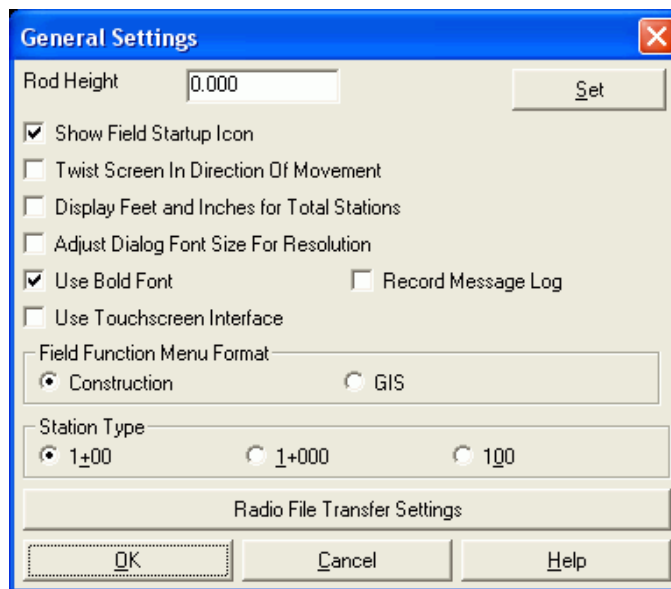
The *Twist Screen In Direction Of Movement* will rotate the drawing view so that your current direction of movement is facing straight up in the view. This rotate is for the view only and does not change the coordinates. This option only applies to GPS and robotic total stations in commands that show the arrow icon such as Track Position.

The *Display Feet and Inches for Total Stations* will use a feet and inches display format (1'2 3/4") in most routines when using a total station.

The *Adjust Dialog Font Size For Resolution* will attempt to adjust some dialogs to better fit the resolution you are using.

The *Use Bold Font* toggles between using standard or bold font for the Carlson Field dialogs.

The *Station Type* chooses the format of centerline station labels. Typically 1+00 is used for feet units, 1+000 is used for metric and 100 has no plus symbol in the number.

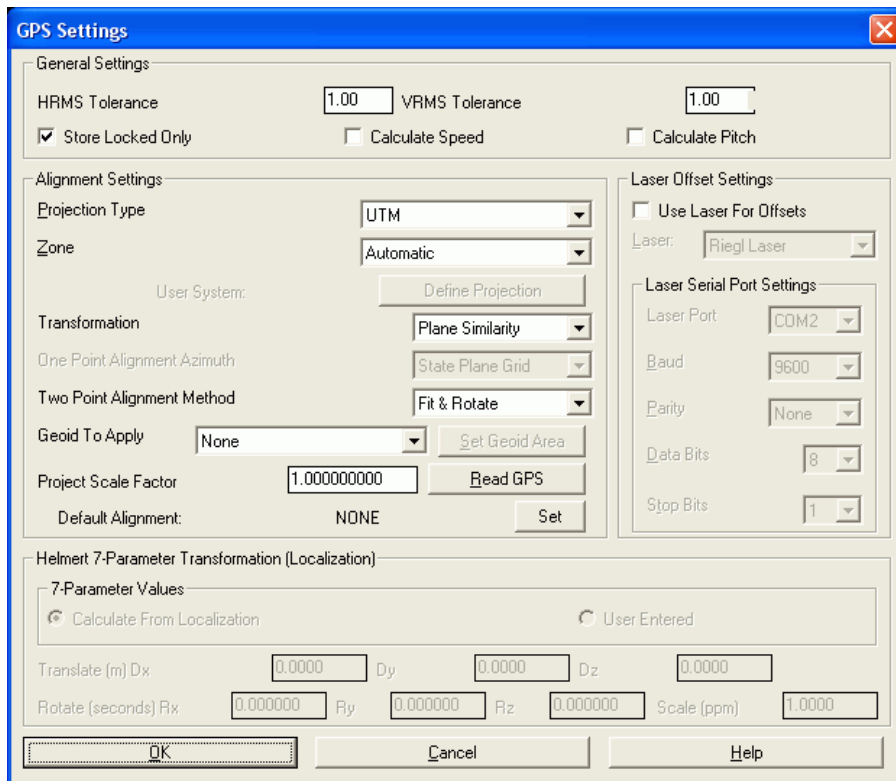


GPS Settings

The *RMS Tolerance* checks the RMS values when reading GPS positions. The RMS is the accuracy value reported by the GPS receiver. There are separate settings for the horizontal and vertical RMS values. The RMS (root mean square) value means that the reported coordinate is within +/- the RMS value of the true coordinate to a certain confidence level. The confidence level depends on the GPS receiver. Typically it is a 98% confidence. If either RMS value exceeds the user-defined tolerance while storing points, Carlson Field will default to "No" when it asks if you want to store the point. You are required to choose yes to override the tolerance check and store the point.

Suggestion: When GPS RTK systems loose lock and go "Float", both the horizontal and vertical RMS values typically jump up to sub-meter (1' or higher) values. In Carlson Field, one foot is the default for the *GPS RMS Tolerance*. Some operators set the *GPS RMS Tolerance* low to 0.2 to check for high RMS values while still "Fixed".

Store Locked Only - The position of the GPS rover is considered either "Autonomous", "Float" or "Fixed" based on the solution status from the GPS base corrections. When you are storing points and the *Store Locked Only* box is checked, Carlson Field will only store points if your position is "Fixed". We suggest you leave this box checked. It ensures that you do not record inaccurate points.



Suggestion: When walking in light to heavy canopy, the rover might remain "Float" and display RMS accuracies of over a foot, sub-meter or more. Setting your *GPS RMS Tolerance* high and turning off *Store Fixed Only* will allow storing wetland and LOD (limits of disturbance) points under canopy that require only sub-meter tolerances. (USCG beacon DGPS sub-meter RTK GPS will always use these settings.)

Projection Type - defines the datum coordinate system to be used for converting the latitude/longitude from the GPS receiver into cartesian coordinates. For the United States two separate horizontal control systems have been developed by the Federal Government: State Plane 1927 and State Plane 1983. For international use the UTM (Universal Transverse and Mercator System) should be selected. The Lat/Lon option will convert the latitude/longitude from degrees minutes seconds format into decimal degrees. This option is useful when working in a decimal degrees lat/lon coordinate system.

Zone - For State Plane projections, you must select the correct state zone that you are working in. For UTM, the Automatic Zone option will have the program automatically use the correct UTM zone for your location. Otherwise for UTM, you can manually set a specific UTM zone. This manual option applies to working on the border between zones and you want to force the program to always use one of those zones.

Important: Coordinates of surveyed points will be inaccurate if the Projection Type and Zone settings are wrong. If you have done survey work and then realize that they are set wrong, then your point coordinates are wrong, but your work is not wasted. Carlson Field records the latitude, longitude and height of every point in a *.RW5 file. You can input the correct projection zone settings later and reprocess your data using the Edit-Process Raw File command.

Model - For UTM, this option sets the ellipsoid constants for converting the lat/lon to UTM coordinates. The following is a list of the models:

Model Earth Radius(m) Flattening factor

Airy 1830 6377563.396 0.00334085064038

Modified Airy 6377340.189 0.00334085064038

Bessel 1841 6377397.155 0.00334277318217

Clarke 1866 ellipsoid 6378206.4 0.00339007530409

Clarke 1880 6378249.145 0.00340756137870

Everest(EA-India 1830) 6377276.345 0.00332444929666

Everest(EB - Brunei & E.Malaysia) 6377298.556 0.00332444929666

Everest(ED - W.Malaysia & Singapore) 6377304.063 0.00332444929666

International 1924 6378388.0 0.00336700336700
Helmert 1906 6378200.0 0.00335232986926
Hough 1960 6378270.0 0.00336700336700
Geodetic Reference System 1980 6378137.0 0.00335281068118
South American 1969 and Australian National 6378160.0 0.00335289186924
World Geodetic System 1972 6378135.0 0.00335277945417
World Geodetic System 1984 6378137.0 0.00335281066475

Transformation - The transformation in the Align Local Coordinates command can either be by plane similarity or rigid body methods. Both methods use a best-fit least squares transformation. The difference is that the rigid body method does a transformation with a translation and rotation and without a scale. The plane similarity does a rotation, translation and scale. This option only applies when two or more points are used in Align Local Coordinates.

One Pt Align Azimuth - This option applies to the rotation when using one point in Align Local Coordinates. For this alignment method, the state plane coordinate is translated to the local coordinate. Then the rotation can use either the state plane grid or the geodetic as north. No scale is applied in this transformation. The state plane and geodetic true north diverge slightly in the east and west edges of the state plane zone. This option allows you to choose which north to use.

Two Point Align Method - This option applies only two point alignments. Possible values are *Fit & Rotate* and *Rotate Only*. *Fit & Rotate* (the default) will use the second alignment point for rotation, translation, and scale (depending on the value set for *Transformation*). The *Rotate Only* option will use the second point of a two point alignment for rotation only.

Geoid To Apply - This option will account for the geoid undulation in determining the orthometric elevation of the measurement. The definition of the geoid model as currently adopted by the National Geodetic Survey is the equipotential surface of the Earth's gravity field which best fits, in a least squares sense, global mean sea level. Orthometric elevation measurements are used in survey calculations. In order to convert ellipsoid heights (H_e) as measured by GPS into orthometric elevations (E_o), you must provide for a correction between the GPS-measured ellipsoid (reference ellipsoid) and a constant level gravitational surface, the geoid. This correction is the geoid undulation (U_g). The formula is $H_e = E_o + U_g$.

The Geoid models are essentially large elevation difference models in grid format. Carlson Field has two geoid models available. Geoid99 covers the United States at 1 minute grid intervals. EGM96 covers the entire globe at 15 minute intervals. These Geoid models are huge and take a lot of disk space and memory. The Geoid model files are not installed automatically and instead need to be installed by going to the Geoid folder on the Carlson Field installation CD. Once installed onto Carlson Field, you then need to specify your location by lat/lon so that the program only needs to load a local portion of the Geoid model. To set your local Geoid area, pick the *Set Geoid Area* button. Setting the Geoid area will carve out a Geoid model around the specified lat/lon covering a square area of 2 degrees by 2 degrees which is about 100 miles by 100 miles.

Carlson Field applies the Geoid model by subtracting the Geoid undulation from the GPS elevation.

The resulting elevation is then used and displayed. In the Monitor function, the Geoid undulation is displayed.

In practice, the Geoid model is most applicable to two types of alignment scenarios. One of these types is when setting up the base over a known point and having no alignment control points. The other is when there is one alignment control point. When using multiple alignment control points, the Geoid model is not as important because Carlson Field can model the elevation difference which can generally pick up the local Geoid undulation.

Project Scale Factor - After converting the LAT/LONG from the GPS to the state plane coordinates and applying the Align Local Coordinates, the Project Scale Factor is applied as the final adjustment to the coordinates. This adjustment is used on the X,Y and not the Z. The Project Scale Factor is applied by dividing the distance between the coordinate and a base point by the Project Scale Factor. The coordinate is then set by starting from the base point and moving in the direction to the coordinate for the adjusted distance. The base point is the first point in Align Local Coordinates. If there are no points specified in Align Local Coordinates, then 0,0 is used as base point. The Project Scale Factor can be entered directly or calculated using the grid factor and elevation for the current position. When using the current position, the program will read the LAT/LONG from the GPS receiver. The scale

factor is then calculated as: (State Plane Grid Factor - (Elevation/Earth Radius)).

Default Alignment - This option sets the alignment file to use by default for new drawings. This feature applies when you will keep working at the same site with the same base receiver setup.

Helmert 7-Parameter Transformation - These settings apply when the Transformation is set to Helmert. The Helmert 7-parameters can either be calculated by the program using the control points in the localization or user-entered.

Laser Offset Settings - There is an option to use a laser for reading the distance and angle for offset points. When this option is enabled, you can choose the laser equipment type and communication parameters. The serial port for the laser must be different than the GPS which requires at least two serial ports on the computer. When using a laser for offsets, the program will read the current position from the GPS and then read the laser for the distance and angle to the point. This combination allows you to calculate points that cannot be directly reached by the GPS. There are two methods in the Point Store command to use the laser when this option is enabled. The Point Store dialog will have a new Laser button which will bring up another dialog that allows you to take multiple shots from the laser. The other method is to click on the Offset toggle in the Point Store dialog. Then when you do the Read function, the program will read the GPS position and then pop-up a dialog for taking one offset shot.

Point Settings

Beep for Store Point - This option will make a triple beep to indicate when a point is stored in the coordinate file.


User-Entered Point Notes - Point Notes are additional descriptions that can be stored with a point. A regular point consists of a point number, northing, easting, elevation and 32 character description. These points are stored in a .CRD file. Point Notes are a way to add an unlimited number of lines of text to a point number. With Point Notes ON in the Store Point command, the program will prompt for notes after collecting a point. The notes are stored in a file that uses the name of the coordinate file with a .NOT extension. For example, a coordinate file called JOB5.CRD would have a note file called JOB5.NOT.

Point Settings

Store Options

- Beep for Store Point
- User-Entered Point Notes
- Coordinates in Point Notes
- GPS RMS in Point Notes
- GPS DOPs in Point Notes
- Speed in Point Notes
- Rod Height in Point Notes
- Project Scaler in Point Notes
- Time/Date in Point Notes
- Florida DOT Feature Notes

Drawing Options

Symbol: 

- Prompt For Symbol Rotation
- Label Elevations
- Label Descriptions
- Locate on Real Z Axis
- Layer:

Point Averaging

Number of Readings:

Direct-Reverse Horiz Angle Tolerance (Seconds):

Direct-Reverse Vert Angle Tolerance (Seconds):

Direct-Reverse Distance Tolerance:

Auto Points At Interval

- Stop Linework When GPS Loses Lock Status

Store Point

- Start In Continuous For GPS

Field to Finish

Use Code Table For: Symbols Layers Descriptions Linework Real Z

- Split Multiple Codes
- Check Descriptions With Code Table

Code File:

Coordinates in Point Notes - When storing a point, this option will store the point number, northing, easting, elevation and description in the point notes as well. This may be used as a backup or reference to coordinate data as it was originally stored.

GPS RMS in Point Notes - When storing a point, this option will store the horizontal and vertical RMS values in the note field for the point. This offers a good check on the quality of the shot.

GPS DOPs in Point Notes - When storing a point, this option will store the DOP (dilution of precision) values as reported from the GPS receiver.

Rod Height in Point Notes - When storing a point, this option will store the rod height value in the note field for the point.

Project Scaler in Point Notes - When storing a point, this option will store the project scale factor in the note field for the point.

Time/Date in Point Notes - This option will store the time and date that the point was stored in the note file. Carlson Field will read the time from the computer.

Drawing Options control how points are drawn by default. It controls the layer, symbol number and whether points will be drawn with descriptions and elevations. Carlson Field's Field to Finish code table can override these defaults.

The symbol used for default points is displayed. You can choose another symbol by changing the *Symbol* name or by selecting one from the table that the *Select Symbol* button brings up. Default point settings are used for points whose descriptions don't correspond to any category on the Field to Finish code table.

Label Descriptions and *Label Elevations* Control whether these two items of information appear on your drawing next to each point.

Locate on Real Z Axis will record points with their true elevations. If this setting is off, all points recorded will have an elevation of zero.

Layer for Points indicates the layer where all default points will be drawn. For points using a code on the code table, the code table will determine their layer.

Number of Readings specifies how many times Carlson Field will read from the instrument in the Read function of the *Point Store* command. This applies to both GPS and total stations. The readings will be averaged to find a more accurate position.

Direct-Reverse Tolerances are used with total stations to check the pairs of direct and reverse horizontal angles, vertical angles and distances. When these values are off by more than the tolerance, the program will display a warning.

Field to Finish is explained fully in the Field to Finish command definition. Basically it uses a code table which holds information on types of points (ie. Man Hole or Edge of Pavement). When the settings *Use Code Table* is selected, Carlson Field will look to the code table for how to draw points of a particular code description.

The file containing the active code table appears after *Code File*: You can change this with the button *Select File*.

The *Split Multiple Codes* option will draw multiple points from the same point when that point description has multiple codes. For example, a point with description "EP DR" will draw the point twice: once with the properties of code EP and a second time using code DR. When this option is off, the program will use the first code and draw the point once.

The *Check Descriptions With Code Table* option will display a warning before storing a point if that point description is not found in the code table. With this option off, the program will go ahead and store the point and the point will be drawn using the default point properties.

Stakeout Settings

Display GPS RMS in Stakeout causes Carlson Field to report the constantly updating horizontal RMS accuracy values while staking a point. The only disadvantage to having this option active is that it slows down a little the stakeout position update.

Draw Trail displays a line in the stakeout screen showing where you've been as you move towards the stakeout point. This option only applies to GPS.

Prompt For Snap On Screen Pick controls whether you are prompted to select an object snap when picking points from the screen during stakeout.

Auto Zoom will zoom the drawing display in or out so that both your current position and stakeout target are visible on the screen.

Zero Horizontal Angle To Target will set the horizontal angle of the total station to zero in the direction towards the stakeout point. When stakeout is completed, the horizontal angle is set back to the original value. This option only applies to total stations.

Automatic Turn To Point Type For Robotic Total Stations does two things. First, it controls whether or not stakeout automatically turns to the stakeout point. Second, it allows the user to select what type of turn is performed: Horizontal Angle only, or Horizontal and Vertical Angle.

Default Stakeout Mode allows the user to select a default stakeout mode to be used when entering the stakeout routine. Choices include the stakeout modes: Station-Offset, Point Number, and Pick Point. If Pick Point is selected as the default stakeout mode, the user can also define the default object snaps.

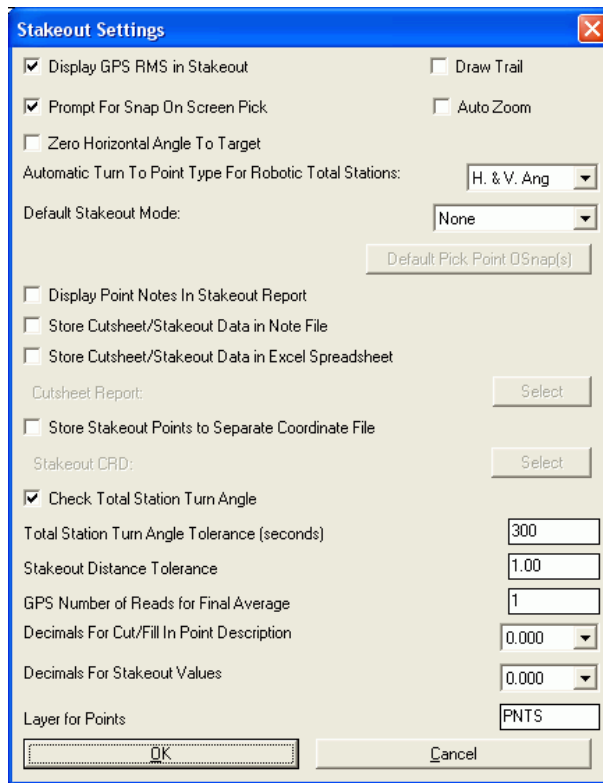
Store Cutsheet/Stakeout Data in Note File will store stakeout data in the note file (.NOT) for the current coordinate file. At the end of staking out a point, there is an option to store the staked coordinates in the current coordinate file. This stakeout note file option allows you to store more stakeout data in addition to the staked coordinates. This additional data includes the target coordinates and horizontal and vertical difference between the staked and target points. This stakeout note data can be used in reports with the List Points or CutSheet Report commands.

Store Cutsheet/Stakeout Data in Excel Spreadsheet will display a cutsheet report in an Excel spreadsheet. The spreadsheet will pop-up at the end of each point stakeout. The report can be saved in Excel format and processed by Excel.

Store Stakeout Points To Separate Coordinate File will store the staked points to a different coordinate file besides the current coordinate file. This allows you to use the same point number for the target and staked points. The staked point coordinate file can be specified by picking the Select Coordinate File button.

Check Total Station Turn Angle will compare the angle from the instrument and the angle to the target point. If this difference is greater than the specified tolerance, then Carlson Field will display a warning message.

Stakeout Tolerance controls the maximum difference between the target location and actual staked point. When the staked point is beyond the tolerance, Carlson Field displays a warning dialog.



GPS Number of Reads for Final Average specifies how many times Carlson Field will read the GPS receiver position for the final staked point. These readings are averaged. Averaging several readings while occupying one point yields a more accurate result, but inevitably takes longer.

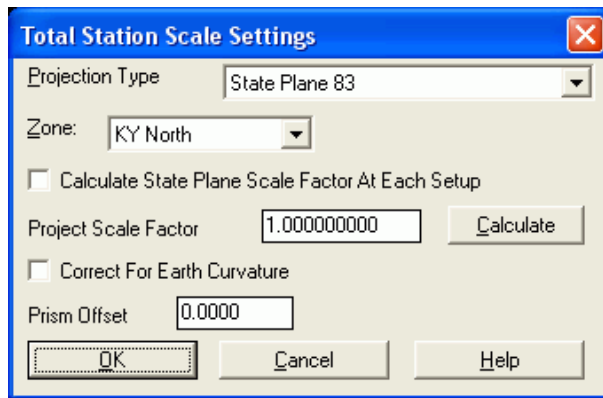
Total Station Scale Settings

These settings apply only to total stations. The *Project Scale Factor* is multiplied by the measured distance from the total station when calculating the foresight point coordinates. A typical project scale factor for working in state plane coordinates is slightly less than one. Factors greater than 2.0 or less than 0.5 are not allowed. The Project Scale Factor can be entered directly or choose the Calculate button. The Calculate function takes a state plane coordinate and calculates the project scale factor as the state plane grid factor minus the elevation factor (Grid Factor - elevation/earth radius). The state plane coordinate is specified by a point number from the current coordinate file.

The *Calculate State Plane Scale Factor At Each Setup* option will calculate the scale factor for each shot as the combined grid and elevation factors (see above equation). The scale factor is calculated at both the occupied and foresight points and then averaged. To use this option, you must be working in state plane coordinates and set the state plane zone in this dialog.

The *Correct For Earth Curvature* option adjusts the horizontal distance and vertical difference to the foresight point to account for the earth curvature.

Prism Offset is for use with total stations to account for the offset (in mm) of the prism in use. It is recommended to keep this at zero, and set the prism offset in the instrument.



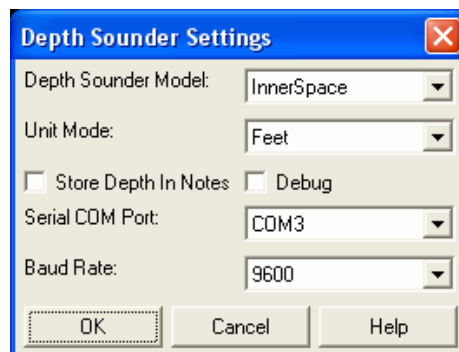
Depth Sounder Settings

Carlson Field can use depth sounders in combination with GPS to collect points of underwater surfaces. Carlson Field supports depth sounders that output standard NMEA data. There are several models to choose from: Hydrotrac, Horizon, Odom Digitrace, InnerSpace, and Generic. For the Odom Digitrace and InnerSpace models, you also need to specify the depth unit mode that the instrument is set to.

The *Store Depth In Notes* option will record the water depth in the current note file (.NOT) when a point is stored to the coordinate file.

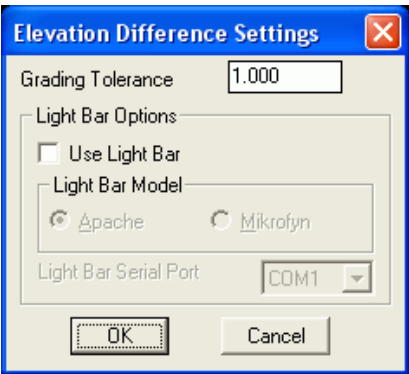
The *Debug* toggle can be used when contacting technical support to diagnose communications issues between the depth sounder and Carlson Field.

The depth sounder must be connected to a separate serial port than the GPS. The *Baud Rate* between the computer serial port and the depth sounder is also specified here.



Elevation Difference Settings

These settings apply to the Elevation Difference command. Grading Tolerance is the target difference between the actual elevation and the design surface. Carlson Field can use an external Light Bar to indicate whether your current position is in cut, fill or on-grade. Currently Carlson Field supports light bars made by Apache and Mikrofyn. The Light Bar must be connected to a separate serial port than the GPS.

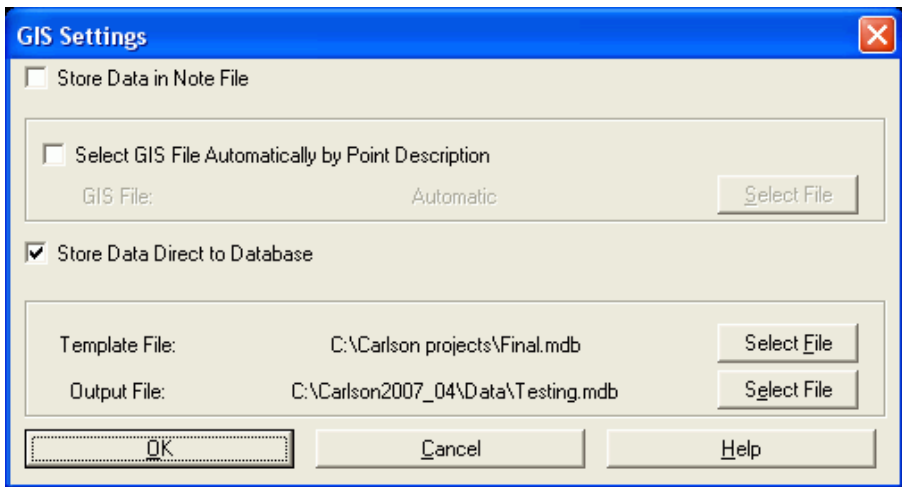


GIS Settings

A standard point is stored in the coordinate file with a maximum 32 character description. The GIS Settings allow you to store more data with each point.

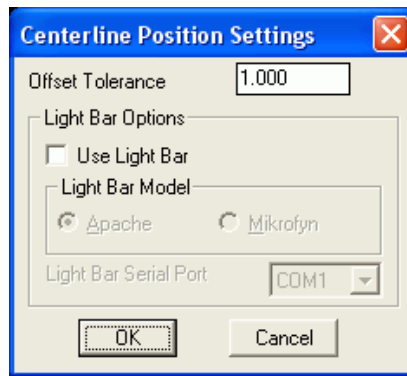
The *Store Data In Note File* option will record additional fields for each point in the note file. The note file has the same name as the current coordinate file except with a .NOT instead of .CRD file extension. The fields that are recorded are defined by the GIS File (.GIS). This file defines a sequence of field names and prompts. For example, a GIS file for manholes could contain Location, Depth and Condition fields. Choose the Select File button to choose the GIS file to use. Or use the *Select GIS File Automatically by Point Description* to use different GIS files depending on the point description. With this option, the program will look for a GIS file with the same name as the point description. For example, if the point description is MH, then the GIS file will be MH.GIS. See the Define Note File Prompts command for more information.

The *Store Data Direct To Database* option will store additional fields for each point in a Microsoft Access database. The database to store the data is set in the Output File line. The Template File is a database that defines the fields to record. See the Define Template Database command for more information.



Centerline Position Settings

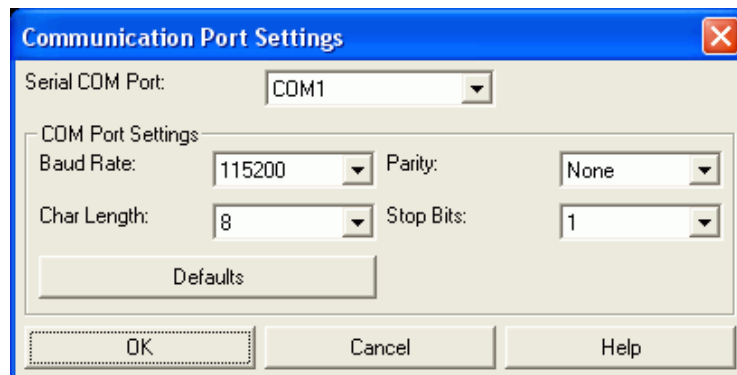
Similar to Elevation Difference Settings only for the Centerline Position command.



Communication Port Settings

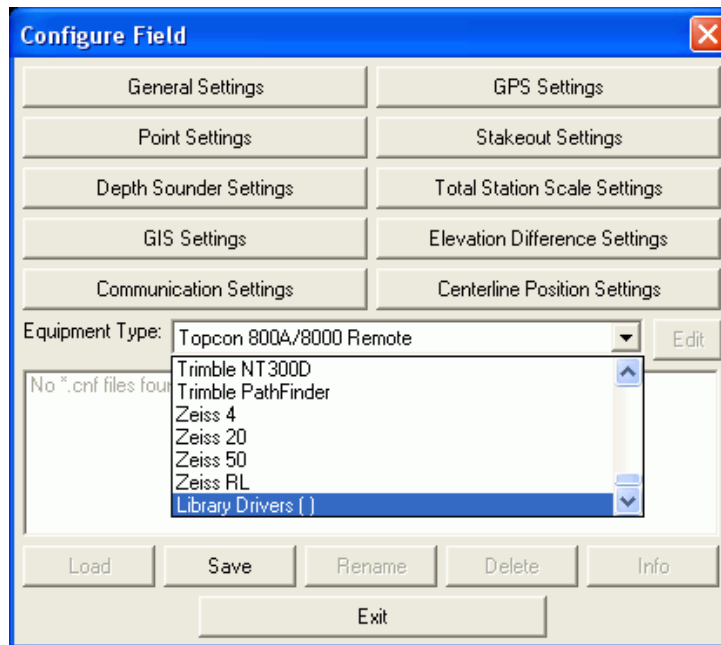
Serial COM Port - The GPS receiver or total station attaches to your Carlson Field computer using a serial cable. This cable is plugged into a serial COM port on your computer called 1, 2, 3 or 4 . Check the circle denoting the COM Port to be used.

The *Baud Rate*, *Parity*, *Char Length* and *Stop Bits* are the serial port communication parameters for the Carlson Field computer. These parameters need to match the parameters on the instrument that you are using. The Defaults button will set these communication parameters to the standard parameters for the current equipment type.

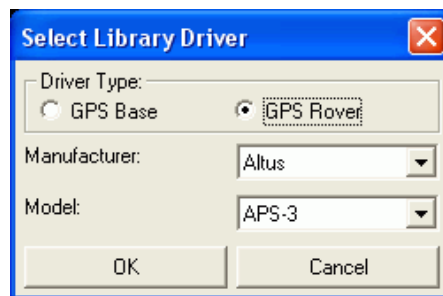


Equipment Type

The *Equipment Type* droplist allows the user to select the instrument driver to use. The very last entry is for "Library Drivers" which are instrument drivers that follow a new Carlson architecture and support many advanced features (ex. NTRIP). These "Library Drivers" are different from the legacy Carlson Field instrument drivers; they are shared with other Carlson products (ie. SurvCE and Carlson Grade). To select a "Library Driver", select that item and press the *Edit* button. Please note, that the *Edit* button is only active for "Library Drivers".

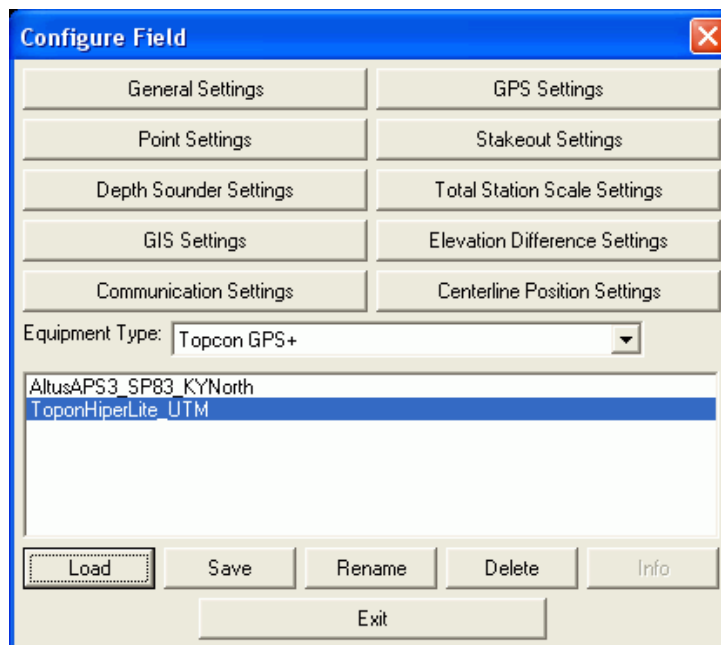


Selection of a "Library Driver" is split into two types: *Base* and *Rover*. After selecting the *Driver Type*, select the proper manufacturer and model to match the equipment being used.



Carlson Field also allows the user to save and recall equipment configurations. If you use the same equipment in different configurations, saving the configuration is an easy way to recall it later without missing a setting. First, select the *Equipment Type* you wish to save a configuration for. Next, make sure all the instrument settings are as desired by running through *Equipment Setup*. Finally, press the *Save* button and enter a description configuration name that will help remind you what the configuration is used for. Keep in mind, saving a configuration will also save the current *GPS Settings* (ie. projection type, etc).

To recall a saved configuration, just select the configuration and press the *Load* button.



Equipment Setup

Function with Total Stations

Selecting the Equipment Setup command will send the user directly to a settings window that corresponds with the instrument selected in Configure Field. Equipment Setup for total stations will be discussed first, followed by GPS equipment.

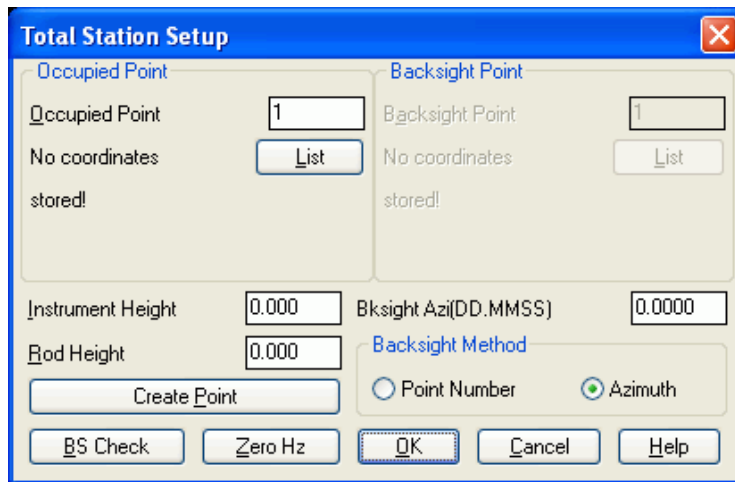
This function for Total Stations lets you tell Carlson Field how you have positioned your total station. The setup information in this command is required before taking shots. Besides running this command from the Field pull-down menu, you can also reach this command with the Setup(F3) button from many of the other Carlson Field functions.

Occupied Point refers to the point your total station is setup on. This point is defined by a point number that references the current coordinate file. The coordinates and description of this point are displayed below the point number. The *List* button will bring up a list of the points in the coordinate file which you can review or select from. If the coordinates for the occupied point are not yet in the coordinate file, then you can pick the *Create Point* button to enter these coordinates.

The backsight can reference either a point or an azimuth. *Backsight Point* is only used if *Point Number* is selected as your *Backsight Method*. If you want to use an azimuth instead of a backsight point, select the *Azimuth* toggle and specify the azimuth in the *Bksight Azi* box.

Set the *Instrument Height* and *Rod Height*. These values will use whatever units your drawing uses: feet or meters.

Carlson Field expects the instrument to have the horizontal angle zeroed on the backsight. Part of the station setup procedure needs to include zeroing the instrument on the backsight. To do this, first specify the occupy point and backsight in this dialog. Then orient the instrument to the backsight and pick *Zero Hz* to zero the gun.

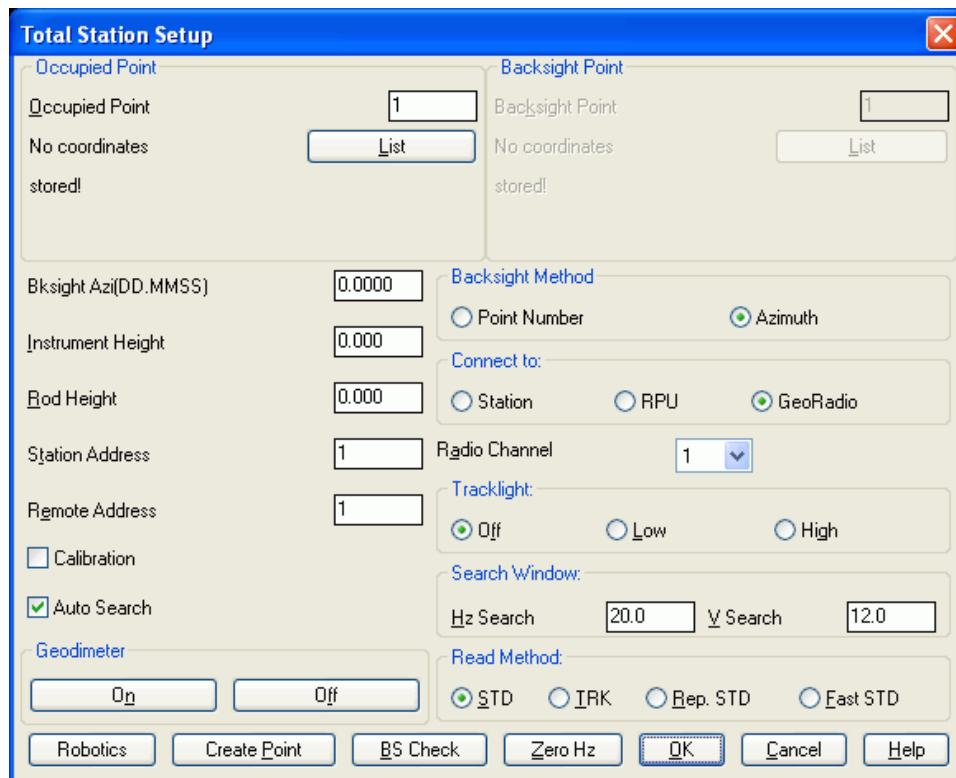


The *BS Check* button runs a backsight check. The program will take a shot and compare the calculated point to the expected backsight point and report the results to you. This will help you establish if the point you are using as the backsight point is really the point that you think it is. For some robotic total stations, the Backsight Check routine has an option to automatically turn the instrument to the backsight. Then after the check is done, the instrument can be automatically turned back to the previous direction. The purpose of this auto turn is to speed up the steps to check the backsight in the middle of surveying points in a different direction.

For some types of total stations, the Total Station Setup dialog will also contain different options that are specific to that type of total station.

Geodimeter Total Station Setup

The three methods of connecting to the Geodimeter include: *Station*, *RPU* and *GeoRadio*. The *Station* option is for connecting directly to the instrument. The *RPU* is a remote control panel. The *GeoRadio* is a radio for remote control of the instrument. For the *GeoRadio*, the *Station Address* and *Remote Address* set the radio addresses and the *Radio Channel* sets the radio channel.



The intensity of the instrument Tracklight can be set to Off, Low or High.

The Geodimeter *On* and *Off* buttons are for putting the instrument in sleep mode to save power.

There are four different read methods. STD mode has a 3.5 second measurement time for each point. It is usually used when a normal degree of angle and distance accuracy is required. TRK mode uses automatic, measured values that are updated 0.4 seconds after making a contact with the prism. Rep STD mode measures distance automatically every 4 seconds. Fast STD mode measures distance in 1.3 seconds. It is used when the demands on precision are low.

Leica Total Station Setup

The *Connection Mode* chooses between connecting Carlson Field directly to the instrument or to a radio for remote control.

The *EDM Mode* sets the instrument distance measurement mode for standard shots. All the possible modes are listed in this dialog including tracking and reflectorless. Be sure to choose a mode that is supported by your instrument. When using the reflectorless mode, the Rod Height should typically be set to zero. When tracking is selected in Carlson Field functions, the program will automatically put the instrument in IR Rapid Tracking mode during tracking and then return to the specified EDM Mode when tracking is done.

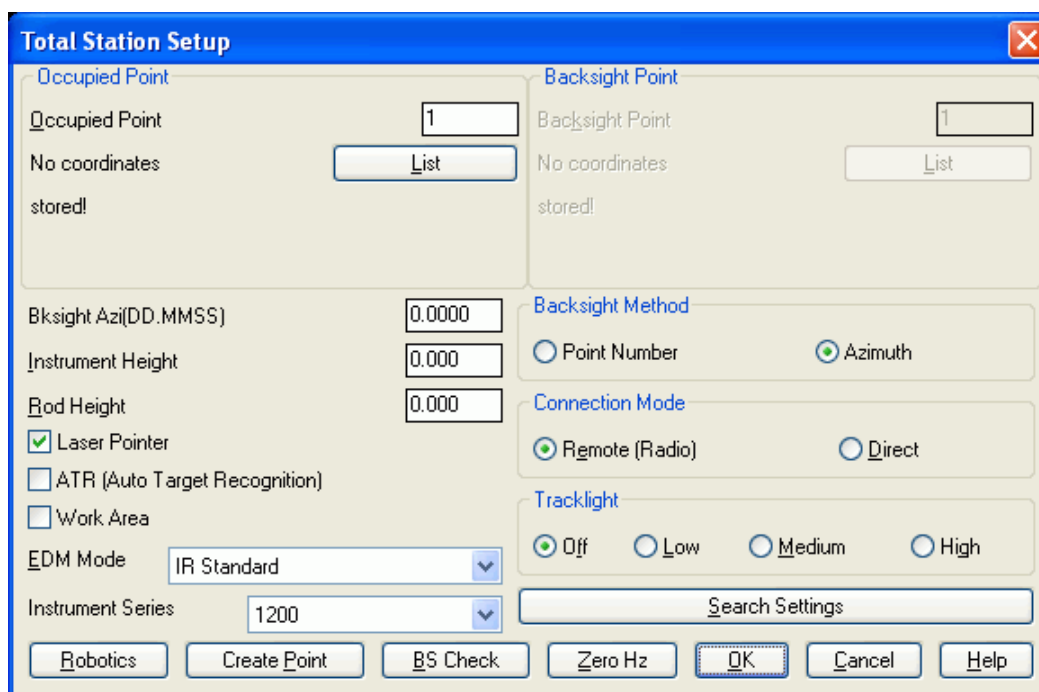
The intensity of the instrument *Tracklight* can be set to Off, Low, Medium or High.

Laser Pointer allows the user to toggle the instrument's laser pointer on and off, if the instrument is so equipped.

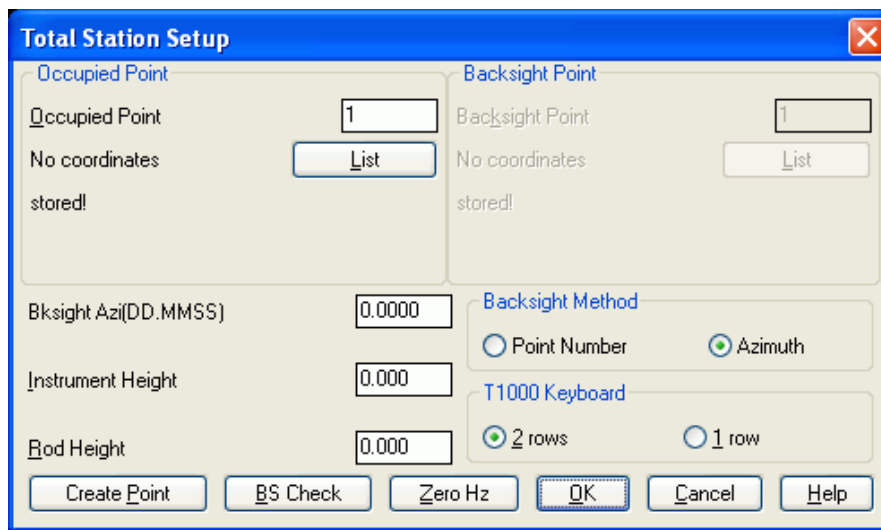
ATR (Auto Target Recognition) allows the user to choose whether or not to use the instrument's ATR system, if the instrument is so equipped. This option is not available when using a reflectorless EDM mode.

Use *Instrument Series* to select the proper Leica TPS model you are using. There are only two choices: *1200* (which uses the GeoCOM protocol) or *100/300/400/700/800/1000/1100* (which uses the GSI protocol).

The *Robotics* button can be used to access the joystick screen, where the user can access convenient controls for robotic total stations.

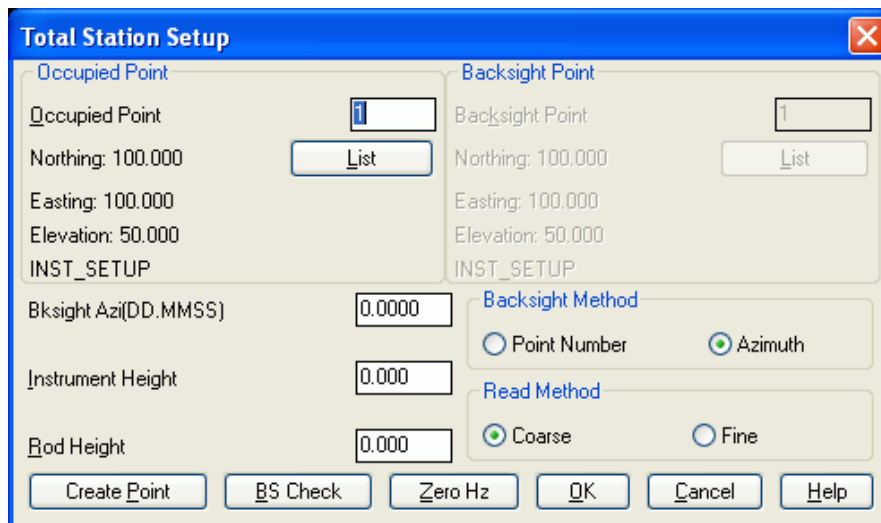


The older Leica total station driver allows the user to select what type of keyboard emulation to use, 1 or 2 row.



Topcon 800A/8000 Direct Total Station Setup

The *Read Method* sets the instrument distance measurement mode for standard shots. When tracking is selected in Carlson Field functions, the program will automatically put the instrument in Coarse mode during tracking and then return to the specified EDM Mode when tracking is done.



Topcon 800A/8000 Remote Total Station Setup

The *Radio Type* can either be Satel 3AS, Satel 2AS, RC-2, Stream, or Other. With Other, Carlson Field does not send any radio setup commands. So these radios must be configured before running Carlson Field. For the Satel 3AS radios, you can set the radio frequency by Channel ID or by manually typing a frequency between 468.5 and 470.5. Stream can be used in any situation where Other would be used. However, Stream is different in that the instrument streams measurements to Carlson Field thereby decreasing the probability of a failed reading. It should be noted, if Stream is used over a very poor radio connection, it is possible for Carlson Field to get out of sync with the instrument and fail to stop streaming when needed.

The *EDM Mode* sets the instrument distance measurement mode for standard shots. When tracking is selected in Carlson Field functions, the program will automatically put the instrument in Coarse mode during tracking and then return to the specified EDM Mode when tracking is done.

Wait Time is switched on when the instrument cannot track a prism due to an obstruction. If after the wait time have elapsed the instrument does not switch back to tracking mode, then searching mode is set.

Vertical range and Horizontal range set the search area. Vertical range can be anywhere from 0-90 degrees, and horizontal range can be anywhere from 0-180 degrees.

Track Indicator On if checked turns on the light which is mounted below the telescope.

Joystick Speed sets the instrument turning speed from the arrow keys in Robotic control.

Topcon APL1/APL1A Total Station Setup

The *Radio Type* can either be Satel 3AS, Satel 2AS or Other. With Other, Carlson Field does not send any radio setup commands. So these radios must be configured before running Carlson Field. For the Satel 3AS radios, you can set the radio frequency by Channel ID or by manually typing a frequency between 468.5 and 470.5.

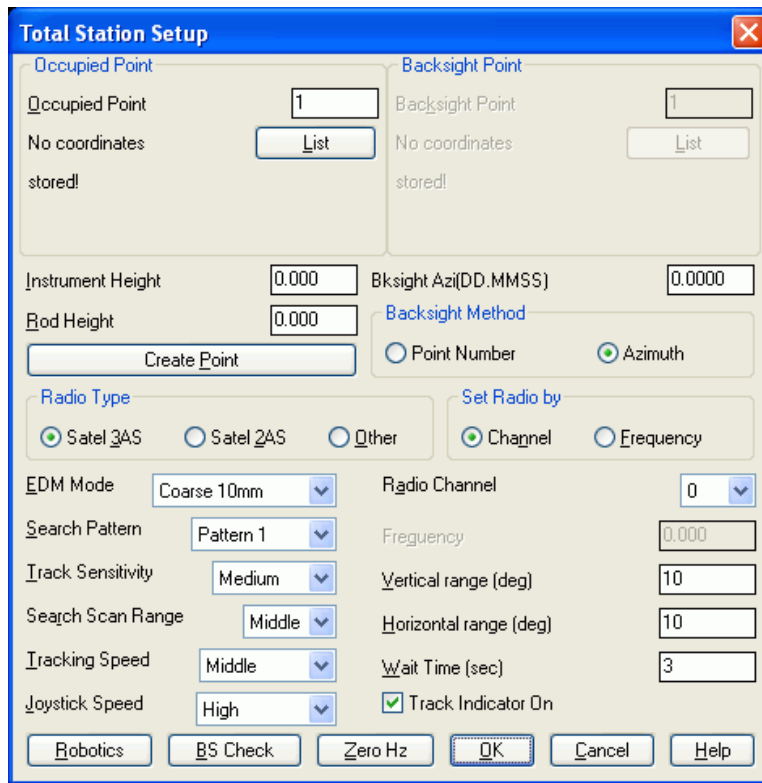
The *EDM Mode* sets the instrument distance measurement mode for standard shots. When tracking is selected in Carlson Field functions, the program will automatically put the instrument in Coarse mode during tracking and then return to the specified EDM Mode when tracking is done.

Wait Time is switched on when the instrument cannot track a prism due to an obstruction. If after the wait time have elapsed the instrument does not switch back to tracking mode, then searching mode is set.

Vertical range and Horizontal range set the search area. Vertical range can be anywhere from 0-90 degrees, and horizontal range can be anywhere from 0-180 degrees.

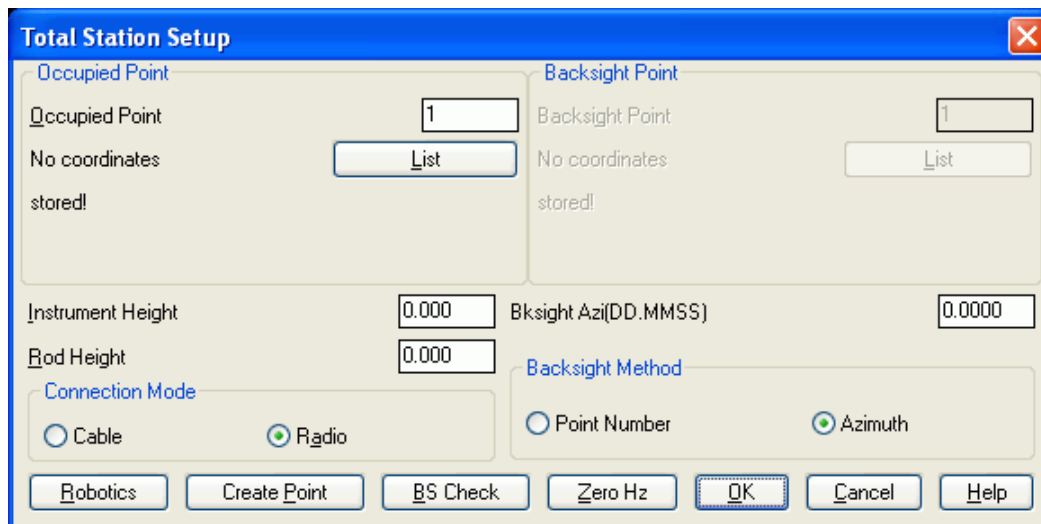
Track Indicator On if checked turns on the light which is mounted below the telescope.

Joystick Speed sets the instrument turning speed from the arrow keys in Robotic control.



Zeiss20 Total Station Setup

The *Connection Mode* chooses between connecting Carlson Field directly to the instrument (cable) or to a radio for remote control.



Equipment Setup with GPS

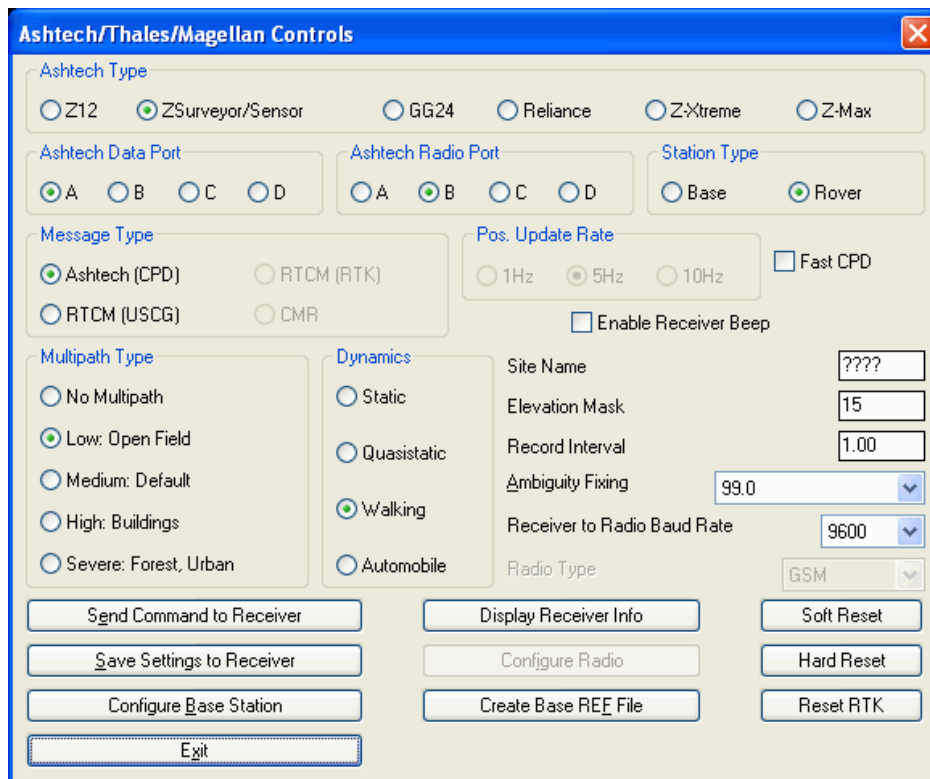
Carlson Field works with the following RTK GPS manufacturers: Ashtech, Javad, Leica, Novatel, Sokkia and Trimble. Each RTK GPS brand has its own GPS Setup control window. To get the window which matches the GPS equipment you are using, go to Configure Field and under the Equipment Type pulldown menu select the correct equipment. A brief explanation is given below for each brand's controls.

For RTK (real-time kinematic) GPS work, the base sends GPS corrections to the rover. To setup a base receiver, you should attach the computer running Carlson Field to the base receiver and run the Equipment Setup. After this is done and the base is outputting corrections, you should detach the base receiver and attach the rover receiver and do Equipment Setup again.

If your base radio has a TX light, it should be flashing while it's sending out corrections. This is a convenient way to tell if the base is configured.

Ashtech GPS Setup

The *Ashtech Type* specifies the model of Ashtech equipment to be used. Carlson Field works with the following Ashtech high precision, centimeter accurate RTK GPS equipment: Z12, Z-Surveyor/Sensor, GG24, Z-Extreme, and Z-Max. Carlson Field also works with the Ashtech Reliance USCG/DGPS RTCM sub-meter RTK GPS receivers.



The previous Ashtech Control settings are default. Changing these settings will change the internal settings of the Ashtech receiver.

Ashtech Data Port is the port on the GPS receiver where the Carlson Field computer is connected, usually Port A.

Ashtech Radio Port is the port on the receiver where the radio modem is connected, usually Port B. For the Z-Extreme, Port D is usually used for the radio port.

Message Type for high precision centimeter RTK GPS set message type to Ashtech (CPD). If you are using the USCG/RTCM DGPS message type for sub-meter accuracy then set the message type to RTCM (USCG).

Multipath Type is used to filter out interference in the satellite signals caused by nearby objects. The choices are No Multipath, Low: Open Field, Medium: Default, High: Building and Severe: Forest, Urban.

Dynamics settings are Static, Quasistatic, Walking and Automobile. Static is selected only when the Rover receiver is stationary. The default is Walking.

Elevation Mask is the cutoff vertical angle above the horizon. Any satellites below this angle will be left out of calculations.

Site Name and *Record Interval* are all setting for post processing use only, not for use with RTK GPS. Site Name is the Point ID name for post processing. Record Interval is the epoch interval to record post processing information. RTK GPS updates every second but post processing epochs are usually 5, 10, 15, 20 or 30 second intervals.

Ambiguity Fixing Parameter (90 - 99.9): controls the confidence level of fixed positions. The default is 99.0. At a lower confidence interval the system solves much faster. If the system incorrectly solves the position, then the position error will be much greater than the reported RMS value.

Position Update Rate is the frequency that GPS positions are calculated and reported.

Fast CPD is a toggle On or Off. Fast CPD toggled On will allow approximating the rover's position if your position is lost briefly. Off is the default. Fast CPD is generally toggled on when Dynamics is set to Automobile.

When Carlson Field functions start, the program uses the settings specified in Equipment Setup to configure the GPS receiver. The *Save Settings to Receiver* uploads the settings in the Carlson Field dialog to the receiver so that the next time the receiver is turned on these settings are still set even without connecting to Carlson Field. Otherwise, Carlson Field must be connected to the receiver to setup these options.

Send Command to Receiver allows experienced users to type in commands using Ashtech GPS receiver commands to set or report internal settings. (See the Ashtech operations manuals for a complete list of Ashtech GPS receiver commands.)

Reset Sensor Memory will reset the receiver memory, reinitialize the communications ports and reset the modem. Saved settings on the receiver will be returned to their default values.

Radio Baud Rate allows you to change Pacific Crest radio baud settings through the receiver. The default baud rate is 9600. (Note: If there are communication problems with either port A or B on the Ashtech ZSurveyor receiver, turn off receiver and turn it back on with both keys depressed to reset receiver to factory defaults.)

For the Z-Extreme, the *Configure Internal Radio* button allows you to change the radio channel and settings. This function will attempt to establish a connection with the internal radio, reporting an error if it is unable to do so. Otherwise, it will open a dialog which will display the current radio channel as well as the valid range of radio channels. Enter the desired radio channel in the edit box and then click on "Program Radio" to set the changes to the radio. Carlson Field will communicate with the radio for a few seconds, and will then request that you power the receiver down, then turn it on again before continuing. It is very important that this is done, or else Carlson Field will be unable to communicate with the ZExtreme. Also note that if the programming of the radio is canceled for any reason, the receiver will still need to be powered down, then powered up again in order for Carlson Field to be able to communicate with it.

Create Base REF File takes a reading from the GPS receiver and stores this lat/lon to a reference file (.ref) that can be used later in Configure Base Station. The purpose is to allow moving the base station based on the current base setup. In this case, Create Base REF File would be run from the rover receiver while in "fixed" position. Then the base could be moved to this point without having to redo the local coordinate alignment.

Configure Base Station initiates the receiver connected to Carlson Field to be a base and begin broadcasting its stationary position and satellite corrections to the rover. (See Configure Base Station for All GPS Brands at end of this section.)

Topcon GNSS/Javad GPS Setup

Radio Port on the Javad base and rover receiver is usually C. Data Port is always A. When using Pacific Crest radios, Javad recommends the new PDL Pacific Crest radios. These must be set to 38,400 baud rate. Javad also uses Spread Spectrum radios which work at 119,200 baud rate.

Receiver Model selects between *Legacy*, *Odyssey* and *Regency*. Currently the Receiver Model does not affect the Carlson Field interface except to determine the default Antenna Type.

Position Update Rate sets the frequency that the receiver calculates and reports position. The faster rates are an option that must be purchased for the receiver.

Antenna Type chooses between an internal and external GPS antennas. This option applies to receivers with built-in antennas.

The *RTK Message Type* determines the format of the GPS correction message that is used from the base to the rover.

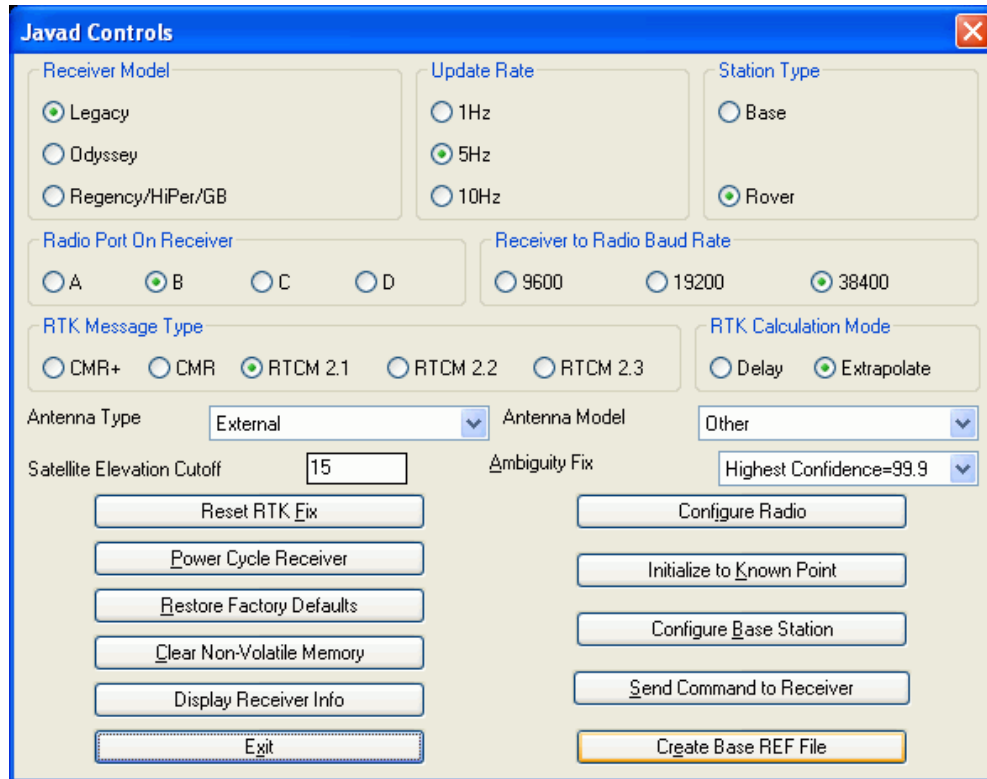
RTK Calculation Mode chooses between Delay and Extrapolate. The Extrapolate mode is needed for fast Position Update Rates.

Satellite Elevation Cutoff is the cutoff vertical angle above the horizon. Any satellites below this angle will be left out of calculations.

Ambiguity Fixing Parameter (95 - 99.9): controls the confidence level of fixed positions. The default is 99.0. At a lower confidence interval the system solves much faster. If the system incorrectly solves the position, then the position error will be much greater than the reported RMS value.

Power Cycle Receiver is the same as turning the Javad receiver off and then on.

Restore Factory Defaults resets the Javad receiver to factory settings the receiver stops acting as base or rover. The baud rate of Port A will be set to 115,200. Reset this to 9600 by turning the receiver off and then on while holding down the FN button. Watch the REC light go from orange to green to red and then let up the FN button. This method can be used if Carlson Field cannot establish communications at any time.



Clear Non-Volatile Memory does everything *Restore Factory Defaults* does and also wipes out the almanac data that tells it where to look for the satellites. The receiver then downloads a new almanac from the satellites.

Send Command to Receiver allows experienced users to type in commands using Javad GPS receiver commands to set or report internal settings. (See the Javad operations manuals for a complete list of Javad GPS receiver commands.)

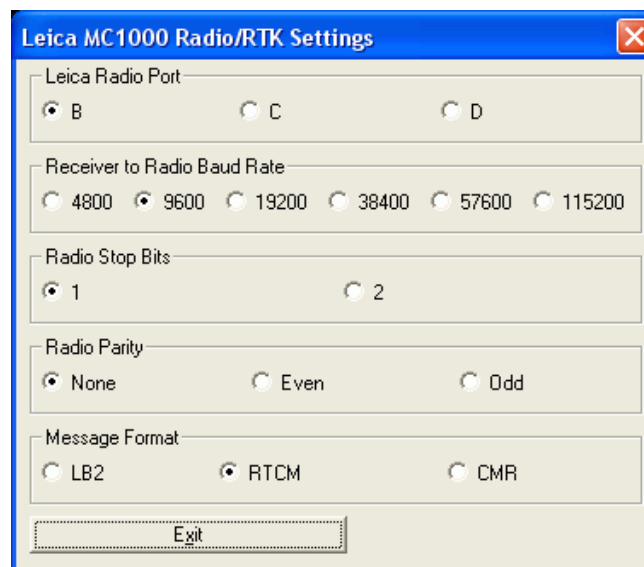
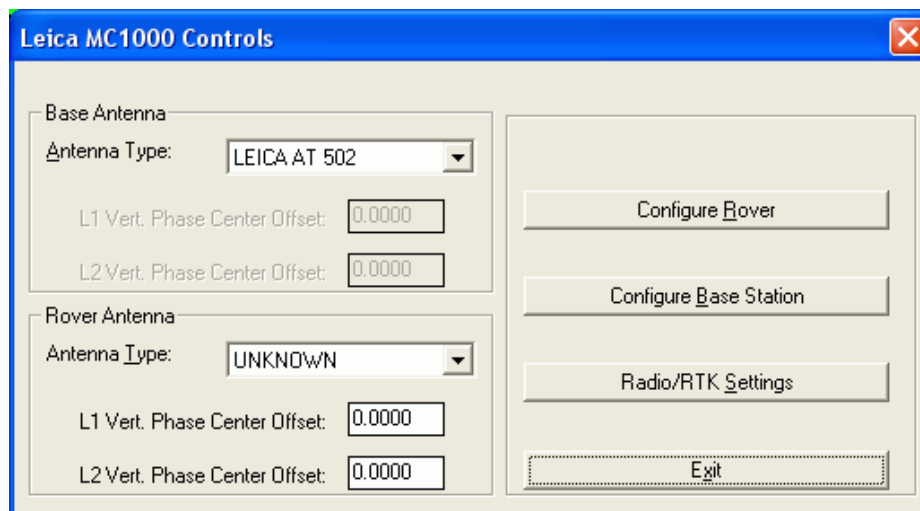
Create Base REF File takes a reading from the GPS receiver and stores this lat/lon to a reference file (.ref) that can be used later in *Configure Base Station*. The purpose is to allow moving the base station based on the current base setup. In this case, *Create Base REF File* would be run from the rover receiver while in "fixed" position. Then the base could be moved to this point without having to redo the local coordinate alignment.

Configure Base Station initiates the receiver attached to be a base and begin broadcasting its stationary position and satellite corrections to the rover. (See *Configure Base Station for All GPS Brands* at end of this section.)

Leica GPS Setup

Carlson Field works with the following Leica GPS receivers: System 500, GS50, MC1000 and MK31. The type of Leica receiver is set in the *Configure Field* command. The options available in the *GPS Setup* dialog depend on the current type of receiver.

Leica Radio Port is the port on the receiver where the radio is attached, usually 2 or 3. Port 1 is usually the one attached to the computer. For the System 500 receivers, you can also set the radio baud rate, stop bits and parity parameters.



For the system 500 receivers, you need to specify the antenna types used at both the base and rover receivers. This antenna type sets the phase center offsets for the antennas which can affect the reported elevations by as much as 0.25 foot if not set properly.

Cell phones can also be used with Leica GPS equipment instead of radios for RTK work.

For GS50 receivers, you can choose between US Coast Guard or Racad for the corrections.

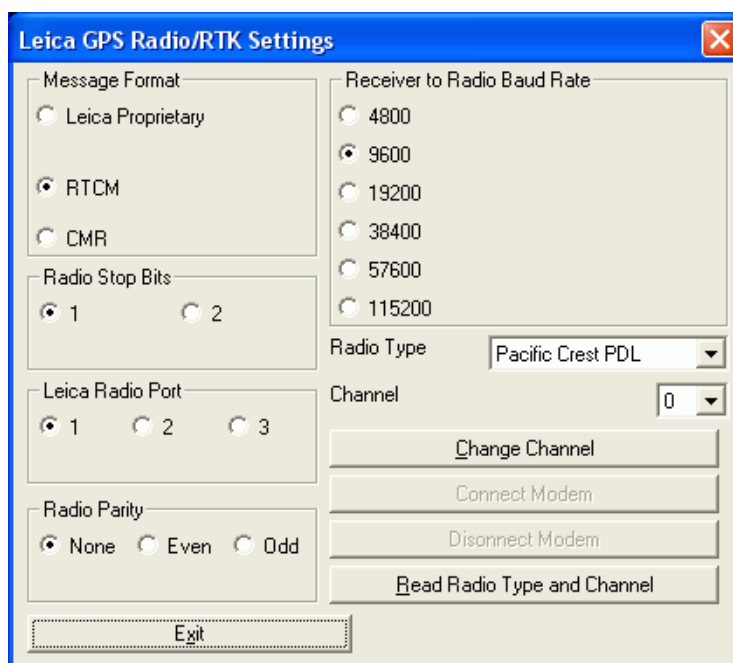
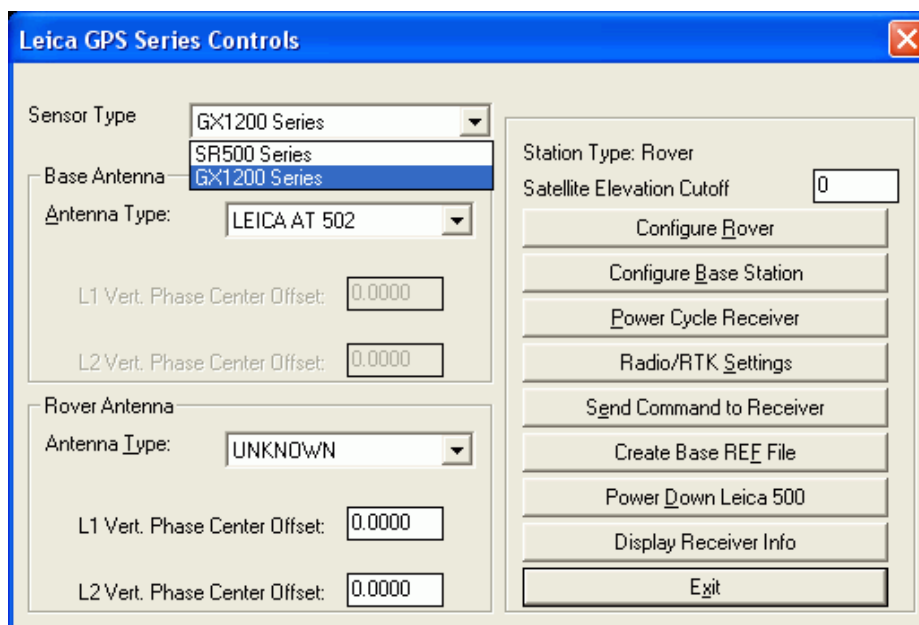
Power Cycle Receiver shuts the receiver off and turns it back on. This forces the receiver to reinitialize tracking satellites and the position solution. This routine is useful if the receiver is stuck in float solution.

Send Command to Receiver allows experienced users to type in Leica commands or send a file to set or report internal settings. (See the Leica operations manuals for a complete list of Leica GPS receiver commands.)

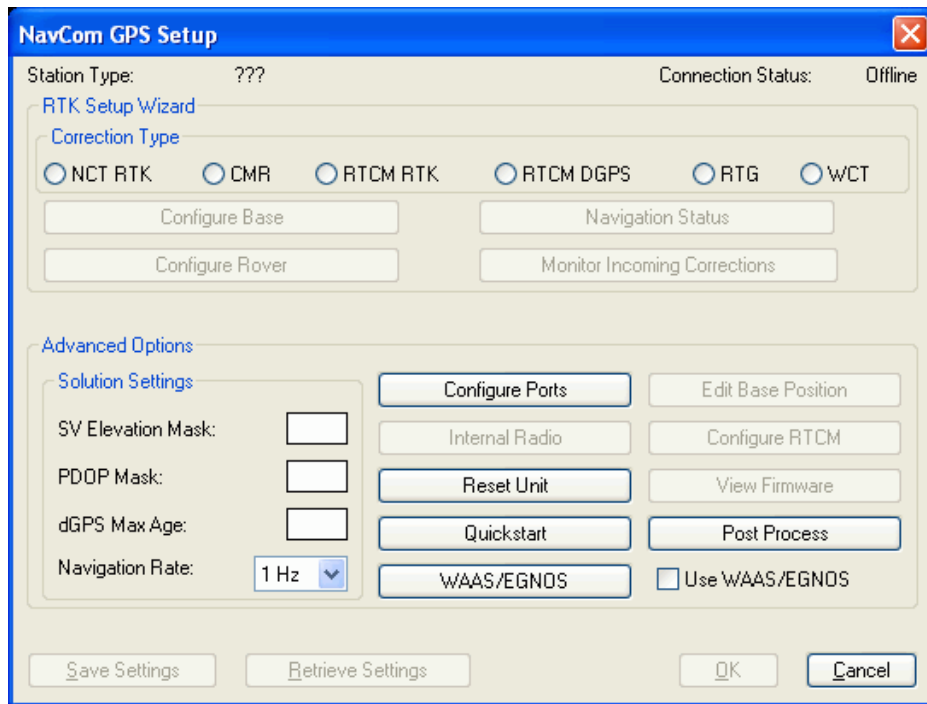
Create Base REF File takes a reading from the GPS receiver and stores this lat/lon to a reference file (.ref) that can be used later in Configure Base Station. The purpose is to allow moving the base station based on the current base setup. In this case, Create Base REF File would be run from the rover receiver while in "fixed" position. Then the base could be moved to this point without having to redo the local coordinate alignment.

Configure Rover sets the receiver to rover mode.

Configure Base Station initiates the receiver attached to be a base and begin broadcasting its stationary position and satellite corrections to the rover. (See Configure Base Station for All GPS Brands at end of this section.)



Navcom GPS Setup



Novatel GPS Setup

Carlson Field works with the original Novatel Outriders and the just released Outrider DL's including the centimeter accurate RT-2 RTK receivers and the sub-meter accurate Gismo USCG/satellite RTCM/DGPS beacon receivers.

Radio Port for external radio connection is typically COM 2 on the receiver. The Data Port connected to Carlson Field is typically COM 1.

Differential Mode toggles the Novatel GPS receiver to use RTCA, RTCM or CMR message types. RTCA is proprietary to Novatel and is used only for centimeter accuracy RTK GPS surveying. RTCM can be used with USCG/DGPS beacon signals for sub-meter accuracy. Novatel receivers work with Trimble CMR proprietary message signal type and can be either a base or rover working with Trimble RTK GPS receivers.

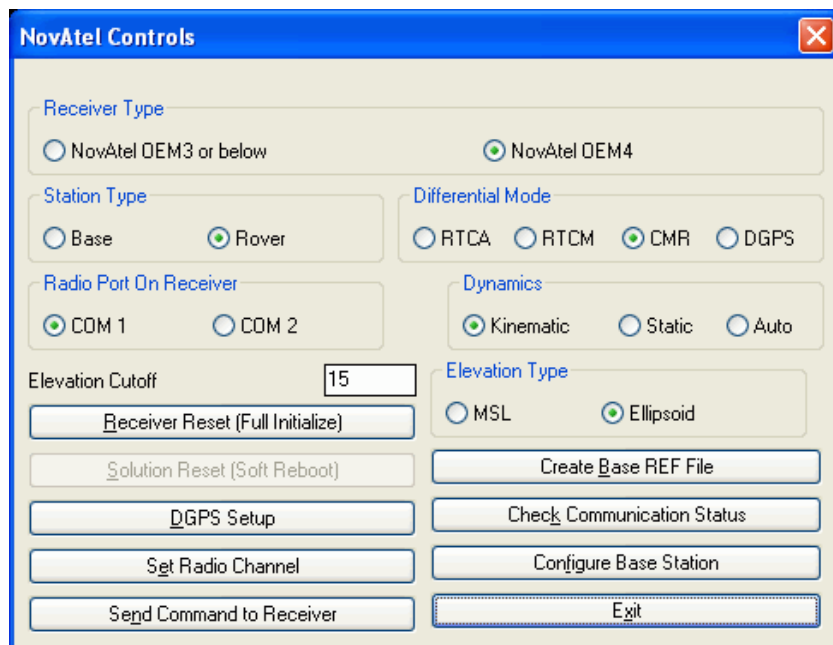
Dynamics toggles the rover between Kinematic or Static. The base is always in Kinematic mode. Kinematic is used for surveying while walking with the receiver. Static is for stationary use only at the rover and gives better accuracies. Since Static mode is for more precise measurements, it can be used for GPS alignment points and for any control points. The receiver should not be moved while in Static mode.

Elevation Cutoff is the vertical cut-off angle above the horizon. Any satellites below this limit will be ignored in calculations. 15 is a common setting.

Elevation Type chooses between Mean Sea Level or Ellipsoid for the elevation model used by the receiver.

Solution Reset (Soft Reboot) resets the Novatel receiver in a few seconds. This is used when the rover receiver is locked up or not properly reporting its position in the Monitor function.

Receiver Reset (Full Initialize) essentially does a factory reset and a power off and on cycle.



A Receiver Reset (Full Initialize) takes three to five minutes to get back on line and become fixed after a full initialize.

Set Radio Channel allows you to change Pacific Crest radio channels through the receiver. The base and rover must operate using the same radio channel.

Send Command to Receiver allows experienced users to type in commands using Novatel GPS receiver commands to set or report internal settings. (See the Novatel operations manuals for a complete list of Novatel GPS receiver commands.)

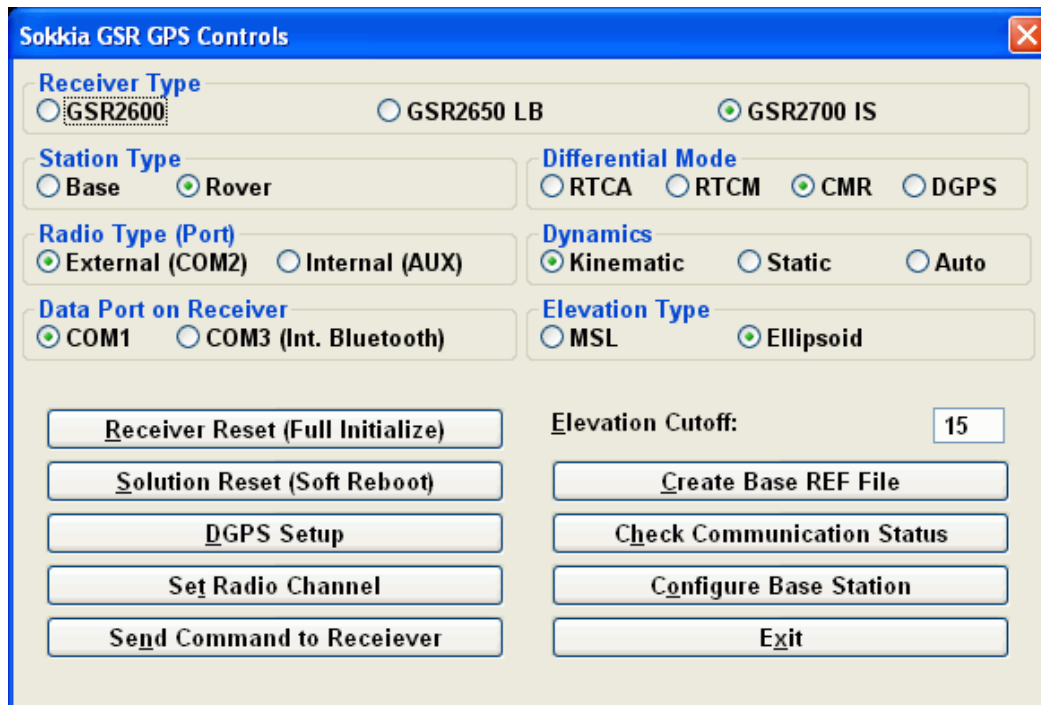
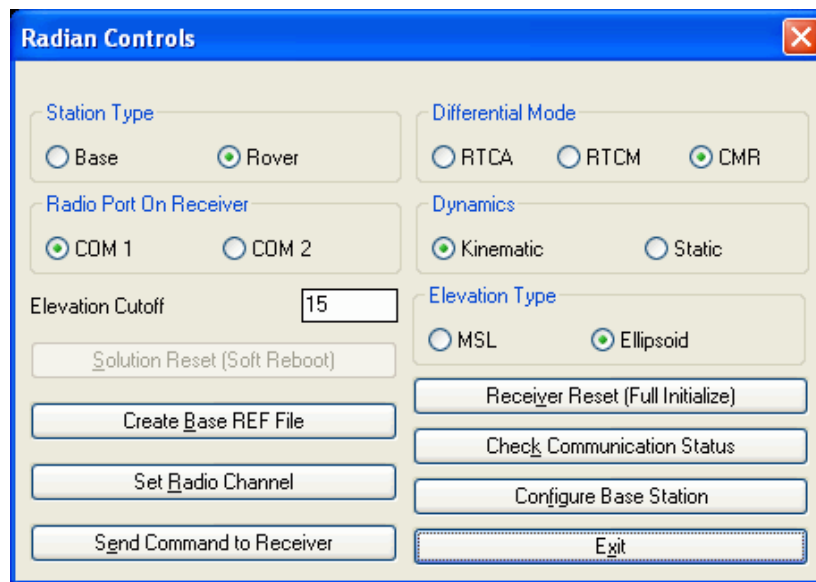
Check Communication Status checks the radio port operation and reports the status as working or not communicating.

Configure Base Station initiates the receiver connected to be a base and begin broadcasting its stationary position and satellite corrections to the rover. (See Configure Base Station for All GPS Brands at end of this section.)

Sokkia Radian GPS Setup

Radio Port for external radio connection is typically COM 2 on the receiver. The Data Port connected to Carlson Field is typically COM 1.

Differential Mode toggles the Sokkia GPS receiver to use RTCA, RTCM or CMR message types. RTCA is proprietary to Sokkia and is used only for centimeter accuracy RTK GPS surveying. RTCM can be used with USCG/DGPS beacon signals for sub-meter accuracy. Sokkia receivers work with Trimble CMR proprietary message signal type and can be either a base or rover working with Trimble RTK GPS receivers.



Dynamics toggles the rover between Kinematic or Static. The base is always in Kinematic mode. Kinematic is used for surveying while walking with the receiver. Static is for stationary use only at the rover and gives better accuracies. Since Static mode is for more precise measurements, it can be used for GPS alignment points and for any control points. The receiver should not be moved while in Static mode.

Elevation Cutoff is the vertical cut-off angle above the horizon. Any satellites below this limit will be ignored in calculations. 15 is a common setting.

Solution Reset (Soft Reboot) resets the Novatel receiver in a few seconds. This is used when the rover receiver is locked up or not properly reporting its position in the Monitor function.

Receiver Reset (Full Initialize) essentially does a factory reset and a power off and on cycle. A Receiver Reset (Full Initialize) takes three to five minutes to get back on line and become fixed after a full initialize.

Set Radio Channel allows you to change Pacific Crest radio channels through the receiver. The base and rover must operate using the same radio channel.

Send Command to Receiver allows experienced users to type in commands using Sokkia GPS receiver commands

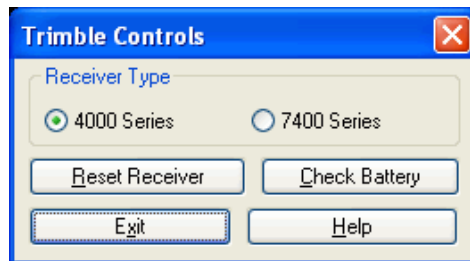
to set or report internal settings. (See the Sokkia operations manuals for a complete list of Sokkia GPS receiver commands.)

Check Communication Status checks the radio port operation and reports the status as working or not communicating.

Configure Base Station initiates the receiver connected to be a base and begin broadcasting its stationary position and satellite corrections to the rover. (See Configure Base Station for All GPS Brands at end of this section.)

Trimble GPS Controls

Carlson Field works with the following Trimble receivers: 4000 series, 4700, 4800, 7400, NT300D, GeoExplorer and Pathfinder. The type of receiver is set in the Configure Field command. The options available in the GPS Setup dialog depend on the current type of receiver.



For the Pathfinder and GeoExplorer, the *Altitude Measurement Type* chooses between using Ellipsoid or Mean Sea Level as the elevation model in the receiver.



With the Pathfinder, Carlson Field will activate the receiver when the first Carlson Field command is run and the receiver will stay active until Carlson Field is exited. The reason is that the Pathfinder will turn off as soon as the COM port is turned off. If you need to make Carlson Field turn off the receiver, then use the Close Communication

With Pathfinder button.

With the Pathfinder, *DGPS Correction Source* selects whether the Pathfinder will get its Corrections from a local Coast Guard Radio Beacon or from the Racal Satellite Correction service. Note that the Racal option must be enabled on the receiver in order to use Racal satellite corrections. (See your dealer for details as to how to do so). If *Racal Service* is selected as the correction source, the *Racal Region* selection will be enabled. The region corresponding to the relative location of the receiver should be selected to ensure proper reception of corrections.

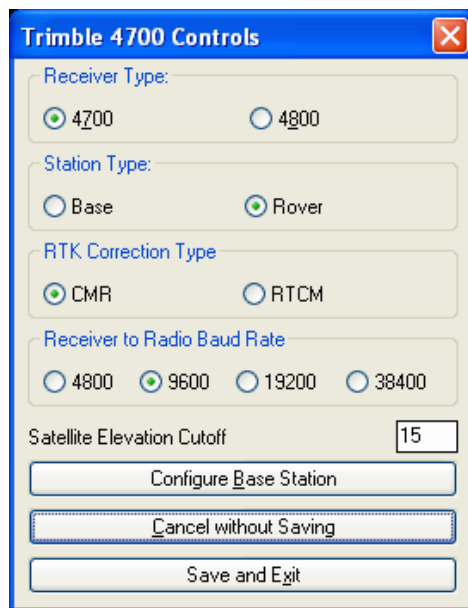
The screenshot shows the 'Trimble GPS Controls' window with the following settings:

- Station Type:** Base, Rover
- Receiver Type:** 4700, 4800, 5700/5800, MS Series
- Data Port:** 1, 2 Serial, 3, 4 BT
- Position Update Rate:** 1Hz, 5Hz, 10Hz
- Antenna Parameters:** Antenna Type: 4800 Internal; Antenna Height: 0.0000; Height Measured To: Hook By 4800 Tape
- RTK Type:** RTCM, CMR, CMR+
- Radio Parity:** Odd, None, Even
- Radio Port On Receiver:** Port 2, Channel 1
- Receiver to Radio Baud Rate:** 4800, 9600, 19200, 38400
- Satellite Cutoff:** 15
- PDOP Cutoff:** 7

Buttons at the bottom: Set Radio Station, Configure Base Station, Reset Receiver, Power Receiver On/Off, Cancel without Saving, Save and Exit.

The Pathfinder and 4700/4800 also feature the ability to select a *Satellite Elevation Cutoff*. All satellites with elevations below this setting will not be used in the final position calculations, even if they are otherwise visible to the receiver.

For 4700/4800 series receivers, the *Receiver Type* option must be set to the correct model in order for Carlson Field to communicate with the receiver. *RTK Correction Type* selects what format of RTK corrections between the Base and Rover receivers. CMR and RTCM formats are available. *Radio Baud Rate* should be set to the same setting as the communication port of the radio connected to the receiver. 4800 bps, 9600 bps, 19,200 bps and 38,400 bps rates are supported. *Configure Base Station* will configure the receiver as a base and begin transmitting corrections via the radio.



Library Drivers - GPS

For library drivers, unlike the legacy Carlson Field GPS drivers, you select a Base or Rover driver from the Configure Field command (instead of inside the Equipment Setup dialog). A GPS Base Equipment Setup dialog is shown below.

General receiver options are configured under the *Receiver* section. There are input boxes for *Antenna Height* and *Elevation Mask*. The antenna height is the rod height of the GPS receiver. The elevation mask is specified in whole degrees (above the horizon, or 0 degrees). The *Position Rate* specifies the rate at which the GPS receiver will output positions or corrections. The available position rates vary from receiver to receiver.

The *RTK* section is used to setup the RTK configuration. First, select the *RTK Device* to be used, and click the *Configure* button next to it if you need to configure the radio (ie. channels, etc). Next, select the *RTK Network* (if applicable). If available, select the correct *RTK Port* and *RTK Baud* to define the connection between the receiver and the RTK device. Then, select the desired *Message Type* (correction type). Finally, enter a *Base ID*, or leave it blank for 'any base', and continue by. In the example below, the base receiver will be configured to output *CMR* corrections (*Message Type*) through the *Internal Radio* (*RTK Device*). The *RTK Port* and *RTK Baud* are grayed out because these are known values when using the internal radio on the Altus APS-3. *RTK Network* is not available in the example because it doesn't apply to the chosen RTK device. The Rover would be configured similarly to accept CMR corrections on it's internal radio.

Altus APS-3 Base Setup

Receiver

Antenna Type: Internal

Antenna Height: 0.000

Elevation Mask: 7

Position Rate: 10 Hz

RTK

RTK Device: Internal Radio

RTK Network: None

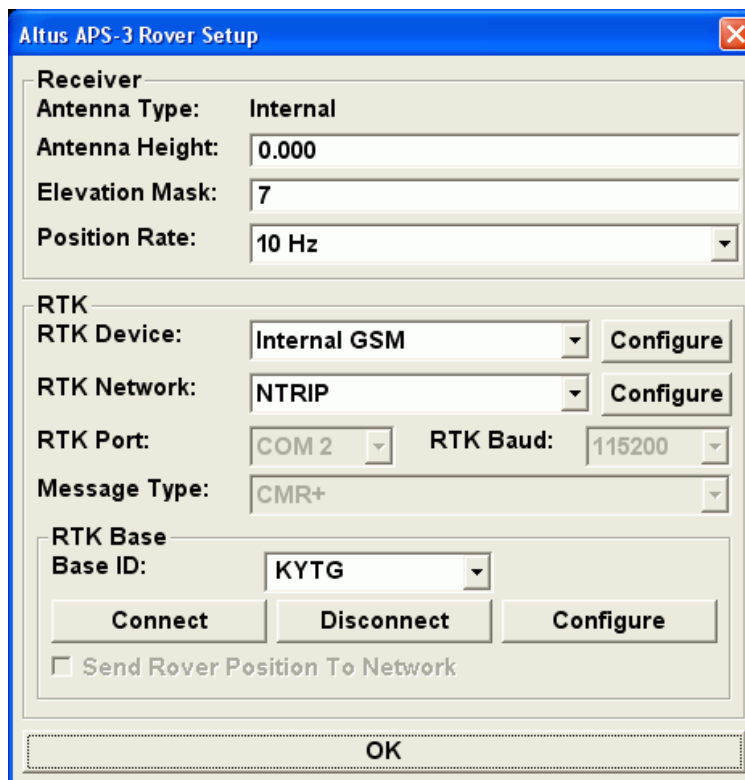
RTK Port: COM 2 RTK Baud: 115200

Message Type: CMR

RTK Base

Base ID:

To switch to the Rover, you must return to Configure Field > Equipment Type > Library Drivers and select the GPS Rover Manufacturer and Model. Similar to configuring a Base, general receiver options are configured under the *Receiver* section. In the example below, the rover receiver will be configured to receive *CMR+* corrections from the *KYTG NTRIP base* over an internet connection using the receiver's *Internal GSM* modem (PLEASE NOTE: This is a separate example from configuring a GPS Base. The examples are not intended to show how to configure the same Base/Rover pair to work in a differential configuration, but are intended to show two entirely different types of configurations.). The *RTK Port* and *RTK Baud* are grayed out because these are known values when using the internal GSM modem on the Altus APS-3. The *Message Type* is grayed out because this is the only message type available from the *KYTG NTRIP base*. Also, in the example below, the user would be required to click the *Connect* button before exiting the Rover setup to actually connect to the *NTRIP base* and start receiving corrections.



Normally, the *Configure* button next to the *RTK Network* is grayed out, and the *Configure* button in the *RTK Base* section is used to configure any details about the Base that the Rover is receiving corrections from. However, *NTRIP* is slightly different. When using *NTRIP* as the *RTK Network* the *Configure* button next to the *RTK Network* is used to configure the *NTRIP Broadcaster*. The *NTRIP Broadcaster* is the server that actually broadcasts it's available *NTRIP* bases and their information. *Save and Exit* will connect to the displayed *NTRIP Broadcaster* and download the base list. *Exit* will keep the broadcaster you have selected, but will attempt to load the base list from memory and will not attempt to connect to the broadcaster to download the base list.



When using an *NTRIP RTK Network*, the *Configure* button in the *RTK Base* section will display *NTRIP Base* information to help the user select the best base choice. As shown below, *NTRIP Base* information usually contains a short name (*Name*), a descriptive name (*Identifier*), the *NTRIP Base* type (*Type*), available *RTK* message type (*Format*), and approximate *NTRIP Base* location (*Position*). The *Send Rover Position To Network* toggle is always grayed out, but displays whether or not the selected *NTRIP Base* requires the Rover position to be sent back (ex. this is typically required by virtual base systems). Once the preferred *NTRIP Base* is selected, click the *Connect* button to actually connect to the base to begin downloading corrections. After a successful connection is created, exit the Rover Setup dialog using the *OK* button. It is recommended to goto the *Monitor/Skyplot* routine to verify that the Rover is receiving corrections. The Rover's status will be *Float* or *Fixed/Locked* if receiving corrections.

Configure Base for All RTK GPS Brands

Within Equipment Setup, the Configure Base Station button is the command that starts the base receiver broadcasting GPS corrections to the rover. You must click the *Configure Base Station* button in Equipment Setup while your are connected to the base receiver. The base needs a set of coordinates to use as its stationary position. There are five methods to set the stationary base position: Read from GPS, Enter Lat/Lon, Enter State Plane Coord, and Read From Reference File and Read From Alignment File.

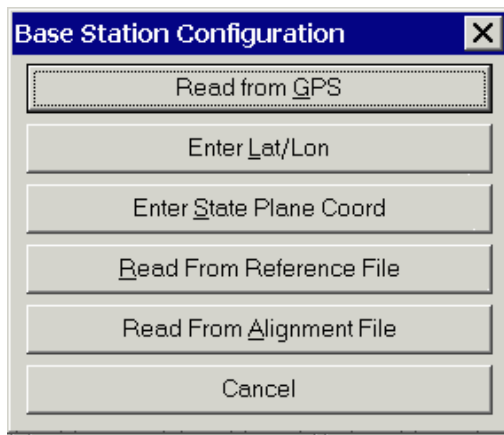
Read from GPS - This method takes one GPS reading from the base receivers autonomous position and uses it as its "true" position. The autonomous position can be off of the actual position by 200 feet. The base will calculate corrections based on this autonomous position. If you set up the base with this method, the rovers must be aligned since the corrections they are using are based on a "true" position that is not really true.

Enter Lat/Lon - requires you to enter the latitude and longitude for the position of the base antenna. This is useful if you are setting up over a USGS monument whose lat/lon you know. It can also be used over a control point whose position is known from GPS post-processing.

Enter State Plane Coord - requires you to enter the State Plane northing and easting for the point that the base is occupying. This is useful if you are setting up over a USGS monument whose coordinates you know.

Read From Reference File - reads a previously saved base position file. All of the other methods of setting up the base let you save the base position at the end of setup. If you return to a site, set up the base in exactly the same position, use Read From Reference File to use the same base position and you don't have to re-align the rover: the old alignment is still valid.

Read From Alignment File - reads a position file from one of the control points in an alignment file. This allows you to setup the base on one of the control points from the alignment. Then you don't have to re-align the rover: the old alignment is still valid.



Method 1 - Read from GPS

Step 1

Pick *Read from GPS* Step 2 - Station ID (Optional)

If you plan on doing post-processing, you can input a Station ID for the base GPS Antenna location. Otherwise just hit *OK*.

Reminder Pop-Box

You are reminded to connect the radio to the correct port.

Base GPS receiver's autonomous position

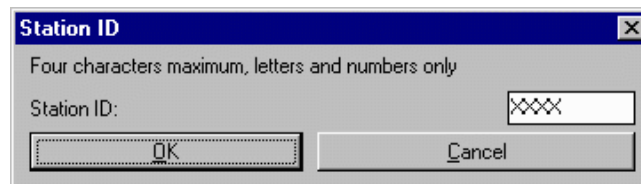
Carlson Field takes a reading and displays the latitude, longitude and ellipsoid height. This is the position that the base will use as its "true" position. The base is now configured. If you are using Pacific Crest radios, the TX light on the radio should begin blinking.

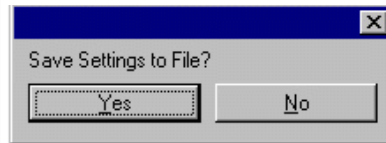
Error Message if incorrect

If the GPS receiver is not properly connected, is turned off, or hasn't determined a position yet, you will see an error message. Check all connections and try again.

Step 3 - Save Settings to File?

You have the option to save this base position as a file. You'll be able to use this file if you set up in the same spot in the future.





Method 2 - Enter Lat/Lon

Step 1

Pick *Enter Lat/Lon*.

Step 2 - Enter Lat/Long/Ellipsoid Height

Input the Latitude, Longitude and Ellipsoid Height for the base position. Pick North or South for the Latitude and East or West for Longitude. Important Note: The Latitude and Longitude entered must be within 100 meters of its true location on the globe. Ideally the entered base position should be a Latitude, Longitude and Ellipsoid Height from an accurate post processed static GPS point or a published NGS monument.

Step 3 - Station ID (Optional)

If you plan on doing post-processing, you can input a Station ID for the base GPS Antenna location. Otherwise just hit *OK*.

Reminder Pop-Box

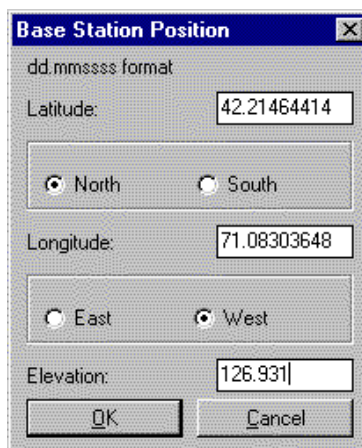
You are reminded to connect the radio to the proper port.

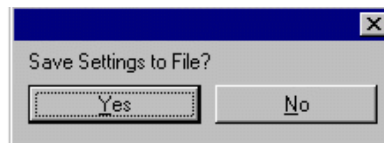
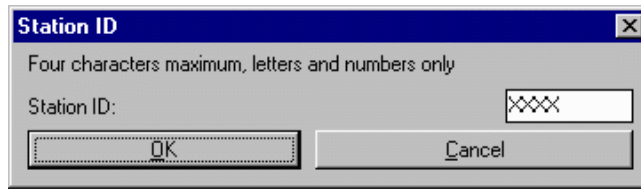
Base's Lat/Lon/Hgt position

The Lat/Long and Ellipsoid Height for the base position are displayed. These will be used for corrections and broadcast to the rover. If your radio has a TX light, it should begin flashing.

Step 4 - Save Settings to File?

You have the option to save this base position as a file. You'll be able to use this file if you set up in the same spot in the future.





Method 3 -Enter State Plane Coord Step 1

Pick *Enter State Plane Coord*.

Reminder Pop-Box - Current Zone & Datum

You are reminded what State Plane Zone and Datum is loaded. If this is incorrect, exit Equipment Setup and input correct State Plane Zone and Datum in Configure Field>GPS Settings.

Step 2 - Enter Northing/Easting/Elevation

Input the State Plane coordinates (northing, easting and elevation) for the base position. Important Note: The State Plane coordinates entered must be within 100 meters of its true location on the globe. Ideally, the entered State Plane coordinates (N,E,Z) should be from an accurate post processed static GPS survey point or from a published NGS monument data sheet.

Step 3 - Station ID (Optional)

If you plan on doing post-processing, you can input a Station ID for the base GPS Antenna location. Otherwise just hit *OK*.

Reminder Pop-Box

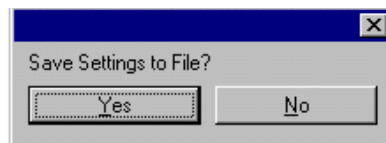
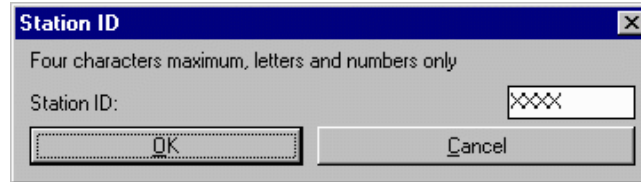
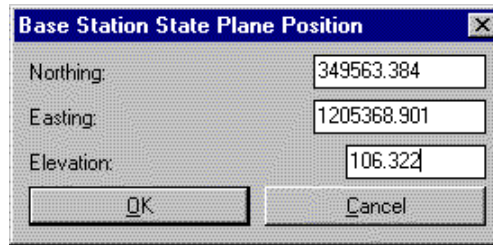
You are reminded to connect the radio to the correct port.

Base's Lat/Lon/Hgt position

The Lat/Long and Ellipsoid Height for the base position are displayed. This position will be used for the corrections that are sent to the rover. If your radio has a TX light, it should begin flashing.

Step 4 - Save Settings to File?

You have the option to save this base position as a file. You'll be able to use this file if you set up in the same spot in the future.



Method 4 - Read From Reference File

Step 1

Pick *Read From Reference File* to select an existing base position REF file.

Step 2 - Select Base Reference File to Load

Pick the base position REF file to be loaded. Use the up arrow folder to browse elsewhere for the REF file.

Position as Read from File

The latitude, longitude and elevation are read from the selected file and displayed.

Step 3 - Base Antenna Height

Enter the vertical height of the base antenna.

Step 4 - Station ID (Optional)

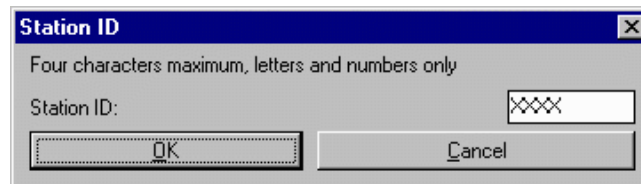
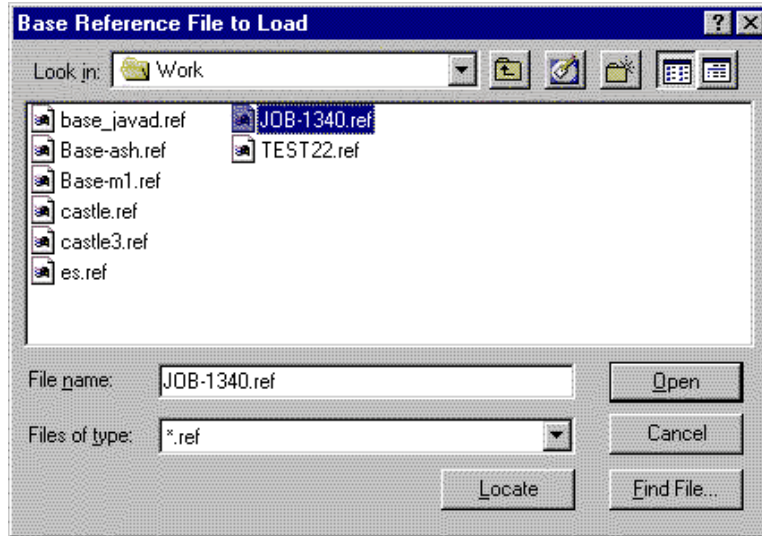
If you plan on doing post-processing, you can input a Station ID for the base GPS Antenna location. Otherwise just hit *OK*.

Reminder Pop-Box

You are reminded to connect the radio to the selected port.

Base's Lat/Lon/Hgt Position

The Lat/Long and Ellipsoid Height for the base position are displayed. This position will be used to calculate the correction that are sent to the rover. If your radio has a TX light, it should begin flashing.





Method 5 - Read From Alignment File

Step 1

Pick *Read From Alignment File*.

Step 2 - Select Alignment File to Load

Pick the alignment DAT file to be loaded. Use the up arrow folder to browse elsewhere for the DAT file.

Step 3 - Select Alignment Point

The program will display a list of points in the alignment file. Pick the point from this list.

Step 4 - Base Antenna Height

Enter the vertical height of the base antenna.

Step 5 - Station ID (Optional)

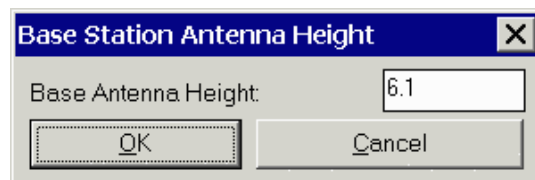
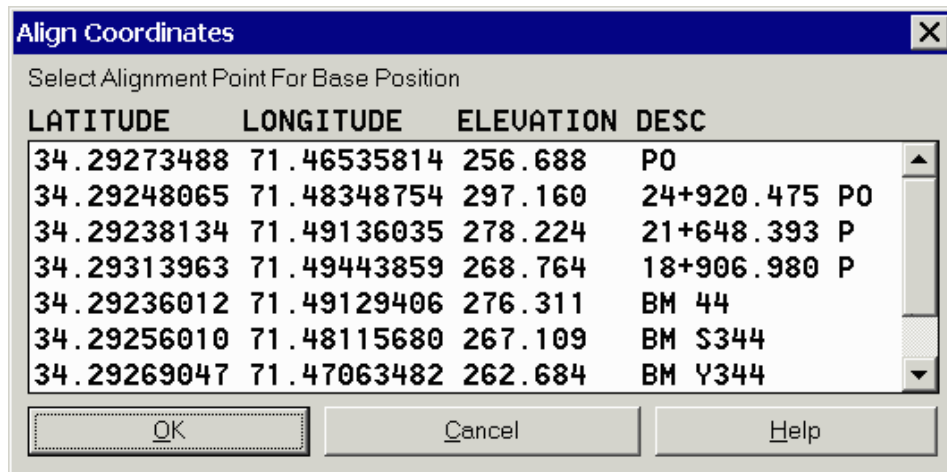
If you plan on doing post-processing, you can input a Station ID for the base GPS Antenna location. Otherwise just hit *OK*.

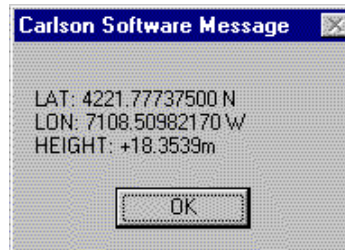
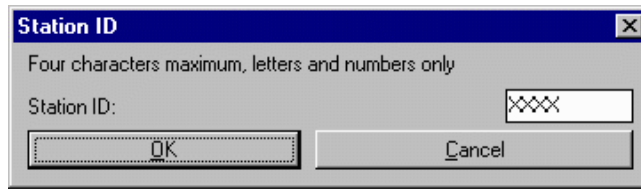
Reminder Pop-Box

You are reminded to connect the radio to the selected port.

Base's Lat/Lon/Hgt Position

The Lat/Long and Ellipsoid Height for the base position are displayed. This position will be used to calculate the correction that are sent to the rover. If your radio has a TX light, it should begin flashing.





Saving Base Settings to a File

It is always recommended to save the base position to a file if you are going to return to the same site survey again. You can setup on the same base position, recall the base REF file and enter the new antenna height. Then you can use the alignment file from the first day in the rover and not have to re-align.

When you save the base antenna position to a file it is stored with a REF extension denoting base reference file. By default, it goes in the Data directory. Input reference filename and pick *Save* and *OK*.

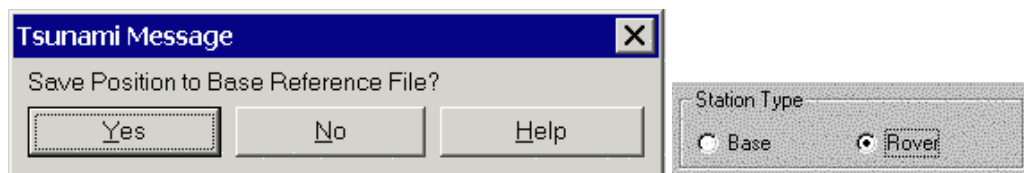
Configuring the Rover

After the base is configured, unplug the base receiver from the Carlson Field computer and plug in the rover receiver. In Equipment Setup, toggle the *Station Type* from *Base* to *Rover*. Then pick *Exit*. This will configure the receiver as a rover.

From the Field drop-down, pick the command Monitor GPS Position. The Status is reported as either Autonomous, Float or Fixed.

If the rover is Autonomous, it is not getting any corrections from the base.

If the status is Float, the rover is receiving corrections, but has not found the fixed solution. Once the solution becomes Fixed, the rover is locked on to the base corrections and is calculating an accurate position.

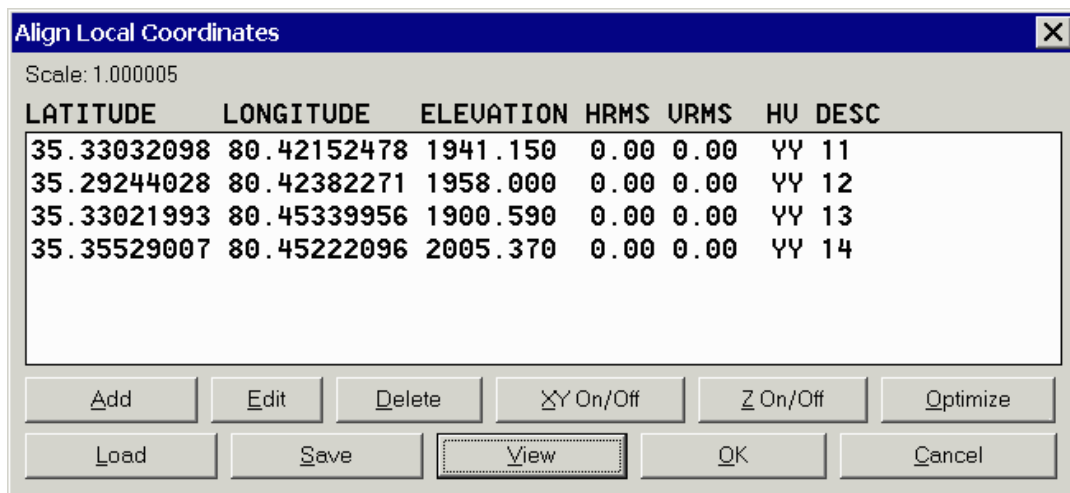
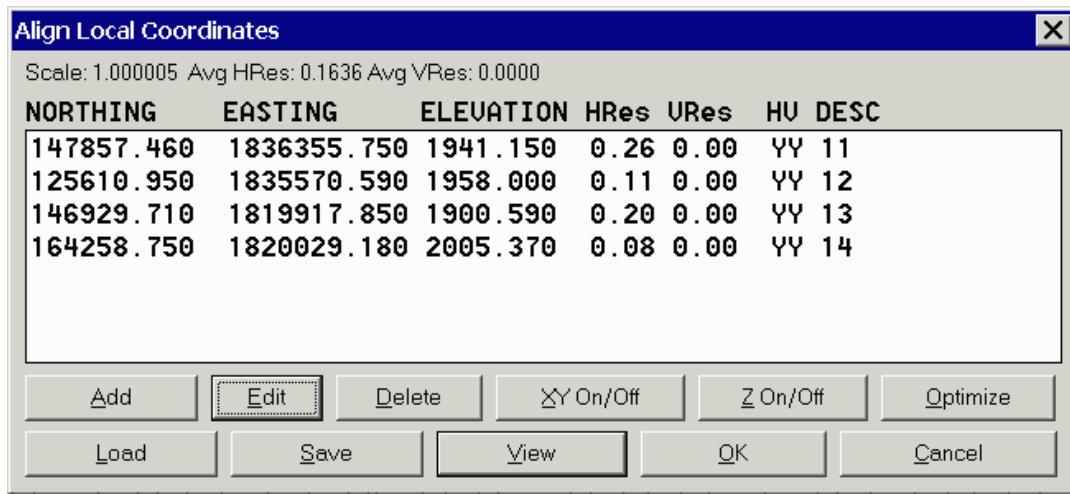


Align GPS To Local Coordinates

Carlson Field reads a latitude, longitude and height position from the GPS rover receiver and converts these values to State Plane or UTM coordinates for the current zone as set in Configure Field. Using local coordinates and their corresponding GPS position, Align Local Coordinates applies a transformation to convert the state plane or UTM

coordinate to the local. Carlson Field can operate in three different modes depending on the Align Local Coordinate settings:

- 1) No points - No Adjustment
- 2) One point - Translation Only
- 3) Two or more points - Translate, Rotate and Scale



Without any alignment points set, Carlson Field will operate with no alignment which directly uses the state plane or UTM coordinates. In order for the coordinates to be the true state plane coordinates in this alignment mode, the GPS base receiver must be set up over a known point and the true Lat/Long for the point must be entered in the base as the base position. Otherwise, if the base is set over an arbitrary point, then the coordinates will not be true state plane.

In one point alignment mode, one pair of GPS and local coordinates is specified. The differences between the GPS and local northing, easting and elevation for these points are used as the translation distances in the transformation. The rotation will use either the state plane grid or the geodetic as north. No scale is applied in this transformation.

A two or more point alignment is used to align to an existing local coordinate system. At least two pairs of local and GPS coordinates must be entered.

In addition to the northing and easting transformation, SurvStar will also translate the elevation from the GPS system to the local. The elevation difference between the two systems is modeled by a best-fit plane.

An alignment is only valid if the base receiver setup has not changed since the alignment points were recorded. In order to use an alignment when returning to a site, you must set up the base receiver in the same position and enter

the same LAT/LONG coordinates for the base.

The *Align GPS to Local Crds* menu item brings up the alignment dialog box. There is more information than to fit in one window, so use the *View* button to switch between viewing the local coordinates and the GPS Lat/Lon.

Each line in the box represents one alignment point. Each point in an alignment file relates a specific Lat/Lon/Elv to a specific Northing/Easting/Elevation for your local coordinate system. Carlson Field will use the current alignment file every time that the GPS is read. It provides the necessary adjustment to properly convert that position to your coordinate system.

In the local points view, the HRes column shows the horizontal residual and the VRes column shows the vertical residual. The residual is the difference between the actual point and the point calculated using the alignment transformation. In GPS points view, the HRMS and VRMS columns show the horizontal and vertical RMS values when that point was recorded.

The *On/Off* buttons allow you to switch whether the highlighted point is used for the horizontal and/or vertical alignment. The *HV* column shows a 'Y' if this point is used in the calculations. Otherwise it shows an 'N'. The H column represents horizontal control and the V column vertical control. For example, you may wish to use 2 points for horizontal alignment and one for vertical.

The *Optimize* button will find the combination of turning alignment points on/off for horizontal and vertical such that the horizontal and vertical residuals are minimized.

The *Desc* field shows an optional description of the alignment points.

The scale factor and average horizontal and vertical residuals appear at the top of the window. These values serve as a check that the alignment is valid. The scale factor should be closed to 1.0 (in range of 0.9 to 1.1). The average residuals should be less than 0.2.

XY On/Off toggles the highlighted alignment point horizontal component off or on. Alignment points with the horizontal component toggled off will not use the northing and easting of that point for adjustment calculations.

Z On/Off toggles the highlighted alignment point vertical component off or on. Alignment points with the vertical component toggled off will not use the elevation of that point for adjustment calculations.

Note: When you toggle either the XY or Z component off or on for any alignment point the scale factor and Horiz/Vert residuals are recalculated automatically. Briefly toggling XY or Z components off or on and reviewing the scale factor and residuals changes is a quick approach to finding the best alignment points. Carlson Field can handle an unlimited number of alignment points.

Highlight an existing alignment point entry and pick *Delete* to delete that alignment point.

Pick the *Add* button to create an alignment point. The Add Alignment Point dialog box appears. There are two ways to enter the local coordinate points: by entering the N, E, Z, or by using an existing point number stored in the current coordinate CRD file. The GPS values can also be specified by two methods: by entering in the Latitude, Longitude and Height or by occupying the control point with the rover and taking a GPS reading at this location. Manually entering the Lat/Lon can only be done when the base is setup on a known location using a true lat/lon position. Otherwise Carlson Field needs to use the Read GPS method. For this method, the base can be setup with a lat/lon that only needs to be close (within 100 feet) of the actual lat/lon. This type of position can be read from an autonomous GPS position. With the base setup on this approximate lat/lon, go with the rover to the control points and use the Read GPS option in the Add function. The rover GPS solution must be in "fixed" status when the alignment point is added. By reading the rover GPS position for the alignment points, the alignment will transform the coordinates from the GPS system of the current base setup to your local coordinate system.

Load allows you to open an existing alignment file. Only one alignment file can be open at a time. Alignment files have a DAT extension and stored in the Data directory by default. This function also loads .LOC files from SurvCE, .DC files from Trimble and .GC3 files from Topcon.

Save stores alignment files (DAT extensions) to a file. Files are by default stored to the Data subdirectory.

The *OK* button will set the current alignment to the settings in the dialog.

Alignment Methods

Carlson Field can operate by the following Alignment methods:

Alignment Method 1) - No alignment points

Alignment Method 2) - One point alignment

Alignment Method 3) - Two or more alignment points

Alignment Method 1

With no alignment of the rover, Carlson Field will report Northing and Easting as State Plane or UTM coordinates. In order for this method to give accurate State Plane or UTM coordinate values, the GPS base receiver must be set up over a known point and configured using the true Lat/Long/Hgt or true State Plane coordinates. If the base is set up over an arbitrary point, configured by reading the GPS, the RTK GPS stored coordinates will be translated up to a 200 feet but accurate in relation to each other.

When using this method, you can skip *Align GPS to Local Crds* and start surveying immediately once the base is configured and transmitting its position and the rover is fixed.

In most cases, you cannot use Method 1 because you will not have setup the base on a point whose exact true position you know. Therefore the base corrections are going to be off a certain distance north/south and a certain distance east/west. This is why you want to do an alignment. You are showing Carlson Field how to correct for the north/south and east/west offsets. Any points surveyed with the alignment file active will be translated to their proper position.

To gather alignment points, you put the GPS antenna over a point with known coordinates and Carlson Field records the GPS Lat/Lon/Elv and the Northing/Easting/Elevation you give it. This point can be a local coordinate, for example a stake you are calling 5000,5000. It can also be a true State Plane point. Using one or more State Plane points will give you an alignment to true State Plane (even if your base is not using its own true position.)

Alignment Method 2

This method uses one alignment point to translate the GPS coordinates to local or true State Plane coordinates.

Remember that if the base is set up over an arbitrary point, the GPS coordinates can be off from true state plane by up to 200 feet. This alignment method can be used to correct for this by translating the system onto true state plane coordinates.

You can choose if you want the coordinate system North to be Geodetic North or State Plane Grid North under *Configure Field>GPS Settings*. If you specify a scale factor in that dialog box, it will be applied to all points recorded.

One point alignment is useful for data collection on a new site. In this case you can set up the GPS base receiver anywhere convenient. Then position the rover over a point (preferably one you can find again) and add this point as your one alignment point by reading the GPS point and entering a local coordinate like 5000,5000,100. Now the local coordinate system is set around this first point at 5000,5000,100.

This method is commonly used for small topo or stockpile RTK GPS surveys. When collecting or staking data at distances greater than 2 miles from the base, both the horizontal and vertical errors will begin to increase gradually. Therefore, you should use a multiple point alignment for large projects.

Alignment Method 3

This method is useful if you are arriving on a job which has already been surveyed. It assures that your survey is in the same coordinate system as the original survey.

Using control points, this method transforms the GPS coordinates to an existing local coordinate system. This method takes pairs of GPS coordinates and the corresponding local coordinates to define the translation, rotation and scale of the alignment.

In *Configure Field>GPS Settings*, there is a choice for the transformation as Plane Similarity or Rigid Body. Plane Similarity will apply a scale factor to the transformation. The scale factor will be based on the alignment points and should always be very near 1.0 to be correct. The Rigid Body option will align by translate and rotate but no scale. Any difference in scale between the GPS and local coordinate systems will be distributed equally between the two alignment points. These differences will appear as horizontal residuals in the Alignment dialog.

Two pairs of points are sufficient to define the translation, rotation and scale for the transformation. But adding more alignment points yields the most accurate results for aligning to existing coordinate systems. Since two pairs of coordinates are sufficient to define the transformation, there is extra data when there are three or more pairs. The program uses a least-squares best-fit routine to find the transformation that minimizes the residuals. This one best-fit transformation is used to convert from the GPS to the local coordinate system for all the points. The residuals are the differences between the transformed GPS coordinates and the actual local coordinates.

A multiple point alignment is especially helpful on a survey which covers a large area. The error in raw GPS coordinates increases as you get farther from the base. Taking alignment points around the perimeter of your job site as alignment points will give you the best geometry for the alignment.

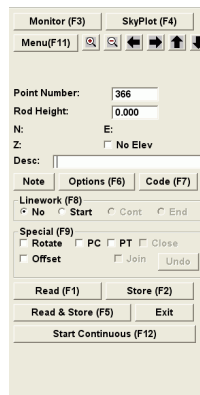
Point Store

This function creates points by reading from GPS or total station equipment. The new points are stored in the current coordinate and simultaneously drawn in the drawing. The measurement data is also stored to the current raw file which has the same name as the coordinate file except with a .RW5 instead of .CRD file extension.

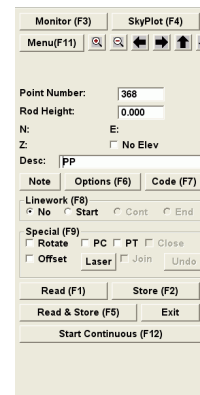
The Point Store dialog docks on the side of the drawing window. This allows you to see the drawing view as you collect points. You can use the arrow keys to pan the drawing and the Page Up/Page Down keys to zoom out and in. There are also icons for the pan and zoom functions at the top of the dialog. Also, besides clicking the function buttons, most buttons have an associated function key such as F1 that you can use to run the routine.

Before taking measurements, make sure that the rod height is correct.

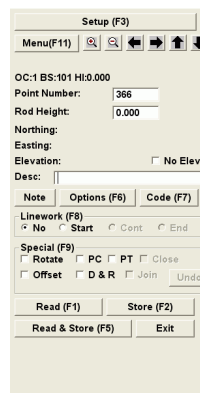
To take a measurement from the survey equipment, pick the *Read* button. The calculated northing, easting and elevation will be displayed in the dialog and a temporary icon will be shown in the drawing at the point location.



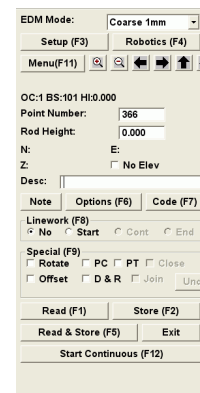
Point Store dialog for GPS



Point Store dialog for GPS with a Laser



Point Store dialog for non-robotic total station



Point Store dialog for robotic total station

Before storing the point, make sure that the point number and description are set in the *Point Number* and *Description* fields. The point number is a required field for storing to the current coordinate file. If the point number specified

already exists in the coordinate file, then a dialog will pop-up with options to overwrite the existing point number, to use another point number or to cancel storing the new point. The *Point Number* field will automatically increment after storing the point.

The *Description* is an optional field for identifying the point. The maximum length of the description is 32 characters. Besides naming the point, the description can also be used to with Field-To-Finish to draw linework and to determine the symbol of the point in the drawing. When the Field-To-Finish option is set on in *Options*, the program will lookup the description in the current code table. If the description matches one of the codes, then the code can determine the symbol, layer, format of the point when it is drawn. Otherwise the defaults in the Point Setting section of the Options dialog are used for the point symbol, layer and format.

To store the new point to the coordinate file and draw the point, pick the Store button. At the time that Store is applied, the program uses the point number, description, linework options and special options currently set in the dialog.

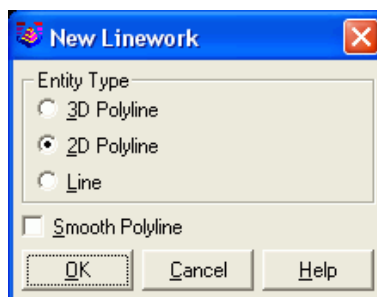
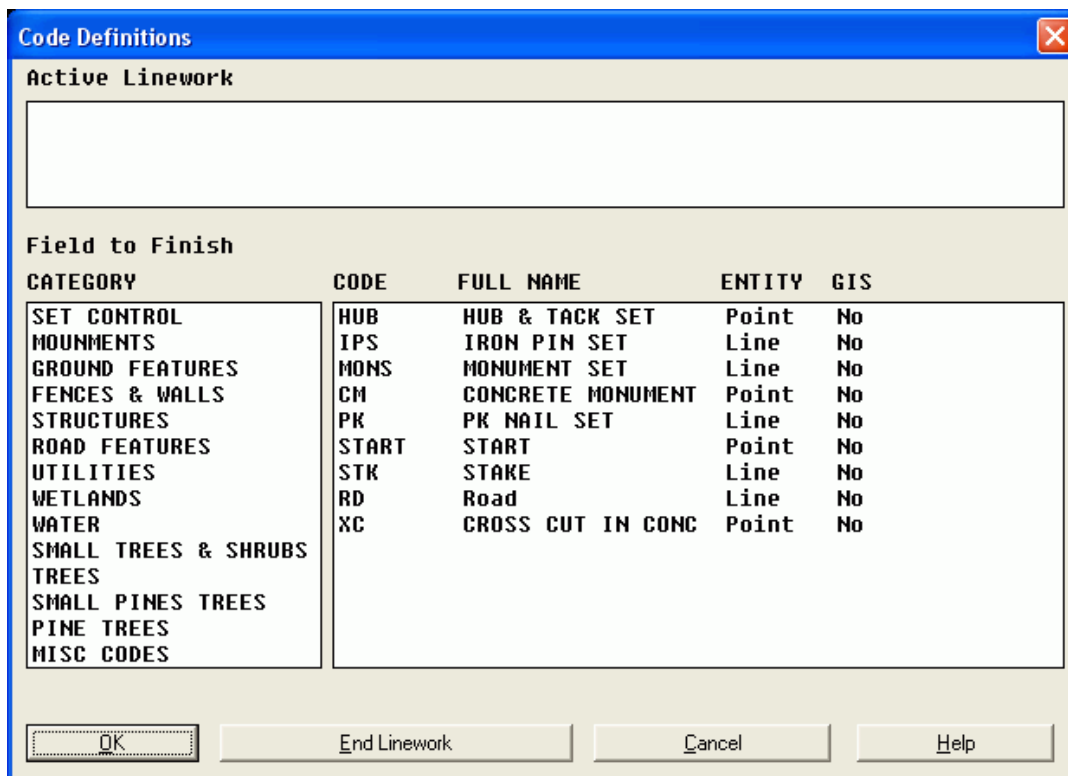
You can also use the *Read & Store* button to do both functions in one step. With this method, the program will take a measurement and if the measurement is successful, then the point will be stored immediately.

The *Code* button brings up a list of point descriptions from the current Field-To-Finish code table. You can select a code from the list to set this code as the current point description. This function also shows a list of all the descriptions of currently active linework. You can end a currently active linework by highlighting the linework description from the Active Linework list and pressing the End Linework button.

Many of the options for storing points can be set in the Configure Field>Point Settings command. The *Options* button in this dialog is a shortcut to these point settings.

If you want lines or polylines to connect the points that you are about to record, select the *Start* button under *Linework*. After the first point, the *Linework* selection will change itself to *Cont* meaning continue. Leave this selected while you are recording points in the same line. Before shooting the last point in your line, change it to *End*. If you want the line to close itself onto its first point, check the *Close* button.

The Field-To-Finish Linework option is an automatic way to start linework. The program will lookup the point description in the code table. If the description matches a code and the code is defined to create linework, then the *Start* toggle in the *Linework* options is turned on. Otherwise you can begin new linework by toggling on *Start* manually.



When a point is stored and *Start* is on, Carlson Field pops-up a dialog for choosing between a line, 2D polyline or 3D polyline. A 3D polyline can contain points with different elevations, but a 2D polyline always has an elevation of zero. The *Smooth Polyline* option will create Bezier smooth polyline through the points.

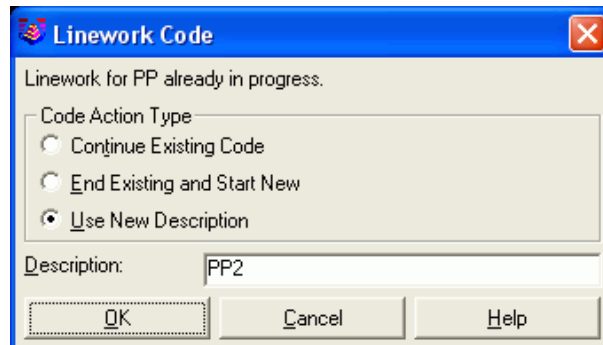
Carlson Field can keep track of several lines being drawn at once. Each line corresponds to a set of points with a different description. Let's say you are shooting a line of points called "fence" and you want to shoot some points on a curb, but you're not finished with the fence. You change the *Desc* box from "fence" to "curb". Carlson Field lets the fence line go for now. It changes the *Linework* selection to *No*. You want a line for your curb, so you select *Start*. The points you shoot now will form a new curb line. To go back to recording fence points, change the description back to "fence". The fence line you were working on will continue to include any new fence points you shoot. If you want to end this fence line, select *End* under *Linework* and Carlson Field will not connect any future "fence" points to this line. If you start a new linework with a description that already has linework, then Carlson Field pops-up a dialog with three options as shown. The *Continue Existing Code* option is the same as using *Cont* instead of *Start*. The *End Existing and Start New* option will end the active linework and start the new linework with the same description. The *Use New Description* option will keep the existing linework and start another linework with another description. For example if you are surveying two edge of pavement lines, you can have one with the description "EP" and the other with "EP2".

The *PC* and *PT* options are for drawing curves. If you want to plot a curve, check the *PC* box before recording the first point on your curve. Shoot as many points along the curve as you need. Carlson Field can handle compound curves as well as simple curves with this function. Before shooting the last point on the curve, check the *PT* button. If you don't specify a *PT*, Carlson Field will assume a three point curve.

Similar to Field-To-Finish linework, when looking up the point description in the code table, if the description matches a code and the code is defined to use data collection codes, then the *Offset* and/or *Rotate* toggles in the *Special* options are turned on. Otherwise you can create an offset and symbol rotation by toggling them on manually. *Offset* is for defining a left/right offset or an azimuth based offset. It also has the option for a vertical offset. *Rotate* will control the rotation angle of the symbol that is drawn when the point is stored; if enabled, the drawn symbol will be rotated based on the current heading of movement.

The *Undo* button will remove the last point number created. The point is removed from both the coordinate file and the drawing.

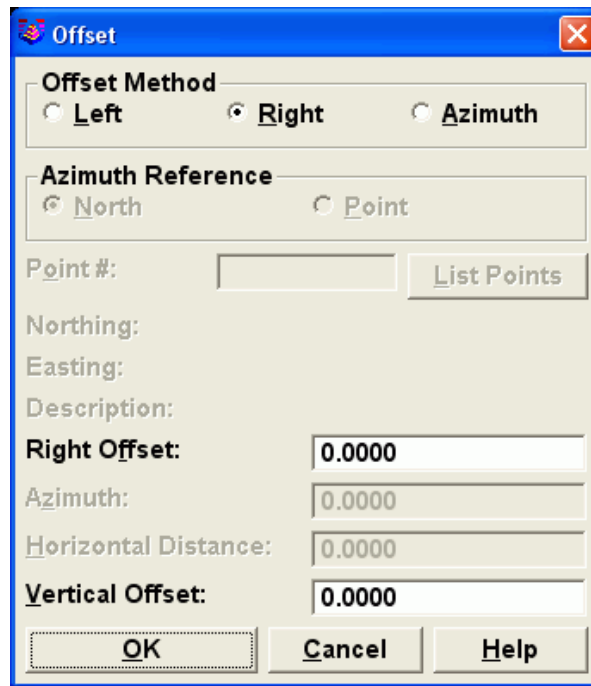
For GPS and tracking total stations, there is a *Start Continuous* button which makes Carlson Field continuously read from the instrument. The coordinates are displayed in the dialog and your position is shown with an arrow icon in the graphics view. To store a point, you can use the *Store* button without using the *Read* button first. Once continuous reading is active, the button changes to *Stop Continuous* which will put you back in standard reading mode.



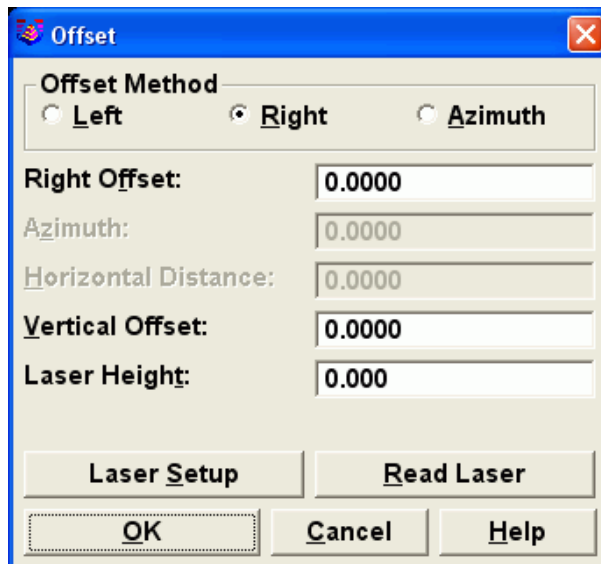
Point Store with GPS

When using GPS equipment, Carlson Field will also report the RMS values and solution status when you take a reading. If Carlson Field gives you a message that your RMS values are too high when you try to read a point, you can click on the *Monitor* button to bring up the *Monitor* window which will give you information on how accurately your position is determined and how many satellites you are tracking. The *Skyplot* button will bring up the window showing you where in the sky the satellites are.

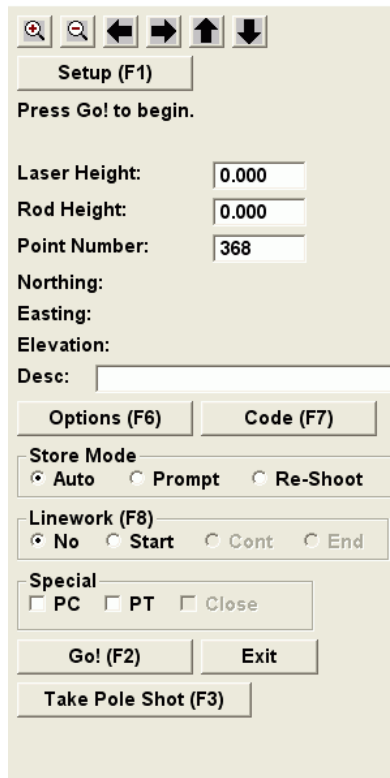
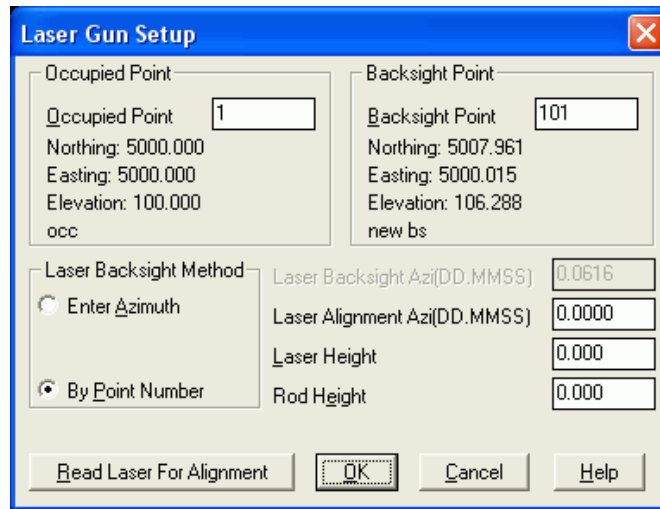
For points that are hard to reach directly by GPS, you can use the *Offset* option. This option can be used in areas of limited satellite communication such as high walls or under a tree. This allows you to setup the rover in a clear area and read the coordinate. The point that is actually stored is offset from the rover position. To create an offset point, turn on the *Offset* toggle and then choose *Read*. The offset direction can be entered as left, right or azimuth. The left and right offset is relative to the rover position at the previous read. The offset distance is entered in the dialog. A Vertical Offset can also be specified. Choose *Store* to store this point after the offset is done.



Offsets can also be done with laser guns when the laser option is setup in Configure Field>GPS Settings. There are two methods for taking laser offsets. One method is to use the *Offset* toggle and the *Read* button. In the Offset dialog, there is button for Read Laser for using the laser measurement for the offset distance and/or angle. This method creates a single offset point.



The other method is to use the *Laser* button which can create many offset points. This method brings up another dialog. The Setup button can be used to set the *Laser Alignment Azimuth*. This alignment applies to laser guns that use a magnetic compass for the horizontal angle. The magnetic north can vary from the north of your coordinate system. The *Laser Alignment Azimuth* is added to the measured laser azimuth to adjust for the difference. To set the alignment azimuth, specify a reference backsight direction by either entering an azimuth or by point number. Then choose the *Read Laser For Alignment* button and take a laser shot towards the backsight. The program will compare the azimuth from the laser with the reference backsight to figure the alignment azimuth. When the alignment azimuth is set, pick the *Go* button. Carlson Field then listens for measurements on the laser gun port. To take a shot, sight the target point and press the laser trigger. Carlson Field will read the laser measurement and read the GPS position. The laser angle and distance are combined with the GPS position for the new point coordinates. To return to regular GPS Point Store, choose the *Exit* button.



Point Store with Total Stations

Before taking measurements with total stations, you need to specify the occupied point coordinates of the instrument, the backsight and the height of the instrument. This current setup data is shown in the "OC:# BK:# HI:#" line in the dialog. Also icons are draw to show the occupied point and backsight direction in the drawing view

The *Setup* button at the top of this dialog brings up the *Total Station Setup* dialog, where you can change your occupied point, backsight and instrument height.

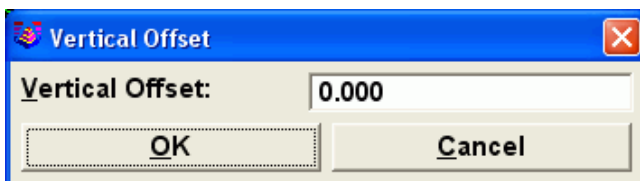
For robotic total stations, there is also a *Joystick* button to turn the instrument, search for the prism and set tracking or standby mode.

Carlson Field can shoot points with offsets. To shoot a point with an offset, check the *Offset* button on the Point Store dialog box. Click Read or press F1. A window appears to let you choose the type of offset to shoot. The choices are Distance/Angle and Enter Offset Distances. The *Offset Vertical* option will prompt for an elevation difference to apply to the point.

To do a Distance/Angle offset, you first take a distance shot and then angle shot. For the distance measurement, have the rodman stand to the side of the point. The prism and the point should both be the same distance from the total station. Carlson Field takes the first shot and gets the distance from it. It then prompts you to read the angle. Turn the gun so that it is aimed at the point. The prism is not needed for this step. Click OK and Carlson Field reads the horizontal angle from the gun and combines this with the distance from earlier to calculate the coordinates of the point. Also for combining these shots, there is an option whether to use the vertical angle from the distance or from the angle shot.



With the Enter Offset Distances method, you can supply both a left/right offset and an in/out offset. To do a In/Out offset, have the rodman stand a measured distance in front of or behind the point. The total station will take the shot and then Carlson Field will ask you how to move the point: in or out and the distance. If the prism is in front of the point, choose out. If it's behind the point, choose in. To do a Left/Right offset, have the rodman stand a measured distance to one side of the point. After taking the shot, Carlson Field will ask whether to offset right or left. If you are at the total station, looking at the prism, and the point you are after is to the right of the prism as you're looking at them, choose right offset. Otherwise, choose left offset.

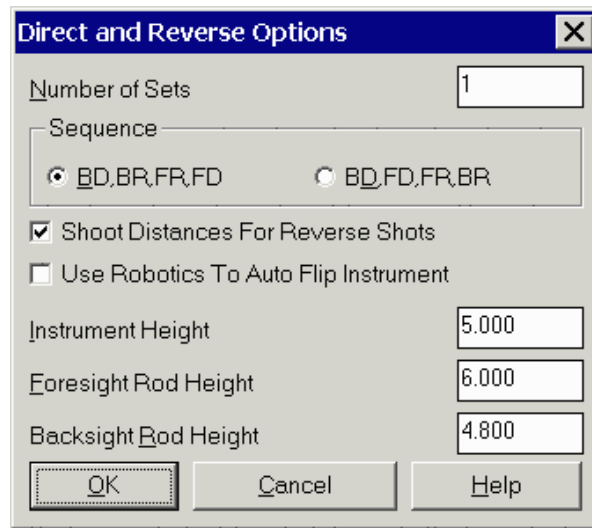


Choose *Store* to store this point after the offset is done.

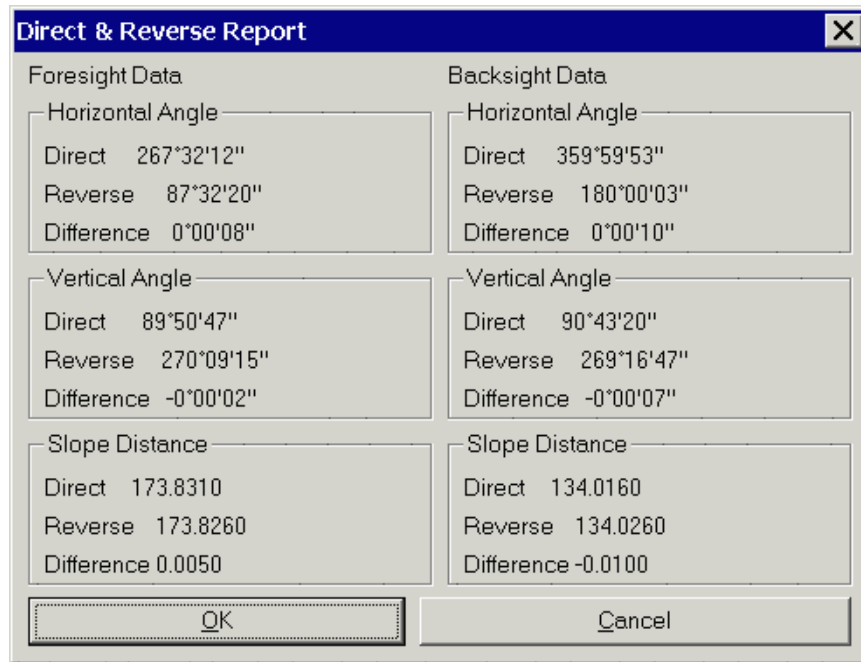
The *D&R* option stands for Direct and Reverse. When this box is checked, Carlson Field will take sets of four shots to determine the coordinates of the next point. Two shots are taken for both the backsight point and the foresight point: one direct shot, one shot with the total station reversed. This yields a more accurate reading. Two options are available for the order of shots when doing a D&R. The first is Backsight Direct, Backsight Reverse,

Foresight Reverse, Foresight Direct. The other option is Backsight Direct, Foresight Direct, Foresight Reverse, Backsight Reverse. Carlson Field also offers the option of shooting multiple sets of Direct & Reverse for even greater accuracy. The *Shoot Distances For Reverse Shots* option determines whether to take distance measurements on the foresight reverse and backsight reverse shots. When this option is off, the program will still use the reverse shots to mean the angles. Otherwise the program will also use the reverse shots to mean the distances. The *Use Robotics To Auto Flip Instrument* option applies to robotic total stations to have the program automatically turn the instrument for reverse shots.

To shoot a point as a Direct & Reverse, check the *D&R* box and click on *Read*. A dialog box appears, offering the choice of orders for the shots. Before each shot, Carlson Field tells you what kind of shot is being taken. After each shot, Carlson Field reports the measurements and allows you to confirm the measurement or to re-shoot. After all four shots are taken, Carlson Field does the math and reports the accuracy of each part of the measurement.



Choose *Store* after completing the Direct & Reverse to store this new point.



StakeOut

The Stakeout function is used to find a specific point in the field. Once you tell Carlson Field the point that you are looking for, pick *Start* and the program draws an X-marks-the-spot bullseye on that point in the drawing. Carlson Field also draws a triangle on the drawing for where you are currently standing. These icons help to guide to the target point graphically. Carlson Field also reports in the dialog box how far you need to move to reach the point.

There are several options for Stakeout defined in Configure Field>Stakeout Settings. These options should be set by Configure Field before running Stakeout. See the Configure Field section of the manual for a description of the stakeout options.

There are four ways to define the target point for stakeout. The first method is to specify a point number from the current coordinate file. To do this, click on the *Point Number* button and type in the point number in the dialog. The second method is to give a station and an offset from a centerline. The program will prompt for a centerline file (.CL) and then the station and offset. You can also specify the station interval for automatically incrementing to the next stakeout point. See the Roads section of this manual for how to create centerline files. The third method is to graphically pick the point from the drawing. Select *Pick Point* and a dialog box allows you to pick different snaps: endpoint, midpoint, center, node (point), or intersection. This will help you pick your desired point more accurately. For example, you can select endpoint and then pick on a polyline corner to stakeout that the polyline endpoint. See the Object Snap command in this manual for more on snaps. The fourth method is to simply type in the target point coordinates in the *Northing*, *Easting* and *Elevation* fields.

Once the stakeout point is set, click the *Start* button and Carlson Field begins the stakeout routine. The format of the stakeout screen that appears depends on whether you are using total stations or GPS as described below.

When you reach the target point, click the *Store* button. Carlson Field reports the difference between your current position and your target position. At this point you can choose to store this staked-out point as a new point in the coordinate file.

Monitor (F3)	SkyPlot (F4)
Menu(F11)	⊕ ⊖ ⬅ ➡ ⬆ ⬇
Rod Height	0.000
Point Number (F7)	
Station-Offset (F8)	
Pick Point (F9)	
Point Number:	17
Desc:	TRV/NS
Northing:	19460.903
Easting:	17779.736
Elevation:	105.320
Start (F1)	Exit

Station-Offset for Stakeout

Station: 0.000

Offset: 12.000

Next Interval: 50.000

OK Cancel

Store Stakeout Point

Point Number: 123

Description: Stake 23

Draw Stakeout Point

OK Cancel

When the target stakeout point has an elevation, Carlson Field also reports the elevation difference between the target and current elevations. This cut/fill is also in an edit box that allows you to change the value for labeling. For example, you may want to round the cut/fill number to an even number to label on the stake with a mark to indicate where this even number occurs. When you change the cut/fill label from the original value, Carlson Field will report the offset for this mark. For total stations, Carlson Field will also report the zenith angle for locating this mark. There are also fields in the report dialog for entering vertical offsets to get additional cut/fill values. For example, if the target point is for the road surface and you want to also get the cut/fill to an 18 inch subgrade, then enter -1.5 as the vertical offset.

GPS Stakeout

After you click *Start* to begin staking the point, Carlson Field changes the dialog box to the one shown below. The dialog shows the target point, the current position northing, easting and elevation and the GPS HRMS/VRMS. The distance, azimuth and cut/fill from the current position to the target are also reported. Carlson Field also breaks down this distance into how far north/south and how far east/west to go. Finally based on your current heading, the program tells you whether to turn right, turn left or that you are on-line.

In the graphics view, the large "X" shows the point being staked-out and the triangle represents your position. A temporary line is drawn between your current position and the target. In Configure Field>Stakeout Settings, there is an option to auto zoom in as you approach the target point. Otherwise you can use the arrow keys to pan the display and the Page Up/Down keys to zoom out and in.

Stakeout Report

POSITION: 5043.504,5041.101,1002.523

TARGET: 5043.336,5041.267,0.000

DIFFERENCE: South:0.168 East:0.166

DISTANCE: 0.236

Store Point Exit

Stakeout Report [X]

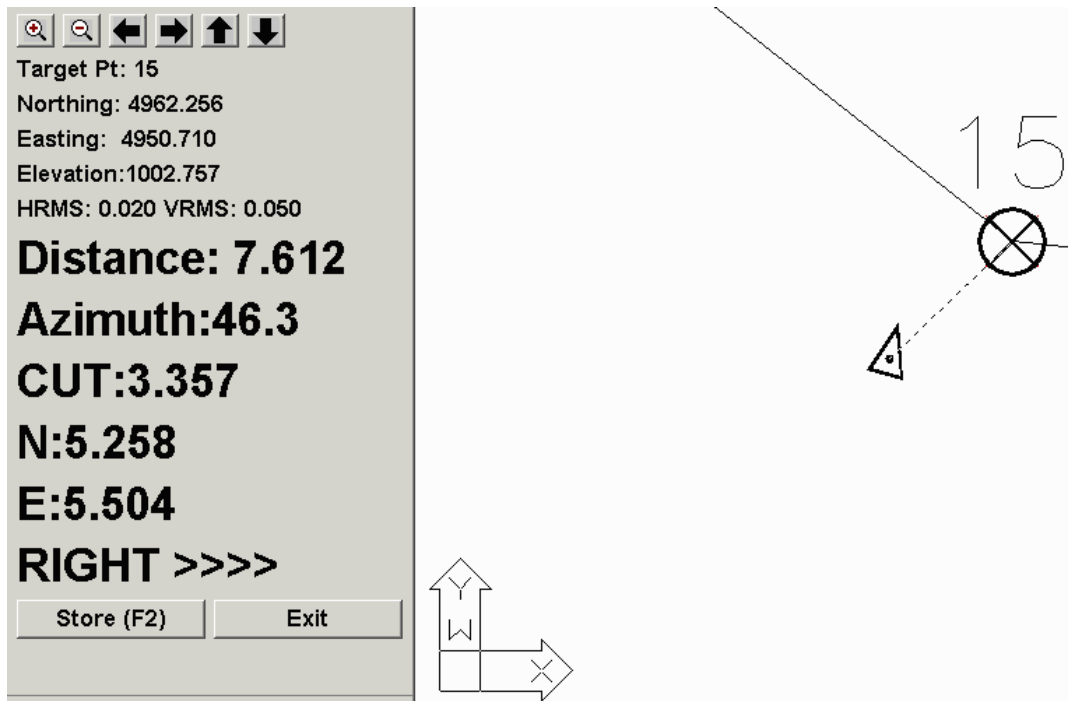
POSITION: 5033.700,4970.740,1001.830
 TARGET PT: 25
 TARGET: 5033.449,4970.718,1002.720
 DIFFERENCE: South:0.251 West:0.022 Fill:0.890
 DISTANCE: 0.252
 Target Vert Offset 1
 Target Vert Offset 2
 Store Stake Point
 Label Stake FILL

OK Cancel

Total Station Stakeout

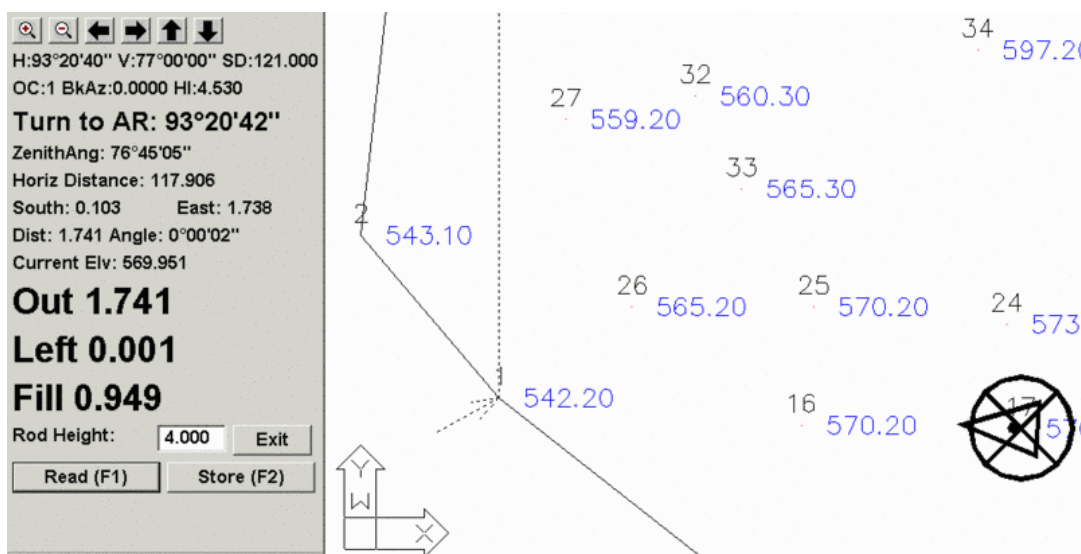
Before starting the stakeout, be sure that the instrument is setup with correct occupied point, backsight and instrument height. This setup data is displayed in the third line of the dialog. You can pick the Setup button to change the instrument setup.

After you click *Start* to begin staking the point, Carlson Field changes the dialog box to the one shown below. The dialog shows the angle to turn the gun and the horizontal distance to the target. Turn the instrument to this angle and position the rodman at this angle and distance. Then pick the Read button to take a measurement. Carlson Field will then report the horizontal distance and cut/fill from the current position to the target. This distance is also reported as how far north/south and how far east/west to go and as how far in/out and left/right to go. To in/out and left/right distances are relative to the rodman facing the instrument. Keep moving the rodman and picking the Read button until you reach the point. Then pick the Store button.



For robotic total stations operating remotely, there is a Continuous button that puts the instrument in tracking mode with continuous measurements.

In the graphics view, the large "X" shows the point being staked-out and the triangle represents your position. Also the location of the instrument is shown with an icon and the backsight is shown as temporary line.



Auto Points at Interval

This command stores a point whenever the distance or time from the previous point exceeds the user-specified interval. This command only applies to GPS and robotic total stations. If you will be collecting a large number of points at once, *Auto Points at Interval* can be a useful tool. For example, you may want to plot the edge of a road. Once you start *Auto Points*, you can walk along that edge of road and let Carlson Field record your position automatically.

The *Auto Points at Interval* dialog box resembles the *Point Store* dialog box with the addition that you can set the interval to record points. You can set it to store a point every time you move a certain distance by selecting *Distance* and entering the distance you choose in the *Interval* box. The distance will be taken in feet if your project is using English units, or meters if your project is in Metric. If you select *Time*, the number in the *Interval* box will refer to

the number of seconds between creating points.

Check the *Draw Linework* box to have your points connected by a line or polyline. You can enter a description or choose from the code table just like in *Point Store*.

The *Offset* toggle will apply an offset to the calculated coordinates. The horizontal offset is applied perpendicular either left or right to the direction of movement. There is also an option for a vertical offset.

The *Rotate* toggle will rotate each drawn symbol in reference to the current heading.

If the description contains a Field-to-Finish code, the *Offset* and *Rotate* toggles will be selected/unselected automatically based on that code's Data Collection Code settings.

Pick *Start* to begin storing points. Carlson Field will take a reading and store the first point. Then Carlson Field will continuously read the GPS or total station. For distance interval method, as each point is read the distance from the last point is calculated. When the distance is greater than the specified interval, a point is created and the point number is displayed in the dialog. In practice, the actual distance between stored points will be greater than the distance interval. For example, if the distance interval is 10 and the current distance is 8.9, then no point is stored. Then you keep moving and the next distance is 11.4 which will store a point.

For time interval point storing, after reading and storing the first point, Carlson Field will wait for the interval time to pass, then read and store again.

The screenshot displays the Carlson Field software interface. At the top, there are two buttons: "Monitor (F3)" and "SkyPlot (F4)". Below these are navigation controls: "Menu(F11)", a magnifying glass icon, a minus sign icon, left and right arrow icons, and up and down arrow icons. The main area contains several input fields: "Point Number:" with the value "359", "Rod Height:" with "0.000", "Northing:", "Easting:", "Elev:", and "Desc:". Below these are two buttons: "Options (F6)" and "Code (F7)". The "Interval Type" section has two radio buttons: "Distance" (selected) and "Time (sec)". Below this are two more input fields: "Interval(x/y):" with "5.000" and "Interval(z):" with "10.000". At the bottom, there are three checkboxes: "Offset", "Rotate", and "Draw Linework", all of which are currently unchecked. Finally, there are three buttons at the very bottom: "Start (F1)", "Stop (F2)", and "Exit".



The new points are both stored to the current coordinate file and drawn in the drawing.

When using GPS, if the RMS values of the position read are above the tolerance set in Configure Field, then the point will not be stored.

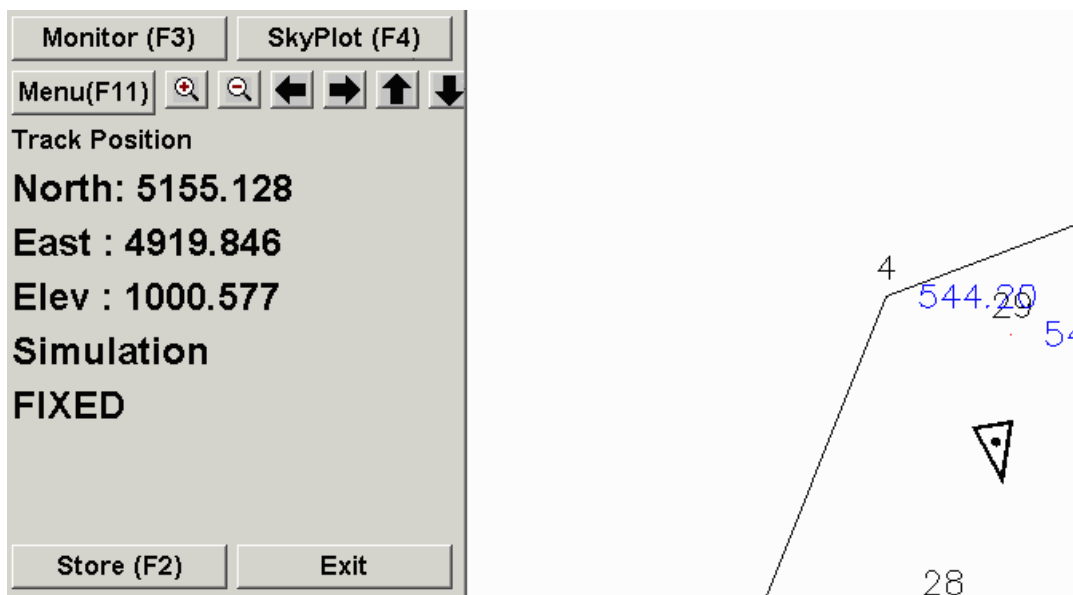
Carlson Field will continue to record points until you click on *Stop*.

Track Position

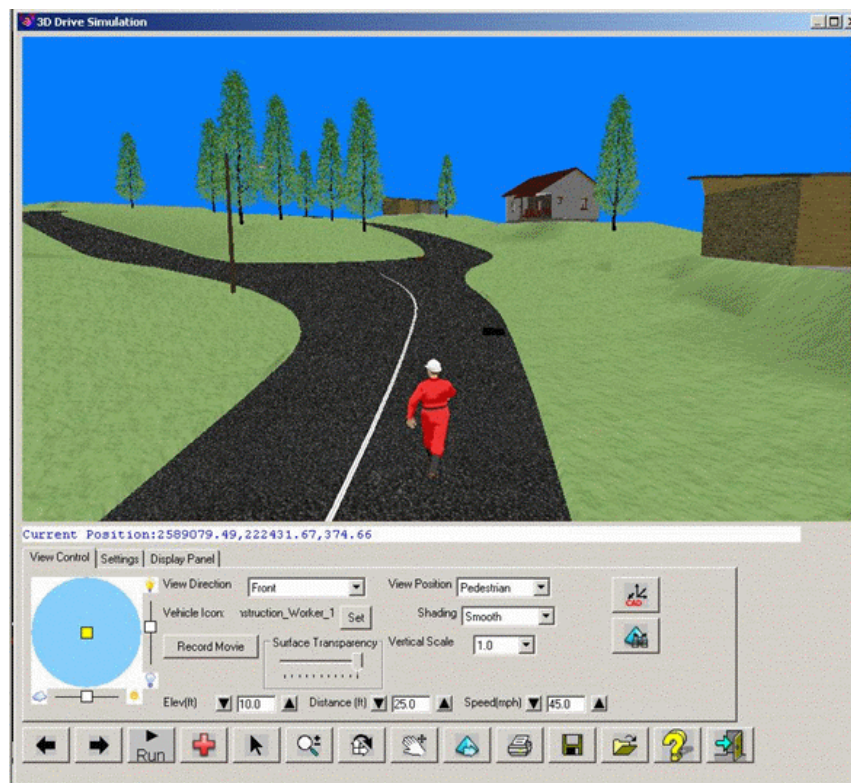
This command shows your current position in either the drawing view or a 3D view. For the drawing view, the coordinates of your current position are shown in a dialog and an arrow icon is shown in the drawing view. For the 3D view, use the Run button to get position updates and use the Stop button to stop updates and use the viewer controls to change to view. When the tablet computer has tilt and compass sensors, the 3D view uses these sensors to control the view direction.

This command only applies to GPS and robotic total stations. As you move along, the arrow icon will move through your drawing showing your position in real-time. If the arrow icon gets near the edge of the screen, Carlson Field will automatically pan over.

A dialog box also appears in Track mode. The dialog shows your current northing, easting and elevation. For GPS mode, the dialog displays the *HRMS* and *VRMS* values and solution status. There are buttons to take you to *Monitor* and *Skyplot*. There is also a *Store* button which will store your current location as a point and plot it, similar to the *Point Store* function.



Track Position in Drawing view mode



Track Position in 3D view mode

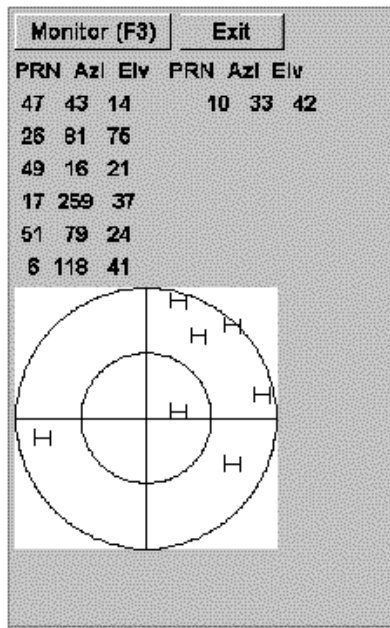
Satellite SkyPlot

When using GPS, it is important to know how many satellites you are tracking and their position in the sky. *Satellite Skyplot's* visual and graphical screen aids in identifying when satellites are being masked by surrounding structures, trees and mountains. Satellites close to the horizon, under fifteen degrees, are less helpful resolving the rover position because of extra atmospheric interference. If there are too few satellites present, the receiver will be unable to resolve its position. Typically five satellites are required to resolve position and four are needed to maintain locked solution. *Satellite Skyplot* can be an invaluable tool to help you monitor the current satellite configuration.

The skyplot screen appears at left. The top half of this window displays the visible satellite information in chart form. *PRN* is the satellite identification number. *Azi* is an abbreviation for azimuth; the horizontal angle from due north, in degrees measured clockwise, to the satellite position (0 to 360 degrees). *Elv* is an abbreviation for elevation; the vertical angle above the horizon where the satellite can be found (0 to 90 degrees). One entry appears for each satellite that the receiver is tracking.

The image on the lower half of the window displays the same information graphically. It shows a map of the sky with North at the top, East to the right. The centerpoint, where the lines cross, is straight up. Each satellite appears as a symbol resembling an "H". As you can see, most of the visible satellites were in the Northeast when this image was captured. The inner circle represents an elevation of sixty degrees. The outer circle is the horizon. Roughly speaking, any "H" touching this circle is too low in the sky to be of much use. For GPS receivers that support GLONAS satellites, Skyplot will show these satellites with a "G" symbol.

For some types of GPS receivers, the receiver will report which satellites are being used for calculating the position and which are only being tracked. A satellite might be only tracked and not part of the solution if the satellite is too low on the horizon or when the signal is not clear. The skyplot will highlight the satellites that are part of the calculations.



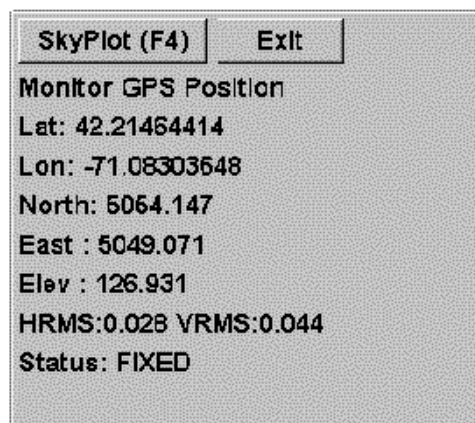
Monitor GPS Position

This command reports the current GPS Lat/Lon, local coordinates and GPS solution status. The latitude and longitude are reported in the DD.MMSSSSSS format. In this example, the latitude is 42 degrees, 21minutes, 46.4414 seconds north. The longitude is 71 degrees, 8 minutes and 31.5699 seconds west. Negative longitudes indicate longitudes west.

The next three items are state plane or local coordinates depending on the transformation in the Align Local Coordinates command. The *HRMS* and *VRMS* are measures of the reliability of the position that the receiver has calculated. They correspond to the position horizontally and vertically, respectively. If the receiver is autonomous, not receiving corrections from a base, the RMS can be up to a few hundred feet. If this rover is computing a "Fixed" position, the RMS values should be less than one foot, probably close to a tenth of a foot. If the receiver loses the fix and becomes "Float", the RMS values will jump to between one and ten feet.

Depending on the type of GPS receiver, the Monitor screen will also show more values like radio link status and receiver battery status.

The *Skyplot* button will jump you to that window so you can see the satellites the receiver is using.



Benchmark

This command takes a measurement to a benchmark point with a known elevation in order to calculate the elevation at the occupied point. This command only applies to total stations.

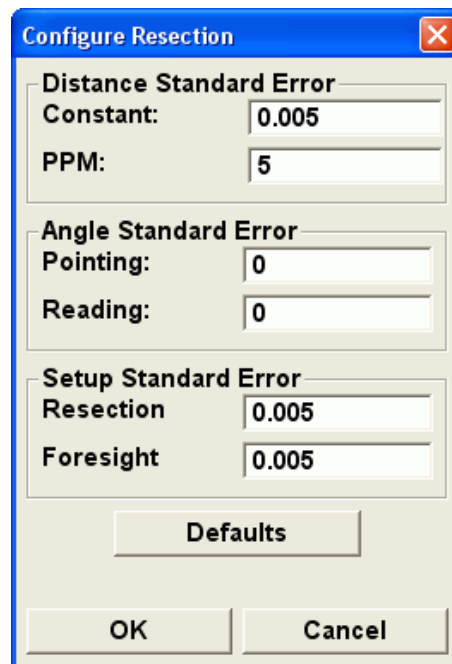
In the Benchmark dialog, fill in the instrument and rod heights. The benchmark elevation is specified in the Target Elevation field. This field can be filled out by entering a target point number which reads the elevation from the current coordinate file for the specified point. Or you can simply type in the target elevation directly. There is a choice between calculating the occupied point elevation or the instrument height. For calculating the instrument height, you need to enter the occupied point elevation. When calculating the occupied point elevation, there is an option to store this elevation to the coordinate file for the occupied point number. When all the options are set and the target benchmark is sighted, pick the Read button to take a measurement. After the reading, the program will display in the dialog the calculated occupied point elevation or instrument height depending on the calculation mode.

Resection

Resection is used when setting up a Total Station on an unknown point. The occupied point coordinates are calculated from two or more angle and distance measurements to known reference points. This command only applies to total stations.

The dialog above has options for *EDM Mode* and *Robotics*. These options will only show up for supported total stations (ie. robots, etc).

Configure Std Errors can be used to set standard errors for the measurements, as shown below. These settings allow you to take advantage of network least squares and weighting in the resection calculation. The default values are shown below:



Distance Standard Error	
Constant:	0.005
PPM:	5

Angle Standard Error	
Pointing:	0
Reading:	0

Setup Standard Error	
Resection	0.005
Foresight	0.005

Defaults

OK Cancel

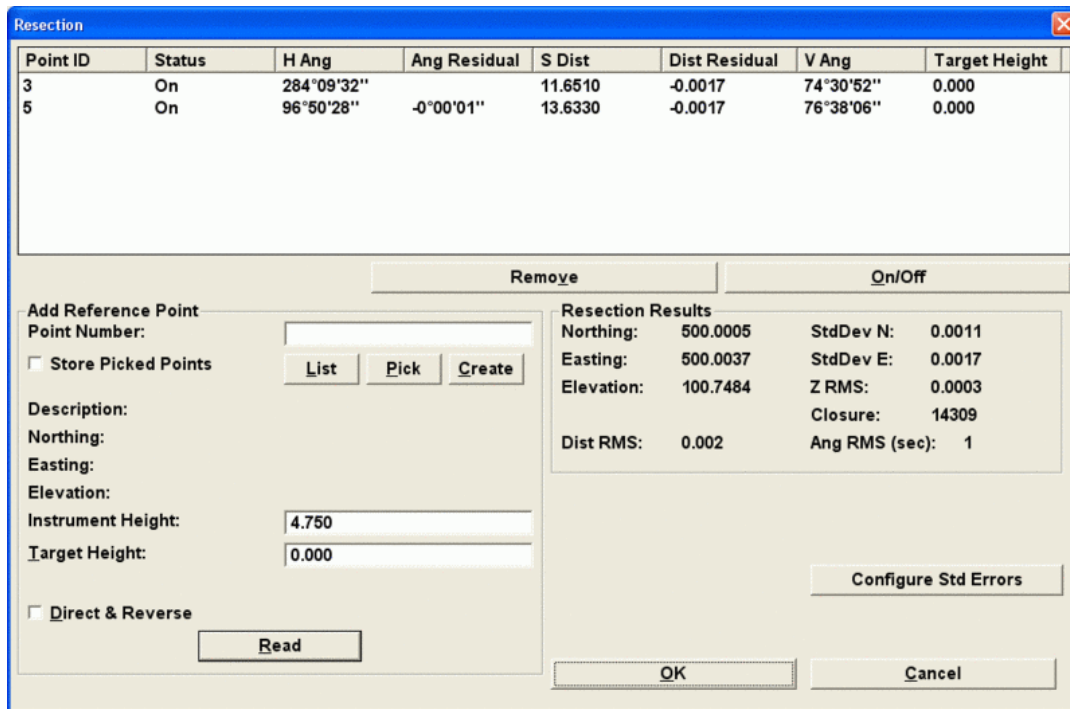
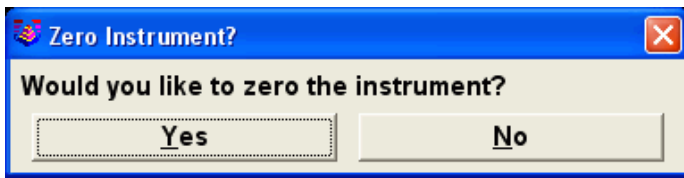
Reference points are distinct known points that are measured to calculate the unknown occupied point. Direct and Reverse measurements are optional by using the toggle *Direct and Reverse* located in the *Add Reference Point* section.

Reference points can be specified in several ways:

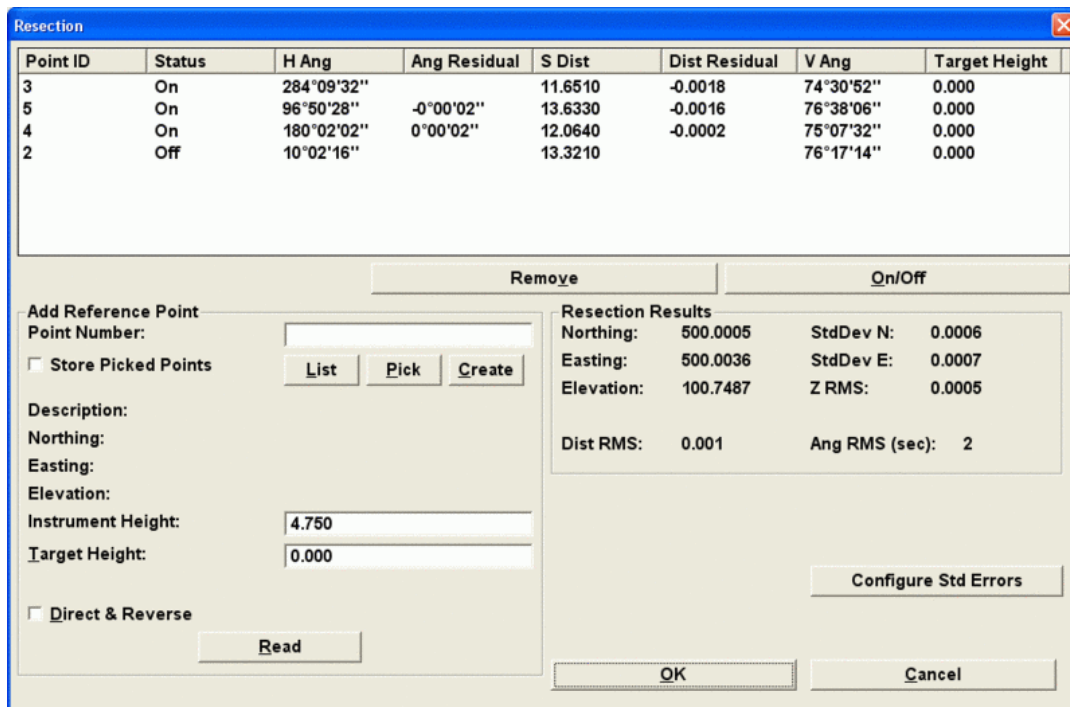
1. Manually entered point number
2. Selected from a list of points stored in the coordinate file (via *List* button)
3. Picked from the CAD screen (via *Pick* button)
4. Manually entered coordinates (via *Create* button)

There are input boxes for the target (or rod) height, and instrument height. The *Store Picked Points* toggle controls whether or not the user is prompted to store screen picked points that are not found in the coordinate file.

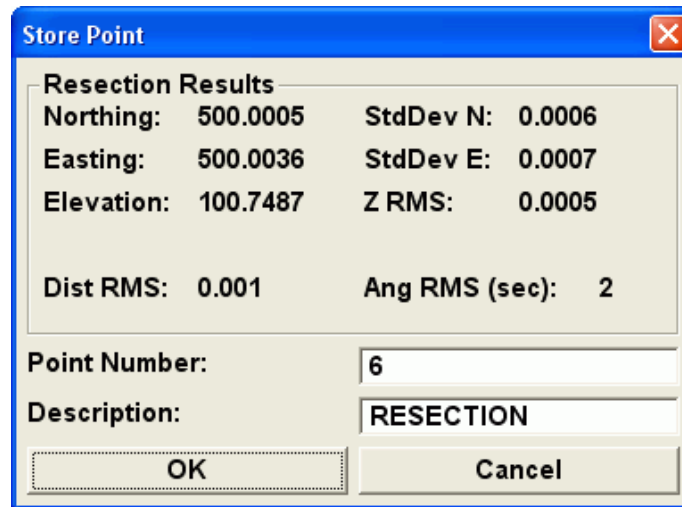
After a reference point is chosen, it's description and coordinate information are displayed. Press the *Read* button to initiate the measurement for the currently displayed reference point. After a successful measurement, the user will be prompted to verify the measurement information. This gives the user a chance to verify the target (or rod) height, and the option to discard the data and take another measurement. The user will be prompted to determine whether or not to zero the instrument on the first reference point.



A minimum of two distinct reference points and measurements are required to calculate the occupied point. The calculated occupied point information is displayed in the *Resection Results* section. The input data is displayed in a list structure and the user has the option to turn certain measurement on or off using the *On/Off* button or remove an unwanted measurement using the *Remove* button. The resection results are calculated automatically anytime a reference point measurement is added, a list item is removed, or a list item is toggled on or off. A resection with three inputs 'on' and one input 'off' is displayed below:



The *Resection Results* section shows the calculated occupied point's northing, easting, and elevation and the difference between the calculated coordinate and the individual solutions as the residuals. The residual indicate the quality of the data. High residuals suggest a problem with the input data. When you press the *OK* button, you will be prompted to verify the calculated occupied point. If accepted, the calculated point will be stored to the coordinate file and you will be setup on the calculated point backsighting the first point shot in the resection.



The 'Store Point' dialog box displays the following information:

Resection Results			
Northing:	500.0005	StdDev N:	0.0006
Easting:	500.0036	StdDev E:	0.0007
Elevation:	100.7487	Z RMS:	0.0005
Dist RMS:	0.001	Ang RMS (sec):	2

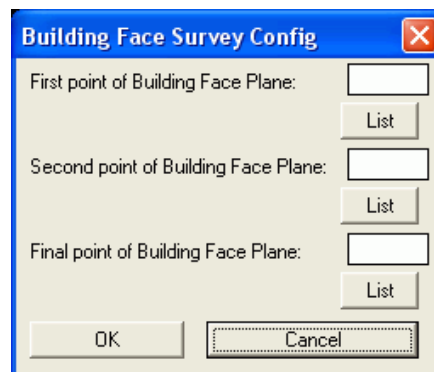
Point Number:

Description:

Buttons: OK, Cancel

Building Face Surface

Used to project all points onto a surface or plane. Upon executing the function, a menu will open, prompting the selection of three points to define the plane/surface. Note that there must already be three points along the plane in the CRD file in order for this function to work properly. After selecting the three points (the "List" buttons will bring up a list of available points), select "OK" to proceed. A screen similar to the Store Points dialog will now open. Every point which is read will be plotted along the plane defined by the three points selected, even if it is at a different distance. When finished, simply exit out of the menu as with any other function.



The 'Building Face Survey Config' dialog box contains the following fields and buttons:

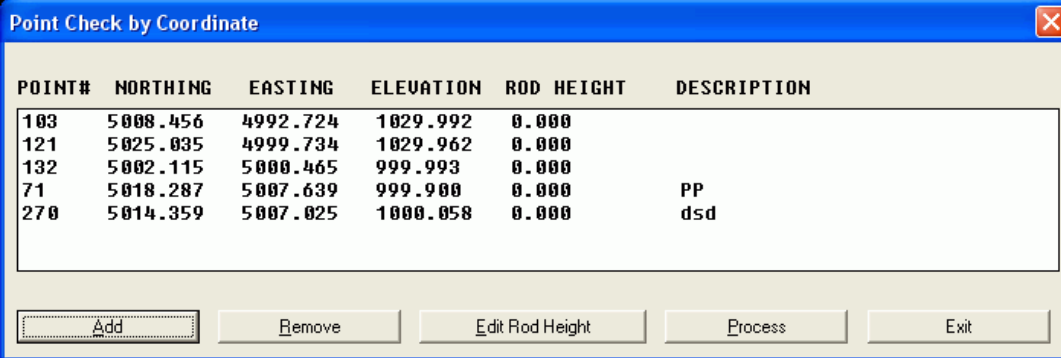
- First point of Building Face Plane: List
- Second point of Building Face Plane: List
- Final point of Building Face Plane: List
- Buttons: OK, Cancel

Pattern Point Survey

Used with a reflectorless Total Station. This function is used to shoot a regular, rectangular "pattern" of points across an area. It is useful when periodic measurements of an area are required. Upon starting the function, a query box will ask for two point defining a rectangle, the lower left corner and the upper right corner. For each of these, aim the gun at the corner of the area to be scanned and click "Read". After reading both points, a menu will prompt for several other parameters. Enter the first point number to shoot, as well as any desired description for the points, and both horizontal and vertical increments. These are angle increments, given in seconds. Once all of the above is entered, select "OK" to begin the survey. The total station will now begin turning automatically to the bottom-left corner of the area, and will begin shooting points. Upon reaching the right-hand limit of the area, it will begin a new row of points, starting at the left.

Point Check By Robotics

This command works with robotic Total Stations made by Leica, Topcon or Geodimeter. This function is used to shoot and record a series of known target points. Before running this command, the instrument setup must be set (occupy point, backsight) with the Equipment Setup commands. After selecting Point Check by Robotics you will be prompted with a dialog box. Choose the points you want to check and click process. The Total Station will then go from point to point and take new measurements. When it is all done, a report will be given with the new measurements and any deviation.



POINT#	NORTHING	EASTING	ELEVATION	ROD HEIGHT	DESCRIPTION
103	5008.456	4992.724	1029.992	0.000	
121	5025.035	4999.734	1029.962	0.000	
132	5002.115	5000.465	999.993	0.000	
71	5018.287	5007.639	999.900	0.000	PP
270	5014.359	5007.025	1000.058	0.000	dsd

Pipe Network

This set of commands is to measure pipe networks and store as-built values. The pipe network model is created by the Carlson Hydrology module and is stored in a .sew file.

Set Model

This command selects the pipe network model file (.sew) to process.

Prerequisite: Pipe network model file .sew

Keyboard Command: set_sew

Plan View Label Settings

This command defines how to draw the pipe network. See the Hydrology chapter for a description of these options. All the options for the Field version of this command are the same except a few on the General Settings tab. The Use As-Built Values option uses the measured values for the labels instead of the design values from the original .sew file. The Measured Color sets the label color for values that have been measured. The Unmeasured Color sets the label color for values that are not yet measured.

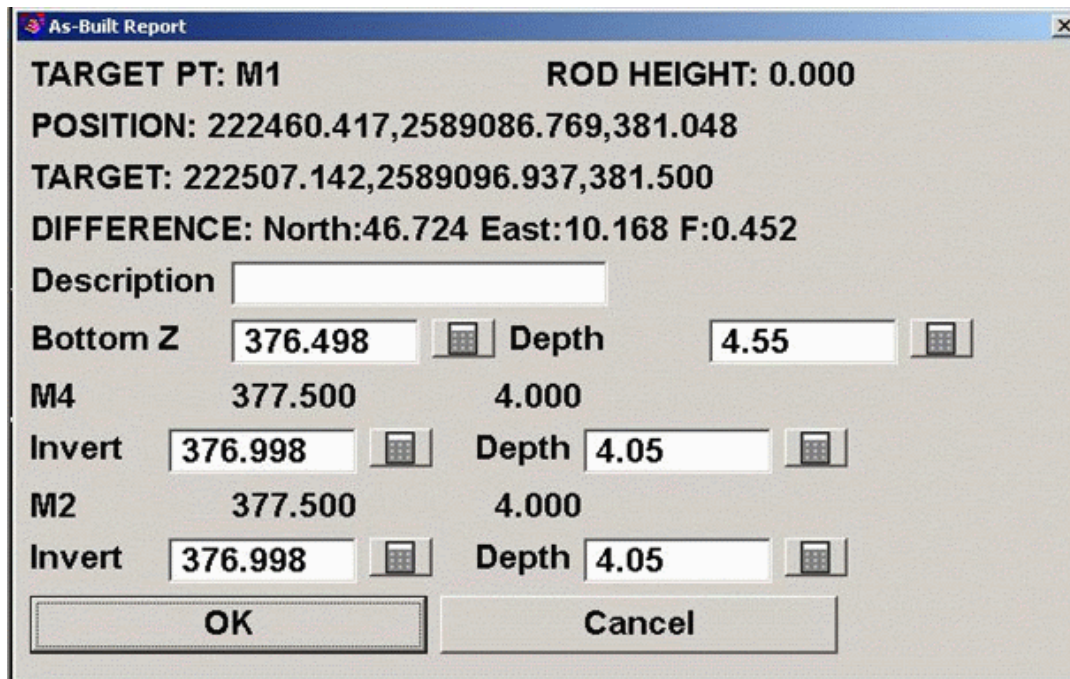
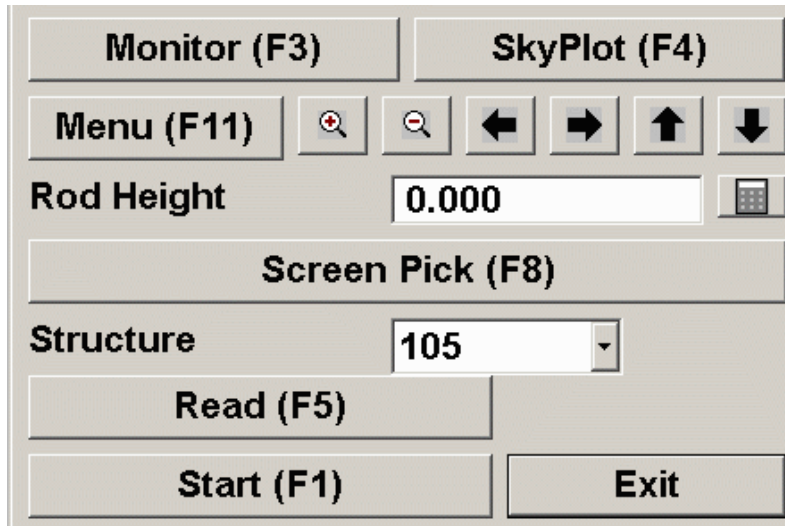
Prerequisite: None

Keyboard Command: setup_sew

Measure

This command measures as-built structures. On the first dialog, select the structure to measure by either Screen Pick where you pick on the drawing or from the pull-down list of structure names. Then pick the Start button to begin measuring. Then the program takes readings from the instrument and reports the difference and direction to the structure like the Stakeout command. Once at the structure, pick the Store button. Then the As-Built Report

dialog shows the difference between the structure design and measured positions. In this dialog, you can measure the bottom of the structure either by entering a depth or bottom elevation. Likewise you can measure the inverts for pipe connection by either depth or bottom elevation.



Prerequisite: set_sew

Keyboard Command: stake_sew

Edit

This command allows to edit the bottom elevation and inverts of a structure. After selecting the structure to edit, the As-Built Report dialog is shown for making the edits.

Prerequisite: stake_sew

Keyboard Command: edit_sew

Report Structures

This command reports structure parameters including the design and measured positions and differences.

Prerequisite: stake_sew

Keyboard Command: report_struct_sew

Report Pipes

This command reports pipe parameters including the design and measured slopes.

Prerequisite: stake_sew

Keyboard Command: report_pipe_sew

Draw Plan View

This command draws lines, labels and symbols for the pipe network. See the Hydrology chapter for more info.

Prerequisite: set_sew

Keyboard Command: plan_sew

Draw 3D

This command draws the pipes and structures as 3D Faces. See the Hydrology chapter for more info.

Prerequisite: set_sew

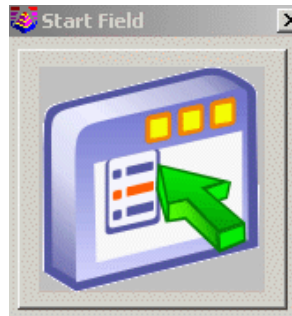
Keyboard Command: 3d_sew

Carlson Field Icon Menu

The Carlson Field Icon Menu lets you select Carlson Field functions by pressing a function key F1-F10 or by picking the icon button. The set of commands that are available in this menu depends of the type of survey equipment that you are configured to. Before running these Carlson Field functions, you need to run Configure Field to set the equipment type and communication parameters.

There are two ways to bring up the icon menu. One way is by picking the Start Carlson Field icon. This start icon is displayed in the lower right of your screen when the Show Startup Icon option is on as set in Configure Field under General Settings. You can close the Start Carlson Field icon for the current drawing session by clicking the X in the icon title bar. To bring back the Start Carlson Field icon you can use the F11 key. The Start Carlson Field icon is only displayed when no commands are running. The other way to show the Carlson Field function menu is to pick the Menu(F11) button while running other Carlson Field commands. This method allows you to switch between Carlson Field functions without having to exit back to the CAD menu. For example, you can switch from

Point Store directly to Stakeout.



This is the function menu when using GPS equipment:



Typical Alignment Scenarios

Scenario: New site. In this case, there are no established coordinates on the site.

Alignment: Choose a point on site and do a one point alignment. For the local alignment point, enter the coordinates that you would like to use (ie 5000,5000,100). Under Configure Field>GPS Settings, The One Pt Align Azimuth option chooses between using true north (geodetic) or state plane north (grid). To use real world ground distances, set the Project Scale

Factor under Configure Field>GPS Settings. Otherwise the default scale factor of 1.0 will collect points on state plane distances.

Scenario: One known state plane coordinate and you want to work in the state plane coordinate system.

Alignment: Either setup the base over the known state plane coordinate or do a one point alignment on this known state plane point. In Configure Field>GPS Settings, set the One Point Align Azimuth to Grid and set the scale factor to 1.0.

Scenario: Multiple known control points.

Alignment: Choose two or more control points to align to. It is best to use control points around the perimeter of the site. Use as many control points as are available or enough to envelope the site. In Configure Field>GPS Settings, set the Transformation to Plane Similarity to fit the GPS points onto the control points and set the Project Scale Factor to 1.0. After making the alignment, stake out another control point (ideally one the is not used in the alignment) to make sure the alignment is good.

Surface Menu

The Surface pull-down menu has Elevation Difference and surface creation commands that are described in the Civil Design manual.



Elevation Difference

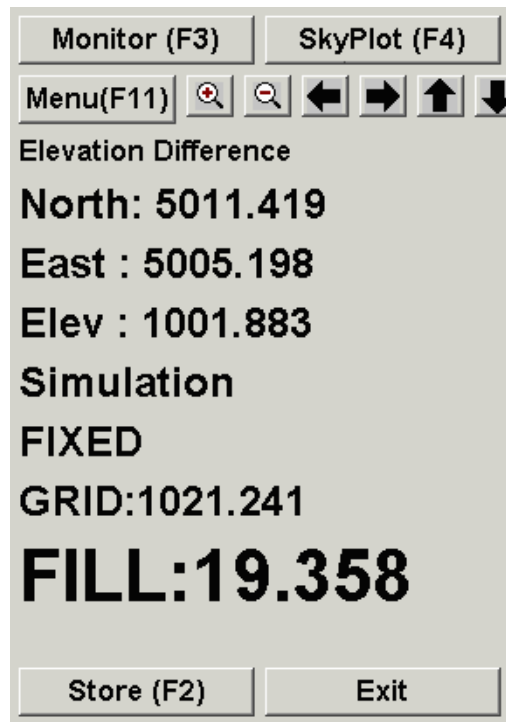
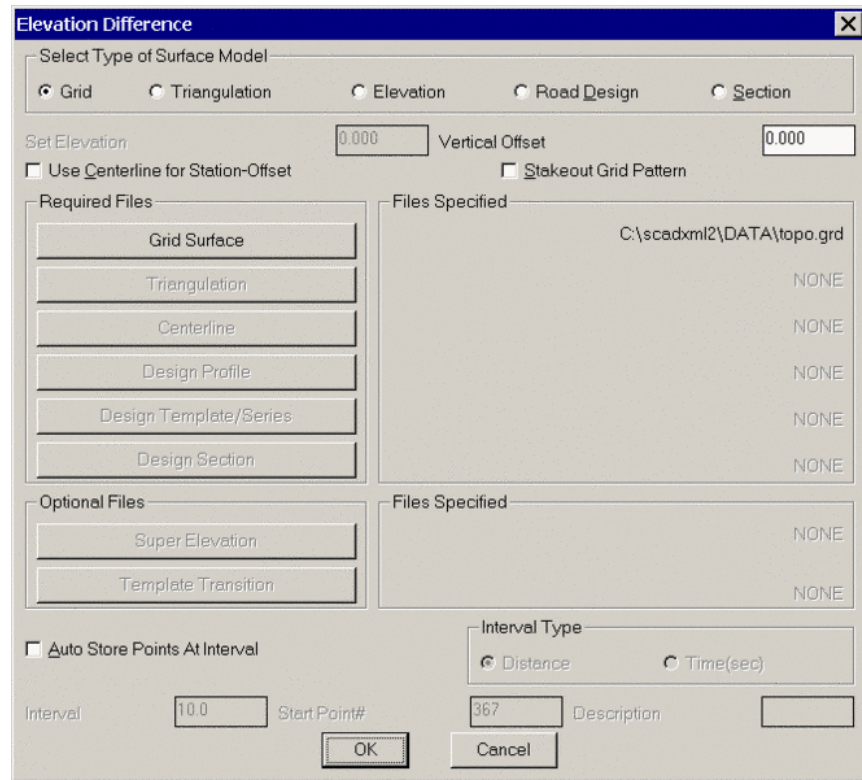
This command reports the cut or fill between your current position and a design surface. The design surface can be one flat elevation, a grid file, a triangulation file, a road design file, or a section file.

The type of design surface is set in the dialog shown. The Vertical Offset in this dialog can be used to modify the design surface by adding this value to the design surface. For example, if you have a design surface for the top of a road and you want to get cut/fill values to a 1.5 subgrade, then enter -1.5 in the Vertical Offset field. The Use Centerline For Station-Offset option will report the station-offset of your current position in addition to the cut/fill. When this option is active, the program will prompt you for the centerline file (.CL) to reference. For GPS and robotic total stations, the Auto Store Points At Interval will creates points whenever your position moves by more than the specified distance or time interval. This option is similar to the Auto Points At Interval command with the addition that the default description will include to cut/fill to the design surface. When all the options are set, pick OK and the program will then prompt you for a grid file or triangulation file if you have selected these types of design surface.

Elevation Difference with GPS

Carlson Field will continually read your current position from the GPS receiver. A dialog box appears displaying your current position. Carlson Field finds the design elevation for this point and compares it to the elevation being reported by the GPS receiver. It then tells you how much cut or fill is required to reach the design elevation from your current position. An arrow icon will appear on the drawing showing your location. You can move around the

site while in *Elevation Difference* mode and Carlson Field will report the necessary cut or fill in real-time. If you move off the area covered by the design surface, then the program will stop reporting cut/fill and instead will report "Off Surface". When using a grid or triangulation surface, the bottom of the dialog shows a profile of the surface in real-time in the direction of your movement.



The Store button will create a point at the current position. The default description will include the current cut/fill. When Store is selected, a dialog box will appear for entering the point number and description.

Elevation Difference with Total Stations

Elevation Difference uses a dialog box that is very similar to the *Point Store* dialog. Under the Setup button, make sure that the occupied point, backsight and instrument height are set. Then have your rodman set the prism over the point you are interested in. Pick *Read(F1)* or *Read & Store(F5)* and the total station will take a shot.

After the shot is taken, the dialog box looks like the one at right. Carlson Field found the design elevation for this point (557.535) and compared it to the actual current elevation (530.0). Based on the current and design elevations, Carlson Field reports to how much cut or fill is required to get to design elevation. In this case, it is fill 27.535. The cut/fill also appears in the *Desc* box. If you click *Store*, Carlson Field will record this point and plot it on the drawing, including the *Desc* as a label.

Setup (F3)	
Menu(F11)	[Icons]
H:123°00'00" V:90°00'00" SD:30.000	
OC:1 BkAz:0.0000 HI:0.000	
Point Number:	2
Rod Height:	0.000
Northing:	4983.661
Easting:	5025.160
Elevation:	530.000
Desc:	FILL:27.535
Options (F6)	Code (F7)
GRID:557.535	
FILL:27.535	
Read (F1)	Store (F2)
Read & Store (F5)	Exit

Pulldown Menu Location: DTM

Prerequisite: None

Keyboard Command: grd_gps

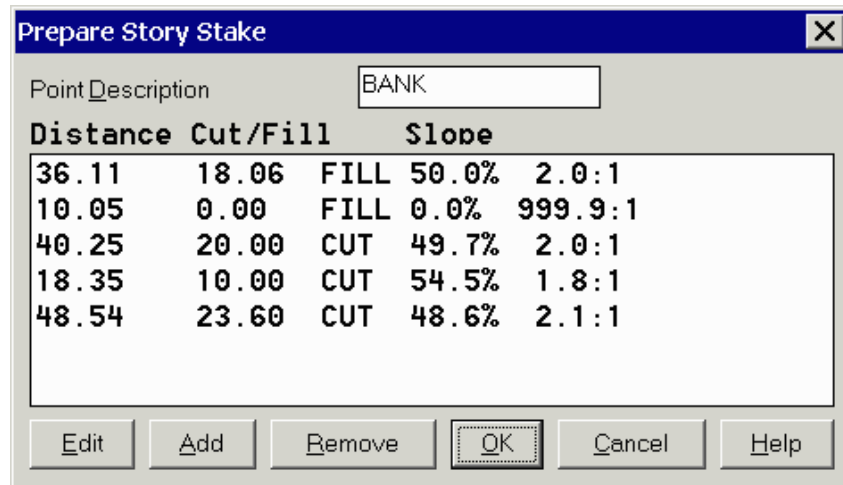
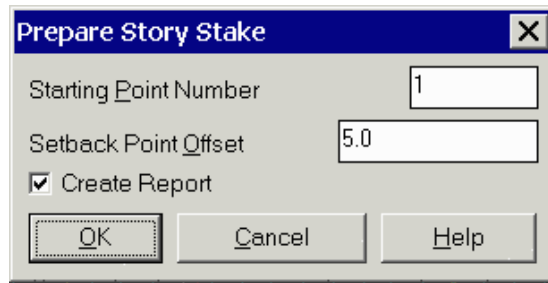
Prepare Story Stake

This command creates points with cut/fill information stored in the note fields for the points. Beginning at a point and facing a specified direction, the cut/fill information describes a design surface that is defined by contours and 3D polylines in the drawing. The program prompts you to pick the starting point followed by a direction point. Then the intersections for all the contours and 3D polylines between these two points are calculated and the resulting horizontal distances and slopes are shown in a dialog. In this dialog, you can edit, add or remove these slopes descriptions. The Point Description can also be specified. When OK is clicked, a point in the coordinate file is created at the starting point with this information stored in the note file. An offset point is also created at the specified offset distance back from the starting point. At the end of Prepare Story Stake, a report of all the created points and the corresponding cut/fill data is shown if the Create Report option was set. Prepare Story Stake does not draw the points in the drawing. These points can be drawn using the Draw-Locate Points command.

The cut/fill information in the note file can be used in the Stakeout routine. In Configure Field>Stakeout Settings there is an option to Display Point Notes in Stakeout Report. With this option active, the cut/fill data in the note file will be displayed when the point is staked out.

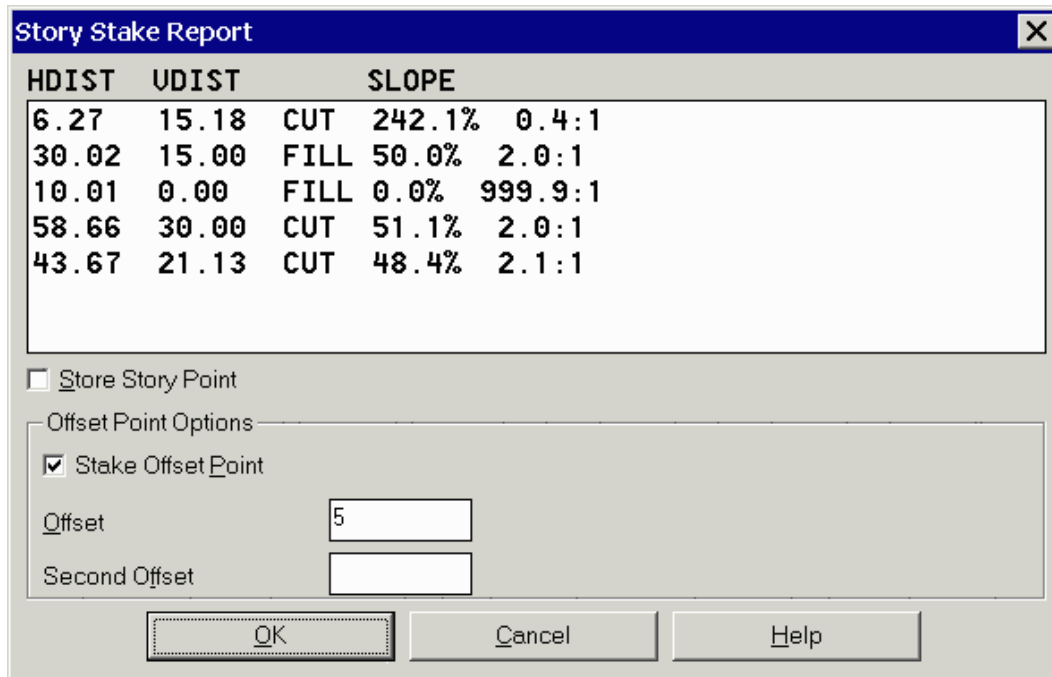
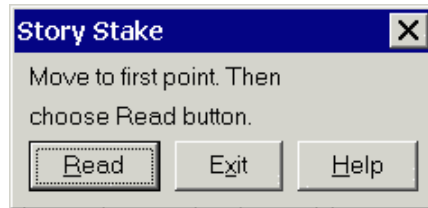
Prepare Story Stake is not a prerequisite for Story Stake By Points or Story Stake By Polyline. In fact, working in

combination with Stakeout, Prepare Story Stake is an alternative to these other routines.



Story Stake By Points

This command creates a report of cut/fill slopes and distances of a design surface from a starting point. First you move to the starting point and then take a reading from the instrument to get the starting point coordinates. This starting position is shown in the drawing. Next you pick a point in the drawing to define the direction. The drawing should contain the design surface entities. The program will then calculate all the intersections with contours and 3D polylines between these two points. The resulting horizontal distances and slopes are shown in a report dialog. From this dialog, there is an option to stakeout one or two offset points set back from the starting point at the specified offsets.



Story Stake Along Polyline

This command creates a report of cut/fill slopes and distances of a line across a design surface. The line is defined as perpendicular from a polyline starting at a specified station and going a specified distance. The drawing should contain design surface entities. The program will calculate all the intersections with drawing contours and 3D polylines along the line. For example, the polyline could be a toe of slope and this routine would be used to create story stakes at an interval along this polyline.

The routine starts by selecting a 3D polyline from the drawing. Then there is a dialog to specify the settings. The Station is the distance along the polyline for the starting point of the story stake. The Next Interval is used to increment the station for the next default station. The Story Offset is the length of the story stake line from the starting point. To have the story stake line go perpendicular right from the polyline, enter a positive offset value. To go left, enter a negative offset. The Read Current Position button will take a measurement from the instrument to find the station of your current position. This current station is put in the Station field. The Pick Point button will prompt you to pick a point in the drawing view. The station of this point is used to fill out the Station field.

After specifying the stakeout station and story offset, then program runs the stakeout routine to guide you to that station on the polyline. When that point is staked, the program calculates the story stake and the resulting horizontal distances, cut/fill and slopes are shown in a report dialog. From this dialog, there is an option to stakeout one or two offset points set back from the starting point at the specified offsets.

Story Stakeout [X]

Station:

Next Interval:

Story Offset:

Story Stake Report [X]

HDIST	UDIST	SLOPE		
6.27	15.18	CUT	242.1%	0.4:1
30.02	15.00	FILL	50.0%	2.0:1
10.01	0.00	FILL	0.0%	999.9:1
58.66	30.00	CUT	51.1%	2.0:1
43.67	21.13	CUT	48.4%	2.1:1

Store Story Point

Offset Point Options

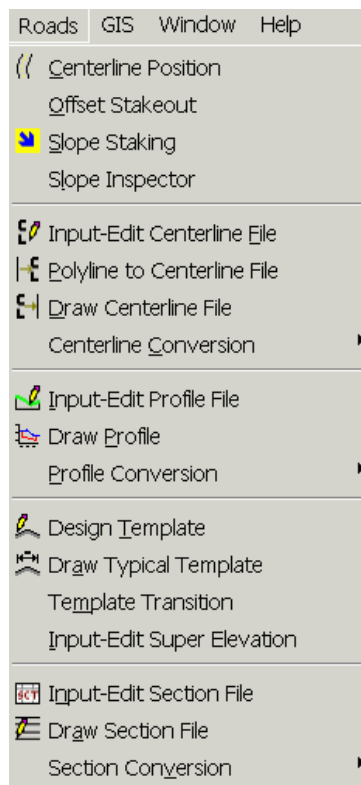
Stake Offset Point

Offset

Second Offset

Roads Menu

The Roads pull-down menu has commands for road stakeout and preparing the road design files. Most of these commands are described in the Civil Design manual.



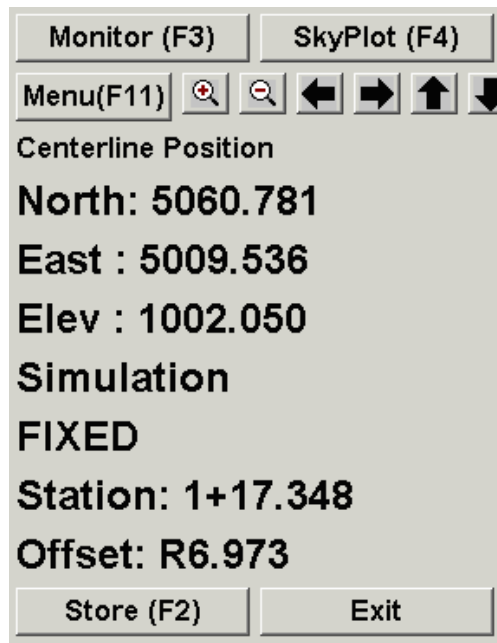
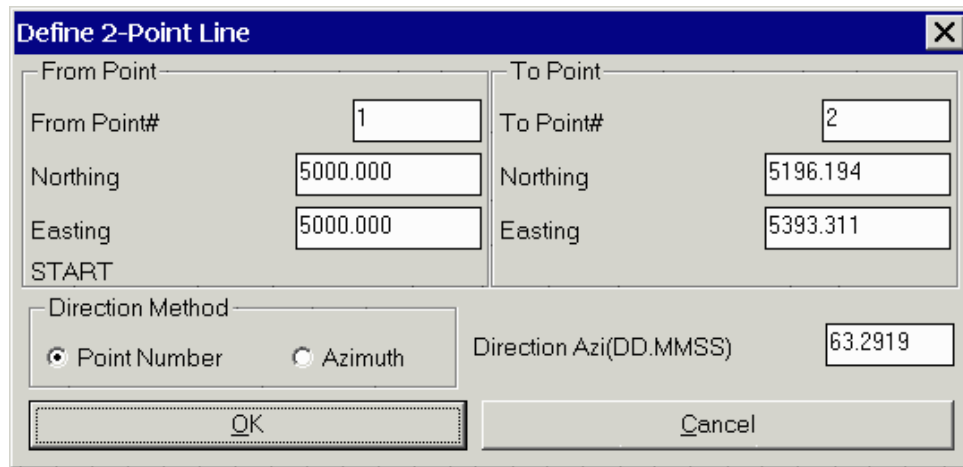
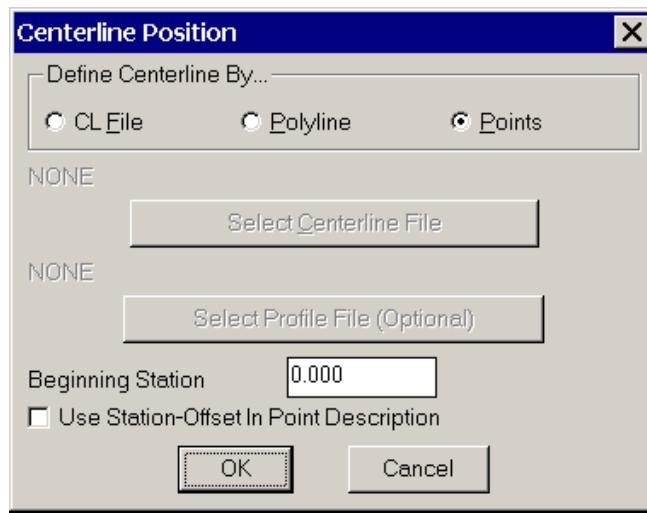
Centerline Position

This function determines the station and offset relative to a centerline for a point. The centerline can be defined by a centerline file (.CL), a points, or by a polyline. The centerline file can be created with commands in the Roads menu. One advantage of centerline files is that it allows you to use profile files which can report cut/fills. For the points method, you can either give two points or a starting point and azimuth. The points can be defined by a point number from the current coordinate file or by simply entering the northing and easting. The first dialog for Centerline Position has the choice for centerline file or points method. With the centerline file option, the dialog shows the last centerline file name used. If this file is correct, then click OK. Otherwise use the Select button to choose the centerline file name.

Light bars are useful for left-right guidance. To enable the light bar option go to Configure Field, then to Centerline Position Settings and pick Use Light Bar.

Centerline Position with GPS

Carlson Field will continually read your current position from the GPS receiver. A dialog box appears displaying your current position. Carlson Field finds and displays in the dialog the station/offset for this point.



An arrow icon will appear on the drawing showing your location. You can move along the centerline and Carlson Field will report the station/offset in real-time. If you move beyond the ends of the centerline, then the program will stop reporting station/offset and instead will report "Off CL".

The Store button will create a point at the current position. The default description will include the current sta-

tion/offset. When Store is selected, a dialog box will appear for entering the point number and description.

Centerline Position with Total Stations

Centerline Position uses a dialog box that is very similar to the *Point Store* dialog. Under the Setup button, make sure that the occupied point, backsight and instrument height are set. Then have your rodman set the prism over the point you are interested in. Pick *Read(F1)* or *Read & Store(F5)* and the total station will take a shot.

After the shot is taken, the dialog box looks like the one at right. Carlson Field reports the current coordinates and the station/offset. The station/offset also appears in the *Desc* box. If you click *Store*, Carlson Field will record this point and plot it on the drawing, including the *Desc* as a label.

The screenshot shows a dialog box titled "Setup (F3)". At the top, there is a "Menu(F11)" button and a set of navigation icons (magnifying glass, zoom out, left arrow, right arrow, up arrow, down arrow). Below these are several lines of text: "H:12°00'00" V:90°00'00" SD:90.000", "OC:1 BkAz:0.0000 HI:0.000", "Point Number: 1131", "Rod Height: 0.000", "Northing: 5088.033", "Easting: 5018.712", "Elevation: -0.000". A "Desc:" field contains the text "1+43.115 L5.681". Below the description are two buttons: "Options (F6)" and "Code (F7)". Further down, it shows "Station: 1+43.115" and "Offset: L5.681 IN: 11.299". At the bottom, there are four buttons: "Read (F1)", "Store (F2)", "Read & Store (F5)", and "Exit".

Pulldown Menu Location: Roads

Prerequisite: None

Offset Stakeout

This function stakeouts a point at a given station and offset of a centerline and reports the cut or fill to a design elevation. The centerline and design elevation can be defined by four methods as set in the dialog show. The Design Files method uses a centerline file (.CL) for the horizontal alignment and a profile file (.PRO) for the vertical alignment. A template file (.TPL) for the design cross section is optional for the cross slope. Without a template file, the program will use the elevation of the profile along the centerline. A superelevation file (.SUP) and a template transition file are optional. These design files can be created with the routines in the Roads menu. The Section File method uses a centerline file for the horizontal alignment and a section file (.SCT) for the design elevation. The section file consists of cross sections of offset/elevation points for a series of stations. Section files can be used instead of the Design Files method when a road design is too complicated to model using design files. For example, if the road contains special ditches at various offsets and varying lane widths, then it may be easier to enter a final section file than to define the template and template transitions. The Points method uses two points to define both the horizontal alignment and design elevations. The design elevation is linearly interpolated between the points. The points to used are specified in the next dialog by entering point numbers from the current coordinate file or by directly entering the coordinates. The 3D Polyline method uses a 3D polyline for both the horizontal and vertical alignment. With this method, the program will prompt you to select the 3D polyline from the drawing. For both the Points and 3D Polyline methods, you can specify the starting station of the horizontal alignment. When using the Design Files and Section File methods, the horizontal alignment starting station comes from the centerline file.

Offset Stakeout Settings [X]

Surface Definition Method

Design Files Section File Points 3D Polyline

Required Files

1> CenterLine for Job

2> Design Profile

3> Design Section File

Files Specified

D:\scdev\DATA\control.cl

C:\scdev\DATA\rd.pro

NONE

Optional Files

4> Design Template

4> Super Elevation

5> Template Transition

Files Specified

NONE

NONE

NONE

Beginning Station:

OK Cancel

Offset Stakeout [X]

Station: Station List

Next Interval:

Design Offset:

Stake Offset From Design: From CL:

Vertical Offset: Slope %:

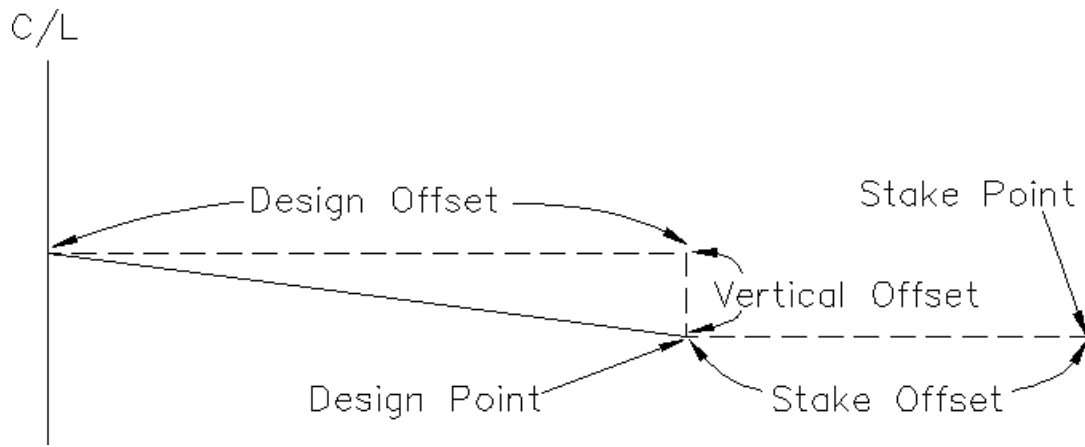
Side For Stakeout

Left Right

Read Current Position

Pick Point

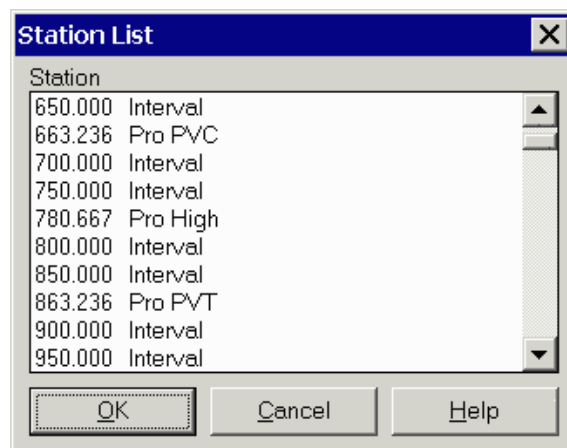
OK Exit Help



After specifying the offset stakeout method, Carlson Field prompts for the station and offset to stakeout as shown in the dialog. The station should be entered as a number without the "+" symbol. The Next Interval field is used to increment the stakeout station for the next stakeout point. In addition to incrementing to the next interval, Carlson Field will also pick up special profile or centerline points between intervals. Centerline special points include: start point, end point, curve (PC, PT) and spiral (TS, SC, CS, ST). Profile special points include: start, end, vertical curve (VC, VT), high points and low points. For example, if the current station is 100 and the interval is 50 and there is a centerline PC at 112.4, then the next station after 100 will be 112.5 followed by 150. The Station List button brings up a list of all the station intervals and special stations. You can select a station to stakeout by selecting the station from the list and pressing OK.

There are two offsets to allow for separate offsets for the design elevation and stake location. The Side For Stakeout toggle selects between left and right offsets. The Design Offset is where the stake point elevation is calculated. The Stake Offset determines X,Y position of the stakeout point by finding this offset at the stakeout station along the horizontal alignment. Having Design and Stake offsets applies, for example, to staking the back of a curb, where the Design Offset is 12, but the stake offset is 17 (5' behind the back of curb, with the elevation reference to the actual back of curb design elevation). The Stake Offset can be specified either as an offset from the design point or as an offset from the centerline. There is also an optional vertical offset that applies to the elevation of the design point. With the Design Files and Section File methods, the vertical offset works as an offset from the template or cross section surface. For example, a vertical offset of -0.5 could be used to stakeout the bottom of a 0.5 subgrade. With the 3D Polyline and Points methods, the vertical offset adjusts the elevation from the along the centerline at the stakeout station.

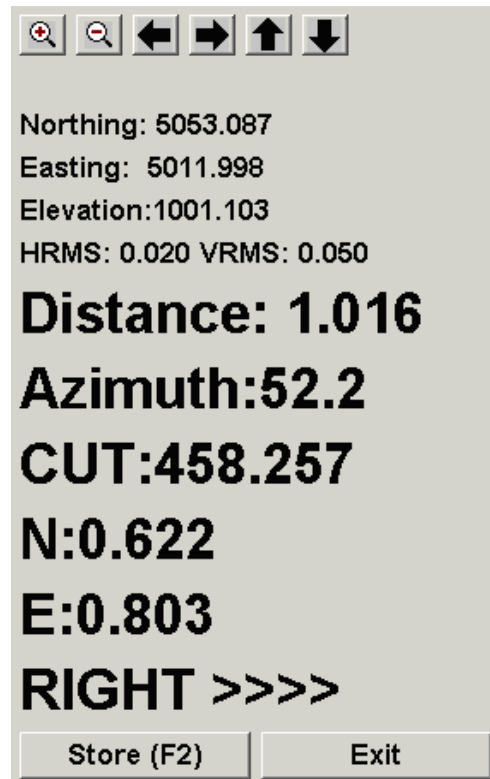
The Read Current Position button will take a measurement from the GPS or total station to find the station of your current position. This current station is put in the Station field.



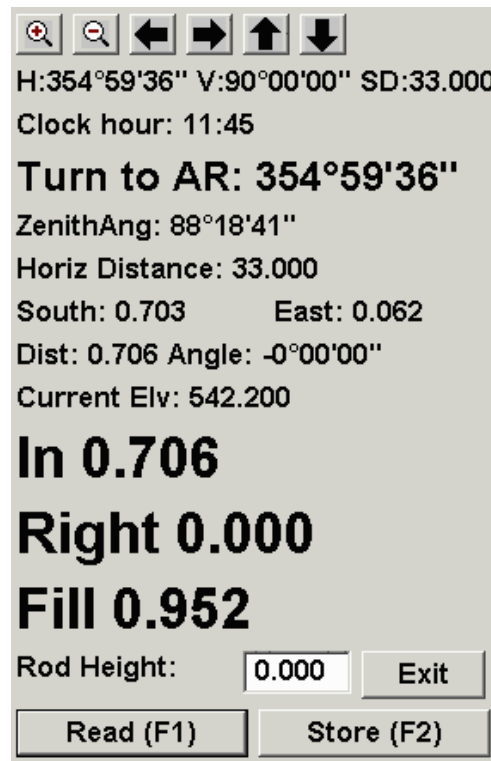
The Pick Point button will prompt you to pick a point in the drawing view. The station and offset of this point are used to fill out the Station and Offset fields.

After specifying the stakeout stations and offsets, Carlson Field uses the same stakeout function as used in the Stakeout command. This stakeout function guides you to the stakeout point and reports the cut/fill to the design elevation. You can store the stakeout point. When the stakeout is done, the station/offset dialog appears for staking the next point. Either enter the next station/offset or pick Exit to end Template Stakeout.

For total stations, you should run the Equipment Setup command before Template Stakeout to set the occupied point, backsight and instrument height.



Stakeout dialog for GPS



Stakeout dialog for total stations

Pulldown Menu Location: Roads

Prerequisite: None

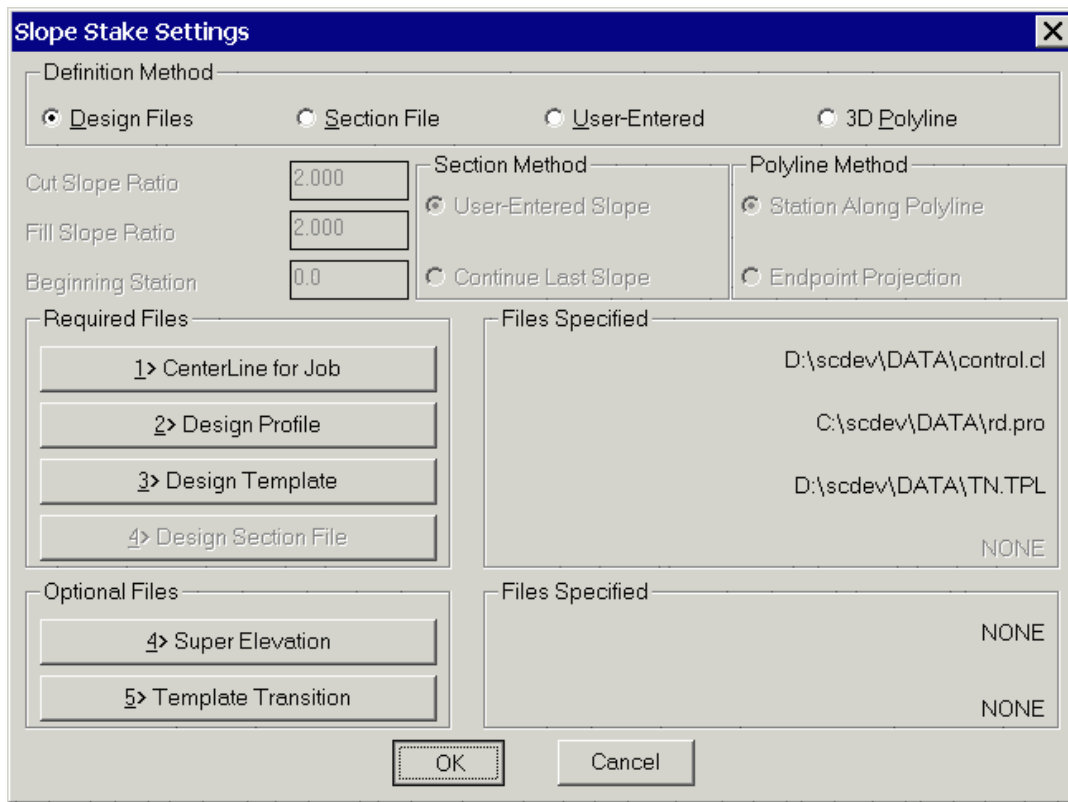
Slope Staking

This command guides you to the catch point where the cut/fill slope intersects the existing ground. Coordinates from the GPS receiver or total station are used to model the existing ground. There are four methods for defining the cut/fill slopes:

- Design Files
- Section File
- User Entry
- 3D Polyline

Design files include a centerline file (.cl), profile file (.pro) and template file (.tpl). The centerline defines the horizontal alignment, the profile defines the vertical alignment and the template defines the cross slopes and cut/fill slopes. Superelevation (.sup) and template transitions (.tpt) files can also be used. Using the design files, any station along the centerline can be slope staked. These design files can be created with commands in the Roads menu.

Section files (.sct) can be used instead of design files when the road is too complicated to model using design files. For example, if the road contains special ditches at various offsets and varying lane widths, then it may be easier to enter a final section file than to define the template and template transitions. A section file consists of offset-elevation points at different stations. At a minimum, each station should contain the pivot point offset-elevations. The slope staking routine will start the cut/fill slope from the furthest offset point in the section.



For example, when staking the right side, the right most offset will be used as the pivot point. The section file can optionally contain additional offsets such as centerline and edge of pavement. The program can then report the horizontal and vertical distances from the catch point to these additional offsets. The section pivot offsets can also be assigned a description which the program reports before starting the slope staking. For example, a pivot offset could be "2:1 from flat bottom ditch" which is reported to the operator. When using section files, a centerline file is also required to establish the horizontal alignment. Any station along the centerline can be slope staked because the program will interpolate between entered section stations. The cut/fill slopes from the section can be either User-Entered or Continue Last Slope. The User-Entered option will use the cut/fill slope ratios as entered in the dialog. The Continue Last Slope option will use the last two points in the section file as the cut/fill slope. This Continue Last Slope option applies to section files that contain pivot point to ground segments whereas the User-Entered option is for section files that end at the pivot points.

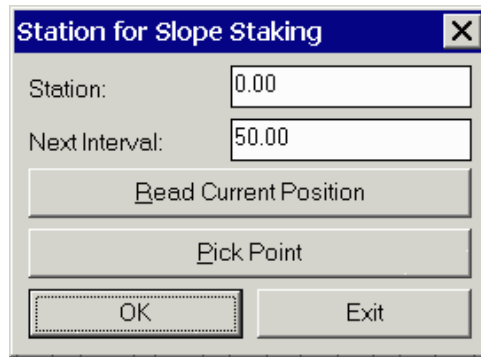
User entry is a simple method for slope staking that only requires a centerline file. With this method, the program prompts for the cut/fill slopes and the pivot offset and elevation. The program finds this offset-elevation for the stake station along the centerline and begins the cut/fill slope from this point.

The 3D Polyline method uses a 3D polyline for both the horizontal and vertical alignments. The program will prompt you to select the 3D polyline from the drawing. There are two polyline methods. The Station Along Polyline method does slope staking perpendicular to the polyline like the other slope staking methods. The Endpoint Projection is a special method that slope stakes from the selected end of the polyline. This method is described at the end of this section.

The first dialog in Slope Staking chooses the design method. For Design Files method, the files are specified in this dialog. For the other methods, the cut and fill slope ratios are also defined in this dialog.

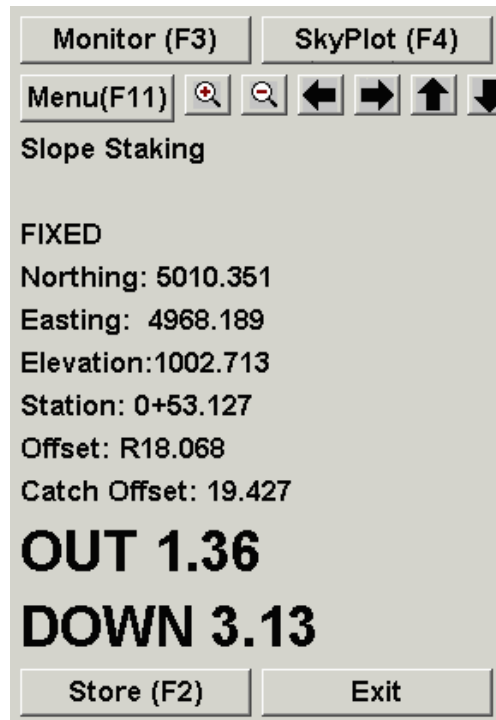
The next dialog sets the station to slope stake. The station should be entered as a number without the "+" symbol. For the 3D Polyline method, the starting station of the polyline is specified in the first dialog. For all the other methods, the starting station of the alignment is set in the centerline file. The Next Interval field is used to increment the stakeout station for the next stakeout point. The Read Current Position button will take a measurement from the GPS or total station to find the station of your current position. This current station is put in the Station field. The Pick Point button will prompt you to pick a point in the drawing view. The station of this point is used to fill out the Station field. For the User-Defined method, this dialog also contains the offset and elevation of the pivot point. For the 3D Polyline method, this dialog also contains the pivot point offset and vertical offset from the 3D polyline to

the pivot point.



The dialog box titled "Station for Slope Staking" contains the following elements:

- Station: 0.00
- Next Interval: 50.00
- Read Current Position button
- Pick Point button
- OK button
- Exit button



The screen displays the following information:

- Monitor (F3) SkyPlot (F4)
- Menu(F11) [Zoom In] [Zoom Out] [Left Arrow] [Right Arrow] [Up Arrow] [Down Arrow]
- Slope Staking
- FIXED
- Northing: 5010.351
- Easting: 4968.189
- Elevation: 1002.713
- Station: 0+53.127
- Offset: R18.068
- Catch Offset: 19.427
- OUT 1.36**
- DOWN 3.13**
- Store (F2) Exit

For the design file method, the centerline elevation at the stakeout station is calculated using the design profile and then the template is applied to calculate the pivot point. For the section file method, the pivot offset is interpolated from the section file. For example, if the stakeout station is 75 with offset right and the section file has offset-elevation of 18.0 right, 100.0 elevation at station 50 and has 20 right, 102.0 elevation at station 100, then the pivot offset for station 75 would be 19.0 right, 101.0 elevation. For the user entry and 3D polyline methods, the pivot point is specified Station For Slope Stake dialog.

After the slope stake station and pivot point are specified, Carlson Field begins to read the GPS receiver or total station to get the current position. The existing surface to tie into is defined by the elevations from these current position coordinates. The point where the cut or fill slope from the pivot point intersects the existing ground is called the catch point. As each coordinate is read, an existing surface cross section is built and the catch point is calculated. Carlson Field will automatically determine whether to find the catch point on the right side or left side of the centerline depending on the side of your current position. The program displays, in real-time as you move, the northing-easting and station-offset-elevation of your current position and the offset of the catch point. The distance from the current position to the catch point is reported as the offset difference as either "IN" or "OUT". The OUT means you should move out from the centerline. The IN means that the catch point is closer to the centerline. Based on this offset difference, you move perpendicular to the centerline either towards or away from the centerline to reach a new offset from the centerline while maintaining approximately the same station. The difference between your current station and the stakeout station is reported as the "UP" or "DOWN" distance. The UP means that your

current station is less than the stakeout station and you should move up the centerline. Likewise, the DOWN means that your current station is greater than the stakeout station and you should move back down the centerline.

When the catch point is located, press the Store button to end the slope staking. A report dialog is then displayed. The Catch Pt is the actual station, offset and elevation of the target catch point. The Stake Pt is the as-staked station, offset and elevation of your current position. The dialog also reports the horizontal and vertical distances from the catch point to the pivot point and the other template points. The Store Catch Point option will record the as-staked coordinates of the catch point to the current coordinate file. The Stake Offset Point is an option to locate an offset point. The offset to stake can be entered as a distance from the catch point or as an offset from the centerline.

	Station	Offset	Elevation
CATCH PT:	50.00	18.07	1002.04
STAKE PT:	51.22	16.26	1002.04

Stake to Pivot PIVOT
H: 4.26 U: 3.04 CUT

Store Catch Point

Offset Point Options

Stake Offset Point

Station: 50.00

Offset Type: CL Offset Delta Distance

Offset: 5.00

OK Exit

	Station	Offset	Elevation
OFFSET PT:	99.83	23.42	1000.15

	HDIST	UDIST	SLOPE
OFFSET to CATCH	4.46	2.33	FILL 52.3% 1.9:1
CATCH to PIVOT	6.96	3.48	CUT 50.0% 2.0:1

OK

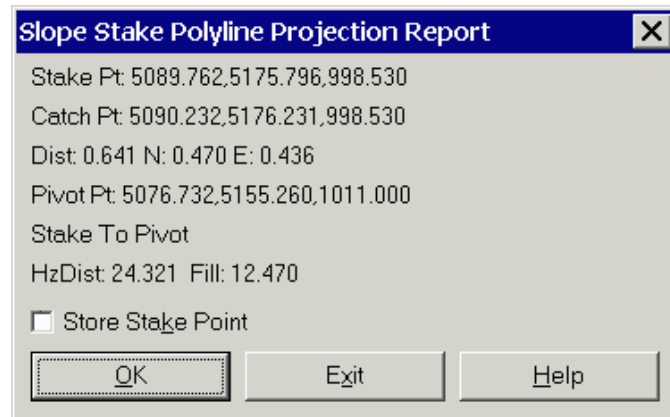
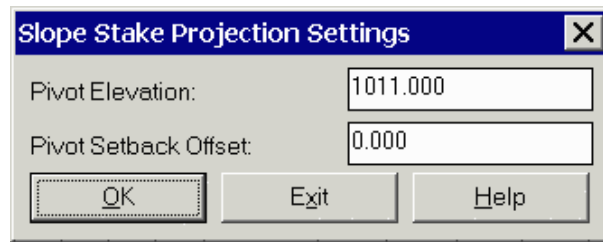
To locate the offset point, the same stakeout function from the Stakeout command is used. This function will guide you to the offset point. When the offset is reached, pick the Store button. Then an Offset Point Report dialog pops up containing the station, offset and elevation of the offset point and the horizontal distance, vertical distance and slopes from the offset point to the catch point, from the catch point to the pivot point and from the pivot point to the template points.

After locating the offset point, the station to stakeout dialog appears. You can enter the next station to stakeout or pick the Exit button to end Slope Staking.

Endpoint Projection

This is a special case of the 3D Polyline method that slope stakes from the end of the polyline. The program will prompt to pick a polyline and the end to stake from is the end nearest to the pick position. The direction of slope staking is in the direction of the polyline as if extending the polyline. The program prompts for the elevation of the pivot point which defaults to the elevation at the polyline endpoint. There is also an option to offset the pivot point along the polyline back from the endpoint.

After the pivot point is specified, the program starts the stakeout routine to guide you to the catch point. Then there is a report to show the difference between the staked and the calculated catch point.



Pulldown Menu Location: Roads

Prerequisite: A centerline file or 3D polyline

Slope Inspector

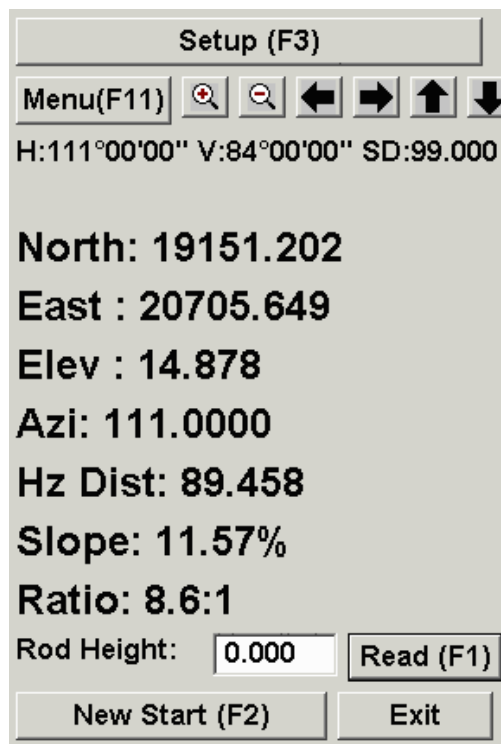
This command reports the azimuth, distance and slope between your current position and a starting point. The command starts by prompting you to move to the starting point and take a reading. This sets the starting point.

Slope Inspector with GPS

Carlson Field will continually read your current position from the GPS receiver. A dialog box appears displaying your current position and the azimuth, distance and slope from the starting point to your current position. An arrow icon will appear on the drawing showing your location. Pick the New Start button to set a new starting point.

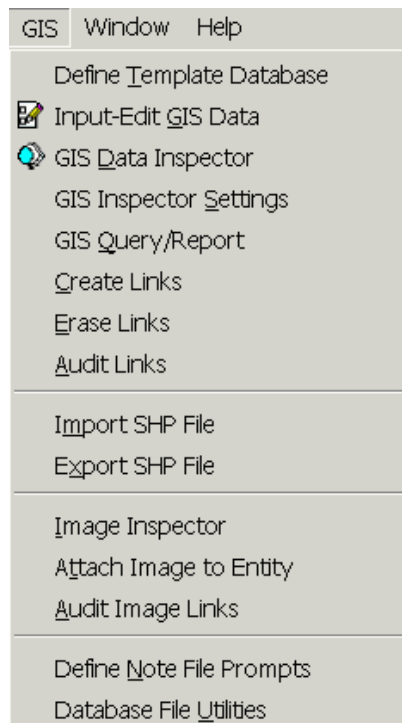
Slope Inspector with Total Stations

Before running Slope Inspector, make sure that the occupied point, backsight and instrument height are set correctly in the Equipment Setup command. Then start Slope Inspector and read a measurement for the starting point. A dialog box appears displaying your current position. Pick the Read button to take another measurement. Carlson Field will calculate the new point and report the azimuth, distance and slope from the starting point to the new point. An arrow icon will appear on the drawing showing your location. Pick the New Start button to set a new starting point.



GIS Menu

The GIS menu commands for Field are described in the GIS manual.



Equipment Menu

Apache Lightbar

Carlson Field can use an external Light Bar for determining elevation differences and centerline offsets.

Light Bars can indicate whether your current position is in cut, fill or on-grade when set vertically. When set horizontally, Light Bars can give centerline left/right offsets. Currently Carlson Field supports a light bar made by Apache, as well as by Mikrofyn, that has arrows for up/down, or left/right, and a row of lights for on-grade. The Light Bar must be connected to a separate serial port than the GPS.



CSI GBX Pro

Hardware Setup

1. Connect the receiver to the antenna by coaxial antenna cable if it is not already connected, and ensure that the receiver has ample power.
2. Ensure that the antenna is tracking corrections from an MSK Radio Beacon. The easiest way to do this is to use the antenna's automatic frequency scanning when first powering on the receiver
 - a) To do this, enter the [SETUP] menu, and select the option [AUTO BX SEARCH]

Note: The beacon automatically selected by this scan will be saved to the receiver's memory and used automatically in the future, until either the scan is executed again, or until a new beacon is specified manually. Thus, it is not necessary to scan each time the beacon is used, provided it is still operating in the same general area.

- b) A scan can be performed again in the event that the beacon is lost to scan for the next nearest beacon.
3. Enter the [Setup] menu, then select [Options] then [NMEA ON/OFF]. This menu allows the enabling or disabling of various NMEA messages. The only ones which are necessary are the GGA, GSV and GSA messages. All others should be disabled.

Software Setup

4. In Carlson Field, no further setup is necessary to make use of the CSI GBX Pro. Simply use the other Carlson Field functions as normal. Note however, that the elevations reported by the CSI GBX Pro are MSL (Mean Sea Level).

Depth Sounder

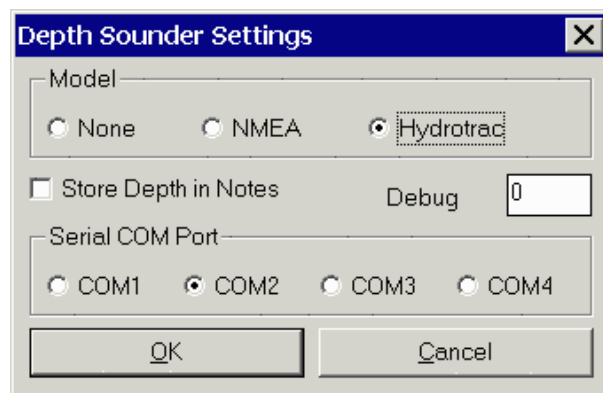
Carlson Field data collection can be used in conjunction with a depth sounder to survey the beds of rivers and lakes. Carlson Field takes input from both a GPS receiver and a depth sounder to determine and record the elevation of the terrain directly below the surveying boat or barge.

All of Carlson Field's routines work with the depth sounder to let you collect points on the underwater terrain. The elevation stored for each point is the elevation of the bed. Modeling of the bed surface works as easily as modeling any surface using Carlson Software. Carlson Field can be a powerful tool for marine surveying and construction.

Settings

To modify the Field depth sounder settings, go to the Field menu and select *Configure Field*. Choose the *Depth Sounder Settings* button.

The *Depth Sounder Settings* menu appears. At this point, Hydrotrac by Odom is the only equipment-specific depth sounder interface. Carlson Field works with other depth sounders that have NMEA standard interface. If you want to use Carlson Field without the depth sounder, make sure the Model is set to None.



For the Hydrotrac model, the depth sounder should be set so it outputs message DESO25 I/O. This is done using the Hydrotrac software. Odom should be contacted with any problems involving setting this message (www.odomhydrographic.com or 225-769-3051). The draft setting on the Hydrotrac should also be set. This will account for the height difference between the water surface and the working sensor of the Hydrotrac.

On the next line appears a box labeled *Store Depth in Notes*. Carlson Field saves point data in a coordinate file and in a text note file. By checking this box, the note file will record the water depth at each reading along with the other information about that point. (Settings to control the rest of the information saved in this file can be found in the menu *Configure Field > Point Settings*.)

The window labeled *Debug* should be set to zero for normal use.

The row of buttons labeled *Serial COM Port* refer to the COM port on your computer where the depth sounder is plugged in. Carlson Field requires two serial points on the computer when working with a depth sounder (one for the GPS and the other for the depth sounder). The depth sounder serial port must be separate from the GPS serial point.

Starting Out

Before working with the depth sounder, we suggest that you make sure the GPS system is working properly with Carlson Field. De-activate the depth sounder by setting the Model to None in the *Depth Sounder Settings* dialog box. Set up the GPS system that you are using and plug the rover receiver into the COM port for the GPS. Go to *Monitor GPS Position* under the *Field* menu. Check that the information being output is correct: Are the latitude and longitude readings what they should be? Are the north and east coordinates aligned to your job coordinate grid? Are the HRMS and VRMS low enough (less than one)? Is the status fixed? If it's autonomous or float, this rover

could be having trouble receiving the radio corrections the base receiver should be broadcasting. If everything is working properly, exit the monitor screen and start the depth sounder setup.

Measure the vertical distance from the GPS antenna to the surface of the water. This distance will be called the rod height. Go to the *Configure Field>General Settings* window and enter this measurement in the *Rod Height* box.

Plug the depth sounder into the depth sounder COM port on your computer. Go to the *Configure Field>Depth Sounder Settings* window and set the depth sounder Model. Set the rest of these settings as you want them and click OK.

Go back to *Monitor GPS Position*. Everything should appear as before, except there should be a new entry called *Depth* and *Elevation* should have changed to *Bottom Elv*. The correct depth should be showing and the *Bottom Elv* should be showing the elevation of the bed.

The usual Carlson Field functions will all work with the depth sounder active. The windows for *Monitor*, *Point Store* and *Auto Points at Interval* will display the depth when the depth sounder is set as active.

Geodimeter

Geodimeter 600 For Remote Mode

Note: Firmware version 696-03.xx or higher is required on the instrument. To check the version, pick MNU-5-4-1.

SET-UP

1. Connect the instrument to the battery pack. There is no need to connect the keyboard to the battery if it is going to be turned off, or attached to the unit.
2. Connect the prism to the top port of GeoRadio.
3. Connect the bottom port of the GeoRadio to Carlson Field. Then turn on the radio.
4. Turn on the Geodimeter. The Geodimeter starts with the screen for leveling the instrument. When the instrument is leveled press [ENT] key to continue to the next step. Now the instrument starts compensator calibration. You can wait for calibration to finish or turn it off. To turn calibration off press on [F] 22, enter 0 for comp. This needs to be done when the instrument is turned on and before [ENT] is pressed.
5. Next Geodimeter will ask for different values for pressure, offset, etc. They can either be left like they are by pressing on [ENT] or they can be changed.
6. Press [F] 79, it is the End of Transfer character, which should be set to 4.
7. To set radio, and station channels, press [MNU], and enter 1 for "Set". After set press 5, which will give the user opportunity to change channel, station, and remote address.
Note: The channel, station and remote address on the Geodimeter should match the channel, station and remote address in Carlson Field.
8. To set the Geodimeter for remote mode, press on RPU, then 3 for remote and 1 for ok, you can answer [NO] to "Define Window?" If [ENT] is pressed, the instrument will ask "Aim to A Press Ent", for which the user have aim to upper/lower left boundary and press [ENT], for "Aim to B Press Ent", aim to the upper/lower right boundary and press [ENT]. For "Measure ref obj?" press [ENT] if you want a reference object, otherwise press [NO]. Than the instrument is going to say remove keyboard however the keyboard can stay on.
9. After Geodimeter display screen turns itself off, it's ready for Carlson Field.

CARLSON FIELD

1. In Configure Field, under equipment type there should be Geodimeter. In Communication Settings Baud Rate should be set to 9600.
2. After Configure Field go to Equipment Setup and make sure GeoRadio is checked, and the channel, station and remote address is the same as it is in the total station.
Note: We recommend using channel 3.
3. If calibration box is checked the instrument will calibrate, to turn of calibration the box should be unmarked.
4. In setup there is also an option to turn on/off-tracking lights.

Geodimeter 600 For Direct Connection

1. Connect the instrument to the battery pack, and the control unit to Carlson Field.
2. Under Field go to Configure Field and place Geodimeter in Equipment type.
3. Click on General Settings make sure that the baud rate is set to 9600.
4. Exit Configure Field.
5. Go to Equipment Setup and check Connect to Station and click OK.

Now you are ready.



InnerSpace Tech depth sounder

The communication settings for the InnerSpace Tech depth sounder are 9600-N-8-1.

Laser Atlanta

To setup Laser Atlanta select Menu on the instrument, then Serial, and set Baud rate to the same as Carlson Field's and Format to Laser Atlanta.



Leica Disto

The communication settings for the Leica Disto are 9600-E-7-1.

Leica GPS System 500

Setting Up a 500 Series Receiver

1. Connect the antenna cable to the ANT Port on the front of the receiver, and to the antenna.
2. If you are using the PacCrest radio module, screw it in place over Port 1 on the receiver and attach its antenna cable. Otherwise, connect any radio being used to Port 1, 2 or 3.
3. If an external power source is being used, be sure to plug it into the PWR Port on the front of the receiver.
4. If external power is not being used, ensure that there are batteries in one or both of the batter slots on the bottom of the receiver.
5. Plug the 9 pin serial connection cable into the serial port of the computer running Carlson Field and into the Terminal Port on the front of the receiver.

Configuring Carlson Field for Use With a 500 Series Receiver

1. Select "Configure Field" from the Field pull-down menu. This will open a new window with several buttons on it, as well as a pulldown list labeled "Equipment Type." Select "Leica 500 Series" in the Equipment Type menu, then select "Communication Settings."
2. Ensure that the COM port is set to the one that the serial cable is plugged into, and that the Baud Rate is 9600, the Char Length 8, the Stop Bits 1, and the Parity None. Close this menu and the Configure Field menu.
3. In the Field pull-down menu, select "Equipment Setup." This will open another menu with several selectable options and several buttons.
4. Use the radio buttons on the top right to select whether the receiver will be a rover or a base station. Also be sure to select the antenna types being used from the pulldown menu at left.
5. Enter the desired Satellite Elevation Cutoff in the text box above the column of buttons. All satellites with elevations less than this number will not be used in position calculation(receiver default is 15).
6. Select the "Radio Settings" button. This will open another window with several selectable settings. Select the Port number the radio is attached to on the front of the receiver, the baud rate of the radio, number of radio stop bits and radio parity. These last three settings should be listed in the documentation for the radio being used. Also, select the desired format to use for sending and receiving messages from the bottommost option. Exit this menu.
7. If the GPS receiver is being configured as a base station, select the "Configure Base Station" button from the Equipment Setup menu, and proceed with step 8. Otherwise, the receiver is ready for use.

8. There will now be a menu with a few buttons to select a method of determining the base station's present location. The options are:

Read From GPS- Read one or more position readings from the GPS and use this position or the average of several positions for the base station corrections.

Enter Lat/Lon- This option will bring up a menu to enter the exact Latitude, Longitude, and elevation of the receiver's position by hand.

Enter State Plane Coord- This option will bring up a menu to enter the coordinates of the position of the base station according to the state plane coordinate system.

Read from File- This option will read a coordinate set from a file already saved to the computer.

Select whichever method will be used, and enter any necessary data. The receiver is now configured and ready for use.

Other Buttons In the Setup Menu:

1. Power Cycle Receiver will shut the receiver down and then power it up again. Used to clear the receiver's memory.
2. Power off Receiver shuts the receiver down. Note that if this button is pressed, any settings changes made while in this menu will not be saved to the receiver.
3. Send command to receiver allows for sending messages to the receiver. The user must enter the message by hand. This feature is only intended for use in conjunction with the technical support provided with Carlson Field.

Troubleshooting the Leica 500 Series in Carlson Field

Several possible errors can occur in the course of using a 500 Series Leica receiver with Carlson Field. Carlson Field will use all its standard error messages to report usual types of error messages, such as an inability to communicate with the satellites that are being tracked. In addition, the Leica 500 Series of receivers will have their own set of error messages unique to themselves. This type of error message is reported if there is an error during the transmission of various configuration messages to the receiver to set up the base station settings. Such messages will say "Set Port Message Rejected", or "Set Base Antenna Message Rejected" or "Set Antenna Height Message Rejected" or "Set RTK Message Rejected." Each indicates which particular facet of the configuration failed. If one of these messages is rejected, it is likely a momentary transmission error. If, on the other hand, several (or all) are rejected, it is possible there is a problem in the communication line between the computer and the receiver, which should be checked.

Leica TC Series

Leica TC Series Instrument Setup

On the instrument, make sure that the communication settings have CR/LF for the terminator.

Remote Mode

1. Turn on Leica
2. Connect Leica to rover radio, and connect the radio to the larger battery.
3. Connect the base radio to Carlson Field, and the smaller battery.
4. In the Field menu go to Configure Field, and under equipment type put Leica TC
5. To make sure the baud rate matches, under the Field menu go to Configure Field and click on Communication Settings and check if the baud rate is 19200. When Leica is turned on under Main Menu enter 5 for "Configuration", and 2 for "Communication Mode", then enter 1 for "Gsi parameters", and check if the baud rate is also set 19200.
6. Line Terminator in "Gsi parameters" should be set to CR/LF
7. Press [F1] for "cont" when done.
8. When back in Communication Mode screen enter 5 for "RCS (Remote) ON/OFF" and make sure it's NOT set for remote mode.
9. In Main Menu press [F6] for "meas" when ready to measure.

10. In the Field menu go to Equipment Setup and for Connection Mode check remote.

11. When done click on OK.

To put Leica in Tracking: On Gun press "FNC" then ATR+ and LOCK+

TCA 1800

1. Turn on Leica

2. Connect Leica to Carlson Field

3. In Field go to Configure Field, and under equipment type put Leica TC

4. To make sure the baud rate matches, under the Field menu go to Configure Field and click on Communication Settings and check the baud rate. When Leica is turned on press [F3] for "conf", then enter 3. The baud rate can be changed by pressing [F6] for "list", when done enter [CONT]. In addition to baud rate parity, char length, and stop bits should also match.

Note: Default in Carlson Field is not the same as default in Leica.

Leica 1100

To set up the Leica 1100 total station select the following commands on the instrument: Main Menu>Configuration>communication mode> GSI parameters. In the GSI parameters command copy the following settings: (baud=to Carlson Field's, protocol=GSI, parity=to Carlson Field's, Terminator=CR/LF, Data Bits=to Carlson Field's, Stop Bit=1), Geocom Param (baud=Field's) RCS Param (baud=Field's). Also, make sure RCS mode is OFF.

Leica 700

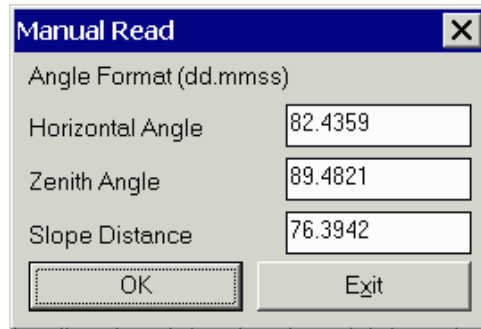
To set communication settings for **Leica 705s**, go Shift key then Prog (Menu) key and then to All Settings and last to PC Comm.



Manual Total Station

This method allows you to run Carlson Field in total station mode without being connected to equipment. The program will prompt you to enter the horizontal angle, zenith angle and slope distance. This method can be used for demonstration purposes or to work with total stations that cannot connect to Carlson Field. For these total stations, instead of the automatic connection, you can take a shot, read the instrument and then manually enter the data into Carlson Field.

As with other total stations, the first step is to run Equipment Setup to establish the occupied point, backsight and instrument/rod heights before running Carlson Field functions. Then in Carlson Field functions, when you pick the Read button, the program will bring up a dialog for entering the angles and distance. The angles should be entered in dd.mmss format (degrees.minutes seconds).



Manual Read	
Angle Format (dd.mmss)	
Horizontal Angle	82.4359
Zenith Angle	89.4821
Slope Distance	76.3942
OK	Exit

Mikrofyn Lightbar

Carlson Field can use an external Light Bar for determining elevation differences and centerline offsets. Light Bars can indicate whether your current position is in cut, fill or on-grade when set vertically. When set horizontally, Light Bars can give centerline left/right offsets. Currently Carlson Field supports a light bar made by Mikrofyn named RD-4 1137551, as well as by Apache, that has arrows for up/down, or left/right, and a row of lights for on-grade. The Light Bar must be connected to a separate serial port than the GPS.



Navcom Configuration Guide

This guide will walk you through the setup process for your Navcom units. It covers individual unit setup as well as base/rover setup under the simplest possible configuration. If you want to customize the configuration, consult the reference manual

A) Preliminary setup steps

Perform the following preliminary steps to initialize your computer for communication with your Navcom units:

- 1) Startup the Carlson product you intend to use.
- 2) Select CONFIGURE FIELD, from the FIELD Menu.
- 3) Select NAVCOM from the EQUIPMENT TYPE dropdown
- 4) Click the COMMUNICATION SETTINGS button. Verify that the SERIAL COM PORT is set to the port you intend to use to communicate with your Navcom unit. (usually COM1).
- 5) In the COM PORT SETTINGS box, click DEFAULT, and verify that baud rate=19200, parity=NONE, char length=8, and stop bits=1.
- 6) Click OK, and then click the GPS SETTINGS button.
- 7) Under GPS SETTINGS, set your HRMS and VRMS tolerance. For single-unit setup, these numbers should be at least 10. For base/rover configuration, they should be around 0.01.
- 8) Under PROJECTION TYPE, select the coordinate plane you wish to use. For state plane, make sure you choose the proper ZONE.
- 9) Click OK and then click EXIT

B) Single-unit setup (no corrections)

Before attempting a multi-unit setup, it is recommended that you first try setting up your Navcom unit to output an uncorrected position. The steps to do so are explained here:

- 1) Mount your GPS Antenna on a tripod in a place where its view of the sky is not obstructed.
- 2) If your antenna is separate from your receiver, connect your antenna to your receiver's AN TENNA port. (This step can be skipped for the RT-3010S, and other all-in-one models)
- 3) Plug your receiver into a power supply, or insert fully charged batteries into the battery ports. (Not all units have battery ports).
- 4) Turn your receiver on by holding down the power button for a few seconds, or until the status lights flash on.
- 5) Use the serial port cable to connect your computer to port A of your Navcom unit. Make sure the port on your computer that you use corresponds to the one you chose during preliminary setup.
- 6) Under the FIELD menu, choose EQUIPMENT SETUP.
- 7) If a PORT SETUP window pops up, set CONTROL PORT to PORT A, and RTK DATA PORT to RADIO PORT.
- 8) Setup is now complete. Steps that follow are optional.
- 9) Click the NAVIGATION STATUS button. From here you can monitor the progress of your Navcom unit as it calculates its position. Click AUTOREFRESH to view continuously updated status reports.
- 10) It may take a few minutes for the unit to calculate its position, if the unit was reset, or recently turned on. When the calculation is complete, VALID NAVIGATION will read YES. When this occurs, the Navcom unit is ready for use. Click CLOSE, then click CANCEL WITHOUT SAVING.
- 11) To monitor your position, choose MONITOR GPS POSITION from the FIELD menu, and you will see your current position. All Carlson Field GPS functions should now work.

C) Multi-unit setup (using corrections from a base)

- 1) Base Setup
 - a) Perform the preliminary and single-unit setup steps described above.
 - b) Attach the radio antenna to the radio port of your base unit.
 - c) Select EQUIPMENT SETUP from the FIELD menu.
 - d) Select a CORRECTION TYPE. We recommend NCT RTK.
 - e) CLICK the CONFIGURE BASE button.
 - f) When prompted to enter a position, enter the exact position of the base unit. Note that the accuracy of your rover's calculation depends on this position being completely accurate.
 - g) Enter a station ID of 0. The station ID is only used in RTCM mode.
 - h) Verify that STATION TYPE now reads BASE.
 - i) Base station setup is complete. Click SAVE SETTINGS AND EXIT.
- 2) Switch the device you're plugged in to:

After configuring the base, unplug your serial cable from your base's port A, and plug it into your rover's port A. Note: Whenever you switch the device you're plugged into be sure to close the Equipment Setup window first.

- 3) Rover Setup

- a) Perform the preliminary and single-unit setup steps described above.
- b) Attach the radio antenna to the radio port of your rover unit.
- c) Select EQUIPMENT SETUP from the FIELD menu.
- d) Select a CORRECTION TYPE. We recommend NCT RTK. Note that this selection must match the selection made during base setup.
- e) CLICK the CONFIGURE ROVER button.
- f) Verify that STATION TYPE now reads ROVER.
- g) Rover setup is complete. Steps that follow are optional.
- h) You can verify that the rover is receiving correction by clicking the MONITOR INCOMING CONNECTION, and then clicking AUTOREFRESH. The open window shows the time since each correction type was last received (delta time). In NCT RTK mode, the delta time of 5b, should stay around 1 second, and the delta time of message 5c should not go above 30 seconds. If these numbers are high, or if they read NEVER, try repeating the setup process or calling Carlson Software technical support.
- i) Click SAVE SETTINGS AND EXIT, and then choose MONITOR GPS

POSITION from the FIELD menu. The STATUS display should eventually go to LOCK.

Troubleshooting Base/Rover Configuration:

If you've configured a base to output corrections, and your rover does not appear to be receiving the corrections, try each of the following in order:

- 1) Verify that your BASE and ROVER are both set to the same correction type.
- 2) Under *Configure Radio*, check that your ROVER is set to slave, and that your BASE is set to master.
- 3) Under *Configure Ports*, check that both your base and your rover's RTK Data Ports are set to the proper value (Usually *Radio Port*).
- 4) Under the *Edit Base Position*, check that your BASE is set to a valid position. Note that if the given position is too far away from the position the BASE is reading, the BASE will not send corrections.
- 5) If you're trying to use RTCM, make sure the BASE and ROVER have the same station ID's.
- 6) Try increase the RTK Max Age constraint.
- 7) Under *Navigation Status*, verify that the Navigation is valid on both units. If either unit does not have a valid position solution, correction will not work.
- 8) Under *Monitor Corrections*, verify that the corrections you're using are arriving regularly. If they aren't you may need to reset both units.
- 9) Try configuring the BASE and ROVER again.
- 10) If all else fails, *Soft Reset* both units through the *Reset Unit* menu. After doing so, you will have to reconfigure the port settings of each device through the *Configure Ports* menu, and wait a few minutes for the devices to recalculate their position.
- 11) If none of these steps work, contact Carlson Software Technical Support.

Navcom GPS Setup

Carlson Field supports Navcom's NCT-2000D GPS message protocol, firmware versions 2.6 and later. If your Navcom unit has an earlier firmware version, contact Navcom for a free upgrade. Carlson Field has been tested extensively with Navcom models RT-3010S and RT-3020M.

From the Navcom GPS setup menu, or any of its submenus, the current device settings can be obtained by clicking the *Retrieve Settings* button. New settings can be saved by clicking the *Save Settings* or the *Save Settings and Exit* button. To cancel your changes, click *Cancel without Saving*.

By changing the *SV Elevation Mask*, you can prevent the Navcom Unit from using any satellite below a specified elevation angle (Range: 0-90).

By changing the *PDOP Mask*, you can prevent the Navcom Unit from using any GPS solution with a PDOP above a specified value (Range: 1-25).

By changing the *RTK Max Age*, you can prevent the Navcom Unit from using any RTK corrections older than a specified number of seconds. (Range: 0-1275, Multiple of 5).

By changing the *Base Station ID* on a base, you can provide your base with a unique identifying number so that rovers can specify which base they want to use for corrections. By changing this setting on a rover, you can specify which base unit you want to use. If 0 is specified, the rover will use any base station it can find. The base station ID only applies when using the RTCM correction format. (Range: 1-1023)

You can choose between 4 different *Correction Types*: NCT (Navcom Proprietary), CMR (Trimble's format), RTCM RTK (Messages 18-22), or RTCM DGPS (Message 1 and 9). When configured to BASE, changing the correction type changes the type being sent. When configured to ROVER, changing the correction type changes the type the unit is listening for. A ROVER will ignore all incoming correction messages except those of the type specified.

Configure Ports Submenu:

The *Control Port* should be configured to *Port A* or *Port B*, depending on which of the Navcom units' ports you are plugged into. Note that the *Control Port* refers to the number of the port on the Navcom unit, NOT the number of the COM port on your computer. If the *Control Port* is configured improperly, you will not be able to communicate with your Navcom unit.

The *RTK Data Port* refers to the device port out of which RTK corrections will be sent. This value should be set to *Radio Port*, unless you want to set up a non-wireless Base/Rover connection through Port A or Port B. The *RTK Data Port* cannot be the same as the *Control Port*.

Configure Radio Submenu:

The *Radio ID* is the value used to identify a unit on a wireless network of Navcom units. Make sure that no other Navcom unit in your vicinity shares the same *Radio ID*. By default, the *Radio ID* is the same as your Navcom unit's serial number. This value can be changed, although there isn't usually any need to do so.

The *Local Radio Type* can be set to either *Master* or *Slave*. Radio communication will only work between *Masters* and *Slaves*. Only one unit on your network should be set to *Master*. It makes sense to make the base unit a *Master*, and all rovers *Slaves*. These settings will be handled automatically by the *Configure Base* and *Configure Rover* routines. So there generally isn't any reason to set the *Local Radio Type* manually.

The *Local Antenna Power Level* allows you to configure your radio to use more or less power. The less power the radio has, the less it will be able to communicate over longer distances. It may be useful to change the power level if you're rover is not traveling far from your base, and you're trying to conserve battery power.

Within the *Navcom Radio Setup* menu, you will be able to access the following status information for all visible Navcom units on the network:

External Power: Indicates whether the unit is plugged into an external power source (On or Off).

Battery A: Indicates whether a well charged battery is plugged into Battery Port A (On/Good or Off/Low)

Battery B: Indicates whether a well charged battery is plugged into Battery Port B (On/Good or Off/Low)

Status: Indicates whether the unit is sending out corrections. (BASE or ROVER)

If more than two units are present, you can access this information for the additional units by selecting the desired unit's radio ID from the *Remote Radio ID* dropdown menu.

Configure RTCM Submenu:

Note: To access this menu, first configure the unit as a BASE and set the *Correction Type* to either *RTCM RTK* or *RTCM DGPS*.

Choose *message 18/19* to make your RTCM RTK base broadcast RTCM message types 18/19.

Choose *message 20/21* to make your RTCM RTK base broadcast RTCM message types 20/21.

Choose message 1 to make your RTCM DGPS base broadcast RTCM message type 1.

Choose message 9 to make your RTCM DGPS base broadcast RTCM message type 9

Edit Base Position Submenu:

Note: To access this menu, first configure the unit as a BASE.

If your BASE already has a GPS position set, it will be shown here. (If you don't see it, trying pressing Retrieve.) To edit this value, change the displayed number and press the *Lock* button.

Click *Survey* to read a new GPS position from the Navcom unit.

Click *Empty*, to clear the GPS position from the unit.

Reset Unit Submenu:

Click *Soft Reset* to send a reset command to the Navcom unit. If the command is successful, all three status lights on the unit should go solid temporarily. After performing a soft reset, you will have to go to the *Configure Port* Submenu to reconfigure the control port.

Click *Factory Reset* to send an emergency reset command to the Navcom unit. However, in nearly all cases, it is only necessary to use the *Soft Reset* button. After performing a factory reset, you will have to go to the *Configure Port* Submenu to reconfigure the control port.

View Firmware Submenu:

This submenu displays the Navcom firmware version your unit is using, along with the hardware serial numbers and the hardware model name.

Navigation Status Submenu:

If *Valid Navigation* reads *Yes*, your unit has successfully solved its position. If it reads *No*, the unit's position has not yet been calculated, and an error message explaining why will be displayed in the *Error* field. A rover will not try to use RTK corrections unless its navigation is valid. Similarly, a base will not broadcast correction unless its navigation is valid.

Navigation Status will read *AUTONOMOUS* if it is not receiving the type of corrections it has been configured to use. It will read *FLOAT* if it is receiving the right kind of corrections, but hasn't finished using them to calculate its position. It will read *LOCK* when it is receiving corrections and has successfully used them to calculate its position.

Navigation Mode displays the specific type of correction that is currently being used.

of Satellites Used shows the number of satellites the unit is able to use in its solution.

All *DOP* values are also shown here (GDOP, PDOP, HDOP, VDOP, and TDOP).

Click *Refresh* to load the latest values from the device.

Monitor Incoming Corrections Submenu:

Note: To access this menu, the local unit must be configured as a ROVER.

This menu displays the number of seconds since the arrival of each RTK correction type. At the top, the correction type currently being used is displayed.

In *NCT Correction Mode*, the relevant messages are 5B (correction), which should be arriving every second, and 5C (base position), which should be arriving every 16 seconds.

In *CMR Correction Mode*, the relevant messages are cmr0 (correction), which should be arriving every second, and cmr1(base position), which should be arriving every 30 seconds.

In *RTCM RTK Correction Mode*, the relevant messages are RTCM message 22, and either messages 18 and 19, or messages 20 and 21, depending on your base's RTCM setup. Messages 18-21 should be arriving every second. Message 22 should be arriving every 6 seconds.

In *RTCM DGPS Correction Mode*, the age of correction messages (1 and 9) cannot be monitored here.

Click *Refresh* to load the latest values from the device.

Configure Base Submenu:

Before clicking *Configure Base*, first choose the type of corrections you want to use. When you click *Configure Base*, all steps necessary to configuring a base will be performed. You will be prompted for a Base Position and a Radio ID. Upon completion, the unit status should read BASE. If it does not, or if an error occurs during base configuration, try again, or consult the Base/Rover configuration troubleshooting section below.

Configure Rover Submenu:

Before clicking *Configure Rover*, first choose the type of corrections you want to use. When you click *Configure Rover*, all steps necessary to configuring a base will be performed. Upon completion, the unit status should read ROVER. If it does not, or if an error occurs during rover configuration, try again, or consult the Base/Rover configuration troubleshooting section below.

Switching the device you're plugged in to:

Whenever you switch the device you're plugged into be sure to either close the Equipment Setup window, or click *Retrieve Settings* from the top level Equipment Setup menu.

Troubleshooting Invalid Navigations:

If the *Navigation Status* menu reports an invalid navigation, you're unit has not yet been able to calculate it's position. The unit may need more time, if less than 4 satellites are visible, or an error is reported. If you can't get a valid solution for a few minutes, try raising the PDOP mask, or lowering the Satellite elevation mask.

Troubleshooting Base/Rover Configuration: If you've configured a base to output corrections, and you're rover does not appear to be receiving the corrections, try each of the following in order:

1. Verify that your BASE and ROVER are both set to the same correction type.
2. Under *Configure Radio*, check that your ROVER is set to slave, and that your BASE is set to master.
3. Under *Configure Ports*, check that both your base and your rover's RTK Data Ports are set to the proper value (Usually *Radio Port*).
4. Under the *Edit Base Position*, check that your BASE is set to a valid position. Note that if the given position is too far away from the position the BASE is reading, the BASE will not send corrections.
5. If you're trying to use RTCM, make sure the BASE and ROVER have the same station IDs.
6. Try increase the RTK Max Age constraint.
7. Under *Navigation Status*, verify that the Navigation is valid on both units. If either unit does not have a valid position solution, correction will not work.
8. Under *Monitor Corrections*, verify that the corrections you're using are arriving regularly. If they aren't you may need to reset both units.
9. Try configuring the BASE and ROVER again.
10. If all else fails, *Soft Reset* both units through the *Reset Unit* menu. After doing so, you will have to reconfigure the port settings of each device through the *Configure Ports* menu, and wait a few minutes for the devices to recalculate their position.
11. If none of these steps work, contact Carlson Software Technical Support.

Troubleshooting when you cannot establish communication with the unit:

If all of your commands in the Equipment Setup menu are failing, try opening the *Configure Ports* submenu, selecting the proper Control Port, and saving the new settings. Make sure that you're plugged into the port you have chosen to be the control port.

If this does not work, issue a soft reset command. If this fails, try a factory reset command. If even this fails, call Carlson Software Technical Support.

Nikon Total Stations

Nikon A-Series

Nikon A-Series includes the A5LG/A5, A10LG/A10 and A20LG/A20. Also the C-100 and D-50 have the same communication as the A-Series and should be used in the SET mode.

Nikon 500 Setup

1. Turn on Nikon

2. Turn it Horizontally and Vertically to set it.

3. Connect Nikon to Carlson Field

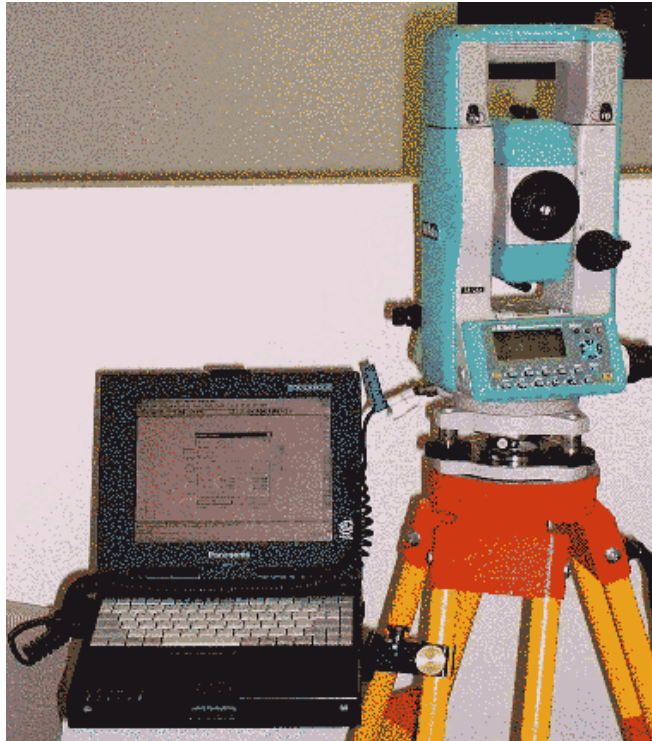
Note: 9-pin serial cable from Nikon to Carlson Field should be NGT type and not SOKTOP.

4. In Field go to Configure Field, and under equipment type put Nikon 300,400,500 series.

5. To make sure the baud rate matches, under the Field menu go to Configure Field and click on Communication Settings and check the baud rate. On Nikon press [MENU], then 3 for "sett", and 6 for "comm". The baud rate can be changed using the arrow keys.

6. Exit the Configure Field menu.

7. To check if units (Ft /M) matches for correct results, in Carlson Field under Settings go to Drawing Setup and select the appropriate button. On Nikon, press [MENU] and 3 for "sett" again, but now press 5 for "unit".



Nikon 310

Set the same baud rate in the Nikon 310 station as you did in Carlson Field and set the Nikon instrument to the record format by selecting on the instrument Fnc->5(Set)->6(other)->3rd screen.

OmniStar Otto

In Field go to Configure Field and under equipment type select CSI GBX/OmniStar Otto and in Communication Settings set the baud rate to 9600.

Simulation GPS

Simulation GPS mode is for demonstration purposes to show or practice Carlson Field functions. This mode allows you to run Carlson Field without being hooked up to any equipment. The program will automatically generate a position. This position is the first point in the alignment. If there is no alignment, then the starting point is 5000,5000,1000. There are keyboard commands to control the simulation position during continuous read commands such as Stakeout and Track Position.

Here are the Keyboard Command:

L - Turn Left
R - Turn Right
F - Go Faster
S - Slow Down
U - Up
D - Down
W - Switch Direction

Sokkia

Sokkia Radian IS

Hardware Setup

1. Make sure that the Radian IS has fully charged batteries installed, as described in the receiver documentation.
2. Connect the Radian IS serial cable to "COM1" on the Radian IS, plugging the other end into the controlling computer's serial port.
3. If the Radian IS is to be used as a base, connect a PDL base radio to the "COM2" port of the receiver. If the IS is to be used as a rover, connect a PDL rover radio to the "COM2" port of the receiver.
4. Power the Radian IS on with its external power switch.
5. Once the receiver finishes its self-initialization (when all the lights on the side panel go out and then the battery light lights in just one position), it is ready for use with Carlson Field. However, positions will not be able to be logged until the receiver has acquired a few satellites. The receiver has enough satellites when the center light is at the second or higher level (when it is orange instead of red).

Software Setup

6. To configure the IS for use, select "Equipment Setup" from the Field pulldown menu. This will open a menu with several options:
 - a) Radio Baud Rate: This radio button sets the baud rate for COM2, the radio COM port. Make sure this number and the number the PDL's are set for is the same.
 - b) Station Type: This sets whether the Radian IS is to be configured as a base station or a rover.
 - c) Elevation Type: This allows selection of Geoid (MSL) or Ellipsoidal measures for height/altitude.
 - d) RTK Dynamics: This sets the dynamics mode of the receiver. In general, this setting should be set to "Dynamic/Kinematic".
 - e) Message Type: This sets what format of corrections this receiver will send/receive for RTK.
 - f) Motion Dynamics: This is used to set the receiver's calculations appropriate to the motion of the receiver.
 - g) Elevation Mask: This is the satellite elevation cutoff. No satellites with elevation less than this number will be used in corrections. This allows filtering out of satellites close to the horizon, which provide less accurate calculations for positions.
 - h) Send Command to Receiver: This allows a specific user-entered command to be sent to the receiver. Mostly used for troubleshooting with Technical support.
 - i) Configure Base: This configures the parameters of a base station for the receiver (Ex: Current position, etc.)
 - j) Power Cycle Receiver: This powers the receiver down and then turns it back on, clearing the main memory.
 - k) Save and Exit: This saves all settings changes and exit this menu.
 - l) Cancel: This restores original settings and exit this menu.

To set the Radian IS up as a Rover:

7. Select "Rover" for Station Type, and set the Radio Baud to match the PDL's which are being used. Also, set "RTK Dynamics" to "Dynamic/Kinematic", and set Motion Dynamics to the appropriate option.
8. Select "Exit and Save". The receiver is now ready for use as a rover.

To set the Radian IS up as a Base:

9. Select "Base" for station type, and set the Radio Baud to match the PDL's which are being used.

For most jobs, set RTK Dynamics to "Dynamic/Kinematic" (unless you are *sure* that static is more appropriate-even small fluctuations from wind on the pole can cause problems in Static mode). Set motion dynamics to Foot/Walking, and then select "Configure Base Station"

10. In the menu dialog that opens, there are a few buttons:

- a) Read from GPS: Read a position from the GPS and fix to that position
- b) Enter Lat/Lon: Fix to a manually entered Lat/Lon position
- c) Enter State Plane Coord: Fix to a manually entered State Plane Northing/Easting position
- d) Read From File: Fix to a position read from a *.ref file.
- e) Cancel: Cancel base setup

If Read From GPS is selected, the software will read once from the GPS receiver, and then fix to that position. If Enter Lat/Lon is selected, a dialog box will open and a Latitude and Longitude must be input manually. If Enter State Plane Coord is selected, a dialog box will open allowing the input of a set of Northing/Easting coordinates by hand.



Read from File will open a File>Open dialog and ask for the file name of the file to open.

Regardless of which option is selected, after the position is determined, this position will be displayed, and dialog boxes will open to enter a station id and the measured base antenna height. Once these values are entered, base setup is complete and the "Exit and Save" button can be selected to exit the Equipment Setup menu.

Sokkia 500 Series

1. Turn on Sokkia
2. Turn it Horizontally and Vertically to set it.
3. Connect Sokkia to Carlson Field
4. In Field go to Configure Field, and under equipment type put Sokkia
5. To make sure the baud rate matches, under the Field menu go to Configure Field and click on Communication Settings and check the baud rate. On Sokkia press [ESC], then [CNFG]. Scroll down or enter 4 for "Comms setup." The baud rate can be changed using the arrow keys, when done press [ESC].

6. Exit the Configure Field menu.

7. To check if units (Ft /M) matches for correct results, in Carlson Field under Settings go to Drawing Setup and select the appropriate button. On Sokkia, in [CNFG] scroll to or enter 5 for "unit" and select appropriate unit using the arrow keys.

Topcon Total Stations

The Topcon instrument must have CR/LF (carriage return/linefeed) turned on for communication with Carlson Field.

Topcon 200 Series

To set CR/LF with 200 series:

1. Turn instrument off
2. Turn instrument on while holding F2 key
3. Choose F3 (Others set)
4. Press F4 (Page down)
5. Choose F3 (CR/LF) and set it on

To set this with 700 series:

1. Choose Parameter from the main screen
2. Scroll down until you find CR/LF and set it on

Topcon ITS

The command echo on the instrument must be turned off to work with Carlson Field.

Topcon GTS-A4

To setup the instrument hold down F-2 as you switch it on. This will bring up a parameters menu, press F-3 for Data Out. Hit Select to browse through the settings options, and make sure CR, LF: is ON and that Echo back: is OFF. Setup is complete.

Topcon GTS-700

To set the instrument to work with Carlson Field, press [F2] for "std" on the instrument.





Topcon 800-A Remote Setup

Topcon Setup:

Note: The instrument needs to be set to REC-A, not REC-B mode

1. Turn on the Topcon
2. Connect the Topcon to one of the radios, and the other radio connect to Carlson Field
3. Under Field menu go to Configure Field, and under equipment type select Topcon800A-Remote.
4. To set Topcon for external mode Press [F1] for "prog", then [F6] for "more". This will lead to more programs. Enter [F2] for "Ext.Link."
5. To select the radio channel, in External Link enter 2 for "settings" and 4 for "parameter (radio modem)", then 3 for "set channel". Using the arrow keys change the channel. When done press for [F1] for set, then press [ESC] until get back to External Link Menu.

Note: Channel on the Topcon should match the channel set in Carlson Field.

6. After channel is set press 1 for "Execute"
7. Topcon is ready.

Note: If the batteries are low either in Topcon or the radios, communication problems will arise.

Carlson Field Setup:

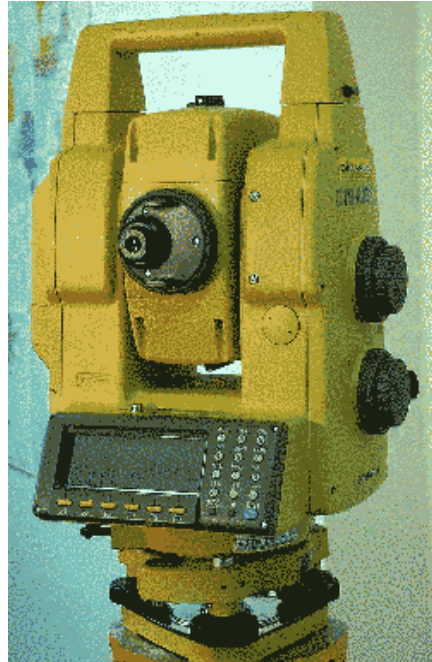
1. In Configure Field, under equipment type there should be Topcon800A-remote. In Communication Settings Baud Rate should be set to 9600.
2. After Configure Field go to Equipment Setup and make sure the radio channel or radio frequency matches the channel and frequency in Carlson Field. Press Ok when done.

Topcon 800A Quick Lock

1. Dismount the handle from the Topcon, and mount RC-2H. Secure it with the fixing screw.
2. Attach RC-2R to the prism, and turn it on.
3. Using the Y cable attach the RC-2H to the radio and Carlson Field.

4. In Joystick click on Quick Lock and Topcon will do angle turn until it finds a prism in which it will lock to, and will start tracking.

5. If RC-2H is not attached to the radio with Y cable, when Quick Lock is pressed the big yellow button on RC-2H needs to be pressed in order for the Topcon to search for the prism.



Trimble

Trimble NT300D

1. In order to properly configure the NT300D to work with Carlson Field, it must first be powered up in Setup mode (by holding down the [Setup] button on the front panel of the receiver while powering it on) so that the advanced setup options are available. Once the NT300D is powered up in this mode, bring up the Setup menu via the [Setup] button. Page down using the *More* menu option until the *I/O* menu item is available, and select it.

a) In the *I/O* menu, select whichever port is to be used to interface the receiver with the computer running Carlson Field (Port 1 by default). Next, set both the input and the output to transmit/receive in *NMEA*, at *9600* baud rate. The final option, Remote Select, should be set to *Primary*.

b) Now the *NMEA* sentences must be configured. From the *I/O* menu, enter the *NMEA Sentences* submenu. Disable all sentences, save for the *GGA* sentences and the *GSA* sentences. Ensure the Talker ID is *GP*. From here, *Return* to the *I/O* menu.

c) The *NMEA Control* menu item, reachable from the *I/O* menu, has three options. The Output Rate here should be set to *1 second*, the Position Output Rate set to *Output Rate*, and the *NMEA Output Version* to *2.1*.

Next, the *GPS* settings must be configured, and can be found in the *GPS* menu under the main *Setup* menu.

The *GPS* Mode should be set to *3D*, and the *DGPS* mode set to *Auto*. The *DGPS* source should be toggled to *Internal*, and the Pos/Vel Filter should be *Off*. Mask Values should be left at *Default*, and the SNR at *M*.

Finally, the Beacon Receiver configuration (under *Beacon Receiver* on the Setup Menu) needs to have its Search Mode set to *Auto-Dist* Mode. All other values in all menus ought to either be left at their default settings, or configured as necessary to the local conditions (in the case of antenna height, etc.).

2. The RMS value reported in Carlson Field is the RMS value of the standard deviation of the range inputs to the navigation process including pseudoranges and *DGPS* corrections.

The NT300D is now properly configured, and if connected to a computer running Carlson Field, will transmit position fix data to the computer automatically. Before using it, however, it is best to power it down and then turn it back on normally, as running it in Setup Mode is not recommended.

Trimble 4000 Series

Hardware Setup

1. Setup the antenna and GPS receiver as normal. The radio should be on I/O Port 2.
2. Connect the Computer that Carlson Field is running on to I/O Port 1 by the appropriate cable.

Front-Panel Configuration:

Base Station:

1. After powering on the receiver, press the [Control] Button. From the selections available, select **MORE**. This will bring up a second page of options. Select **MORE** again. The front panel screen should now be on **RECEIVER CONTROL "3 of 7"**.
2. Select **BAUD RATE/FORMAT**, and from the menu that this creates, select **SERIAL PORT 1 SETTINGS**.
3. Ensure that the port is set to 38400 baud, 8-Odd-1 Format, with no flow control.
4. Similarly, make sure that the settings for I/O Port 2 agree with those of the type of radio being used (typically 9600 8-None-1).
5. Return to the **RECEIVER CONTROL** menu, and go to page 4 of 7. Select **REFERENCE POSITION**.
6. Enter the Lat/Lon of the position the base is located at. Alternately, select **HERE** to have the GPS unit read the current position and use that as the base reference point.
7. On page 1 of the **RECEIVER CONTROL** menu, select **RTK OUTPUT CONTROL**.
8. Set the **RTK OUTPUTS** to Port 2, and the **ANTENNA HEIGHT** to the measured height of the antenna.
9. Ensure that all other forms of output (Cycled Output, 1PPS output, Event Markers, etc.) are disable. These options may all be accessed with the submenus accessible through the [Control] button.
10. Ensure that the Synch time of the Rover and Base are the same. This setting may be accessed by first pressing [Control] and then cycling through the menus until the **MASKS/SYNCH TIME** option is available.

Rover Station:

1. After powering on the receiver, press the [Control] Button. From the selections available, select **MORE**. This will bring up a second page of options. Select **MORE** again. The front panel screen should now be on **RECEIVER CONTROL "3 of 7"**.
2. Select **BAUD RATE/FORMAT**, and from the menu that this creates, select **SERIAL PORT 1 SETTINGS**.
3. Ensure that the port is set to 38400 baud, 8-Odd-1 Format, with no flow control.
4. Similarly, make sure that the settings for I/O Port 2 agree with those of the type of radio being used (typically 9600 8-None-1).
5. Return to the **RECEIVER CONTROL** menus, and go to page 2.
6. Select **RTK ROVER CONTROL**.
7. Toggle the **ENABLE** setting to **L1/L2**.
8. Push the [Status] button, and select **POSITION**. There should now be an **RTK** option. Select it. This will bring up a screen displaying delta Northing/Easting, correction status, etc.
9. Ensure that the **STATIC** option appears at the right. This means you are in kinematic/rover mode. If instead the **ROVE** option is available, select it.
10. Ensure that all other forms of output (Cycled Output, 1PPS output, Event Markers, etc.) are disable. These options may all be accessed with the submenus accessible through the [Control] button.
11. Ensure that the Synch time of the Rover and Base are the same. This setting may be accessed by first pressing [Control] and then cycling through the menus until the **MASKS/SYNCH TIME** option is available.

Trimble 4700/4800

Hardware and Equipment:

1. Make sure that the computer's serial port is connected to the 4700/4800 in it's COM1 port (typically the port that a data collector is normally plugged into). Power should be supplied on COM2, and any radio used for RTK should

be plugged into COM3.

2. All other equipment (antenna, wires, etc.) should be set up as normally directed by the manuals.

Software Configuration:

1. After selecting the Trimble 4700 equipment type from the "configure field" menu, open up "Equipment Setup."

This should bring up a new window/dialog box with the following options:

- a. Receiver Type: Select whether you are using a 4700 or 4800 receiver.
- b. Station Type: Choose what type of RTK station you are setting this receiver up as a base or rover.
- c. RTK Correction type: Select the type of Corrections you would like a base station to transmit. Note that CMR messages should be used for most precision applications, as RTCM is only capable of producing less-accurate floating precision positions
- d. Radio Baud Rate: The baud rate of the radio port. This should be left at the default setting of 9600 in general
- e. Satellite Elevation Cutoff: All satellites with elevation from the horizon of less than this number will *not* be used in calculating a position. This allows less accurate low elevation satellite to be factored out of a position.
- f. Configure Base Station: Will configure the receiver to act as a base. See "Configuring Base Station" below.
- g. Cancel without saving: Will exit this menu without saving any changes that have been made.
- h. Save and Exit: Will save these settings to the receiver and to Carlson Field's setup and exit out of this menu.

Configuring Rover:

No real configuration is necessary, aside from setting up the equipment and setting the appropriate Receiver Type, Station Type, and Satellite Elevation Cutoff.

Configuring Base Station:

1. After selecting all the appropriate settings in "Configure GPS," click on the "Configure Base Station" button.
2. In the menu dialog that opens, there are a few buttons:
 - a) Read from GPS: Read a position from the GPS and fix to that position
 - b) Enter Lat/Lon: Fix to a manually entered Lat/Lon position
 - c) Enter State Plane Coord: Fix to a manually entered State Plane Northing/Easting position
 - d) Read From File: Fix to a position read from a *.ref file.
 - e) Cancel: Cancel base setup

If Read From GPS is selected, the software will read once from the GPS receiver, and then fix to that position. If Enter Lat/Lon is selected, a dialog box will open and a Latitude and Longitude must be input manually. If Enter State Plane Coord is selected, a dialog box will open allowing the input of a set of Northing/Easting coordinates by hand. Read from File will open a File>Open dialog and ask for a file name of a reference file (*.REF) to open for use in corrections.

Regardless of which option is selected, after the position is determined, this position will be displayed, and dialog boxes will open to enter a station id (used by the base to identify itself to the rover(s)) and the measured base antenna height. Once these values are entered, base setup is complete and the "Exit and Save" button can be selected to exit the Equipment Setup menu. At this point, whenever looking at a menu that displays the connection status, "REFERENCE" will be displayed, instead of Float, Fixed, or Autonomous.

Trimble 5800

Carlson Field Configuration:

In Configure Field, set the Equipment Type to Trimble Generic.

In Equipment Setup, be sure to set the Data Type to match your receiver setup. This Data Type is the port on the receiver that communicates with Carlson Field. Typically, the Data Type should be set to 2 for the serial connection and to 4 for Bluetooth.

When configuring the Base receiver, use a base station id number in the range from 1 to 32.



Point Clouds Module

13

Point Cloud Recommended Specifications

The most important aspect for processing Point Clouds is RAM: for 250 million points about 12GB of RAM is needed (the cloud itself is half that but the user will want to be able to hold a source and a destination). More RAM allows more clouds to be held in the memory.

A SSD (solid state disk) instead of an HDD (hard disk drive) allows clouds to load faster. If the SSD on your computer doesn't have a large capacity, it is recommended to have an additional drive or backup server for extended storage.

As for processing power required, Point Cloud is primarily a single-core program but some select routines are multi-threaded. Therefore, you can focus on buying a processor with more speed (GHz) rather than the number of cores if you are only using the Point Cloud software and other applications such as Survey and Civil. Now then, if you are or will be using Carlson PhotoCapture for drone image processing, you are better off buying a multi-core processor computer as PhotoCapture is a multi-threaded application and more cores are better.

It is necessary to have a 64-bit operating system. Windows 7, 8, 8.1, 10 or 11 are all fine but please note that XP is no longer supported.

It is recommended to have a separate graphics card, preferably nVidia GeForce.

Summarized Specifications:

1. 64-Bit Operating System - Windows 7, 8, 8.1, 10, 11
2. Single/Multi Core Processor - preferred higher GHz and number of cores
3. 32GB RAM or greater - for photogrammetry and point cloud processing
4. SSD Drive
5. Optional additional HDD/SDD for storage

Point Clouds Getting Started

Carlson Point Cloud (not to be confused with Carlson Coordinate Cloud) is an application designed to run alongside IntelliCAD or AutoCAD and offers processing tools for large point datasets. It provides several methods to reduce, clean, extract from, and manipulate Point Cloud datasets so that they can be processed within CAD software without the typical memory limitations found in CAD applications.

Carlson Point Cloud comes in two varieties - Basic and Advanced.

Point Cloud Basic - designed to work within a CAD environment to manipulate and extract data from a point cloud. It has a very large capacity (up to a billion points) and is a great program for performing a "virtual survey" directly in a cloud. It allows the creation of Carlson points and 3D polylines utilizing a typical Carlson Field to Finish code table (FLD file) and draws entities directly into an open CAD drawing with no additional export function. A notable feature is Survey by Grid which sets points at any specified grid spacing.

Point Cloud Advanced - contains all of the functionality of Carlson Point Cloud Basic plus the ability to extract features such as curbs, parking lot stripes, road markings, power lines, trees, hydrants and poles. It also allows the creation of TIN/Grid files, solids, and utilizes typical Carlson cross section alignments (mxs files) to extract cross sections and profiles. You can also calculate volumes between surfaces. Cloud data can be selected by color or

elevation enabling you to create regions for isolating portions of a cloud for editing or extracting features. A notable feature is the ability to simplify a TIN and create smoother looking contour lines.

All actions performed in Point Cloud are initiated through the Point Cloud **Project Manager**. This manager can be brought up from your CAD application (assuming that the Point Cloud module is loaded) via the **Point Clouds -> Point Cloud Project Manager** menu option or with the **PC_PROJECT_MANAGER** command. Work in Point Cloud is arranged into projects and each project maintains a tree structure of all the data it contains.

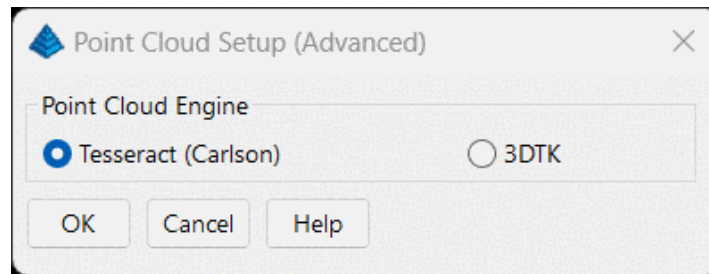
If you have not used Point Cloud before, you may want to go through the **Point Cloud Step By Step Tutorial**.

If you need to activate one of the legacy engines you can do so in the Point Cloud Setup via the **Point Clouds -> Point Cloud Setup** menu option.

Please view the Recommended Specifications.

Point Cloud Setup

The Point Cloud application comes with two point cloud engines: **Tesseract (Carlson)** and **3DTK**. Point cloud setup, located in the Point Cloud menu in CAD, allows the user to choose which point cloud engine to use.



Tesseract (Carlson): The Tesseract engine is the recommended Carlson engine for all projects. All automated feature extraction tools including paint stripes, parking lines, and color edges, and new bare earth tools require the Carlson Tesseract engine. The Tesseract engine is the default engine for the Point Cloud application.

3DTK: The 3DTK engine is a legacy engine that should only be used when working on older projects that were created using this engine. Other limitations include the fact that it doesn't support the Leica .pts or .ptx formats.

Note: When opening a point cloud legacy project created with the 3DTK engine, with the **Tesseract (Carlson)** engine active, you will be prompted to upgrade the project to use the Tesseract engine.

Menu Location: Point Clouds > Point Cloud Setup

Keyboard Command: pc_setup

Prerequisite: None

Point Clouds Step-by-Step Tutorial

Tutorial Setup

This is an introductory tutorial for Carlson Point Clouds. If you wish to follow along with the tutorial by using the same Scan files, you can download them from the Carlson website at http://update.carlsonsw.com/tutorials/Carlson_Office.laz

You can also follow along with any other scans that have proper target points and control points.

Launch Point Cloud Manager

Work done in Carlson's Point Cloud module is done on a per-project basis. To create a new project, you must first have a CAD drawing open in AutoCAD or IntelliCAD. To enable Point Clouds, on the top toolbar click **Settings** => **Carlson Menus** and select **Point Clouds Menu**. This will display the Point Clouds menu on the top toolbar. Click on **Point Clouds** and then select **Project Manager**.

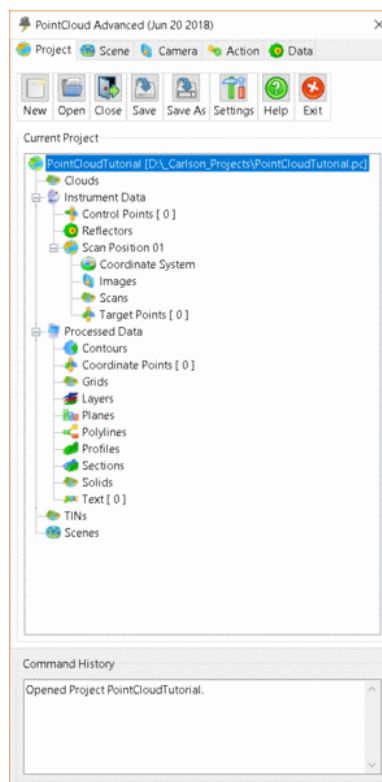
Alternatively, on the top toolbar click **Settings** and select **Toolbar Setup**. In the **Select Toolbars** window, ensure the **PointCloud** option is checked. This will now display the **Lightning Bolt** icon on the toolbar. Click the **Lightning Bolt** icon to launch Point Cloud Manager.



Project Setup

To create a new project, in the Point Cloud Advanced window, click **New**. In the New PointCloud Project window, select the file location to save the new project, enter in a File Name and then click on **Save**.

Your initial project should look like the following image. In the tree structure you will see your Project with various defaults.

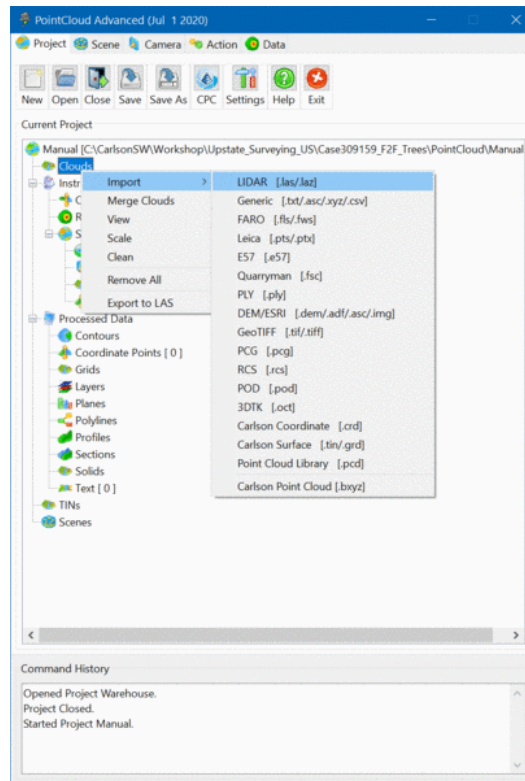


The program will attempt a recovery in the case of an error, but it is still recommended to **Save** often using the Save button at the top of the **Project Tab**.

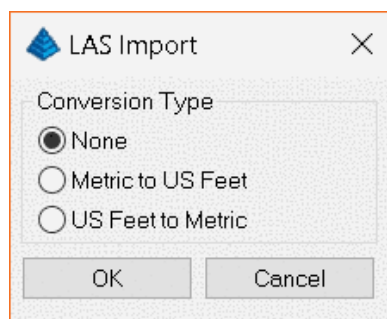
For more information see the **PointCloud Project Manager** entry.

Cloud Import

Let's begin by importing a Point Cloud. From the Project tree, right click on **Clouds** > **Import** > **LIDAR**.



Find and select the TutorialData.laz file, or use any other .las or .laz file. The program will ask if you want to convert units. This data is in feet, so if you want to work in Metric, select the third option. Otherwise leave this as "None" and click **OK**.



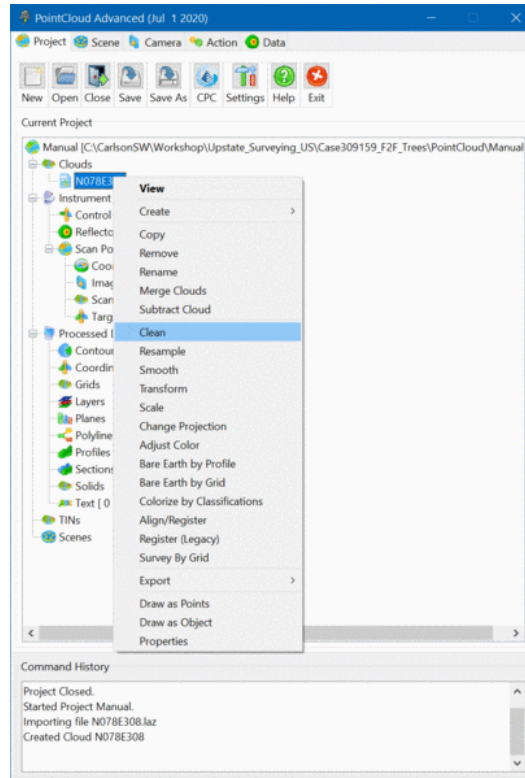
A new Cloud will now be displayed in the Project Tree.

For more information see **Import Cloud**.

Clean Cloud

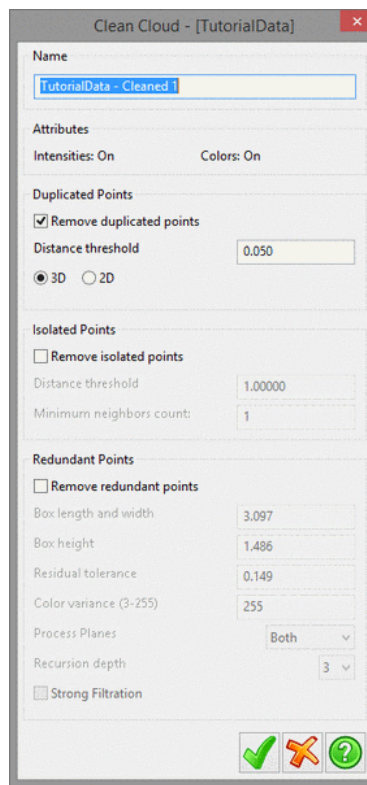
Many clouds which are imported directly from a laser scanner or exported from the manufacturer's software contain unnecessary points which can be removed.

Right-click on the newly-imported Cloud and select **Clean**.



For this type of scan, there are more points than necessary close to the scanner, so we remove them via the "Remove Duplicated Points" method.

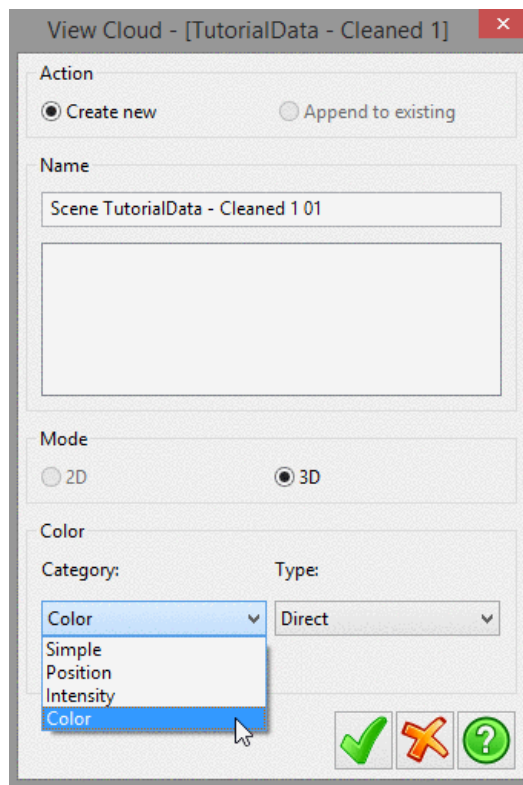
For this Cloud, we want to make sure that the "Remove Isolated Points" method is not enabled. Click the **Green Tick** to run the Clean operation.



For more information see **Clean Cloud**.

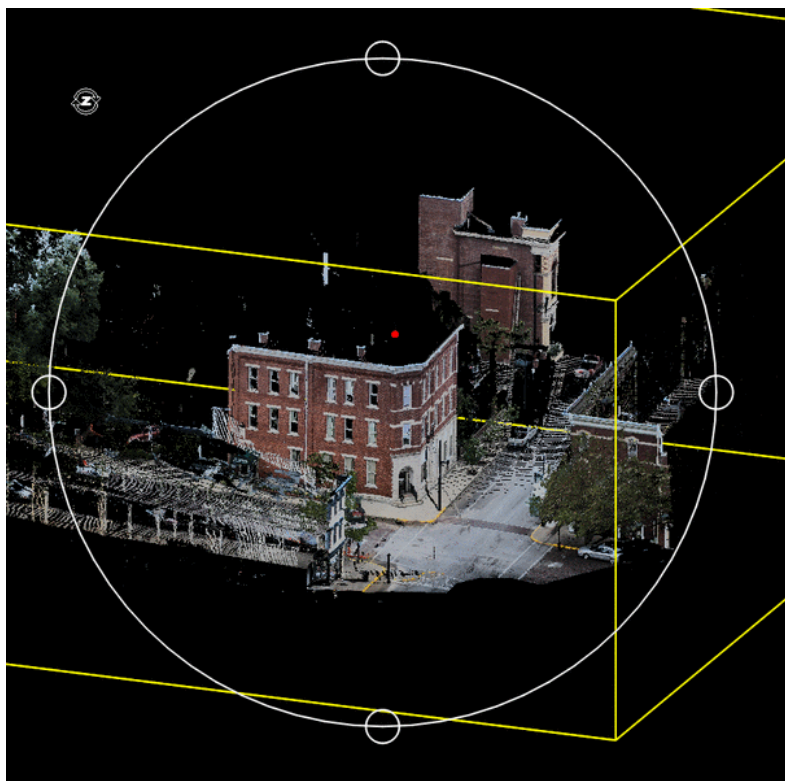
View Cloud

A new "Cleaned" Point Cloud will be available in the Project Tree. Double-click on the new Cloud or right-click and go to **View**.



This Cloud has color in it so we will choose **Color category: Color** and **Type: Direct** for our color mode. If this viewing mode is not available it may indicate that the Cloud does not have color in it. Click on the **Green Tick** to open the **View** window.

For more information, see **Scenes**.



For this screenshot, the Orbit has been displayed (via the Viewer Settings section of the **Project Settings Manager**).

By default, a left-click-and-drag within the bounds of the Orbit will change the y-angle of the scene, relative to the red pivot point.

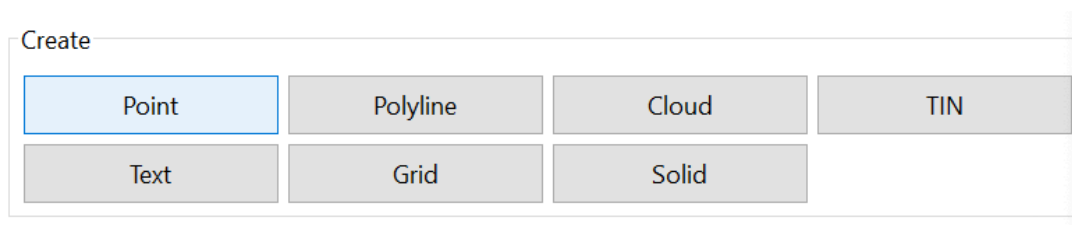
A left-click-and-drag outside of the Orbit will rotate the scene relative to the z-axis and the pivot point.

A middle-click-and drag will pan the scene.

The mouse scrollwheel can be used to zoom in and out.

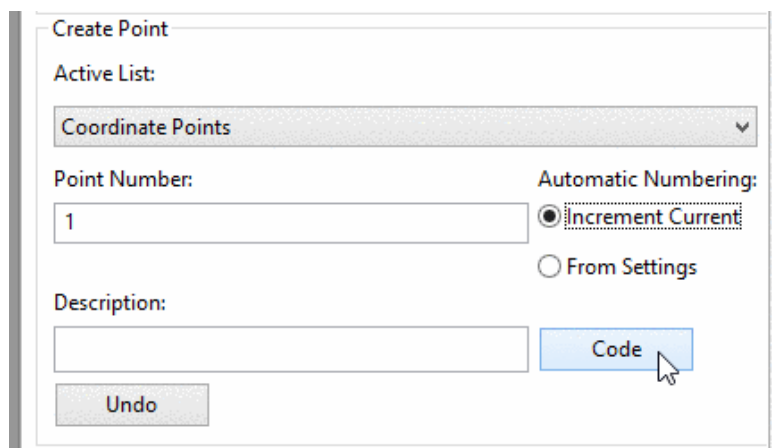
Virtual Survey

To enter the Create Point mode, select **Point** under **Create** on the **Action** tab of the **Project Manager**.



PointCloud will read your current .fld Code Table into a dialog which allows you to graphically select the code to apply to the next point.

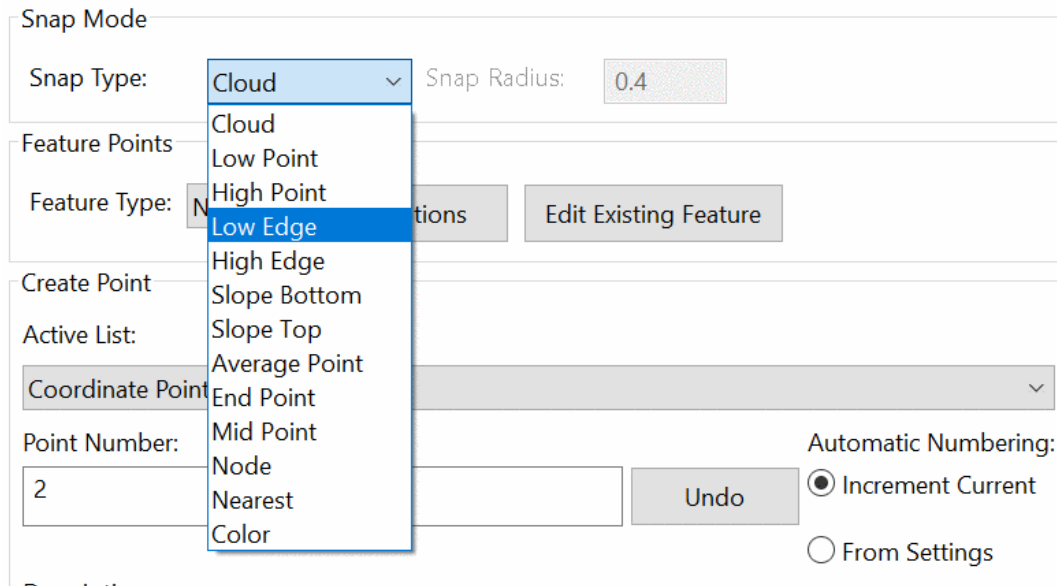
This dialog can be accessed via the **Code** button.



If the **Code Table** dialog is empty you may need to select a FLD File. To select a FLD File, click the **Code** button. In the **PointCloud - Code Table** window, click the **Pick FLD File** button at the top of the dialog then navigate to your FLD file and click **Open** to populate the data.

You can change the Snap Mode depending on what you are attempting to shoot a point of. In this example

we will select the **Low Edge** snap type in order to create some points for the bottom of the curb. The Snap Radius should be changed from 0.25 to 0.5 feet (if you are using meters, use a Snap Radius of 0.15 instead).



The code to use for this will vary depending on your code table, but it is important to pay attention to the Entity Type, as this controls whether a selected point shows up as an isolated point or as part of a polyline.

When the red pick dot appears at the bottom of the curb, hold down the **Ctrl** key and click the left mouse button to create a point. The program will automatically label this point as the start of the line and subsequent points with the same code continuing the line, until you end the line with **End Linework** or Ctrl-E.

Sending to CAD

After you are done with the Virtual Survey you may want to add the created points to your current CAD drawing.

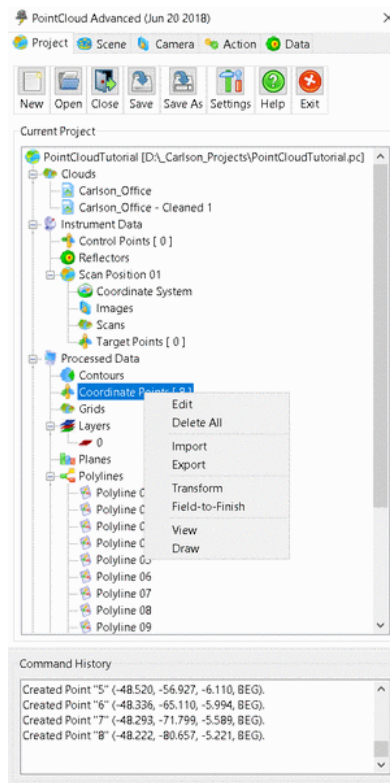
Return to the **Project** tab and select the entry for **Coordinate Points** from **Processed Data**.

To edit the points or view their coordinates you can right-click and select **Edit**, or double-click to open the editor window.

Right-clicking gives a few more options:

Field-to-Finish will use your Carlson F2F settings for styles and colors when sending to CAD.

Draw will simply send the points to CAD as points, based on the settings of the chosen Layer.

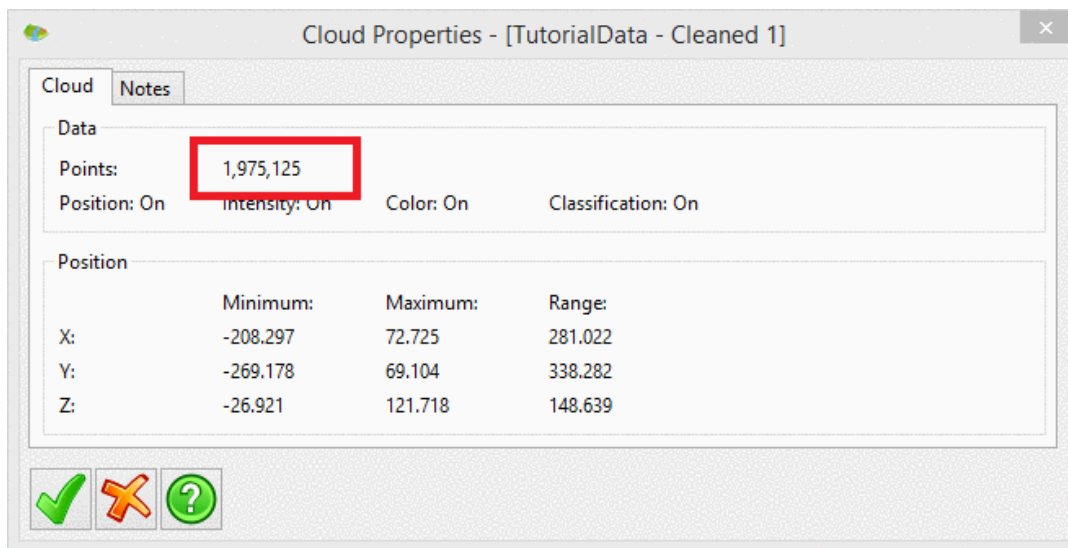


For more information see **Coordinate Points**.

TIN Creation

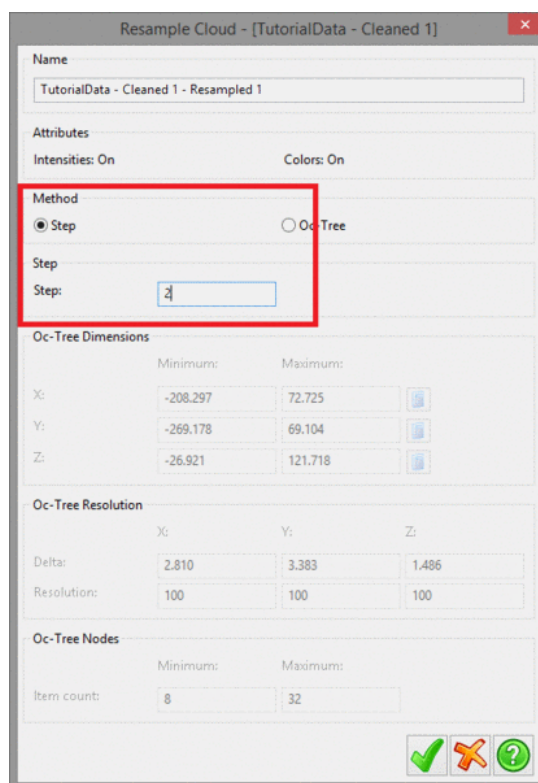
TIN files work best with about 1,000,000 or less points, and in most cases where they are used, this is more than enough points to be useful. So it can be helpful to reduce the number of points in a Cloud before creating a TIN.

First, right-click on the already-cleaned Cloud file in the Project Tree and click **Properties**.

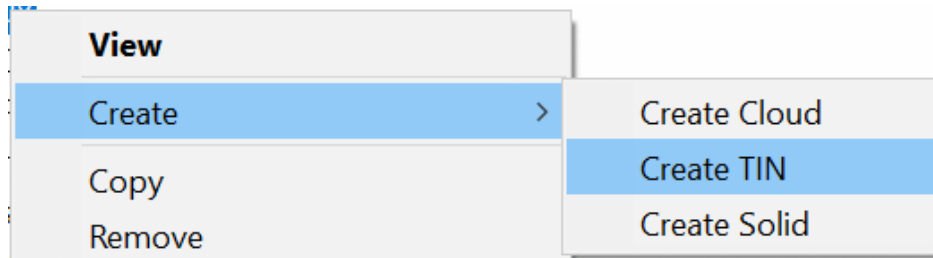


Here you can see the number of points in the current Cloud so you know how many need to be removed. Click the **Green Tick** to exit the **Properties** window.

Right-click on the Cloud again and select **Resample**. We are using a step size of 2 because we have approximately 2 million points, and having already Cleaned the Cloud we assume the density is relatively even. Click on the **Green Tick** to run the Resample operation.

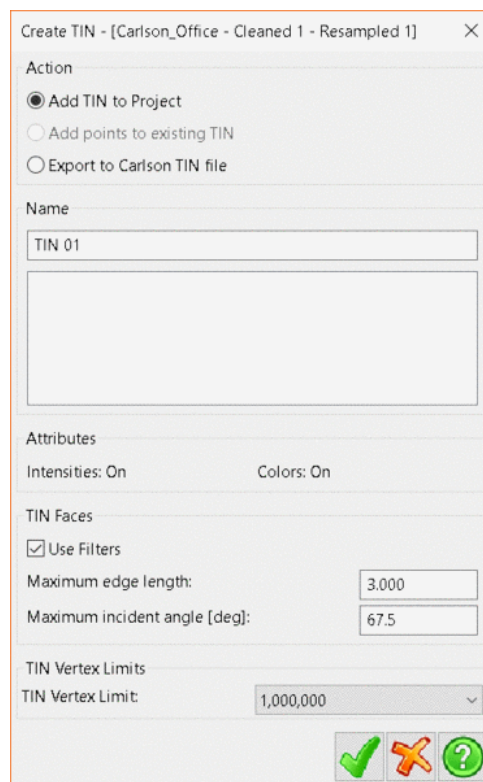


Now that we have created a more manageable Cloud, we are ready to create a TIN. Right-click on the Resampled Cloud in the Project Tree and select **Create > Create TIN**.



The maximum edge length should be a few times longer than the normal distance between points in your Cloud. The incident angle controls how tilted a triangle in the TIN can be relative to the Normal. In this Cloud, setting this below 90 will cut off the sides of buildings.

Click the **Green Tick** to create the TIN. You can also choose to Export the TIN at this stage.



The newly-created TIN can be used in other functions which work on triangulated data.

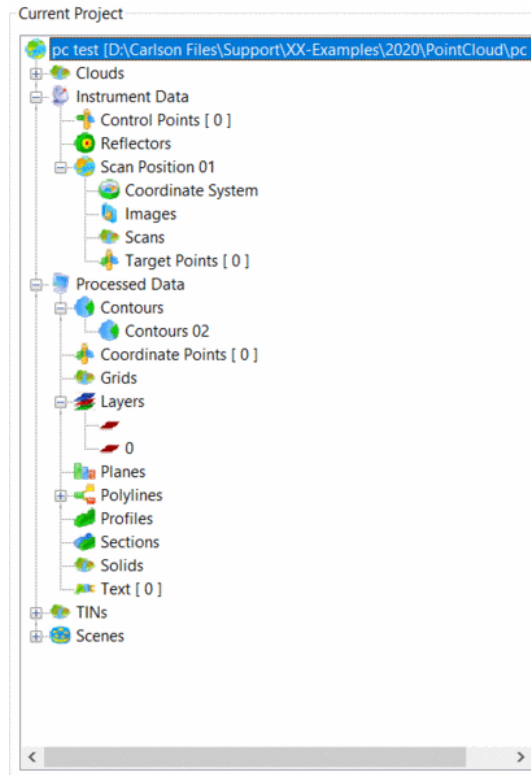
This completes the tutorial, but this is only as small sample of the functionality available in Carlson Point-Cloud.

If you would like further assistance, please contact support@carlsonsw.com.

Point Clouds Project Manager

The project manager is the central place of operations for Carlson Point Cloud. To bring up the Project Manager, first ensure that the Point Cloud module is loaded by clicking on the lightning bolt icon in your CAD application. The manager can be brought up by either clicking the **Point Clouds > Project Manager** or by entering the command **PC_TOOLBAR1** into your CAD applications command window. The title bar of the Point Cloud Manager window will display the version, build date of the software and the active Point Cloud engine. Tesseract (Carlson) is the default Point Cloud engine.

Every function and operation that can be performed in Point Cloud is accessed through the Project Manager. These functions are divided up among five major categories represented as tabs in the project manager: **Project, Scene, Camera, Action, and Data**.



Most Point Cloud features require you to have a Point Cloud Project open. These are stored as .pc files. You can manipulate them on the **Project Tab**.

Command History

The **Command History** panel maintains a list of all major functions used since execution of Point Cloud began, and is also the central place where information regarding the results of all functions is posted, much like the command window in AutoCAD or IntelliCAD. For instance, after execution of the TIN simplification function has finished, details about the number of faces removed from the TIN will be appended to the Command History.

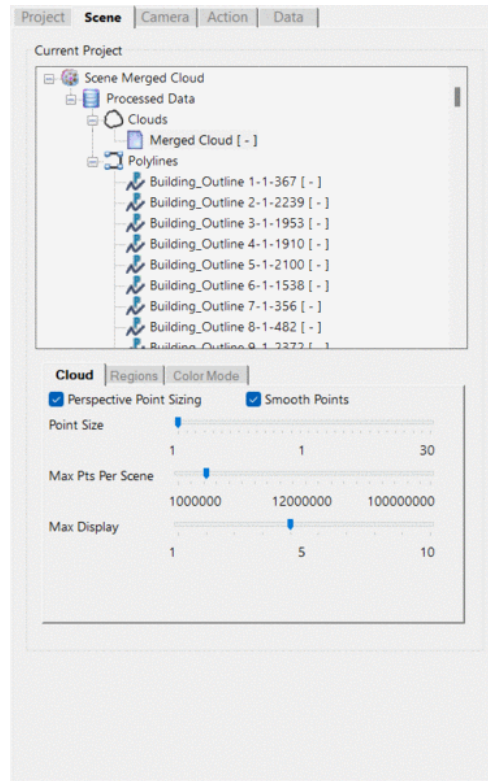
Pulldown Menu Location(s): Point Clouds

Keyboard Command: pc_toolbar1

Prerequisite: None

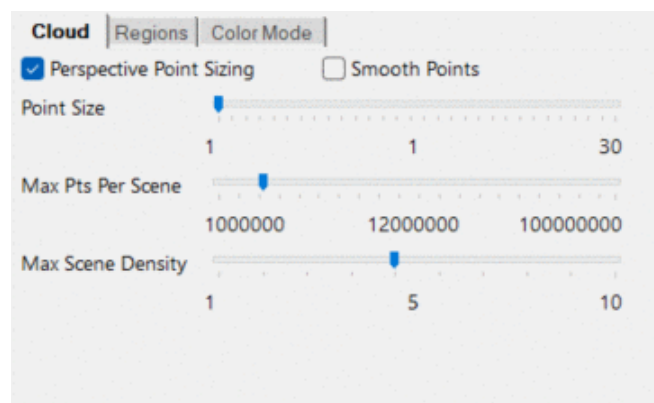
Scene Tab

The **Scene Tab** allows you to customize the active scene's display options. The **Scene Tab** will be blank until you open a scene from the **Project Tab** using the *View* command. It is important to note that the controls in the Scene Tab vary depending on whether objects are selected in the Current Project tree or selected in the active scene window.



To display the cloud controls, **select a cloud** from the **Current Project** tree. Using the cloud controls you can set the point size and display, hide/show regions (classifications), and set the color mode to display the cloud in the scene (direct color, intensity, elevation, etc).

Point Size



Perspective Point Sizing: will make points farther from the camera appear smaller than points closer to the camera when the Scene viewer is set to *Perspective Mode*.

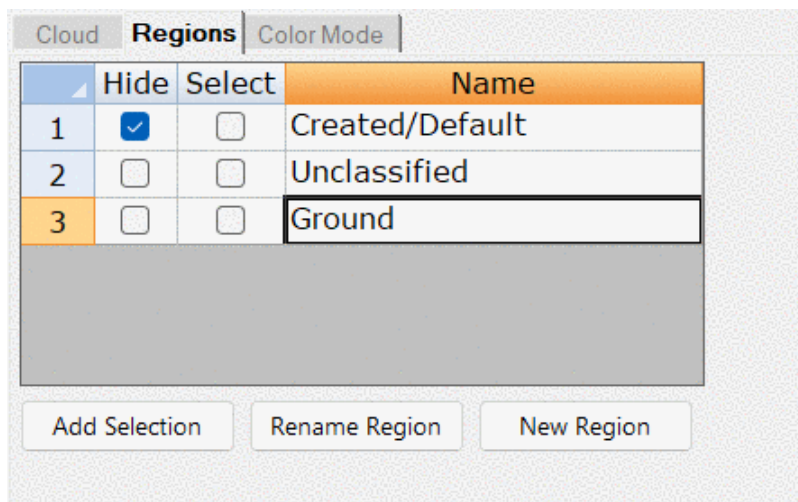
Smooth Points: Changes the way the points are displayed from a square to a circular appearance which makes the point display on screen appear fuller. The default is on. If you prefer viewing the points at standard size, uncheck Smooth Points.

Point Size: **Point Size** controls the size of points in the Scene viewer and can be set from 1-30. Default is set to 3, if you prefer displaying the points at standard size, set the size to 1.

Max Pts Per Scene: Controls the maximum number of points that can be displayed in the scene allowing to cull the cloud display and improve the graphic performance.

Max Scene Density: Controls the density of points in the view prior to limiting the overall scene. If the density is low, the cloud may appear thinner but will display faster. If the density is high, the display will show more points in the denser areas but it may reach the *Maximum Points Per Scene* quicker.

Regions



Hide: toggle on/off to hide or display the classified region.

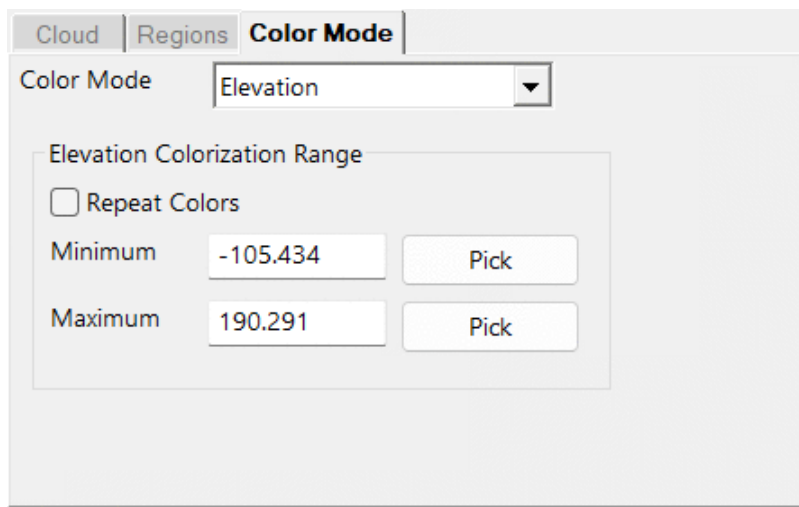
Select: toggle on/off to select all the points in the region.

Add Selection: adds the selected points in the scene to the highlighted region. Point that were in a different classification will be moved to the target region.

Rename Region: renames the highlighted region.

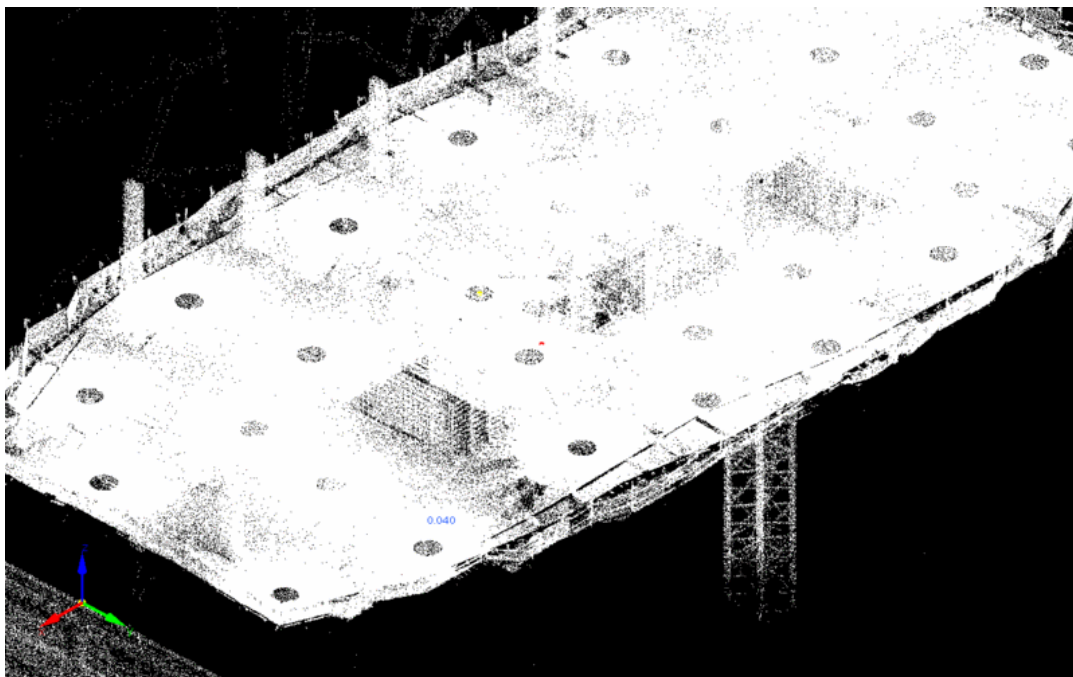
New Region: a new region can be created by first making a selection from one or more existing regions, or by selecting a portion of the cloud from the Action tab and then creating a region from that selection. If no points are selected, the new region will contain no points.

Color Mode

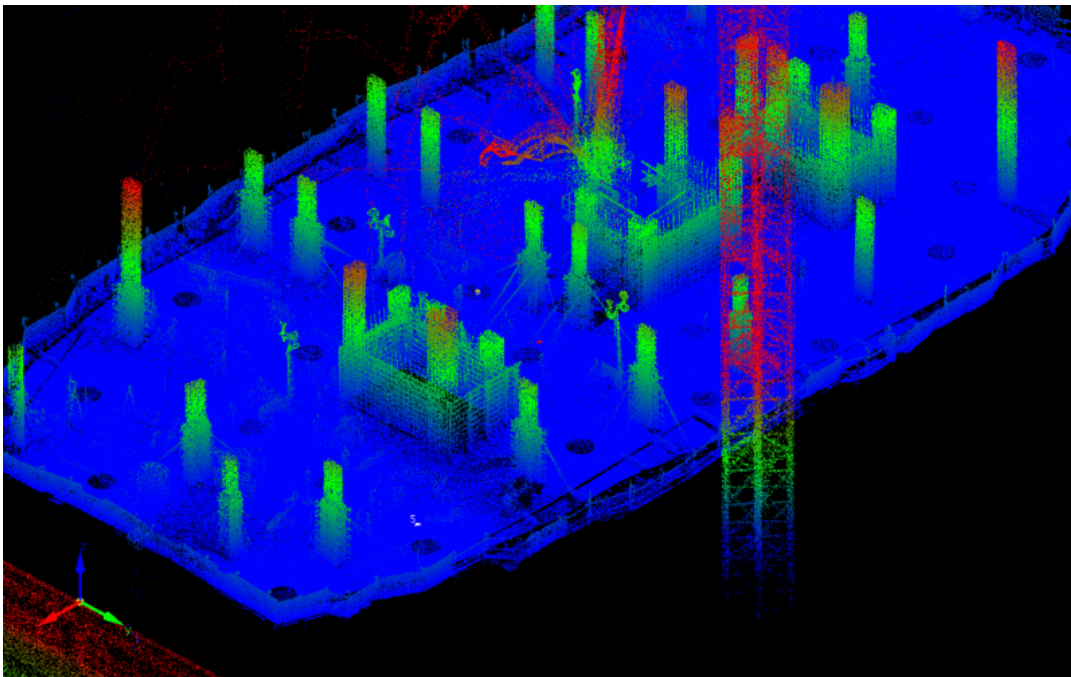


The **Color Mode** determines the mode to color the data in. There are eight modes that are contextually available depending on the type of data and selected mode to be drawn in. These color modes are divided into categories based on how the colors are determined, these categories are: **Single Color**, **Elevation**, **Intensity**, **Colorized Intensity**, **Direct Color**, and **None**.

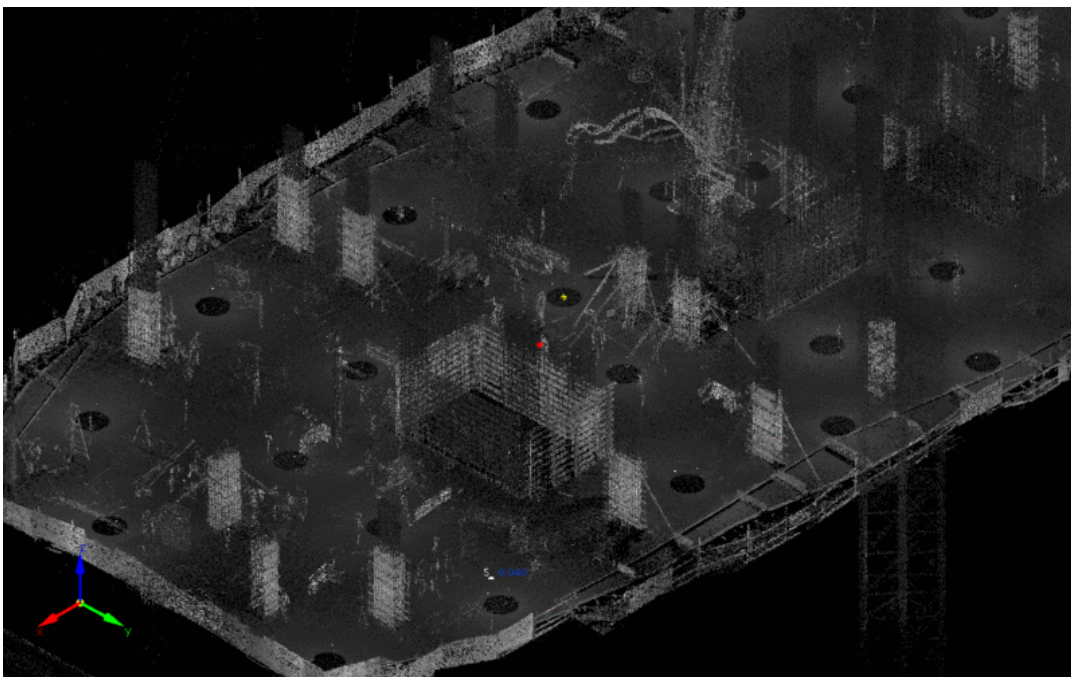
Single Color: will color the entire dataset in the same color. **Click** the **Select** button to choose the color. Default is white.



Elevation: colors each point based off its elevation data in a range from blue being the minimum elevation to red being the maximum elevation. By default, the program will scan the cloud to determine the minimum and maximum elevations and apply the colorization. You can enter a **Minimum** and **Maximum** elevation by typing the values in the corresponding fields or by clicking the **Pick** button to select points in the cloud to affect the colorization. Toggle **Repeat Colors** to allow the software to repeat the color bands.

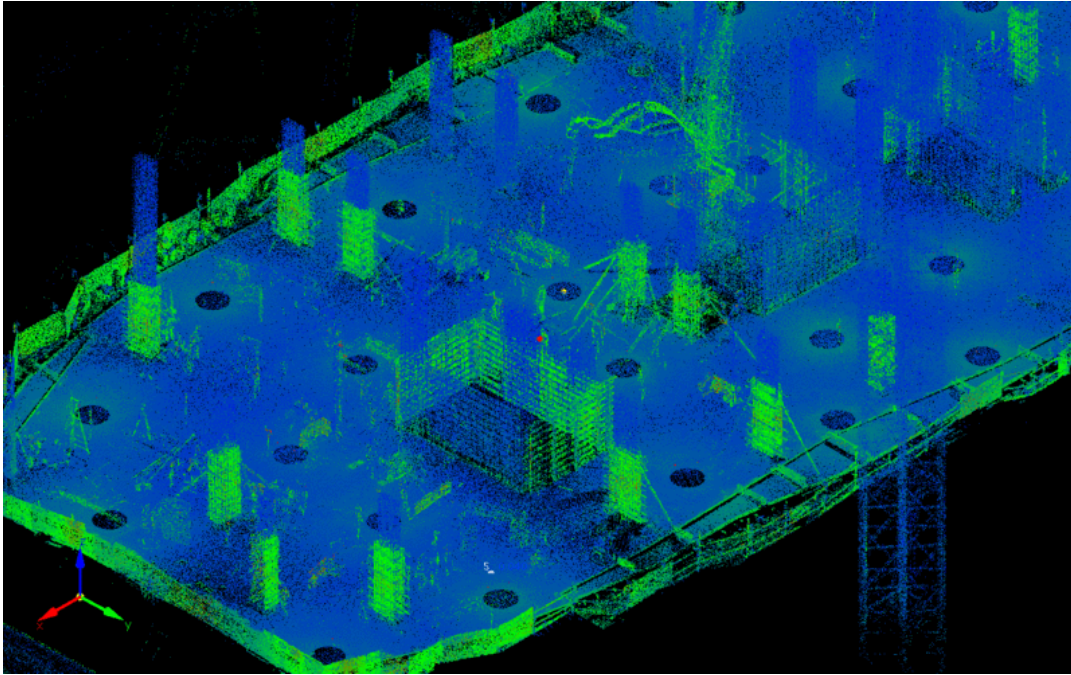


Intensity colors each point based off its single intensity value. Points with a low intensity will be colored pure black while colors with high intensity will be colored pure white. The **Set Range to** options allow you to customize the intensity range to be applied. **Full** will set the Min value to 0 and Max to 255. **Cloud** will analyze the cloud for the lowest and highest intensity values and automatically set the Min/Max values. **Optimized** will analyze the cloud for the lowest and highest intensity values and then apply a Min/Max set of values that will emphasize the intensity differences the most. **Custom** will allow you to enter the Min/Max values. Note that some ground scanner clouds may only have low/high intensity values therefore the range will be limited.

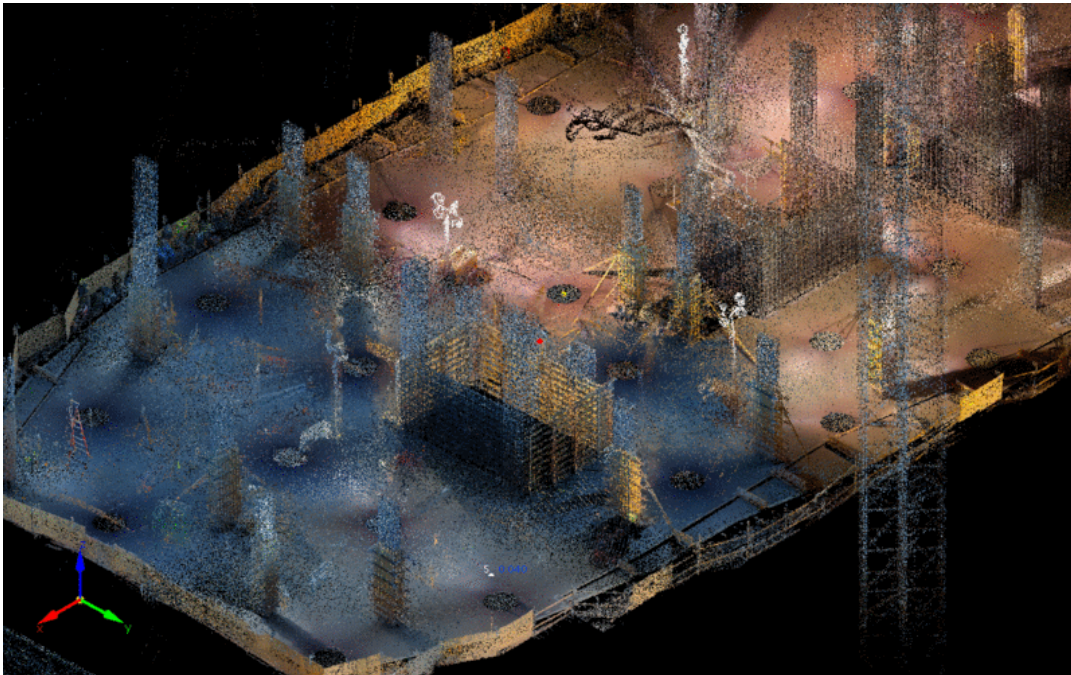


Colorized Intensity: colors each point based off its single intensity value using red, green, and blue col-

orization. The **Set Range to** options allow you to customize the intensity range to be applied. **Full** will set the Min value to 0 and Max to 255. **Cloud** will analyze the cloud for the lowest and highest intensity values and automatically set the Min/Max values. **Optimized** will analyze the cloud for the lowest and highest intensity values and then apply a Min/Max set of values that will emphasize the intensity differences the most. **Custom** will allow you to enter the Min/Max values. Note that some ground scanner clouds may only have low/high intensity values therefore the range will be limited.

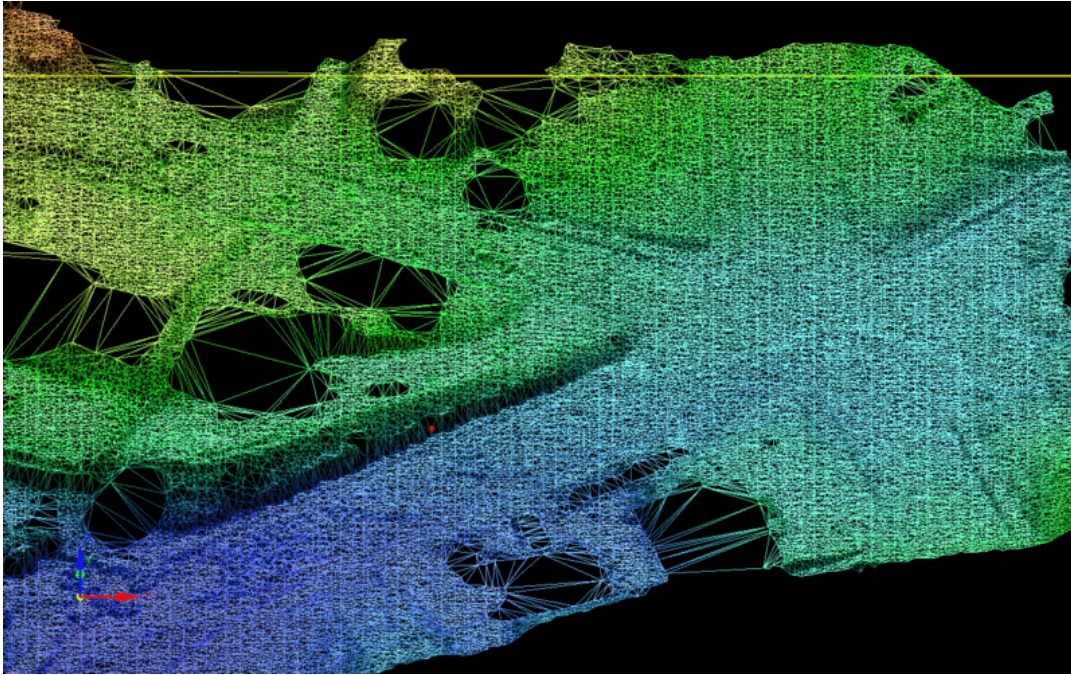


Direct Color: colors the points based off the color information in the imported cloud data.

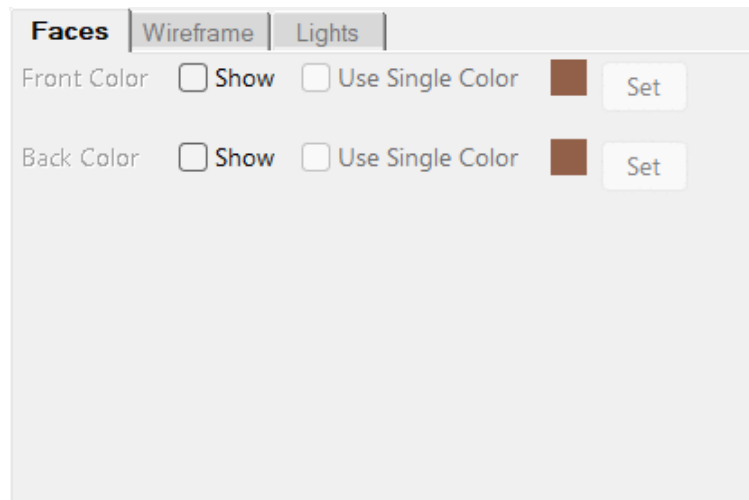


None: removes colors.

To display the TIN controls, **select** a **TIN** from the **Current Project** tree. Using the TIN controls you can set how to display the TIN faces, wireframe, and lights.



Faces

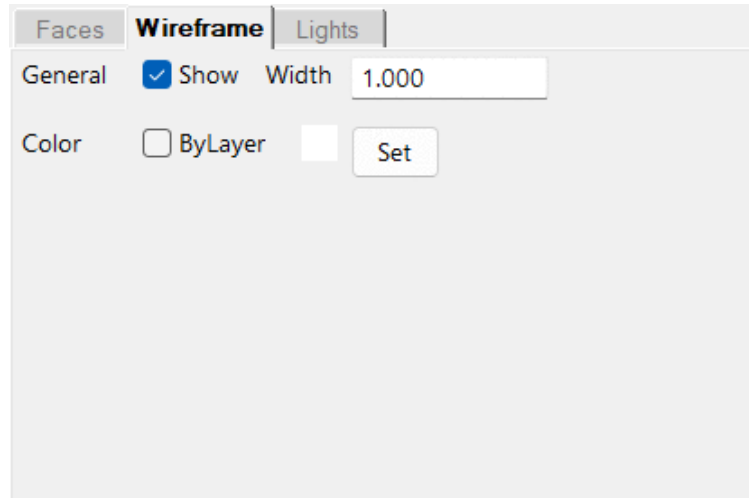


Show Front: toggles the display of the front TIN faces in the active scene viewer.

Show Back: toggles the back (or underneath) faces of the TIN in the active scene viewer.

The **Front** and **Back** colors can be set separately and different colors can be selected when toggling **Use Single Color** and then **clicking** the **Set** button.

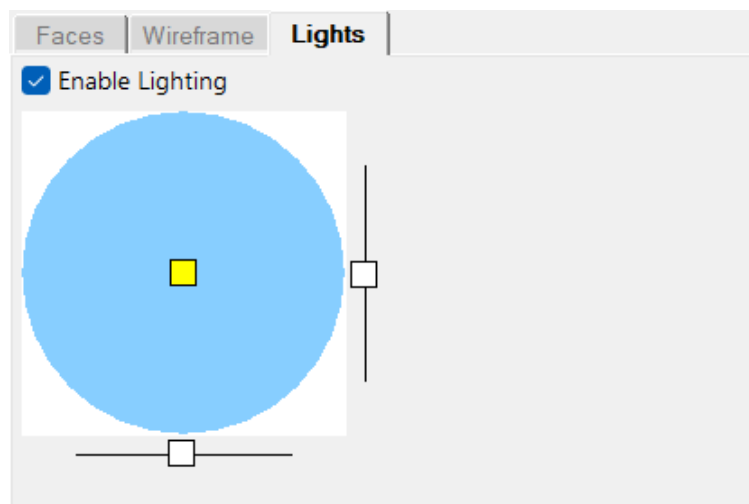
Wireframe



Show: Toggles on/off the display of the TIN wireframe (or mesh).

Color: Allows you to set the wireframe color whether to a single color or multiple colors based on elevation ranges.

Lights



The two sliding scales control the Light Intensity (vertical control) and the Ambient Intensity (horizontal control).

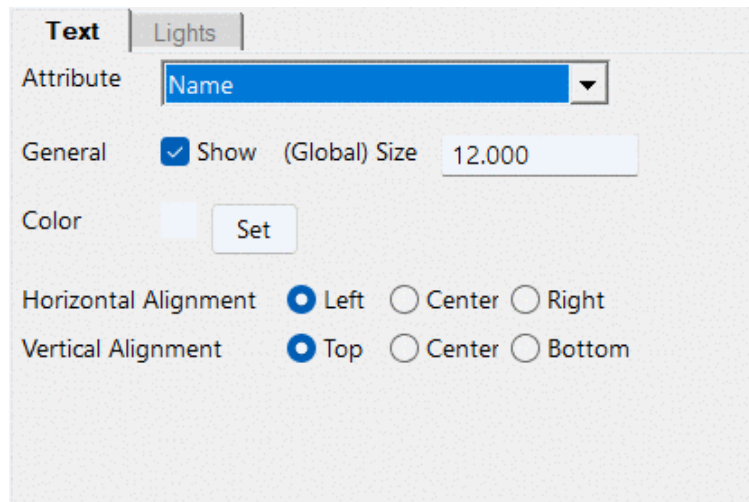
Light Intensity: Controls the intensity of the directional light, which is a light, positioned to simulate the effect of a distant light source like the sun.

Ambient Intensity: Controls the intensity of the ambient lighting, which is the base light level that affects everything

regardless of the surface normal.

Center Light control can be moved freely to change the position of the light source.

To display the Carlson Point Attribute controls, **select Coordinate Points** from the **Current Project** tree under **Processed Data**. You can set to display the attributes, their text size, color, and justification.



Attribute: use the pull-down menu to select **Name**, **Description**, or **Elevation**.

Show: toggle on/off to set if the attribute will be displayed in the scene.

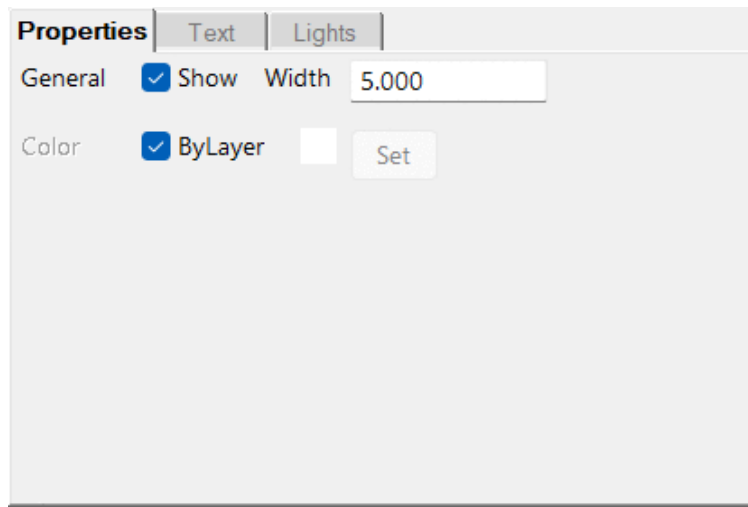
Global Size: adjust the attribute text size.

Color: set the attribute color by **clicking Set**.

Horizontal Alignment: set the attribute's horizontal justification.

Vertical Alignment: set the attribute's vertical justification.

To display the Polyline controls, **select Polylines** from the **Current Project** tree under **Processed Data**. You can turn on/off the polylines, set the line width, and set the polyline colors.

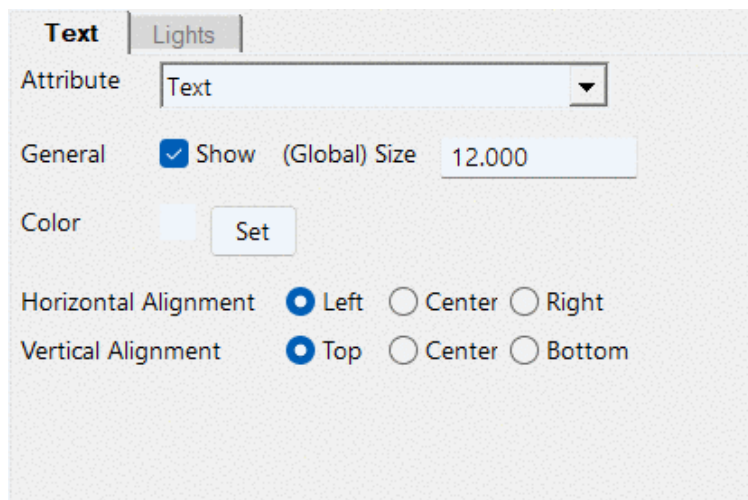


Show: toggle on/off the display of the polylines.

Width: set the line width.

Color: you can set a color to display the polylines.

To display the Polyline controls, **select Text** from the **Current Project** tree under **Processed Data**. You can turn on/off the text, set the text size, color, and justification.



Show: toggle on/off to set if the attribute will be displayed in the scene.

Global Size: adjust the attribute text size.

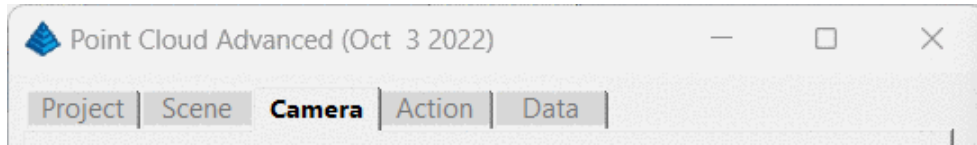
Color: set the attribute color by **clicking Set**.

Horizontal Alignment: set the attribute's horizontal justification.

Vertical Alignment: set the attribute's vertical justification.

Camera Tab

The **Camera Tab** is the third tab in the Project Manager and has several functions that govern the current Scene's camera behavior and positioning as well as functions that allow access to some camera presets. The Camera Tab will only be active if there is a Scene currently opened, otherwise all of its controls will be greyed out.



The Mouse Motion Panels

The **Left Motion**, **Middle Motion**, and **Right Motion** panels control the behavior of the left, middle and right mouse buttons, respectively. The options in the pull-down menu are Cub Orbit, Orbit, Pan, Swivel and Zoom.



Cube Orbit rotates the camera around the center of the scene's bounding box, the default behavior for the left mouse button.

Orbit will orbit the camera around the pivot, and the pivot's behavior can be defined in the **Pivot** panel.

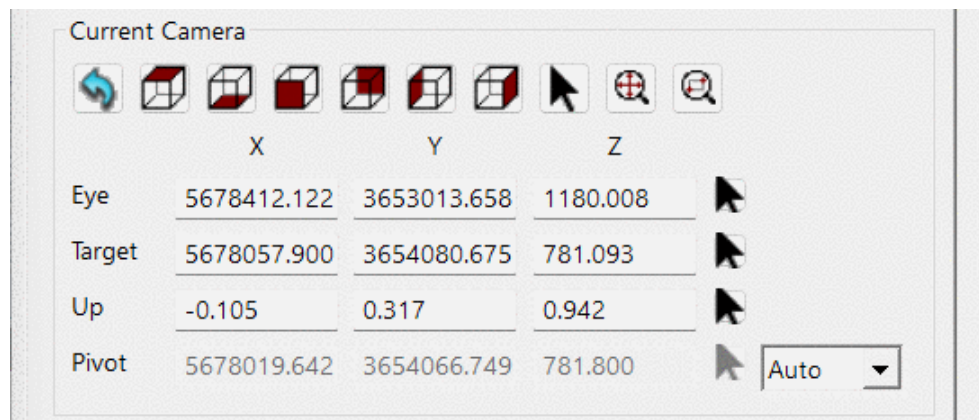
Pan pans the camera, moving the camera laterally or vertically from its perspective, the default behavior for the middle mouse button.

Swivel rotates the camera view direction around its current position.

Zoom will zoom the camera, moving the mouse up will zoom in, moving the mouse down will zoom out. This is the default behavior for the right mouse button.

The Current Camera Panel

The **Current Camera** panel has options for manually modifying each of the camera's position and view direction properties. Each option features a target button which, when pressed, will allow you to CTRL+click into the scene to set that property to a value from the scene. The preset views for the Camera are accessed by clicking the icons along the top of the panel. From left to right they are; Reset View, Top, Bottom, Front, Back, Left, Right, Pick, Zoom Extents, and Zoom Window.



Eye is the position of the current camera.

Target is the current facing direction of the camera.

Up is the current camera's up vector.

Normal is the 90 degree vector to the current view plane.

The **Pivot** determines the behavior of the pivot point when in **Orbit** mode. Use the drop-down menu to select the different Pivot behaviors.

Auto, the default behavior, moves the pivot to the valid data nearest to the cursor whenever orbit mode is activated.

Center will move the pivot point to the center of the scene's bounding box.

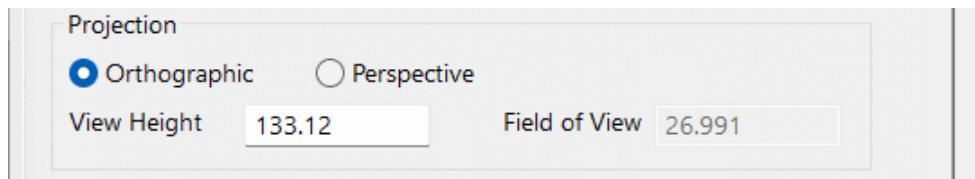
User will enable the **Position** controls and allow entry of the Pivot point manually (or via a CTRL+click in the scene if you press the green target button).

Origin will move the pivot point to the origin of the scene's coordinate system.

Target will place the pivot point at the nearest valid data to the center camera's view.

The Projection Panel

The **Projection** panel determines the projection mode of the current camera.



Orthographic projection

mode will not apply perspective distortion (to simulate the effect of a real-world camera lens) and the **View Height** option will determine the height of the view (in world units).

Perspective projection mode will apply a perspective distortion, and the **Field of View** option determines the vertical viewing angle of the cone that defines the camera's visibility.

The Clipping Panel

The **Clipping** options allow the user to manually set up of the view's clipping planes. Clipping planes will cut out any objects in the Scene closer than the front clip plane and any objects further than the back plane. **Note:** By default, custom clip planes are disabled.

In some cases, it may be useful to manually manipulate the clip planes.

Front defines the distance from the front of the Bounding Box that the camera clip will hide. Adjust the **Front** slider to clip more or less of the Scene, starting from the front.

Back defines the distance from the back of the Bounding Box that the camera clip will hide. Adjust the **Back** slider to clip more or less of the Scene, starting from the back.

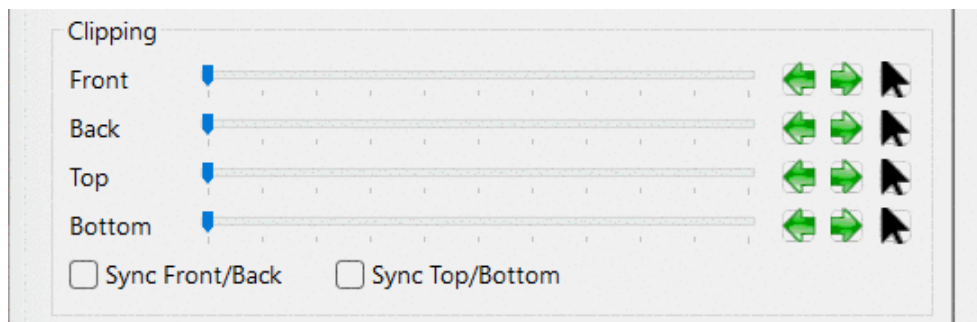
Top defines the distance from the top of the Bounding Box that the camera clip will hide. Adjust the **Top** slider to clip more or less of the Scene, starting from the top.

Bottom defines the distance from the bottom of the Bounding Box that the camera clip will hide. Adjust the **Bottom** slider to clip more or less of the Scene, starting from the bottom.

Sync Front/Back - clicking in the check box will mean that as the Front or the Back slider is adjusted, the opposite value will move in the opposite direction by the same amount.

Sync Top/Bottom - clicking the check box will that as the Top or the Bottom slider is adjusted, the opposite value will move in the opposite direction by the same amount.

The **Target** buttons next to each value can be used to define the value of each clip by clicking directly into the Scene.

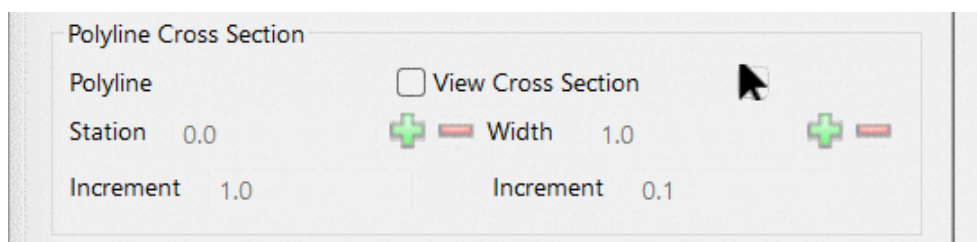


Polyline Cross Section

This clipping method uses an existing polyline to clip the point cloud. Select the desired polyline using the **Polyline** drop-down menu. To undo the Cross Section view, select the **Blank** option from the drop-down menu and click anywhere in the Scene.

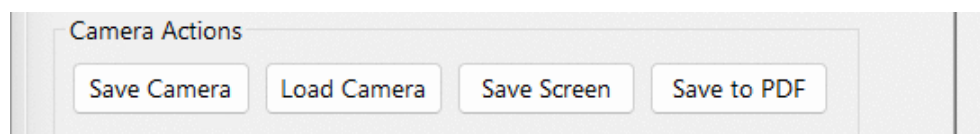
Station - Increase or decrease the **Station** value to move the cross section through the Cloud.

Width - Increase or decrease the **Width** value to widen or narrow the cross section.



Camera Actions

Additionally, there is an option to save the current camera settings to the current scene (which can later be accessed through the scene page) using the **Save Camera** button and an option to save an image of the current contents of the viewer using **Save Screen**, which will bring up a save file dialog. Pressing **Save Screen** will bring up the standard Carlson save file dialog and you can choose where to save the image as a BMP file. The final option is to Save to PDF. Pressing the **Save to PDF** button opens the PDF options. The user can choose size of image, Border and title block fields to be included in the PDF file. The current view in the scene is written to the PDF.

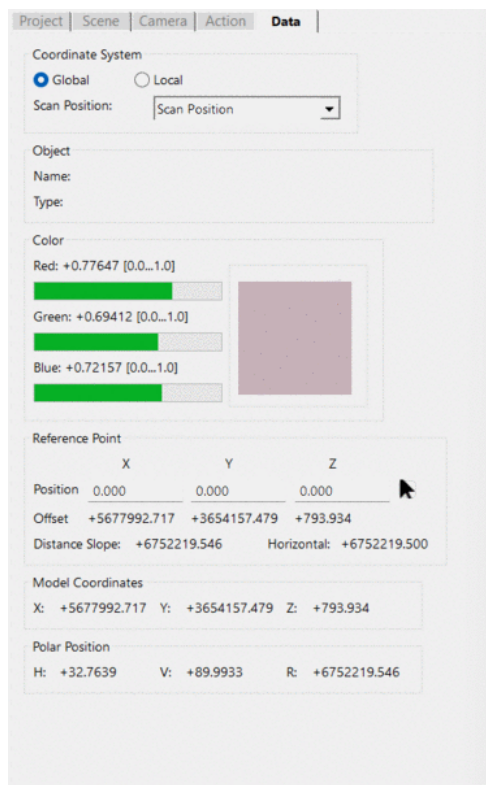


Tab Location(s): Camera Tab

Prerequisite: Open Scene

Data Tab

The Point Cloud Data tab is useful for visual inspection of data in the current scene. It allows the user to examine, color, range, and position values in the scene's coordinate system as well as in coordinate systems relative to the scanner positions or a reference point.



The **Coordinate System** panel displays the Cartesian coordinates (in the currently selected coordinate system) of the point under the cursor if it is a valid data point. By default, all data inspection is done in the Scene's global coordinate system. However, it is also possible to work in coordinate systems relative to the scanner positions by clicking the **Local** radio button and choosing a scan position to use as basis for the coordinate system.

The **Object** panel displays the name and object type of the current object under the cursor.

The **Color** panel displays the color values of the pixel under the cursor, if valid. Keep in mind that this value will not be the exact color value of the dataset if lighting is enabled or if the object under the cursor is selected.

The **Reference Point** panel displays options for placing the reference point and information about the point under the cursor's relative position to the reference point. This is useful for rough estimation of range values without having to create a polyline in the project. Clicking the small green target icon will put the current Scene into reference point positioning mode. CTRL+Click in the Scene to place the reference point at the valid data nearest to where the user clicks.

The **Polar Position** panel displays the current polar coordinates of the point under the cursor if it is a valid data point.

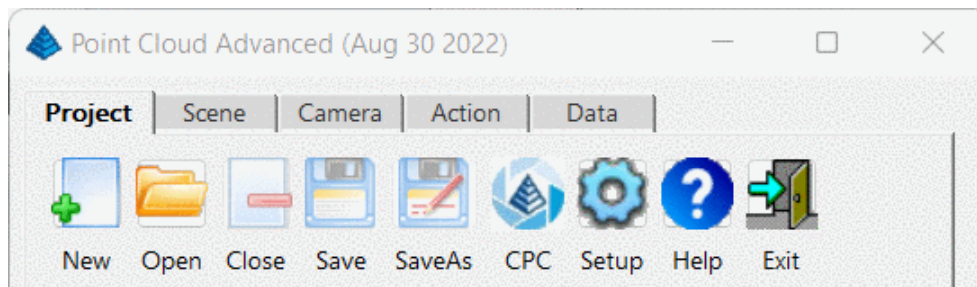
Project Tab

The **Project Tab** is the main tab of the Point Cloud Project Manager, and should be the first thing you see after pulling up the Project Manager. It is from here that you can manage your project files, as well as perform most of the operations on your datasets that affect them in their entirety. Interaction with items in the project is done by right-clicking them to bring up a contextual menu with all the actions that can be performed on them.



The Project Files Panel

This group of buttons all relate to the current working project. If there is no current working project the **Close**, **Save** and **Save As** buttons will be inactive.



New Begins a new project. If there is already a project open the user will be prompted to save changes, close it, or cancel.

Open Opens an already existing project. If there is already a project open the user will be prompted to save changes, close it, or cancel.

Close Closes the current open project. If there are any unsaved changes, the user will be prompted if they wish to save them.

Save Saves the currently open project.

Save As Saves the currently open project to a new name specified by the user. Take note, this will not create new copies of the TIN, Scan, and Cloud files.

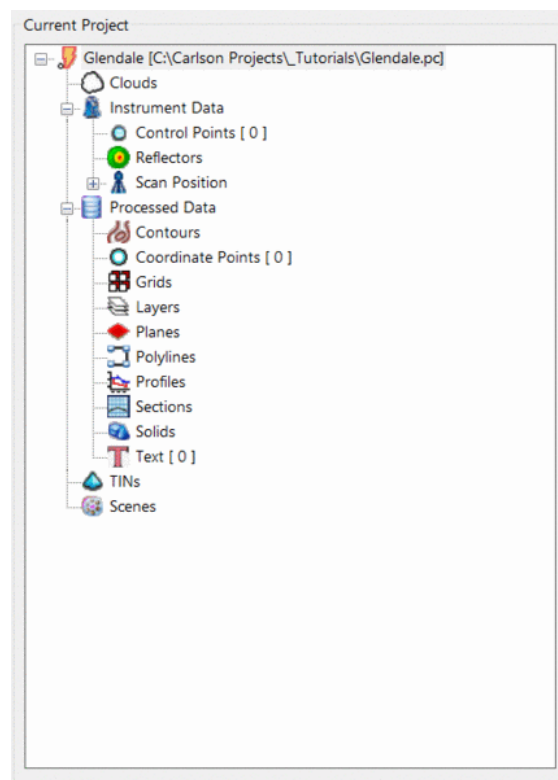
CPC opens Carlson Photo Capture online, Carlson's photogrammetry program that will process image files from a drone into a point cloud. CPC will also generate an orthoimage, DEM, DTM, and project quality report.

Settings Opens the settings dialog box - See **Settings** for further information

Help Opens the Help files for Point Cloud

Exit Leaves the Point Cloud manager

The Project Tree



Projects in Carlson Point Cloud are arranged into a tree structure. Items in projects are divided into two separate

categories: **Instrument Data** and **Processed Data**. **Instrument Data** consists of data in its raw form directly from the scanner that needs to be registered to a common coordinate system. This largely consists of raw **Scans** and the data associated with them (**Target Points**, **Control Points**, **Reflectors**, **Images**, etc). **Processed Data** is data that has been registered to a common coordinate system. **Clouds**, **Contours**, **Coordinate Points**, **Grids**, **Layers**, **TINs**, **Planes**, **Polylines**, **Profiles**, **Sections** and **Text** all fall under this category. **Clouds** are the central data objects to Carlson Point Cloud and upon which most operations are performed.

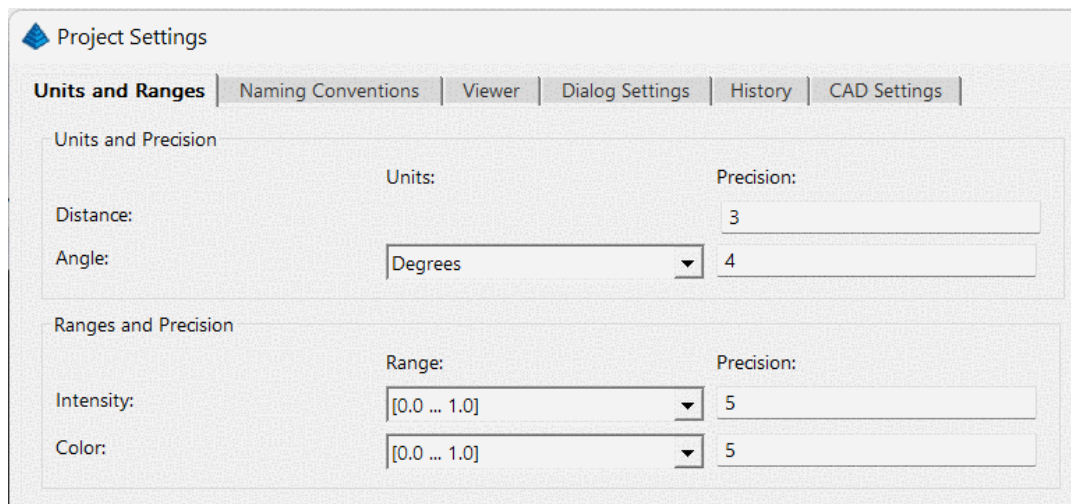
If you have not done anything else yet, you will need to **Import** a point cloud to get started.

Project Settings

The Project Settings dialog allows the user to modify various project-wide settings that fall under several categories: Units and Ranges, Naming Conventions, Viewer, Dialog Settings, History, and CAD Settings. It is usually best upon creation of a project to first go to this dialog and make sure that these settings are ideal for the work to be done.

Units and Ranges

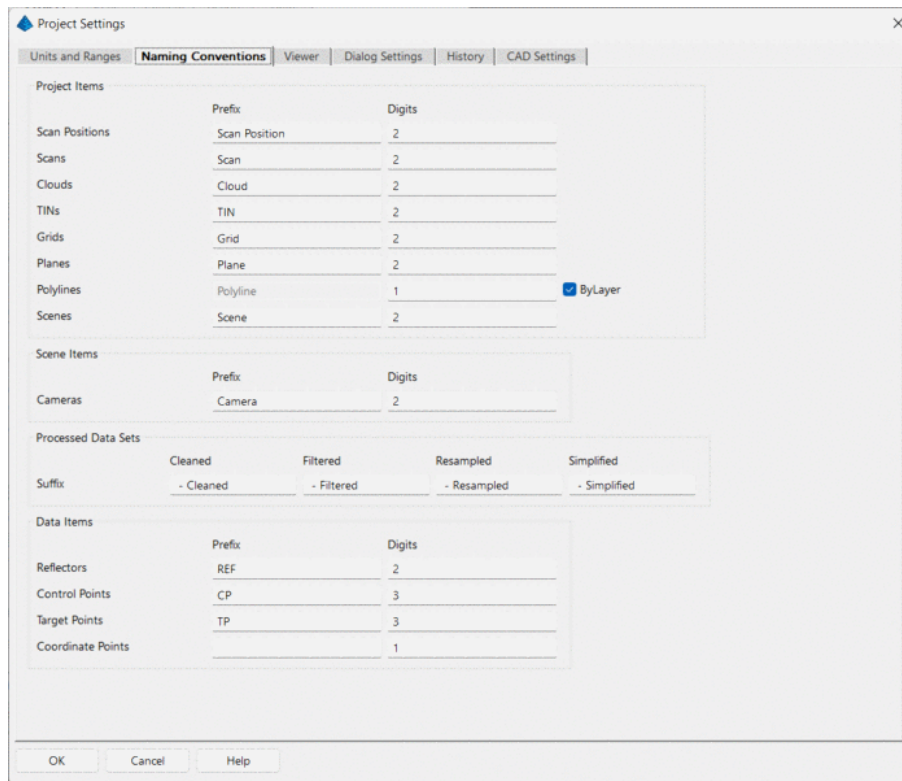
The Units and Ranges page of the Project Settings dialog allows the user to specify their desired working units and their desired ranges and precision for color values retrieved by the scanner.



These values will be internally used by Point Cloud and any values exported from Point Cloud or drawn to any CAD software will be in these units. Point Cloud no longer differentiates between feet and meters because most scanners do not include this information in their exported scans making it impossible to automatically convert. If you need to convert between distance units, use the **Scale Cloud** command.

Naming Conventions

The Naming Conventions page of the Project Settings dialog allows the user to specify the default naming behavior for each of the major item types in Point Cloud.



By default, when objects are created they will be given the names specified in the Naming Conventions dialog followed by the smallest number with the specified number of digits that will give it a unique name. For instance, using the above settings, if a project already has "TIN 01" and "TIN 02" objects and the user creates a third TIN, this third TIN will be given the default name of "TIN 03". Additionally, the user can also specify suffixes to be added to datasets after they have been processed in some manner. Given the above settings, if a TIN, "TIN 01", were simplified with the above settings its simplified TIN would be given the default name of "TIN 01 - Simplified 01". In most cases where new objects are created, the user will have the option to change the new object's name from the one automatically generated from these settings.

Viewer

The Viewer page of the Project Settings dialog allows the user to customize various settings of how the Point Cloud viewer window looks.

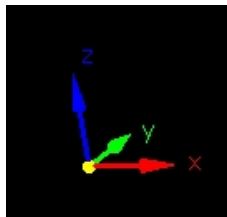


General

General settings allow you to define the background and selection colors. Toggling View Multiple Scenes will allow you to have multiple scene viewer windows open simultaneously - default is off which will keep a single scene window open which makes it easier to know which point cloud is being edited at any time. Show Warnings will enable displaying messages (default is on).

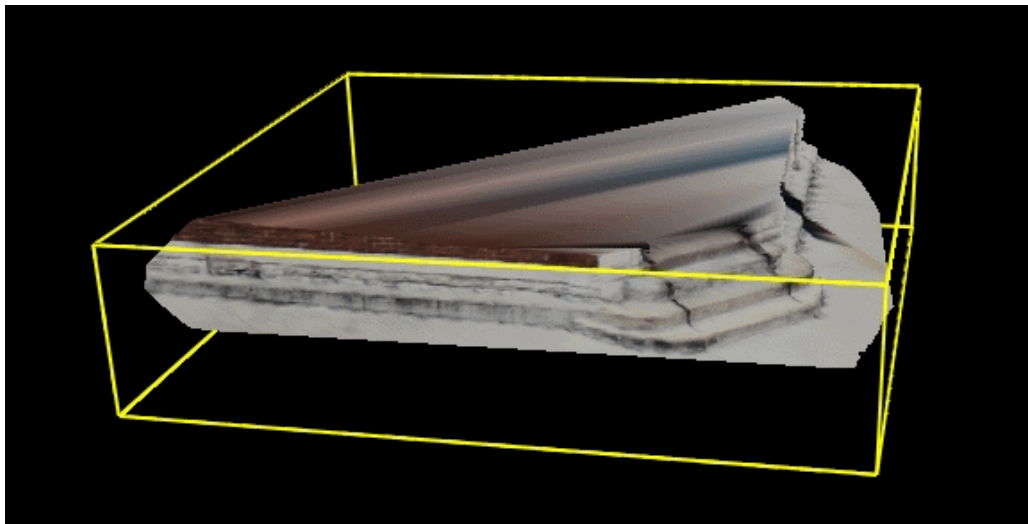
Orientation Axes

The Axes options specify the size of the orientation axis as well as its relative position in the viewer. The axes show the orientation of the three axes with respect to the current camera's view direction. Its size is in pixels, so the axes will not scale with the window size.



Bounding Box

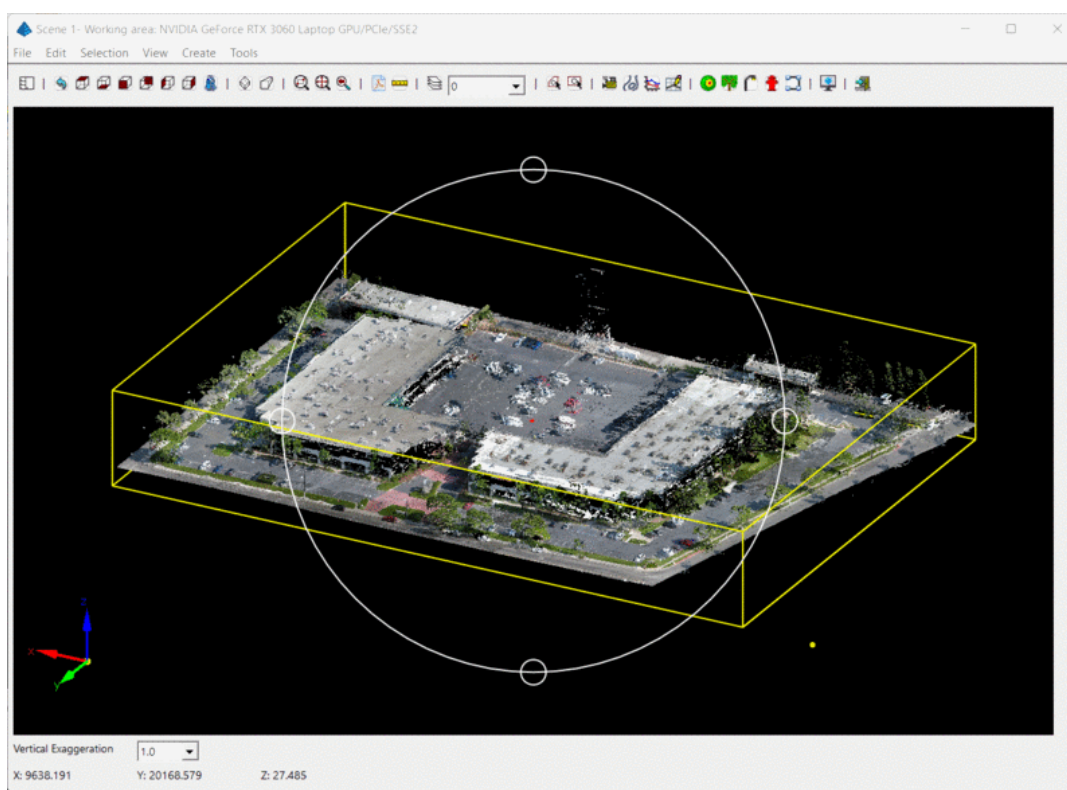
The bounding box options specify the size of the lines (in pixels) that outline the extents of the current object in the viewer as well as their colors.



TIN with its Bounding Box

Orbit

The Orbit options are for drawing a graphical orbit on screen, which is useful for restricting camera movement along a specific axis. When working with the orbit control, dragging with the mouse starting in the small circles on the left and right locks the camera to the x axis, while starting in the circle on the top and bottom locks it to the y axis. Starting a mouse drag outside the big circle locks the camera to the z axis.



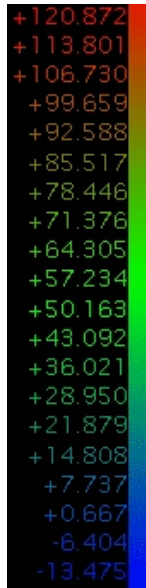
Scene with the Orbit Control Enabled

Pivot Point

The Pivot Point and Reference Point settings specify the size and color of the points drawn to represent the current pivot and the current reference point, respectively. The pivot is the current pivot for the camera, which the camera rotates around in Orbit mode and the reference point is the current point of reference which is used in the data page for relative distance checks and some other functions.

Elevation / Range Legend

The Elevation / Range Legend options specify whether to draw the legends for these values (if the current dataset being viewed consists of Elevation or Range data). These legends show the elevation / range values associated with colors values for objects colored by elevation / range. Enable Selection Tooltips are helpful tips that appear when selecting data.



CTRL Key Settings

Ability to choose to use the CTRL key or not with mouse clicks. Three options: Use CTRL Key to Pick, Use CTRL Key to Navigate or Do not use CTRL Key (default). Sound Alert for Failed Snap will warn of missed snaps (default is on)

Default Cloud Point Size

Ability to set the default size of each point cloud node. Value from 1-100. Default Polyline Width allows you to set the polyline width.

Active Polyline Highlight

Ability to display active polyline as either Dashed, Thickened or None (normal). You can also set the Elevation Label and Index Label colors.

Cloud Display Limits

Ability to set the Maximum Cloud Points Per Scene and Maximum Display Density which may improve the software performance for large datasets.

Dialog Settings

Allows you to save and reload settings.



History

Allows you to track and save a history of commands and data with timestamps. You can also auto save at your

desired time interval.

The image shows three stacked settings panels. The top panel, titled 'Command History', contains two checkboxes: 'Log Command History to File' and 'Show Timestamp in Command History Panel'. The middle panel, titled 'Data Objects', contains two checkboxes: 'Show Cloud Timestamps' and 'Sort Clouds by Time'. The bottom panel, titled 'Autosave', features a text input field with the value '15' and the label 'minutes'.

Cad Settings

Allows you to draw all points, polylines, and text to the CAD automatically. Note that if you are planning on creating a large amount of data toggling on this feature will slow you down. If that is the case, use the draw polylines and draw using field-to-finish options from the project tree for faster results. Toggling on Draw Polylines in 2D will force the polylines to zero elevation.

Toggling on Use CAD's CRD for Point Cloud's Coordinate Points will link the CAD project coordinate file (CRD) with the point cloud project. This is practical when you have existing points in the CAD file as it will prevent the need to renumber points and accidentally overwriting point data.

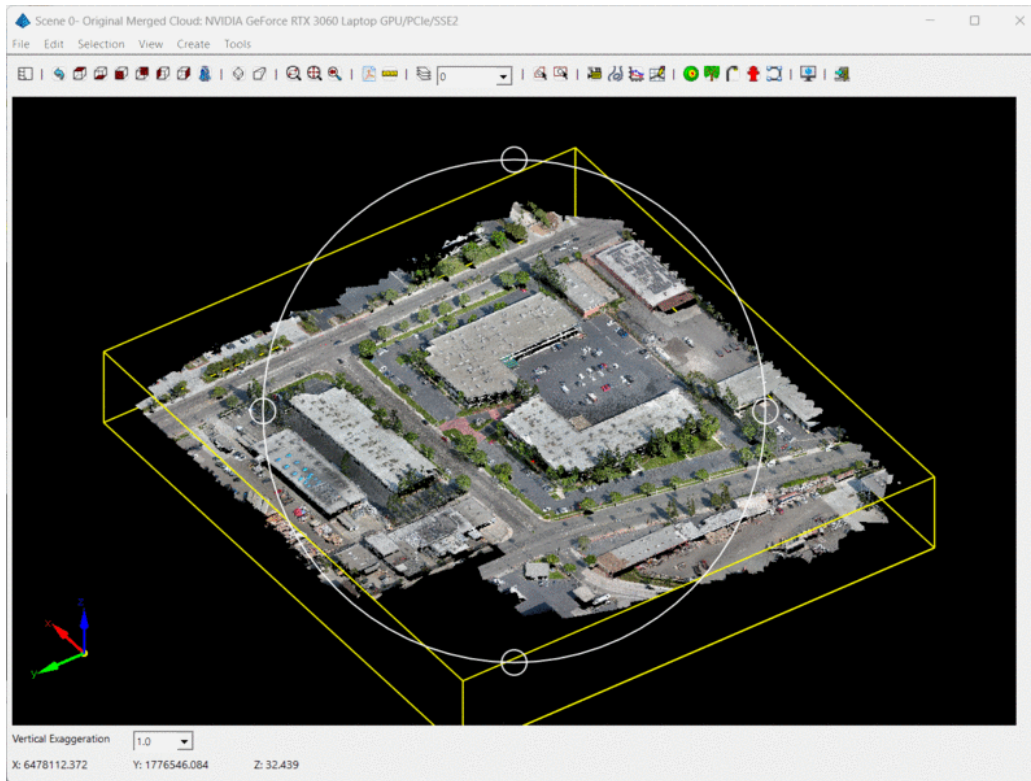
The image shows two stacked settings panels. The top panel, titled 'Auto Draw', contains two checkboxes: 'Automatically Draw All Points, Polylines, and Text to CAD' and 'Draw Polylines in 2D'. The bottom panel, titled 'CAD CRD', contains one checkbox: 'Use CAD's CRD for Point Cloud's Coordinate Points'.

Tab Location(s): Project Tab

Prerequisite: Open Point Cloud project

PointCloud Viewer

The Carlson Point Cloud viewer is where all virtual surveying and visual inspection of data is done. It has both 2D view mode functionality for viewing images and scans (as a flat image) as well as 3D view mode for viewing three dimensional data. Drawing in the viewer is divided into two different modes, dynamic and static. Dynamic mode draws a greatly reduced version of the current scene to improve draw speeds, while static mode will draw the full detailed scene. Typically, the viewer will run in dynamic mode when the user is attempting to do something interactive that involves camera, such as rotating the scene or zooming in, and it will draw in static mode when the camera isn't being manipulated. Options that pertain to how much detail is drawn in the current scene in dynamic and static mode can be changed in the Scene page of the Project Manager.



By default, the left mouse button operates in an orbit mode similar to the 3D viewer in the Carlson Civil Suite, the middle mouse button pans, and the right mouse button zooms. Mouse behavior for the current viewer can be changed in the **Camera Tab** of the Project Manager.

The toolbar and the menu at the top of the viewer have several shortcuts to functions commonly used in the Camera page. These icons and their functions are as follows (from left to right):



Tile Windows will align the Project Manager window with the Viewer window next to each other on screen.



View Reset will reset the view to its original extents and camera position.



View Top places the camera directly above and set to the extents of scene's bounding box.



View Bottom places the camera directly below and set to the extents of scene's bounding box.



View Front places the camera directly in front (negative Y axis) and set to the extents of scene's bounding box.



View Back places the camera directly in the back (positive Y axis) and set to the extents of scene's bounding box.



View Left places the camera directly to the left (negative X axis) and set to the extents of scene's bounding box.



View Right places the camera directly to the right (positive X axis) and set to the extents of scene's bounding box.



Scanner position moves the camera to the position of a scan in the scene, if there is more than one scan this button will cycle through each of the scanner positions. If the project has no scanned positions, the scene will not change.

Orthographic Projection sets the camera to parallel, orthographic, projection mode.

Perspective projection sets the camera to perspective projection mode. When in this mode, you can set the points to be displayed using *Perspective Point Sizing* from the Scene tab > Cloud. This will make the points closer to the camera look larger than those farther away.

Zoom To Window allows the user to select an area to zoom into. First click will place the first vertex of the zoom window and the second click will complete it and zoom the viewer to fit the window.

Zoom extents moves the camera to show the extents of the full scene, if in parallel projection mode, changes the width of the projection to fit the scene.

Zoom Previous resets the Point Cloud viewer to the previous view limits and orientation.

Save PDF will save the current view to a PDF format with borders and user specified title block information.

Distance Click this icon to check distances between points. Distances will be reported in Command history window. Click this icon a second time to end the distance function.

Layer Manager button opens the **Layer Properties Manager** dialog box.

Layer Selector Allows the user to select which layer to display.

Select Window Inside instigates the *Selection Set by Window* action using inside cloud points method. You will be prompted to pick the first and second point for the window and the selected cloud points will be highlighted when completed.

Select Perimeter Inside instigates the *Selection Set by Perimeter* action using inside cloud points method.

Extract Bare Earth opens the *Extract Bare Earth* window. You can choose from three different methods: *By Expansion, By Grid, or By Profile*. For more information open the **Bare Earth Extraction** help page.

Extract Contours opens the *Extract Contours* window. For more information open the **Extract Contours** help page.

Extract Profile starts the profile extraction tool. For more information open the **Extract Profile** help page.

Extract Sections starts the cross section extraction tool. For more information open the **Section Extraction** help page.



Create Point starts the create point tool. For more information open the **Point Creation** help page.



Extract Tree starts the extract tree feature tool - under the *Point Creation* actions. For more information open the **Point Creation** help page.



Extract Pole starts the extract pole feature tool - under the *Point Creation* actions. For more information open the **Point Creation** help page.



Extract Hydrant starts the extract hydrant feature tool - under the *Point Creation* actions. For more information open the **Point Creation** help page.



Create Polyline starts the create polyline tool. For more information open the **Polyline Creation** help page.



Set CAD view to current view will set your CAD drawing view to match the current scene view.



End Action will end or exit any active tools. Note that some modes can also be ended via right-click and/or the Escape key.

File Menu

- **Save PDF** Exports the current onscreen view to a PDF.
- **Exit** Closes the current viewer window only. To close the point cloud project and all open windows use the Exit button on the *Project tab* of the *Point Cloud Project Manager* window.

Edit Menu

- **Show All** shows any points which have been hidden via the Hide command.
- **Hide** hides selected points.
- **Delete Selected** deletes selected points from a Cloud or TIN.
- **Crop to Selection** crops the cloud the cloud to the selected points.
- **Transform Wizard** will open the *Transform Cloud* where transformations including translations and rotation can be applied.
- **Scale** grows or shrinks selected items.

Selection Menu

All of the same functions that can be found on the **Action Tab** along with the ability to show more information about the current selection by clicking on **Information**.

View Menu

Shows all of the available options to change the current view - similar to the options in the *Toolbar* along with the option to open the **Layer Properties Manager**.

Create Menu

In here the user has all of the functions that can be found in the Shortcuts Toolbar along with options that are found in the Create Panel within the **Action Tab**.

Tools Menu

- **Distance** Click on this to check distances between points. Slope, Horizontal and Vertical Distances will be reported on screen and in the Command history window. Click this icon a second time Right click to end the distance function.
- **Cloud**
 - **Clean** Will open the **Clean Cloud** dialog.
 - **Resample** Will open the **Cloud Resample** options.
 - **Bare Earth** Changes the mode to **Bare Earth Extraction**. Hold the Ctrl key while clicking on the scanner position to start extraction. Bare Earth works only in Plan View (viewing from Top).
- **TIN** - See **TINs** for more information
 - **Clean** To remove spikes from a TIN or noise points from a cloud
 - **Simplify** Reduces the size of the TIN surface so that it can be transferred over to CAD software
 - **Smooth** Allows the user to create a new TIN or modify the existing TIN
 - **Image** Ability to import an existing Geo-referenced image (bmp, jpg, tiff, tiff) onto a TIN

Orbit

The Orbit guide displays a primary circle with circles at the cardinal directions. These circles help guide the user when performing three dimensional rotations of the scene.

The Bounding Box

The scene bounding box is an axis-aligned box that encompasses the full range of the data currently in the scene. The bounding box is used indirectly in several functions that affect the entire scene.

The Pivot Point

The current camera pivot is represented by a point in the scene, which is red by default. When the camera is working in Orbit mode, this point is the point that the camera orbits around when you drag the mouse. By default, this point moves to the valid data nearest to your cursor when you press the left mouse button in Orbit mode, but this behavior can be changed in the **Camera Tab**.

The Reference Point

The current reference point is represented by a point in the scene, which is yellow by default. It is used for preliminary data checking without having to create any objects in the project itself. All functionality that uses the reference point can be found on the **Data Tab**.

The Elevation and Range Legend

The elevation and range legends display the range of elevation/range colors and the values associated with those colors in the current scene. The values associated with each color are not absolute and can change from scene to scene. Elevation coloring ranges from Blue, as the minimum elevation, to Red, as the maximum elevation, while range coloring ranges from white being the closest point to black for the furthest. The ranges for range coloring are calculated from the distance between the origin to the current point in clouds and TINs, while for scans the scanner position is used.

The Orientation Axes

The orientation axes are the red, blue, and green arrows that are located in the bottom left of the viewer by default. These show how the current camera's view direction relates to the world axes and can be useful when determining what direction the camera is looking in. The red arrow is the x axis, the green arrow is the y axis, and the blue arrow is the z axis. The size and position of the orientation axes can be changed in the **Project Settings** dialog.

Vertical Exaggeration

You can select different vertical exaggeration values from the pull-down menu to affect the display of vertical distances in the scene viewer.

The Status Bar

At the bottom of the viewer is a status bar that is divided up into four blocks. The leftmost three of these blocks display the x, y, and z position information of the data under the cursor (if it is valid data), and the fourth block displays any prompts to the user that may be important.

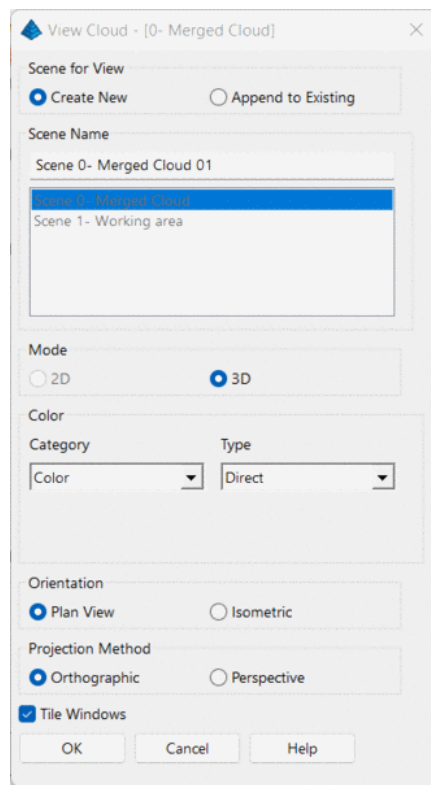
Customizing the Viewer

There exists several options for customizing the Point Cloud viewer; these options can be found in the **Project Settings** dialog, which can be accessed from the **Project Tab** of the **Project Manager** at the top of the window. See documentation on the Project Settings for more information about these options.

Scenes

Scenes are 2D or 3D displays of various data types in the viewer. Multiple objects can be viewed in a single scene or you can set each item is an independent scene. To create a scene:

- **Right-click** the object you want to view in the new scene.
- **Select View** and the View window will open.



note: If a scene already exists for the selected object, you will be prompted to view the existing scene or create a new one. The program will prompt to close the currently open scene before opening the new scene. These options can be changed in Project Settings.

The Scene for View panel determines whether to create a new scene or to append the selected object to an existing scene.

Create New: with this option selected, you will need to enter the **Scene Name**. The program will suggest scene name using the conventions found in Project Settings.

Append to Existing: with this option selected, you will append the selected object to an existing scene. Select the target scene from the list.

The **Mode** panel determines the dimensions to view the data in, **2D** or **3D**. Viewing the data in **2D** mode will only be available if the dataset can be laid out on a two dimensional plane (scans, profiles, and images). Additionally, selecting **2D** mode will not add a new scene to the project, it will only create a temporary scene to view the data in, because there is no reason to view multiple objects in the same 2D view (the objects would all obscure each other).

The **Color** panel determines the mode to color the data in. There are eight modes that are contextually available depending on the type of data and selected mode to be drawn in. These color modes are divided into categories based on how the colors are determined, these categories are: **Simple**, **Position**, **Intensity**, and **Color**. Selecting **Simple** will color the entire dataset in the same color, **Position** colors each point based off its world positioning data, **Intensity** colors each point based off of its single intensity value, **Direct** colors the points based off their color information.

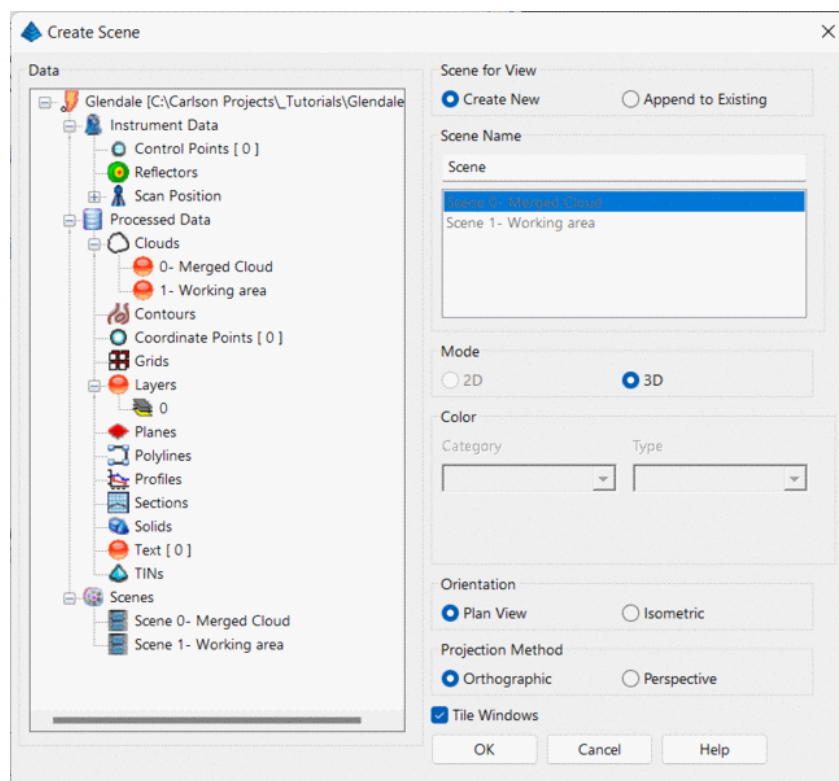
The **Simple** category only has the **Direct** color mode which colors the entire dataset a single color. Once the scene has been created, you can choose the color from the Scene tab of the Project Manager, under the Color Mode options.

The **Position** color category has three color modes: **Elevation** which colors each point in a range from blue being the minimum elevation value to red being the maximum color value, **Range** which colors each point based off its distance from the origin (or scanner if the current object being viewed is a scan), black being the minimum distance and white being the maximum, and **Normal** which colors each point based of the normal of the surface at that point. In the **Normal** mode, each color dimension (red, green, and blue) are assigned a spatial dimension (x, y, and z) to represent that direction in the objects finally color. Note that some of these options are dependent on the data set containing pertinent information.

The **Intensity** color category has three color modes: **Direct** will color each pixel in the range from black to white based off its intensity value, **Scaled** will scale the range of intensities so that the minimum intensity in the data set matches up with zero and the largest intensity in the data set will match up with the maximum possible intensity value, which can make minor changes in intensity more noticeable, and **Equalized** will perform a similar scaling operation but do it in a manner that will equalize the histogram of the intensity values, making it so that all intensity values occur equally often, this method can also make minor changes in the intensity much more noticeable.

The **Color** category has two color modes: **Direct**, which colors each point directly by its internal color value, and **Grayscale** which colors each point by the grayscale representation of its internal color value.

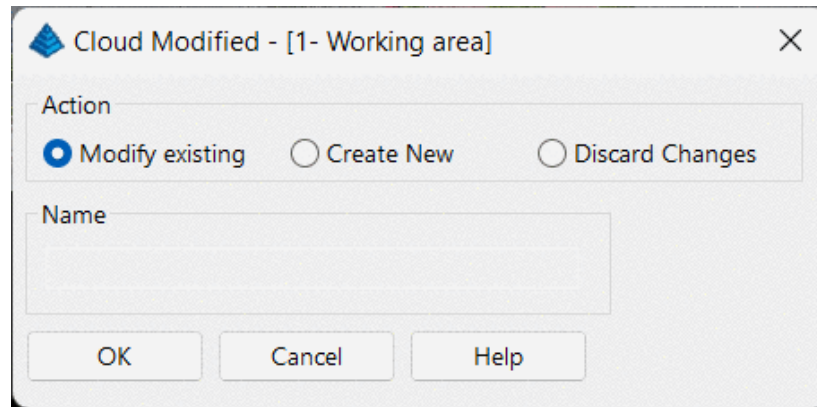
In addition to the above method of scene creation, you may also create a scene by right clicking the scenes folder and selecting **Create New** from the menu. This method will allow you to create a scene with multiple objects in it without having to go through the dialog several times.



The right half of the dialog is the same as single object viewing, but the left half of the dialog shows a tree structure representing objects in the project that can be added to the scene. Click on the red/green bubble icon next to an object to toggle on/off the inclusion of that object in the scene.

When viewing any of the large object types (scans, TINs, and clouds) there is only one copy of that object retained

in memory at all times. This is done to preserve memory as much as possible when opening up multiple views of the same object. This also means that any changes made to the object in one view will be reflected in all other views of that object. For instance, if you're viewing a scan in two views and wish to clean up its data, deleting points from the scan in one view will change the scan in any other views of it. When all views of an object have been closed, the object modified dialog will be displayed prompting the user for an action to perform with the modified data.



This dialog is very similar to the action panel found in many other dialogs. **Modify existing** will save the changes to the original object and **Create new** will allow the user to specify a name for a new object of that type with the modified data. **Discard Changes** will close the scene without saving changes.

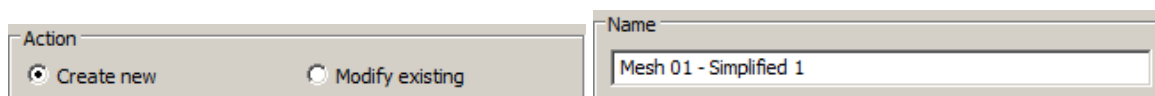
Data Objects

There are several data types in Carlson Point Cloud including instrument and processed data. These items are organized into a tree structure in the project tree and can be manipulated by right clicking to bring up a menu specific to that item type. Depending on the complexity of the selected object, a menu may not pop up or a menu with several options may pop up.

There are several functions that are shared among these object types. For brevity, these functions are listed here.

Object Action Panel

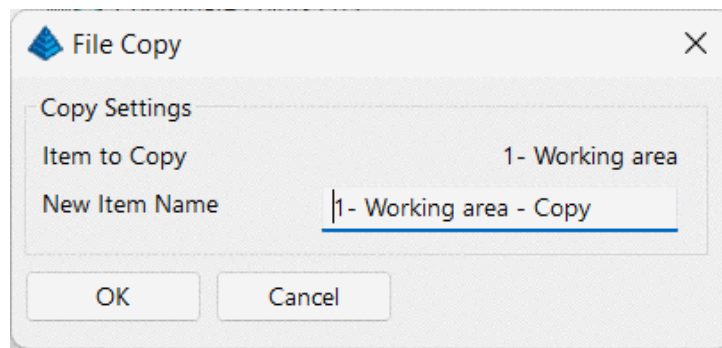
In functions that can manipulate the dataset, for example clean, resample, and bare earth point cloud filters, the topmost set of options in the dialog will allow you select whether you want to create a new point cloud or modify the point cloud or specific object as well as chose a name for the new data set when create new is selected.



Modify Existing will modify the current underlying object in the project, while the **Create New** will add a new object to the project in the same folder as the object being manipulated, with the **Name** option specifying the name of the new object.

Copying an Object

If an object has external data, such as a cloud, scan, or TIN, it may be useful to preserve the original data in case changes are made to it that are unrecoverable. Selecting **Copy** will bring up the copy object dialog.



The new item will be added to the same folder as the source item, with the name chosen.

Removing an Object

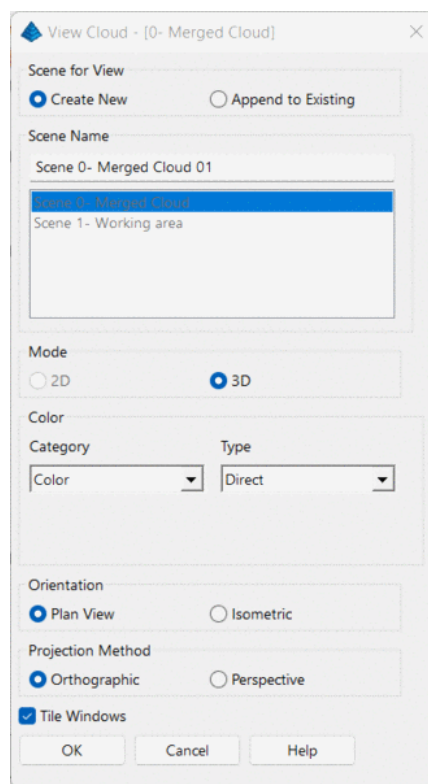
If an object can be removed from the project, there will be a **Remove** option in the right-click menu for that object. If the object has external files (scans, TINs, and clouds) you will be prompted if other files associated with this item are to be deleted or not.

Renaming an Object

To rename an object, right-click it and select **Rename**. If you try to assign that object a name that already exists elsewhere in the folder that object is in, Point Cloud will revert the change back to its old name.

Viewing an Object

If an item has data that can be viewed (in either two dimensions or three dimensions), there will be a **View** option in the right-click menu for that object. Upon clicking **View**, the **View Object** dialog will be displayed.



With **Create new** selected, you can input the name for the new scene in the **Scene Name** field. After clicking **OK**, the program determines whether a scene is already associated to that object. If none is found, the new scene window will open. If a scene is already associated to the object, you will be prompted whether to open the existing scene or

create a new one - if you do not want to be prompted this can be changed under Settings. If **Append to existing** is selected, the list of scenes in the project will become enabled allowing you to select the existing scene unto which the object will be added.

The **Mode** panel determines if the data will be viewed in 2D or 3D dimensions - defaults is 3D. Viewing the data in **2D** mode will only be available if the dataset can be laid out on a two dimensional plane (scans, profiles, and images). Additionally, selecting **2D** mode will not add a new scene to the project, it will only create a temporary scene to view the data in, as there is no reason to view multiple objects in the same 2D view (the objects would all obscure each other).

The **Color** panel determines the mode to color the data in. There are eight modes that are contextually available depending on the type of data and selected mode to be drawn in. These color modes are divided into categories based on how the colors are determined, these categories are: **Simple**, **Position**, **Intensity**, and **Color**. Selecting **Simple** will color the entire dataset in the same color, **Position** colors each point based off its world positioning data, **Intensity** colors each point based off of its single intensity value, **Direct** colors the points based off their color information.

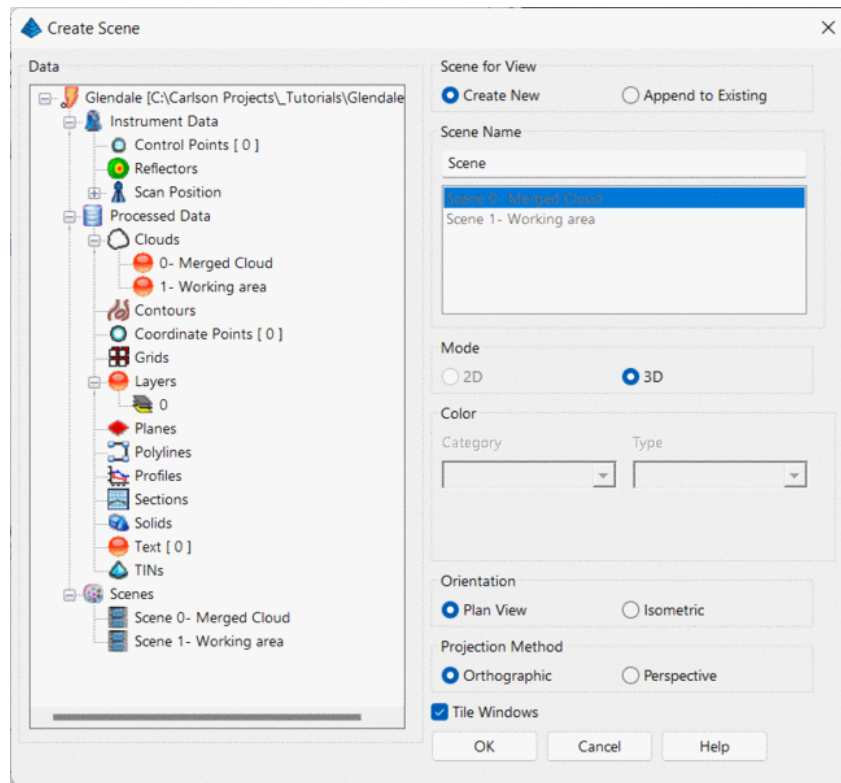
The **Simple** category only has the **Direct** color mode which colors the entire dataset a single color. Once the scene has been created, you can choose the color from the Scene tab of the Project Manager, under the Color Mode options.

The **Position** color category has three color modes: **Elevation** which colors each point in a range from blue being the minimum elevation value to red being the maximum color value, **Range** which colors each point based off its distance from the origin (or scanner if the current object being viewed is a scan), black being the minimum distance and white being the maximum, and **Normal** which colors each point based of the normal of the surface at that point. In the **Normal** mode, each color dimension (red, green, and blue) are assigned a spatial dimension (x, y, and z) to represent that direction in the objects finally color. Note that some of these options are dependent on the data set containing pertinent information.

The **Intensity** color category has three color modes: **Direct** will color each pixel in the range from black to white based off its intensity value, **Scaled** will scale the range of intensities so that the minimum intensity in the data set matches up with zero and the largest intensity in the data set will match up with the maximum possible intensity value, which can make minor changes in intensity more noticeable, and **Equalized** will perform a similar scaling operation but do it in a manner that will equalize the histogram of the intensity values, making it so that all intensity values occur equally often, this method can also make minor changes in the intensity much more noticeable.

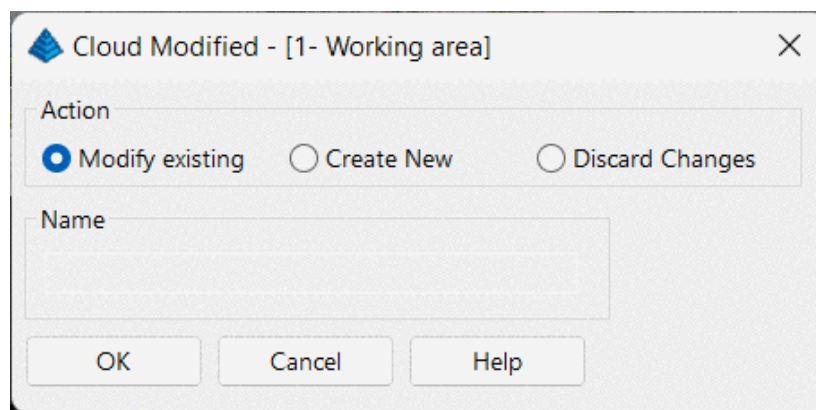
The **Color** category has two color modes: **Direct**, which colors each point directly by its internal color value, and **Grayscale** which colors each point by the grayscale representation of its internal color value.

In addition to the above method of scene creation, you may also create a scene by right clicking the scenes folder and selecting **Create New** from the menu. This method will allow you to create a scene with multiple objects in it without having to go through the dialog several times.



The right half of the dialog is the same as single object viewing, but the left half of the dialog shows a tree structure representing objects in the project that can be added to the scene. Click on the red/green bubble icon next to an object to toggle on/off the inclusion of that object in the scene.

When viewing any of the large object types (scans, TINs, and clouds) there is only one copy of that object retained in memory at all times. This is done to preserve memory as much as possible when opening up multiple views of the same object. This also means that any changes made to the object in one view will be reflected in all other views of that object. For instance, if you're viewing a scan in two views and wish to clean up its data, deleting points from the scan in one view will change the scan in any other views of it. When all views of an object have been closed, the object modified dialog will be displayed prompting the user for an action to perform with the modified data.



This dialog is very similar to the action panel found in many other dialogs. **Modify existing** will save the changes to the original object and **Create new** will allow the user to specify a name for a new object of that type with the modified data. **Discard Changes** will close the scene without saving changes.

Drawing an Object to CAD

If an object can be drawn to the CAD, there will be a **Draw** option in the right-click menu for that object. Clicking **Draw** will bring up options specific to the selected object. For example, if points are selected, you will have an

option to use Field-to-Finish or Draw points; if Draw is selected, options for point range, to add to the Carlson project CRD file and to renumber will be displayed. If polylines are selected, a layer selection dialog will display allowing you to select the layers that contain the polylines to be drawn in CAD. Objects will be drawn in their current coordinate system (if two scans are properly registered, they will be properly aligned with each other when drawn to the CAD engine).

Points

There are three kinds of point item types in Carlson Point Cloud: control points, target points, and coordinate points.

Target points always exist as a sub-item of a scan position and are in that scan position's local coordinate system. This means that a project can have several sets of target points (one for each scan position).

Control points are the global positions of the target points, and are a sub-item of the **Instrument Data** folder. There can be only one set of control points for a project.

Coordinate points are points extracted from the final registered data sets and are a sub-item of the **Processed Data** folder. Coordinate point positions are in the same coordinate system of the data from which the extraction is performed on.

All point types have the same right-click menu options.

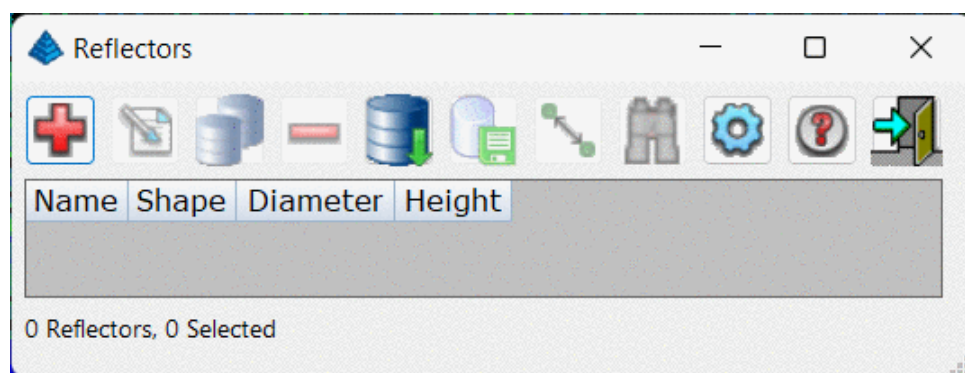
Polylines

Polylines are usually extracted from the registered data, but they can also be imported or manually entered with the **Import** and **Edit** commands. Polyline importation is very similar to importing a set of points; see the documentation for data points for more information. The edit brings up the Point Editor; see the documentation on the point editor for more information.

Reflectors

Reflectors in a Point Cloud project represent a disk, sphere, or cylinder reflector within the set of scans for the current project. Each reflector can be linked to a control point which will enable Point Cloud to account for the shape of target points that are to be linked with that control point during the scan registration process.

These reflectors can either be imported from an ASCII text file with the **Import** command or entered in manually with the **Edit** command. Pressing **Edit** will bring up the point editor dialog with some specialized options for reflectors.



The only difference between this point editor and the one used for the various point objects is that the **Add Reflector** button will bring up a different dialog that allows you to specify the properties of the new reflector.

Name will specify a unique name to be used for the reflector in the registration process.

Shape is a dropdown with three options relating to the three reflector shape types (disk, cylinder, and sphere).

Diameter allows you to specify the diameter of the sphere, cylinder, or disk objects.

Height is accessible only for the cylinder reflector type and specifies the height of the cylinder.

After all the reflectors have been added to your project, linking them to their corresponding control points is done through the control points editor. See the documentation for the Point Editor for more information.

Images

Images are stored in a folder as a sub-item of a scan position. There can be multiple images per scan-folder, and typically are pictures taken from the position of the scanner. Images can only be viewed in 2D mode.

Coordinate System

The Coordinate System is the current transformation applied to a Scan Position. This transformation is applied to any scans that belong to the current scan position. Typically, manual modification of the coordinate system should not be necessary, as the proper coordinate system to align scans properly should be determined during the registration process. To modify the coordinate system for a scan position, right-click it and select edit. This will bring up the Coordinate System editor.

The transformation panel displays the transformation matrix that is applied to the scan position. The two buttons along the bottom are **Import**, and **Export**. See the documentation on the Point Editor for information about these functions.

Contours

A contour item consists of several contour polylines that have been extracted from a TIN. Contour objects can only be viewed in 3D mode and drawn to the CAD engine. Contour extraction is done through the Action Tab, for more information on how to extract contours, see the documentation for Extract Contours in the Action Tab.

Grids

Grids are rectangular representations of a cloud typically used to instead of TIN. The grid file that is created by Point Cloud has a PCG extension. Point Cloud grids are saved to the Grids folder under Processed Data in your project folder. To share grids created in Point Cloud with other Carlson modules right-click the grid you want and select export. A GRD file will be exported. Grids created in other Carlson modules can be imported into Point Cloud by right-clicking Grids and selecting **Import**. For more information on how to create a grid, see the documentation for Create Grid in the Action Tab.

Profiles

A profile consists of a set of height values and distances taken along a polyline extracted from a cloud. Profiles can be viewed in 2D and 3D mode and drawn to the CAD engine. Additionally, profiles can be extracted to the .pro file format. Profile extraction is done through the Action Tab of the Project Manager. For more information on how to extract profiles, see the documentation for Extract Profile in the Action Tab.

Sections

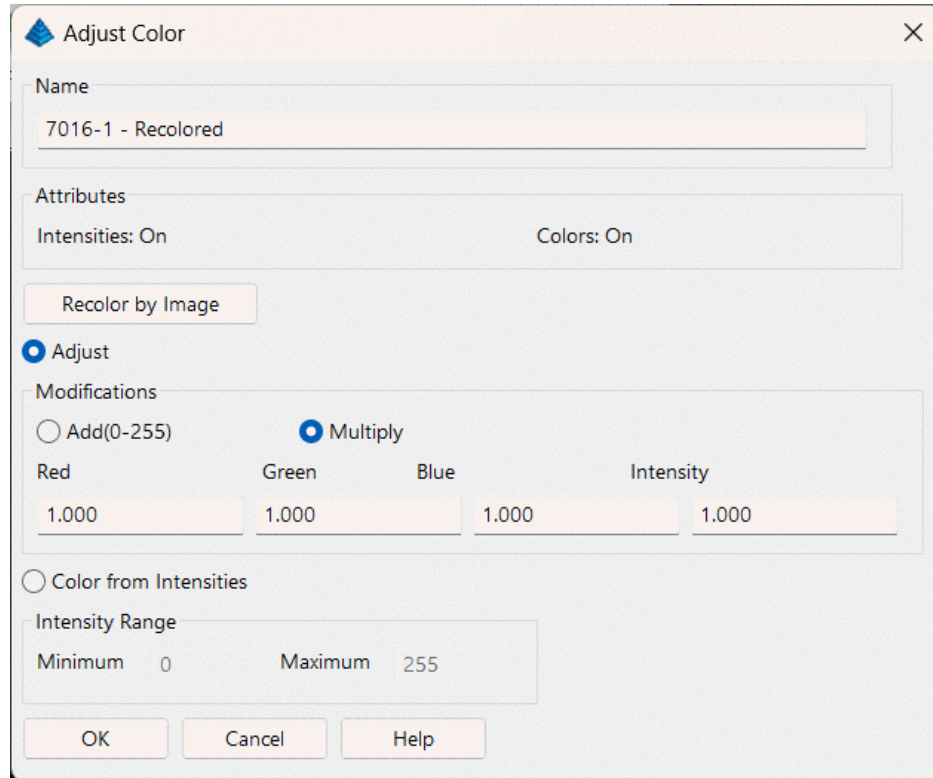
A section object consists of a list of profiles taken at certain distances along a polyline. Sections can be extracted from a cloud using the Action Tab in the Project Manager. Sections can only be viewed in 3D mode and can also be drawn to the CAD engine. Additionally, sections can be extracted to the .sct file format. See the Extract Section documentation in the Action Tab for more information about how to extract sections.

Scenes

Scene objects maintain a list of objects that are in the scene, graphics properties for those objects, and camera information for the scene. If a scene is closed and then reopened, these properties should still be the same.

Adjust Color

To adjust the color, **right-click** on a Cloud in the project tree and select **Adjust Color**. This will open the **Adjust Color** window.



Enter a name for the new Cloud.

By clicking on **Recolor by Image**, the software will ask the user to select a Geo-referenced image to use to color the pointcloud. If the image does not have an associated georeferenced world file, the software will use a best guess method to drape the image over the points. **Note:** doing this will amend existing RGB values or assign an RGB value to each point.

Under the **Modifications** panel, the user can select **Add** or **Multiply**. By selecting **Add**, Red, Green, Blue and/or the Intensity values can be altered by adding the entered value to each entity. By choosing the **Multiply** option, the existing values will be magnified by the values entered.

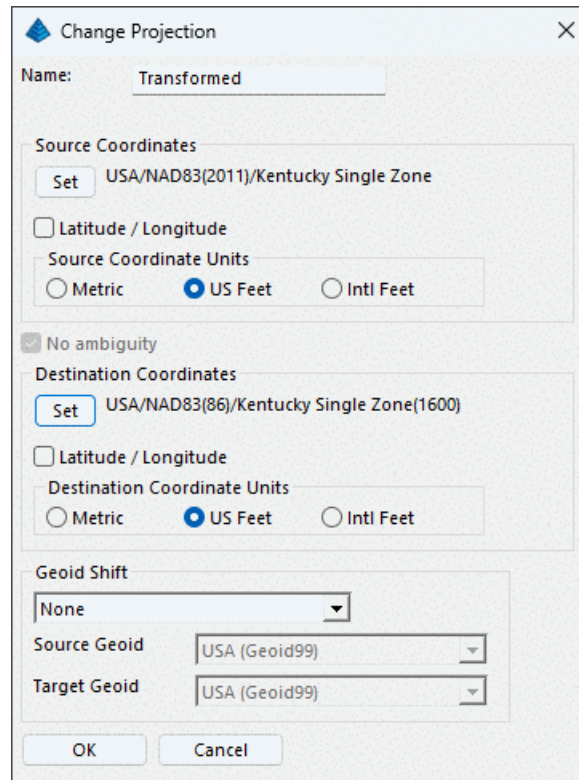
By selecting **Color from Intensities**, you can adjust the point cloud color by a **Minimum** and **Maximum** intensity range.

Click the **OK**. The new cloud will be added under the Clouds section in the project tree.

If you have an open **Scene**, the program will prompt if you want to view the newly created cloud and then prompt if you want to close the current Scene window. You can change the prompts in the Display Settings from the main Project tab.

Change Projection

To create a new Cloud using a different Coordinate projection, **right-click** on the cloud you want to change, select **Transform** then chose **Change Projection**.



Enter a name for the new cloud.

To specify **Source Coordinates**, click the **Set** button to select the **Projection**. If the source coordinate are not loaded, you will need to add a predefined system, add a project from a file, or enter a user-defined system. Alternatively, you can select using Latitude/Longitude. Make sure to select the appropriate **Source Coordinate Units**.

Then, in the **Destination Coordinates** panel, click the **Set** button to select the destination **Projection**. Similarly, if you the destination coordinates are not loaded, you will need to add a predefined system, add a project from a file, or enter a user-defined system. Alternatively, you can select using Latitude/Longitude. Make sure to select the appropriate **Destination Coordinate Units**.

Click the **OK**. The new cloud will be added under the Clouds section in the project tree.

If you have an open **Scene**, the program will prompt if you want to view the newly created cloud and then prompt if you want to close the current Scene window. You can change the prompts in the Display Settings from the main Project tab.

Control Point Editor

Control points are the global positions of the target points, and are a sub-item of the **Instrument Data** folder. There can be only one set of control points for a project. There are two ways to manipulate Control Points: the **Right-Click menu** or the **Control Point Editor**.

Note: Several commands are available in both the **Right-Click menu** and in the **Control Points Editor**. A command in the Right-Click menu operate on all control points. The Control Points editor commands operate on only the selected Control Points. If no control points are selected the users is asked if they wish to perform the operation on all control points.

Edit
Delete All
Import
Export
Transform

View

Draw

Launches the Control Point Editor

Deletes all Control Points. The user is prompted via a dialog box to confirm the deletion.

Launches the ASCII file import dialog box.

Launches the ASCII file export dialog box.

Launches the Transform dialog box to allow the user to define a transformation sequence to apply to the control points.

Launches the Scene creation dialog box to create a scene for viewing the control points.

Draws the Control Points in CAD. The user is prompted via a dialog box for the layer to draw the Control Points on.

To open the **Control Point Editor**, right-mouse click on **Control Points** under **Instrument Data** in the **Current Project** tree and then select **Edit**.

Along the top is a toolbar with several icons:

Activate Control Point toggles the selected control point(s) to be active in the current project.

Deactivate Control Point toggles the selected control point(s) to be inactive in the current project.

New Control Point brings up a dialog that allows users to specify the properties of a new control point to add.

Edit Control Points brings up the point edit dialog which allows users to change properties of the currently selected **Control Point**.

Copy Control Points allows users to copy the currently selected Control points to another point set (such as from the control points to the target points of a scan).

Delete Control Points deletes the currently selected control points.

Import Control Points brings up the ASCII file import dialog.

Export Control Points brings up the ASCII file export dialog.

Transform Control Points allows users to define a transformation sequence to apply to the control points.

View Control Points brings up the scene creation dialog to create a scene for viewing the current control point set.

Settings allow users to configure what properties of the control points are visible in the spreadsheet control.

Help brings up help documentation.

New Control Point

Name will specify a unique name to be used for the Control Point in the Project.

X, **Y** and **Z** are the coordinates of the Control Point.

Description allows users to add information that describes the new Control Point.

Target Points allows the user to specify **Target Points** from any **Scan** to be referenced to the new **Control Point**.

Add adds a new reference **Target Point**.

Edit allows the user to edit the selected reference **Target Point**

Delete deletes the selected reference **Target Point**

Import Control points

Clicking the **Import** button opens the **Import Control Points** window. Navigate to the correct file and click **Open**.

Currently supported file import types are **TXT**, **CSV**, **CRD** and **CRDB**.

Once the file has been opened, the **Import Control Points** dialog will display.

Click on the **Preset** drop-down to select the format that matches coordinates in the control points file you will import. Once a preset is selected, you will see the column headers populate with the corresponding titles. Verify that the **Delimiter Type** is set correctly if the data fields/columns are not populated correctly. Click on the **Import** check mark on the left to select or deselect points, then click **OK** to import the selected control points.

On the **Import Control Points** window, selecting **Rename all items** will rename all imported points to the settings specified in the **Naming Conventions** settings. If the **Rename all items** toggle is not selected the names listed in the file being imported will be used. If there are no names in the file being imported and **Rename all items** is not selected, the control points will be named using the settings in **Project Settings** naming conventions. The **Duplicate Entries** panel allows users to specify the action to take when a point being imported has a name that already exists in the set of points being imported to. Click **OK** to import the points.

Exporting Points

Clicking the **Export Points** button will open the **Export Control Points** window, which allows users to export the data to several different file formats.

In the **Columns** panel, click on the check boxes to include or exclude the data elements. To change the order of the data elements in the file, click on the element and then click the up or down arrows to adjust its position.

The **File** panel determines which file format to export to and also allows you to specify an extension other than the default for a given file format.

The **Text** panel contains options when exporting to the different text file types, **ANSI Text** or **Unicode Text**. The **Delimiter Type** determines the divider between data elements in the file. The **Write header** option specifies whether to write a header line to the file detailing the data ordering of the file.

Microsoft Excel allows the user to define the name of the Excel worksheet that the points will be exported to. **Note:** only available when exporting as **Format: Microsoft Excel**.

XML allows users to define the **Root** and **Collection** values and choose to **Format data**. **Note:** only available when exporting as **Format:XML**.

GCP - when exporting as GCP, click on the **Set** button to choose the Coordinate Projection. Users can choose to select a Pre-Defined projection, add a projection from a file or create a User-Defined projection on the fly. Click **OK** to select and return to the previous window. Users can select to export the coordinates in Latitude/Longitude format and also to select the appropriate units; **Metric**, **US Feet** or **International Feet**.

Users can select a preset export format by selecting it from the **Presets** drop-down menu. To define a new preset, include or exclude the desired data elements in the **Columns** panel, organize the elements into the desired order and then click the **Green Plus** button next to the drop-down. The new preset will be added to the drop-down menu for

future use.

To delete a preset, select it from the **Presets** drop-down menu and then click the **Green Dash** button next to the **Presets** drop-down menu.

Transform Points

The **Transform** button opens the **Transform Coordinate Points** window where users can define a selection of transformation parameters to apply to the selected Coordinate Points.

Initially, the dialog will have no transformations specified. To add a transformation, press the **Add** button, which will open the **New Transformation** window. Use the Type drop-down menu to select one of the four Transformation types:

Translate allows the user to define a translation of the current data set by specifying offsets in X, Y and Z.

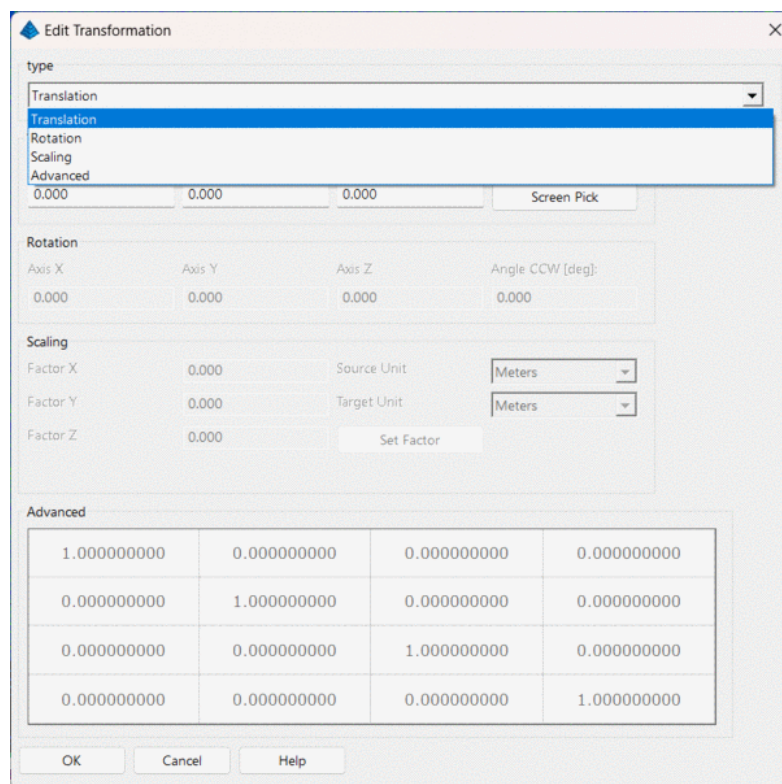
Rotation allows the user to rotate the current selection set by specifying rotations around the X Y and Z axis.

Scaling allows the user to scale the current selection by a set Factor in the X, Y and Z axis.

Advanced allows the user to specify a transformation matrix to be applied to the data.

After clicking the check button, users will be taken back to the **Transform Control Points** window. From here users can add more transformations, modify the currently selected one by pressing the **Edit** button, delete them by pressing the **Delete** button, or change their order of application to the data using the **Up/Down** buttons. Press the **OK** to apply the transformations.

Note: Transformations are applied in top-to-bottom order.



Settings

The Settings button will bring up the **Control Point Editor Settings** window, which allows the user to configure which data elements of the current point set are visible, as well as the order that they are displayed in.

To turn off a data element so it will not be displayed, click in the check box next to that element in the **Columns** panel. To change the order that data elements are displayed, select the element to move and then click the green up and down arrows to adjust the elements position.

Users can select a preset export format by selecting it from the **Presets** drop-down menu. To define a new preset, include or exclude the desired data elements in the **Columns** panel, organize the elements into the desired order and then click the **Plus** button next to the **Presets** drop-down. The new preset will be added to the drop-down menu for future use. To delete a preset, select it from the **Presets** drop-down menu and then click the **Dash** button next to the **Presets** drop-down menu.

Tab Location(s): Project Tab

Panel and Button: Current Project and Control Points

Prerequisite: None

Coordinate Points

Coordinate points are points extracted from the point cloud data sets - it could be a registered scan data set, LiDAR or the result of photogrammetry processing - and are a sub-item of the **Processed Data** folder. There can be only one set of coordinate points for a project. There are two ways to manipulate Coordinate Points:

1. The Right-Click menu
2. Coordinate Point Editor

Edit

Launches the Coordinate point Editor

Delete All

Deletes all Coordinate Points. The user is prompted via a dialog box to confirm the deletion.

Import

Launches the ASCII file import dialog box.

Export

Launches the ASCII file export dialog box.

Transform

Launches the Transform dialog box to allow the user to define a transformation sequence to apply to the Coordinate points.

Field-to-Finish

Draws all Coordinate Points in CAD. The user is prompted to select a Field-to-Finish (.FLD) file.

View

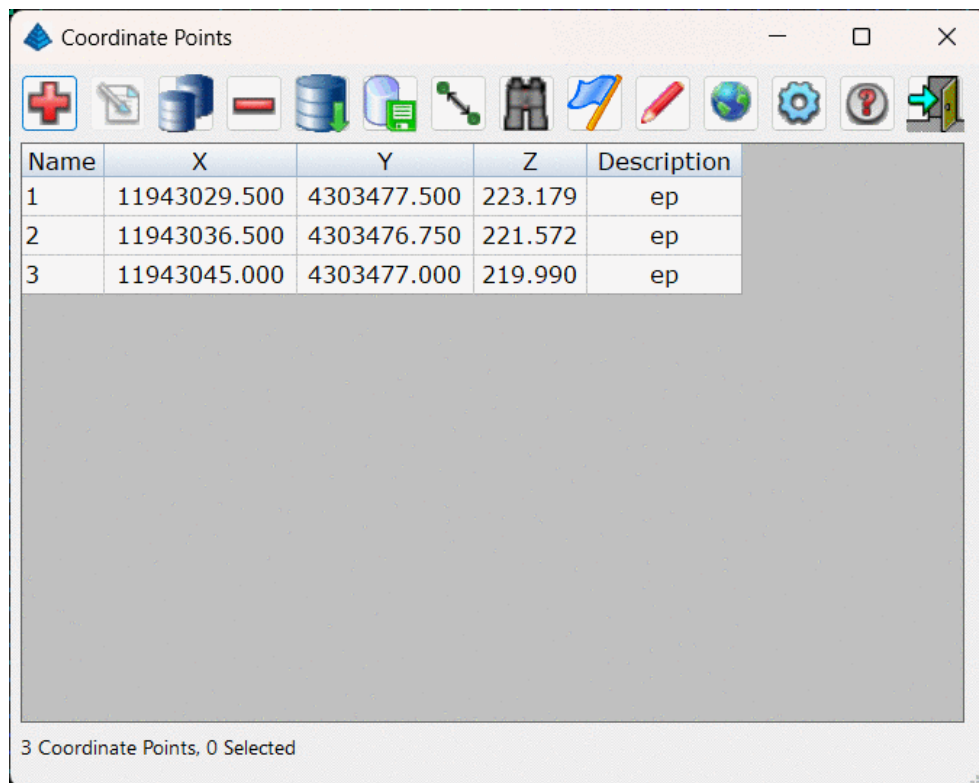
Launches the Scene creation dialog box to create a scene for viewing the Coordinate points.

Draw

Draws the Coordinate Points in CAD. The user is prompted via a dialog box for the layer to draw the Coordinate Points on.

Notes: Several commands are available in both the Right-Click menu and in the Coordinate Points editor. The difference between a command in the Right-Click menu and in the Coordinate Points editor is the Right-Click menu commands operate on all Coordinate points. The Coordinate Points editor commands operate on only the selected Coordinate Points. If no Coordinate points are selected the users is asked if they wish to perform the operation on all Coordinate points.

The Point Editor can be opened by right-mouse clicking on a set of points in the **Current Project**tree and selecting **Edit** or by double clicking on a set of points. In the **Coordinate Points** window, there are several base functions that are shared across all point types, with some extended options added for certain points types.



Along the top toolbar are several option buttons:



Add Point brings up a dialog that allows you to specify the properties of a new point to add.



Edit brings up the point edit dialog which allows you to change properties of the currently selected point.



Copy Points allows you to copy the currently selected points to another point set (such as from the control points to the target points of a scan).



Delete deletes the currently selected points.



Import Points brings up the ASCII file import dialog.



Export Points brings up the ASCII file export dialog.



Transform allows you to define a transformation sequence to apply to the points.



View brings up the scene creation dialog for viewing the current point set.



Field to Finish Draws the selected points in CAD using the current FLD file and field to finish settings.



Draw to CAD brings up the Point Draw Settings dialog specify a layer to draw the selected points on in CAD and specify if the points should be added to the current CRD file.



Coordinate System allows you to choose the coordinate system the positions values are in (Global or any current scan positions).



Settings allow you to configure what properties of the points are visible in the spreadsheet control.



Help brings up help documentation.

Adding Points

Clicking the **Add Point** button will open the **New Coordinate Point** window.

Name is a unique name for the point to be used to identify it in Point Cloud.

X, **Y**, and **Z** are the x, y, and z coordinates values for the new point to be added.

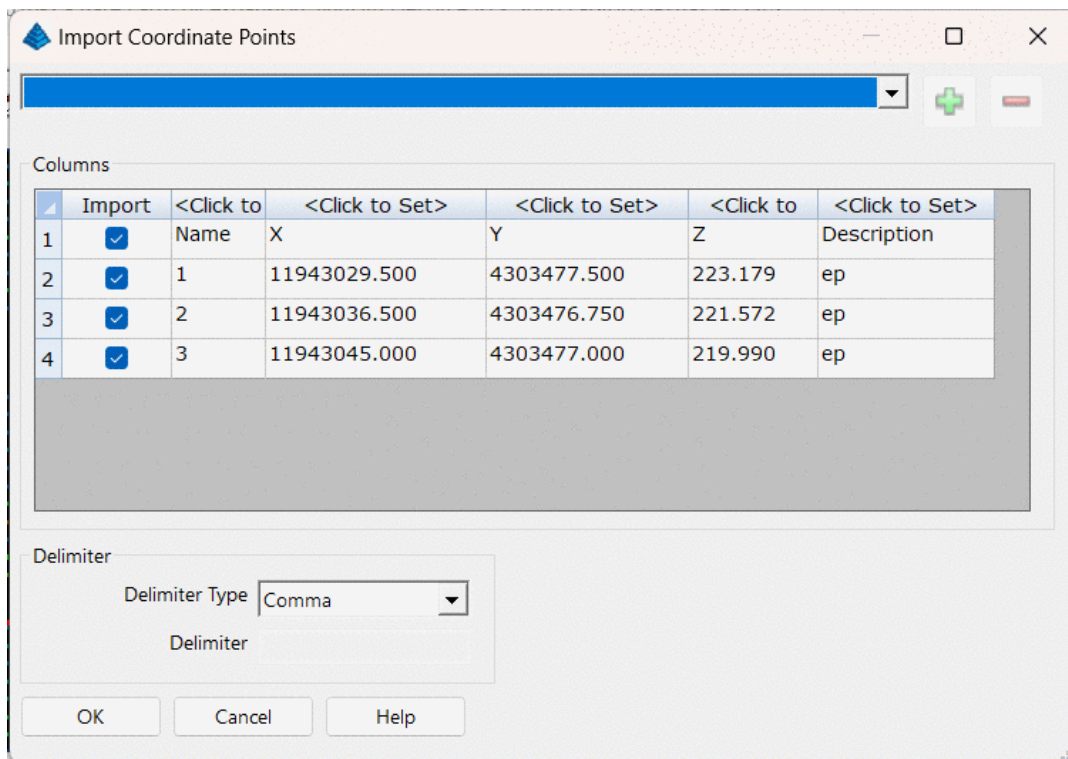
Description is a description for the point.

The image shows a dialog box titled "New Coordinate Point". It has a title bar with a blue diamond icon and a close button. The main area contains the following fields and controls:

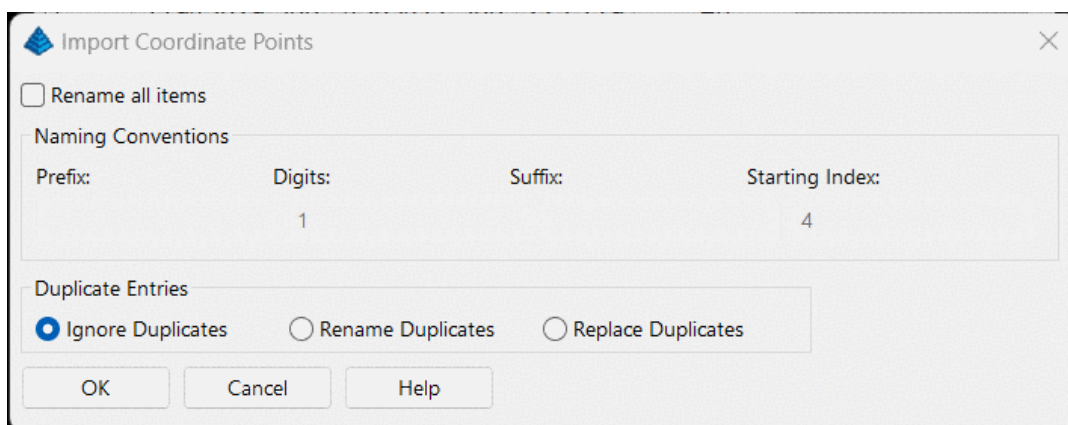
- Coordinate Point** (Section Header)
- Name:** 1001
- Description:** GCP
- X:** 0
- Y:** 0
- Z:** 0
- Screen Pick** button
- Dropdown menu: All
- OK** button
- Cancel** button
- Help** button

Importing Points

Clicking the **Import Points** button will open the **Import Coordinate Points** file selection dialog. The file formats currently supported for import are **TXT**, **CSV**, **CRD** and **CRDB**. Navigate to the correct file and click **Open** to display the **Import Coordinate Points** window.



Selecting **Rename all items** will rename all imported points to the settings specified in the **Naming Conventions** settings. The **Duplicate Entries** panel will specify the action to take when a point being imported has a name that already exists in the set of points being imported to. Click **OK** to import the points.



Exporting Points

Clicking the **Export Points** button will open the **Export Coordinate Points** window, which allows users to export the data to several different file formats.

In the **Columns** panel, click on the check boxes to include or exclude the data elements. To change the order of the data elements in the file, click on the element and then click the up or down arrows to adjust its position.

The **File** panel determines which file format to export to and also allows you to specify an extension other than the default for a given file format.

The **Text** panel contains options when exporting to the different text file types, **ANSI Text** or **Unicode Text**. The **Delimiter Type** determines the divider between data elements in the file. The **Write header** option specifies whether to write a header line to the file detailing the data ordering of the file.

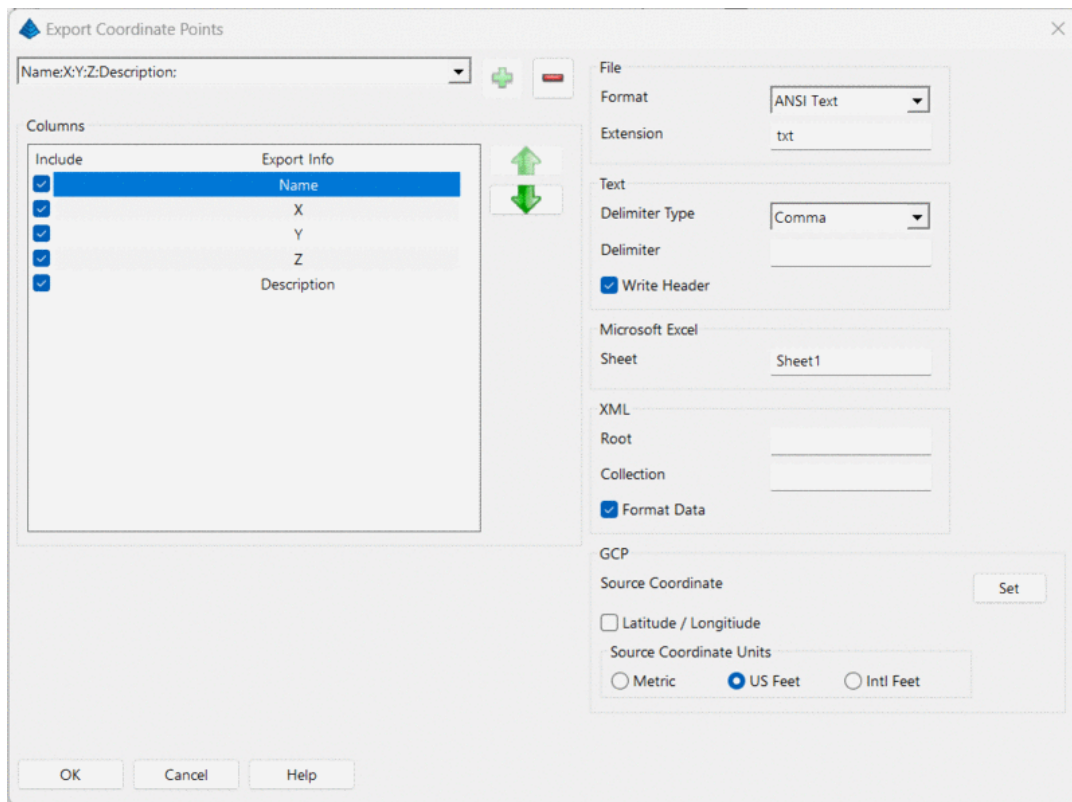
Microsoft Excel allows the user to define the name of the Excel worksheet that the points will be exported to. **Note:** only available when exporting as **Format: Microsoft Excel**.

XML allows users to define the **Root** and **Collection** values and choose to **Format data**. **Note:** only available when exporting as **Format:XML**.

GCP - when exporting as GCP, click on the **Set** button to choose the Coordinate Projection. Users can choose to select a Pre-Defined projection, add a projection from a file or create a User-Defined projection on the fly. Click **OK** to select and return to the previous window. Users can select to export the coordinates in Latitude/Longitude format and also to select the appropriate units; **Metric**, **US Feet** or **International Feet**.

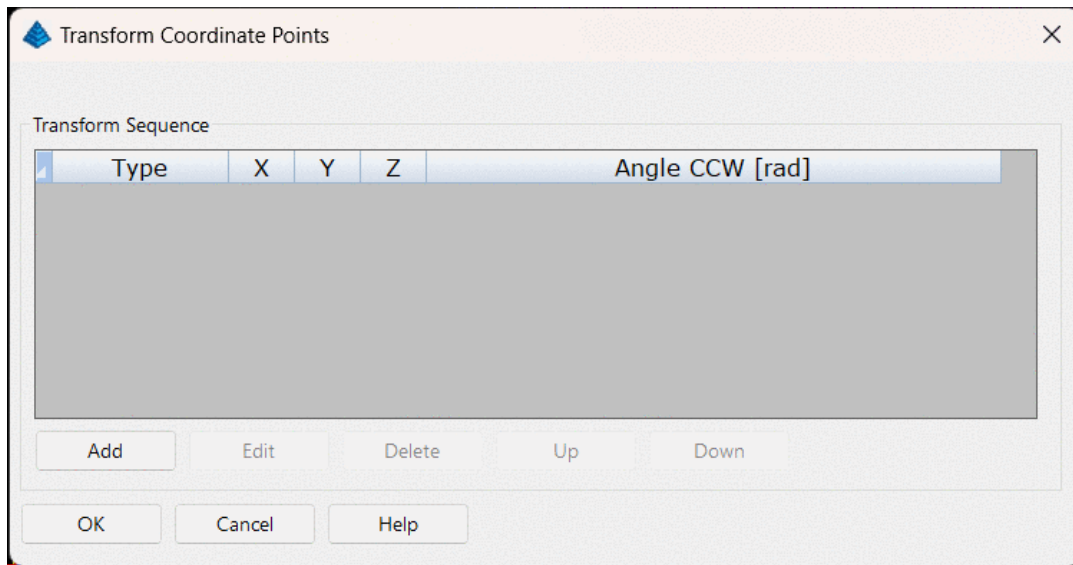
Users can select a preset export format by selecting it from the **Presets** drop-down menu. To define a new preset, include or exclude the desired data elements in the **Columns** panel, organize the elements into the desired order and then click the **Green Plus** button next to the **Presets** drop-down. The new preset will be added to the drop-down menu for future use.

To delete a preset, select it from the **Presets** drop-down menu and then click the **Green Dash** button next to the **Presets** drop-down menu.



Transform Points

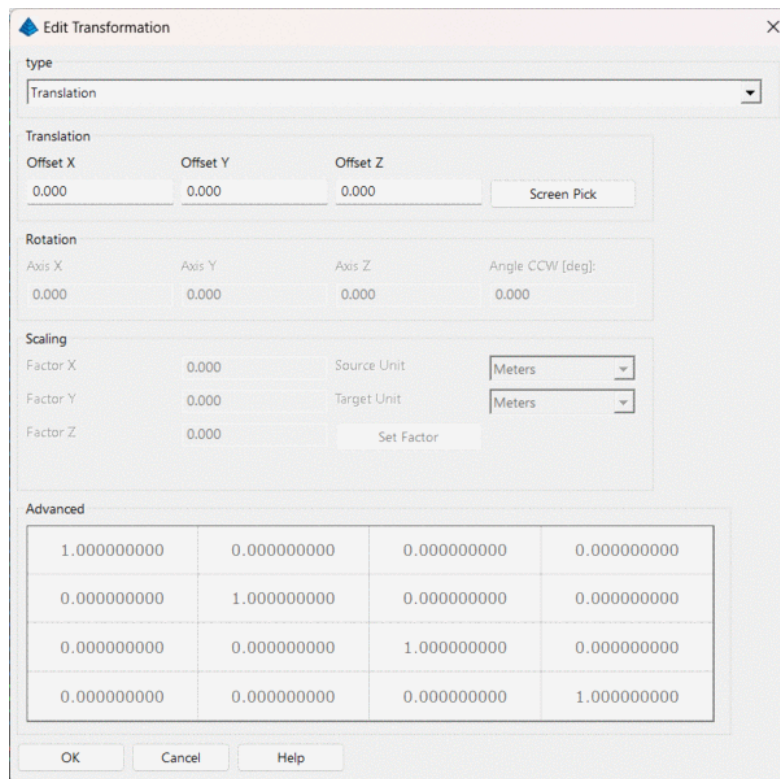
The **Transform** button opens the **Transform Coordinate Points** window where users can define a selection of transformation parameters to apply to the selected Coordinate Points.



Initially, the dialog will have no transformations specified. To add a transformation, click the **Add** button which will open the **New Transformation** window.

Use the **Type** drop-down menu to select one of the four Transformation types:

- **Translate** allows the user to define a translation of the current data set by specifying offsets in X, Y and Z.
- **Rotations** allow the user to rotate the current selection set by specifying rotations around the X Y and Z axis.
- **Scaling** allows the user to scale the current selection by a set Factor in the X, Y and Z axis.
- **Advanced** allows the user to specify a transformation matrix to be applied to the data.



After clicking the **OK** button you should be taken back to the **Transform Coordinate Points** window. From here users can add more transformations, modify the currently selected one by pressing the **Edit** button, delete them by pressing the **Delete** button, or change their order of application to the data using the **Up / Down** buttons. Press the **OK** button to apply the transformations.

Note: Transformations are applied in top-to-bottom order.

Settings

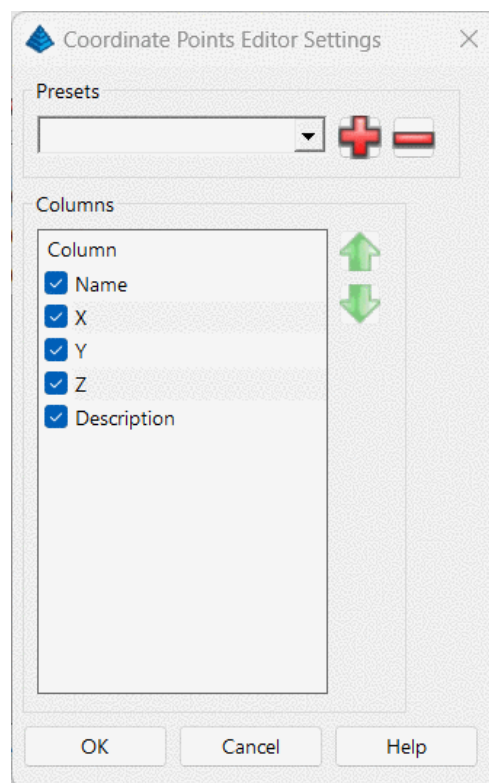
The Settings button will bring up the **Coordinate Point Settings** window, which allows the user to configure which data elements of the current point set are visible, as well as the order that they are displayed in.

To turn off a data element so it will not be displayed, click in the check box next to that element in the **Columns** panel.

To change the order that data elements are displayed, select the element to move and then click the green up and down arrows to adjust the elements position.

Users can select a preset export format by selecting it from the **Presets** drop-down menu. To define a new preset, include or exclude the desired data elements in the **Columns** panel, organise the elements into the desired order and then click the **Green Plus** button next to the **Presets** drop-down. The new preset will be added to the drop-down menu for future use.

To delete a preset, select it from the **Presets** drop-down menu and then click the **Green Dash** button next to the **Presets** drop-down menu.



Tab Location(s): Project Tab

Panel and Button: Current Project and Coordinate Points

Prerequisite: None

Target Points

Target points always exist as a sub-item of a **Scan Position** and are in that scan position's local coordinate system. This means that a project can have several sets of target points (one for each scan position). There are two ways to manipulate Target Points:

1. The Right-Click menu
2. Target Point Editor

Edit

Delete All

Import

Export

Transform

View

Draw

Launches the Target Points Editor

Deletes all Target Points. The user is prompted via a dialog box to confirm the deletion.

Launches the ASCII file import dialog box.

Launches the ASCII file export dialog box.

Launches the Transform dialog box to allow the user to define a transformation sequence to apply to the target points.

Launches the Scene creation dialog box to create a scene for viewing the target points.

Draws the Target Points in CAD. The user is prompted via a dialog box for the layer to draw the Target Points on.

Notes:

Several commands are available in both the Right-Click menu and in the Target Points editor. The difference between a command in the Right-Click menu and in the Target Points editor is the Right-Click menu commands operate on all target points. The Target Points editor commands operate on only the selected Target Points. If no target points are selected the users is asked if they wish to perform the operation on all target points.

The Target Point Editor is activated when the user *right-clicks* **Target Points** and selects **Edit**. In the **Target Point** editor there are several base functions that are shared across all point types.

The title bar will display the scan position that the target points are part of. Just below that is a toolbar with several icons. The icons and their corresponding functions are as follows:

Activate Target Point toggles the selected target point(s) to be active in the current project.

Deactivate Target Point toggles the selected target point(s) to be inactive in the current project.

Add target Point brings up a dialog that allows you to specify the properties of a new target point to add.

Edit Target Point brings up the target point edit dialog which allows you to change properties of the currently selected target point.

Copy Target Points allows you to copy the currently selected points to another point set (such as from the target points to the control points of a scan).

Delete Target Points deletes the currently selected target points.

Import Target Points brings up the ASCII file import dialog.

Export Target Points brings up the ASCII file export dialog.

Transform Target Points allows you to define a transformation sequence to apply to the target points.

View Target Points brings up the scene creation dialog for viewing the current target point set.

Settings allow you to configure what properties of the points are visible in the spreadsheet control.

Help brings up help documentation.

New Target Point

Name will specify a unique name to be used for the Target Point in the project.

X, Y and Z may be in feet (ft) or Meters (m). These are the coordinates of the Target Point.

Description allows you to optionally add information that describes the new Target Point.

Control Point allows the user to specify a **Control Point** for the new **Target Point**.

Import Target points

The settings of the import dialog must be configured to match the data ordering of the file being imported. First, ensure that the **Delimiter** is set to the correct value, if it is there should be multiple columns in the dialog, if it isn't, all of the data should only be in one column. After the delimiter is properly set, the data values for each column must be set. This can be done by clicking the header of each column (which say *Click to Set* by default) to bring up a menu with all the available values that can be set for that column. After setting these values toggle the importing of lines you wish to leave out by clicking the green buttons in the Import column to turn them red. At the top of the dialog is the presets panel, which will allow you to save the current settings if you wish to use them again later. After configuring the dialog to match your data, click the OK to import and continue to the next step. The next dialog displayed will allow you to configure the naming conventions for the target points to be imported.

Selecting **Rename all items** will rename all imported points to the settings specified in the options. The **Duplicate Entries** panel will specify the action to take when a point being imported has a name that already exists in the set of points being imported to. If the **Rename all items** toggle is not selected the names listed in the file being imported will be used. If there are no names in the file being imported and **Rename all items** is not selected, the control points will be named using the settings in **Project Settings** naming conventions.

Exporting Target Points

Clicking the export button will bring up the Export Target Points dialog, which will allow you to export the data to several different file formats.

Simply click the check boxes next to the data elements that you wish to export. To change the order of the data elements in the file, select the data element you wish to move and click the up or down arrow to move it.

The **File** panel determines which file format to export to and also allows you to specify an extension other than the default for a given file format.

The **Text** allows you to specify the **Delimiter** to be used as the divider between data elements in the file and whether to use the **Write header** to specify whether to write a header line to the file detailing the data ordering of the file.

Transform Target Points

The Transform Target points dialog box allows specifying a sequence of transformations to apply to the currently selected target points.

Initially the transformation list is empty. Press the green plus button to add a new transformation, which will bring up a new dialog.

There are three types of transformations, as well as an advanced transformation where the user can specify the transformation matrix to apply.

1. **Translation** transformations move the points linearly along the offset values specified.
2. **Rotation** transformations rotate the points around the axis specified the angle specified.
3. **Scaling** transformations scales the selected points (using the global origin as the origin for the scaling) in each dimension by the amount specified.

After the transformation's settings have been configured, pressing the green check mark will add it to the current list of transformations. In the Transformation sequence dialog the following functions apply:

Add will allow you to add transformations.

Edit will allow you to edit the currently selected transformation.

Delete will delete the currently selected transformation.

Up / Down will change the order of applications of the transformations. Transformations are applied in top-to-bottom order.

Settings

The Settings button will bring up the Point Editor settings dialog, which will allow the user to configure which data elements of the current point set are visible, as well as the order that they are displayed in.

Display of a property can be changed by toggling the check box next to that property, changing the order can be done by first selecting the property to move and then pressing the green up or down arrows to move the selected property up or down. Saving the current settings can be done clicking the plus button in the **Presets** panel.

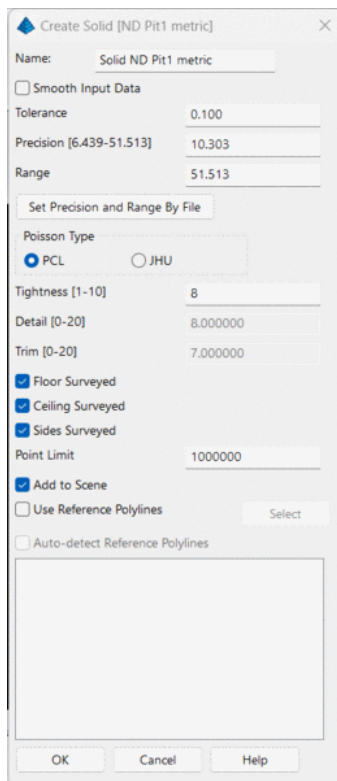
Tab Location(s): Project Tab

Panel and Button: Current Project and Target Points

Prerequisite: None

Create Solid

You can create a solid model from data objects such as point clouds and tins. The program uses Poisson surface reconstruction techniques that will create watertight surfaces from the point data. To create a Solid from a cloud, right-click the cloud you want to create a solid from the project tree and select **Create > Create Solid**. If only the floor, ceiling, or sides have surveyed, a 2D solid model can be created.



Smooth Input Data: The program applies an smoothing algorithm to remove some of the noise in the point data to facilitate flatter surfaces.

Tolerance: Helps remove outlier points. If the resulting surface is too spiky, adjust this value higher.

Precision: Uses tighter calculations for pre-processing for tighter grid.

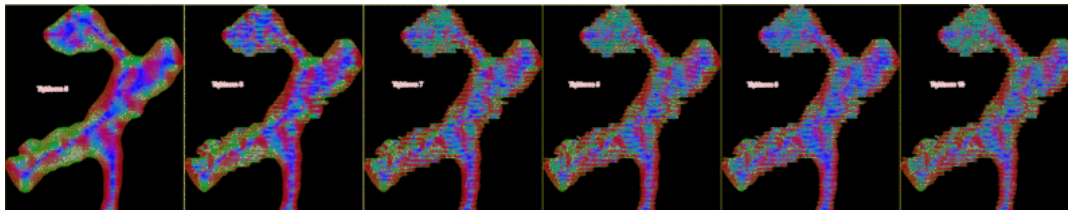
Range: If the mesh does not span over expected gap areas, raise this value higher. If the surface spans over unexpected areas, lower the value.

Set Precision and Range By File: will calculate the precision and range values from by scanning the file.

Poisson Type: **PCL** (industry standard Point Cloud Libraries) or **JHU** (Johns Hopkins University) method can be selected. Program defaults to the PCL method.

Tightness: Used for determining the density of the triangles when the PCL method is selected. The higher the value, the smaller the triangles of the resulting solid. Note that tightness will not be used if a 2D solid model triangulation is done.

To generate a watertight 3D solid model **check** the **Floor Surveyed**, **Ceiling Surveyed**, and **Sides Surveyed** boxes.



Note: High Tightness can lead to banding in certain situations. If the resulting cloud exhibits banding, consider reducing the Tightness setting.

Point Limit ensures that the resulting 2D or 3D solid model does not have more points than the value entered in this field, with a default of 1,000,000.

Add to Scene will create and open a Scene window displaying the resulting solid model.

As the the Poisson surface generation process is highly sensitive to the normals of the points in the cloud, **Use Reference Polylines** and **Auto-detect Reference Polylines** provide a method to to set the normals for the cloud based on a set of reference polylines such that all the point normals in the cloud face away from the nearest reference polyline prior to creating the solids. **Use Reference Polylines** allows you to **Select** your drawn polylines. By checking **Auto-detect Reference Polylines**, you will allow the program will automatically calculate a group of reference polylines. As an example, when creating a 3D solid model for a tunnel, it is possible that not all the point normals will be facing away from center, by drawing custom polylines or using auto-detect, the program will ensure that all the normal are aligned away from center.

Click **OK** to create the solid. Once created, a **View Solid** dialog will display. Confirm the settings and click **OK** to view the new solid in a Scene. The new Solid will be added under the **Solids** menu in the **Current Project** tree.

Cloud Objects

Clouds are the most basic of the three large dataset objects supported by Carlson Point Cloud. Point clouds consist of a large amount of points with positions that can also contain color, intensity, and classification associated with each point.

Point Cloud has several functions that can help reduce (decimate), clean, and generally prepare clouds to be manageable in a CAD application. In the **Current Project** tree structure, clouds are categorized under their own folder, **Clouds**. Clouds have a couple of unique functions that make them distinct from other data objects, in addition to the general actions common to other data elements with 3D data.

Import a Cloud
(from an external file)

Create a Cloud
(from a selection of points, etc)

Clean a Cloud
(eliminate noise points and duplicated points)

Resample a Cloud
(to reduce total point count)

Smooth a Cloud

Transform a Cloud
(rotate, translate, etc)

Scale a Cloud
(from feet to meters etc)

Change the Projection
(Lat/Long to State Plane etc)

Adjust Color
(brighten, darken, or drape image)

Bare Earth by Grid
(keep only the low points from the cloud)

Register a Cloud
(match pre-entered control points to hand-picked target points)

Merge Clouds

Subtract Clouds
(remove one cloud's area from another cloud)

Export Cloud
(to LAS, ASCII, or PDF)

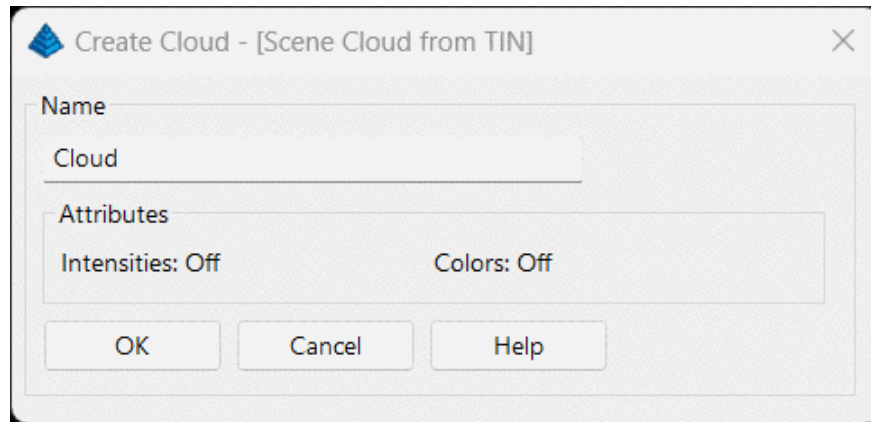
Create a Solid

Create Cloud

New clouds can be created many ways, for example, new clouds are created as the result of applying filters including the Clean and Bare Earth algorithms. Clouds can also be created from TINs and Scans by **right-clicking** on the TIN or Scan in the project tree and selecting **Create** then choosing **Cloud**. A new cloud can also be created by selecting a cloud in the project tree, the resulting cloud is copy of selected cloud.

Commonly and more practically, a new cloud can be created by selecting a section of another cloud using the **Selection Set** tools in the **Action tab** when a Scene is open. Once the data is selected in the Scene view, click **Cloud** in the **Create panel** under the **Action tab**.

In the Create Cloud window, enter the **Name** of the new cloud.



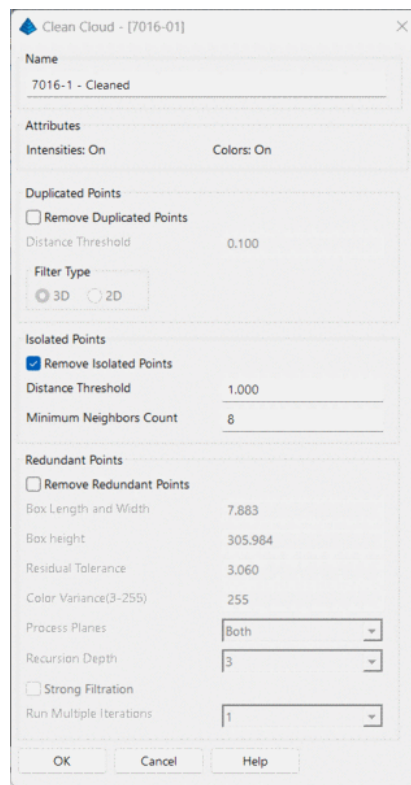
Click the **OK**. The new cloud will be added under the Clouds section in the project tree.

If you have an open **Scene**, the program will prompt if you want to view the newly created cloud and then prompt if you want to close the current Scene window. You can change the prompts in the Display Settings from the main Project tab.

Clean Cloud

Cleaning or decimating the cloud can be helpful to reduce the size of the point cloud that you are working with and make it easier to use any of the extraction tools, create TINs, calculate volumes, and extract profiles and sections. You can clean a complete cloud, a selection within the cloud, or multiple clouds at the same time.

To clean the complete cloud, **right-click** the cloud in the project tree and select **Clean**. To clean a portion of the cloud, open the cloud scene. Then, select the part of the cloud that you want to clean using any of the tools in the **Selection Set** panel of the **Action** tab; then, click **Clean** from the **Edit** panel in the **Action** tab. To clean multiple clouds, right-click **Clouds** in the project tree, then select **Clean**; the **Select Clouds to Clean** window will be displayed allowing you to select multiple clouds and then **click OK**.



The Clean function offers three different cleaning methods: Duplicate Points, Isolate Points, and Redundant Points. Note that the Duplicate Points and Isolate Points routines may be used in combination. Redundant Points must be used individually. It is also important to note that the size of the cloud or clouds being filtered and the options that are selected will affect the execution time for this routine.

Remove Duplicated Points method will search for any places where two or more points in the cloud are within the **Distance threshold** of each other. If any such points are found, they are deleted from the cloud. This method is to help remove redundant data that, for all intents and purposes, are the same given the current data set (say a data set that spans several miles having two points within inches of each other) and reduce the size of the data set. This can be carried out on a 2D or 3D basis. The 3D method runs faster. Usually clouds that are produced by stationary scanners have more points than necessary right next to the scanner, and need this kind of cleaning.

Remove Isolated Points method will search for any points in the cloud that have less than the **Minimum Neighbors Count** points within the **Distance Threshold** from them and delete them. This will remove points that are likely to be outliers, which could be a result of bad data such as the beam on a ground scanner hitting a dust particles in the air.

Redundant Points method uses a series of user parameters to clean points from the cloud. This is most useful before creating a TIN, because it will reduce flat parts down to one point. Using a box of user defined size, points are checked for fitting the plane and/or color match. If no points match the box is subdivided and the check repeated. These sub-boxes will again be subdivided if no points match. If points are found that meet the criteria a point closest to the average of those in the box is copied to the clean cloud. If no match is found, all the points from that box are copied into the clean cloud unless Strong Filtration is on.

- **Box Length and Width** Sets the horizontal distance to search for points to compare. In general a small values will produce best results.
- **Box height** Sets the vertical component completing a 3D cube search area for points to compare. In general a small values will produce best results.
- **Residual tolerance** is the measure of how far from a best fit plane points can be. A typical value would be 1/10 of the smallest box dimension.

- **Color variance** determines how much the color of a point can vary. When set to 255 color variance is not used.
- **Process Planes** allows the user the ability to choose to clean vertical or horizontal data only or to clean both.
- **Recursion depth** is the number of times the initial box is subdivided to find points that meet the parameters. Typical values used would be 3 or 4.
- **Strong Filtration** when checked on, the program will copy the point closest to the average for plane and/or color to the new cloud even if no exact plane or color match was established.

Click the **OK**. The new cloud will be added under the Clouds section in the project tree.

If you have an open **Scene**, the program will prompt if you want to view the newly created cloud and then prompt if you want to close the current Scene window. You can change the prompts in the Display Settings from the main Project tab.

Import Cloud

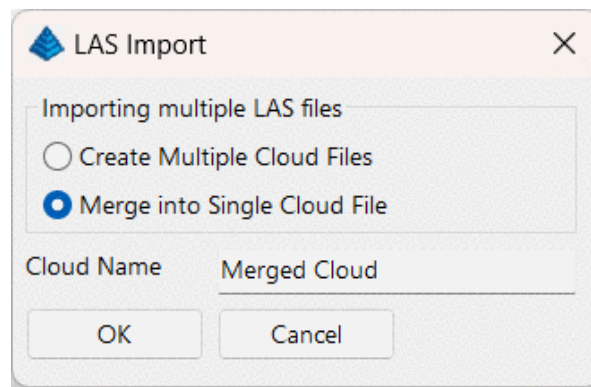
To import a **Cloud** from a file right-click the **Clouds** folder in the **Current Project** tree and select **Import**. Carlson Point Cloud supports file formats generated by multiple manufacturers. Formats include:

- LiDAR (.las/.laz)
- Generic (.txt/.asc/.xyz/.csv)
- FARO (.fls/.fws)
- Leica (.pts/.ptx)
- E57 (.e57)
- Quarryman (.fsc)
- PLY (.ply)
- DEM/ESRI (.dem/.adf/.asc/.img/.tif)
- GeoTIFF (.tif/.tiff)
- PCG (.pcg)
- RCS (.rcs)
- POD (.pod)
- 3DTK (.oct)
- Carlson Coordinates (.crd/.crdb)
- Carlson Surface (.tin/.grd)
- Point Cloud Library (.pcd)
- Points from CAD
- Carlson Point Cloud (.bxyz)

Notes: the 3DTK engine does not support .pts or .ptx files. Carlson Quarryman [.fsc] can be imported directly from a Carlson Quarryman system. Choosing **Points From CAD** allows the user to select any points in their CAD environment to import into the Point Cloud project.

After selecting a format, the **Select Clouds to Import** window will open. Browse to the folder with you point cloud files. You can select multiple files at a time for some file types. Click **Open** to import the data.

If multiple files were selected a new **Import** window will open prompting if you want to **Create Multiple Cloud Files** - each file selected will be displayed as a separate object in the project tree - or **Merge into Single Cloud File**. If merging the cloud files, you will need to enter a **Cloud Name** for the combined point cloud. Click **OK**.

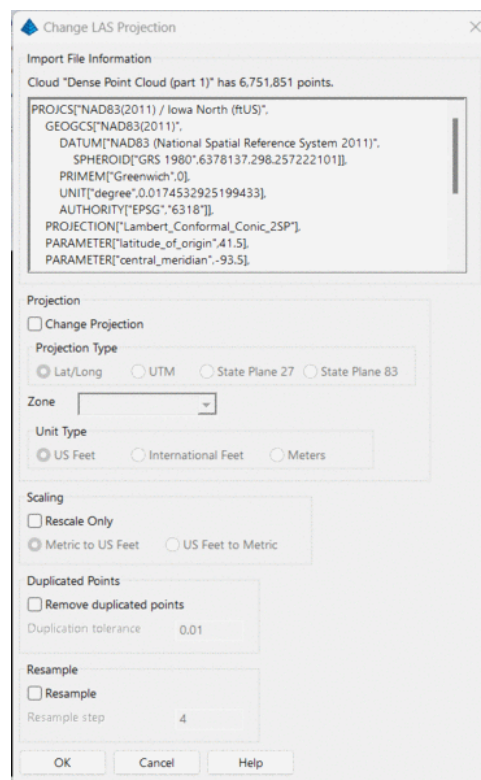


The **Change Projection** window will open. The **Import File Information** panel will display the projection information found in the file(s) that will be imported. You can use **Change Projection** to change the projection of the imported file to match your Carlson project system.

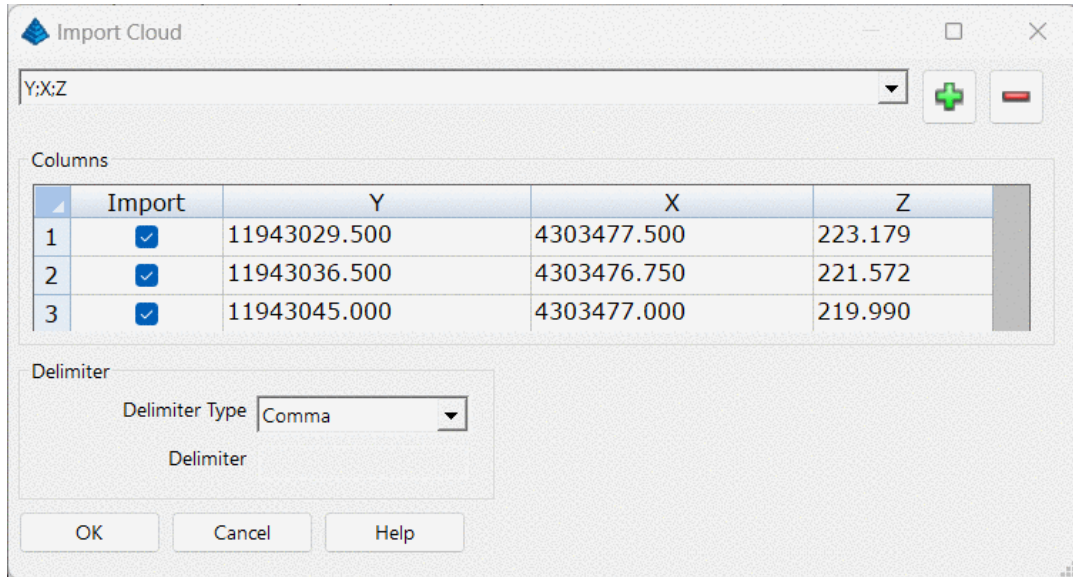
Use **Rescale Only** to convert the file from **Metric to US Feet** or **US Feet to Metric**. Note that commonly the software used to register the ground scanner clouds will export the data in metric units, therefore this tool is very handy if your project is set in feet.

Use **Remove duplicated points** to clean (decimate) the file to be imported. The program will search for any places where two or more points in the cloud are within the **Duplication tolerance** of each other. If any such points are found, they are deleted from the cloud. For example, this is recommended when the cloud was created using a ground scanner as the density of points in close proximity to the scanner will be large and unnecessary. By using this filtering tool, you can reduce the size of the cloud significantly without affecting the accuracy.

Using **Resample** will also reduce the size of the imported point cloud. The resample method though is a fast but naive method as it will reduce the cloud to 1/**Step** its current size by keeping only one out of every nth point found. Click **OK** to import the file.



Note: If a Generic format (ASCII) is selected, the **Import Cloud** window will be displayed. In order for the data to be imported correctly the user must define what **Delimiter Type** is used within the ASCII file. If the data is not shown organized in columns (all the data is in one column for example); another delimiter will need to be chosen. Clicking the header of each column will bring up a menu with all the possible values that can be assigned to it, click each column's header to set its proper value. At the top of the dialog is the **Presets Panel**, which allows users to save the current column assignments to be used in the future. The import column is used to determine whether to import that particular column (often files will have text or header information that isn't actually part of the data set). Save the current configuration by clicking the + button in the panel, presets can also be removed by clicking - button. Presets are saved in the global program settings so they can be used across multiple projects. Click the **OK** to begin the import.



After importing a cloud, you will have access to multiple methods to decimate the cloud, from the tools found in the **Clean** options to the bare earth filters to the automated feature extraction actions. r

Menu Location: Right click on Clouds entry in Project tree >Import

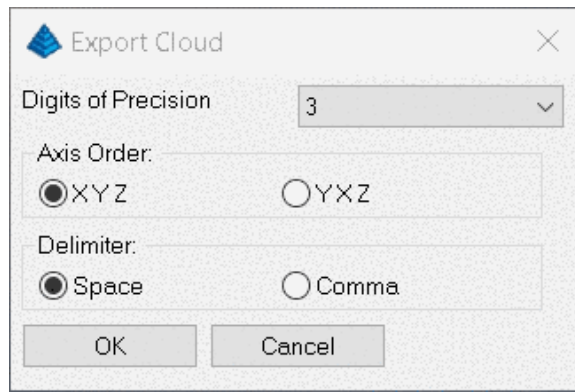
Keyboard Command: No **Prerequisite:** Supported cloud file available, a project open in the Project Tab

Export Cloud

Clouds can be exported to a variety of formats, accessed by right-clicking on the cloud to export in the project tree, then selecting **Export** and choosing the format for the exported file. Options include: **ASCII**, **LAS**, **E57**, **RC5**, **PDF**, and **PCD**.

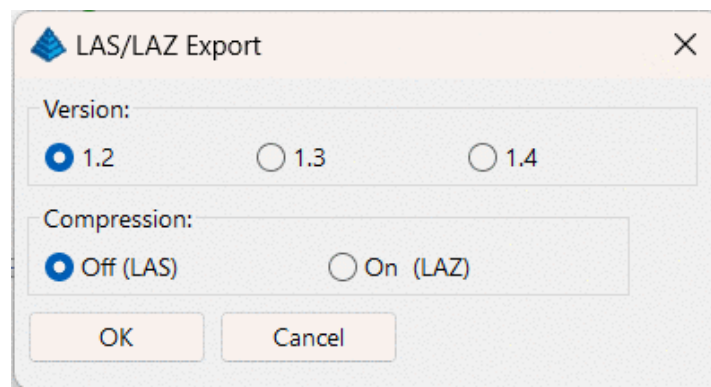
Export to ASCII - Creates a space or comma delimited text file (.txt) of the point cloud.

Select **Export to ASCII**. Enter the name for the ASCII file and click **Save**. In the **Export Cloud** window, you can set the **Digits of Precision**, the **Axis Order**, and the **Delimiter format**. Click **OK** to export.



Export to LAS - Creates an .las or .laz file.

Select **Export to LAS**. Select the version for the exported file and whether it should be uncompressed (LAS) or compressed (LAS). Click **OK** and then enter the name for the new file. Click **Save** to export the file.



Export to E57 - Creates an .e57 file. e57 files are generic and can be read by most if not all applications that support point clouds.

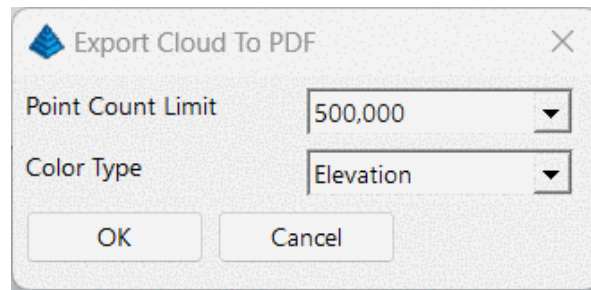
Select **Export to E57**, enter the name for the new file. Click **Save** to export the file.

Export to RCS - Creates an .rcs file. RCS the file format used by Autodesk ReCap.

Select **Export to RCS**, enter the name for the new file. Click **Save** to export the file.

Export to PDF - Creates a 3D PDF of the cloud that can be opened in Adobe Acrobat Reader.

Select **Export to PDF**, enter the name for the new file. Click **Save**. In the Export Cloud to PDF window, select the **Point Count Limit** and **Color Type**. The **Point Count Limit**, limits the number of points to be used to create the 3D PDF by resampling the selected cloud - values range from 250,000 to 2,000,000. The **Color Type** can either be Automatic (which will use the RGB values for each point in the cloud), Intensity, or by Elevation. Click **OK** to export the file.



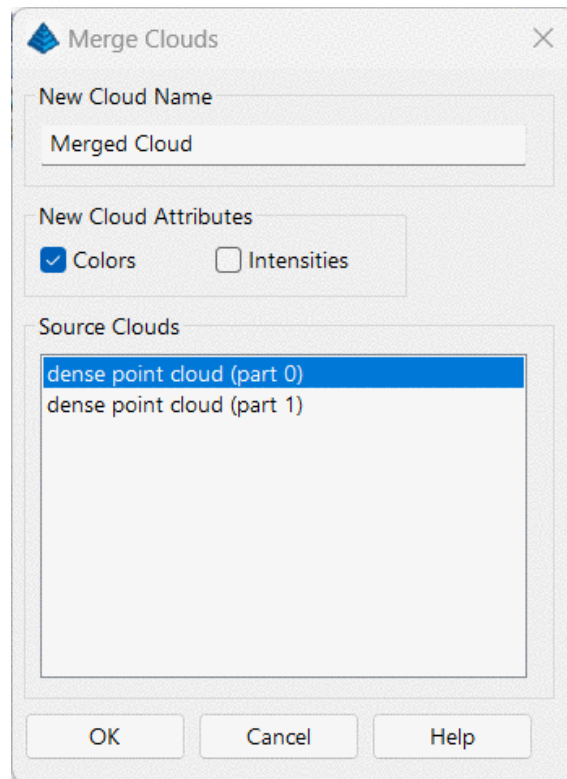
Export to PCD - Creates a .pcd file

Select **Export to PCD** to open the file selection window. Enter a name for the PCD file and click **Save** to export.

Merge Clouds

Two or more existing clouds can be merged into a single cloud. **Right-click** on one of the clouds in the **Current Project** tree and then select **Merge Clouds**.

In the **Merge Clouds** window, enter the **New Cloud Name**. Select whether to keep and merge the **Color** and **Intensity** values from the existing clouds. Under the **Source Clouds** panel, use the CTRL and SHIFT keys to select the remaining clouds to merge. Note that a maximum of 20 clouds can be merged together in a single operation. Click **OK** to begin the merge. The new cloud will be displayed in the **Clouds** sub-menu in the **Current Project** tree. You will be prompted if you would like to view the new cloud and if you want to close any opened Scene windows.



Tab Location(s): Project

Tree Folder: Clouds

Prerequisite: Existing Clouds

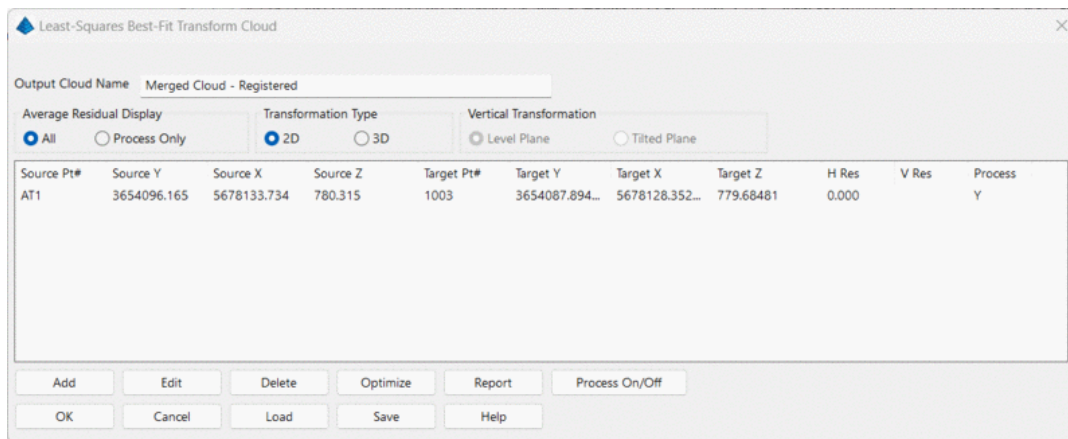
Register Cloud

Using **Align/Register** transforms coordinates between local, State Plane 27, State Plane 83, Latitude/Longitude, Universal Transverse Mercator (UTM) and many other projections, including regional and user-defined projections.

To align the cloud, **right-click on** the cloud in the **Current Project** tree and select **Align/Register**. Enter a name for the new, registered Cloud. You will be prompted to select the Alignment/Registration method.

Least-Squares Best-Fit

The **Least-Squares Best-Fit** option is used when there are more than two pairs for translation points. Since two pairs of points are sufficient to define the translation and rotation, more than two pairs of points provides more than enough information. You will need to enter the **Output Cloud Name** for the resulting cloud.



Over **Determination by Plane Similarity** is used to find the least squares best fit transformation for all the given source and destination points. Besides doing a translation and rotation, this option will also scales the points during the transformation. The **Rigid Body Transformation** also does a best fit least squares transformation, but applies only translation and rotation with no scale.

The **Helmert 7-Parameter** method can also be used for local transformations. The 7-Parameter Values can be calculated from control points or entered by the user. The program uses the Helmert Transformation, Strict formula:
 $[X_t \ Y_t \ Z_t] = [dX \ dY \ dZ] + M * R * [X_s \ Y_s \ Z_s]$
where R is the rotation matrix.

The Transformation Type chooses between doing a 2D transformation and 3D transformation. For the 3D transformation, the program transforms the x/y using the same method as the 2D transformation, and the z is transformed using an elevation difference model that is modeled by either a best-fit level plane or tilted plane as set by the **Vertical Transformation** setting.

The **Add** button is used to define the source and destination coordinates for the points that define the transformation. Pressing this button brings up the following dialog box. Clicking the **List** button will allow you to select points from the Point Cloud **Coordinate Points**, Drawing Coordinate Points (.crd/.crdb), **Control Points** (presumed to be correct in World Space), and Scan Position lists.

Source Coordinate	
Point Number	MH1
Northing	3654106.651
Easting	5678167.63
Elevation	779.756

Target Coordinate	
Point Number	1002
Northing	3654099.5742
Easting	5678179.7620
Elevation	779.49481

The **Edit** button is used to edit existing data.

The **Delete** button removes the source and destination pairing from the transformation setup.

The **Process On/Off** button allows source and destination pairings to be turned on and off. This is useful when wanting to inspect different results using different pairings.

The **Optimize** option chooses which point pairings would yield the best transformation results by turning off the processing of pairings with higher residuals. This minimizes the average residual for the control points.

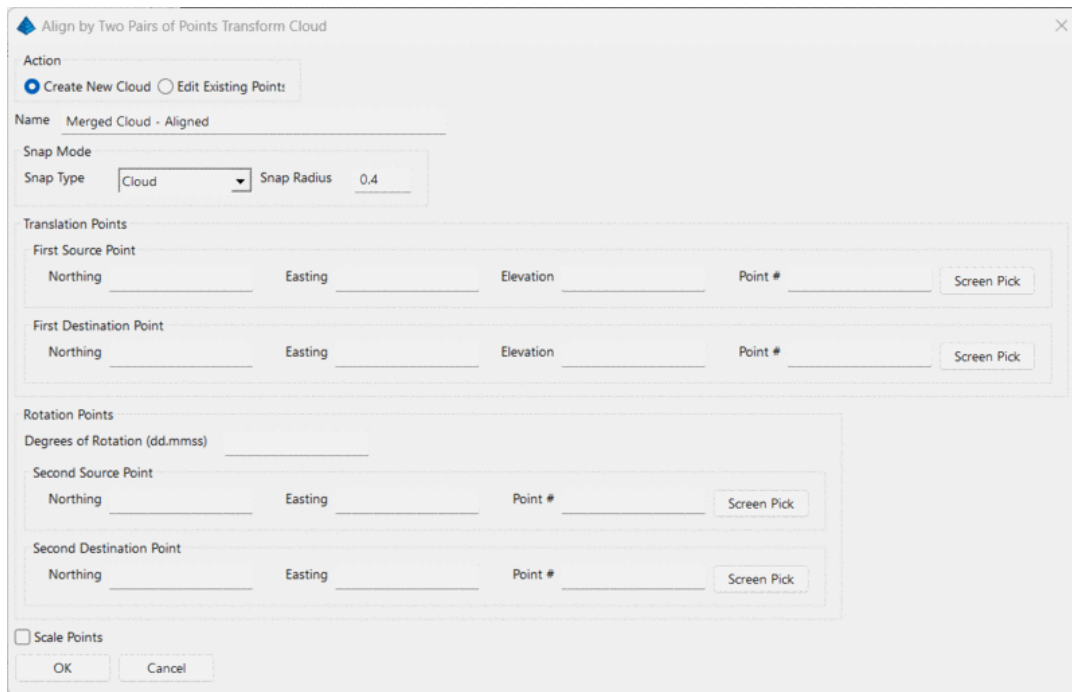
The **Report** option displays a report of the transformation point pairings, their residuals, processing status, transformation scale and avg. residual.

The **Load** and **Save** options allow for saving and recalling local coordinate transformation pairings and settings.

Align by Two Pairs of Points

The **Align by Two Pairs of Points** option uses two pairs of source and destination coordinates. The first pair defines the translation as the difference between the source and destination northing and easting. You will have the option to **Create New Cloud** or **Edit Existing Points**.

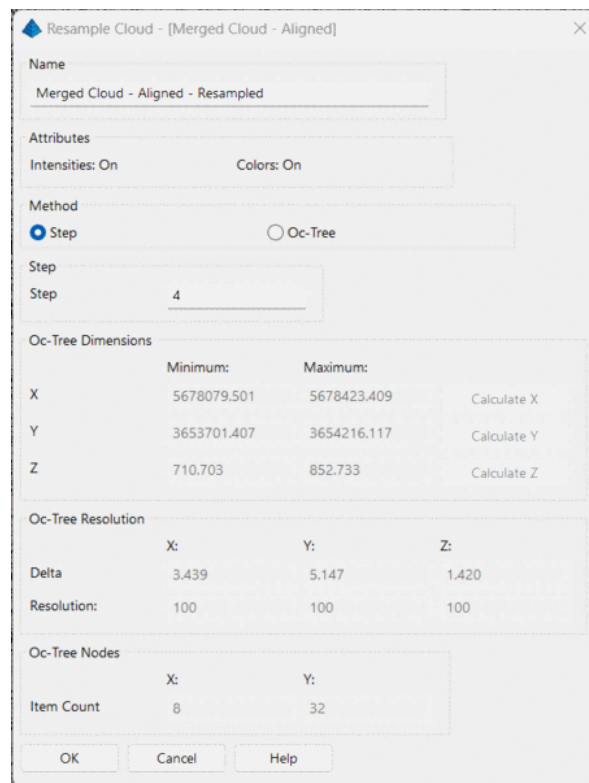
This destination point is also the pivot point for rotation. Rotation can be entered directly or defined by a second pair of points where the bearing between the first and second source points is rotated to align with the bearing from the first and second destination points. There is an option to also apply scaling. The scaling holds the angle between points and adjusts the distances by the scale factor. The scale factor is calculated for each point as the elevation factor at the first source point times the grid factor at the first destination point averaged with the elevation factor at the transform point times the grid factor at the transform point.



Resample Cloud

Point Cloud has two methods of cloud reduction **Step** and **Oc-Tree**. There is a fast, naive method, and a slower, more intelligent method.

To resample a Cloud, **right-click** the cloud in the **Current Project** tree and select **Resample**. This will open the **Resample Cloud** dialog.



Step Method

The **Step** method is much faster at the cost of being less intelligent. It will simply reduce the cloud to 1/Step its current size by only keeping one out of every nth vertices. For example, a step size of four, Resample will traverse the cloud and only keep every fourth vertex, deleting three for each one it keeps.

Oc-Tree Method

The **Oc-Tree** method is a more intelligent process, but it can have significantly longer run times than the Step method. The Oc-Tree creates a bounding box based on the **Oc-Tree Dimensions** settings. This bounding box is then subdivided into blocks (nodes) based on the **Oc-Tree Resolution** settings. The points in each block (node) in the Oc-Tree is then filtered according to the **Oc-Tree Nodes** settings.

- **Oc-Tree Dimensions:** The extents ("Minimum" <-> "Maximum") in the x,y, and z dimension of the Oc-Tree's bounding box.
- **Oc-Tree Resolution:** Defines the number of blocks (nodes) included in the Oc-Tree such that resolution*delta=extents (in x,y, or z dimension).
 - **Delta:** Extents of one of the Oc-Tree's blocks (nodes) in the x,y, or z dimension.
 - **Resolution:** Number of blocks (nodes) in the x, y, or z dimension.
- **Oc-Tree Nodes:** Specifies how the Oc-Tree filters points. If a block (node) in the Oc-Tree contains fewer than "Minimum" number points, the contents of that block (node) will be deleted. If a block (node) has more than "Maximum" number points, random points within that block (node) will be removed until it has "Maximum" number points inside it.

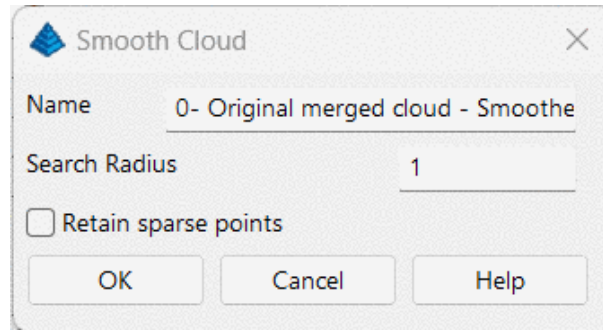
Click the **OK** to resample the Cloud. The new Cloud will be displayed in the **Clouds** sub-menu of the **Current**

Project tree.

See also: **Clean Cloud**

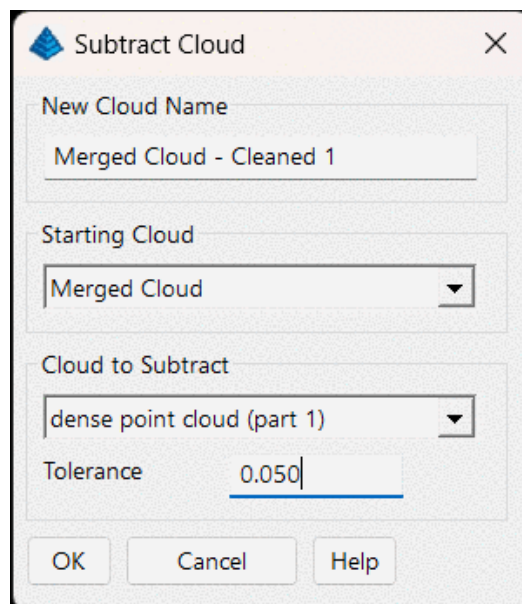
Smooth Cloud

The Smooth Cloud command moves each point to be closer in height to its neighbors within the search radius. If a point doesn't have any neighbors within the radius, it will be removed, unless "Retain sparse points" is checked.



Subtract Cloud

A cloud that is part of a combined or merged cloud, can be subtracted from another. **Right-click** on the combined or merged cloud in the **Current Project** tree and select **Subtract Cloud**. Enter the **New Cloud Name**. Use the **Starting Cloud** drop-down menu to select the original cloud. Use the **Cloud to Subtract** drop-down menu to select the cloud that is to be subtracted from the original. A **Tolerance** can be entered - recommended value of 0.05 will normally be sufficient. Click **OK**. The new cloud will be displayed under the **Clouds** sub-menu in the **Current Project** tree.

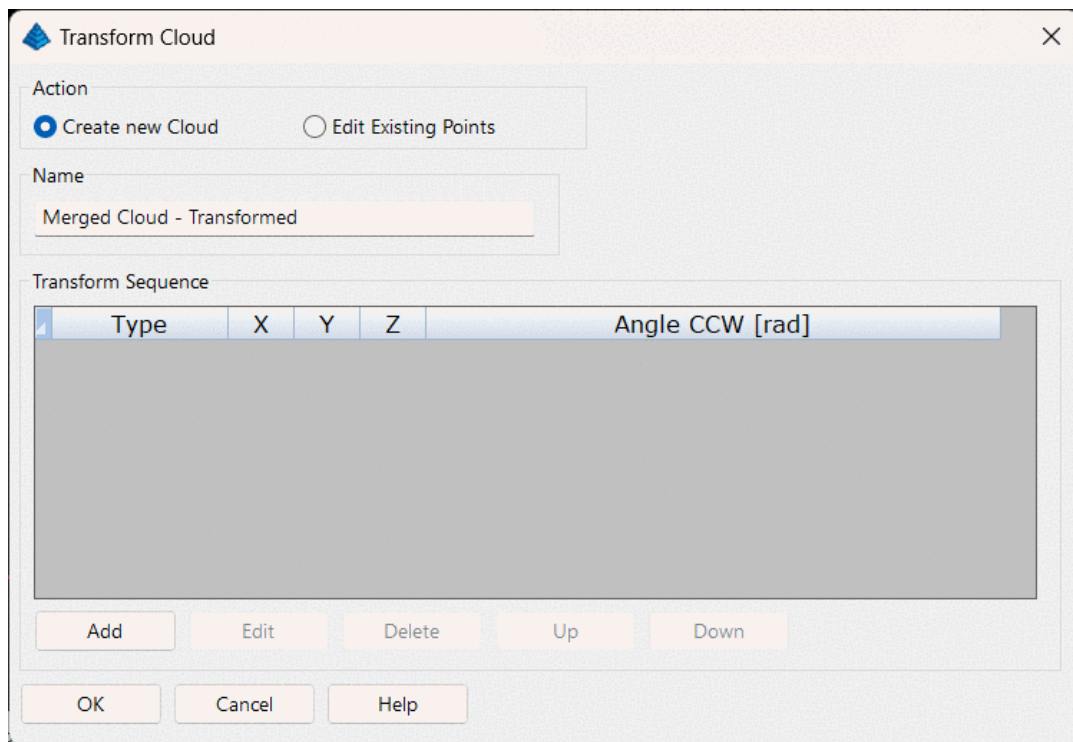


Transform Cloud

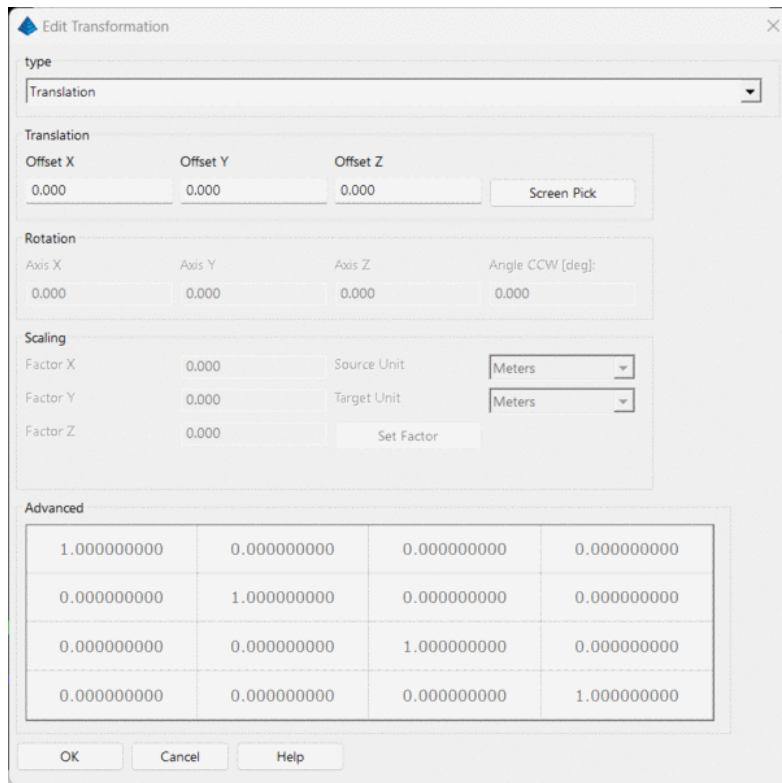
You can apply various transformations to the point cloud:

- **Translate** allows the user to define a translation of the current data set by specifying offsets in X, Y and Z.
- **Rotations** allow the user to rotate the current selection set by specifying rotations around the X Y and Z axis.
- **Scaling** allows the user to scale the current selection by a set Factor in the X, Y and Z axis.
- **Advanced** allows the user to specify a transformation matrix to be applied to the data.

Right-click on the cloud in the **Current Project** tree, select **Transform**, then chose **Transformation Wizard**. In the **Action** panel, you can choose to **Create New Cloud** or to **Edit Existing Points**. If you select **Create New Cloud**, you will need to enter the **Name** for the new cloud.



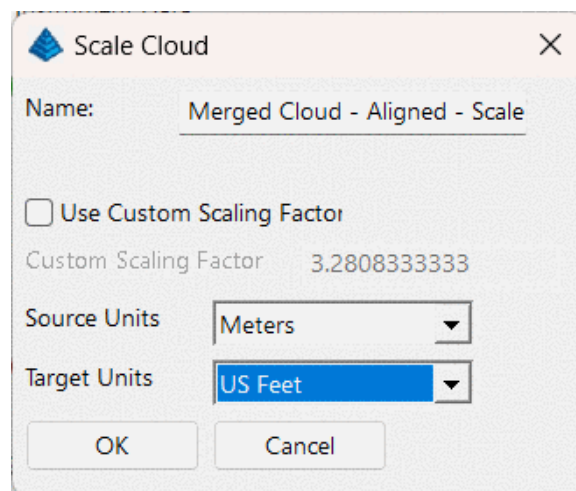
Initially, the dialog will have no transformations specified. To add a transformation, click the **Add** button which will open the **Edit Transformation** window. Use the **Type** drop-down menu to select one of the four transformation **Types**. Enter the appropriate parameters and click **OK**.



You can add more transformations to the list, modify the currently selected one by clicking the **Edit** button, **Delete**, and change their order of application to the data using **Up/Down** buttons. Press the **OK** button in the **Transform Cloud** window to apply the transformations to the selected data.

Scale Cloud

To scale a cloud from Metric to US Feet and vice versa, **right-click** on the cloud in the **Current Project** tree, select **Transform**, and choose **Scale**. This will display the **Scale Cloud** dialog. Enter a **Name** for the new Cloud. Select the **Source Units** and **Target Units** from the pull-downs. Alternatively, check **Use Custom Scaling Factor** and enter the **Custom Scaling Factor** you want to apply. Click **OK** to begin the scaling. The new Cloud will be displayed under the **Clouds** sub-menu in the **Current Project** tree.



Instrument Data Project Items

There are two data types in Carlson Point Clouds. These are Instrument Data and Processed Data. Within each data type there are various items and objects that can be added, edited or created. These Data types and items are organized into a tree structure and each can be manipulated by right clicking to bring up a menu specific to that item type. The menu options presented when an item is right-clicked will depend on what the item is. The data types listed here are specific to the Instrument Data type. Common functions can be found in the Data Objects section.

Points

There are three kinds of point item types in Carlson Point Clouds:

- Control Points
- Target Points
- Coordinate Points

Coordinate Points are processed data and are discussed in the Processed Data section.

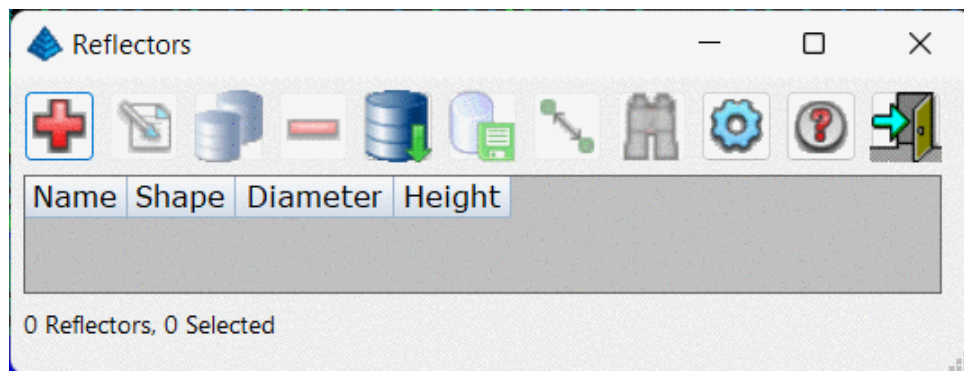
Target points always exist as a sub-item of a **Scan Position** and are in that scan position's local coordinate system. This means that a project can have several sets of target points (one for each scan position).

Control points are the global positions of the target points, and are a sub-item of the **Instrument Data** folder. There can be only one set of control points for a project.

Reflectors

Reflectors in a Point Clouds project represent a disk, sphere, or cylinder reflector within the set of scans for the current project. Each reflector can be linked to a control point which will enable Point Clouds to account for the shape of target points that are to be linked with that control point during the scan registration process.

These reflectors can either be imported from an ASCII text file with the **Import** command or entered in manually with the **Edit** command. Pressing **Edit** will bring up the Reflectors editor dialog box.



Below is a list of the Reflectors Editor options:



New Reflector brings up a dialog that allows you to specify the properties of a new reflector to add.

Name will specify a unique name to be used for the reflector in the registration process.

Shape is a dropdown with three options relating to the three reflector shape types (**Disk**, **Cylinder**, and **Sphere**).

Diameter allows you to specify the diameter of the sphere, cylinder, or disk objects.

Height is accessible only for the cylinder reflector type and specifies the height of the cylinder.



Edit Selected Reflector brings up the Reflectors edit dialog which allows users to change properties of the currently selected reflector.



Copy Reflectors will copy the reflectors.



Delete Selected Reflector deletes the currently selected reflectors.



Import Reflectors will import reflector information in text (ASCII) format.



Export Selected Reflectors will export the reflectors. When pressed, it will bring up the ASCII file export dialog for options.



Transform Reflector will transform the reflector.



View Reflector will display the reflector.



Settings allow users to configure what properties of the reflectors are visible in the spreadsheet control.



Help opens the help documentation.



Exit will close the Reflectors dialog.

After all the reflectors have been added to the project, linking them to their corresponding control points is done through the Control Points Editor.

Coordinate System

The Coordinate System is the current transformation applied to a Scan Position. This transformation is applied to any scans that belong to the current scan position. Typically, manual modification of the coordinate system should not be necessary, as the proper coordinate system to align scans properly should be determined during the registration process. To modify the coordinate system for a scan position, *right-click* it and select **View**. This will bring up the Coordinate System editor.

The transformation panel displays the transformation matrix that is applied to the scan position. Use **Import** to import a coordinate system file (.dat). Use **Export** to export the coordinate system from the point cloud project.

Images

Images are stored in a folder as a sub-item of a scan position. There can be multiple images per scan-folder, and typically are pictures taken from the position of the scanner. Images can only be viewed in 2D mode.

1. *Right-Click* **Images** in the **Current Project** tree.
2. Select **Add > Existing**.

3. In the **PointCloud - Add Image** window, select an existing image to add and click **Open**. Current supported file types are **BMP, GIF, JPG, JPEG, PNG, TIF** and **TIFF**.
4. The **Transfer File** dialog specifies if a copy of the image is to be save to the target location or move the source file and whether to use an existing target file if available or replace target file.

The **Action** panel allows the user to choose to copy the selected image to the target location or move the selected image. Moving the image will delete it from the source location and it will only be available in the target location.

The **If target file exists** panel determines what to do if the file name of the image being added already exists in the project folder.

Processed Data Project Items

There are two data types in Carlson Point Clouds. These are Instrument data and Processed data. Within each data type there are various items and objects that can be added, edited or created. These Data types and items are organized into a tree structure and each can be manipulated by right clicking to bring up a menu specific to that item type. The menu options presented when an item is right-clicked will depend on what the item is. There are several functions that are shared among multiple object types. These common functions are discussed under Data Objects. This section deals with Processed Data objects. The following data types are considered Processed Data:

- Clouds
- Contours
- Coordinate Points
- Grids
- Layers
- TINs
- Planes
- Polylines
- Profiles
- Sections
- Text

Clouds

Clouds can be created several ways.

1. You can *right-click* a scan and select **Create** > Cloud.
2. You can *right-click* **Clouds** and select Import > then select point cloud format.
3. You can *right-click* **on a Cloud** and select Create > Create Cloud.
4. You can right-click on a Cloud and select Copy.

5. You can *right-click* on a Cloud and select Merge.
6. When applying some filters to a point cloud, you can select to create new cloud.

Contours

A contour item consists of several contour polylines that have been extracted from a TIN or cloud. Contour objects can only be viewed in 3D mode. Contour objects can be drawn to the CAD engine. Contour extraction is done through the **Action Tab** of the project manager.

Coordinate Points

There are three kinds of point item types in Carlson Point Clouds:

1. Control Points

2. Target Points
3. Coordinate Points

Control Points and Target Points are Instrument Data and are discussed in the Instrument Data section.

Coordinate points are points extracted from the final registered data sets and are a sub-item of the **Processed Data** folder.

Grids

Grids are a rectangular representation of a TIN. They are created from Cloud scenes or imported. For more information about creating Grids see Create Grid under the **Action Tab**.

Layers

Layers allow the user to control visibility and color for entities drawn in the Point Cloud scenes. For additional information see Layer Properties Manager under Common Utilities.

TINs

TINs are used to extract profiles, sections and can be exported as .TIN files for use with other Carlson modules. For more information about TINs see TINs under Processed Data.

Planes

Planes may be created as part of the process of Contour extraction, Sections and/or Profiles. Planes may also be created by right-clicking **Planes** on the Project tree and selecting **Add > New**. Planes created from the Project Tree will need to be edited to establish location and normal direction. To edit a Plane **double click it** in the project tree or **right-click it** and select **edit**. Planes created by extracting data can also be edited.

Plane extraction may also be initiated by selecting **Plane** from the **Extract Panel** of the **Action Tab**. These planes are a best fit of cloud or scan point data based on user entered parameters and can be used to illustrate wall positions with the draw polyline option.

Polylines

Polylines are usually extracted from the point cloud data working through the Action tab, but they can also be imported from CAD or manually entered with the **Import** and **Add** commands. If you **Add** a polyline it will appear under **Polylines** in the project tree. To edit the new polyline you can **double click it** or **right-click it** and select **Edit**.

Polyline > Import opens the file selection dialog. Users can select TXT, PLN (Carlson polyline file) or CL (Carlson centerline) files to import. The **Edit** option brings up the Polyline Editor; see the documentation on the Polyline Editor for more information.

Profiles

A profile consists of a set of height values and distances taken along a polyline extracted from a cloud, scan or TIN. Profiles can be viewed in 2D and 3D mode and drawn to the CAD engine. Additionally, profiles can be extracted to the .PRO file format. Profile extraction is done through the **Action Tab** of the project manager.

Sections

A sections object consists of a list of profiles taken at set intervals or distances along a polyline. Section extraction is done using the **Action Tab** in the project manager. Sections can only be viewed in 3D mode and can also be drawn to the CAD engine. Additionally, sections can be extracted to the .SCT file format.

Scenes

Scene allow you to visualize the point cloud project data. Scenes maintain a list of objects that are visible in the view that you have selected, for example a point cloud with imported polylines from CAD, graphics properties for those objects, and camera information for the scene. If a scene is closed and then reopened, these properties should still be the same.

Scans

Scans represent the raw data taken from a scanning device. In addition to the position, intensity, color and normal values found in a Cloud, a Scan also maintains the scanner position, a mapping of scanner pixels to points in the cloud, and a mesh structure that creates each point in the scan with its neighbors. Each Scan is associated with one scanner position, which defines the location of the scanner when the scan was taken.

Scan Positions

A scan position encapsulates the entirety of the information needed to match scans taken from a single position with the control points of a project. This data includes any scans taken from that position, the target points associated with that position, any images taken from that position, and the current coordinate system for that scan position.

Importing a Scan

Each of the point engines that are included with Point Cloud support a variety of formats for importing.

To import a scan, right-mouse click on an existing scan position in the **Current Project** tree and select **Import** then click **Scan**. The **Import Scan** file selection dialog will be displayed. Currently **TXT** is the supported file import type.

Navigate to the desired file and click **Open**. The **Import Scan** window will be displayed.

Note: If importing a non ASCII file, the file will be converted to Point Cloud's internal format and saved to the project.

The settings in this **Import Scan** must be configured to match the properties of the scan being imported. First, verify that the correct **Column Delimiter** is being used.

Next, clicking the header of each column to open a menu with possible values that can be assigned to that column. Click on the correct value in the menu to assign it to that column.

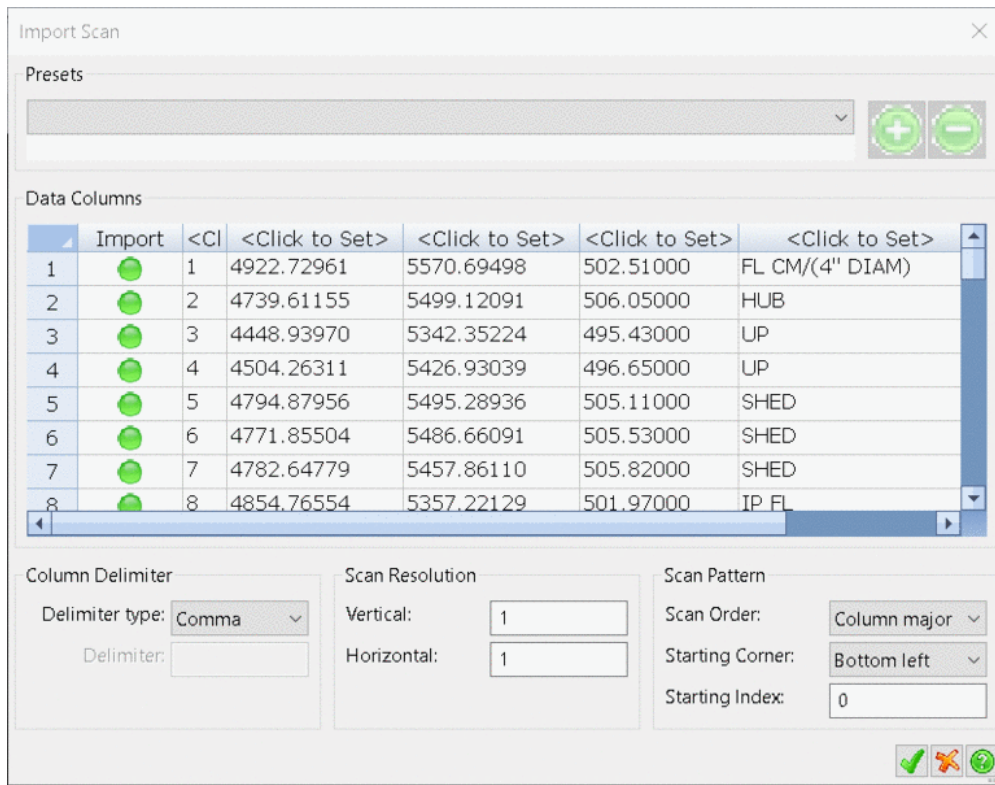
Note: Typically these values can be found in the header of the text file or in a file accompanying the text file.

Scan Resolution allows a user to define the **Vertical** and **Horizontal** resolution for the new scan.

The **Scan Pattern** options determine the ordering method of the scan's data, as well as the starting corner of the scan's data, the data for these values can often be found in the same place as the scan's resolution information, but if not, the default, or the default with the starting corner set to Top Left are the most common scan pattern types.

Users can select a preset export format by selecting it from the **Presets** drop-down menu. To define a new preset, define all of the **Import Scan** options as required including the column values and then click the **Green Plus** button next to the **Presets** drop-down. The new preset will be added to the drop-down menu for future use.

To delete a preset, select it from the **Presets** drop-down menu and then click the **Green Dash** button next to the **Presets** drop-down menu.

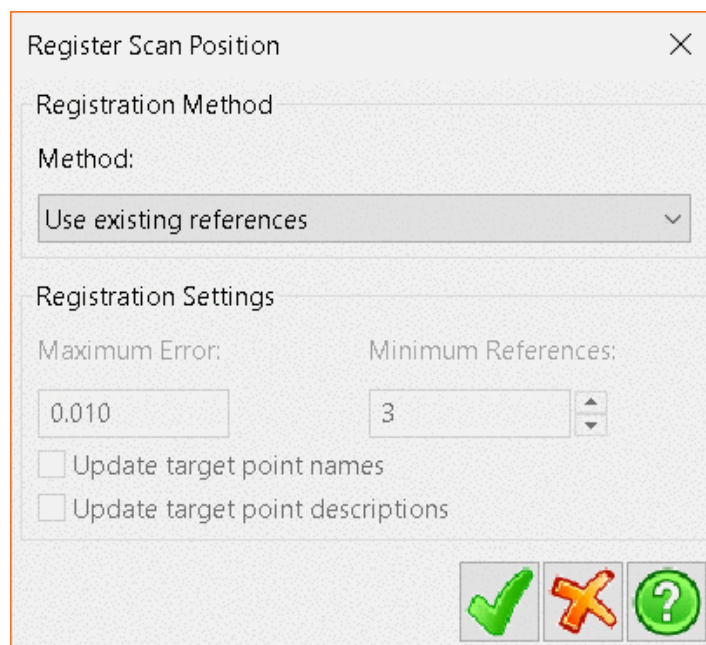


Importing Target Points

Target points can be manually entered or imported via a similar process to scan importing. To enter the target points in manually, right-click the target points object belonging to the scan position to modify and select **Edit**. This will bring up the **Edit Points** dialog for the target points.

Registering Target Points

The scan registration process begins with pressing the **Register** button in the toolbar of the **Target Points Editor**.



Carlson Point Cloud supports four different methods of scan registration:

Use existing references will make Point Cloud only use the references that already exist to perform the registration process (these references can be set manually in the Points Editor, or could have been set in a previous registration process).

Match point names will match target points up with control points with the same name.

Match point description will match target points with control points with the same description.

Minimize position error will attempt to automatically transform the current target points so that they match up with the control points, creating control point-target point pairs. If such a transformation that meets these conditions exists, the target points will have their reference control points set to the control point they were paired with.

Once satisfied, click the **Green Tick**. A dialog will display informing the user if the registration was successful. After all scans have been registered, verify the registration visually by viewing the scans together in the same Scene.

Creating a Scan

Unlike clouds and meshes, Scans can only be created from other Scans. This is due to extra information required for the indexing of the cloud data into the scanner's image.

To create a new scan from an existing one, right-mouse click the source Scan in the **Current Project** tree and select **Create > Scan**. This will display the **Create Scan** window.

Create Scan from single source

Note: Typically, the default options give the best results.

Scan Dimensions determines the height and width (in angles) of the new scan to create:

From Scan will just the ranges from the original scan

Specify will allow the user to specify their own custom angles.

Scan Resolution determines the data density of the new scan, smaller **Delta Angle** values will give denser data, while larger resolutions values will do the same.

From Scan will just the ranges from the original scan

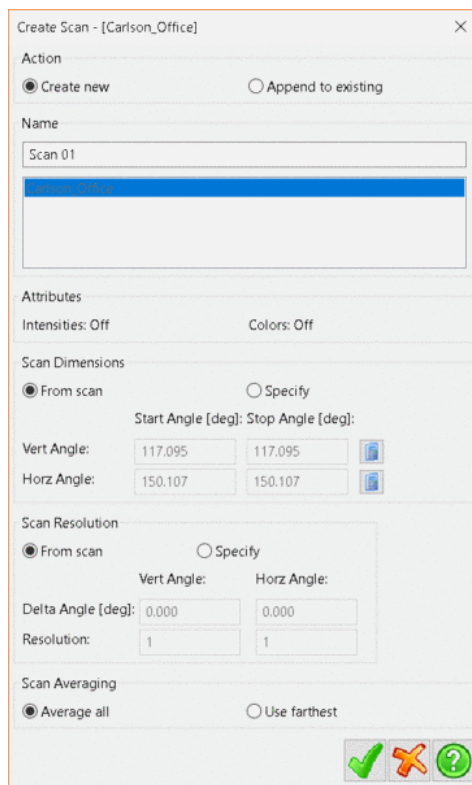
Specify will allow the user to specify their own custom angles

The **Scan Averaging** determines the action to be taken when it is found that multiple scan points in the source data set fall under the same scan pixel in the new scan data set.

Average All will average all color, position, and intensity values, discarding the original data.

Use Farthest will only use the data point furthest from the scanner and discard all other data points.

Click the **Green Tick** to create the new scan.

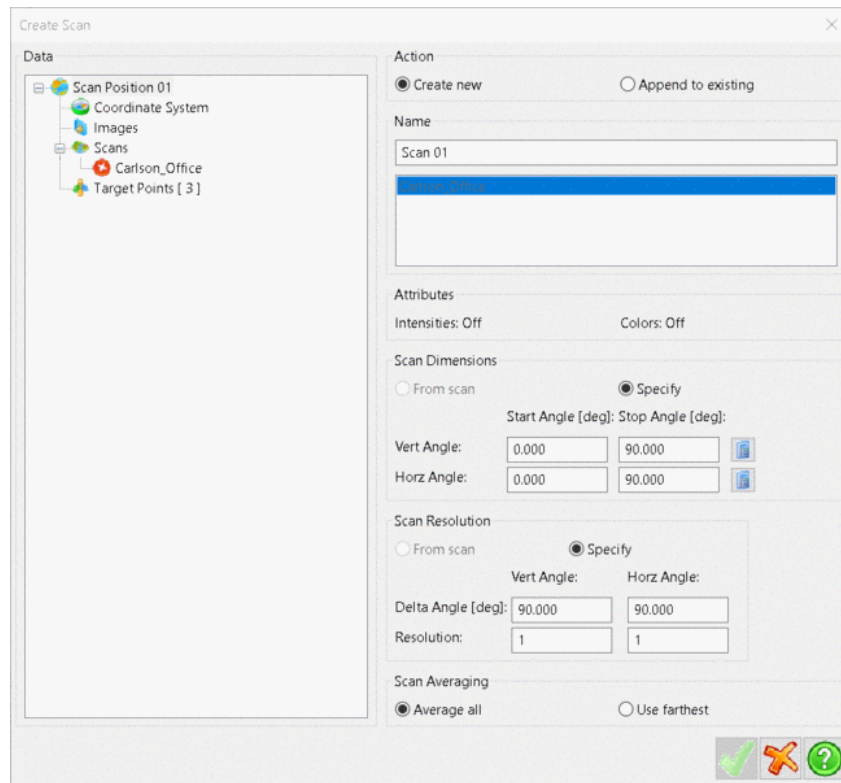


Create Scan from multiple sources

To create a scan from multiple source scans, right-mouse click on the **Scan Position** in the **Current Project** tree and select **Add > New**.

The options on the right side of the **Create Scan** window have the same functionality as the single source dialog. The tree structure on the left side of the dialog allows the user to choose the source scans to use for data. Clicking the **red x** next to a scan will turn it into **green circle**, signifying to use that scan as a data source.

Click the **Green Tick** to create the new scan.



Cleaning a Scan

To clean up scanner data to make it more uniform and remove possible outliers, right-mouse click the scan in the **Current Project** tree and select **Clean**.

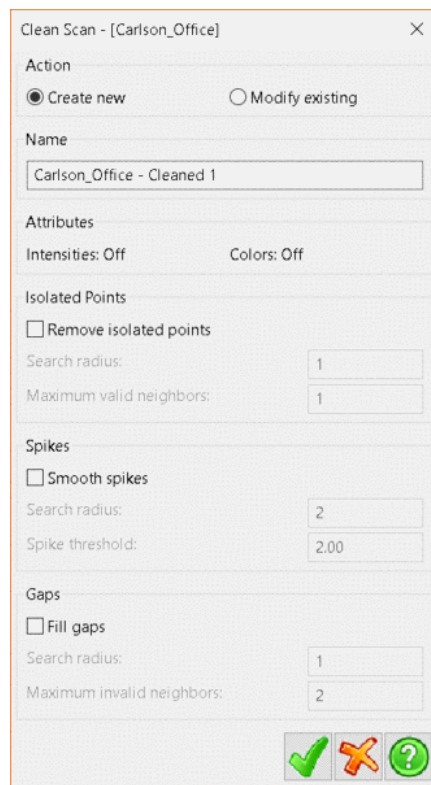
Carlson Point Cloud has three separate methods of cleaning scans:

Remove Isolated Points searches all neighbor data points within a square with a side length of **Search radius** pixels around each pixel in the scanner image for valid data. If less than the maximum amount of valid neighbors specified is found, the point is deleted.

Smooth Spikes reduces spikes in the mesh by finding any points whose distance from the scanner is larger than **Spike threshold** standard deviations from the average of the pixels in the search radius.

Fill gaps searches each invalid point's neighbor's for the specified amount of invalid neighbors. If less than the specified amount is found, then the invalid reading is changed to a valid reading with a range equal to the average of the neighbor points and color values equal to the average color values.

Click the **Green Tick** to clean the scan.



Resampling a Scan

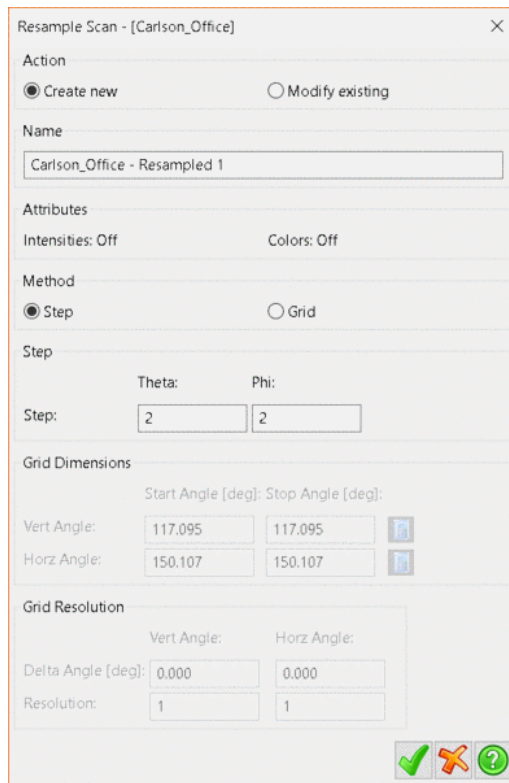
To resample a scan, right-mouse click on the scan in the **Current Project** tree and select **Resample**.

There are two methods of resampling:

The **Step** method of resampling steps through the scan using the step values provided for the step size. With a **Theta** and **Phi** value of 2 each, the scan will be reduced to 1/4th of its original size.

The **Grid** method divides the scan up into a grid where each grid block is of the specified size. It then averages the values found inside each block to a new single value for that entire grid block.

Click the **Green Tick** to resample the scan.



Tab Location(s): Project Tab

Prerequisite: Scanner Data

TINs

A TIN or Triangulated Irregular Network is the final result of processing a scan or cloud and are the basis upon which most data extraction is performed. The TIN structure itself is a set of vertices and a set of edges and faces that connect those vertices as well as any color or intensity values that were in the cloud or scan that the TIN was created from.

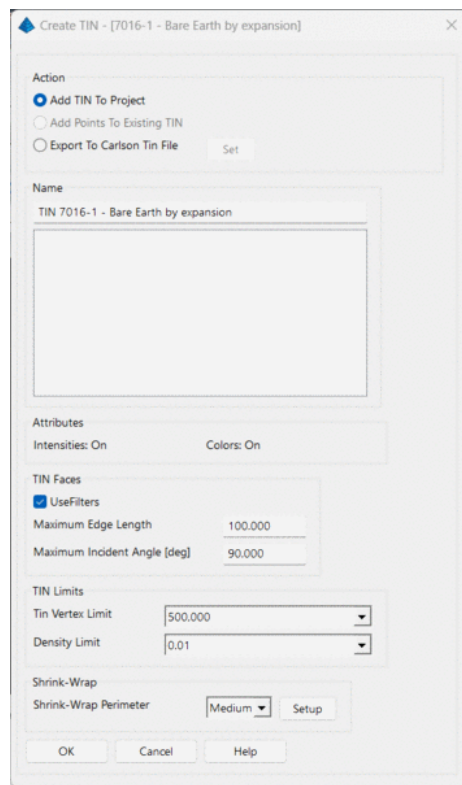
Importing a TIN

One can import a .tin file simply by right-clicking the **TINs** folder and selecting **Import** and navigating to the .tin on disk.

Creating a TIN

There are several methods for creating a TIN, but all of these methods use one of two cloud creation dialogs. The most common and direct method of creating a TIN is to right click the source data object (either another TIN, a cloud or a scan) and select **Create > Create TIN**. You can also access the **TIN** dialog from the **Action Tab**. Creating a TIN from another TIN will re-triangulate the points in that TIN based on the new normal information provided in the **Create TIN** dialog, so it is entirely possible that your new TIN can look completely different.

This dialog operates in two different modes: multiple source mode and single source mode. The single source mode is typically the result of right-clicking a data object in the Point Cloud tree structure and selecting **Create TIN** or from the **Action Tab**, this mode only allows the TIN to use the originally selected data as its source.



The **Action** panel gives you options to **Add TIN to Project**, adding a TIN to point cloud manager project, **Add Points to Existing TIN**, and **Export to Carlson TIN File** to save a .TIN file that can be read by other Carlson programs including Survey, Civil, Takeoff, Mining.

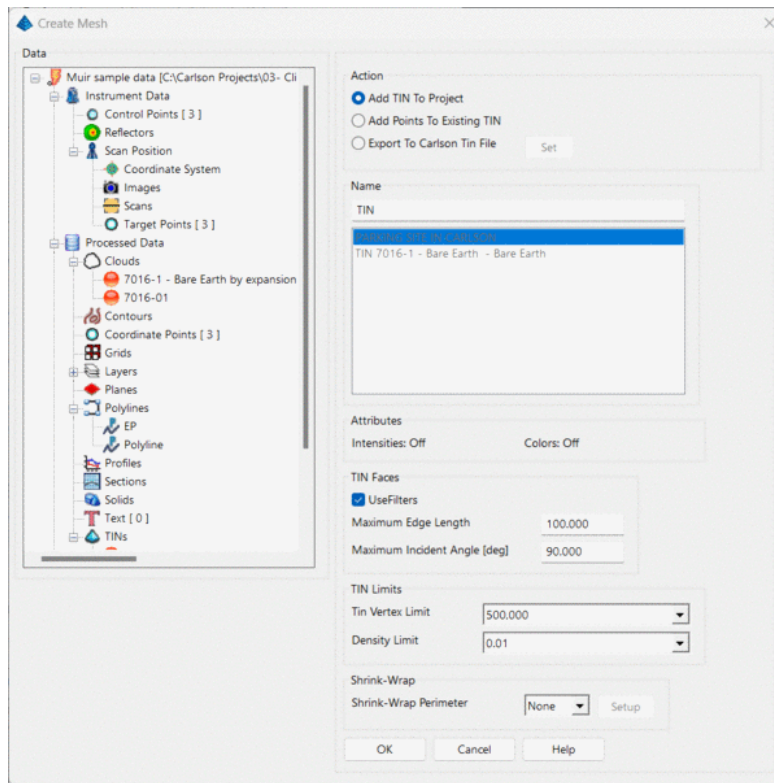
In the **Name** panel you can enter the name for the new TIN. If Add Points to Existing TIN is selected, you can choose the TIN to edit from the panel's list.

The **TIN Faces** panel allows you to set restrictions on the triangulation process. **Use filters** determines whether or not to filter out vertices based on **Maximum Edge Length** and **Maximum Incident Angle [deg]**. No edge in the TIN will be created if it will exceed the **Maximum edge length** or if the angle between the two faces it connects exceeds the **Maximum incident angle**.

In the **TIN Limits** panel, **TIN Vertex Limits** specify the maximum number of TIN vertices that will be used in a TIN. Setting this value too low may result in no TIN being created. **Density Limit** will limit the size of the smallest triangles.

In the case of scans, there will be a **From Scan** radio button that you can use, which will automatically use the scanner position information from the scan as the normal. The **Normal** is the direction to use for the Delaunay TIN triangulation; typically you want to use an axis that is representative of the direction that the data was taken from, such as the view direction of the scanner.

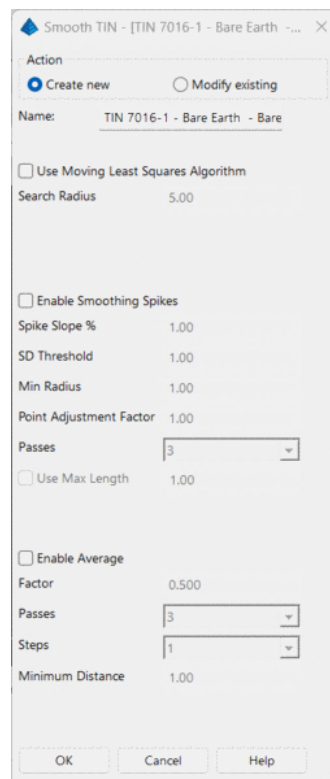
One can also create a TIN with multiple sources by **right-clicking** the **TINs** folder and selecting **Create New**, this will bring up a dialog similar to the one above with an extra tree control to the left.



All of the controls on the right half of the dialog are the same, but you can now toggle inclusion of objects into the TIN by clicking the **red ball** next to their name in the tree, turning the icon to a **green ball**.

Smooth TIN

Smooth TIN allows you to create a new TIN or modify the existing TIN. There are three methods for smoothing which may be used separately or in combination



Use Moving Least Squares - If toggled on, create a Moving Least Squares (MLS) approximation of TIN. The MLS algorithm fits a curved surface using least-square to each vertex of the TIN based on the neighbors within the **Search Radius** of that vertex. Each vertex is then projected onto its least-squares curved surface.

The **Search Radius** is a unit-less dimension used to determine how many neighbors a point has when generating a least-squared surface approximation for the TIN at that vertex. Any other vertices within the search radius distance of a vertex are considered neighbors.

Enable Smoothing Spikes - If toggled on, the routine will iterate over every vertex of the TIN and delete vertices that are considered spikes based off of the profiles in the xz and yz planes at that vertex and the smoothing settings.

Spike Slope % is the angle, in degrees, of the profile slope along the the xz and yz planes after which a vertex will be considered a spike. A value of 90% will disable this test

SD Threshold is the unit-less distance from the average plane of the neighbor points past which qualifies a vertex as a spike.

Min Radius is the unit-less distance from a vertex on the TIN to search for neighbor vertices.

Point Adjustment Factor specifies how far a point should move toward the the average plane of its neighbors. 1.0 meaning "move vertex's position to the plane" and 0.0 meaning "Do not change the vertex's position at all".

Passes are the number of times to apply the spike smoothing algorithm to TIN. Value 1-9.

Use Max Length, if toggled on, use "Max Length" alternative to the "Minimum Radius" option. Should only be used is there are a significant number of breaklines in the TIN.

Enable Average - If toggled on, smooths vertices of TIN by moving them towards the average elevation of their neighbor vertices' positions.

Factor is the factor to adjust a vertex's elevation by. 1.0 meaning "make vertex's elevation equal to the average of its neighbors' elevation), 0.0 meaning "do not change vertex's elevation".

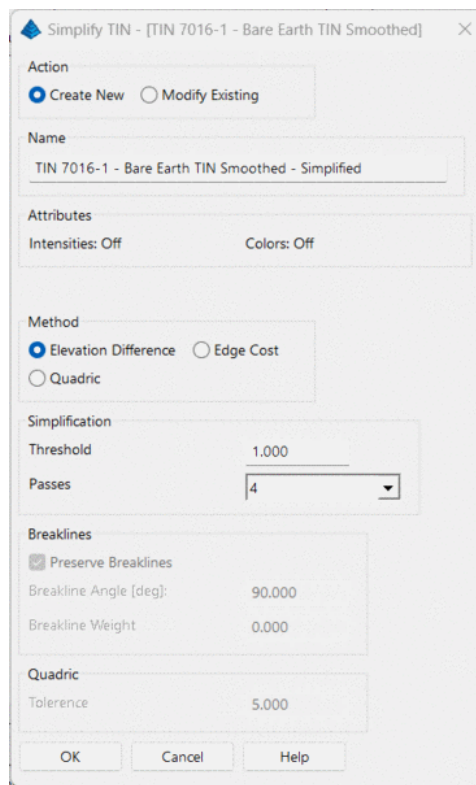
Passes is the number of times to apply "averaging" algorithm to TIN. Value 1-9.

Steps is used to specify the vertices considered neighbors of a vertex when applying the "averaging" algorithm to that vertex. Any vertex connected to the a base vertex (by an edge) is considered a 1-step-neighbor vertex to that base vertex. Any vertex connected to any of those 1-step-neighbor vertices that is not a 1-step-neighbor vertex or the base vertex is a 2-step-neighbor-vertex. In total, n-step-neighbor vertices will be used in the "averaging" algorithm for a base vertex.

Minimum Distance is the minimum elevation change that can be applied to a vertex's position when "averaging." If the distance from a vertex's elevation to its neighbors' average elevation is less than "Minimum Distance" it will not be moved.

Simplify TIN

Simplifying a TIN is one of the key ways to reduce data down to a state that it can be transferred over to the CAD software or to make it more manageable in Point Cloud without losing much surface quality. **Right-click** the TIN to simplify and select **Simplify**. This will open the **Simplify TIN** dialog.



There are three methods of TIN simplification available. Elevation Difference, Edge Cost, and Quadric.

The **Elevation Difference** method will loop over all the vertices in the TIN a number of times based on the number of **Passes**. The program will delete vertices whose deletion would not lead to a deviation in the current TIN of less than the **Threshold** value entered. Note that this method may be slower than the others.

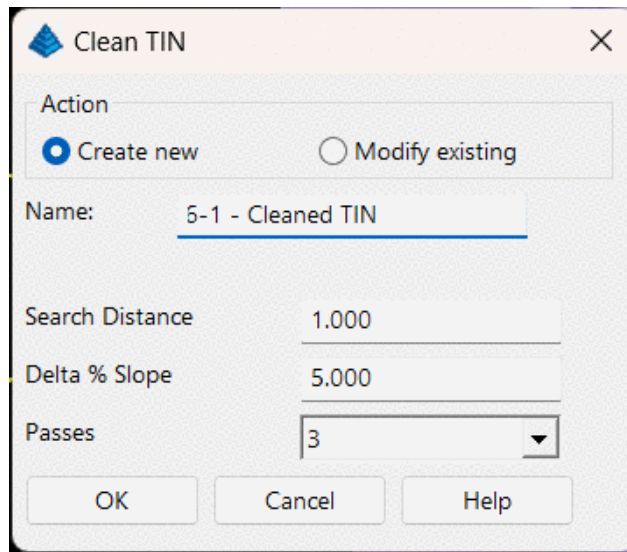
The **Edge Cost** method determines the total deviation in the TIN that result from each edge removal by merging its two vertices and removes all edges whose removal would result in a deviation of less than the **Threshold** value entered. In addition, the preserve breaklines options for the **Edge Cost** method will multiply the deviation value calculated by the **Breakline Weight** if the angle between the two faces it borders is greater than the **Breakline Angle**, this can be used to help preserve corners.

The Elevation Difference method is best used in largely flat data (such as a scan of a large open area), while the Edge Cost method is best for complex data with lots of corners. Additionally there is a memory trade off, the Elevation Difference method is generally slower at higher numbers of passes (which gives better results), while the Edge Cost method consumes more memory.

Clean TIN

To remove spikes from a TIN, **right-click** the target TIN and select **Clean**. Vertices that meet the parameters of this dialog will have their positions adjusted to meet the shape of the vertices around them, smoothing out the TIN.

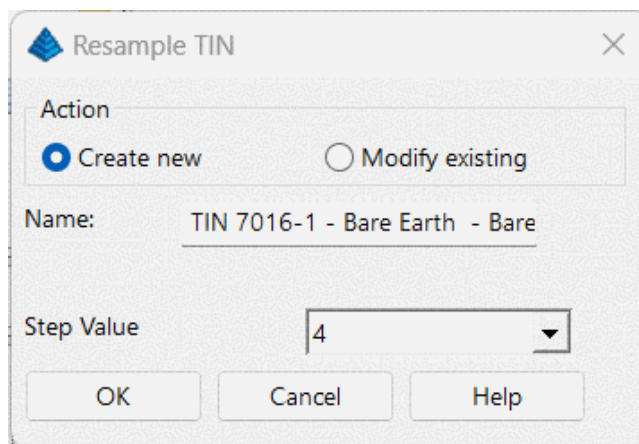
Search Distance determines the distance for a vertex to search for vertices that exceed the Delta % Slope. **Delta % Slope** is the maximum change in slope that is allowed. Vertices that exceed this value are removed. **Passes** determines the number of passes to make over the TIN.



Resample TIN

Resampling a TIN goes through the list of points in a TIN and takes one out one point at the **Step Value** interval. Therefore the number of points in the resulting TIN will be $(1 / \text{Step Size})$ times the original number of points in the TIN.

To run this command, **right-click** on the existing TIN and select **Resample**. In the Resample TIN window, the user can either choose to modify the existing TIN or create a new TIN with the resampled data. The **Step Value** can be varied from 2 to 10, depending on the final size of the output.



For example:

Original TIN

Cloud	Mesh	Notes
Data		
Vertices:	171,226	
Edges:	913,652	
Faces:	342,427	

10Mb

TIN Resampled Step 2

Cloud	Mesh	Notes
Data		
Vertices:	85,612	
Edges:	296,801	
Faces:	171,190	

5Mb

TIN Resampled Step 10

Cloud	Mesh	Notes
Data		
Vertices:	17,122	
Edges:	51,302	
Faces:	14,211	

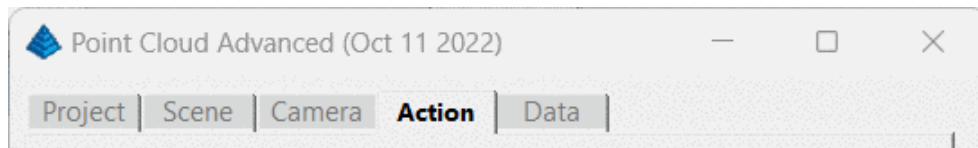
1Mb

Approximate File Sizes of Exported TIN

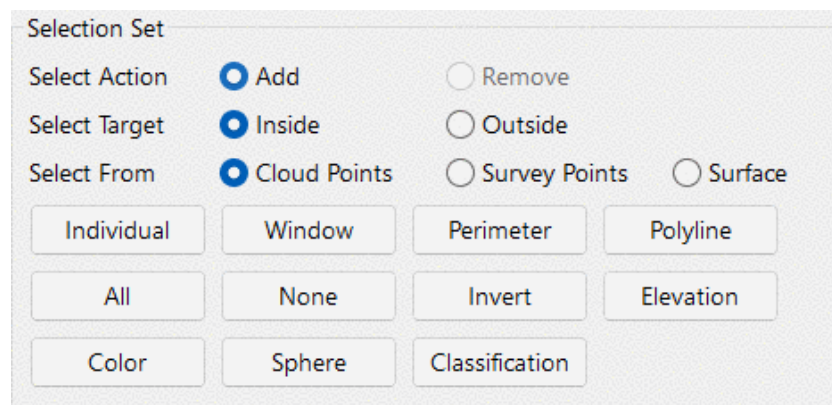
Tab Location(s): Project Tab
Tree Folder: TINs
Prerequisite: An Existing TIN

Action Tab

The **Action Tab** is the fourth tab in the **Point Cloud Manager**. All data extraction, as well as virtual surveying, is done here. A Scene with 3D data must be currently opened for the controls in the Action tab to be enabled.



The Action functions that can be used on the objects in a scene are broken up into five different Panels: **Selection Set**, **Edit**, **Transform**, **Create**, and **Extract**.



The group of radio buttons at the top of the **Selection Set** panel operate as a group to determine the behavior of the *Individual*, *Window*, *Perimeter*, and *Polyline* selection modes.

Add/Remove determines whether the objects that are selected in the view are to be added or removed from the current selection set.

Inside/Outside determines whether to perform the selection operation on entities inside or outside the boundary of the current selection set.

The next set of three radio buttons determines whether the entities to be selected are *Cloud Points*, *Survey Points*, or *Surface*.

There are four selection modes: **Individual**: every mouse click attempts to select the object currently under the cursor - useful for precision selection of individual points.

Window: you will be prompted to pick a base point and then pick a second point that defines the opposite corners of a selection window.

Perimeter: you will be prompted to pick several times on the scene to define the shape of the perimeter and then right-clicks to close the perimeter.

Polyline: you will be prompted to select an existing polyline in the scene. The polyline(s) can be drawn using the *Create Polyline* command or can be *Read from CAD*.

There are also seven global selection buttons:

All: selects all entities in the scene.

None: deselects all entities in the scene.

Invert: inverts the current selection set in the scene.

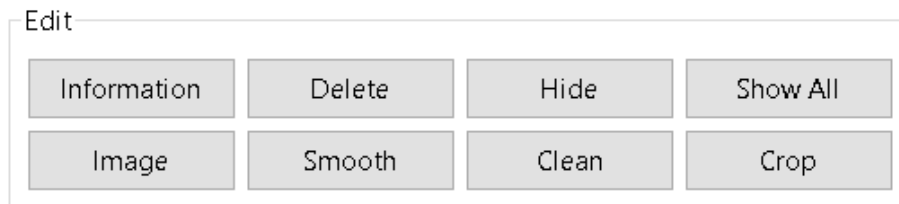
Elevation: you can specify maximum and minimum elevation values to build the selection set in the scene.

Color: selects all points with a matching RGB value to the point selected. Adjust the *Color Tolerance* to also select points with closely-matching RGB values from the base color. Users also have the ability to click on the **Screen Pick** button to set the tolerance by clicking **Screen Pick**, then choosing a first and second points to calculate the tolerance. Within the action, you can use the *Add/Remove*, *Inside/Outside* parameters as well as *Invert the Selection* and *Create a Region* from it.

Sphere: selects any points within a radius of the *Specified Point Position*.

A **Region** of selected Points can be saved to be hidden or to re-select at a later stage. See the Scene Tab page for more information.

The **Edit** panel contains controls for modifying the currently visible set of points.



Information button displays a properties dialog with some general statistical information about the current selection set, including the number of points selected, ranges of their positions, color ranges and intensity values. Click **Show Advanced** to display the Mean and Standard Deviation.

Delete button deletes the selected points from the scene and from any objects that contained them. The User will be prompted to confirm whether to Delete selected entities.

Hide button temporarily hides the current selection set, isolating the current area of interest without making permanent changes to the dataset.

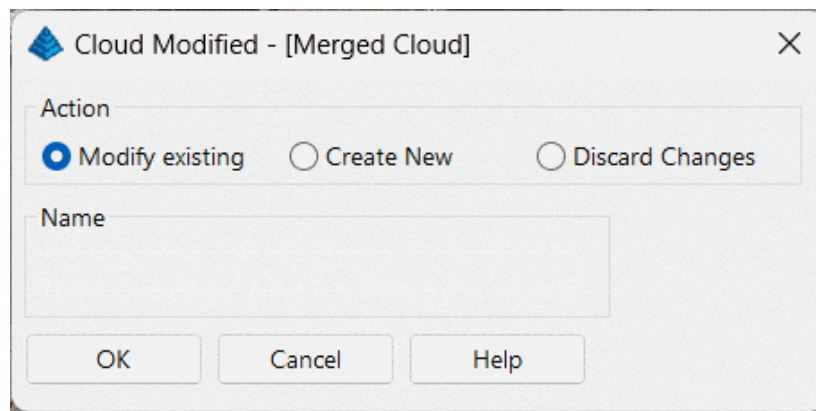
Show All will display all hidden points in the scene.

Image allows the user to drape an image over a TIN.

Smooth and **Clean** will act on the Cloud or TIN in the current Scene. See Smooth Cloud, Clean Cloud, or the TINs entry.

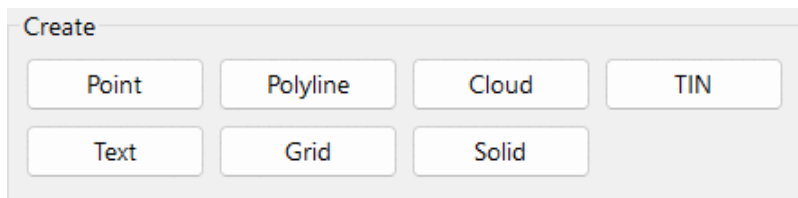
Crop deletes any points outside of the current selection.

As you make edits to the cloud, it is recommended that you Save the project. If you close the Scene viewer and haven't saved the changes yet, the program will prompt if you want to **Modify existing** cloud, **Create New** cloud, or **Discard Chnages**. If you select **Create New**, you will need to enter the **Name** for the new cloud.



The Transform panel contains two functions that allow you to apply translate, rotate, and scale adjustments to the selection set.

The **Create Panel** allows a user to manually create new project items including points, polylines and text, or by creating a new data set from the currently selected data points such as clouds, TINs and grids.



Create Point: will open the Point Creation action panel.

Create Polyline: will open the Polyline Creation action panel.

Create Cloud: will bring up the Point Cloud creation dialog. Enter the *Name* for the new cloud and click *OK* to create the new point cloud from the selected data in the current scene.

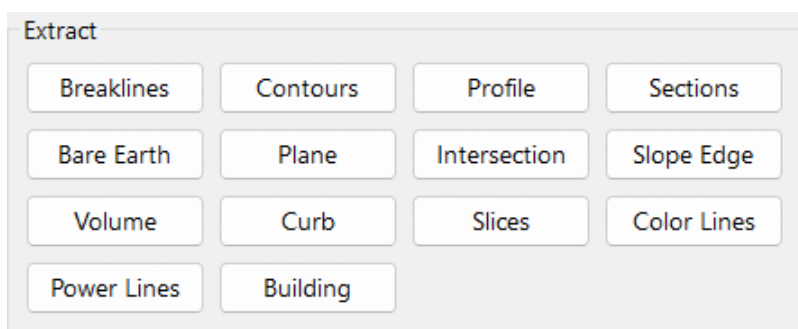
Create TIN: will bring up the TIN Creation dialog to create a TIN from the selected cloud data in the current scene.

Create Text: will open the Text Creation action panel.

Create Grid: will open the Grid Creation dialog to create a grid from the selected data in the current scene.

Create Solid: will bring up the Create Solid dialog to create a solid from the selected data in the current scene.

In the Extract Panel you will find functions to automatically extract features from point clouds such as curbs, parking lines, road markings, color edges, building and roof footprints, and power lines. Also in this panel are the bare earthing routines and tools to obtain profiles, cross sections, contours, and run volume calculations.



For more information on the specific extract tools, click on the links below to open the corresponding help entry:

- Breaklines
 - Contours
 - Profile
 - Sections
 - Bare Earth
 - Plane
 - Intersection
 - Slope Edge
 - Volume
 - Curb
 - Slices
 - Color Lines
 - Power Lines
-
- Building

Note that extracted features can be easily transferred to CAD. *Points* can be drawn in CAD using the **Field-to-Finish** routine or drawn using **Locate Points** by **right-clicking** on the **Coordinate Points** from the **Current Project Tree**. *Polylines* can be drawn by **right-clicking** on **Polylines** from the **Current Project Tree**.

Bare Earth

Clicking the **Bare Earth** button from the **Extract** panel in the **Action tab** will display the Bare Earth options. It can also be accessed by right-clicking on the point cloud in the Project Tree.

There are three methods for Bare Earth extraction, **Expansion**, **Grid** and **Profile**.

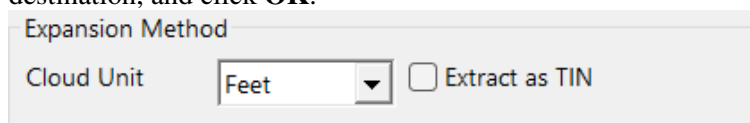
Expansion mode works by inspecting the cloud by dividing it into small cells, resolving the ground plane for individual cells from the bottom up, and then clustering the cells. Once it has determined what is most likely to be ground it will expand the point clusters to other cells. Lastly, it will create a TIN from the ground clusters to resolve isolated points. Benefit of this mode is that it has no parameters that need to be entered.

Grid mode works by inspecting the cloud as a grid to find the lowest valid points in each grid cell and keeping only those points.

Profile mode works by drawing a 3D line away from the scanner and seeing if it slopes upwards too sharply, removing any vertical segments and keeping any flat segments.

Expansion Method

Select the **By Expansion** radio button. This method only requires confirmation that the units are correct, choose the destination, and click **OK**.



Expansion Method

Cloud Unit: Feet Extract as TIN

Extract Destination: Can create either a new **Bare Earth Cloud**, a **Non-Ground Cloud** or **Bare Earth Region**. The Regions option creates regions within the current Cloud instead of creating a new Cloud. Regions are usually slower than creating a new Cloud.

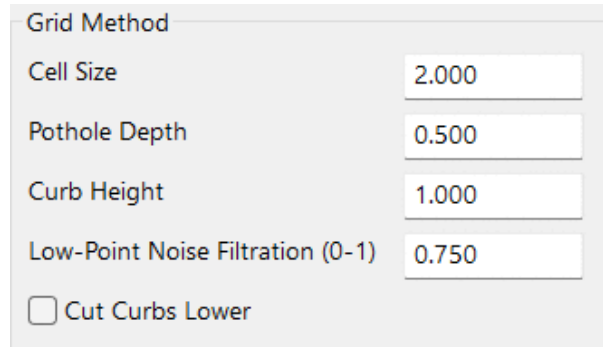
Name: Name for the newly created cloud

Cloud Unit: as the parameters for this method are internal, make sure that the selected unit matches the point cloud units.

Extract as TIN: will bare earth the point cloud and save the results as a TIN file rather than a new point cloud.

Grid Method

Select the **By Grid** radio button. Enter the desired cell size, pothole depth, curb height, and low-point noise filtration values. To begin the **Grid Method** Bare Earth extraction, click the **OK** button.



Grid Method	
Cell Size	2.000
Pothole Depth	0.500
Curb Height	1.000
Low-Point Noise Filtration (0-1)	0.750
<input type="checkbox"/> Cut Curbs Lower	

Extract Destination: Can create either a new **Bare Earth Cloud**, a **Non-Ground Cloud** or a **Bare Earth Region**. The Regions option creates regions within the current Cloud instead of creating a new Cloud. Regions are usually slower than creating a new Cloud.

Name: Name for the newly created cloud

Cell Size: Determines the grid size when analyzing the point cloud. It is recommended to use small cell size values.

Pothole Height: The tolerance for a point to be considered within the level that is to be kept for each grid cell.

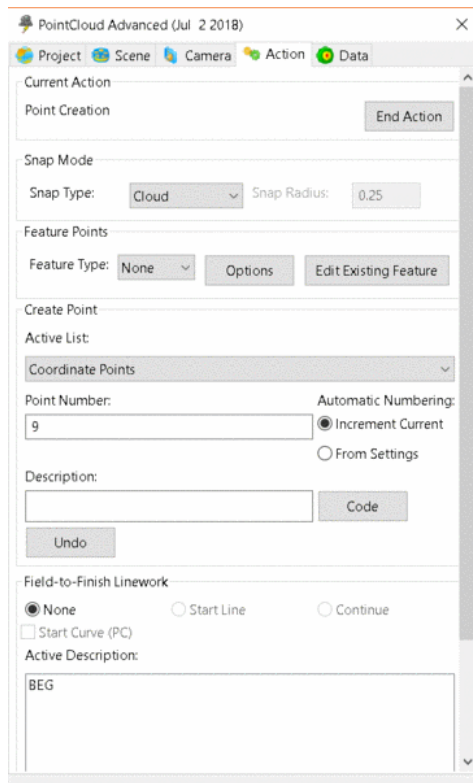
Curb Height: The tolerance between adjacent cell elevations to determine which cells should be kept.

Low-Point Noise Filtration: Filters out points below the level for each grid cell. A value of 0 will just accept the lowest point in each cell even if it's an outlier due to noise.

Cut curbs lower: When this is checked, doesn't take adjacent cells into account when deciding which points to take from each grid cell.

Profile Method

Select the **By Profile** radio button. Move the cursor to the Scene window and CTRL+click to specify the "scanner" position or center of the Bare Earth extraction. The scene must be in **Plan** view (View from Top) to run the Profile Method Bare Earth extraction. Points are examined and either included or excluded for the Bare Earth Cloud based on the extraction parameters.



Snap Mode Panel

The Snap Mode Panel offers twelve different options for snapping to points in the cloud in the open scene.

Cloud - No snap function is active. The vertex of the polyline will be placed at the cloud point nearest to the selected location.

Low - The point is placed at the lowest point (smallest z value) within the Snap Radius.

High - The point is placed at the highest point (largest z value) within the Snap Radius.

Low Edge - Snaps to the low edge of a feature, like a curb. A dynamic window in the upper left of the scene displays a cross section of the area within the Snap Radius and displays the high and low edge as red squares.

High Edge - Snaps to the high edge of a feature, like a curb. A dynamic window in the upper left of the scene displays a cross section of the area within the Snap Radius and displays the high and low edge as red squares.

Slope Bottom - Snaps to the bottom edge of a slope that is less than 45 degrees. This is useful for finding low edges on mountable curbs and other non-vertical features.

Slope Top - Snaps to the top edge of a slope that is less than 45 degrees. This is useful for finding upper edges on mountable curbs and other non-vertical features.

Average Point - This snap averages Northing, Easting and Elevation for all points within the Snap Radius and uses the averaged values for the coordinates for the new point.

End Point - Snaps the new point to the endpoint of an existing polyline.

Mid Point - Snaps the new point to the midpoint of an existing polyline.

Node - Snaps the new point to points placed in the drawing using Carlson Point Cloud Create Point.

Nearest - Snaps the new point to the point on an existing polyline nearest to the cursor location.

Feature Points Panel

Feature Points

Feature Type:

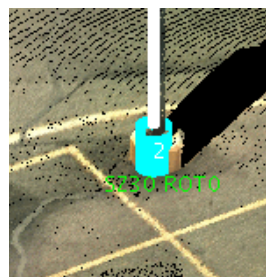
Feature points are specific types of features that Carlson Point Cloud can automatically extract. These points are extracted as 3D elements to reflect the feature selected and display the general shape in the Point Cloud scene. The screen captures below show the various 3D elements for each feature type. There are six options in the Feature point drop list.

None - No Feature type is selected.

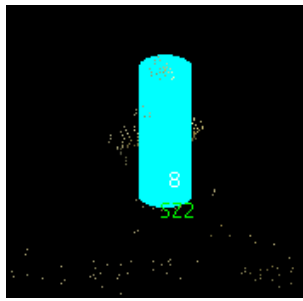
Tree - The user picks a point near the bottom of a tree in the scene and the trunk diameter, height, and drip line diameter are extracted and displayed as the description for the point.



Pole - The user picks near the base of a pole in the scene and the diameter and height are extracted and displayed as the description for the point.



Hydrant - The user picks near the base of a hydrant in the scene and the diameter and height are extracted and displayed as the description for the point.



Circle - The user picks anywhere on a vertical structure such as a pole and the diameter of the pole is extracted. The program then prompts for an elevation and the user can pick a second point for the actual elevation of the pole. This is useful when the bottom of the pole is obscured.

Corner - This feature places a point at the intersection of two building faces. The user picks the first flat building face followed by the second. The program then prompts for a third location to use as an elevation. The result is a point set at the intersection of the two building faces at the elevation picked such as the actual ground.

Clicking **Options** from the **Feature Points** panel will display the **Feature Options** window.

Hide Feature Points - When toggled on this command will "hide" or remove from the scene the points that make up the feature selected. For example, if feature type Pole is active and you select the base of a pole, the points that display as the pole in the scene will be hidden. A coordinate point is placed showing the diameter and height of the pole.

Use SZ Code - When toggled on this command will change the description displayed in the scene and stored in the Coordinate list. The new description is coded to take advantage of special codes in Field to Finish to more accurately display the features. Size or height, rotation and diameter or width. By setting up the Field to Finish file (.FLD) to place 3D blocks and use the special codes placed in the description by Point Cloud a more realistic and accurate 3D view can be created in CAD. **Draw Solid Feature** - draws the item as a solid

For the **Tree, Pole, Hydrant, Fence, Building, Circle, Corner** settings, fill them out to suit the scene.

Note: If the Hide Feature Points is toggled on and features are selected the user will be prompted to update the cloud, save a new cloud or discard changes when closing the scene.

Create Point

The Create Point panel displays options for creating a point.

Create Point

Active List:

Coordinate Points

Point Number: Automatic Numbering:

Increment Current

From Settings

Description:

In the Create Point panel you can choose the destination of points created using the Active List drop list. Points created will be added to the list displayed in the Active List. Three options are available.

- Control Points
- Scan Position
- Coordinate Points

Point Number determines the number to be assigned to the point. There are two options for numbering.

Increment Current will follow the standard numbering convention of Carlson Civil Suite (increment the number without any filler digits).

From Settings will use the prefixes and digits from the naming Conventions in the **Project Settings**

Clicking **Code** will bring up the **PointCloud - Code Table** dialog.

PointCloud - Code Table

FLD File
C:\Carlson Projects\Settings\carlson.fld

Categories

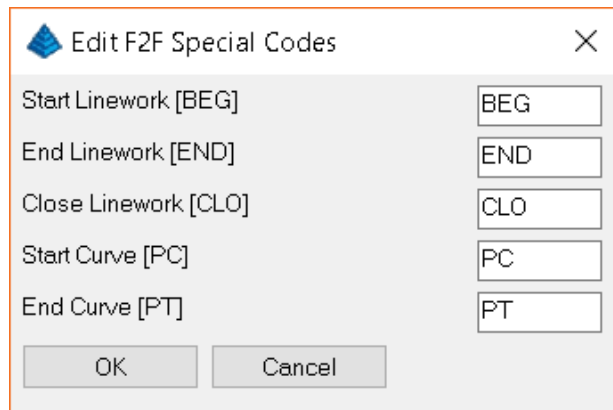
- FENCES & WALLS
- GROUND FEATURES
- MISC CODES
- MONUMENTS
- PINE TREES
- ROAD FEATURES
- SET CONTROL
- SMALL PINES TREES
- SMALL TREES & SHRUBS
- STRUCTURES
- TREES
- UTILITIES

Codes

Code	Full Name	Enti
2D	2"	Point
2P	2" PINE	Point
4D	4"	Point
4P	4" PINE	Point
6D	6"	Point
6P	6" PINE	Point
8D	8"	Point
8P	8" PINE	Point
9P	9" PINE	Point
10D	10"	Point
10P	10" PINE	Point
12D	12"	Point
12P	12" PINE	Point
14D	14"	Point

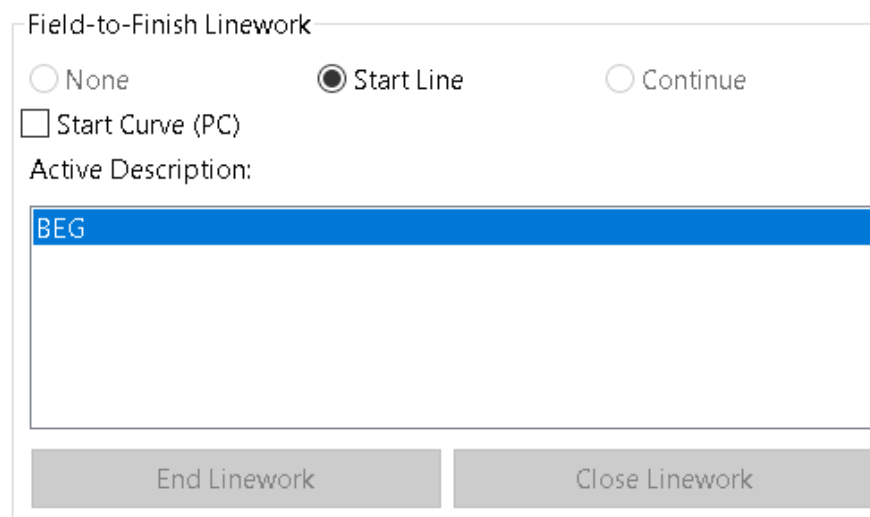
Pick FLD File allows the user to set the current field-to-finish file (.FLD). Once an FLD file is open the user can select a code to use to create points by click the desired code and then clicking the Green check mar.

Special Codes allows the user to adjust the F2F Special Codes used for Start, End and Close of Linework and the Start and End of Curves.



Field-to-Finish Linework

The Field-to-Finish Linework panel allows you to specify properties of the next field-to-finish linework segment (if the current code table entry has linework associated with it). If your code does not specify linework you will receive a message that your code does not have linework associated. Once you clear the message dialog your point will be placed and the linework options will be set to **None**.



Once a code is selected from the Create Point panel the user may select one of the Field-to-finish Linework radio button options.

None no linework is created. This is the default when a Point code without linework is selected.

Start Line Begins a line using the parameters in the code table. Lines are only created for point codes that have linework associated with them.

Continue is selected as you continue adding points to the current field-to-finish linework.

Start Curve (PC) Sets the starting point for an arc. The arc will be drawn using the parameters in the code table.

Prompts

Specify Vertex Position: Places a point at the selected location. If the point Code is associated with line work the point is connected to the previous point with a line.

Survey by Grid



The survey by Grid command sets points throughout the entire cloud at a preset interval. ie. 25' grid.

Point Code - Specify a single code for all grid points

Starting Point Number - Specify the point number to begin numbering sequence with

Cell Size - Enter the size of the grid spacing

Point Within - Specify the radius for the program to consider for the Elevation Method Snap setting

Elevation - Similar to above, specify the snap settings for elevations within the radius specified.

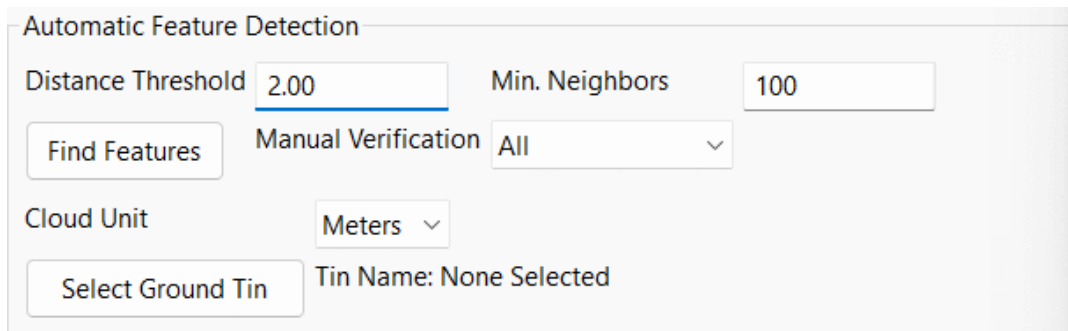
Example: The above settings would set points at the lowest elevation found within a 0.25 radius. (The drawing units are considered ie. meters, feet)

Automatic Feature Detection - *Experimental Feature* -

This requires a lot to setup - all of the Features listed above (tree, pole, etc) need to have their Settings set correctly in order for this to run correctly.

NOTE: the objects to be extracted need to be, in general, not touching each other. So a tree touching a fence would probably not work. Fences touching a building are OK. **The features in the cloud need to be well separated and isolated from one another.**

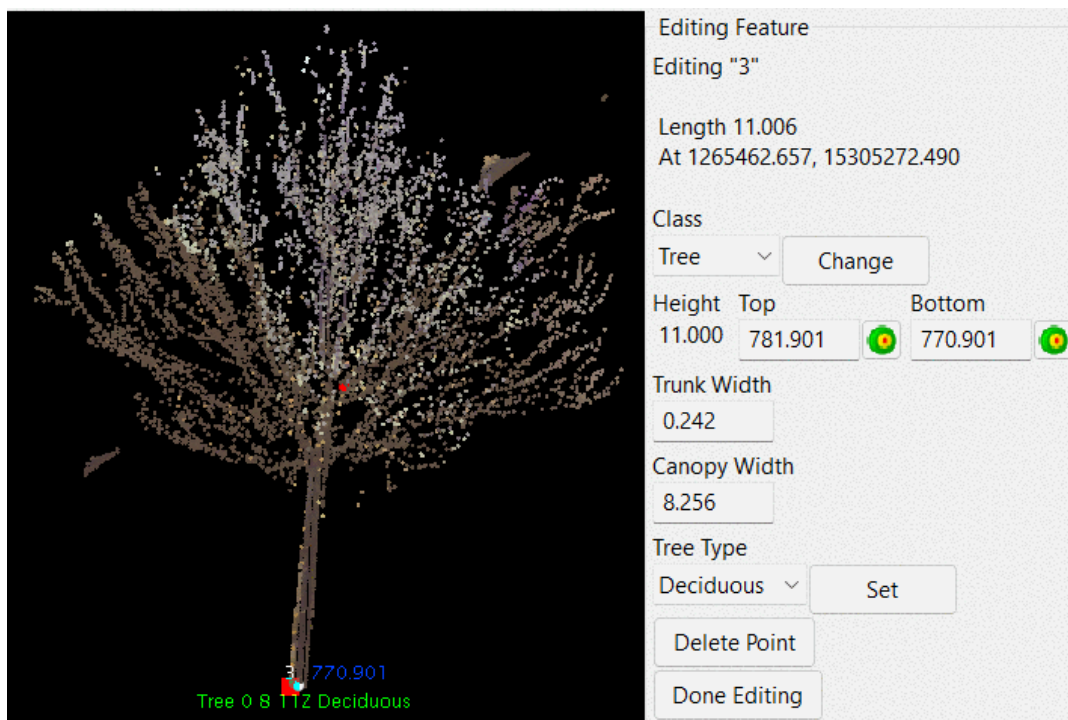
In most cases, it would be worth extracting the bare earth and creating a non-ground cloud. Then create a TIN ground surface that can be used in this routine. By having a TIN set as the ground reference, it will allow the software to better differentiate between the ground and the feature. The TIN can be turned off when loaded into the scene - with the **Show Front** unchecked, then TIN will be hidden but still used in the routine.



Manual Verification being set to ALL forces the user to run through each feature that has been found to confirm that they are indeed correct. If set to **Low Confidence** it will only pause on items that are uncertain.

Once all the settings for each feature has been configured, click on **Find Features**. It ask for the TIN file then run through the routine finding all the features.

It will now run through each feature for confirmation (if set to **All**).



When editing, the **Class** can be changed along with the **Top** and **Bottom** elevation of the tree. Click on the icon to the right of each box to pick the point on screen.

NOTE: this

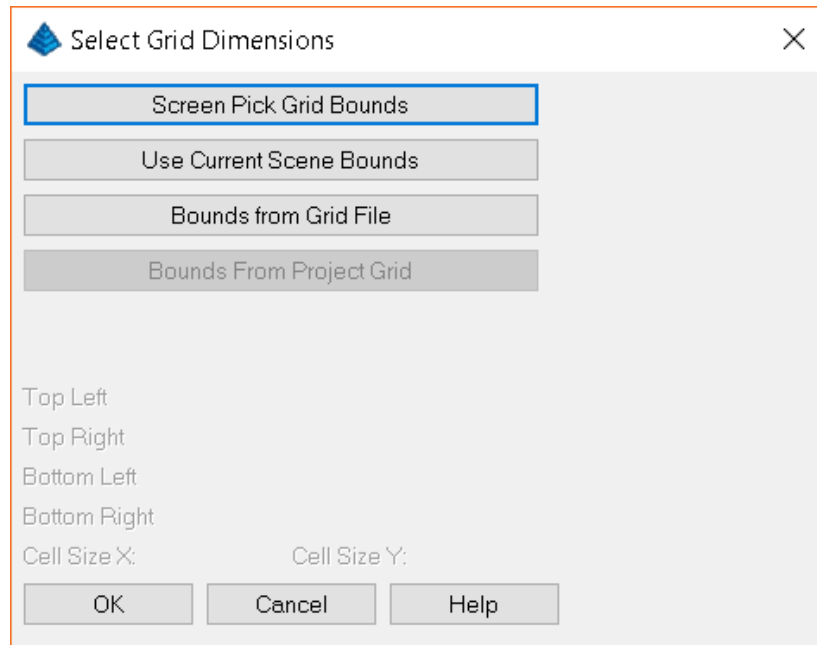
Tab Location(s): Action Tab

Panel and Button: Create and Point

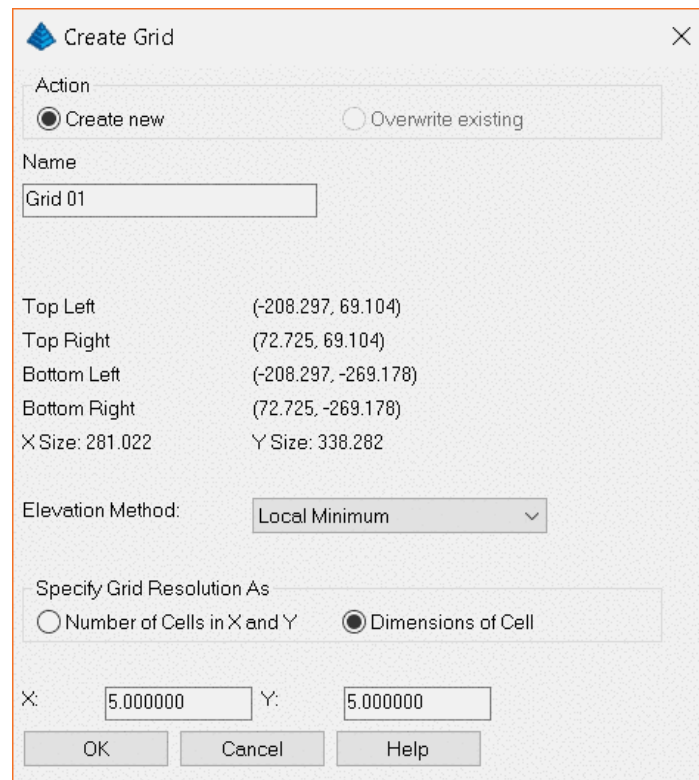
Prerequisite: An open scene of a point cloud

Create Grid

Selecting the **Grid** button from the Create panel in the **Action Tab** will open the **Select Grid Dimensions** window. Users can define the desired Grid manually onscreen, by using the boundary extent of the current scene or by selecting a Grid file (.grd).



To define the extent of the Grid onscreen, click **Screen Pick Grid Bounds**. The Scene will default to a Top-down view. Click once to pick the first corner, move the mouse to expand the box and click again to pick the other corner. The **Create Grid** window will now be displayed.



Name

The default name of the new grid is controlled through **Settings** on the **Project** panel. Statistical information about the selected area will be shown below the name field. For best results work in Plan view with the X axis pointing to the right and the Y axis pointing up.

Elevation Method

Elevation Method:

Local Maximum - Uses the maximum elevation within a grid cell as the cell elevation.

Local Minimum - Uses the minimum elevation within a cell as the cell elevation.

Cell Average - Uses the average of the cell elevation as the elevation for the cell.

Specify Grid Resolution As

Specify Grid Resolution As

Number of Cells in X and Y Dimensions of Cell

X: Y:

The grid Resolution may be specified in either the number of cells in the X and Y direction or by the size of each cell. After selecting a method you may adjust the values for X and Y. The total number of cells that will be created is shown below the X and Y values.

Note: If the size of your cell is less than the distance between points in your cloud you will have holes or void areas in your grid. If this occurs you can select the same area and create a grid with larger cell dimensions or a fewer number of cells.

To end the Grid Creation mode click the End Mode button at the top of the Action tab.

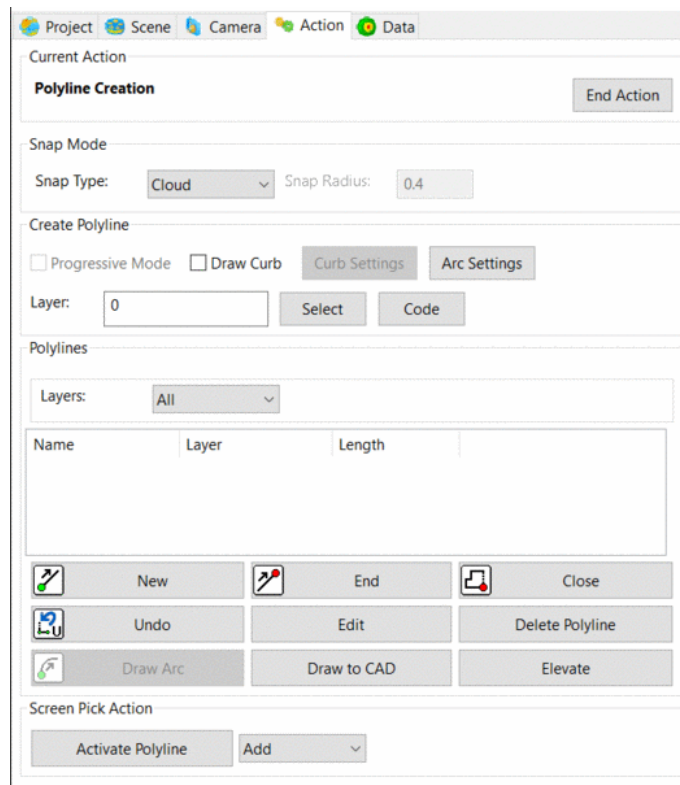
Tab Location(s): Action Tab

Panel and Button: Create and Grid

Prerequisite: An open scene of a point cloud

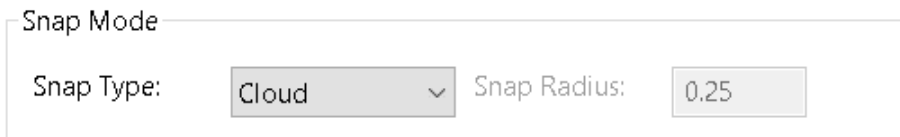
Create Polyline

Selecting the **Polyline** button from the **Create** panel in the **Action** tab will open the **Polyline Creation** dialog.



Snap Mode Panel

The Snap Mode Panel offers twelve different options for snapping to points in the cloud in the open scene.

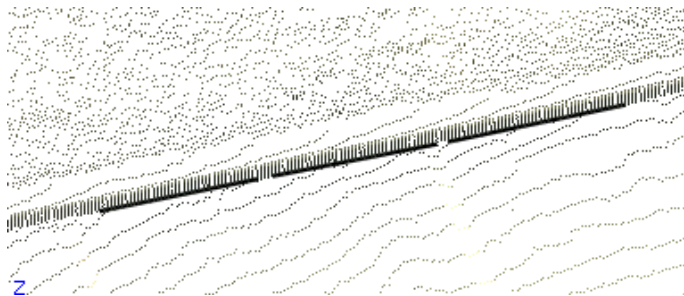


Cloud - No snap function is active. The vertex of the polyline will be placed at the cloud point nearest to the selected location.

Low Point - The polyline vertex is placed at the lowest point (smallest z value) within the Snap Radius.

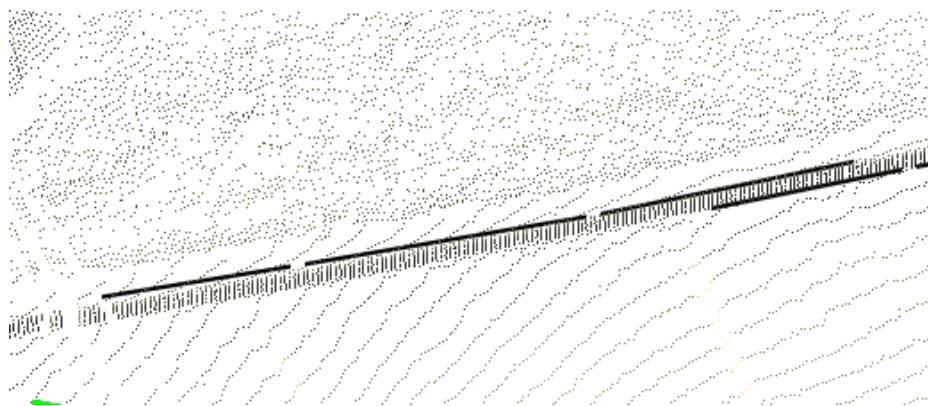
High Point - The polyline vertex is placed at the highest point (largest z value) within the Snap Radius.

Low Edge - Snaps to the low edge of a feature. A dynamic window in the upper left of the scene displays a cross section of the area within the Snap Radius and displays the high and low edge as red squares.



High Edge - Snaps to the high edge of a feature. A dynamic window in the upper left of the scene displays a cross

section of the area within the Snap Radius and displays the high and low edge as red squares.



Slope Bottom - Snaps to the bottom edge of a slope that is less than 45 degrees. This is useful for finding low edges on mountable curbs and other non-vertical features.

Slope Top - Snaps to the top edge of a slope that is less than 45 degrees. This is useful for finding upper edges on mountable curbs and other non-vertical features.

Average Point - This snap averages Northing, Easting and Elevation for all points within the Snap Radius and uses the averaged values for the coordinate for the new polyline vertex.

End Point - Snaps the new polyline vertex to the endpoint of existing polylines.

Mid Point - Snaps the new polyline vertex to the midpoint of existing polylines.

Node - Snaps the new polyline vertex to points placed in the drawing using Carlson Point Cloud.

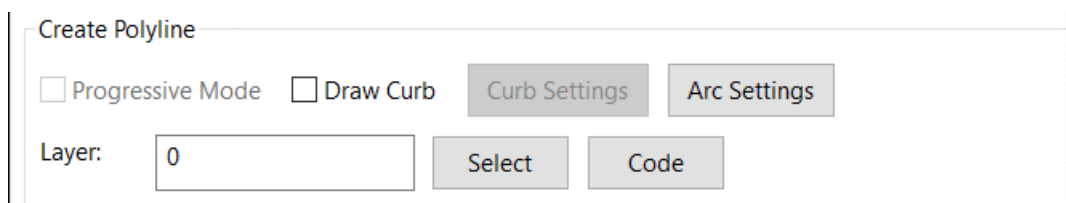
Nearest - Snaps the new polyline vertex to the point on an existing polyline nearest to the cursor location.

Color - Snaps the new polyline vertex to the selected color.

High/Low Edges - Snaps to the high and low edges of a vertical curb or wall simultaneously drawing two polylines.

Create Polyline Panel

The Create Polyline Panel sets options for creating polylines from the Point Cloud. Each option is discussed below.



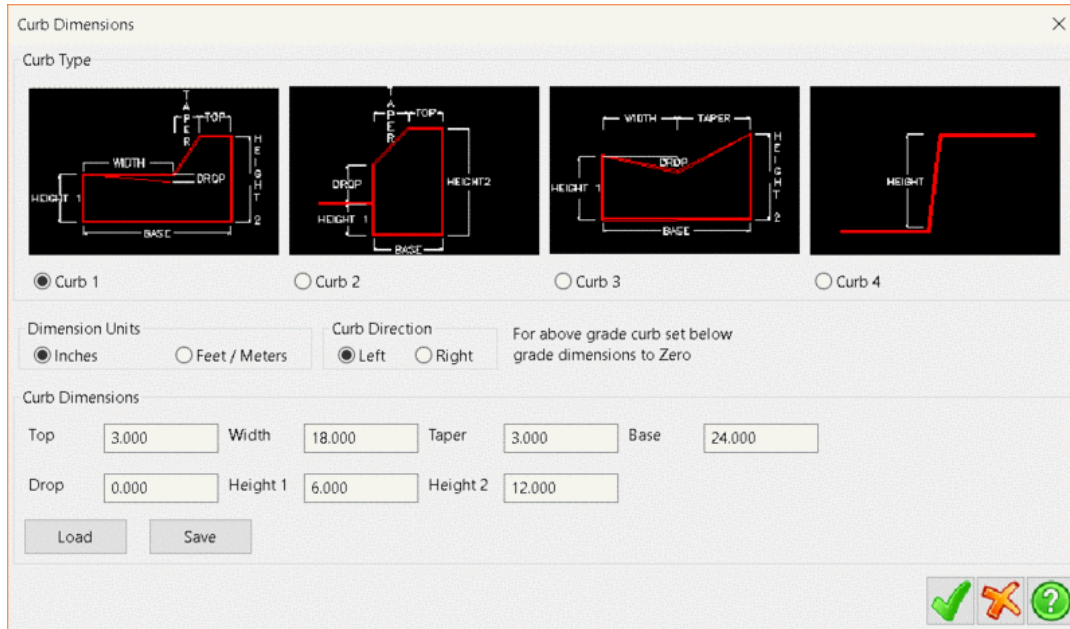
Progressive Mode - Note: Progressive mode can only be used when the Snap Type is set to High Edge or Low Edge. Progressive Mode creates a best fit polyline between the two selected points by breaking the distance into smaller segments with length equal to the snap radius. A point is found at each new segment location and a best fit line created through the resulting points.

Polyline Layer - Users can specify the layer to draw the new polyline on by typing a name in the Polyline layer field. Users may also click the Select button and select any layer currently in the scene or they may create a new layer to be used by typing it at the bottom of the Select Polyline layer dialog box.

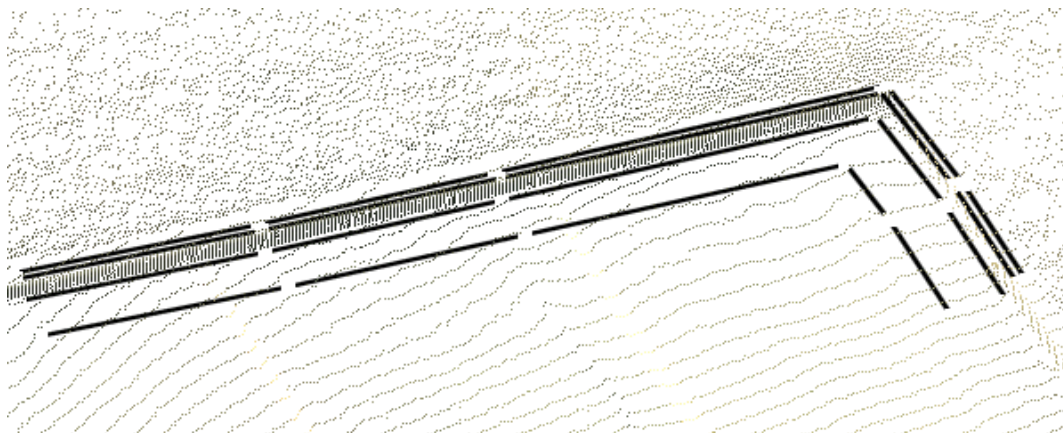
Draw Curb - The draw curb toggle when selected activates the Curb Settings Button

Curb Settings window

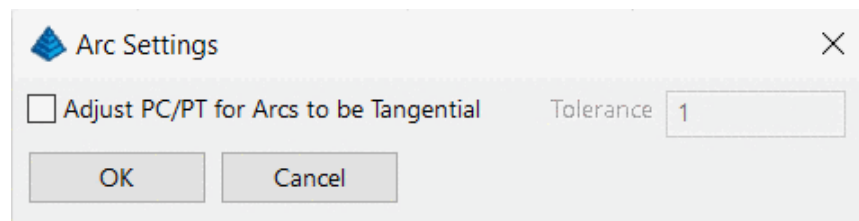
There are four curb types to choose from by clicking the radio button below the desired type. The dimensions for each curb can be specified using either feet or inches. The curb direction is relative to the direction you select the curb points in.



The polyline created from the picked points will be named like any other polyline. The additional polylines created from the draw curb routine will include the name of the main polyline and a suffix number. Below is an example of polylines created using Curb Type 1.

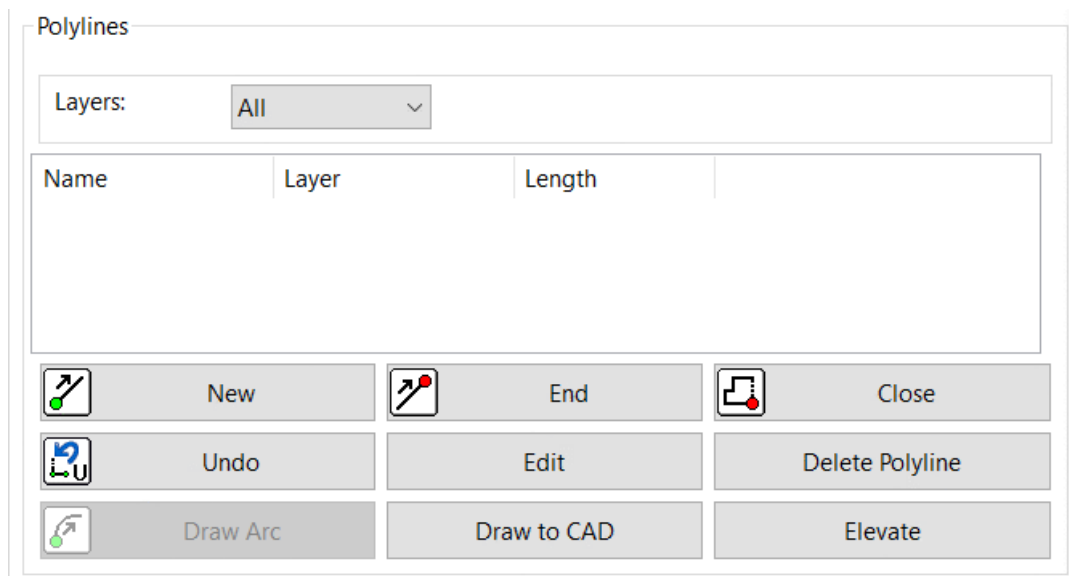


Arc Settings



Allows the creation of polylines with tangential arc. If selected, a tolerance can be set.

Polyline Control Panel



New - Starts a new polyline

End - Ends the polyline that is highlighted in the list of Active Polylines

Close - Closes the current polyline to its starting point

Undo - Removes the last polyline segment drawn and allows the user to continue the polyline creation from the previous endpoint.

Edit - Opens the Polyline Editor dialog box. The Polyline Editor dialog box can also be accessed by double clicking a polyline in the **Current Project** tree.

Delete Polyline - Deletes the selected polyline

Draw Arc - Draws a three point arc in the current polyline using the last point selected as the PC. The user is prompted to specify second point on arc. A line will display for the PC to the POC. After selecting the second point the user is prompted to specify end point of arc. The arc is drawn and the line from the PC to POC is gone.

Draw to CAD - Draws the active polylines into the current CAD file.

Elevate - elevates the selected polyline to:

Cloud

Reference Cloud

ground and road - Cleaned 2

Minimum point spacing: 10.000000

Point Within: 0.250000

Elevation Method: Lowest

Enable Simplification

Window: 8.000000

Threshold: 0.100000

Enable Smoothing

Window: 8.000000

Factor: 0.250000

OK Cancel

Reference Cloud - Specifies the cloud to use for elevating the polyline.

Minimum point spacing - sets the spacing to add vertices along the polyline as it intersects the point cloud

Point Within - specifies the radius of that the command will consider for the snap selection

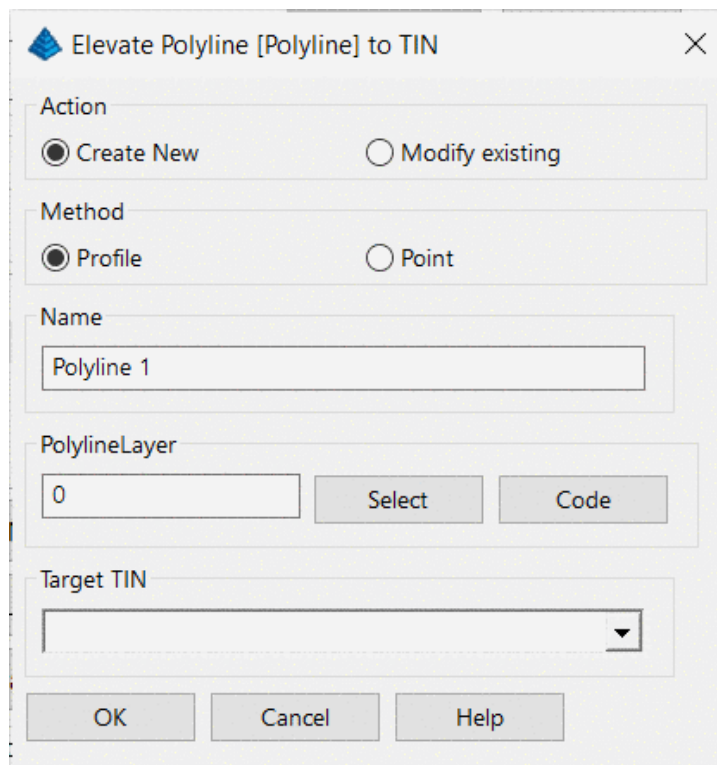
Elevation Method - sets the snap to one of four options for the elevation

- Lowest
- Highest
- Average
- Median

Enable Simplification - Sets controls to simplify the polyline as it is elevated to reduce the amount of vertices

Enable Smoothing - Sets controls to smooth the polyline as it is elevated.

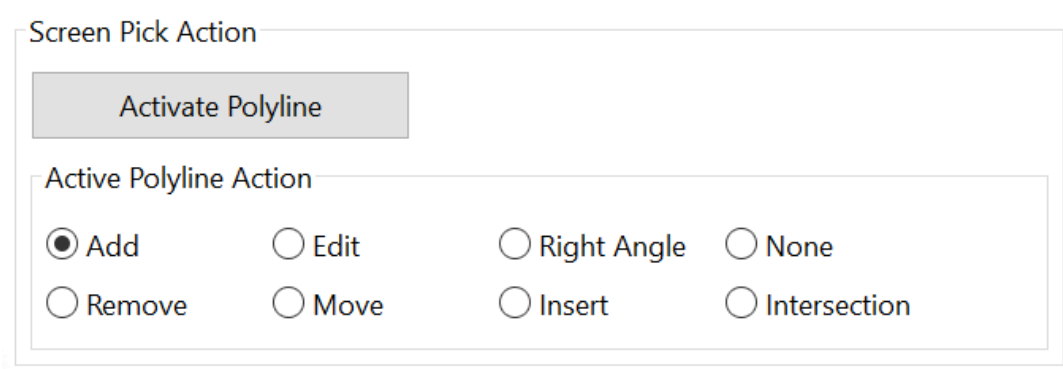
TIN



Can **Create new** or **Modifying existing** polyline using either the **Profile** or **Point** method.
Target TIN - choose the target TIN file

Screen Pick Action

The **Screen Pick Action** Panel provides options for editing polylines in the active scene. Each option is discussed below.



Activate Polyline - Allows the user to select the polyline to activate for the editing process. The Activate button can be used to make a polyline previously drawn active again so you can add on to that polyline. The user is prompted to select the polyline to activate. Multiple polylines can be activated. Click the End Activate button before trying to work with one of the activated polylines.

Add - appends (or continues) the currently selected Polyline.

Keyboard Shortcuts:

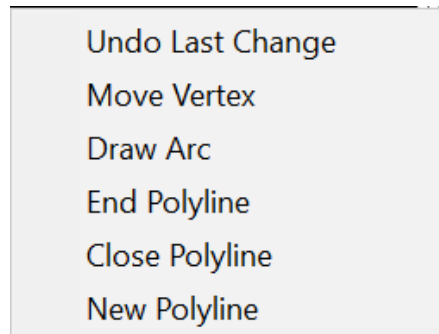
a Start point selection for arc drawing (Same as "Draw Arc" button).

c Make currently active polyline closed and deactivate currently active polyline (same as "Close" button).

e Deactivate currently active polyline (same as "End" button).

ctrl+z Undo most recent change to currently active polyline (same as "Undo" button). Note: This will only work when picking points for a polyline.

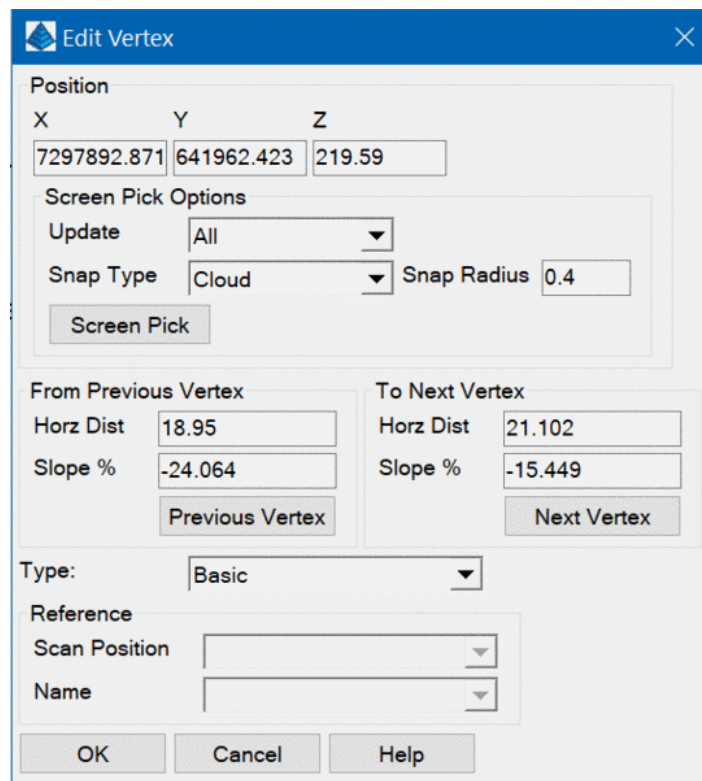
Right-Click Options:



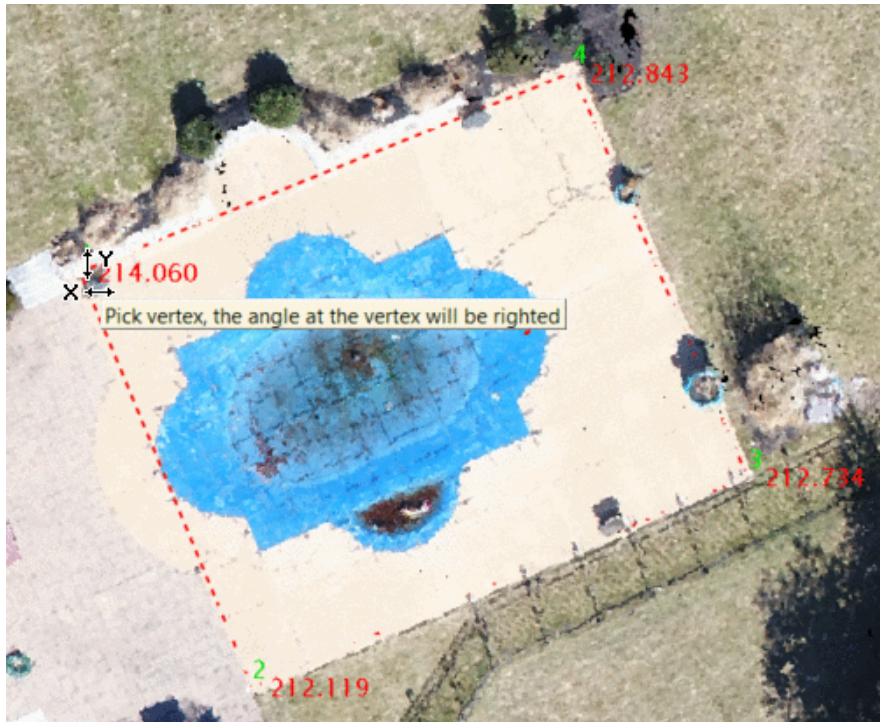
New in 2024 is the ability to right-click while the Polyline Creation is active to choose a range of different options, listed above.

Edit - The user selects the polyline to be edited by clicking on it in the current Scene. After selecting a polyline the user is prompted to pick point on polyline to edit by clicking on the point. The dialog box below is presented to the user.

The user can change the coordinates by typing in the X, Y or Z fields or by clicking on a new position in the current Scene (click the Screen Pick button). There are buttons to advance to the Next vertex or return to the previous vertex along the polyline. The user can also change the point type from Basic to Coordinate Point reference.



Right Angle - Forces a vertex to be at a 90 degree angle from two Polyline segments. The user selects the vertex and the command will create a right angle at that point in the polyline



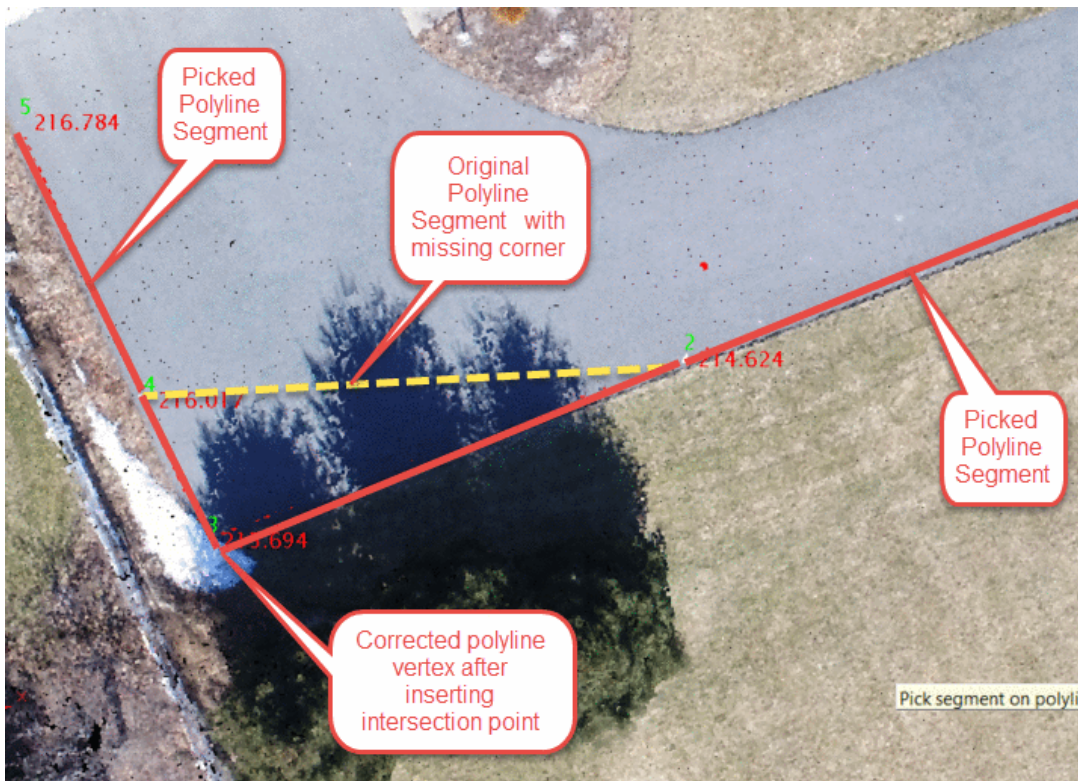
None - sets the Edit Mode to none to prevent unintended edits being made

Remove - Removes selected point from polyline

Move - Allows the user to select a vertex and move it to a new position

Insert - Inserts a new point into the polyline

Intersection - creates a vertex at the point of intersection of two polyline segments and modifies the polyline with the new location.



Tab Location: Action

Panel and Button: Create Polyline

Prerequisite: Open Point Cloud Scene

Create Text

Selecting the **Text** button from the **Create** panel in the **Action** tab will open the **Text Creation** dialog.

Current Action

Text Creation End Action

Snap Mode

Snap Type: Cloud Snap Radius: 0.25

Create Text

Text Leader Segments None

Text Size Alignment Top Left

Snap Mode

The Snap Mode Panel offers twelve different options for snapping to points in the cloud in the open scene.

Snap Mode

Snap Type: Snap Radius:

- **Cloud** - No snap function is active. The text will be placed at the cloud point nearest to the selected location.
- **Low** - The text is placed at the lowest point (smallest z value) within the Snap Radius.
- **High** - The text is placed at the highest point (largest z value) within the Snap Radius.
- **Low Edge** - Snaps to the low edge of a feature. A dynamic window in the upper left of the scene displays a cross section of the area within the Snap Radius and displays the high and low edge as red squares.
- **High Edge** - Snaps to the high edge of a feature. A dynamic window in the upper left of the scene displays a cross section of the area within the Snap Radius and displays the high and low edge as red squares.
- **Slope Bottom** - Snaps to the bottom edge of a slope that is less than 45 degrees. This is useful for finding low edges on mountable curbs and other non-vertical features.
- **Slope Top** - Snaps to the top edge of a slope that is less than 45 degrees. This is useful for finding upper edges on mountable curbs and other non-vertical features.
- **Average Point** - This snap averages Northing, Easting and Elevation for all points within the Snap Radius and uses the averaged values for the coordinate for the text.
- **End Point** - Snaps the text to the endpoint of existing polylines.
- **Mid Point** - Snaps the text to the midpoint of existing polylines.
- **Node** - Snaps the text to points placed in the drawing using Carlson Point Cloud.
- **Nearest** - Snaps the text to the point on an existing polyline nearest to the cursor location.

Create Text Panel

The Create Text Panel sets options for creating text in the current Scene. Each option is discussed below.

Create Text

Text Leader Segments

Text Size Alignment

Text - This is the text that will be placed in the scene. The text will be a single line.

Leader Segments - This value specifies the number of segments a leader will have. The text is placed at the segment endpoint.

Text Size - text size is a fixed unit. This means that as the user zooms in and out the text is always the same size on the screen and not relative to the cloud.

Alignment - There are nine alignment options for the text relative to the text location picked.

- Top Left
- Top Center
- Top Right
- Middle Left
- Middle Center
- Middle Right
- Bottom Left
- Bottom Center
- Bottom Right

Tab Location: Action

Panel and Button: Create and Text

Prerequisite: Open Point Cloud Scene

Extract Breaklines

PointCloud has an automatic breakline extracting function to help facilitate the processing of a dataset.

Breakline extraction is available by clicking the **Breaklines** button under the **Extract** panel in the **Action** tab.

In order to extract breaklines, a TIN must be open in a Scene and a user should first select the area from which to extract the breaklines using a **Selection** button.

Note: For best results, it is recommended to view the TIN in a Scene using the **Color** settings of **Category**: Position and **Type**: Normal.

Click the **Breaklines** button a new panel should be displayed at the bottom of the Action Page which displays all the options for configuring PointCloud's breakline extraction utility. Additionally, the viewer window will remove the coloring of the current selection to prevent it from interfering with the zone flag specification process, the viewer still internally keeps track of the area of the TIN that you have selected.

The screenshot shows the 'Breakline Extraction' panel with the following settings:

- Current Action:** Breakline Extraction (Buttons: Show, Extract, End Action)
- Direction - Color Zones:** X - R, Y - G, Z - B, Color (with a 3D coordinate system icon)
- Active Directions - Colors:** Use X - Red, Use Y - Green, Use Z - Blue
- Extraction Method:** By Vertex, By Face
- Vertex Method Settings:** Use Local Smoothing
- Join Nearest:** Enable, Maximum Distance: 1,000
- Minimum Breakline Length:** Enable, Min Length: 1,000
- Polyline Simplification:** Enable, Window: 8, Factor: 0.10
- Polyline Smoothing:** Enable, Window: 8, Factor: 0.25

To add color zones, *Ctrl + Click* in the 3d view window on the TIN at a location that has the color/normal you want to define your zone. This will add the color to the Color Zones table, and you can manually modify the value if it isn't quite what you want it to be. You can choose which directions you want to be considered when zone classification is being performed using the **Use X**, **Use Y**, and **Use Z** check boxes. Restricting directions can yield better results in certain situations. For instance, if you're trying to extract edges along a curb, it would be best to turn off **Use X** and **Use Y** and only use the **Z** direction of the normal to determine the zone, due to the fact that curbs are usually a corner with a vertical surface and a horizontal surface.

There are two methods of breakline extraction: **By Vertex** and **By Face**.

Extraction Method by Vertex

Note: This method is the default method and generally gives better results in datasets that have hard edges (such as corners of buildings).

Extraction Method

By Vertex By Face

Vertex Method Settings

Use Local Smoothing

Join Nearest

Enable Maximum Distance:

Minimum Breakline Length

Enable Min Length:

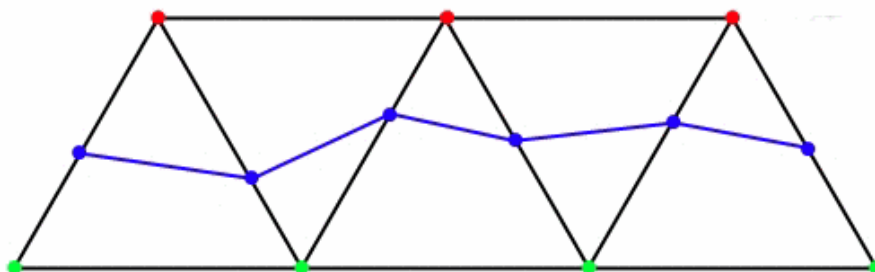
Polyline Simplification

Enable Window: Factor:

Polyline Smoothing

Enable Window: Factor:

In this method, each vertex of the current selection is assigned a zone based on the normal at that vertex. PointCloud then creates new polyline vertices on each edge that connects vertices belonging to different zones. These polyline vertices are positioned based on how the vertex normal's compare to the normal that defines its zone.



Simple example of the Vertex Method

The Vertex method has one unique setting, **Use Local Smoothing**, which averages the normal at each vertex with the normals of its neighbors to provide a smoother set of normals to work with. This helps to give more continuous breakline edges by accommodating for places in the TIN that may be distorted due to missing or outlying data.

Extraction Method by Face

To extract breaklines using the Face Method, select the **By Face** radio button under the **Extraction Method** panel.

Extraction Method	
<input type="radio"/> By Vertex	<input checked="" type="radio"/> By Face
Zone Minimum Size	
Minimum Area:	Minimum Number of Triangles:
<input type="text" value="0.000"/>	<input type="text" value="1"/>
Join Nearest	
<input type="checkbox"/> Enable	Maximum Distance: <input type="text" value="1.000"/>
Minimum Breakline Length	
<input checked="" type="checkbox"/> Enable	Min Length: <input type="text" value="1.000"/>
Polyline Simplification	
<input checked="" type="checkbox"/> Enable	Window: <input type="text" value="8"/> Factor: <input type="text" value="0.10"/>
Polyline Smoothing	
<input checked="" type="checkbox"/> Enable	Window: <input type="text" value="8"/> Factor: <input type="text" value="0.25"/>

In this method, each face of the current selection is assigned a zone based on its normal. Polylines are created from the edges that touch faces belonging to a different zone. The face method has unique options of minimum area and triangle count values for each zone area. This is to prevent a common problem where a single or a small number of faces near the border will be classified as belonging to a zone other than the one they currently reside in, creating an extremely small zone.

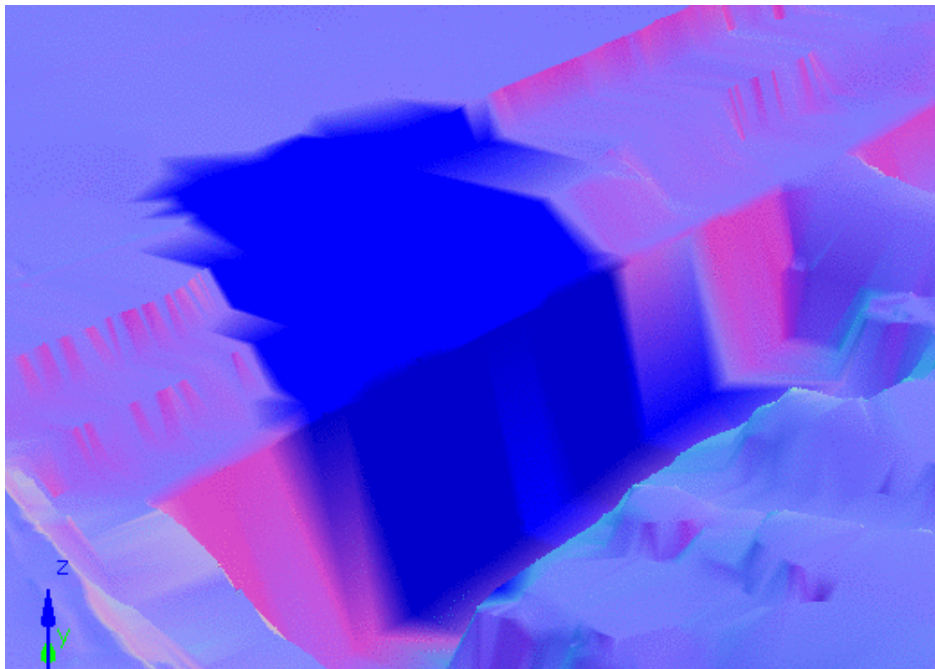
The **Join Nearest** option will join any polylines whose endpoints are within the threshold distance of each other. This can help to bridge gaps in the TIN that could arise from parts of a scan being in shadow.

The **Polyline Simplification** option will remove each vertex that is within a certain distance to the least squares approximation. Window is the number of neighbor vertices to use when generating the least squares approximation. Threshold is a positive number that determines the distance under which vertices will be removed from the polyline.

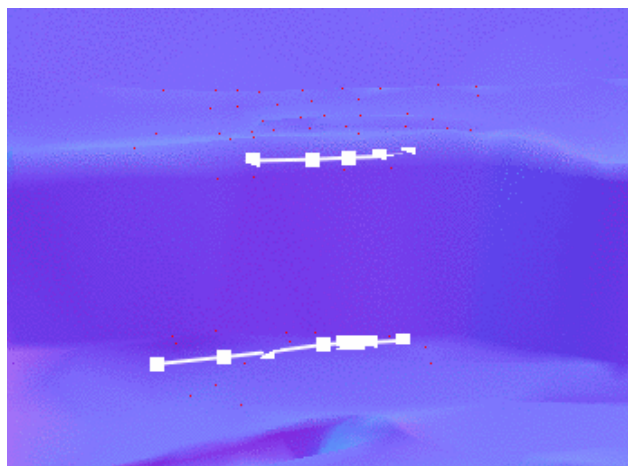
The **Polyline Smoothing** option will smooth each vertex in the polyline using a least squares approximation. Window is the number of neighbor vertices to use when generating the least squares approximation. Factor is a number between 0.0 and 1.0, 0 being no smoothing performed and 1 being fitting each vertex to it's respective least squares line (usually makes the polyline extremely linear given a window size greater than 8).

Pressing the **Show** button at the top of the **Breakline Extraction** page will only show the zones in the viewer given the current configuration, but will not create the polyline at this stage. **Show**

This gives you a chance to review your settings and make sure that the zones are configured properly without adding any extra objects to the project.



Pressing **Extract** will extract the breaklines with the given settings. Each breakline polyline will be added to the project with the a name generated from the current project settings.



Prompts

Specify Zone Flag Position Ctrl-Click in the viewer to select the color zone for breakline extraction

Tab Location(s): Action Tab

Panel and Button: Extract and Breaklines

Prerequisite: Open scene of a TIN with color by Normal

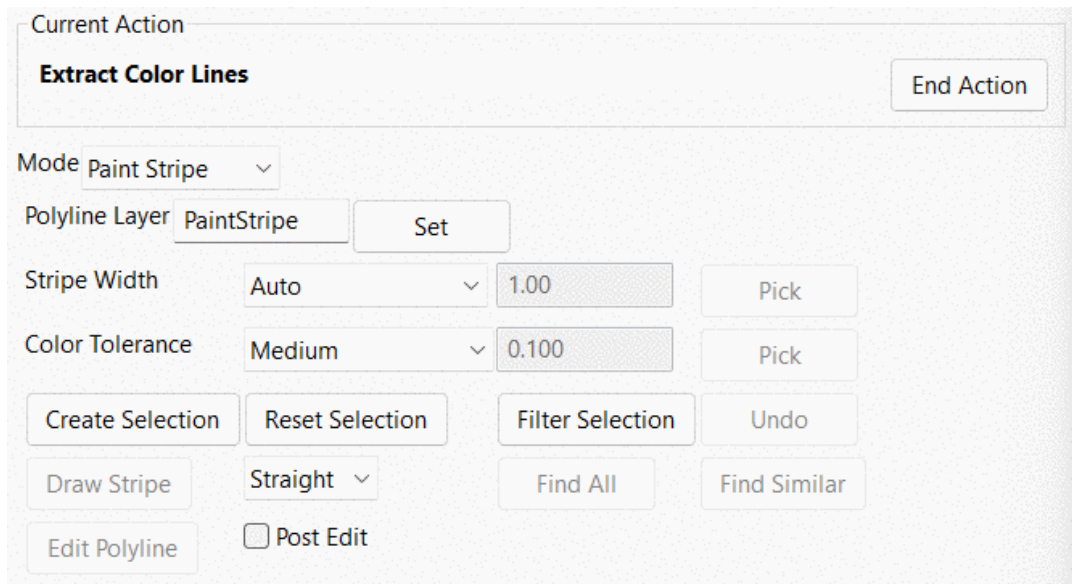
Extract Color Lines

Clicking the **Extract Color Lines** button in the **Extract** panel of the **Action Tab** will put the current mode into Extract Color Lines. There are 4 options to choose from, listed below.

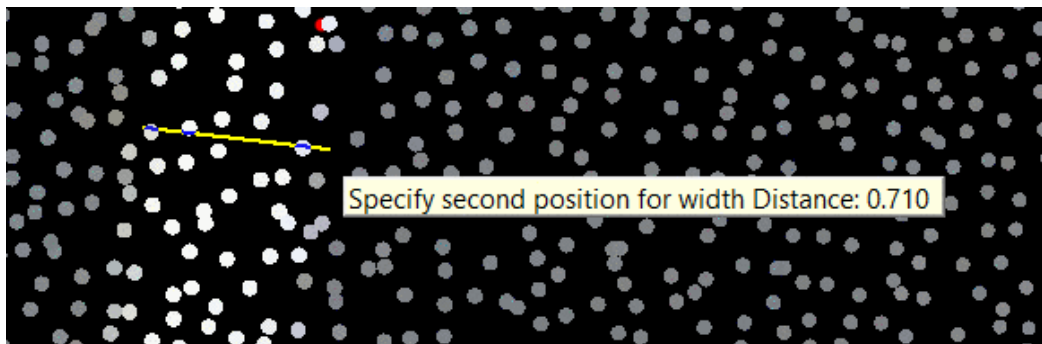
Note - for point clouds that have a high density, the time taken to process using these features could be quite considerable so it is advised that the cloud is **Cleaned** prior to running these routines. For most clouds, the **Duplicated Points Distance Threshold** could be set to 0.04 (which is approx. 0.5 inches) or if using a metric cloud, set the value to 0.012.

Paint Stripe

Used to extract painted lines from a cloud.



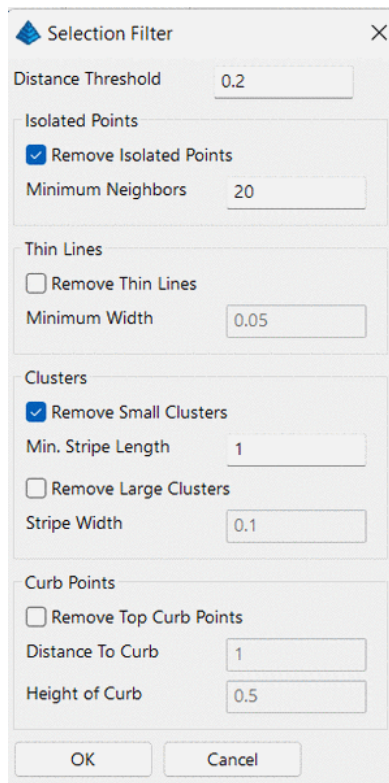
Polyline Layer sets the layer that the newly created data will go into. Click on **Set** to bring up the layer manager. **Strip Width** - setting to **Auto** is best used when using the **Draw Stripe**. Setting this to **Custom** will allow the user to enter in an approximate width of the lines they would like to select. By clicking on **Pick** to the right of this, it allows the user to choose two points in the cloud to populate the Stripe Width value



Color Tolerance will set the color tolerance of the points to be selected. Click on **Pick** to choose two points in the cloud to set this value.

Create Selection chooses the points in the cloud. Click on this then the tooltip in the scene will state "Pick a point to sample color". Once a color has been selected, it will select all of the same colored points in the cloud.

Filter Selection allows the user to filter out the points.



Distance Threshold is the distance between the points in the cloud. This will cluster all the points together. Try to go slightly larger here. For example, if the points are 0.1 to 0.2 apart, set this to 0.3.

Remove Isolated Points will remove any points that are less than the set **Minimum Neighbors**. That is, any cluster of points that contain less than this value won't be selected.

Remove Thin Lines will remove any lines that have a width of less than the value nominated here

Remove Small Clusters - by setting the **Min Stripe Length**, it will remove any clusters that are less than this value. For example, if the lines are parking lines, they're going to be quite long so can set this value to something near the average length.

Remove Large Clusters will do the opposite of the **Remove Small Clusters** - set the **Stripe Width** to the width of the stripe; be conservative on this as this will remove any clusters that are larger in width than this.

Curb Points will remove Selected Points that are a Curb Height higher (specified by the **Height of Curb** parameter) than other points that are less than the **Distance To Curb** away. The idea is that paint stripes will not go over a curb in the majority of cases so if there are some points selected that are on top of a curb then this filter will remove them.

Once the desired filters have been set, click on **OK**. This will remove the points using the filters that were originally selected using the **Create Selection** routine.

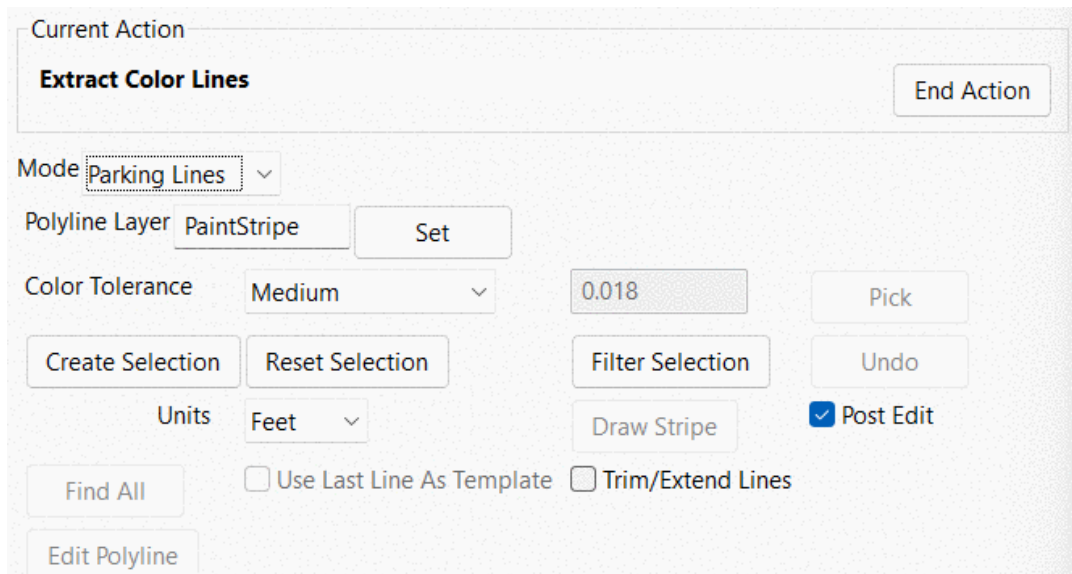
Now that the points have been selected, the option **Draw Stripe** will be available. Set **Stripe Width** to **Auto** then click on a section of straight line in the scene. It should then draw a polyline of this stripe. Repeat for others. For any curved lines, change the stripe type to **Curved** then click a point on the curve (not too close to an intersection of a straight and the curved line).

Find All is an automated process to find and create polylines for any paint stripe using the **Stripe Width** variable; it would be worth setting this to **Manual** and either entering in the width or use the Pick option to set the width.

Edit Polyline allows the user to modify polylines created using this routine.

Parking Lines

Specialized routine for extracting parking stripes. Some of the options are the same that appear in **Paint Stripe**. This routine works best when there is a clear distinction between the parking lines color and the color of the surrounding surface.



Units - this is critical. By choosing **Feet**, it looks for parking lines that are about 18 feet long, which is the typical length of parking lines. The **Metric** option does the same but in meters.

To start the routine, click on **Create Selection** then click on the parking lines in the scene. Multiple points can be selected if necessary. If there is a good distinction between the parking lines and the road surface, the **Tolerance** of **Medium** should work fine. Alternatively, change the value to another per-configured setting or choose **Custom** then click on **Pick** and select 2 points that best show the parking lines to set the tolerance value.

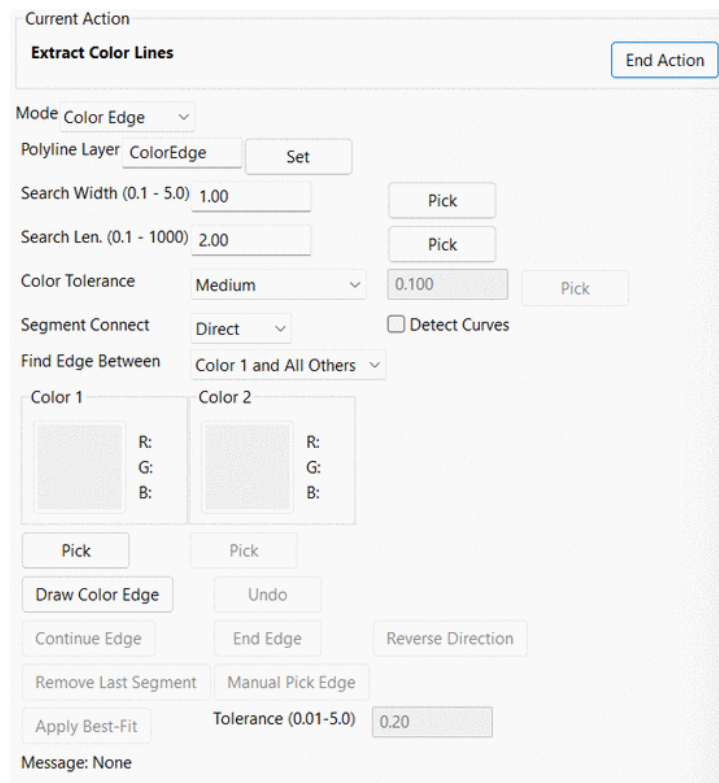
If the routine picks some points that are not parking lines (i.e. an object that is close to the color of the lines) this is OK. Click on **Find All** to draw the polyline for each parking line. The Command History will show the lines being created then the total number of Stripes found.

Note: this routine will normally draw two-point lines if the terrain is flat. However, if there are some bumps in the terrain, the lines will actually have a few points in them so that better follow the curvature of the ground.

Draw Stripe is used when the parking stripe lines are not the pre-configured 18' (or 5.5m) long. First, run **Create Selection** and choose the parking stripes. Next, choose **Draw Stripe** and click anywhere on a well-defined parking stripe. If the user is satisfied with the selection, tick the option **Use Last Line As Template** then click on **Find All**.

Color Edge

This routine extracts an edge of an object. For example, between a roadway and the grass verge. Some of the options are the same that appear in **Paint Stripe**.



Search Width is the width across the two different features in which the routine will examine the points. Users can either enter in a width or click on **Pick** and pick a point on either side of the edge.

Search Length is the length of the search area that the routine will examine. Clicking on **Pick** will allow this value to be populated via picking on two points in the scene.

Note: If either of these values are too small, and not enough points are being examined, the software will warn the user about this.

Segment Connect is how the segments will be joined; **direct, intersect** or **rounded**.

Detect Curves will try and detect the edge at an angle other than the original line.

Find Edge Between - two options: **Color 1 and All Others** allows the user to pick 1 color then tries to find an edge between that color and all other colors while **Color 1 and Color 2** allows the user to pick two colors to compare.

Draw Color Edge runs the routine based on the parameters entered in. Click on a point on the edge and it will create a starting point for the line. Click a second point on the edge to get the line started. The routine will run for as long as it can based upon the values entered.

Continue Edge can be used while within the **Draw Color Edge** routine to continue a line. If the line does not continue as far as it could, the colors may need to be selected again then continue the line work.

Reverse Direction will start the routine looking in the opposite direction to the initial starting point.

Apply Best-Fit will apply a best fit line through the segments. If it does not work, use **Undo Best-Fit** and adjust the tolerance.

Color Polyline

The **Extract Polyline** command "traces" a new Polyline from points within a Cloud, based on color and intensity. Polylines are extracted by holding down the CTRL key and clicking the line in the cloud to extract. Line sets are parallel polylines of equal length such as parking stripes. Line sets can be created by holding down the ALT key while clicking the lines to be extracted to the line set. The polyline length and direction are based on the first line extracted.

Current Action
Polyline Extraction End Action

Evaluation radius:

Chromaticity variance threshold(0-255):

Intensity variance threshold(0-100):

Maximum gap allowed:

Straight line Dynamic base color

Number of intermediate lines:

Undo
Edit Vertex

Polyline Layer: Select

Use CTRL-click to create a polyline
 Use ALT-click to create a line set

Evaluation Radius: specifies the radial distance to use to locate points to evaluate against the remaining criteria.

Chromaticity variance threshold (0-255): specifies how much the color can vary with 0 being an exact match (this almost never occurs) and 255 turning off the color match. The default value of 2 is a good starting value.

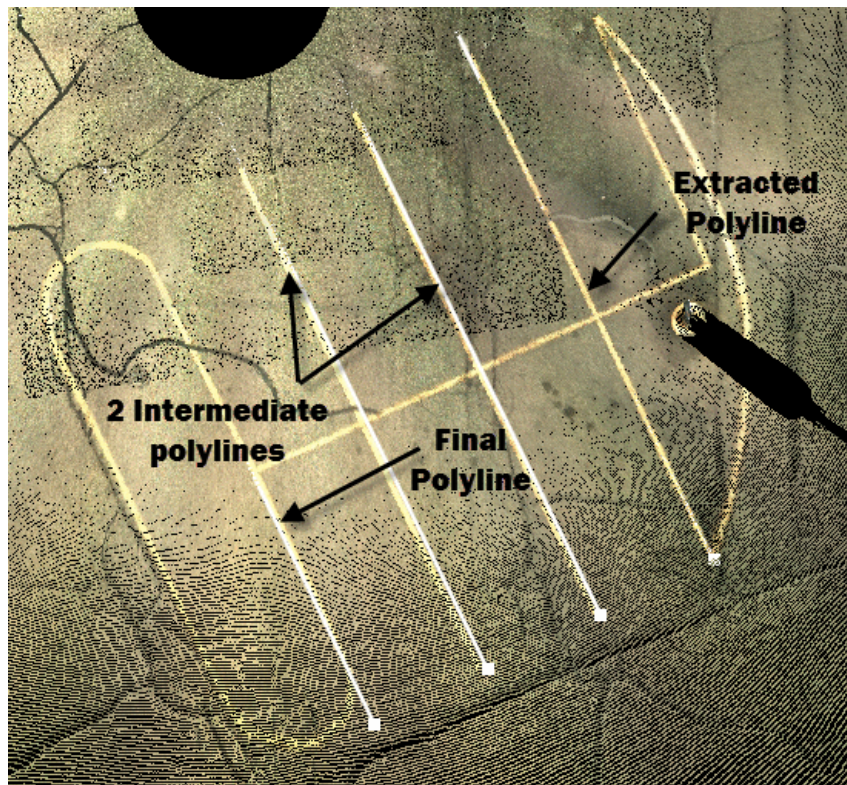
Intensity variance threshold (0-100): specifies how much the intensity value can vary from point to point. Zero (0) would be an exact match and 100 would turn off the intensity match. The default value of 3 is a good starting value.

Maximum gap allowed: Applies only to straight lines. The Maximum Gap Allowed is the maximum gap distance that a straight line may have. The value will depend on the data set being used.

Straight line: This option should be toggled on when extracting straight lines. Points are tested for variation based on the point clicked.

Dynamic base color: When toggled on this option will test points for variation based on the last point deemed to be on the line. This option is more useful when extracting curves.

Number of intermediate lines: This option is strictly for straight line extraction. To activate hold down the ALT key and left click when in the Polyline Extraction mode. The program will use the last line extracted as a guide. Polylines are then created parallel to the last polyline extracted. The new polylines are the same length as the last polyline extracted. The new polylines are placed equidistant from each other and a final polyline is placed on the point selected with the ALT-Click.



Undo will undo only the last command used.

Edit Vertex edits the vertices of the last polyline extracted. Use the CTRL or ALT key in combination with left-click to pick a new location for a vertex. The vertex nearest the location selected will be moved. In some cases you may need to select intermediate locations to move the desired vertex.

Polyline Layer specifies the layer to draw the extracted polyline on. A layer name may be enter directly in the edit box or a layer may be selected from the layer list by click the **Select** button.

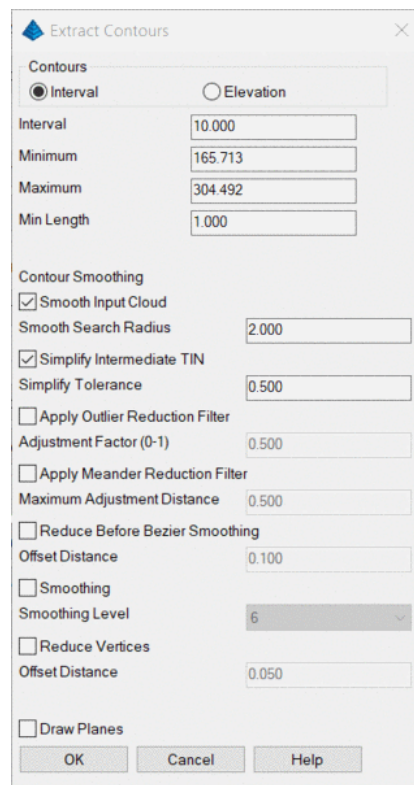
Tab Location(s): Action Tab

Panel and Button: Extract

Prerequisite: Open scene of a scan, cloud or TIN

Extract Contours

The **Extract Contours** window can be reached by clicking the **Contours** button in the **Extract** panel of the **Action Tab**, or by selecting **Create Contours** from the menu bar in a Scene Window. The only requirement for contour extraction is that the data, either a point cloud, a TIN, Solid or a scan is currently opened in a Scene.



The **Interval** extraction method extracts contours at every multiple of the specified interval between the minimum and maximum elevation values (a minimum of 2.5 and an interval of 5.0 would have the first contour elevation be at 5.0).

The **Elevation** method only extracts the contours at the specified elevation value.

By default, the **Minimum** and **Maximum** elevation values are the minimum and maximum elevations of the current scene.

Min Length determines the minimum length of a contour line to be drawn. Upon extraction, contours will be given an automatically generated name and added to the **Contours** folder in the project tree.

Contour Smoothing offers an option to smooth the cloud data prior to creating the contours. The Smooth Radius specifies how far from each point the program will look for neighbors. If a point does not have at least 3 neighbors within that radius, it will not be included in the algorithm.

Simplify Intermediate TIN removes spikes which might cause small circles or jagged edges in contours. The Simplify tolerance is the maximum adjustment distance a point will move to be closer to the same level as its neighbors.

Apply Outlier Reduction Filter reduces spikes in contours from outlier points.

Apply Meander Reduction Filter - leaving this at 0.5 default is often a reasonable choice to assist with contour smoothing but may be adjusted based upon trial and error with the data set.

Reduce Before Bezier Smoothing works in a manner similar to reduce vertices using its own Offset Distance

Smoothing (1-10) - Using the drop-down menu, users can lower the Smoothing setting which will have less looping or less freedom to curve between contour line points. Likewise, increasing the smoothing increases the looping effect. Note: too much smoothing applied in some situations can result in crossing contours. This setting affects the contour lines themselves, not the underlying data.

Reduce Vertices removes vertices from the contour polylines provided the removed vertex is not further away from the new line than the Offset Distance

If **Draw planes** is enabled, plane objects will be added to the project and the scene that cut through the scene at each extracted elevation level. The planes will be horizontal and extend to the limits of the cloud. While this can be useful for a single elevation contour that is extracted, it creates many planes when used with an interval extraction.

Tab Location: Action

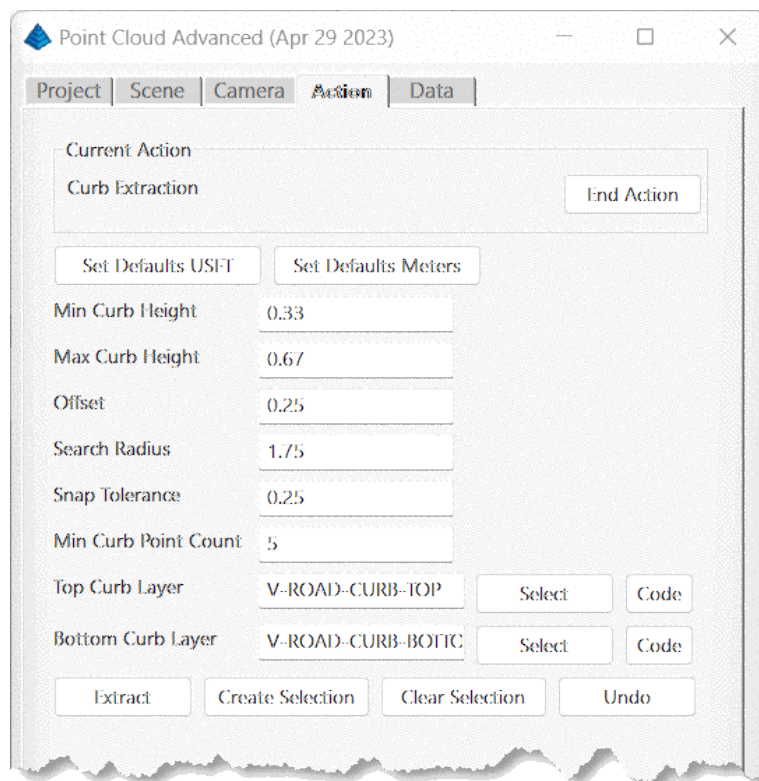
Panel and Button: Extract and Contours

Prerequisite: Open Point Cloud Scene of a scan, cloud or TIN

Extract Curb

The Extract Curb command can automatically generate top of curb and flowline polylines along the curb edges from point cloud data generated by a ground or aerial LiDAR or by photogrammetric processes. The algorithm uses edge clustering based on typical curb heights to find top of curb and bottom of curb points to find and draw the edges.

From the **Action tab**, click **Curb** under the **Extract** panel. Enter the minimum and maximum curb height parameters, flowline offset, search radius, snap tolerance, and minimum curb point count values. Click the **Create Selection** button to select the point cloud points along the curb area. Note that the points will be selected along the reference polyline drawn using the Search Radius as left and right offset range. Click **Extract** to begin the polyline extraction.



Set Defaults USFT: sets all the parameters to default values in feet.

Set Defaults Meters: sets all the parameters to default values in meters.

Min Curb Height: sets a minimum height for points to be used to determine the curb edge.

Max Curb Height: sets a maximum height for points to be used to determine the curb edge.

Offset: sets an offset range to find the bottom of curb edge.

Search Radius: sets the offset distance from the reference polyline when creating a selection set when Create Selection is used.

Snap Tolerance: specifies the radius for the program to consider for the elevation snap setting.

Min Curb Point Count: a low value helps resolve the curb edges better along curves. Click **Select** to choose from existing layers in the project. Click **Code** to select a layer based on field code table codes.

Top Curb Layer: sets the layer to draw on the top of curb polyline. Click **Select** to choose from existing layers in the project. Click **Code** to select a layer based on field code table codes.

Bottom Curb Layer: sets the layer to draw on the bottom of curb polyline. Click **Select** to choose from existing layers in the project. Click **Code** to select a layer based on field code table codes.

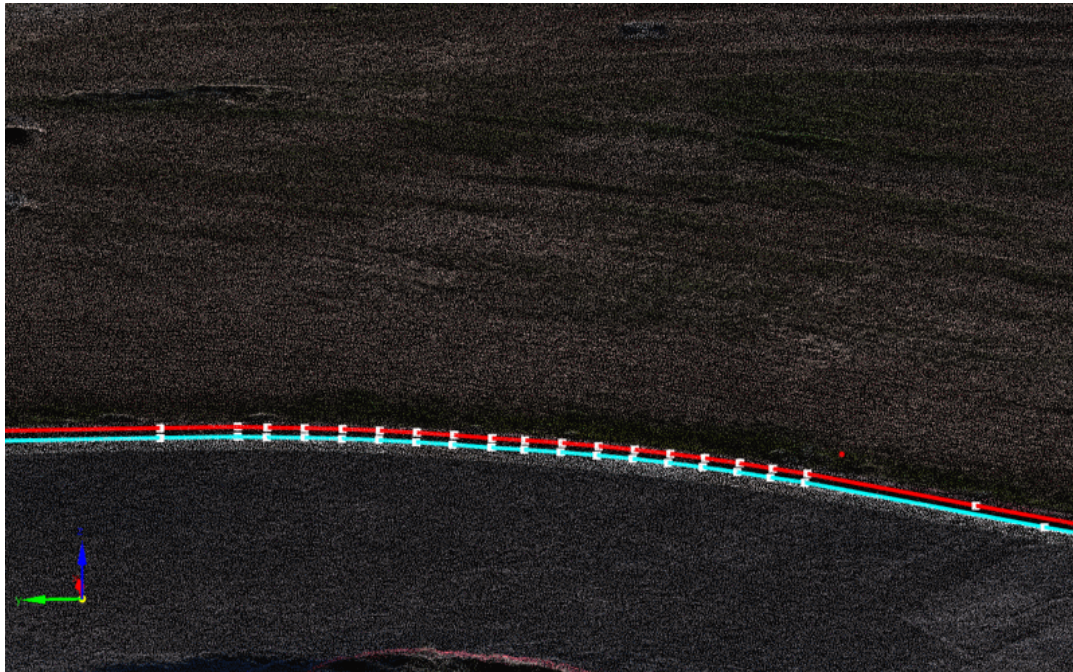
Extract: begins the extraction process.

Create Selection: starts a point cloud point selection mode. Click on the cloud and a reference polyline will be drawn. Points will be selected within an offset range along the polyline based on the Search Radius distance.

Clear Selection: clears the selection set.

Undo: will remove the polylines created by the last curb extraction.

Using the values shown above, below you can see an example of an extracted curb. The point cloud was generated by Carlson PhotoCapture from a drone flight with a ground sampling distance (GSD) of 0.03 ft. If you are working with ground scanner data, it is recommended that you decimate the point cloud first to reduce the number of point cloud points by using any of the methods found in the Clean or Resample filters.



Tab Location: Action tab.

Panel and Button: Extract and Curb.

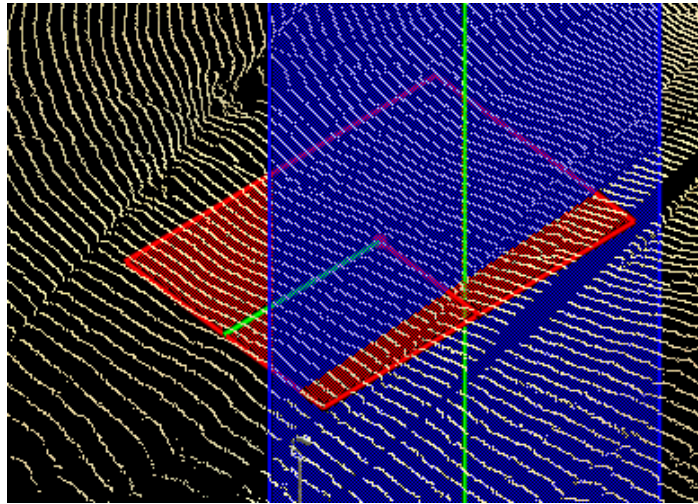
Prerequisite: Point Cloud Scene open.

Extract Intersection

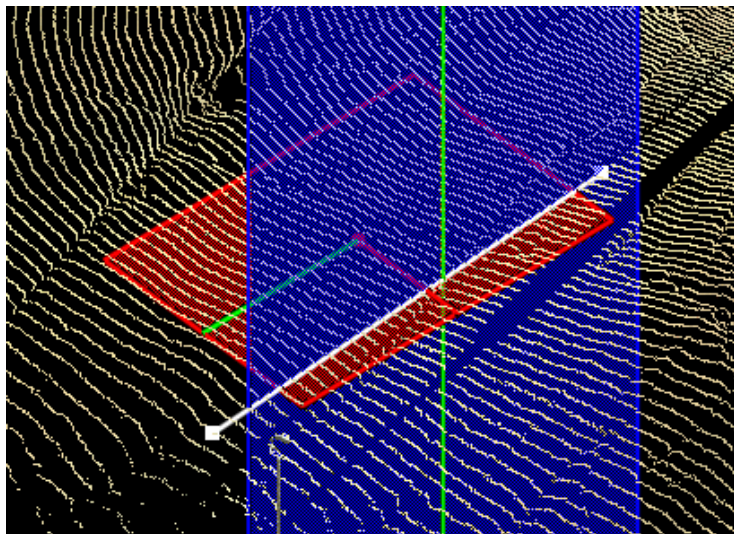
Extract Intersection creates a polyline based on the intersection of two selected planes. Planes can be drawn when extracting Planes, Contours, Profiles or Sections.

To extract intersections, click on the **Intersection** button under the **Extract** panel in the **Action** tab. There are two options to select the planes to be used to extract the intersection.

Use CTRL-click to select a plane Use this option to select the first plane to use to extract an intersection.



Use ALT-click to intersect two planes Use this option to select the planes to intersect with the plane selected using option one. You can also select the first plane using this option. Once the first plane is selected using either method, multiple intersecting planes may be selected using the ALT-click method.



Note:

- The 3D polyline(s) of the intersection is added to the **Project Tab**.

Tab Location(s): Action Tab

Panel and Button: Extract and Intersection

Prerequisite: Open scene with two or more planes

Extract Plane

Clicking the **Plane** button in the **Extract** panel of the **Action Tab** will put the current mode into Plane Extraction mode.

Current Action
Plane Extraction End Action

Evaluation radius:

Standard deviation threshold:

Create shrink-wrap:

Plane points:

Use CTRL-click to create a plane
Use ALT-click to expand the last plane

Note: Planes can be extracted in any rotation of the scene. The scene may be of a Cloud or Scan.

The size of the plane is controlled by two parameters.

Evaluation radius: the distance around each point that PointCloud will search for other points that meet the Standard Deviation Threshold.

Standard deviation threshold: how far a point can be away from the plane for the plane to be considered flat. A value of about 20% of the Evaluation Radius is typical.

The **Create shrink-wrap** droplist has three options.

Full: creates a 3D polyline that encloses the points used to create the plane.

Bottom: creates a 3D polyline

None: No polyline is created. This is the default option.

The **Plane points** droplist has three options.

Select: Adds the points used to create the plane to the active selection set. The points are highlighted in red. This is the default option.

Hide: removes the points used to create the plane from visibility in the scene.

Leave: The plane is created without highlighting or hiding the points used.

The **Undo** option undoes the last function performed only. For example, creating a plane having selected the **Hide** option, Undo would restore the points to being visible and the plane would still be in the Scene. Undo only performs one operation, it cannot undo multiple steps.

After these options have been set to the preferred values, the user can extract the plane by left clicking in the viewer while holding down the CTRL key. The plane is displayed in the active Scene and can be drawn in CAD as a closed 3D polyline from the **Project** tab.

Prompts

Use CTRL-click to create a plane Left-click in the viewer while holding down the CTRL key to create the plane.

Use ALT-click to expand the last plane Left-click in the viewer while holding down the ALT key to expand the last plane created.

Tab Location(s): Action Tab

Panel and Button: Extract and Plane

Prerequisite: Open scene of a scan or cloud

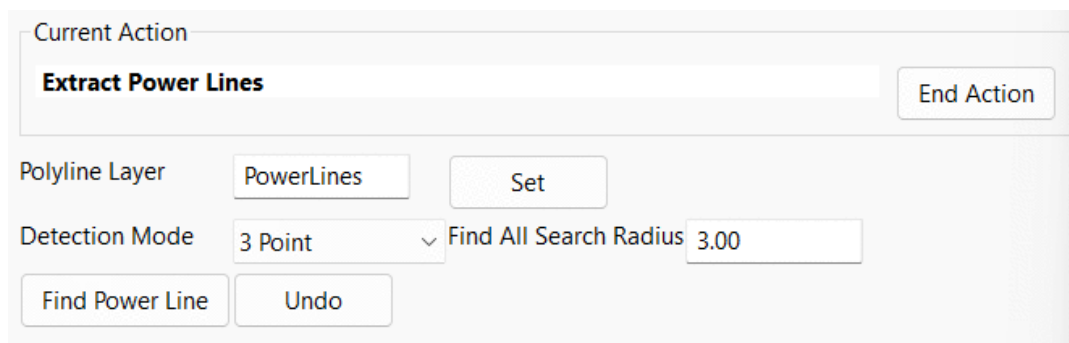
Extract Power Lines

Clicking the **Extract Power Lines** button in the **Extract** panel of the **Action Tab** will put the current mode into Extract Power Lines.

Notes:

1. For point clouds that have a high density, the time taken to process using these features could be quite considerable so it is advised that the cloud is **Cleaned** prior to running these routines. For most clouds, the **Duplicated Points Distance Threshold** could be set to 0.04 (which is approx. 0.5 inches) or if using a metric cloud, set the value to 0.012.

2. This routine works best when working with Non-ground points. To quickly do this, use the **Extract Bare Earth** routine. Under the **Extract Destination** heading, choose **Non-Ground Cloud**. This will reduce the number of points in the cloud making it easier to work with.



Polyline Layer sets the layer that the newly created data will go into. Click on **Set** to bring up the layer manager.

Detection Mode sets the type of routine used to extract the power line.

3 Point uses more manual entry, but is generally more reliable.

1. Specify the wire position on the first pole
2. Specify a point on the wire in between the poles (can be anywhere along the wire, but works best if a point in the middle is chosen)
3. Specify a point on the second pole

A polyline should now be generated that follows the curvature of the power line.

1 Point allows a user to click on a point in the middle of the line. This routine looks in both directions and it can find the power line on its own with just one point. Try and pick a point in the middle that has a good amount of data for the algorithm to look for. This works best where there is a lot of data on the lines and no holes in the data.

1 Point Find All is the same as **1 Point** mode except it uses the search radius parameter to try to identify several power lines all at once. **Find all Search Radius** needs to be large enough to span in between the power lines.

Tab Location(s): Action Tab

Panel and Button: Extract Power Lines

Prerequisite: Open scene of a scan or cloud

Extract Profile

The **Profile Extraction** functionality can be activated by going to the **Action Tab** and clicking the **Profile** button in the **Extract** panel. The only requirement for profile extraction is that the data, either a point cloud, or a TIN is being viewed in a Scene. Pressing the **Extract Profile** button also puts the current viewer into profile extraction mode. A tooltip with instructions on how to extract the profile will display when the cursor is in the scene. Left clicking in the viewer will establish the profile location. To exit profile extraction mode, click the **End Mode** button.

Current Action
Profile Extraction End Action

Extract Profile
 Real-World Camera View

Polyline Simplification
 Enable Window: 8 Threshold: 0.10

Polyline Smoothing
 Enable Window: 8 Factor: 0.25

Planes
 Draw planes

Extract Along Centerline

Profiles can be extracted in two kinds of coordinate systems: **Real-World** and **Camera View**.

Top-Down is the most commonly used method and extracts the profile in the current scene's coordinate system. The z values, or elevation values, are those found in the surface the profile is being extracted from.

Camera View is much less commonly used and uses the current camera as the origin instead of the scene's origin, and elevation values are effectively distances from the camera. This mode can be useful if you're trying to extract a profile out of a vertical surface such as a wall or are trying to get the profile of an object from a direction other than the z axis. **Camera View** mode gives the best results if the current camera is in parallel projection mode (perspective mode will distort the profile).

Interval specifies the spacing between points to be added to the profile. The **Point Within** option specifies the search radius for the Elevation Method which is essentially a cloud Osnap function. The **Elevation Method** can be set to Lowest, Highest Average or Median elevations found within the specified *Point Within* radius.

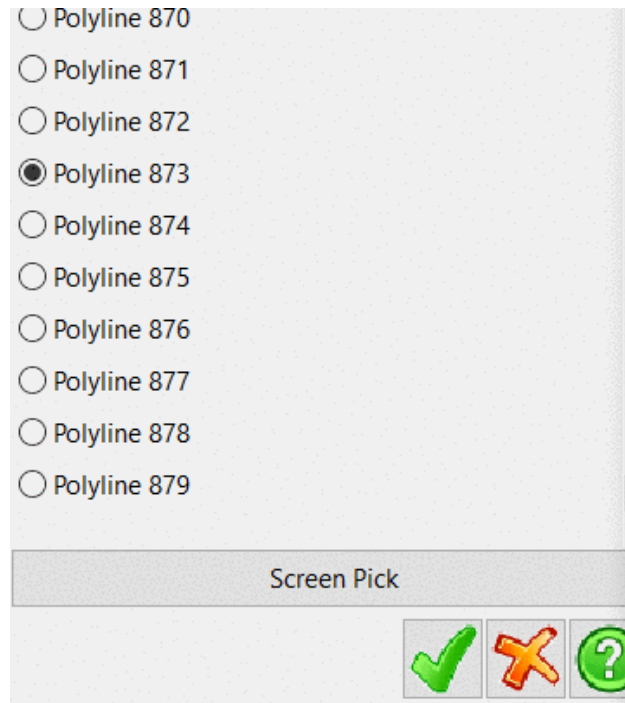
The **Profile Simplification** option will remove each vertex that is within a certain distance, **Threshold**, to its least squares approximation. Window is the number of neighbor vertices to use when generating the least squares approximation. Threshold is a positive number that determines the distance under which vertices will be removed from the polyline.

The **Profile Smoothing** option will smooth each vertex in the polyline using a least squares approximation. Window is the number of neighbor vertices to use when generating the least squares approximation. Factor is a number between 0.0 and 1.0, 0.0 being no smoothing performed and 1.0 being fitting each vertex to its respective least squares line (usually makes the polyline extremely linear given a window size greater than 8).

If **Draw Planes** is enabled, plane objects that follow the path of the initial selection polyline will be drawn into the viewer after the profile is extracted. The planes will extend vertically to the low and high elevation matching the limits of the Bounding box. Each plane drawn will also be added under the Planes heading in the Project Tree.

Extract Along Centerline when clicked the user is prompted for the .CL file to use. The horizontal location of the centerline is extracted from the Cloud or TIN.

Extract Along Polyline when clicked the user is prompted to select an existing polyline from the project or Screen Pick a polyline



After extracting a profile by polyline or .CL file, a dialog will open asking for an action to perform with the extracted profile, this gives you a chance to review the profile and the option to save it to the project or save it to a file.

Current Action
Polyline Extraction End Action

Evaluation radius:

Chromaticity variance threshold(0-255):

Intensity variance threshold(0-100):

Maximum gap allowed:

Straight line Dynamic base color

Number of intermediate lines:

Undo Edit Vertex

Polyline Layer: Select

Use CTRL-click to create a polyline
Use ALT-click to create a line set

Evaluation Radius: specifies the radial distance to use to locate points to evaluate against the remaining criteria.

Chromaticity variance threshold (0-255): specifies how much the color can vary with 0 being an exact match (this almost never occurs) and 255 turning off the color match. The default value of 2 is a good starting value.

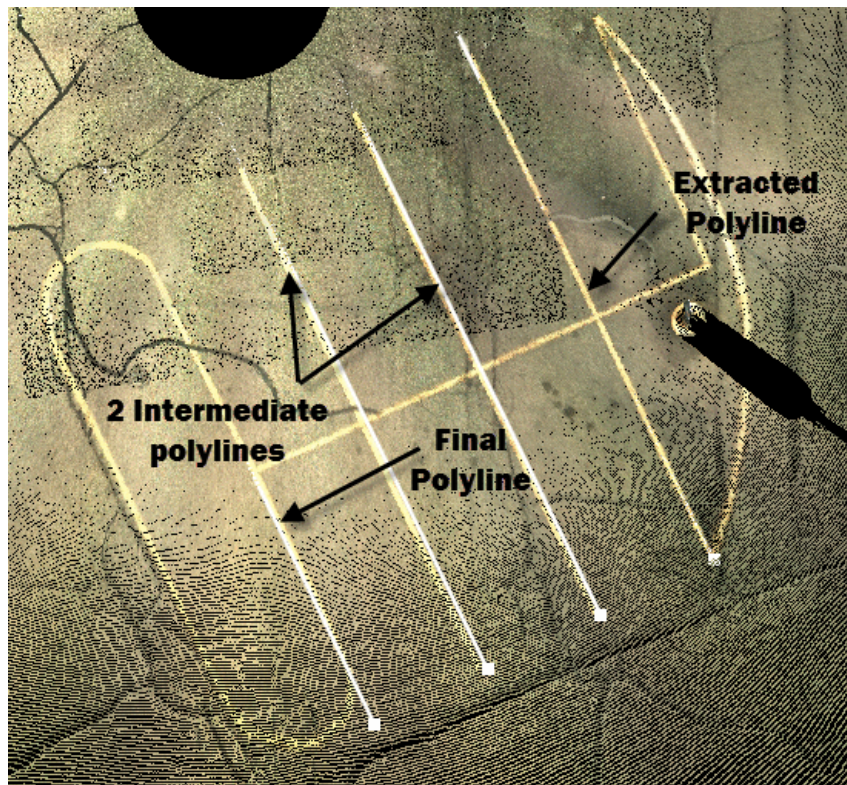
Intensity variance threshold (0-100): specifies how much the intensity value can vary from point to point. Zero (0) would be an exact match and 100 would turn off the intensity match. The default value of 3 is a good starting value.

Maximum gap allowed: Applies only to straight lines. The Maximum Gap Allowed is the maximum gap distance that a straight line may have. The value will depend on the data set being used.

Straight line: This option should be toggled on when extracting straight lines. Points are tested for variation based on the point clicked.

Dynamic base color: When toggled on this option will test points for variation based on the last point deemed to be on the line. This option is more useful when extracting curves.

Number of intermediate lines: This option is strictly for straight line extraction. To activate hold down the ALT key and left click when in the Polyline Extraction mode. The program will use the last line extracted as a guide. Polylines are then created parallel to the last polyline extracted. The new polylines are the same length as the last polyline extracted. The new polylines are placed equidistant from each other and a final polyline is placed on the point selected with the ALT-Click.



Undo will undo only the last command used.

Edit Vertex edits the vertices of the last polyline extracted. Use the CTRL or ALT key in combination with left-click to pick a new location for a vertex. The vertex nearest the location selected will be moved. In some cases you may need to select intermediate locations to move the desired vertex.

Polyline Layer specifies the layer to draw the extracted polyline on. A layer name may be enter directly in the edit box or a layer may be selected from the layer list by click the **Select** button.

Tab Location(s): Action Tab

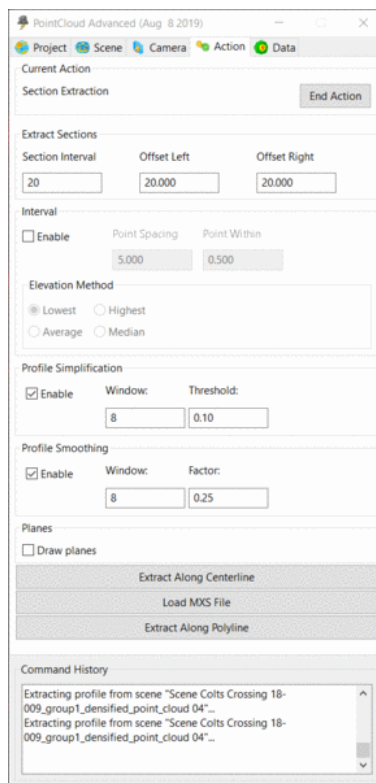
Panel and Button: Extract and Polyline

Prerequisite: Open scene of a cloud

Extract Sections

Extract Sections

Clicking the **Sections** button in the **Extract** panel of the **Action Tab** will put the current mode into Section Extraction mode.



The width of the sections and the frequency, or interval, at which they are extracted along the designated path are controlled by three parameters.

Left Offset is the distance left of the path the sections will extend. Left is determined based on the direction the extraction path was drawn or "up station".

Right Offset is the distance right of the path the sections will extend. Right is determined based on the direction the extraction path was drawn or "up station".

Interval is the value that determines the distance along the path between sections.

The **Elevation Method** sets a cloud "snap" value to specify which point elevations to use for each profile vertex. The **Point Spacing** and **Point Within** options control the search parameters for points to consider for the Elevation Method. The options are Lowest, Highest, Average and Median.

The **Profile Simplification** option will remove each vertex that is within a certain distance to its least squares approximation. Window is the number of neighbor vertices to use when generating the least squares approximation. Threshold is a positive number that determines the distance under which vertices will be removed from the polyline.

The **Profile Smoothing** option will smooth each vertex in the polyline using a least squares approximation. Window is the number of neighbor vertices to use when generating the least squares approximation. Factor is a number between 0.0 and 1.0, 0.0 being no smoothing performed and 1.0 being fitting each vertex to its respective least squares line (usually makes the polyline extremely linear given a window size greater than 8).

If **Draw Planes** is enabled, Plane objects will be added to the project and the scene that follow each section line. The planes will extend from the upper to the lower extents of the cloud. These limits are the same as used with the bounding box option in the viewer.

Extract Along Centerline when clicked the user is prompted for the .CL file to use. The horizontal location of the centerline is used and the sections are extracted perpendicular to the centerline.

Load MXS File loads a Multiple Cross Sections file. These can be created using the "Section Alignment Editor" command under the Sections menu in Carlson Civil.

Extract Along Polyline allows users to select an existing polyline from the project or Screen Select a polyline.

After these options have been set to the preferred values, the user can extract the sections by left clicking in the

viewer to draw the path polyline and right clicking to end it. The sections are saved to the project and can be drawn in CAD or exported to a .SCT file from the **Project Tab**.

Prompts

Left-click to create polyline Left-click in the viewer to create the polyline for the profile, Right-click to end the polyline.

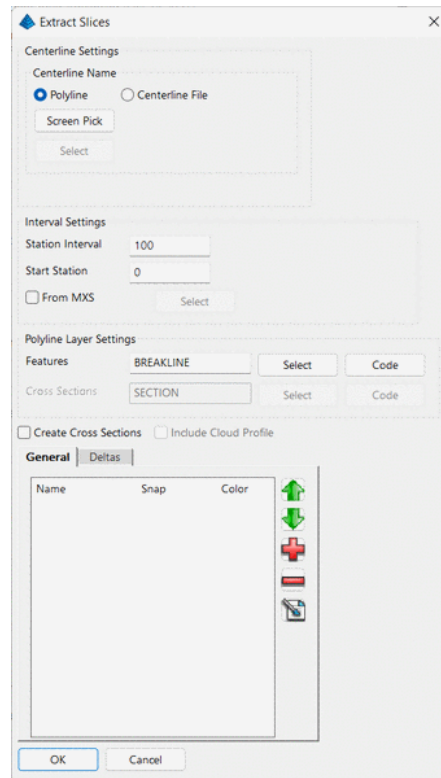
Tab Location(s): Action Tab

Panel and Button: Extract and Sections

Prerequisite: Open scene of a scan, cloud or TIN

Extract Slices

Clicking the **Extract Slices** button in the **Extract** panel of the **Action Tab** will put the current mode into Extract Slices.



Centerline Name has two options;

Polyline which uses an existing polyline in the scene via the **Screen Pick** option and

Centerline File which will load in an existing Carlson centerline (.CL) file

Station Interval is the space between each slice.

Start Station allows a starting station to be set.

Polyline Layer Settings sets the layer that the newly created data will go into. Click on **Select** to bring up the layer manager and **Code** to open the PointCloud Code table.

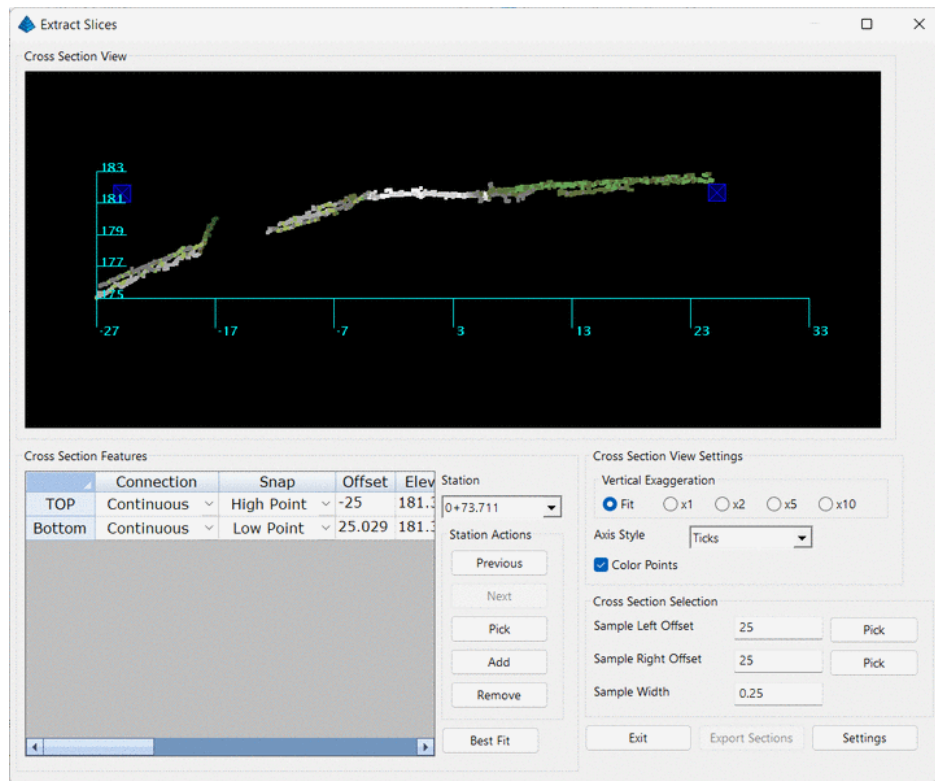
Create Cross Sections enabling this allows the Cross Sections layer to be set along with being able to set the accompanying code.

Include Cloud Profile includes the cloud points in the selection.

General is where Features can be entered in. Click on the red Plus icon to add.

Once OK is selected, the routine runs and the **Extract Slices** dialog box appears. In here, there are a number of options where the user can step through the slices to view data from the loaded scene. By using the **Cross Section Features** ongoing polylines can be created using the different Snap settings available.

The **Sample Offset** can be modified along with the **Sample Width**.



If the option **Create Cross Sections** was enabled, once the user exits out of the **Extract Slices** routine, the cross sections will be available to view in the Project Tree under the **Sections** heading. There is also the option **Export Sections** to save the sections as a Carlson xsct or sct file.

Tab Location(s): Action Tab

Panel and Button: Extract Slices

Prerequisite: Open scene of a scan or cloud

Extract Slope Edge

Clicking the **Extract Slope Edge** button in the **Extract** panel of the **Action Tab** will put the current mode into Extract Slope Edge.

Note: For point clouds that have a high density, the time taken to process using these features could be quite considerable so it is advised that the cloud is **Cleaned** prior to running these routines. For most clouds, the **Duplicated Points Distance Threshold** could be set to 0.04 (which is approx. 0.5 inches) or if using a metric cloud, set the value to 0.012. **Polyline Layer** sets the layer that the newly created data will go into. Click on **Set** to bring up the layer manager.

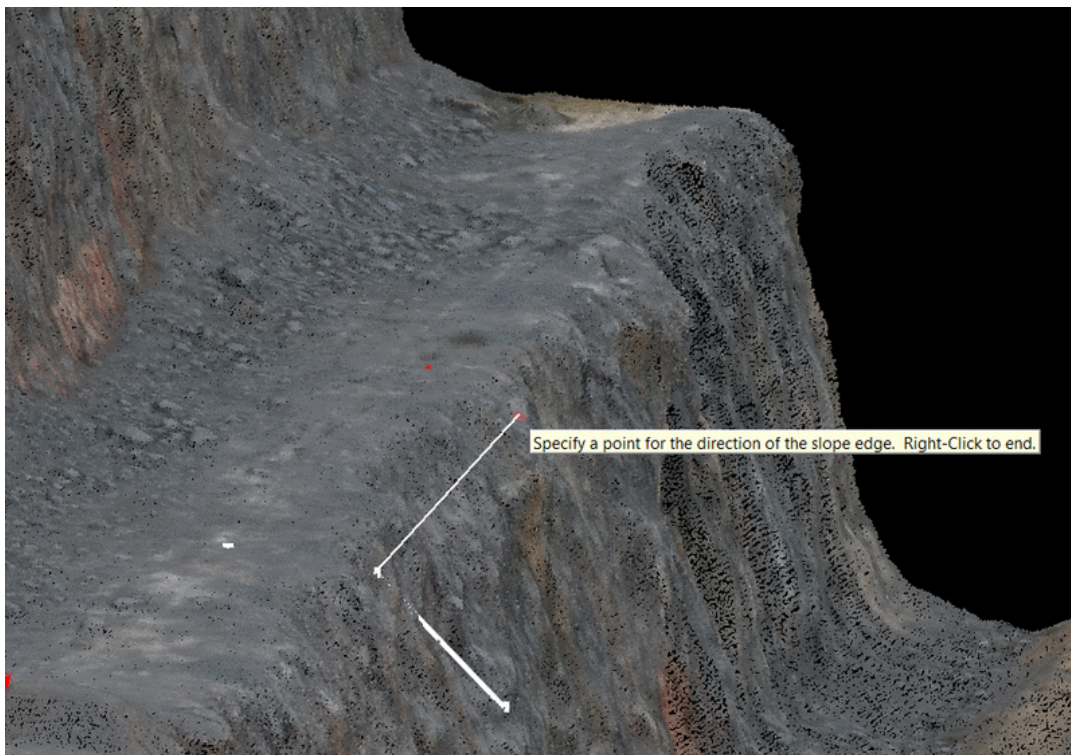
Segment Connect is how the segments will be joined; **direct**, **intersect** or **rounded**.

Search Len is the length at which the routine will search on each slope. When set to **Auto** it tries a wide range of search lengths and then tries to pick the best one to extract data. By setting it to **Custom** you can force the search length. Normally, **Auto** works well but it depends on the clarity of the edge.

Draw Slope Edge

1. The tooltip will ask to pick a point on the face of the 1st slope - pick a point that will contain a number of points for the routine to use
2. Pick a point on the edge of the two slopes - for example, the edge of a cliff
3. Pick a point on the face of the second slope, again ensuring that enough data is included
4. The lines need to be long enough so that the routine considers a decent amount of points along this Cliff edge; we're defining slope one and then slope two with each line respectively
5. Lastly, pick a second point along the edge of the Cliff just to indicate direction
6. It will now search for the edge between the two slopes

If it stops due to an object in the way (or goes off course) use the **Undo Last Segment** button to remove a number of segments then use **Continue Edge** and redefine the template further along.



Reverse Direction will continue the linework in the opposite direction from the initial starting point.

Apply Best Fit can create a line of best fit through the created segments. Once done, click **End Edge**.

If the profile of the slopes change too much, the routine will stop. If it goes around a curve, tick the option **detect curves**. Note: this option works best on well defined edges.

Tab Location(s): Action Tab

Panel and Button: Extract Slope Edge

Prerequisite: Open scene of a scan or cloud

Extract Volume

Extract Volume will let you calculate volumes from a point cloud selection set. Besides the volume report, you can also create triangulation files (TIN) for the resulting surface and the base elevation that can be used in other Carlson desktop programs including Civil, Takeoff, and Mining.

Before you start the Extract Volume action, use the Create Polyline action to draw a closed polyline representing the area where the volume will be calculated. Note that you can also import polylines from the CAD drawing if you have the desired boundaries.

In the **Action Tab**, under the **Extract** panel, **click Volume**.

Current Action
Volume Extraction End Action

Extract Volume

Keep Selection

Use Base Elevation Screen Pick

Save Base TIN View Base TIN

Save Scan TIN View Scan TIN

Extract Units

Feet Meters

Select incision polyline to begin

Keep Selection: When toggled, the program will keep the point cloud points selected after the volume has been calculated.

Use Base Elevation: Toggle it to enter a desired base elevation for the calculation. **Click** the **Screen Pick** button to select a point to set the base elevation.

Save Base TIN: By toggling it, the program will save the calculated base surface TIN both to the Current Project Tree, under TINs, and to the Carlson project directory.

View Base TIN: By toggling it, the program will add the base surface TIN to the current scene view.

Save Scan TIN: By toggling it, the program will save the calculated surface TIN both to the Current Project Tree, under TINs, and to the Carlson project directory.

View Scan TIN: By toggling it, the program will add the surface TIN to the current scene view.

Extract Units: Point cloud files are generally unit-less, use this setting to confirm the project units for the report.

To calculate the volume, **select** the **closed polyline** representing representing the outermost perimeter of the desired area. The program will highlight the points inside selected area and calculates the volume. If you **click Continue**, the **Report Formatter** window will open allowing you export the report to a text file, PDF, and insert a table in CAD.



Prompts

Left-click to select polyline

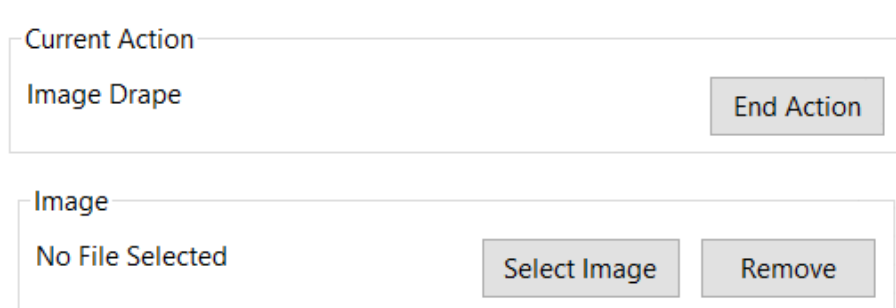
Tab Location(s): Action Tab

Panel and Button: Extract Volume

Prerequisite: Open scene of a scan, cloud or TIN and existing closed polyline.

Image Drape

The Image Drape mode is accessed by clicking the **Image** button in the Edit panel of the **Action Tab**. Two Panels make up the Image Drape dialog box.



Current Action

The Current Action panel displays the current mode, in this case Image Drape. The **End Action** button ends the Image Drape mode and closes the dialog box. The user is returned to the **Action Tab** default dialog box.

Image

If an image is currently draped in the scene, its file name will be listed in the Image panel. The two functions of the Image panel are:

Select Image - This button opens the Select Overlay Image dialog box allowing the user to browse for the image they wish to use. Supported image formats include; BMP, JPG, JPEG, TIF and TIFF.

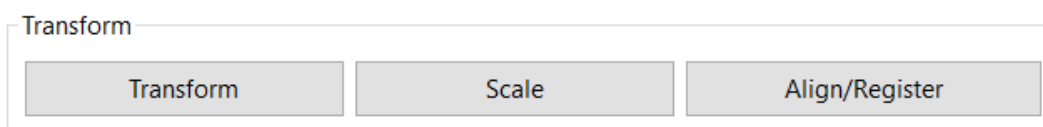
Remove - This button removes the current image from the scene. If there is no image in the current scene this button has no effect.

Tab Location: Action Tab

Panel and Button: Edit and Image

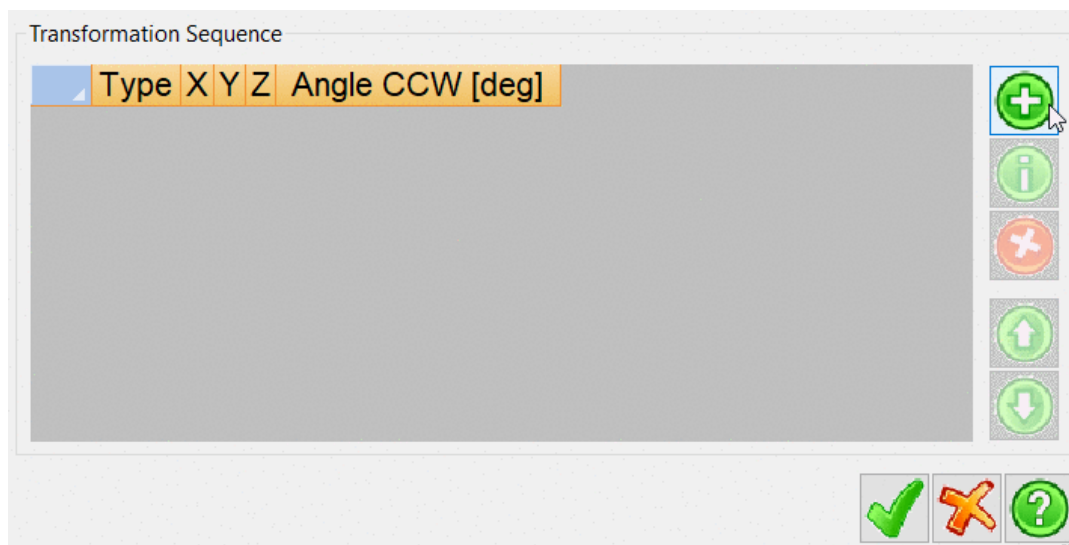
Prerequisite: Open scene of a TIN and an image file that is georeferenced

The Transformation Panel



The **Transform** panel contains three functions that allow for minor adjustments to be made to the selection set if required:

Transform will display the **Transform Cloud** window.



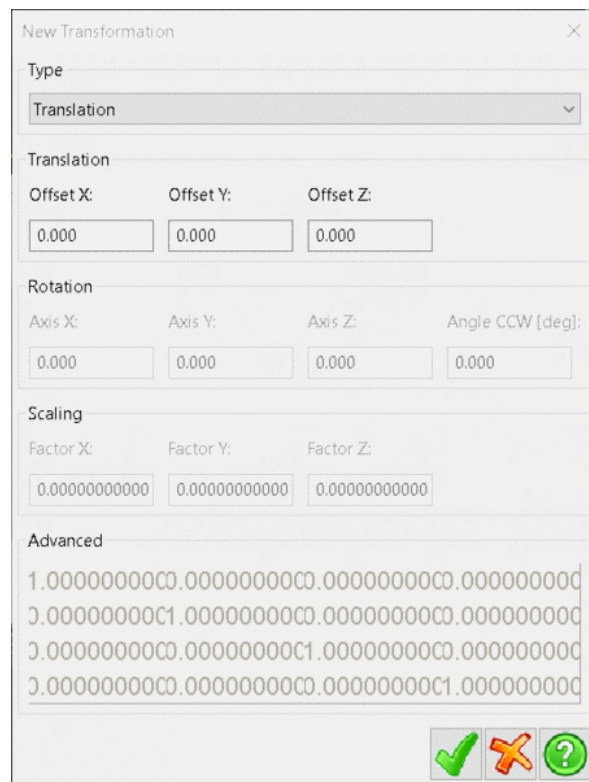
Initially, the dialog will have no transformations specified. To add a transformation, press the **Green Plus** button, which will open the **New Transformation** window.

Use the **Type** drop-down menu to select one of the four Transformation types:

Translate allows the user to define a translation of the current data set by specifying offsets in X, Y and Z.

Rotations allow the user to rotate the current selection set by specifying rotations around the X Y and Z axis.

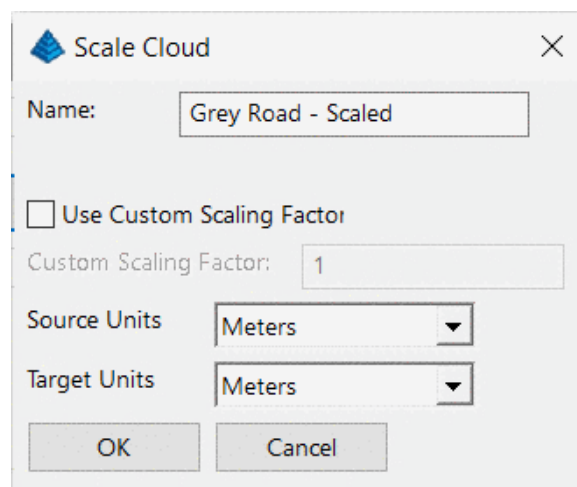
Scaling allows the user to scale the current selection by a set Factor in the X, Y and Z axis. **Advanced** allows the user to specify a transformation matrix to be applied to the data.



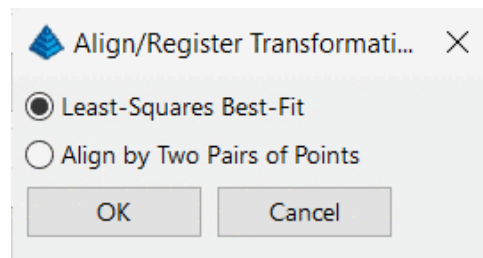
After clicking the check button you should be taken back to the **Transform Cloud** window. From here you can add more transformations, modify the currently selected one by pressing the **Green i** button, delete them by pressing the **Red X** button, or change their order of application to the data using the **Green arrow** buttons. Press the **Green Tick** button in the **Transform Cloud** window to apply the transformations to the selected data.

The transformation will be applied and, if **Create New Cloud** was selected, a new Cloud created in the **Project Tree** - the original Cloud will be unaffected. If **Edit Existing Points** was selected, then the cloud selected will be transformed.

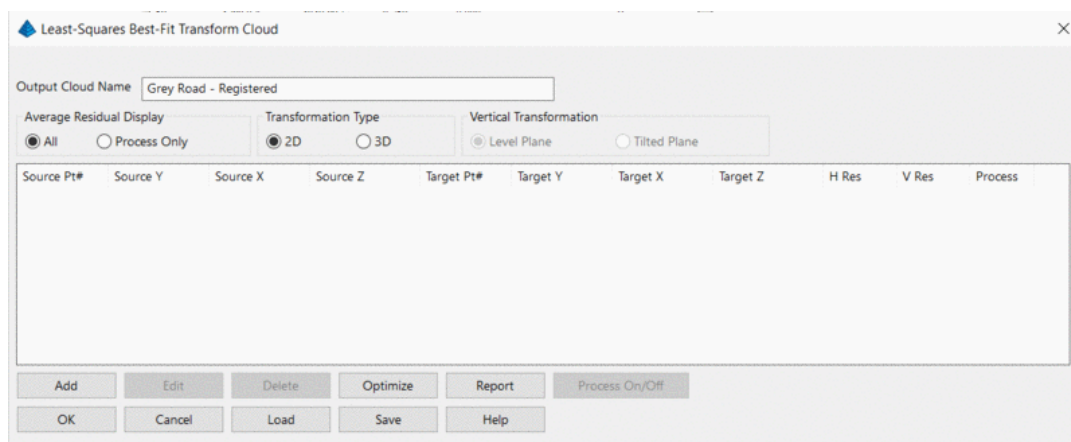
Scale allows the user to **Use Custom Scaling Factor** or to convert between Meters, US Feet and International Feet.



Align/Register performs either a **Least Squares Best-Fit** or **Align by Two Pairs of Points**



Least Squares Best-Fit is used when there are more than two pairs for translation points. Since two pairs of points are sufficient to define the translation and rotation, more than two pairs of points provides more than enough information. A Rigid Body transformation is used for the horizontal aspect.



The **Transformation Type** chooses between doing a 2D transformation and 3D transformation. For the 3D transformation, the program transforms the x/y using the same method as the 2D transformation, and the Z is transformed using an elevation difference model that is modeled by either a best-fit level plane or tilted plane as set by the **Vertical Transformation** setting.

The **Add** button is used to define the source and destination coordinates for the points that define the transformation. Pressing this button brings up the following dialog box.

The dialog box is titled "Add Alignment Point" and features a close button (X) in the top right corner. It is organized into two primary sections: "Source Coordinate" and "Target Coordinate".

Source Coordinate Section:

- Point Number:** An input field with a "List" button to its right.
- Northing:** An input field with a "Screen Pick" button to its right.
- Easting:** An empty input field.
- Elevation:** An empty input field.

Target Coordinate Section:

- Point Number:** An input field with a "List" button to its right.
- Northing:** An empty input field.
- Easting:** An empty input field.
- Elevation:** An empty input field.

At the bottom of the dialog, there are three buttons: "OK", "Cancel", and "Help".

The **Edit** button is used to edit existing data.

The **Delete** button removes the source and destination pairing from the transformation setup.

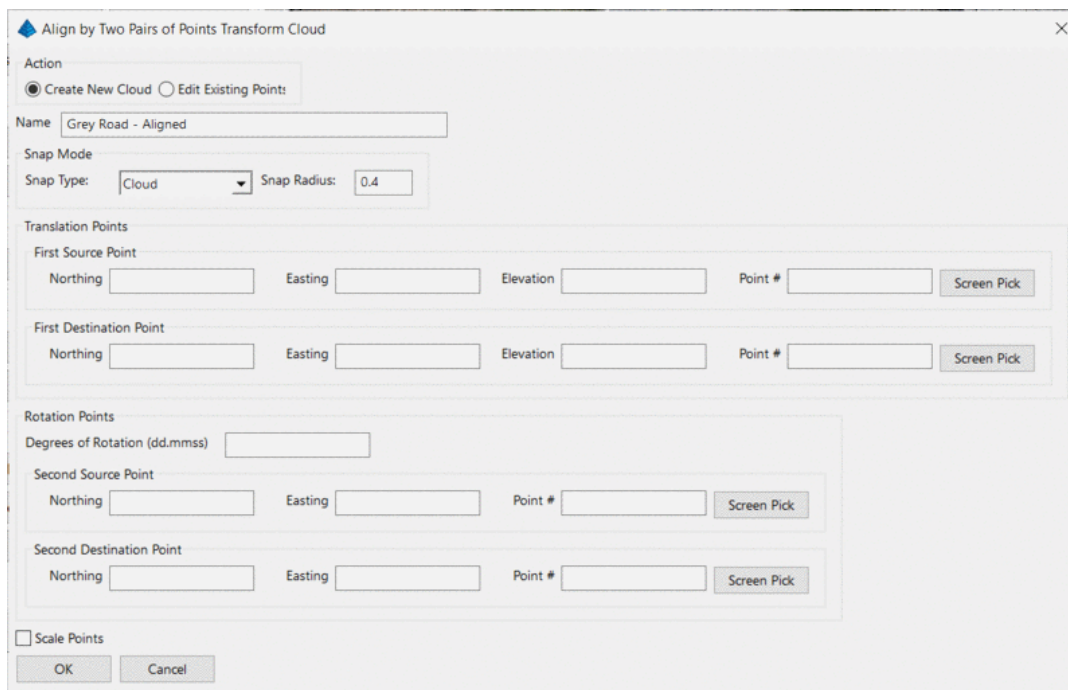
The **Process On/Off** button allows source and destination pairings to be turned on and off. This is useful when wanting to inspect different results using different pairings.

The **Optimize** option chooses which point pairings would yield the best transformation results by turning off the processing of pairings with higher residuals. This minimizes the average residual for the control points.

The **Report** option displays a report of the transformation point pairings, their residuals, processing status, transformation scale and avg. residual.

The **Load** and **Save** options allow for saving and recalling local coordinate transformation pairings and settings.

Align by Two Pairs of Points option uses two pairs of source and destination coordinates.



A new cloud can be created via **Create New Cloud** or the existing cloud can be modified using **Edit Existing Cloud**.

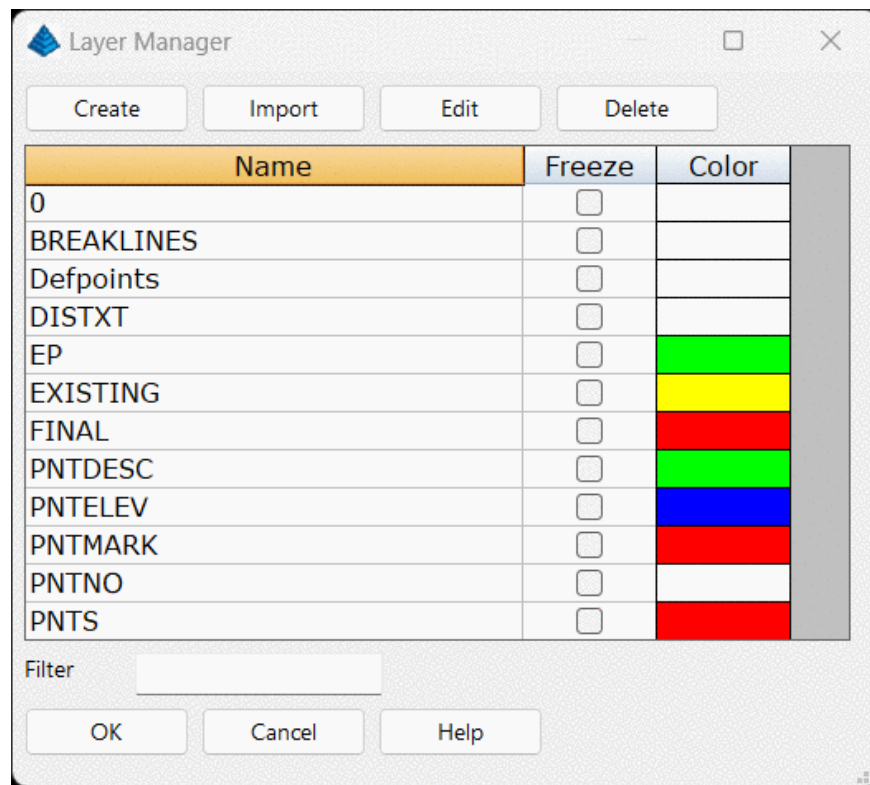
Snap Type sets the type of snap to be used when using the **Screen Pick** option.

The first pair defines the translation as the difference between the source and destination northing and easting. This destination point is also the pivot point for rotation. **Rotation** can be entered directly or defined by a second pair of points where the bearing between the first and second source points is rotated to align with the bearing from the first and second destination points. There is an option to also apply **scaling**. The scaling holds the angle between points and adjusts the distances by the scale factor. The scale factor is calculated for each point as the elevation factor at the first source point times the grid factor at the first destination point averaged with the elevation factor at the transform point times the grid factor at the transform point.

Common Utilities

Layer Properties Manager

The **Layer Properties Manager** can be accessed from the **Project Tab** by **right-clicking** on **Layers** and selecting **Edit**. It can also be opened from the menu bar in an opened **Scene** by selecting **View** and then **Layer Manager** or the toolbar by clicking the **Layer Manager** button.



Create adds a new layer to the list. The default new layer name is Layer # where # begins at 1 and increments as new layers are added.

Import allows you to import the layer names from the CAD project drawing or layer standards file.

Drawing Layers will read the layers in your drawing and open a **Select Layers** window. You can select one or multiple layers in the panel. Click **OK** to import the layers. The program will import the layer name and the layer color.

Layer Standards File will open a file selection window. Browse and select the Layer Standards File (.la) and click **Open**.

Edit Places the cursor in the **Name** field of the *highlighted layer* to allow editing of the layer name. Users can also edit layer names by double-clicking on the layer name field. **Note:** if no layer is highlighted, the cursor will select the first layer in the list.

Delete deletes the currently selected layer. **Note:** users can not delete a layer that has objects drawn on it.

The **Layer Properties** panel

Name - Shows the layer name.

Freeze - Toggles the freeze mode on and off. When toggled on, the objects on that specific layer will not be displayed.

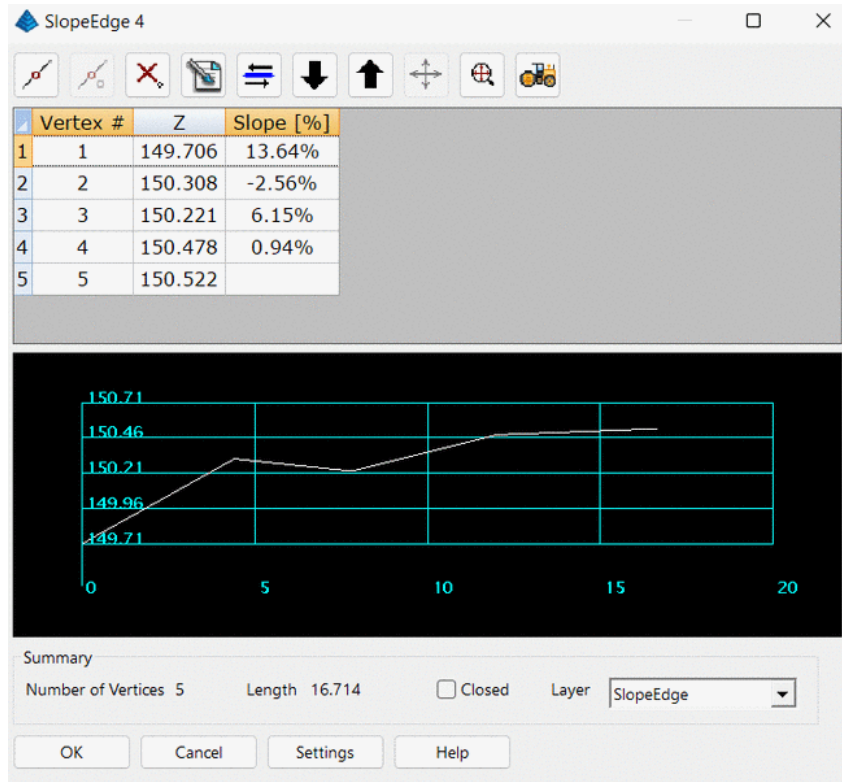
Color - Shows the color assigned to each layer. Click in the **Color** box to select a different color.

Tab Location(s): -Project Tab or Scene Viewer-

Prerequisite: -None-

Polyline Editor

Use the **Polyline Editor** to make changes to the polylines in the point cloud project. The **Polyline Editor** window can be opened by **right-clicking** on a polyline and selecting **Edit** or by **double clicking** on a polyline in the **Current Project** tree. The **Polyline Editor** can also be accessed when creating a new polyline from the **Action** tab. Under the **Active Polyline** panel, highlighting a polyline and clicking the **Edit** button.



Along the top is a toolbar with several icons. The icons and their corresponding functions are as follows:



New Vertex brings up a dialog that allows you to specify the properties of a new vertex to add.

Edit Vertex brings up the edit vertex dialog which allows you to change properties of the currently selected vertex.

Delete Vertices deletes the currently selected vertex or vertices.

Rename Polyline renames the currently selected polyline.

Reverse Polyline reverses the direction.

Import Polyline brings up the Carlson file selection dialog.

Export Polyline brings up the ASCII file export dialog.

Transform allows you to define a transformation sequence to apply to the vertices.

Zoom Extents zooms to the polyline in the current scene.

Flatten allows you to flatten the polyline.

Adding Vertices

Clicking the **New Vertex** button will bring up the **New Vertex** dialog.

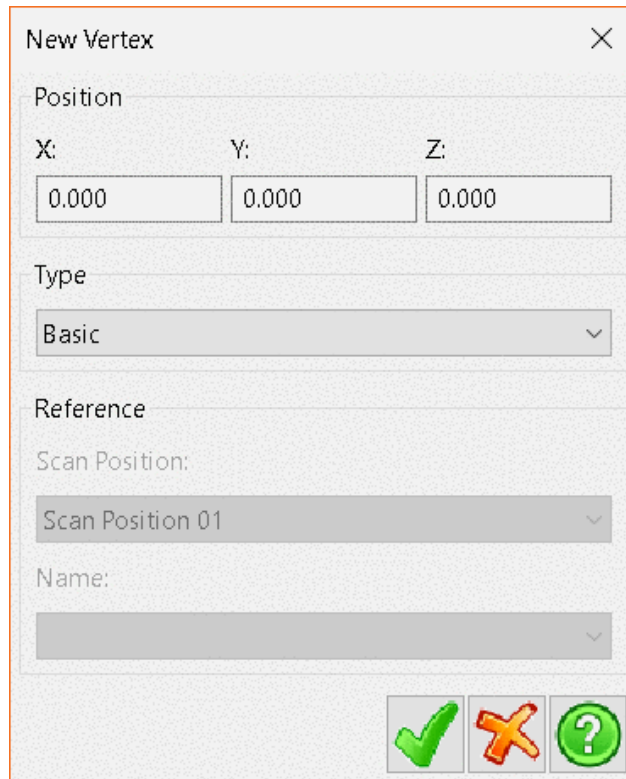
X, **Y**, and **Z** are the x, y, and z coordinates values for the new vertex to be added.

Type can be either **Basic**, **Target Point Reference** or **Coordinate Point Reference**.

The **Reference Panel** is activated when the vertex type is set to **Target** or **Coordinate Point Reference**.

When using **Target Point Reference**, select the desired scan position from the drop-down menu and then select the correct target point from those available in the **Name** drop-down menu.

When using **Coordinate Point Reference**, select the desired coordinate point from those available in the **Name** drop-down menu.



The image shows a 'New Vertex' dialog box with the following fields and options:

- Position:** X: 0.000, Y: 0.000, Z: 0.000
- Type:** Basic
- Reference:** Scan Position: Scan Position 01, Name: (empty)

At the bottom right, there are three icons: a green checkmark, an orange X, and a green question mark.

Importing Polyline

Clicking the **Import** button will bring up the **Import Polyline** file selection window. The currently supported file types for polyline import are **TXT**, **PLN** and **CL**.

Navigate to the correct polyline file and click **Open**.

Exporting Polyline

Clicking the **Export** button will open the **Export Polyline Vertices** dialog, which allows users to export the polyline to several different file formats.

In the **Columns** panel, click on the check boxes to include or exclude the data elements. To change the order of the data elements in the file, click on the element and then click the up or down arrows to adjust its position

The **File** panel determines which file format to export to and also allows you to specify an extension other than the default for a given file format.

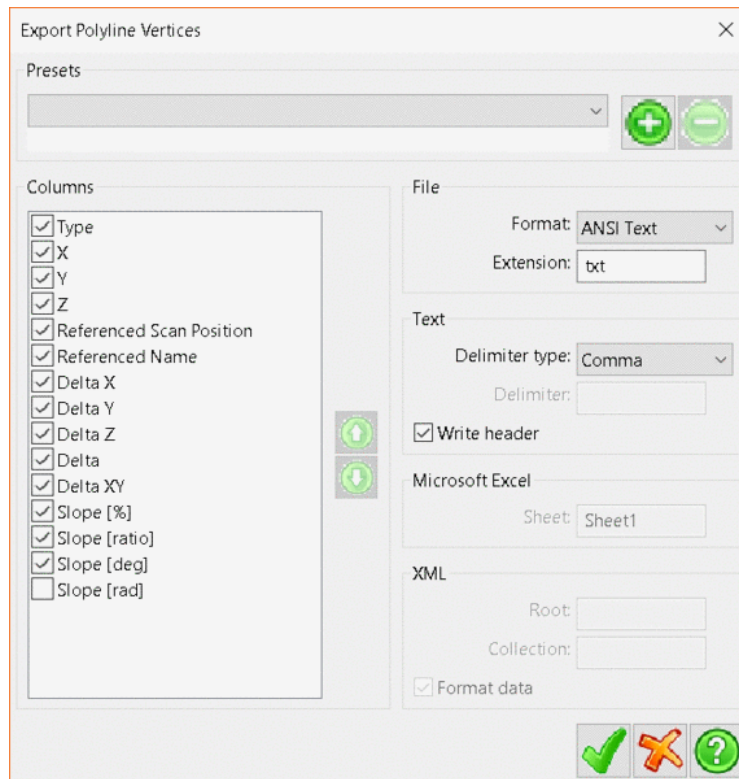
The **Text** panel contains options when exporting to the different text file types, **ANSI Text** or **Unicode Text**. The **Delimiter Type** determines the divider between data elements in the file. The Write header option specifies whether to write a header line to the file detailing the data ordering of the file.

Microsoft Excel allows the user to define the name of the Excel worksheet that the points will be exported to.

Note: only available when exporting as **Format Microsoft Excel XML** allows users to define the **Root** and **Collection** values and choose to **Format data**

Users can select a preset export format by selecting it from the **Presets** drop-down menu. To define a new preset, include or exclude the desired data elements in the **Columns** panel, organise the elements into the desired order and then click the **Green Plus** button next to the **Presets** drop-down. The new preset will be added to the drop-down menu for future use.

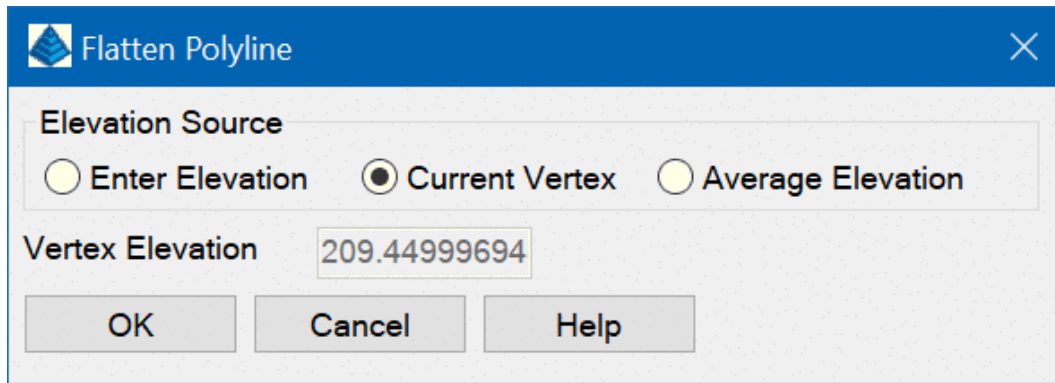
To delete a preset, select it from the **Presets** drop-down menu and then click the **Green Dash** button next to the **Presets** drop-down menu.



Flatten Polyline

Clicking on the **Flatten Polyline** button will bring up the Flatten Polyline dialog box. There are three available methods to flatten the polyline:

- **Enter Elevation** - allows the user to enter in a constant value to be used for all of the vertices.
- **Current Vertex** - sets the polyline elevation to the currently-selected vertex.
- **Average Elevation** - will average all of the vertices. This value is displayed in the box underneath for confirmation.



Settings

The Settings button will bring up the **Polyline Editor Settings** window, which allows the user to configure which data elements of the current vertex set are visible, as well as the order that they are displayed in.

To turn off a data element so it will not be displayed, click in the check box next to that element in the **Columns** panel.

To change the order that data elements are displayed, select the element to move and then click the green up and down arrows to adjust the elements position.

Users can select a preset export format by selecting it from the **Presets** drop-down menu. To define a new preset, include or exclude the desired data elements in the **Columns** panel, organize the elements into the desired order and then click the **Green Plus** button next to the **Presets** drop-down. The new preset will be added to the drop-down menu for future use.

To delete a preset, select it from the **Presets** drop-down menu and then click the **Green Dash** next to the **Presets** drop-down menu.

Tab Location(s): Project Tab and Action Tab

Access Command: Right-Click Points and select Edit (Project Tab) Click Edit button (Action Tab - Polyline Creation mode)

Prerequisite: Scene with a polyline

Point Editor

The Point Editor allows you to edit the points in the point cloud project. **Right-clicking** on **Coordinate Points** in the **Current Project** tree and select **Edit** or by double clicking on a set of points. In the **Coordinate Points** window, there are several base functions that are shared across all point types, with some extended options added for certain points types.

Name	X	Y	Z	Description
1001	5678168.022	3654106.694	779.725	MH BEG
1002	5678180.162	3654099.604	779.455	MH
1003	5678191.182	3654089.144	779.325	MH
1004	5678201.252	3654097.374	779.585	MH END
1005	5678195.772	3654105.054	779.585	WV
1006	5678205.982	3654103.604	779.555	WV
1007	5678204.012	3654073.124	779.295	WV
1008	5678166.512	3654071.484	779.035	SIGN
1009	5678114.112	3654081.464	779.915	SIGN
1010	5678219.232	3654054.984	780.375	SIGN
1011	5678212.672	3654146.024	779.945	SIGN
1012	5678160.702	3654127.324	780.545	FS
1013	5678180.292	3654130.244	780.215	FS
1014	5678197.022	3654129.034	779.945	FS

36 Coordinate Points, 0 Selected

Along the top toolbar are several option buttons:



Add Point brings up a dialog that allows you to specify the properties of a new point to add.



Edit brings up the point edit dialog which allows you to change properties of the currently selected point.



Copy Points allows you to copy the currently selected points to another point set (such as from the control points to the target points of a scan).



Delete deletes the currently selected points.



Import Points brings up the ASCII file import dialog.



Export Points brings up the ASCII file export dialog.



Transform allows you to define a transformation sequence to apply to the points.



View brings up the scene creation dialog for viewing the current point set.



Field to Finish Draws the selected points in CAD using the current FLD file and field to finish settings.



Draw to CAD brings up the Point Draw Settings dialog specify a layer to draw the selected points on in CAD and specify if the points should be added to the current CRD file.



Coordinate System allows you to choose the coordinate system the positions values are in (Global or any current scan positions).



Settings allow you to configure what properties of the points are visible in the spreadsheet control.



Help brings up help documentation.

Adding Points

Clicking the **Add Point** button will open the **New Coordinate Point** window.

Name is a unique name for the point to be used to identify it in Point Cloud.

Description is a description for the point.

X, **Y**, and **Z** are the x, y, and z coordinates values for the new point to be added. Or click on **Screen Pick** to select from the cloud.

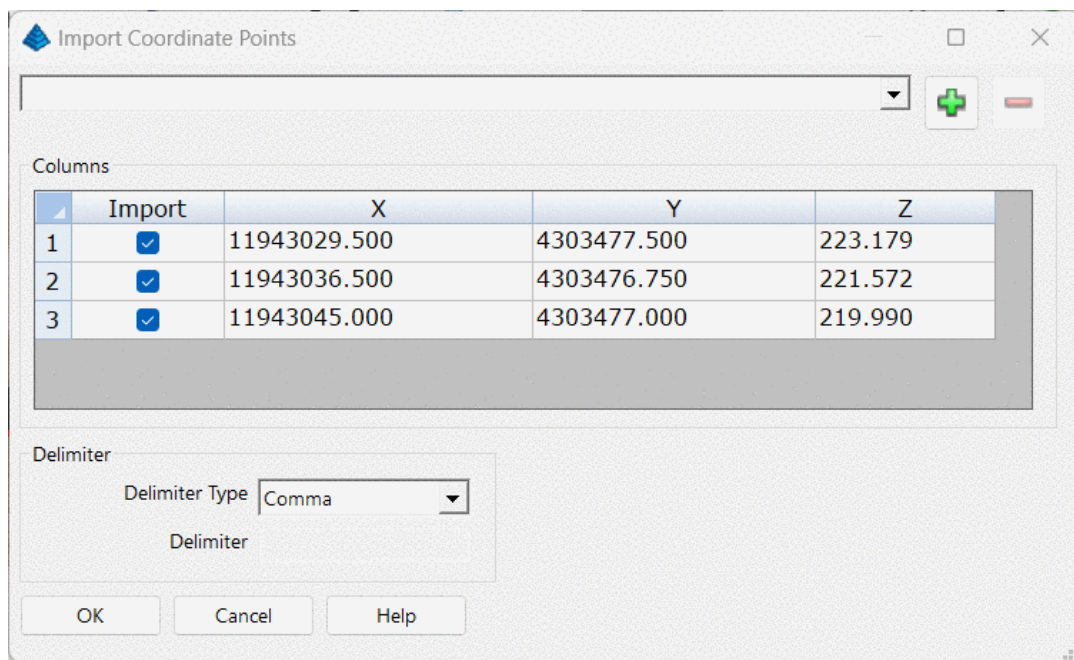
X	Y	Z
5678133.716	3654096.467	780.441

Importing Points

Clicking the **Import Points** button will open the **Import Coordinate Points** file selection dialog. The file formats currently supported for import are **TXT**, **CSV**, **CRD** and **CRDB**. Navigate to the correct file and click **Open** to display the **Import Coordinate Points** window.

Selecting **Rename all items** will rename all imported points to the settings specified in the **Naming Conventions** settings. The **Duplicate Entries** panel will specify the action to take when a point being imported has a name that already exists in the set of points being imported to.

Click the **Green Tick** to import the points.



Exporting Points

Clicking the **Export Points** button will open the **Export Coordinate Points** window, which allows users to export the data to several different file formats.

In the **Columns** panel, click on the check boxes to include or exclude the data elements. To change the order of the data elements in the file, click on the element and then click the up or down arrows to adjust its position.

The **File** panel determines which file format to export to and also allows you to specify an extension other than the default for a given file format.

The **Text** panel contains options when exporting to the different text file types, **ANSI Text** or **Unicode Text**. The **Delimiter Type** determines the divider between data elements in the file. The **Write header** option specifies whether to write a header line to the file detailing the data ordering of the file.

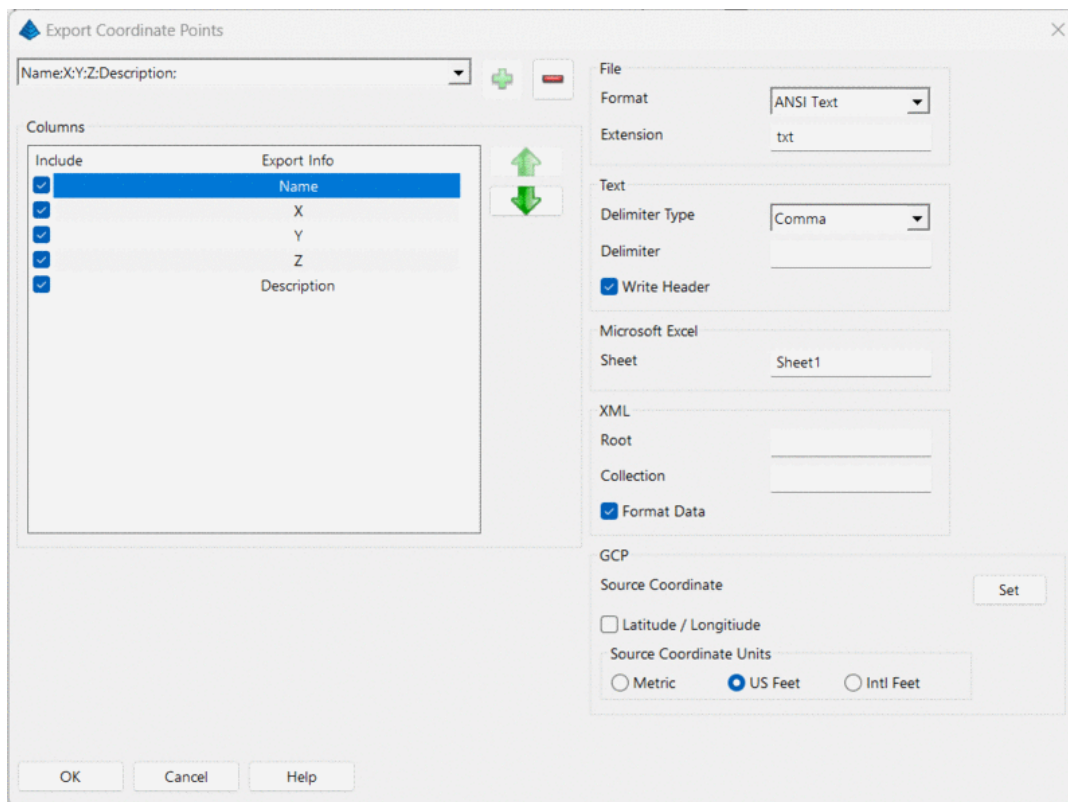
Microsoft Excel allows the user to define the name of the Excel worksheet that the points will be exported to. **Note:** only available when exporting as **Format: Microsoft Excel**.

XML allows users to define the **Root** and **Collection** values and choose to **Format data**. **Note:** only available when exporting as **Format:XML**.

GCP - when exporting as GCP, click on the **Set** button to choose the Coordinate Projection. Users can choose to select a Pre-Defined projection, add a projection from a file or create a User-Defined projection on the fly. Click **OK** to select and return to the previous window. Users can select to export the coordinates in Latitude/Longitude format and also to select the appropriate units; **Metric**, **US Feet** or **International Feet**.

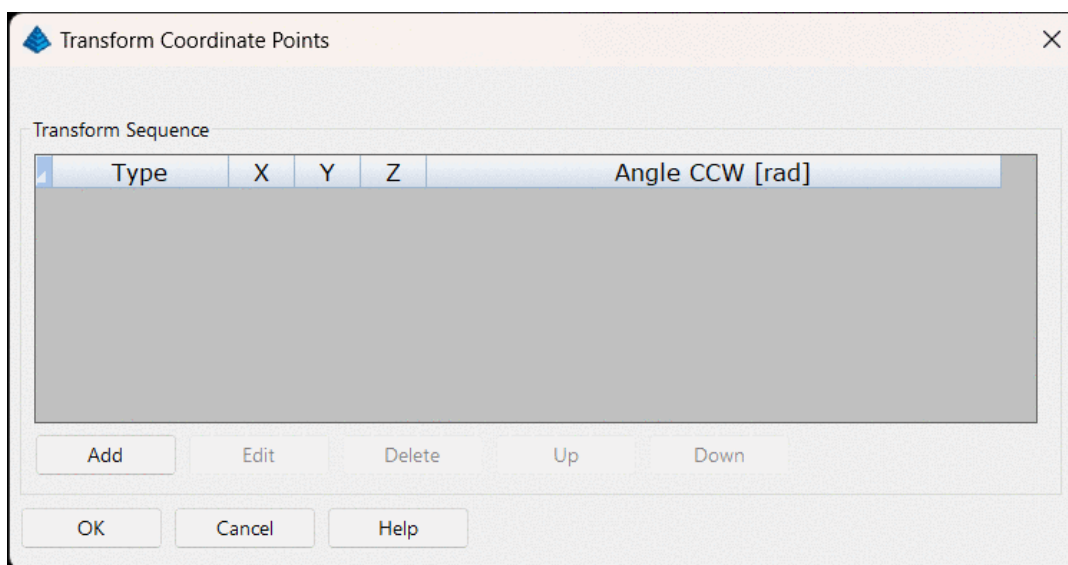
Users can select a preset export format by selecting it from the **Presets** drop-down menu. To define a new preset, include or exclude the desired data elements in the **Columns** panel, organise the elements into the desired order and then click the **Green Plus** button next to the **Presets** drop-down. The new preset will be added to the drop-down menu for future use.

To delete a preset, select it from the **Presets** drop-down menu and then click the **Red Dash** button next to the **Presets** drop-down menu.



Transform Points

The **Transform** button opens the **Transform Coordinate Points** window where users can define a selection of transformation parameters to apply to the selected Coordinate Points.



Initially, the dialog will have no transformations specified. To add a transformation, press the **Add** button, which will open the **New Transformation** window.

Use the **Type** drop-down menu to select one of the four Transformation types:

Translate allows the user to define a translation of the current data set by specifying offsets in X, Y and Z.

Rotations allow the user to rotate the current selection set by specifying rotations around the X Y and Z axis.

Scaling allows the user to scale the current selection by a set Factor in the X, Y and Z axis.

Advanced allows the user to specify a transformation matrix to be applied to the data.

After clicking the check button you should be taken back to the **Transform Coordinate Points** window. From here users can add more transformations, modify the currently selected one by pressing the **Edit** button, delete them by pressing the **Delete** button, or change their order of application to the data using the **Up / Down** buttons. Press the **OK** button to apply the transformations.

Note: Transformations are applied in top-to-bottom order.

Advanced			
1.000000000	0.000000000	0.000000000	0.000000000
0.000000000	1.000000000	0.000000000	0.000000000
0.000000000	0.000000000	1.000000000	0.000000000
0.000000000	0.000000000	0.000000000	1.000000000

Settings

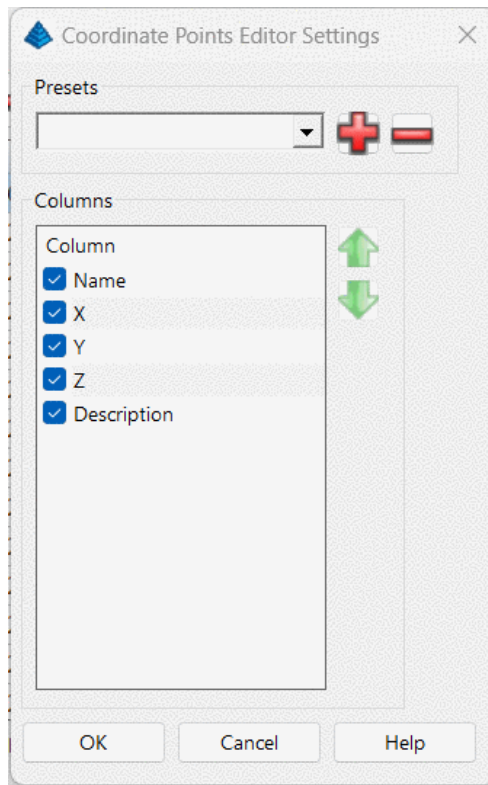
The Settings button will bring up the **Coordinate Point Settings** window, which allows the user to configure which data elements of the current point set are visible, as well as the order that they are displayed in.

To turn off a data element so it will not be displayed, click in the check box next to that element in the **Columns** panel.

To change the order that data elements are displayed, select the element to move and then click the green up and down arrows to adjust the elements position.

Users can select a preset export format by selecting it from the **Presets** drop-down menu. To define a new preset, include or exclude the desired data elements in the **Columns** panel, organize the elements into the desired order and then click the **Red Plus** button next to the **Presets** drop-down. The new preset will be added to the drop-down menu for future use.

To delete a preset, select it from the **Presets** drop-down menu and then click the **Red Dash** button next to the **Presets** drop-down menu.



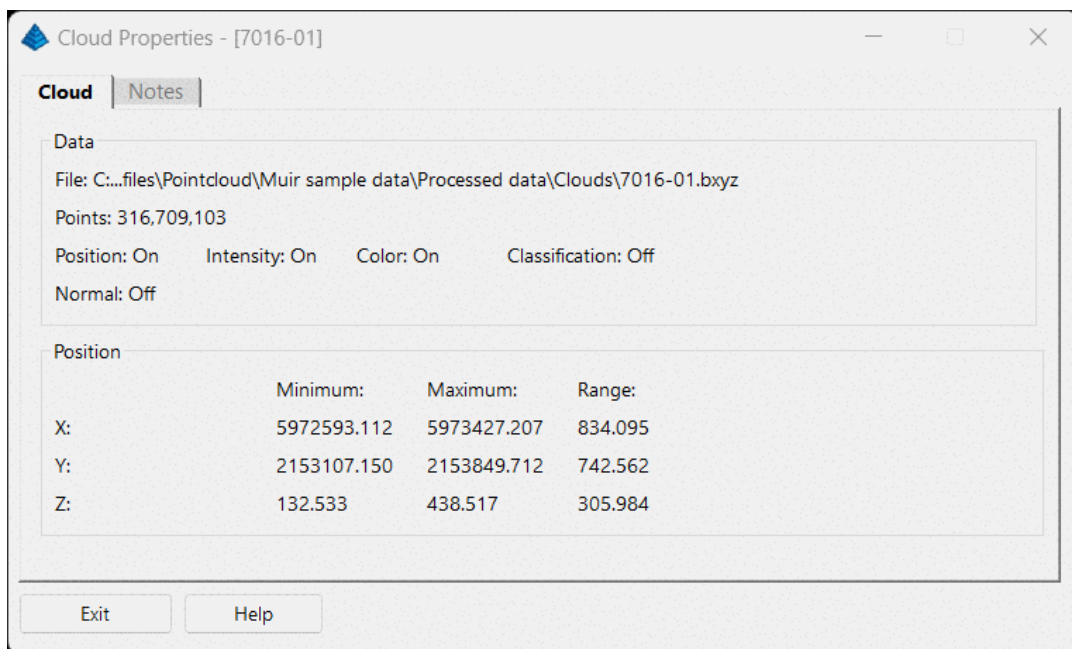
Tab Location(s): Project Tab

Access Command: Right-Click Points and select Edit

Prerequisite: Control Points or Target Points or Coordinate Points

Item Properties

To see statistics and information about the size of Scans, Clouds, TINs, or Scenes, **right-click** the object in the **Current Project** tree and select **Properties** to open the **Properties** window.



The Properties window can have different tabs depending on the data object selected. For example, when a cloud is selected as in the above image, the Properties window will display **Cloud** and **Notes** tabs. In the Cloud tab you will see the location of the cloud file on your drive or network, the number of points found in the cloud, and the minimum, maximum, and range of coordinates.

The **Scan** properties adds a **Scan** tab and a **Preview** tab. The **Preview** tab found in **Scan Properties** will show a small preview of the scan data in 2D.

The **TIN** properties adds a **TIN** tab.

The **Notes** tab features a textbox where users can add notes to the object for future reference.

Tab Location(s): Project Tab

Prerequisite: A Scan, TIN or Cloud



Natural Regrade Module 14

Traditional landscape design is often based on the subjective judgment of landscape appearance or desired land use with little consideration for proper hydrologic function for balanced conveyance of water and sediment from the land surface. Alternately, traditional design methods might use engineering principles to create structural controls for water and sediment conveyance (Bugosh, 2000). Over the last several decades, fluvial geomorphic research has identified distinct relationships among several important factors including climate, discharge, slope, and earth materials that define stable stream channels (Bloom, 1978) (Dunne and Leopold, 1978) (Williams, 1986). The Carlson *Natural Regrade* module applies fluvial geomorphic principles to upland landforms through computer software (GeoFluv™). GeoFluv™ creates a landscape design that mimics the functions of the natural landscape that would evolve over time under the physical and climatic conditions present at the site to convey the water and sediment from the land surface in a stable hydrologic equilibrium.

The following sections summarize why GeoFluv™ benefits reclamation landform design and description of how computerizing the approach provides a value multiplier by allowing detailed designs to be made and evaluated quickly. The "Problems Addressed by *Natural Regrade*" section discusses limitations of conventional approaches to disturbed-land reclamation design. "The Fluvial Geomorphic Solution" section discusses how and why *Natural Regrade's* GeoFluv™ approach solves the problems inherent in the conventional approaches. The "Description of Software" section explains fundamental concepts and terminology used in GeoFluv™ approach, and how and where these are used in the *Natural Regrade* module.

The remaining sections are as follows. The "Links with other Software" section describes other software that the user can use along with *Natural Regrade* to achieve even greater efficiency when constructing the GeoFluv™ design (or any other construction project). The "Software Compatibility" section describes the CAD software that *Natural Regrade* is designed to work with.

Highlight Tractive Force Zones

This Natural Regrade command allows the user to identify zones of tractive force values in the GeoFluv design channels above a user-specified threshold value. This is useful when editing a draft design to reduce tractive force values to meet the designer's criterion for channel 'steady-state' stability, i.e., adequate stream competence to transport sediment through the system and not cause problematic deposition while also not accelerating channel bank and bed erosion. When the user can see the zone where tractive forces exceed the project threshold value criterion, they can consider design edits, e.g., lowering reach slope, reducing reach discharge, that can lower those tractive forces to target values.

The command collects input values from the design in the drawing and uses them in the tractive force calculations. Zones in the drawing on each channel reach that exceed the specified threshold are highlighted by a linework polygon enclosing the zone's upper and lower ends. The user can specify the width of the zone, hatching within the zone, and zone color for both bankfull and flood prone values.

Double-clicking on the Highlight Tractive Force Zones command button on DWG tab causes the Highlight Tractive Force Zones Above Threshold dialog box to appear on the screen.

Bankful Highlight Settings	
<input checked="" type="checkbox"/> Draw zones of bankful tractive force above threshold	
Threshold Value (lbs/sq.ft.)	1,290
Check Interval (ft.)	2,000
Highlight Width (ft.)	5,000
Layer	GF_ThresholdBankfull
Hatch Style	SOLID
Hatch Scale	0.100
Zone Color	Red (1)
Hatch Transparency (0-100)	50

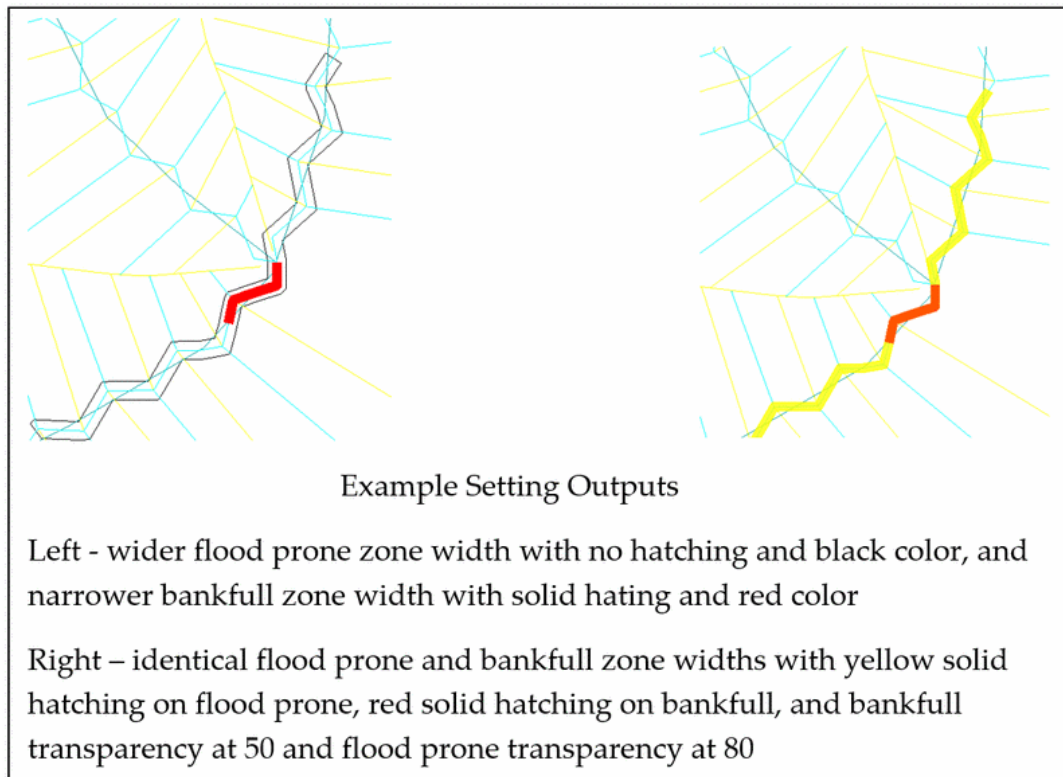
Flood Prone Highlight Settings	
<input checked="" type="checkbox"/> Draw zones of flood prone tractive force above threshold	
Threshold Value (lbs/sq.ft.)	1,290
Check Interval (ft.)	2,000
Highlight Width (ft.)	10,000
Layer	GF_ThresholdFloodProne
Hatch Style	SOLID
Hatch Scale	0.100
Zone Color	Yellow (2)
Hatch Transparency (0-100)	80

Draw zone of bankfull/flood prone tractive force above threshold: The user clicks the appropriate toggle box to specify if they want to draw tractive force zones for bankfull or flood prone or both discharges. The edit boxes in the bankfull and flood prone fields become active when their boxes are toggled on.

Threshold Value: The user enters the tractive force value above which they want identified on the GeoFluvTM design drawing. This and greater values will be located in the highlighted zones.

Check Interval Distance (ft or m): The user enters a distance value for Natural Regrade to collect input values from the design surface for the tractive force calculation. The value must be greater than or equal to 2 ft.

[Note: The tractive force values are calculated from variables measured from the design surface TIN file, which in turn is made from the input coordinate data, so that small irregularities in the resulting surfaces can result in jumps in the calculated variable values, that in turn affect the tractive force calculations over very small distances. We have observed these jumps in testing different drawings based on data sets of varying x, y, and z resolution and density. The value jumps present themselves as breaks within the zone above the threshold value. The user can eliminate any resulting breaks in the tractive force zones by increasing the check interval values until the breaks disappear. Our testing indicated that for many data sets the presented values should be considered accurate within about five percent for this reason.]



Highlight Width (ft/m): The user can specify the width of the polygon enclosing the zone above the specified threshold value. Specifying different widths for the bankfull and flood prone values can make them easier to see when both are selected (left example in settings output figure).

Layer: The tractive force above threshold zone polygon linework is drawn on GF_ThresholdBankfull and GF_ThresholdFloodProne layers by default. These layers are erased by default when leaving "Edit design surface in drawing" mode and returning to "Edit GeoFluv inputs" mode. The user can enter a different name in the edit box window if saving the zone polygons to a different layer is desired.

Hatch Style: The user can select a hatch style pattern from the example icons or by scrolling through the list of hatch styles. Selecting "None" from the list will color the polygon linework a single, color without any design pattern inside the polygon. Selecting "Solid" from the list will color the polygon linework and the area inside the polygon a single, color without any design pattern. The hatch style can be specified for both bankfull and flood prone tractive force above threshold zones, e.g., selecting "Solid" for bankfull and "None" for flood prone zones can display a flood prone outline surrounding a solid bankfull zone (left example in settings output figure).

Hatch Scale: The user can vary the size of the hatch pattern by entering larger and smaller values in the edit box.

Zone Color: The user can specify the zone color by clicking on the box next to the edit box, selecting the

desired color from the pop-up palette, and clicking "OK". The specified zone color will then be displayed on the settings dialog box.

Hatch Transparency: The user can 'see through' an overlying layer to an under-lying layer by entering a relative value on a scale of 0 to 100 for lesser or greater transparency. For example, if the bankfull and flood prone zones have the same width and the bankfull zone lies entirely within the flood prone zone, transparency would cause the bankfull zone to not be entirely obscured by the overlying flood prone zone, but instead both zones could be seen simultaneously (right example in settings output figure).

After entering all the command settings the user clicks "OK" and the command starts running and displays "Calculating and drawing tractive force zones..." at the command prompt line until completed. Complex data sets with tighter Check Interval Distance settings may take some time to complete drawing the zones; the user will know when the command is done when the command prompt line reads 'Done'.

Problems Addressed by Natural Regrade with GeoFluv

Conventional land-shaping practices are often based on conveying or capturing runoff from an extreme event. These conventional practices include grading slopes to a uniform gradient, building gradient terraces across slope faces, and constructing rip-rapped down drains to convey runoff as shown in Figure 1.

Use and Cost Limitations of Conventional Approach

Conventional designs often do not address the hydrologic balance during less extreme flow conditions. This results in problems with reclamation success for vegetation, livestock, and wildlife post-disturbance land uses, high maintenance costs, and reclamation bond complications.



Figure 1. Conventional steep slope reclamation with uniform slope gradient, gradient terrace, and rip-rap lined down drain

The unnatural configuration of these designs does not provide the terrain diversity that creates spatial variation in water harvesting and slope aspect. The result is that vegetation tends toward a monoculture and animal habitat is minimized. The native land in the foreground of Figure 1 has forbes and shrubs growing near minor gullies, whereas the uniformly-graded slopes above them do not favor these plants, despite having been seeded with them.

Conventional land-shaping practices have high construction, maintenance, and liability costs. Terraces can be difficult and expensive to grade on steep side slopes. The rip-rap material may have to be procured off site and transported to the site. After construction, regular maintenance is often required as the terraces and ditches sized for extreme flows become clogged with sediment at lower flows, or are penetrated by burrowing animals. Clogged or burrowed terraces can result in catastrophic diversions of runoff from the terraces straight down the slope, often requiring major repairs.

Bonding Limitations of Conventional Approach

The conventional approach to reclamation landform design affects reclamation bonding liability and costs. The damage to the slope from a blowout and related repair work can result in a reclamation bond clock being restarted, which prolongs the operator's period of liability. The expense of creating land form designs has often limited an operator's ability to propose incremental reclamation bonding for various stages of a project's disturbance. For example, an operator may determine that their greatest disturbance will occur at year four of a five-year permit and they may post a bond for that maximum disturbance, even though their liability will be lower for the first four years. This creates an unnecessary financial burden for the operator.

The Fluvial Geomorphic Solution

This fluvial geomorphic landscape computer-design software (GeoFluv™) uses an algorithm based on fluvial geomorphic principles. The essence of this approach is to identify the type of drainage network, i.e., stream channels and valleys, which would tend to form over a long time given the site's earth materials, relief, and climate to achieve a stable landform, and to design and build that landform. The resulting slopes and stream channels are stable because they are in balance with these conditions (Rosgen, 1996). They are a reclamation alternative to uniform slopes with terraces and down-drains. Rather than fight the natural forces that shape the land, the algorithm helps the user create a landscape that harmonizes with these forces.

The channel and swales in the foreground, and the steep slope ridges, valleys, and channels in the center of Figure 2., are examples of portions of a 115-acre coal mine reclamation project completed using this innovation fluvial geomorphic approach.



Figure 2. Steep slope reclamation using the fluvial geomorphic approach shown during the second growing season

Natural Stability

Over the last thirty-some years hydrologists have observed and measured stable natural streams and determined mathematical relationships that describe these stable stream types. Essential among these determinations is that channel morphology is directly related to a relatively small, but frequently recurring annual flood event. The natural channel is shaped to keep its sediment load and stream flow in balance during these low-flow events, as well as during extreme events. *The GeoFluv™ fluvial geomorphic approach to land reclamation relates the upland landforms to the stream channel form.* Both can be formed similarly by flowing water. Reclamation landscapes created using fluvial geomorphic principles provide stability against erosion with runoff waters capable of meeting water quality criteria, and support a diverse vegetative community. These landscapes offer the benefits of lower initial cost, no long-term maintenance costs, and they promote bond release (Bugosh, 2002, 2003).

Promotes Bond Release

The GeoFluv™ fluvial geomorphic approach provides a high degree of confidence that reclamation projects will demonstrate long-term stability against erosion similar to adjacent undisturbed lands because the reclamation channels are designed to maintain the hydrologic balance, as the natural channel does. This means that the reclaimed land does not have to be regularly disturbed to repair erosion problems. Additionally, the varied landform provides niches for different plants, wildlife, and livestock. These benefits demonstrate to regulatory authorities that the site will remain stable and productive; that demonstrated stability can promote bond release.

Benefits of Computerizing the Fluvial Geomorphic Approach

Previous application of alternative land-shaping practices may have been limited for several reasons, including the limited extent of training in fluvial geomorphic principles of the designers, the complexity of the design calculations to create a thoroughly integrated landform, and the difficulty of guiding the heavy equipment operators to build more sophisticated designs. The *Natural Regrade* module addresses all these potential limitations. GeoFluv™ creates a draft landform based on empirically determined fluvial geomorphic mathematical relationships. The draft landform is an idealized solution that uses the input parameters to create a stable landform. The designer can then modify this idealized draft landform to conform to special site conditions, such as an archaeological site, landmark, or other feature, or to create a more natural appearance.

User Friendly

Existing computer software for earth-moving designs does not incorporate this innovative approach, is often not "user friendly", and does not have the broad applications for landscape designs that are stable against erosion offered by *Natural Regrade*. GeoFluv™ makes "user friendly" computer design software available to a large body of users that do not have advanced training in fluvial geomorphology, as well as to those who do have this background. *Natural Regrade* has been designed to be as "user friendly" as possible; the program commands are organized following a left-to-right and top-to-bottom format that follows the project design work sequence, with minimal input needed and with guidance provided in the "Help" resource and documentation.

Minimizes Training

The *Natural Regrade* module minimizes the training necessary to immediately use the fluvial geomorphic approach for reclamation at disturbed sites, or when evaluating proposed reclamation designs. Users can compress design time and build reclamation landscapes from disturbed earth to seeded reclamation. GeoFluv™'s developer has successfully introduced this reclamation approach to the largest mining company in the world at truck-and-shovel and dragline operations. The *Natural Regrade* module is designed to quickly make the GeoFluv™ design approach available to the widest range of users including professional hydrologists, environmental scientists, and engineers responsible for reclamation design at disturbed sites, and for regulatory personnel responsible for evaluating reclamation designs.

Simplifies numerous complex calculations

An important advantage of the *Natural Regrade* module's GeoFluv™ computerized approach is the ease by which the user can create landscapes that are functional, stable against erosion, low-maintenance, aesthetically pleasing, and cost-effective. The GeoFluv™ computer design software offers several options for developing input parameters from climatic and hydrologic data, and several options for creating landscape features, e.g., ephemeral, intermittent and perennial stream channels, complex slopes, ridges and valleys, and calculating material balances and centroids, and optimum material movement routes, for the resulting design. The user can design channels with appropriate characteristics, including channel patterns, sinuosity, longitudinal profiles, cross sectional areas, width to depth ratios, etc. and their contiguous uplands as functional components of a stable topography for tens of acres of land in minutes. GeoFluv™ allows the user to view topographic maps and three-dimensional images of the resulting landscape design. The GeoFluv™ approach replaces lengthy and tedious manual calculations and allows rapid evaluation of many landscape design alternatives. This allows the user to select the optimum landscape design for his needs.

Promotes Bonding Alternatives

The ability to quickly create and evaluate alternative reclamation designs provides great utility for both industry and regulatory personnel working on reclamation bonds. Because designing a reclamation surface has been such a lengthy and expensive process, often only a 'worst case scenario' design has been created for setting a reclamation bond. For example, this 'worst case scenario' may have been based on the disturbance in year four of a five-year mine permit. The ability to quickly create design surfaces and conduct mass balance comparisons makes it practical for the *Natural Regrade* module user to propose bonds for several stages of mine development, i.e., incremental bonding, that can reduce bond costs and promote release of more acres from bond.

Interface with GPS and Machine-control Software

This software also is ideal for integrating with Global Positioning System and laser machine control to simplify and speed construction and reduce costs. Construction of the complex landforms that are characteristic of stable natural landscapes, and which GeoFluv™ helps the user design, is facilitated by GPS and machine guidance technologies. The need to survey and stake the designs in the field is eliminated using these technologies, as is the need for the construction team to constantly provide guidance to the equipment operators.

Description of Software

GeoFluv™ requires only minimal input parameters to produce a draft surface and the material balance associated with creating that surface. The software outputs a draft landform that provides a solution for a stable landform that satisfies the input parameters. The software also displays the cut/fill balance achieved when building the draft landform, and centroids of material and void for material movement planning.

The *Natural Regrade* module helps the user through the design process by conveniently organizing all the commands that design a draft landform using the GeoFluv™ approach on a 'dockable dialog box' that is activated by the Design GeoFluv Regrade command. When this command is selected from the *Natural Regrade* menu, the dockable dialog box appears on the screen with all the GeoFluv™ design steps organized in a generally left-to-right and top-to-bottom sequence that leads the user through the design process. As a further aid to design sequencing, subsequent GeoFluv™ design inputs/commands are inactive on the dockable dialog box until the prerequisite step has been made. Finally, the commands automate and integrate as many of the calculations as possible to relieve the user of the burden of repetitive command steps.

The user can focus his design energy on testing alternative designs for enhanced suitability to site-specific conditions. Those site-specific conditions can include post-disturbance land use considerations, community relations, equipment constraints, material constraints, bond costs, visual aesthetics, etc. The resulting three-dimensional surface map can be exported in a variety of electronic formats to other programs, or printed as two-dimensional hard copy. The completed design can be taken to the construction site using survey and stakes, or output electronically to GPS and laser-guided construction equipment to further promote project efficiency. The designed topography can then be constructed with available equipment and earth materials.

Discussion of Input Parameters

GeoFluv™ helps the user create a stable landform based on minimal local input variables. These include site elevations (from a survey grid), a GeoFluv™ project boundary, a local stream base level to which the area within the GeoFluv™ boundary drains, a desired drainage density, design maximum discharge velocity, precipitation from the 2-yr, 1-hr and 50-yr, 6-hr storms, and runoff coefficient. The user will also select a desired cut/fill balance tolerance. Figure 3 shows an example of the minimal input data needed for the software to design the landform using this fluvial geomorphic approach.

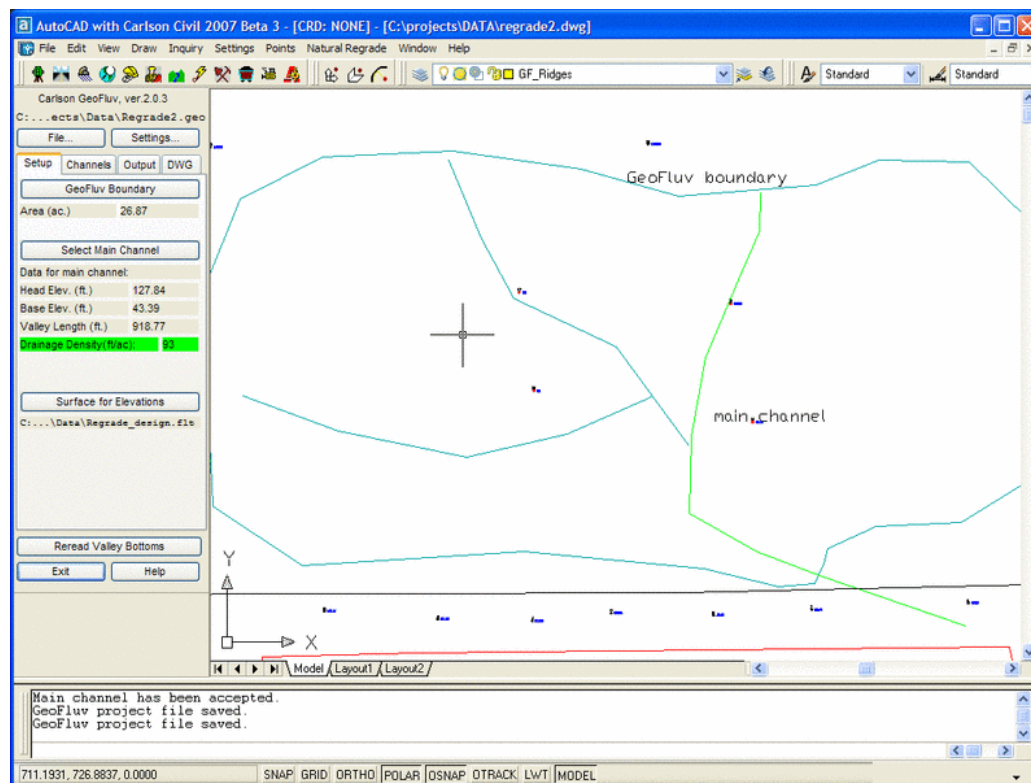


Figure 3. Example of Setup tab input dialog box

The user can then edit this idealized landform for any number of reasons. The site may have boundaries that must be avoided. The user may want to bend a channel around an archaeological or historic site, or local landmark. The user may want to alter slope aspects to promote vegetation diversity, wildlife niches, or to harvest moisture by retaining snow. Aesthetic considerations, such as view sight line, may prompt the user to edit the draft landform. Material movement planning may require the user to evaluate factors including the cut/fill balance and haul distances associated with various alterations of the draft landform. The user may wish to create several interim landform

designs leading to the final design for submission for incremental reclamation bonding. The ease and speed by which the software creates a draft design solution facilitates these and other edits. The *Natural Regrade* module frees the user to focus on site-specific design considerations and finding an optimal solution to creating a stable site landform, rather than being immersed in ponderous calculations for each subwatershed. Following the discussion of input parameters below, the Settings button default settings on the Setup tab will be explained. These settings are one way that the draft landform can be edited.

Drainage Density and Channel Pattern

The drainage density input is the valley length (without meanders) divided by the subwatershed area (Dunne and Leopold, 1978). Its units are length over area (L/L^2). Convenient U.S. units for landscape design work are feet/acre. This value will vary depending on factors such as earth materials, slope aspect, storm intensity, and vegetation type and coverage.

Drainage density is important because it represents the subwatershed size that will be stable for the local conditions. Drainage density and the ridges that form between channel meanders work together to break up the landform into many small subwatersheds, as can be seen in the natural subwatershed shown in Figure 4. The subwatersheds minimize both slope length and catchment area and thereby minimize erosion.



Figure 4. Natural "A" channel meanders and ridges break slope length into a series of subwatersheds

The drainage density is an expression of the amount of erosion that has occurred in the watershed. In a stable watershed, it represents the state at which sediment supply and water runoff are balanced in a state of dynamic equilibrium. Designing the landform using an appropriate drainage density for the project area conditions is an important first step toward achieving a stable landform design.

Watersheds may be disturbed in different ways and those affect reclamation planning differently. For example, mining may break up consolidated rock in the watershed and replace it with unconsolidated material. The result of this change on watershed reclamation design is often a marked change in channel pattern. Channels exploit weak portions of consolidated rock and tend not to form on more resistant portions, that is, the channel pattern has structural control. In the disturbed, unconsolidated material, the channels may form anywhere. Streams that previously had patterns that followed cracks in the consolidated rock can now form a more random pattern in the unconsolidated material. A different drainage pattern with greater drainage density may be expected in the unconsolidated, disturbed material for these reasons.

The effects of land leveling, whether for road building, urbanization, agriculture, or other purpose, may be nearer to the land surface and may not affect structural rock as much as an activity like mining might. The adverse effects of these land disturbances can still be unacceptable. Often these activities result in a decrease in drainage density and associated diversion of runoff from several watersheds into another watershed that is not adjusted to that flow. Runoff water may accumulate in undesirable parts of the leveled land and an undersized receiving watershed may respond to an increased flow with erosion and excess sediment production. Reclaiming these lands disturbed by leveling with an appropriate channel pattern and drainage density can mitigate the effects of the prior disturbance.

GeoFluv™'s default drainage pattern is a dendritic pattern, because this "branching tree" pattern is the type that typically forms in unconsolidated materials (Bloom, 1978; Dunne and Leopold, 1978), such as those existing at a disturbed site. Drainage patterns other than the dendritic pattern generally express structural controls related to

rock (or soil) mineralogy. Streamflow will not tend to maintain variation from the dendritic pattern when reclaiming unconsolidated materials without reestablishment of a structural control, e.g., rock-lined stream banks. Installing structural controls will add cost, will establish a point of weakness subject to attack by flowing water, and can cause disruption in the flow regime up- and downstream of the structure that will require compensation in the channel designs there.

Determining appropriate reclamation drainage density

GeoFluv™ suggests a default drainage density value, but the user can, and must, determine site-specific values to achieve landform stability comparable to surrounding natural land. By using empirically determined drainage density values in GeoFluv™'s input, the user can have a very high degree of confidence that the resulting design will behave similar to the areas from which the drainage density measurements were taken.

The user can determine a desired range of site-specific drainage density values. Local drainage density measurements taken from the undisturbed land with earth materials similar to the project area, and from nearby areas with earth materials that are similar to the project's disturbed earth materials, can define the range. The drainage density measured on undisturbed earth materials provides a lower end-of-range value, while the drainage density measured on nearby areas similar to the project's disturbed materials provides an upper end to the range of desirable drainage density input values.

The recommended procedure for determining drainage density values is to visit the field site with a map and to mark the location and length of each valley feature that, if it were to erode into a finished reclamation landscape, would be large enough to be considered undesirable. Many of these features that will be identified in the field would not be apparent when examining a 7.5 minute quadrangle. It is important to recognize that when different individuals determine drainage density values for the same stable watershed, their results will vary. One individual may map a slightly smaller feature and generate a greater drainage density value than another observer, or vice versa. For this reason, it is recommended that the same individual make all the determinations for design consistency.

Entering Drainage Density Values

The user draws a GeoFluv Boundary and sketches a draft channel pattern inside the boundary. The appropriate pattern for unconsolidated, disturbed material is typically a dendritic pattern. The channels should be spaced so as to divide the GeoFluv™ work area into roughly equal portions. The user then identifies the GeoFluv Boundary in the Setup tab by pressing the Select GeoFluv Boundary button and then selecting the boundary on screen with the cursor. The software calculates and displays the watershed area.

The user then clicks on the Select Main Channel button to identify which channel segment in the drainage pattern will collect discharge from all the tributary channels and convey it out of the watershed at the watershed's base level. Regrade will display on the Output tab the length of the main channel selected and calculate the GeoFluv™ work area drainage density that the user has sketched. If the reclamation drainage density is too low, the natural watershed response would be to erode material until the appropriate drainage density is achieved. If the software's indicated value is too low as compared to the desired design value, the user can lengthen or add channel segments until the desired drainage density is attained. Conversely, if the indicated value is too high, the user can shorten or remove channel segments. If the design drainage density is too high, erosion is not likely, the landform may even be more resistant to erosion, but earthwork costs would increase beyond that which is necessary to create a landform as stable as surrounding natural lands.

Sinuosity

Sinuosity is the ratio of channel length to valley length. A stream flowing in unconsolidated material will typically begin to meander as it flows down slope. Because of this, the distance that the stream flows is greater than the straight line distance from the stream's head to its mouth. Sinuosity is calculated using units of length over length (L/L) and is a dimensionless value greater than 1.0 when any meandering is present. After the user has input the channel pattern and accepted a pattern with the desired drainage density, this software will then draw a draft channel pattern with suggested sinuosity appropriate to the channel slope. Channels on steeper slopes generally are less sinuous than those on lower gradients in stable land forms. The user may edit the draft channel's sinuosity value

using the Channels tab's 'Current Channel Settings...' button.

Channel Longitudinal Profile

Following the development of the channel pattern with sinuosity, GeoFluv™ calculates channel longitudinal profiles for each channel in the draft drainage pattern. The longitudinal profile of a natural channel is typically concave (Dunne & Leopold, 1978), steeper gradient in the headwater reaches and lower gradient near the channel mouth. That is because the headwaters of the watershed have less area, and therefore generate less runoff and erosive energy than the reaches near the channel mouth. Steeper channel gradients can be stable in the upstream reaches and lower channel gradients are appropriate in the downstream reaches for this reason. Stable slope profiles also tend toward this profile as can be seen in Figure 5.



Figure 5. Concave longitudinal profiles in stable natural slopes

Determining appropriate channel reclamation longitudinal profile

GeoFluv™ designs the longitudinal profiles for the draft landform to grade concave profiles to each local base level. For example, the main valley bottom channel in the draft GeoFluv Boundary work area grades to the user-input local base level (the lowest elevation in the design's main channel, typically where all runoff leaves the GeoFluv Boundary). GeoFluv™ grades each valley wall channel, at its confluence with the main valley bottom channel, to the main valley bottom channel slope at their confluence. The headwater slope for the design profile can be automatically determined by the elevation of the design's GeoFluv™ Boundary and a default distance from that boundary over which the ridgeline can have a convex profile and be stable, or can be user specified using the Channels tab's 'Current Channel Settings...' button.

Entering Longitudinal Profile Values

When the user identifies the GeoFluv Boundary in the Setup tab by pressing the Select GeoFluv Boundary button and then selecting the boundary on screen with the cursor, GeoFluv™ uses the boundary elevation to calculate a channel head elevation for each channel in the watershed from the Surface for Elevations file. The user specifies the three dimensional surface file that GeoFluv™ will use as a beginning surface from which to create its fluvial geomorphic landform design using the Surface for Elevations button. Examples of Surface for Elevations include existing post-disturbance topography designs or pre-disturbance topography. The user can type in the file path and name or use the browse button to help locate the desired file. The user may also enter Head Elevation and Base Elevation values using the Channels tab's 'Current Channel Settings...' button manually to gain accuracy; this is highly recommended for the base level elevation. The base level elevation has great effect on watershed response and an interpolated elevation may vary by several feet from the actual elevation. If the sketched channel begins beyond the default maximum distance from the GeoFluv Boundary, a pop-up warning will advise the user of this condition. The user may then either extend the channel to be within the default value or reset the default distance using the Settings button if local conditions permit a greater distance without erosion. The result of GeoFluv™'s longitudinal gradient solution is a network of sinuous channels that have concave profiles and smoothly transition from steeper headwater gradients to the gradient at the design watershed's local base level elevation. Figure 6 shows

an example of a natural stable network of slopes and channels with concave longitudinal profiles graded together from steeper ground to a lower gradient valley bottom.



Figure 6. Stable natural channels and slopes grade from steep to flatter gradient by a network of concave longitudinal profiles

The Profile button allows the user to review the design longitudinal profile for the current channel. It displays the beginning and ending channel elevations, the profile, and by moving the cursor, stationing is depicted along the profile along with the elevation and slope at that station. The viewer allows for vertical exaggeration to aid work on lower relief channels. The viewer also has toggle settings for pan/zoom and tick mark options. The *Natural Regrade* drop-down menu has powerful editing commands for channel and slope longitudinal profiles for special situations.

Channel Cross-section

GeoFluv™ calculates the channel cross-sectional profiles for channel reaches. The bankfull width (Dunne and Leopold, 1978) (Rosgen, 1996) for the mean annual flow is used to create a hydrologically balanced cross section. GeoFluv™ uses the input runoff coefficient, maximum water velocity, 2-yr, 1-hr storm precipitation, and width to depth ratio values to create this cross section. As the watershed area increases downstream, more water is present in the channels and the channel cross sectional area must increase to convey the discharge within the user-specified design velocity range. Other channel pattern dimensions, i.e., meander length, meander belt width, radius of curvature, related to the bankfull discharge (Williams, 1986) increase concurrently. GeoFluv™'s cross sectional area increase occurs simultaneously with the other channel dimensions. Channel flood-prone area has been related to a 50-year recurrence interval event (Rosgen, 1996) and GeoFluv™ uses this value to design the flood-prone area of the channel. The resulting dimensions define the channel banks for the draft landform. The designer can get cross section information by station for any channel in the GeoFluv design using the Channels tab's Report button. The range of design dimensions can also be seen using the Output tab's Summary Report button. A reviewer can get cross section information by station for any channel in the GeoFluv™ design from a completed drawing using the *Natural Regrade* dropdown menu's GeoFluv™ Channel Cross-section Report command.

Ridges, Slopes, and Volumes GeoFluv™ designs ridgelines between the channels at elevations that create side slopes less than a default 5:1 gradient for the draft landform. The Preview button in the Output tab will display the location of the main ridges, and the subridges and subridge valleys that form around the channel bends. The user may alter the elevation and placement of the ridgelines to adjust slope gradient and material balance. The Draw Design Surface button in the Output tab is used to contour the ridgelines and channels to reveal the draft landform. The Save Design Surface button in the Output tab saves the landform drawing as a file.

Figure 7 shows a reclamation project midway through construction that used this fluvial geomorphic approach. The mine pit highwall that ends at the graded gray spoil used to continue trending to the right of the figure and then turned ninety degrees toward the lower right of the figure. The steep slope reclamation with four subwatersheds immediately to the right of the end of the pit is the same slope shown in Figure 2 above. This project was designed over a period of months without the benefit of the computerized software; using *Natural Regrade* with GeoFluv™, the design time would be measured in hours.



Figure 7. Fluvial geomorphic reclamation is underway at the 115-acre Cottonwood Reclamation Project, Farmington, New Mexico. Gray colored material is mine spoil being graded using fluvial geomorphic approach. The U.S. Department of the Interior awarded San Juan Coal Company "National" and "Best of the Best" reclamation awards for 2004 for this project.

GeoFluv™ calculates and displays the material balance needed to create the draft landform. The GeoFluv™ design's material balance is calculated by comparing the GeoFluv™ design surface to the surface file identified as the Comparison Surface. The user could compare the Design Surface to the pre-disturbance surface, another post-disturbance reclamation surface design, the existing disturbed surface, or other surface file. Figure 8 is an example output dialog box that compares the cut to the fill needed to create the landform.

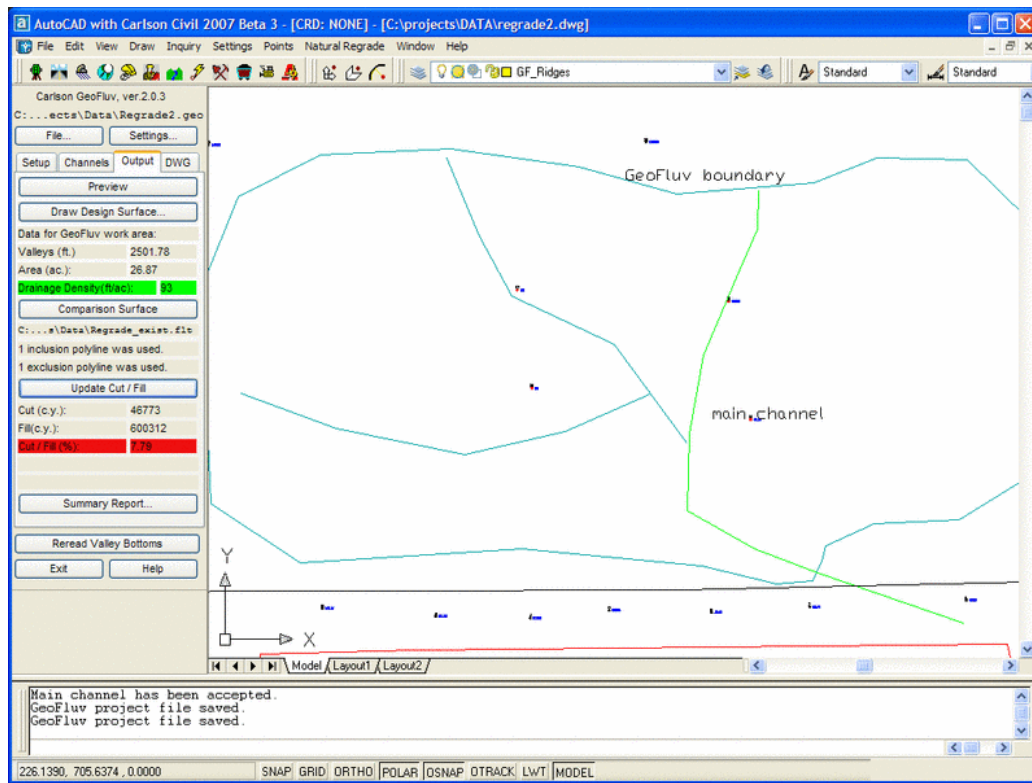
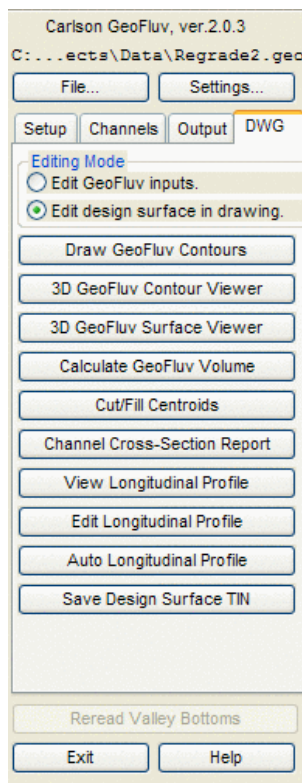


Figure 8. Output dialog box gives immediate cut/fill balance to guide landform design editing

Note that the Output tab also displays the overall drainage density with the GeoFluv work area, which can be compared with the current channel drainage density. The user can compare the drainage density that is displayed in the Channels tab for each channel's subwatershed to the overall watershed drainage density to verify that the drainage density is uniform throughout the watershed. Subwatersheds that have too great or small drainage density values can be corrected by editing ridges or channels to vary areas or channel lengths.

The Output tab and the Summary Report show whether or not the balance is within the user-specified tolerance. The Summary Report button in the Output tab generates a report for the channel showing the design parameter values of the channel and valley slopes. The report also displays the parameter values for natural channel types in the Rosgen stream classification scheme for ready comparison.

The draft landform is an idealized solution to creating a stable landform according to fluvial geomorphic principles based on the user-specified input values. The user may modify the draft landform, for example to reduce the fill volume by lowering a ridgeline using the Edit Longitudinal Profile or Auto Longitudinal Profile commands, and the software can almost instantaneously recalculate the cut/fill balance to meet the user's design.



The DWG tab lists the tools for analyzing the Design Surface as it is represented in the drawing in the channels and ridges layers (GF_Channels and GF_Ridges by default). These are the same commands that are in the Natural Regrade menu plus the addition of Save Design Surface TIN. The Save Design Surface TIN command will save a TIN file of the current Design Surface (as it is represented in the drawing) and is simply the built channels and ridges within the GeoFluv boundary. The Editing Mode toggle helps to clarify the difference between editing a GeoFluv input and editing a Design Surface in the drawing.

The Fluvial Geomorphic Characteristics of the Draft Landform

The fluvial geomorphic characteristics of the draft landform are those that are compatible with unconsolidated materials placed at various slopes, subject to particular storms, and considering special limitations of typical reclamation sites. Those special limitations include a relatively thin topsoil veneer over mixed earth materials (spoil), equipment limitations (e.g., equipment grading capability versus design grading requirements, ability to traverse steep slopes), and desire to minimize cost and maintenance.

The GeoFluv™ fluvial geomorphic approach to building stable reclamation landforms is centered on creating a network of ephemeral drainage channels and associated slopes that are in a state of quasi-equilibrium, i.e., that are "stable." Natural ephemeral channels are the landscape's response to runoff events. By definition, they flow now only in response to direct precipitation. However, they may have formed in response to greater precipitation during wetter climatic conditions, including glacial periods, when they may have flowed as intermittent or even perennial streams. Considering that, their water and sediment transport characteristics would be expected to be consistent with streams that flow perennially in the present climate.

For this discussion, we will use the Rosgen classification scheme for natural channels. The Rosgen scheme classifies perennial streams according to major types based on slope, width to depth ratio, entrenchment ratio, and sinuosity (see glossary for definitions), and stream bed material particle size (1 through 6, where 1=bedrock and 6=silt/clay). The Rosgen classification scheme describes natural channels as major types A through G using characteristics of multiple and single thread channels that form in different geologic settings.

Slope is generally considered the dominant characteristic, and only the type A and A+ channels are associated with

slopes greater than -0.04. For this reason, the A and A+ channel types have a place in many reclamation landscape designs.

Some of these channel types, such as the multiple-thread D and single-thread F and G, are associated with high bank erosion rates and sediment transport and deposition. They tend to exist as transitional channel forms as the channel moves towards a more stable type and for this reason are not favored for use in a stable reclamation landscape design.

The B type is a step-pool stream with low sinuosity, with the steps typically formed by resistant rock strata and narrow rock canyon walls limiting sinuosity. Because both of these structural elements are typically gone in a reclamation landscape, the B type channel is not favored in stable reclamation landscape design either.

The remaining major types, the C and E, differ mainly in their width to depth ratios and sinuosities, and the stable E type's association with dense bank vegetation. The low width to depth ratio of the E-type develops where the combination of cohesive bank material and a dense network of roots from bank vegetation are present. The E type channel is stable, but is very sensitive to disturbance of its bank material and vegetation. The C type has a tendency toward lateral migration through the process of erosion at the cut bank and deposition on the point bars, a tendency that is also exacerbated by bank material and vegetation disturbance.

From this discussion, it can be concluded that the characteristics of the A, C, and E major types have advantages for stable reclamation channel design. Further, the major channel types do not exist as distinct and separate entities, but in an evolutionary continuum from one type to another. For example, a B5c stream would have the major characteristics of a B channel with dominantly sand-size material, but its flatter slope, greater sinuosity, and width-to-depth and entrenchment ratios, would be tending toward those associated with type C streams. The flatter slope of this stream type combined with its greater sinuosity can allow it to transport and balance its water and sediment loads by channel geometry and not require the structural drops associated with the major B-type's step/pool sequences. Its width-to-depth and entrenchment ratios are such that all but extreme events may remain within a flood prone area within its channel banks. When its hydraulic design is correct and its banks are sufficiently protected by vegetation, natural appropriate channel roughness, and bank protection such as rock deflectors or J-hook vanes, this channel type can convey water and sediment discharge with minimal changes to the channel pattern. In other words, the channel can maintain its course and not erode into its banks and through the relatively thin veneer of topsoil because sediment transport and deposition occurs within the channel. When floods greater than the flood-prone capacity (based on entire 50-year, 6-hr recurrence interval precipitation introduced to the channel instantaneously in the GeoFluv™ approach) occur, the additional discharge energy can be rapidly dissipated on an adjacent floodplain.

The default channel-type settings create type A and A+ channels at slopes greater than -0.04 and type Bc channels for slopes less than -0.04 using lower-range channel geometry values for these types. The user can also optionally choose to randomly vary the channel geometry values within the acceptable range using the Channels tab's 'Current Channel Settings...' button. GeoFluv™ displays the channel geometry values for all channels and reaches of longer channels in the watershed in the Summary Report. The user can then edit the channel settings within the ranges of value appropriate to the default channel types or vary the settings to use different channel types.

Links with Other Software

The *Natural Regrade* with GeoFluv™ computerized landform design can further improve operational efficiency by interfacing with computerized machine guidance software. The GeoFluv™ landform design can literally be sent from the designer's computer screen to the machine operator's guidance screen by radio transmission and the designs can be implemented "on the fly". Design editing can also be done expeditiously. For example, if unforeseen conditions emerge, such as shallow bedrock near the edge of disturbance that hinders a dozer cut, the designer can keep the operator working elsewhere, adjust the channel's and related subwatershed design during the shift, and return the operator to work on the revised design.

Additional efficiency can be achieved by integrating the computerized landform design with software, such as Carlson Software's *Productivity Tools*, that provide real-time equipment monitoring and data capture during construction.

This software determines material movement volumes and distances over time for associated equipment. The information that this software provides to decision makers was previously not available and can help them identify the most efficient operational methods for maximum cost savings. *Carlson Telescope* and *Starnet* are two office monitoring products that allow for viewing heavy equipment in real-time, as well as enabling two-way real-time file transfer.

GPS guidance and machine control software can virtually eliminate the need for survey stakeout in the field and greatly enhance production efficiency. The machine operators can follow the project design and complete it to grade on their own as they work. Carlson Software makes *Carlson Dig* for excavators and shovels, *Carlson TruckPro* for haul trucks, *Carlson Autograde* for dozers, loaders, compactors, motorgraders, scrapers, foreman trucks, etc., *Carlson Drillstar* for drills, and *Carlson Grade* that allows cross-platform, multi-equipment functionality.

The link to the RIVERMorph two-dimensional data collection and channel design software allows for near instantaneous conversion of the RIVERMorph user's design into a three-dimensional design. The RIVERMorph two-dimensional software follows natural channel design methods, as developed and taught by Dave Rosgen. The user can render and view the three-dimensional channel design made in *Natural Regrade* with all the Carlson capabilities, rotating, tilting, quick volume calculations of earthwork or pools, etc. The RIVERMorph designs are made into .tin files by default and can be also be made into .flt or grid file formats. The .flt and .tin file formats are commonly used by machine control software, so the *Natural Regrade* design is ready to load into machine control equipment; construction can begin without additional surveying or staking! The RIVERMorph designs can be inserted into any GeoFluvTM design to produce an integrated natural river and upland landform design.

Software Compatibility

The *Natural Regrade* module with GeoFluvTM is a module of the Carlson Civil / Survey family. As such, it functions in tandem with the widely-used AutoCAD drafting software. Carlson Civil / Survey is application software for the civil engineering, surveying, mine engineering, and GIS disciplines that use AutoCAD as the graphics engine and drawing editor. Carlson Civil / Survey's system requirements are no greater than that of the AutoCAD version with which it operates and will work with any AutoCAD-based product from AutoCAD 2000 through AutoCAD 2009 as well as IntelliCAD.

Data Entry

The Carlson Civil / Survey software accepts data downloads from any data collector, or other data file. Once the data are imported, they are stored as a coordinate (.crd) file. The entire *Natural Regrade* module project can then be designed from the .crd files without leaving Carlson Civil / Survey.

DWG Tab

The DWG tab functions to help the user separate tasks that are done using the GeoFluv method in Natural Regrade from other tasks that are done in a CAD environment that do not relate to the GeoFluv method. For example, the user may make minor edits to ridgelines and not upset the projects hydrologic balance whereas editing channel profiles can result in knickpoints and immediate erosion consequences. The command buttons are ordered from top to bottom in logical sequences for working with a draft GeoFluv design that is made when running the Draw Design Surface button in Output tab.

Check Ridgeline Slope: The Check Ridgeline Slope command allows the user to inspect the sub-watershed ridge and swale toe slopes in the entire design. This is useful because it is possible that the user input settings: the Surface for Elevations, GeoFluv boundary line, valley input lines, Global settings and Current Channel Settings may result in areas where the resulting valley wall slopes do not drain into the channels, but instead the storm water runoff runs sub-parallel to the channel for some distance. This could result in rill or gully erosion. The

Check Ridgeline Slope command helps the user identify areas where this may be happening and determine why it is happening so that the design may be edited to correct the problem.

When the command is run, a pop-up dialog gives the user the options to generate a report, draw symbols corresponding to the slope values in the report, and to specify the type of symbol used and the layer it is drawn on, and to scale the symbol. After the user specifies the preferences and clicks, OK, the command runs and displays the specified information.

The naming convention for the lines is to identify the sub-watershed, e.g., main L2R1 (note L and R are capitalized when identifying channels and their subwatersheds), followed by the sub-watershed ridge or swale line identification where lower case 'r' indicates a sub-watershed ridge line and a lower case 's' indicates a sub-watershed swale line. The sub-watershed ridge or swale line letter is followed by a number indicating its sequential order from the head of the sub-watershed. Using this convention, the 'main L2R1 r1' designation identifies the first sub-watershed ridge on the right valley wall (r1) of the sub-watershed that holds the first tributary entering from the right valley wall (R1) of the second tributary entering from the left valley wall (L2) of the main channel.

Summary

Computerization of this fluvial geomorphic approach to land reclamation makes the applied science from a relatively obscure body of knowledge available to a wide range of users. The approach helps the designer to build the landform that would tend to form under the existing physical conditions. The benefits of this approach include stability against erosion, hydrologic function and plant and animal habitat that are similar to undisturbed natural lands, lower construction cost for steep slopes, mitigation of maintenance concerns, and improved aesthetics. This approach also links with machine control and management software to further improve the efficiency of land reclamation. Table 1 summarizes the advantages gained by using the *Natural Regrade* software with GeoFluv™ to reclaim disturbed lands.

Old method	<i>Natural Regrade</i> software with GeoFluv™
Based on conveying a single extreme discharge event	Based on all discharges
Conveys only water discharge effectively at lower Q	Natural channel morphology conveys water and sediment discharge; hydrologic balance
Requires expensive off-site earth material, e.g., rip-rap	Built with on-site materials
Expensive on steep slopes	Cost is significantly lower than gradient terraces and down drains on steep slopes
Requires long-term maintenance	Self-maintaining
Requires maximum backfill to lower slopes	Can reclaim steep slopes in stable and suitable configurations, save money on material moving
Provides minimal slope aspect diversity	Increased slope aspect diversity promotes vegetation success and animal habitat
Visual affront	Natural beauty
Rigid design sideboards limit landscape alternatives	Landscape designs can vary and provide alternatives
Regulatory agencies not satisfied	Regulatory agencies embrace*

Table. 1. Comparison of old methods of reclamation to landscape designed using this computer-aided fluvial geomorphic design method.

*Approach is award-winning: New Mexico *Innovative Practices- 2001*, US Dept. of Interior "*National*" and "*Best of the Best*" reclamation for 2004.

These benefits will be realized for those designing mine reclamation, subdivisions, golf courses, ski areas, parks,

etc.; any site where the land surface has been disturbed. For example, storm water catchments do not have to be a rectangular pond surrounded by a chain link fence, but can serve the storm water control purpose and also be a hydrologically functioning and esthetically pleasing park. This technology will help designers, developers, and regulators evaluate more options, help companies save production and bond dollars, and promote better land reclamation and use.

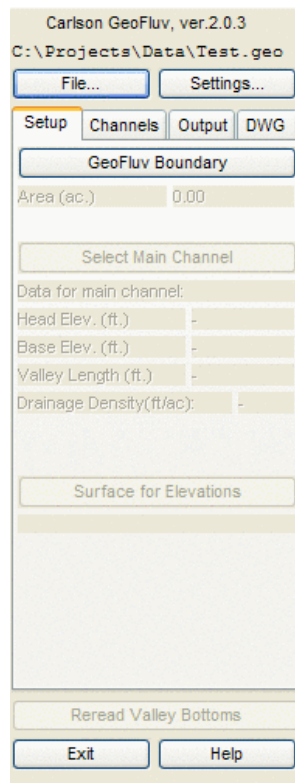
Documentation References

- Bloom, A.L., 1978. *Geomorphology, a systematic analysis of late Cenozoic landforms*. Prentice Hall, New Jersey. 510 p.
- Bugosh, N. 2003. Innovative reclamation techniques at San Juan Coal Company (or why we are using fluvial geomorphic principles and otherwise doing our reclamation differently), at Rocky Mountain Coal Mining Institute national meeting, July 2003, Copper Mt., Colorado.
- Bugosh, N. 2003. Stream channel design reclamation - The fluvial geomorphic approach to hydrologic reclamation, pre-conference workshop at joint conference of the Billings Land Reclamation Symposium and the Annual Meeting of the American Society of Mining and Reclamation, June 2003, Billings, MT.
- Bugosh, N. 2002, Reclamation using fluvial geomorphic principles (or why we are doing our reclamation differently), at Office of Surface Mining Bond Release Forum, August 2002, Bismarck, ND.
- Bugosh, N. 2002. Innovative reclamation practices promoting successful bond release at San Juan Coal Company, NM at Office of Surface Mining Bond Release Forum, August 2002, Bismarck, ND.
- Bugosh, N. 2002. Fluvial geomorphic principles applied to mine reclamation at New Mexico meeting of Rocky Mountain Coal Mining Institute, April 2002, Farmington, NM.
- Bugosh, N. 2000. Fluvial geomorphic principles applied to mined land reclamation at OSM Alternatives to Gradient Terraces Workshop, January 2000, Farmington, NM.
- Dunne, T. and L.B. Leopold, 1978. *Water in environmental planning*. W. H. Freeman and Company, San Francisco. 796 p.
- Rosgen, D., 1996. Applied river morphology. *Wildland Hydrology*, Pagosa Springs, Colorado, 343 p.
- Williams, G.P., 1986. River meanders and channel size. *Journal of Hydrology*, v. 88, Elsevier Science Publishers B.V., Amsterdam, pp. 147-164.

Natural Regrade Menu

Design GeoFluv Regrade

The Design GeoFluv Regrade command on the *Natural Regrade Menu* is used to open the dockable dialog box and access the main GeoFluv™ design commands.



The main GeoFluv™ input edit boxes, buttons, and commands are arranged in GeoFluv™'s main dockable dialog box in four tabs, Setup, Channels, Output, and DWG. The edit boxes and buttons are arranged in the input and operational sequence that the user will usually follow to make a GeoFluv™ design. The general order is left-to-right through the tabs and top-to-bottom within each tab. Prerequisite commands are further noted in the GeoFluv™ dockable dialog box by making prerequisite commands/inputs active (clear image) while subsequent command/inputs remain inactive (faded image) until the prerequisite step is performed.

Pulldown Menu Location: Natural Regrade

Keyboard Command: gf

Prerequisite: Polyline perimeter

Natural Regrade File

The File button at the top of the Design GeoFluv Regrade dockable dialog box provides a convenient way for the user to save GeoFluv™ projects. The File button can be clicked at any time. When the user left-clicks on the File button, the Open and Save Projects dialog box appears.

This dialog box gives the user the options to create a "New" GeoFluv™ project file, "Open" an existing GeoFluv™ project file, or "Save As..." a new project file. Saving the various designs as separate project files allows the user to store and retrieve each design alternative with all of its settings intact. If the user creates a project file, either with the New button or the Save As button, then all changes made to that design will be saved automatically to the project file. If a project file is never selected, then the settings are lost when the drawing is closed or when the dockable dialog box is closed and a new drawing is opened.

Every line and point (and in general, "entity") has a name in AutoCAD. The GeoFluv™ project file remembers the names of the GeoFluv Boundary polyline and the valley bottom polylines that are in the drawing rather than storing complete copies of all the coordinates in those polylines. The advantage of this method is that when the GeoFluv Boundary or the valley bottoms change, the GeoFluv™ design will reflect the changes automatically and will never be inconsistent with the drawing.

Natural Regrade gives the user the ability to rapidly create many design alternatives according to fluvial geomorphic principles for a stable landform. The user can then compare the various GeoFluv™ landform design alternatives considering the how well the designs satisfy land use objectives, and practicality and overall material handling costs.

From these comparisons the user can decide on an optimal design. Saving the design alternatives as projects using the File button saves the user from having to repeat data entries.

Natural Regrade Global Settings

These settings include variables that will remain constant for a GeoFluv™ design, e.g., precipitation event values, and other detail settings specific to a GeoFluv™ design area that the user will typically not change for each design iteration.

The Settings button at the top of the Design GeoFluv Regrade dockable dialog box provides a convenient way for the user to access all these settings in one place. The Settings button is accessible at any time. Left-clicking on the Setup tab's Settings button opens the Natural Regrade Global Settings dialog box. Each of the settings is described below.

Maximum distance between connecting channels (ft.): This is a drawing setting that defines a maximum separation of polylines that Natural Regrade will recognize as channel polylines. The maximum distance should be set as small as the user can comfortably draw. The user types this value into the edit box. Some users may be able to hold and click the cursor more accurately than other users and this setting accommodates those differences.

Maximum distance from ridge line to channel's head (ft.): This is an essential local variable in the GeoFluv™ approach. It is the shortest distance from a ridgetline to the head of a stable channel. The user will determine this value, in the vicinity of the GeoFluv project, for stable landforms that developed in earth materials similar to the disturbed earth materials within the GeoFluv™ boundary. The value is a function of local factors including soil cohesiveness, vegetation canopy, cover, and root density, storm intensity and other climatic factors, and topographic relief. The user types this value into the edit box. The 80-foot default value is for an erosive semi-arid, high-altitude desert region in the southwestern United States.

Maximum Distance for Subridge: The user can vary the Global sub-watershed ridge length in the Natural Regrade Global Settings dialog. The sub-watershed ridges and swale valleys form by similar processes of erosion as do the channel-containing valleys, but the watershed area above them is not great enough to generate discharges that form defined channels. This means that in some areas the landform dimensions of these smaller features may have greater convex lengths than in channel-sized valleys. The user can input a maximum convex length for the sub-watershed ridges that will be applied globally in the GeoFluv design by typing the desired value in the edit box; the remainder of the profile will be concave when possible. The swale distance setting is not changed when the sub-watershed ridge setting is manually input.

Maximum Distance for Swale: The user can vary the Global sub-watershed swale length in the Natural Regrade Global Settings dialog. The sub-watershed ridges and swale valleys form by similar processes of erosion as do the channel-containing valleys, but the watershed area above them is not great enough to generate discharges that form defined channels. This means that in some areas the landform dimensions of these smaller features may have greater convex lengths than in channel-sized valleys. The user can input a maximum convex length for the sub-watershed swales that will be applied globally in the GeoFluv design by typing the desired value in the edit box; the remainder of the profile will be concave when possible. The sub-watershed ridge distance setting is not changed when the swale setting is manually input.

Slope at the mouth of the main valley bottom channel (%): This setting, along with the Channels tab's "Advanced...Specify mouth elevation." setting, may be the most critical value to creating a stable final landform design using the GeoFluv™ approach. The GeoFluv™ design must integrate with upstream and downstream areas to achieve stability. That means that GeoFluv™ channel reaches must have longitudinal profiles that blend smoothly with up-and downstream channel reach profiles. The user will determine the slope downstream of the mouth of the main valley bottom channel by survey, e.g., a point every 25 feet for about 400 feet. The user can plot a longitudinal profile from these points and select an input value for the channel slope upstream that will blend smoothly into the downstream profile. The user types this value into the edit box.

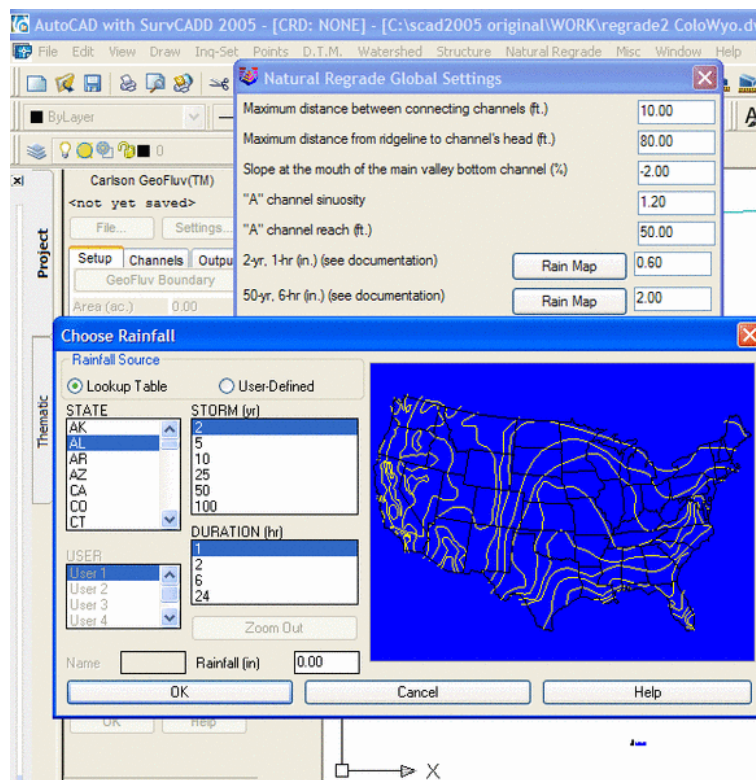
In some cases, disturbance may continue for a great distance downstream of the GeoFluv™ boundary. In those

cases, the user must determine the slope at the eventual downstream, undisturbed tie-in point, extend that profile upstream to the GeoFluv™ boundary, and specify a smooth tie-in slope value.

"A" channel sinuosity: This setting applies to channel reaches with slopes >0.04 . The channel types that form in these steeper reaches have low sinuosity, <1.2 . Steeper reaches may be expected to have lower sinuosity still. Sinuosity will typically increase as slope decreases (inversely proportional). This setting allows the user to specify a maximum sinuosity <1.2 , which may be desirable in certain cases, e.g., for very steep, short channels the user may want to specify a lower value. The user types this value into the edit box.

"A" channel reach (ft.): This is an essential local variable in the GeoFluv™ approach, and reflects many of the same local variables as does "Maximum distance from ridge line to channel's head" above. It is one-half of a meander length. The user will determine this value for stable landforms, in the vicinity of the GeoFluv™ project, that developed in earth materials similar to those within the GeoFluv™ boundary. The user types this value into the edit box.

2-yr, 1-hr (in.) (see documentation): This is where the user inputs the precipitation value for the storm event that determines the bankfull channel dimensions and plan-view channel geometry. This is an essential local variable in the GeoFluv™ approach. The GeoFluv™ approach uses a 2-yr, 1-hr storm event to design these features for ephemeral upland channels, and it can be used for ungauged intermittent and perennial channels as well. The value can be typed into the edit box, or entered by using the Rain Map button for sites in the U.S. and Puerto Rico.



Clicking on the Rain Map button produces a dialog box in which the user will select the state or territory of interest, and the storm frequency and duration using dropdown menus. When the user selects a state or territory, Rain Map zooms in to that selection. The user then moves the cursor to the GeoFluv™ project area, left-clicks on it, and the interpolated value is entered into the "Rainfall (in.)" field in Rain Map. When the user clicks on the OK button at the bottom of the Rain Map dialog box, the dialog box closes and the rainfall value is automatically entered into the Natural Regrade Global Settings precipitation event edit box (next to the Rain Map button).

If the user has data from a stream gauging station sufficient to determine the discharge associated with an annual recurrence interval (bankfull) event, they can directly enter that value into GeoFluv™ using the "Channels" tab's "Advanced . . ." button and "Use manual Qpk.". The user is cautioned that increasing these values beyond the actual event value will not create a "design safety factor", but rather will create a channel that is not competent to transport

sediment during more frequent, lower-discharge events, i.e., it will cause sediment deposition in the channel that can cause channel blockage, etc.

50-yr, 6-hr (in.) (see documentation): This is where the user inputs the precipitation value for the storm event that determines the flood-prone channel dimensions. This is an essential local variable in the GeoFluv™ approach. The user can input the value using methods discussed in the "2-yr, 1-hr" section above.

The dominant channel morphology has been shown to be related to about a 50-year recurrence interval event, rather than some extreme 100-year, 200-year, probable maximum, or other event. The GeoFluv™ approach uses an intense 50-year event to design the flood prone area of the channel and places the entire amount of the 6-hour storm into the channel instantaneously to calculate a peak discharge associated with extreme channel-forming events. All this discharge is contained within the channel banks in Natural Regrade's default GeoFluv™ channel design. The user can also design a floodplain or terrace adjacent to the channel to accommodate greater discharges; when these are relatively wide the tremendous increase in cross sectional area allows the additional discharge to spread across the surface without causing undesirable erosion.

Target drainage density (ft./ac.): This is an essential local variable in the GeoFluv™ approach. It is the total valley length divided by the area within the GeoFluv™ boundary. The user will determine this value for stable landforms, in the vicinity of the GeoFluv™ project, that developed in earth materials similar to those within the GeoFluv™ boundary. (Refer to Introduction section for more detail.) It is a function of local factors including soil cohesiveness, vegetation canopy, cover, and root density, storm intensity and other climatic factors, and topographic relief. It is the extent to which a drainage network will develop given those local factors to achieve a stability comparable to surrounding land areas. The user types this value into the edit box.

Target drainage density variance (%): This is an essential local variable in the GeoFluv™ approach. It captures the range of acceptable drainage density values for the GeoFluv™ project area based on the range of locally-measured drainage density measurements as described above and in the Introduction section. For example, if the lowest drainage density value that the designer has determined can be stable is 80 feet/acre and the highest drainage density value that is measured in similar local earth materials is 120 feet/acre, the user could set a target drainage density value of 100 feet/acre and a target drainage density variance at 20 percent to capture that locally-determined range of 80 to 120 feet/acre. The user types this value into the field.

Force ridges to be lower than GeoFluv boundary: This toggle setting allows the user to specify if any point on a main ridge line can be higher than where the ridge line meets the GeoFluv™ boundary. A main ridge line in the GeoFluv™ approach is a ridge that defines a subwatershed divide within the GeoFluv™ boundary. When this box is checked (toggled on), the elevations on the main ridgeline will all be lower than the elevation on the GeoFluv™ boundary where the main ridgeline intersects the GeoFluv™ boundary. When this box is left blank (toggled off), a ridgeline may have a high point of greater elevation than the GeoFluv™ boundary, e.g., creating a knob or butte feature. In this case all runoff will still remain within the GeoFluv™ boundary and exit at the mouth of the main valley bottom channel. The feature allows the user to create a topographic feature within the GeoFluv™ boundary that may vary from the pre-disturbed surface, but is still a stable landform. This can be used to minimize the movement of piles of earth material when creating a stable landform design.

Angle from subridge to channel's perpendicular, upstream (deg.): This is a local variable in the GeoFluv™ approach. This is the angle that subridgelines make in the upstream direction from the valley bottom to the main ridgeline. The user will determine this value for stable landforms, in the vicinity of the GeoFluv™ project, that developed in earth materials similar to those within the GeoFluv™ boundary. Setting this value similar to surrounding stable landforms will help the GeoFluv™-designed landform blend harmoniously with surrounding natural landforms. Natural Regrade will automatically create all the subridges in the draft landform using this value. The user can edit individual ridgeline orientations from the draft landform to suit site-specific design needs, however, Natural Regrade's ability to create the subridges and subridge valleys saves the user a tremendous amount of design time when producing draft landform designs. The user types this value into the edit box.

North or East straight-line slopes (%): This setting allows the user to specify a global target for a maximum ridge-to-toe slope profile steepness on the north- and east-facing slopes (between 315 and 135 degrees). The north and east-facing slopes are generally steeper in natural landforms because they get less sun and can retain more moisture, factors that favor tree growth and its associated root-binding of slope soils. Because the slope faces can contain an

infinite degree of aspects in the GeoFluv™ approach, Natural Regrade does not produce the identical slope angle for all north or east-facing slopes with this setting; it is rather a best-fit slope adjustment toward the specified target value. The user should remember that changing the slope on one side of a ridgeline will affect the slope on the other side of the ridgeline. The user types this value into the edit box.

Maximum straight-line slopes (%): This setting allows the user to specify a global target for a maximum ridge-to-toe slope profile steepness on all slopes within the GeoFluv™ boundary. As in "North or East straight -line slopes" above, it is impractical for Natural Regrade to control every area of the design with this setting; it is a best-fit slope adjustment toward the specified target value. The user should remember that changing the slope on one side of a ridgeline will affect the slope on the other side of the ridgeline. The default value is 33 percent because agricultural machinery commonly in use for land reclamation has difficulty working across steeper slopes. The user types this value into the edit box.

Maximum cut & fill variance (%): This setting allows the user to specify a global maximum for the cut and fill material balance for the GeoFluv™ design surface as compared to another surface, e.g., Pre-disturbed surface. A value of 100 percent means cut and fill are balanced. A value of 125 percent means that there is 25 percent more material that needs to be removed to create the surface than there are areas requiring fill. The user types this value into the edit box.

Minimum cut & fill variance (%): This setting allows the user to specify a global minimum for the cut and fill material balance for the GeoFluv™ design surface as compared to another surface, e.g., Pre-disturbed surface. A value of 100 percent means cut and fill are balanced. A value of 80 percent means that there is 20 percent less material that needs to be removed to create the surface than there are areas requiring fill. The user types this value into the edit box.

Cut swell factor: This setting allows the user to specify a global swell factor for cut material from bank volume to loose volume. A value of 1.000 means that the loose volume is the same as the bank volume. A value of 1.120 means that the excavated loose material fills 12 percent greater volume than did the same material in place before excavation. The user types this value into the edit box.

Fill shrink factor: This setting allows the user to specify a global shrink factor for fill material from loose volume to fill volume. A value of 1.000 means that the fill volume is the same as the loose volume. A value of 0.900 means that the filled and settled/compacted material fills 10 percent less volume than did the same loose material after excavation. The user types this value into the edit box.

Channel head elevation tolerance: Natural Regrade alerts the user via pop-up dialogs when channels designed from the user-input valley lines on the Surface for Elevations would require changes in elevation or slope to make smooth concave profiles. The alerts can be very precise, to decimal point values, but the user may not be concerned to be alerted for extremely small variances. The channel head elevation tolerance setting allows the user to input an elevation variation that the user finds significant, for example if the user is concerned about lowering or raising the channel elevation by 0.3 units, they would enter 0.3 in the edit window and pop-up alerts will only appear when greater variances exist. Note: When the project area is relatively large, even small differences in elevation over the area can result in significant changes in material volumes.

Channel head slope tolerance: Natural Regrade alerts the user via pop-up dialogs when channels designed from the user-input valley lines on the Surface for Elevations would require changes in elevation or slope to make smooth concave profiles. The alerts can be very precise, to decimal point values, but the user may not be concerned to be alerted for extremely small variances. The channel head slope tolerance setting allows the user to input a head slope variation that the user finds significant, for example if the user is concerned about starting the channel at plus or minus two percent from the specified head slope value, they would enter 2.0 in the edit window and pop-up alerts will only appear when greater variances exist.

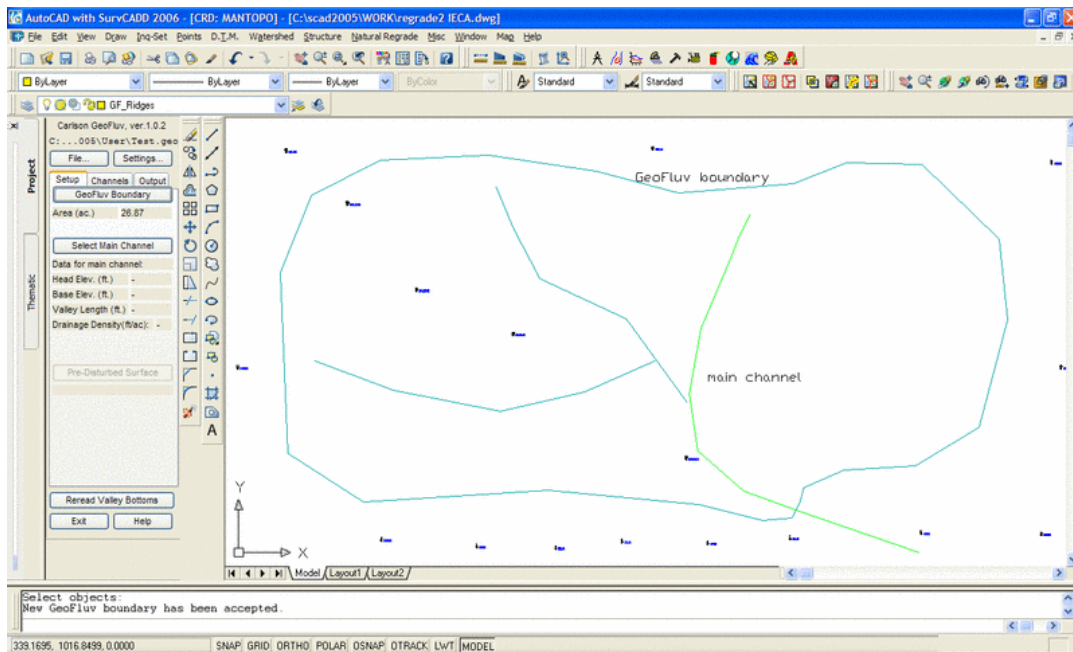
Setup Tab

The Setup tab is used for work that defines the watershed boundary, establishes the general channel pattern, calculates the watershed drainage density, channel head and mouth elevations, channel slopes, and defines the three

dimensional surfaces that GeoFluv™ will use to create the stable draft design watershed and calculate the material balances for the design.

Select GeoFluv Boundary

The user defines the boundary polyline that they have drawn to outline the GeoFluv™ design area on the drawing. GeoFluv™ will automatically calculate and display the watershed area inside the defined boundary. (This area is only the area to which the GeoFluv™ design will be applied. If additional undisturbed or previously reclaimed area lies upstream of the GeoFluv™ design area, runoff from that additional area can be added into the design in the Channels tab using the Add Area window.) The default area units are acres, but metric units can be selected in the Carlson Drawing Setup dialog box accessed through the Carlson toolbar at Settings/Configure /Drawing Setup (use of consistent units in a drawing is recommended).



Command Prompt:

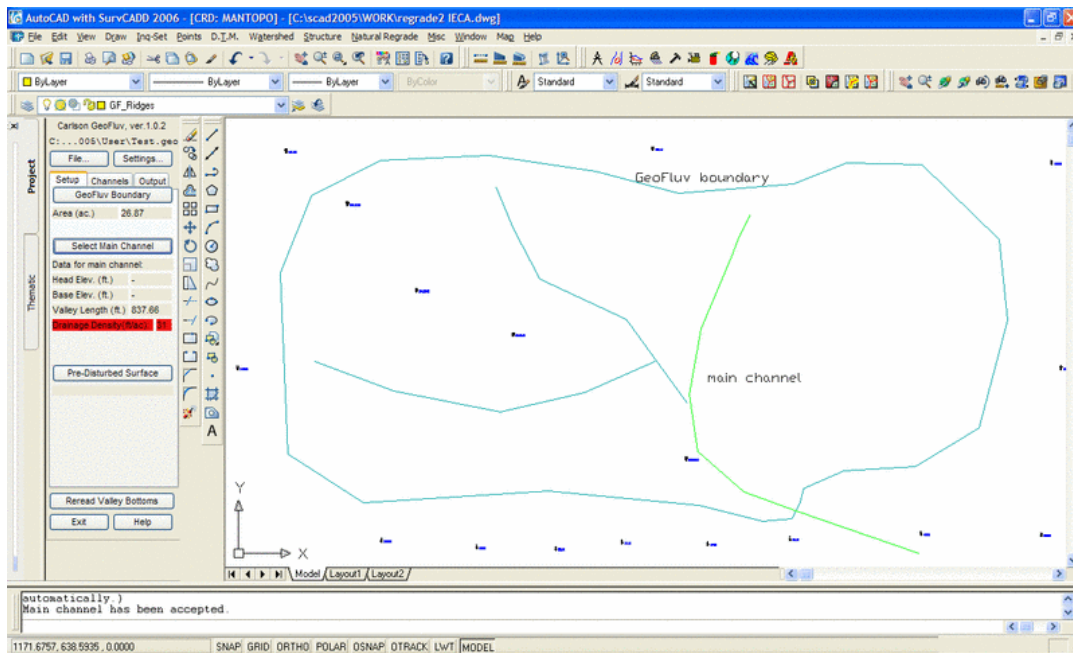
Select GeoFluv boundary polyline. Select objects

The user moves the cursor to cross the GeoFluv Boundary polyline anywhere along its length and left-clicks to define the GeoFluv Boundary to the Natural Regrade module.

The command prompt reads, "New GeoFluv boundary has been accepted.", if the cursor has clicked on a closed polyline, the watershed area is calculated and displayed, and the Select Main Channel button on the Setup tab becomes active. The command prompt reads, "GeoFluv boundary polyline must be closed." if the polyline the user selected is not closed, and the watershed area is not calculated and displayed, and the Select Main Channel button remains inactive.

Select Main Channel

The user defines the channel that is the main valley bottom channel draining all discharge to the watershed's base level. The user also specifies the point at which the main valley bottom channel makes its transition from a lower gradient (<0.04) channel type to a steeper gradient (>0.04) channel type. The Select Main Channel button becomes active after the user has selected the GeoFluv Boundary polyline.



Command Prompt

Select main valley bottom channel polyline. Select objects: The user moves the cursor to cross the main valley bottom channel polyline anywhere along its length and left-clicks to define the main valley bottom channel to GeoFluv™.

When executed correctly, the **command prompt** reads, "**Choose the forced transition point between channel types. (Press Enter to find automatically.)**" if the cursor has covered and left-clicked a closed polyline. The user then either specifies the forced transition point between channel types (reaches with >-0.04 and <-0.04 slope) by placing the cursor crosshairs on the valley bottom polyline and left-clicking, or presses enter to allow GeoFluv to find a transition point based on the Pre-Disturbed Surface file elevations.

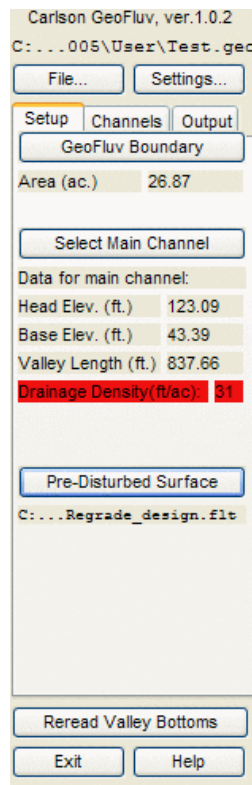
Command Prompt:

Choose the forced transition point between channel types: The user moves the cursor crosshairs to the point at which he wants the main valley bottom channel to transition to its steeper (A/Aa+ type) reach and left-clicks to select that point. Alternately, the user can press the Enter key and GeoFluv will determine a transition point using the Pre-disturbed file data. When executed correctly, the main channel length and drainage density for that channel in the entire GeoFluv Boundary area are calculated and displayed in the "Data for main channel:" fields and the reads, "Main channel has been accepted." GeoFluv will compare this drainage density to the target value in Settings and, will highlight the value in red if it is too low or will highlight the value in green if it is within the acceptable range. If the value is too low, the user can lengthen the main channel, decrease the GeoFluv Boundary area, or add more channels using the Channels tab's Add button.

The command prompt reads, "Main channel polyline must not be a closed polyline" if the polyline the user selected is closed, and the channel length is not calculated and displayed. The command prompt reads, "Main channel must cross watershed boundary" if the user has clicked on a segment that does not leave the watershed at its base level, and the channel length is not calculated and displayed.

Data for Main Channel

The "Data for main channel" fields display information that GeoFluv™ will use to create the main valley-bottom channel that conveys all runoff from within the GeoFluv Boundary downstream to the base level elevation. The GeoFluv™ design is built headwards from the base elevation.



Head Elev. (ft.) displays the elevation at the head of the main valley bottom channel. GeoFluvTM can determine a head elevation from the three-dimensional surface entered using the Pre-Disturbed Surface button. The head elevation will appear in this field when the user enters a three-dimensional surface file. Alternatively, the user may specify an elevation using the Channels tab's Advanced button to access the "Specify head elevation" option on the 'Channel "main" Advanced Settings' dialog box.

Base Elev. (ft.) displays the elevation at the that is the local base level for the main valley bottom channel. GeoFluvTM can determine an approximate base-level elevation from the three-dimensional surface entered using the Pre-Disturbed Surface button. The approximate base-level elevation will appear in this field when the user enters a three-dimensional surface file. **This elevation is adequate for creating draft GeoFluvTM designs, but the user must use an accurate field-surveyed base-level elevation for the final design.** The user may specify the field-surveyed base-level elevation using the Channels tab's Advanced button to access the "Specify mouth elevation" option on the 'Channel "main" Advanced Settings' dialog box. (See also Settings tab's "Natural Regrade Global Settings", Slope at the mouth of the main valley bottom channel (%) for this related critical setting.)

Valley Length (ft.) displays the length of the main channel that the user has identified using the Setup tab's Select Main Channel button. After the user inputs the transition point from the main channel's headwater reach (slope >-0.04) and its valley bottom reach (slope <-0.04), GeoFluvTM will display the length of the selected main channel in this field and use the value to calculate the main channel subwatershed drainage density.

Drainage Density displays the drainage density value for the main valley bottom channel subwatershed as determined from the main channel Valley Length (ft.) and the GeoFluv Boundary Area (ac.). The drainage density is displayed in units of feet/acre in U.S. units, a convenient unit for landform design work.

If the drainage density is within the variance that the user specified in the Design Natural Regrade dockable dialog box's Settings button, Natural Regrade Global Settings, "Target drainage density variance (%)", then the value will be highlighted in green; if the value is outside the user-specified variance it will be highlighted in red. A red warning can mean that the drainage density is too high or too low. If too high, the channel can be shortened or the GeoFluv Boundary area decreased. If the value is too low, the main channel can be lengthened or more channels can be added using the Channels tab's Add button.

Pre-disturbed Surface

The user defines a three-dimensional surface file that GeoFluv™ will use as the reference surface from which the fluvial geomorphic surface will be designed. This surface file could be an existing Approximate Original Contour map, a pre-disturbance map, or any other surface from which the user wants to begin the design.

When the user clicks on the Surface for Elevations button, the Use Existing TIN or Create New TIN? Pop-up dialog box appears to give the user the options, by clicking on radio buttons, for entering a 3-D surface file. The user can type the file name into the File Name field or browse for the file by left-clicking the Browse button to the right of the File Name field. Left-clicking the browse button will open another dialog box that will display file selections and the directory path. The user can search for the surface file in various directories and left-click the Open button when the desired 3-D surface file is found.

When Open is clicked in the file search dialog box, the search dialog disappears and the Carlson Software 3D Viewer window will appear showing the .tin for the area that the user has defined. The user can rotate, fill in the .tin triangles, etc. to preview the .tin to be sure that it is satisfactory, i.e., no unwanted holes, elevation spikes, etc. When the user is satisfied that the .tin correctly represents the comparison surface, the user closes the 3D viewer window and the file name is entered into the File Name field in the Select Surface for Elevations TIN File - (tin;flt) dialog box, the file name appears below the Surface for Elevations button, details of the file loading will be listed on the command line: Loadings edges . . . , the number of points and triangles that were loaded will be listed. Regrade will also conduct an automatic save of the file and the path will be stated. The user is then ready to continue to Channels tab.

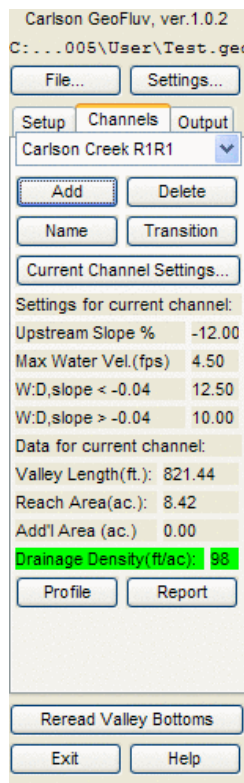
Alternatively, the user can click the radio button for Create new TIN file by selecting entities in the drawing. When the user selects this option and clicks OK, the cursor changes to a selection box that the user can use to define an area with a crossing window. (Note: it is very important that this area encompass the entire GeoFluv project .tin surface. If areas outside the GeoFluv boundary area may be used for material borrow and incorporated into the cut / fill balance, the .tin area should include those areas also.) After the user defines the .tin area and hits the enter key, the Carlson Software 3D Viewer window will appear as above. The remaining steps are also as described above.

When the Surface for Elevations file has been accepted, the file name is listed below the Surface for Elevations button, and GeoFluv™ reads the head and base level elevations from the Surface for Elevations file for the main channel that the user has sketched and displays these elevations in the "Data for main channel:" fields above the Surface for Elevations button. GeoFluv™ designs the main channel from this information and the settings on the Channels tab. The elevation at the heads of all other channels have their defaults set by the Surface for Elevations. The elevations along the GeoFluv Boundary are also set from this surface. This in turn sets the elevations of the main ridges and subridges and subridge valleys that intersect the GeoFluv Boundary.

Channels Tab

The Channels tab is used to input variables that GeoFluv™ will use to design channel geometry dimensions (including radius of curvature, meander length, meander belt width, sinuosity, and channel cross sections that are sized for bankfull and more extreme flood events), to add or delete channels from the design, to name channels, to view channel longitudinal profiles, and to design related upland landforms, and to generate reach-scale reports of the design characteristics of channels.

GeoFluv™'s draft design will have concave longitudinal channel profiles that join together in a smooth hydraulic transition. GeoFluv™ will automatically design drainage-divide ridges between the channels that form subwatersheds for each channel using this information. GeoFluv™ will also automatically design the sub-ridges and sub-ridge valleys in each subwatershed, and calculate and display the subwatershed area, subwatershed channel valley length, and subwatershed drainage density.



Channel Add

The Add button is used to add each channel to the watershed design.

Command Prompt:

Select tributary channel polyline. *Select objects*

The user selects the polyline that represents the valley bottom for a channel that is to be added to the GeoFluv™ design. The selected polyline must meet the following criteria:

- One end must be near another valley bottom polyline that has already been added to the design.
- The other end must be near the GeoFluv Boundary.
- It must not cross the GeoFluv Boundary.
- It must not cross any valley bottom polyline.
- It must not cross itself.
- It must not be closed.

When executed correctly, the command prompt reads, "Creating final design surface . . . DONE", GeoFluv™ adds the channel to the design, lists it in the Current Channel menu, retains the input settings from the previously added channel, designs the channel based on those input settings, and recalculates and displays the channel length, subwatershed area, and subwatershed drainage density. The user can view the channel's vertical curve profile and edit any of the channel design input settings. When the user left-clicks on a channel name in the Current Channel field, GeoFluv™ draws an arrow on the design pointing to that channel.

GeoFluv™ uses a consistent naming convention for channels, because this has been found to be a very important attribute for communication when taking designs to the field. Everyone involved with the project, from the designers to surveyors to equipment operators can clearly know what part of the project is being discussed when following this consistent naming convention, and this minimizes the chances of miscommunication and mistakes. GeoFluv™ uses the main valley bottom channel that drains to the watershed's local base level for the primary name, for example "Carlson Arroyo". The channels tributary to "Carlson Arroyo" have alphanumeric names that follow the hydrologic right- and left-bank convention, that is, right and left bank when facing downstream. The first tributary downstream of the headwaters entering "Carlson Arroyo" on its right bank is labeled "Carlson R1" and the first tributary entering on the left bank is "Carlson L1". This convention is applied to the tributaries themselves, so that the first tributary entering "Carlson R1" on its left bank is labeled "Carlson R1L1", and so on. GeoFluv™ will revise the naming

sequence so that when all the channels in the watershed have been added, each channel will be named correctly following this convention.

To name the channels differently, simply type the preferred name into the edit box in the Change Channel Name dialog box accessed using the Channels tab's Name button.

Channel Delete

The Delete button is used to delete the "Current Channel" from the watershed design.

First, the correct channel must be chosen in the dropdown list on the Channel tab. The user can change the current channel by left-clicking on the menu arrow to the right of the Current Channel list-box and selecting the name of the channel that is to be deleted. The user then left-clicks on the Delete button.

GeoFluv™ deletes that channel name from the list, deletes that channel's input settings, deletes any tributaries that connect to this channel, adjusts the names of the remaining channels according to GeoFluv™'s naming convention, recalculates the ridges and subwatersheds and other aspects of the GeoFluv™ design, and makes the main valley bottom channel to be the "Current Channel."

Channel Name

The Channels tab's Name button allows the user to change the name of a channel within the GeoFluv Boundary that has been named according to GeoFluv™'s default automatic naming convention.

Left clicking on the Name button causes the Change Channel Name dialog box to appear on the screen. The user types the new channel name into the "Channel's name:" edit box and left-clicks the OK button to apply the new name.

The user can toggle the Change Channel Name dialog box's option to "Update tributary channel names" on or off. This option will automatically rename all channel's that are tributary to the channel named in the "Channel's name:" edit box using that new name. Note that if the toggle is off, the selected channel will be renamed, but the channel tributary to it will not, but may need to be manually renamed if its name includes a GeoFluv™ alpha-numeric portion, i.e., if Moose Creek RE1 is renamed Spruce Creek, its tributary's name, Moose Creek R1R1, will still include the now meaningless R1R1.

If the user does not want to use the default GeoFluv™ channel naming convention, they can rename the channels simply by toggling off the Update tributary channels names option and typing in any name.

Channel Transition

This button allows the user to change the transition point from a reach greater than -0.04 slope to less than -0.04 slope in a channel's longitudinal profile.

Command Prompt:

Choose the forced transition point between channel types. (Press Enter to find automatically.)

When the user left-clicks on the Transition button, the cursor changes to a crosshair that the user can place on the valley line to specify a new transition point. The user can elect to allow GeoFluv™ to determine this point automatically based on the elevation data supplied in the Pre-Disturbed Surface file by pressing the Enter key.

Current Channel

The Current Channel list-box displays the name of the current GeoFluv™ channel that the user is designing. The main valley bottom channel was specified in the Settings tab and this channel is the first listed in the Current Channel tab. Channels that are tributary to the main valley bottom channel are added in the Channels tab using the Add button. The user can see a listing of all channels that are built by left-clicking on the down arrow to the right of the list-box. Then left-clicking on a channel name will make that channel be the "current channel".

Current Channel Settings

This button allows the user to specify settings that will vary the channel discharge and the related channel geometry and upland ridge and subridge morphology specific to the subwatershed active in the Channels tab current channel name box. The settings are organized on two tabs, Geometry and Watershed. The Geometry tab has settings for maximum velocity, upstream slope, downstream slope, width to depth ratio, sinuosity, random scale factors on sinusoidal channel, subridge spacing on sinusoidal channel, and channel head and mouth elevation. The Watershed tab has settings for runoff coefficient when using the Rational Runoff Method (the default method), or to allow input of discharge computed by an alternate method, and to add runoff from contiguous land areas.

Left-clicking on the "Settings" button brings up the "Channel 'xxxx' Settings" dialog box that gives the user the options shown below. The optional settings made in the "Channel 'xxxx' Settings" dialog box will apply only to the Channel 'xxxx' subwatershed. The blue subject bar at the top of the dialog box displays the name of the channel's subwatershed to which the Settings will apply. The user will select a different channel in the "Current Channel" window of the "Channels" tab and then left-click on "Settings" to make these changes to other channels and their subwatersheds, e.g., 'Channel yyyy', 'Channel zzzz,' etc. After specifying the settings in the dialog box, the user can apply them by left-clicking the "OK" button at the bottom of the dialog box.

Geometry Tab

Maximum Water Velocity (ft./s.): The user can specify a maximum water velocity for the channel by typing the desired value into the edit box. Velocity is inversely related to channel cross-sectional area for a given discharge according to the relationship $Q/a=v$, where Q is discharge (cubic feet per second), a is area (square feet), and v is velocity (feet per second).

Upstream slope %: The user can specify the upstream slope for the channel using this edit box. This feature can be used to vary the channel's longitudinal profile that will join to a mouth slope dictated by the receiving channel slope at their confluence. It can also be used to tie into the upstream slope when the headwaters of the channel are at the GeoFluv Boundary and join with an upstream channel slope draining "Additional watershed area."

Downstream slope % (Only adjustable on main channel.): The user can specify the mouth slope for the main channel at the GeoFluv Boundary to join smoothly to the downstream channel slope by typing the desired slope into the edit box. If the Channel's tab Settings dialog box is open for any tributary to the main channel, the edit box will read "n/a."

Width-to-Depth, slope > -0.04: xx.xx , < -0.04: xx.xx: The user can specify width-to-depth ratios for channels with slopes greater and less than -0.04 by typing the desired width-to-depth ratio into the edit box. The default values are 10.00:1 for channels with greater than -0.04 slope and 12.5:1 for channels with less than -0.04 slope.

Sinuosity, slope > -0.04: xx.xx , < -0.04: xx.xx: The user can specify sinuosity for channels with slopes greater and less than -0.04 by typing the desired sinuosity into the edit box. The default values are 1.15 for channels with greater than -0.04 slope and 1.48 for channels with less than -0.04 slope.

Random scale factors on sinusoidal channel: The meander pattern of the idealized draft valley bottom channels (<-0.04) will be determined by mathematical constants and thus will be very uniform, changing (enlarging) as a function of flow (related to discharge) and valley bottom orientation. Checking the 'Random scale factors on sinusoidal channel' box will randomly vary the constant values, within their acceptable ranges for stable channels, such that radius of curvature, meander length, and meander belt width vary. This random variation produces a more natural appearance for the channel and related upland landforms.

Subridge spacing on sinusoidal channel: This setting applies to channels with slopes <-0.04. The lower-gradient channels, with slopes <-0.04, may have an adjacent floodplain (or terrace) area and the uplands landform may begin some distance from the channel banks. The user can use this setting to create some of this open floodplain or terrace area by increasing the spacing between subridges. A subridge spacing setting of 3, for example, will create a subridge on every third meander bend of the channel with an opening for the floor of the subridge valley between these subridges.

Note: The user must select odd-number spacing; specifying even number spacing will result in all subridges and subridge valleys being on opposite sides of the valley. Even spacing can be made with manual Carlson editing.

The user can also manually add or delete subridges, or vary subridge longitudinal profiles using Natural Regrade's longitudinal profile editors, to introduce more variation to the draft GeoFluv's landform.

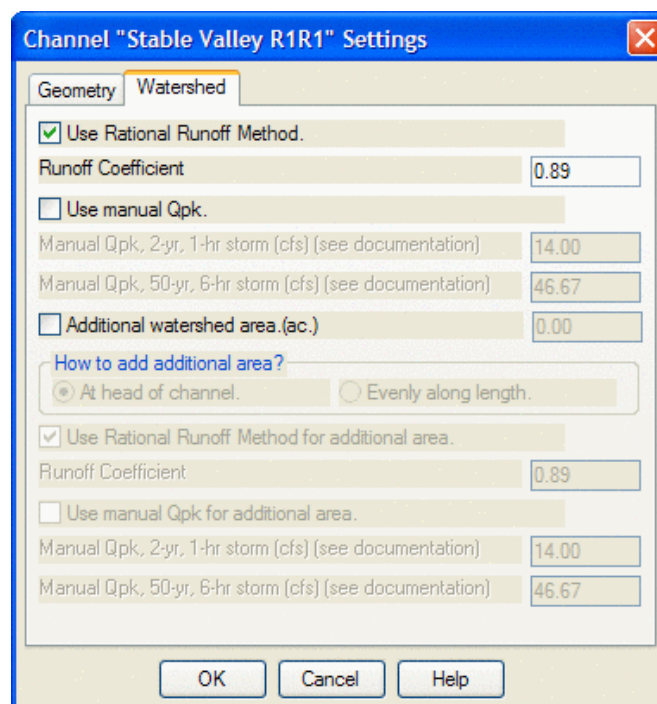
Specify head elevation: The user can specify the head elevation for any channel, rather than accepting an elevation that is automatically determined from the Pre-disturbance file specified in the Settings tab. The user checks the box to select this option and then proceeds in one of two ways. The user can type a desired headwater elevation into the Specify Head Elevation field. Alternately, the user can left-click on the Pick button and then identify a (COGO) point of the desired elevation on the drawing. To use the Pick method, the user left-clicks the cursor near the desired point and then, by moving the cursor diagonally, creates a box around the point. The user left-clicks again to define the opposite corner of the box surrounding the desired point and the point elevation is entered into the Specify Head Elevation field.

Specify mouth elevation: The user can (and should) specify the mouth elevation for the main channel only. This setting becomes inactive on the tributary channels because their mouth elevation is controlled by the main channel's longitudinal profile. The procedures for setting the elevation are the same as in Specify Head Elevation above.

Note: The user should specify the mouth elevation of the main channel in the GeoFluv's project area because this elevation and the channel slope immediately downstream of this point may be the most critical variables for assuring a stable landform design. The elevations that Natural Regrade interpolates from the 'Pre-disturbed surface' specified in the Settings tab are appropriate for creating and comparing draft design alternatives, but a channel mouth elevation interpolated from a map surface can vary from the actual elevation on the order of feet. A channel will be expected to adjust to elevation and slope inaccuracies by erosion.

Specify sub-ridge/swale distance: The user can vary the sub-watershed ridge and swale length for each project sub-watershed in the Natural Regrade Channel's tab's Current Channel/Geometry tab settings dialog. The sub-watershed ridges and swale valleys form by similar processes of erosion as do the channel-containing valleys, but the watershed area above them is not great enough to generate discharges that form defined channels. This means that in some areas the landform dimensions of these smaller features may have greater convex lengths than in channel-sized valleys. The user can input a maximum convex length for the sub-watershed swales and ridges that will be applied to all the design lines in the subwatershed by typing the desired value in the edit box; the remainder of the profile will be concave when possible. The sub-watershed ridge distance setting is not changed when the swale setting is manually input.

Watershed Tab



Use Rational Runoff Method: This is the default setting for calculating runoff to the GeoFluv boundary; channels in Natural Regrade and is the setting that will be used when the box is checked. The Rational Runoff Method calculates a peak discharge using the formula $Q_{pk} = CIA$, where C is the runoff coefficient, I is the rainfall intensity, and A is the acreage. The user enters the appropriate runoff coefficient for the area within the GeoFluv boundary in the Runoff Coefficient field and Natural Regrade does all the related calculations.

Use manual Qpk: The user can choose to input a peak discharge value calculated by some other method by checking the 'Use Manual Qpk' option. When the user checks this box, the runoff coefficient field in the Use Rational Runoff Method setting (and use of that method) becomes disabled. The user then types in the peak discharges to use for the two storm events.

Note: The GeoFluv approach uses the 2-yr, 1-hour storm event to calculate bankfull discharge and the 50-yr, 6-hr event to calculate a flood-prone discharge. Reclamation landforms constructed using the GeoFluv approach that use these inputs have been stable in a very harsh and erosive high-altitude desert environment through extreme storm events. Using other input values may give unsatisfactory results.

Additional Watershed Area: This setting allows the user to incorporate runoff from contiguous lands into the GeoFluv Boundary. When the user checks the Additional Watershed Area box, the fields below become active and offer a choice of how the additional runoff will enter the GeoFluv Boundary. If the head of the GeoFluv channel is downstream of the Additional Watershed Area, as when joining to an upstream channel reach, the user should select the "At head of channel" option. The GeoFluv channel's headwater dimensions will then be sized to accommodate the runoff from the area above the channel headwaters within the GeoFluv Boundary and the Additional Watershed Area upstream of that. If the Additional Watershed Area is subparallel to the GeoFluv channel, checking "Evenly along length" will introduce the runoff from the Additional Watershed Area gradually along the GeoFluv channel reach and the channel dimensions will increase proportionately along the reach. The remainder of the settings are as described above in "Use Rational Method" and "Use manual Qpk."

Data for Current Channel

These are the data that GeoFluv™ uses to calculate the drainage density for the subwatershed containing the channel in the Current Channel field above, and a display window that informs the user if discharge from additional area outside the GeoFluv Boundary is entering the subwatershed. These are not input fields, but real-time displays of the values being used in the GeoFluv™ design for the Current Channel. GeoFluv™ calculates these values from the subwatershed boundary it has built and the drainage pattern that the user sketched, and user-input additional area by the Channels tab's "Advanced . . ." button.

Valley Length (ft.) This value is the straight-line length of the valley (in feet in U.S. units), not the sinuous length of the channel.

Reach Area (ac.) This value is the area (in acres in U.S. units) draining water to that channel or channel reach.

Add'l Area (ac.) This field displays any additional area outside the GeoFluv™ boundary that is draining into the GeoFluv™ Current Channel. This could be contiguous undisturbed land or land that has already been reclaimed. The Channels tab's "Advanced . . ." button allows the user to add additional area.

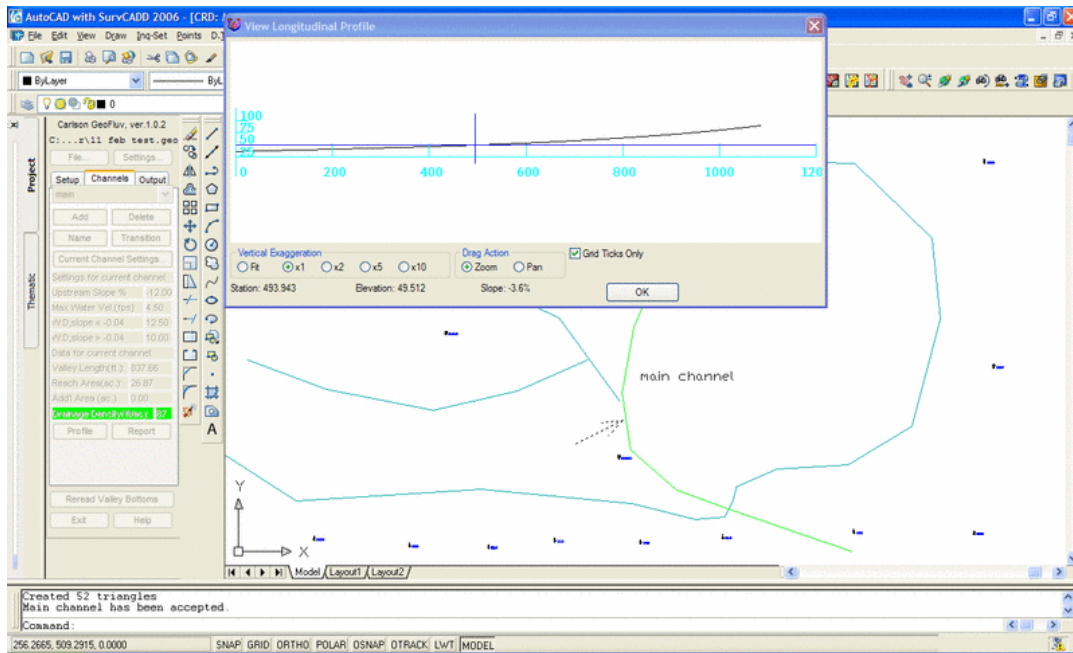
Drainage Density (ft/ac) This value is the ratio of Valley Length to Reach Area in U.S. units of feet per acre, a convenient unit for this parameter in landform design work.

If the drainage density is within the variance that the user specified in the Design Natural Regrade dockable dialog box's Settings button, Natural Regrade Global Settings, "Target drainage density variance (%)", then the value will be highlighted in green; if the value is outside the user-specified variance it will be highlighted in red. A red warning can mean that the drainage density is too high or too low. If too high, the channel can be shortened or the Current Channel's subwatershed area increased. If the value is too low, the Current Channel can be lengthened, or a tributary channel can be drawn in the Current Channel subwatershed and then added using the Channels tab's Add button, or the subwatershed area can be decreased.

If the GeoFluv Boundary polyline or the valley bottom polylines are modified in the drawing, clicking on the Reread Valley Bottoms button will cause the above data to be updated.

Profile

This button activates a popup view window, Profile Viewer, which allows the user to see the Current Channel's longitudinal profile graphically. Moving the viewer cursor along the profile allows the user to obtain the station, elevation, and slope at any point indicated by crosshairs along the profile.



Vertical Exaggeration This toggle button setting allows the user to select profile views at fit, 1x, 2x, 5x, and 10x vertical exaggeration.

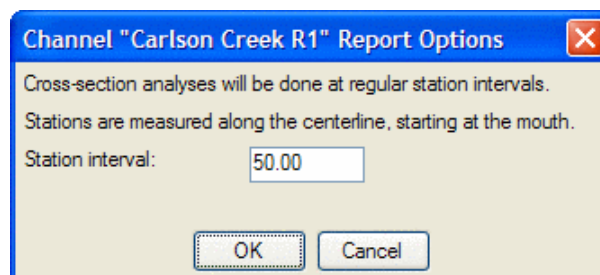
Drag Action This toggle button setting allows the user to select either zoom or pan drag action. Then the user selects the Zoom button, holding down the left-click button on the mouse as the mouse is moved will zoom in and out on the profile display. Similarly, when the user selects the Pan button, holding down the left-click button on the mouse allows the user to pan the profile in the display. Whether in Zoom mode or Pan mode, the middle mouse button can be held down to pan the profile in the display. Thus, if the user has a middle mouse button, staying in the Zoom mode and using the middle mouse button to pan is most efficient.

Grid Ticks Only This toggle button setting allows the user to select either the default x and y grid lines across the profile, or tick marks only on the axes.

Report

The Report button is used for detailed inspection of the GeoFluv™ Current Channel's design characteristics.

The report is for the channel as it would be incorporated into the design after using the Draw Design Surface button on the Output tab. Note that the channels with slopes > -0.04 do not have the meandering channel geometry relationships that lower slope channels do. Their different characteristics are designed differently in the GeoFluv™ approach and because of this radius of curvature, meander length, meander belt width, and meander width ratio are not listed.



The user left-clicks on the Report button and the "Channel 'xxxx' Report Options" dialog box opens. The user can type in the desired station interval for the report in the "Station Interval:" window. The user then left-clicks on the OK button and the report for that channel is generated according to the user-specified stationing.

```
Cross-section reports are done every 50.00(ft.).
Stations are measured along the centerline, starting from the headwaters.
Stations are only reported when there is a real cross-section
formed by the four polylines of the built channel.
The length of the centerline is 908.85(ft.).
Left and right are from the point of view of looking downstream.
```

```
-----
station (ft.):          0.00
slope at station:      -0.12
flood prone width (ft.): 0.39
flood prone depth (ft.): 0.04
flood prone area (sq.ft.): 0.01
bankfull width (ft.):  0.17
bankfull depth (ft.):  0.02
bankfull area (sq.ft.): 0.00
bottom width (ft.):    0.03
Shields shear stress, bankfull width (lbs/sq.ft.): 0.07
Shields shear stress, flood prone width (lbs/sq.ft.): 0.17
right side slope (%):  25.00
left side slope (%):   25.00
-----
```

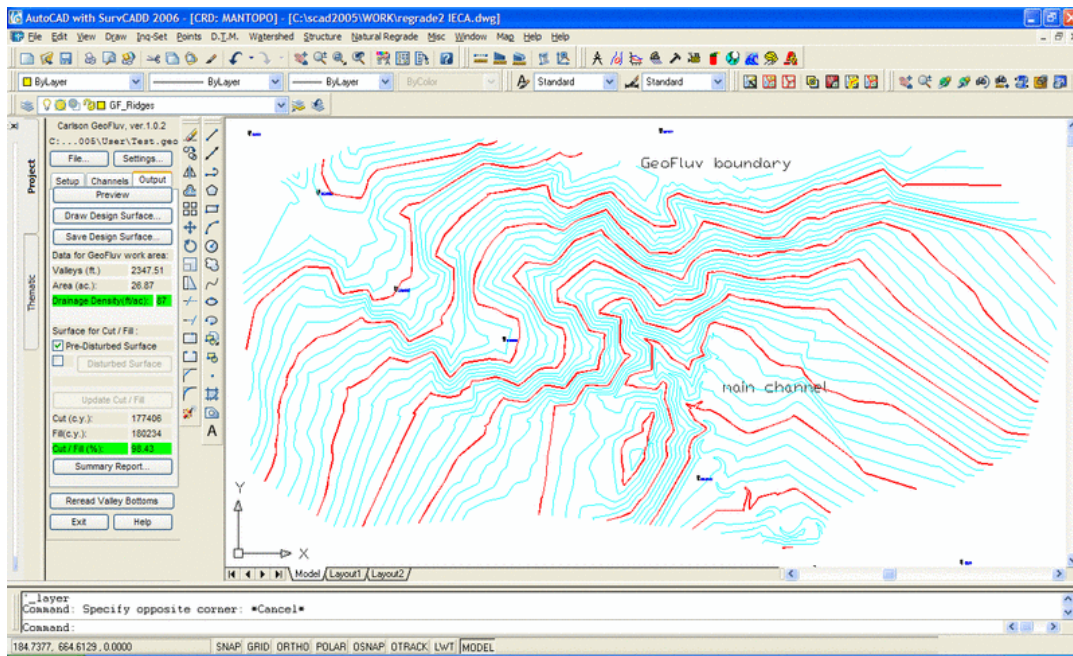
```
-----
station (ft.):          50.00
slope at station:      -0.12
flood prone width (ft.): 1.45
flood prone depth (ft.): 0.17
-----
```

The report gives summary design information for the Current Channel and lists channel-reach detailed information at the user-specified station intervals. The stationing increases in the downstream direction with station 00 being the channel head.

Output Tab

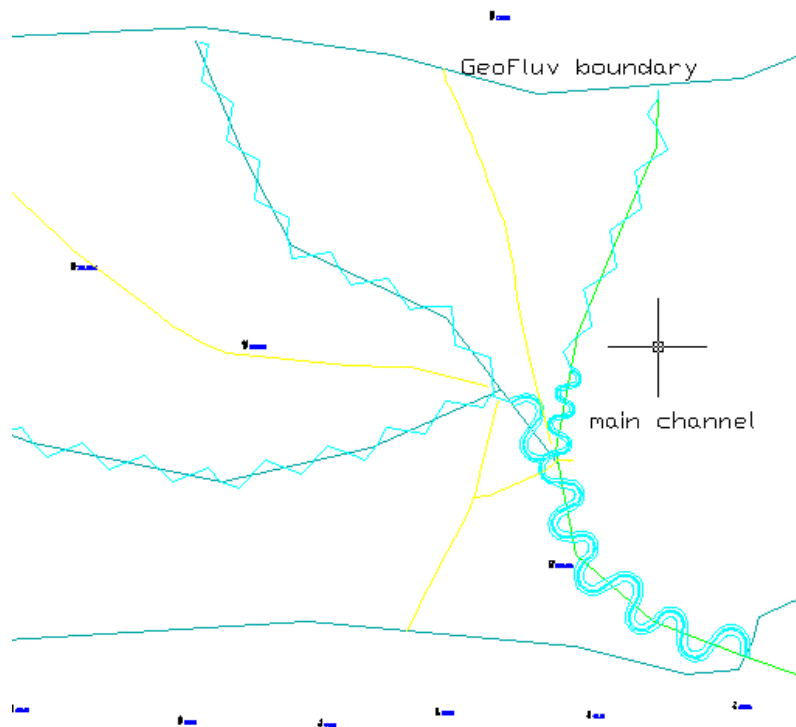
Function

The Output Tab allows the user to preview the channels and ridgelines that will be contoured to reveal the draft GeoFluv™ design landform, to draw and contour the surface, to save the design, to verify the average drainage density within the GeoFluv Boundary, to compare the cut and fill volumes needed to create the design and to verify that the cut and fill volumes balance within a user-specified limit, and to create and view a summary report of the channel settings and design dimensions.



Preview

The Preview button displays the channel and main ridgelines from which GeoFluv™ will base its draft landform design.



Command Prompt:

The command reads, "Preview. Use View menu commands to change views. Press Enter to continue." when the Preview button is left-clicked.

Left-clicking on the Preview button draws the main GeoFluv™ channel and ridge design lines on the drawing. The A and A+ channel reaches (>-0.04 slope) are displayed as zig-zag lines. The valley bottom channel reaches (<-0.04 slope) are displayed as sinuous curved lines. The main ridgelines are shown between the tributary channels and are

sub-parallel to the channels.

Data for GeoFluv Work Area

These are the data that GeoFluv™ uses to calculate the drainage density for the entire area within the GeoFluv Boundary. These are not input fields, but real-time displays of the values being used in the GeoFluv™ design for overall project area within the GeoFluv Boundary. GeoFluv™ calculates these values from the GeoFluv Boundary that the user has drawn and the drainage pattern that the user sketched.

Valleys (ft.) This value is the combined straight-line length of all the valleys (in feet in U.S. units), not the sinuous length of the channel, that the user has sketched within the GeoFluv Boundary.

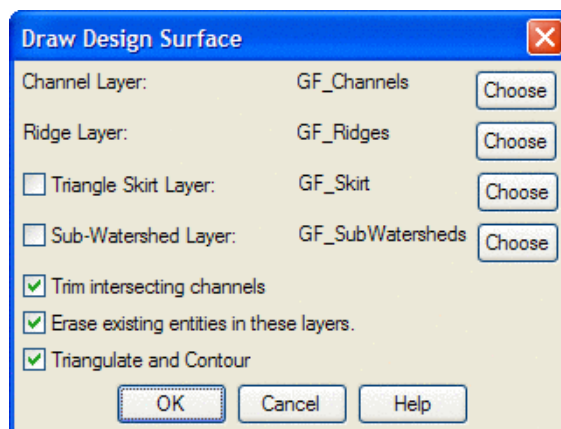
Area (ac.) This value is the area (in acres in U.S. units) within the GeoFluv Boundary.

Drainage Density (ft/ac) This value is the ratio of Valley Length to Reach Area in U.S. units of feet per acre, a convenient unit for this parameter in landform design work.

If the drainage density is within the variance that the user specified in the Design Natural Regrade dockable dialog box's Settings button, Natural Regrade Global Settings, "Target drainage density variance (%)", then the value will be highlighted in green; if the value is outside the user-specified variance it will be highlighted in red. A red warning can mean that the drainage density is too high or too low. If too high, channels can be deleted or shortened, or the area within the GeoFluv Boundary can be increased. If the value is too low, valleys can be lengthened, or a tributary channel can be drawn within the GeoFluv Boundary and then added using the Channels tab's Add button, or the area within the GeoFluv Boundary can be decreased.

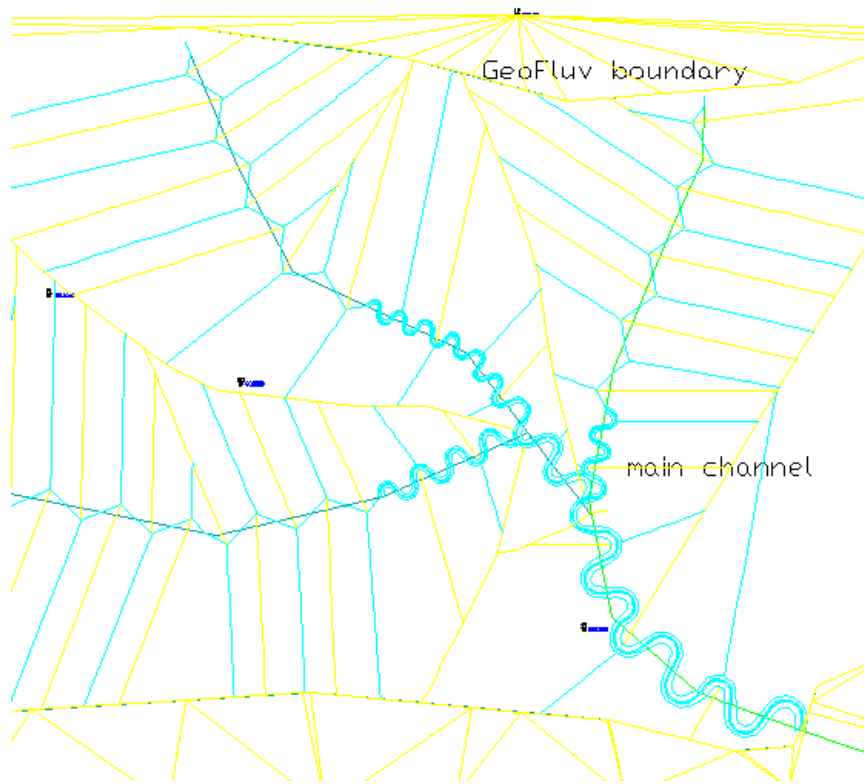
Draw Design Surface

Draw Design Surface integrates all of the GeoFluv™ landform design data that the user has input and outputs it to the drawing.

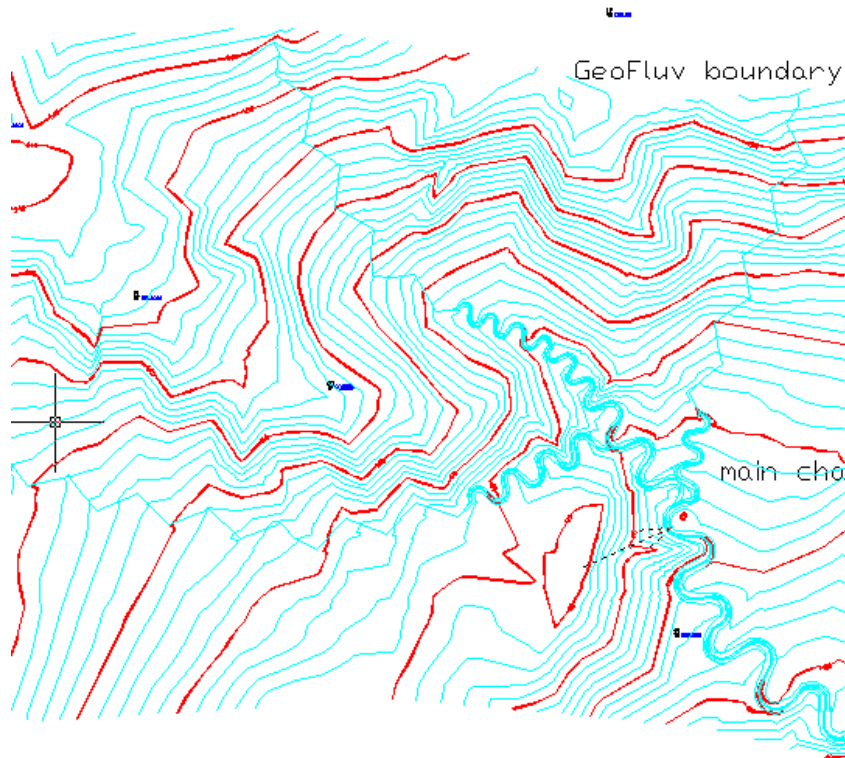


Left-clicking on the Draw Design Surface button causes the Draw Design Surface dialog box to appear. This dialog box allows the user to choose the layer name to which *Natural Regrade* will save the channel and ridge polylines, the triangle skirt layer, and the Sub-Watersheds. Toggles in the dialog box also allow the user to specify:

- If the triangle mesh outside the GeoFluv™ boundary will be drawn.
- If the 2D outlines of subwatersheds will be drawn.
- If intersecting built tributary channels are trimmed at the point of intersection or if they continue all the way to the confluence.
- If existing entities in these layers are erased.



Clicking the OK button in the Draw Design Surface dialog box will capture these settings for the GeoFluv™ design. If the settings are valid, *Natural Regrade* will insert the subwatershed subridge and subvalley breaklines into the drawing, regardless of how many are needed to create the stable draft design and the Triangulate and Contour from TIN dialog box will pop up. The user can edit the settings on the dialog box tabs or accept the default settings. The user clicks OK to begin contouring.



Natural Regrade will draw the draft GeoFluv™ landform contours on the drawing and a pop-up Carlson Edit dialog box will appear that lists any instances of 'crossing barrier lines', if the triangulate and contour settings were

appropriate. The user can use this edit box to review the drawing for possible errors; typically the crossing barrier lines reported in this edit box are intersections of channel and valley lines. (See also the Draw GeoFluv Contours command for more detail about this feature.)

If the 'maximum triangle mesh line length' setting in the Triangulate tab of the Contour selection in the Carlson DTM Triangulate and Contour menu is set to a value less than the required triangle mesh line length for portions of the design, the command line will read "Ignored 'xxx' triangulation lines that exceeded maximum tmesh line length" and only those portions of the design, if any, that did not exceed the maximum tmesh line length will be contoured. If this occurs, the user can reset the maximum triangle mesh setting in the DTM menu to be greater than the 'xxx' distance reported in the command line and then repeat the Draw Design Surface command sequence as described above. GeoFluv™ will then contour the entire drawing as described above. (See also the Draw GeoFluv Contours command for more detail about this feature.)

The subridgelines and the valleys between them extend from the main ridgelines to the channels. The slopes have default settings that create concave slopes, rather than constant gradient or convex slopes that are subject to rill and gully formation.

Save Design Surface

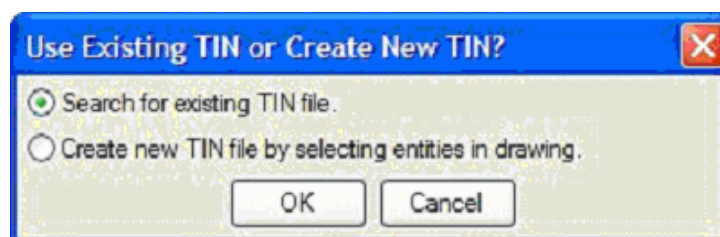
The Save Design Surface button provides a quick means to save the triangulation mesh file of the GeoFluv™ draft design surface. The draft design surface is virtual and is not necessarily represented in the drawing at the moment. The draft design surface is created using the inputs of the 2D valley bottom polylines, the GeoFluv Boundary, the Pre-Disturbed Surface, and the various settings of the current GeoFluv project. The draft design surface is the same one that would be created by the Draw Design Surface button.

Any modifications to ridge or subridge or channel polylines in the drawing using tools such as Edit Longitudinal Profile are not a part of the draft design surface. To create a triangulation mesh file of the design surface in the drawing, use the Draw GeoFluv Contours command in the Natural Regrade dropdown menu and select the Write Triangulation File option on the Triangulate tab.

The user left-clicks on the Save Design Surface button and the Save Design Surface - (.FLT; .TIN) dialog box appears on the screen. The user is offered three options to name and save the surface file. The user can type in the name of a new or existing file in the File Name window, or left-click on the Browse button to the right of the window to get a list of surface files related to the project. If the user wants to save surface file as an existing file (overwrite the file), they can highlight the file name in the Recently Used Files: window at the bottom of the dialog box or in the "Files in that folder" window at the right of the dialog box. The user then left-clicks on the Open button at the bottom of the dialog box to save the file with the selected name.

Update Cut/Fill

The Comparison Surface button is used to enter a three-dimensional surface file (.flt or .tin) for determination of cut and fill volume differences between the comparison surface and the GeoFluv landform design that the user has made.



When the user clicks on the Comparison Surface button, the Use Existing TIN or Create New TIN? Pop-up dialog box appears to give the user the options, by clicking on radio buttons, for entering a 3-D surface file. By clicking OK when the radio button is on Search for existing TIN file, the user can browse through existing files to select

the file to import, or select the file from a list of recently used files. When the user highlights the desired file and clicks the Open button, the file name appears below the Comparison Surface button and the user is ready to begin the Update Cut / Fill command.

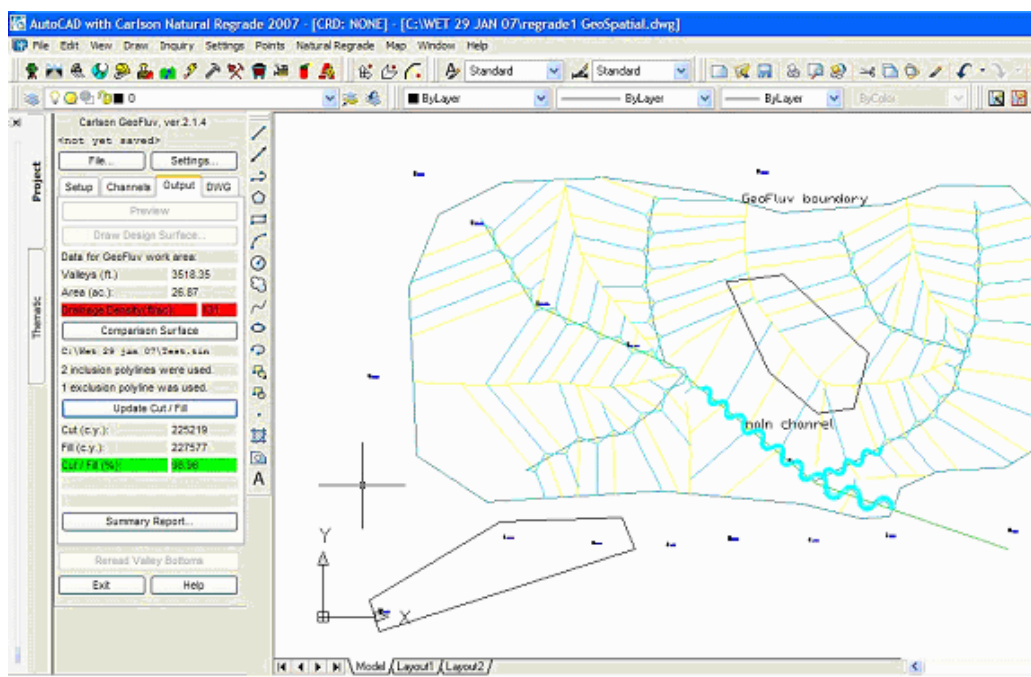
Alternatively, the user can click the radio button for Create new TIN file by selecting entities in the drawing. When the user selects this option and clicks OK, the cursor changes to a selection box that the user can use to define an area with a crossing window. (Note: it is very important that this area encompass the entire GeoFluv project .tin surface. Areas outside the work area that are the same in both the comparison surface and the GeoFluv design will not affect the volume balance so the user does not have to be concerned with defining a slightly larger area.) After defining the new .tin area, the user hits the Enter key and the Select Comparison Surface TIN file - (.tin;.flt) dialog box appears.

The user can choose an existing .tin file by browsing folders or choosing from recently used files, or click on the New tab to create the new .tin file. The user can click on the New tab, enter a name for the new .tin file, and click OK.

The Carlson Software 3D Viewer window will appear showing the .tin for the area that the user has defined. The user can rotate, fill in the .tin triangles, etc. to preview the .tin to be sure that it is satisfactory, i.e., no unwanted holes, elevation spikes, etc. When the user is satisfied that the .tin correctly represents the comparison surface, the user closes the 3D viewer window and the command prompt line reads Select Inclusion Polylines.

The Select Inclusion Polylines option allows the user to define area boundaries for use in the Cut / Fill balance with polylines. By default Natural Regrade will use the GeoFluv boundary polyline as an inclusion boundary to compare against the specified Comparison Surface. Inclusion area boundaries are useful if the user wants to add an adjacent stockpile volume to the calculations, for example. The user clicks on any additional desired inclusion area polylines, including the GeoFluv boundary polyline as needed, that Natural Regrade will use in the calculation. If no inclusion boundary lines other than the GeoFluv boundary are needed, or when all selections are made, the user can hit Enter and is prompted to select Exclusion polylines.

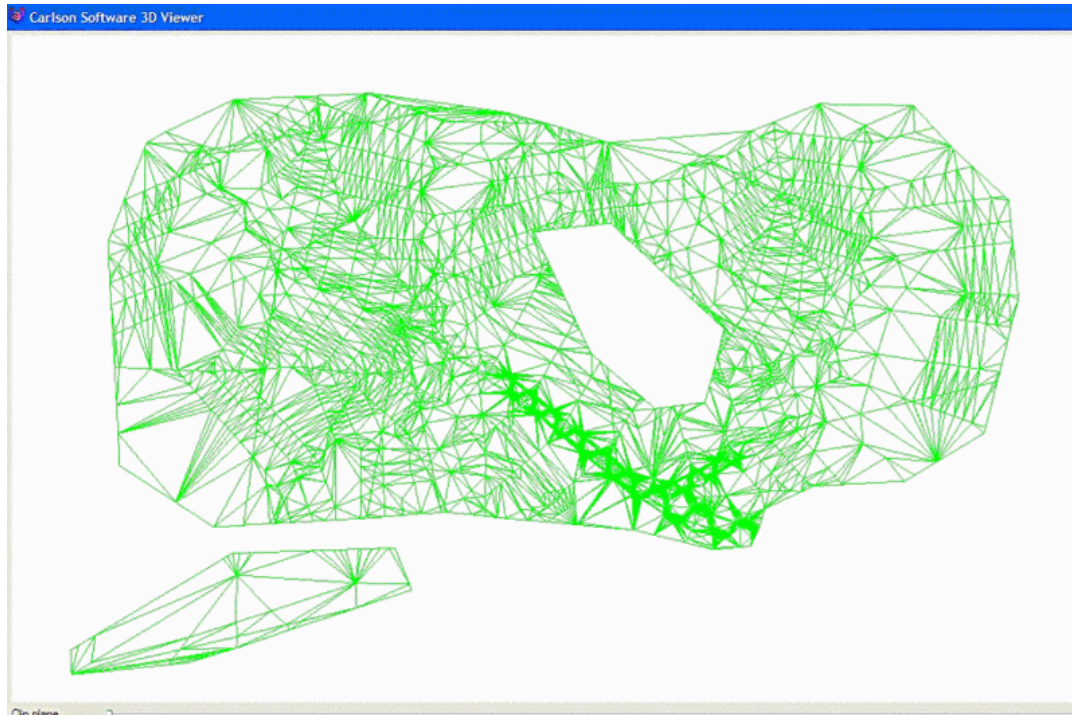
Exclusion polylines are the inverse of inclusion polylines. They can be used to exclude an area like a home-site that is surrounded by the GeoFluv project. If no exclusion boundary lines are needed, or when all selections are made, the user can hit Enter and is prompted to select Exclusion polylines. Natural Regrade will then display the number of inclusion and exclusion polylines that will be used in the cut / fill calculation. The user is then ready to proceed to the Update Cut / Fill button to make the calculation.



In the example, the user has selected two inclusion boundaries, the GeoFluv boundary and a boundary line around a stockpile southwest of the GeoFluv boundary, and one exclusion boundary on top of a ridge near the middle of the GeoFluv design.

The Update Cut / Fill button makes the calculation of the cut and fill needed to make the GeoFluv design from the comparison surface that the user has specified.

When the user clicks on the Update Cut / Fill button, the Carlson Software 3D Viewer window appears showing the .tin that will result from the user's specified file and any inclusion and exclusion boundaries. This helps the user to quickly verify that the specified areas are as desired. The user may also find it helpful to use the 3D Viewer's Toggle shading of surfaces button to fill in the triangle mesh spaces when inspecting the .tin.

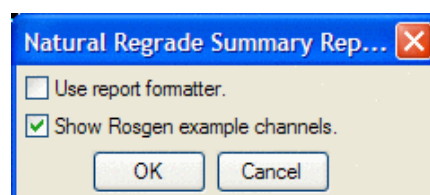


When the user closes the 3D viewer, the calculation results are displayed below the Update Cut / Fill button.

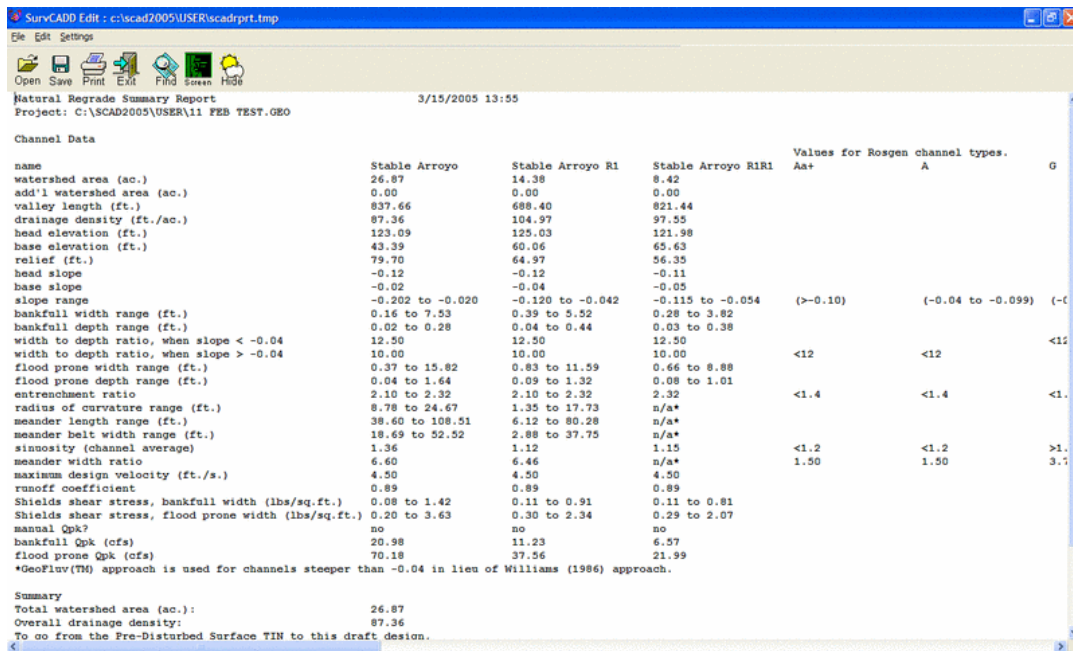
The calculation respects any inclusion and exclusion areas that the user has specified. When the result is within the balance range that the user specified in the Natural Regrade Global Settings (accessed via the Settings button), the Cut / Fill (%) is highlighted in green to give the user quick feedback; conversely, if the result is outside the user-specified range the result is highlighted in red.

Summary Report

The Summary Report button provides the user with a quick summary report of the input parameters and resulting dimensions and material volumes that *Natural Regrade* used to design each GeoFluv™ channel within the project's GeoFluv Boundary.



The user left-clicks on the Summary Report button and the GeoFluv Summary Report dialog box appears on the screen. The dialog box gives the user the options of creating a custom report by checking the "Use report formatter" box (toggle on) and presenting characteristics of Rosgen channel types by checking the "Show Rosgen example channels" box (toggle on; this is the default setting).



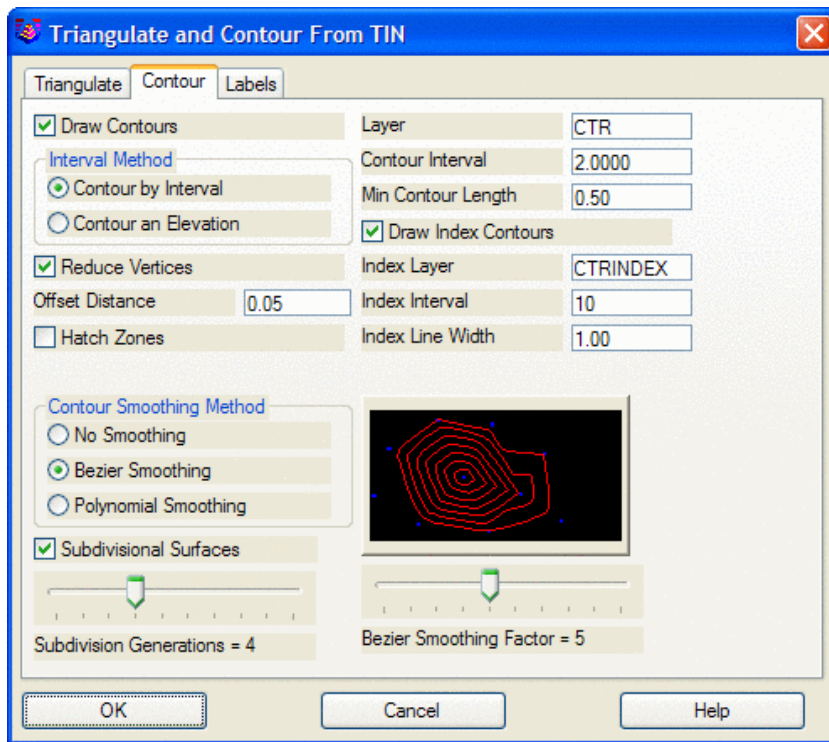
This information can be useful to compare the channels' morphology with other natural channel types when editing the draft GeoFluv™ design to verify that changes to the channel do not exceed the range established for that stable channel type. The summary report information can also be useful for estimating if the user may want to increase channel roughness, install additional bank protection or channel weirs to augment step-pool sequences, etc. Fields that contain information that is not used in the GeoFluv™ approach for designing A and Aa+ channels (>0.04 slope) are marked n/a. Many of the channel characteristics occur within a range at any particular reach in stable natural channels, rather than as a specific value. The dimensions also change in the up-and downstream directions as a function of discharge. The Summary Report presents the range of these values that are applicable to the entire length of each channel. The user can report channel reach-specific information by using the Channels tab's Report button.

DWG Tab

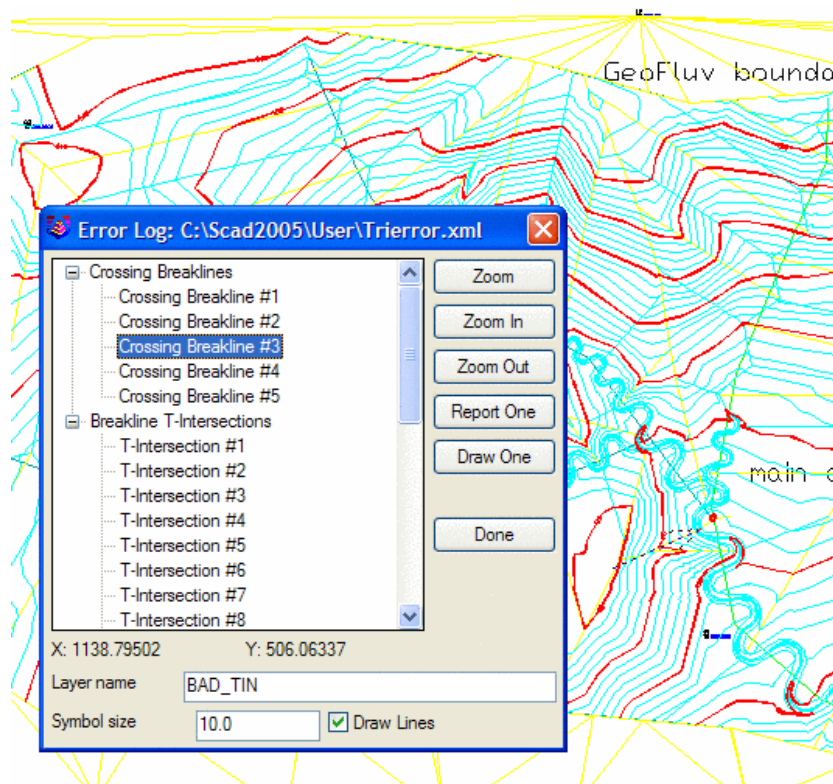
Draw GeoFluv Contours

This command gives the user single-click access to the "Triangulate and Contour from TIN" dialog box via the *Natural Regrade* drop-down menu.

After the user has used the Design GeoFluv Regrade dockable dialog box/Output tab's Draw Design Surface button to create the GeoFluv™ design in the drawing, the user can left-click on the Draw GeoFluv Contours command and produce the Triangulate and Contour from TIN dialog box to contour revisions to the polylines in the layers that the GeoFluv™ design was drawn to.



When the contours have been drawn, the **Error Log: C:\Scad2005\User—Trierror.xml** dialog box will appear. The user can review this error log, or close the box and proceed with the contouring if precise detail is not yet required at this design stage. This Error Log reports any potential errors that Carlson has detected when contouring the drawing.



For example, if the report has a Crossing Breaklines (two polylines with different elevations) field, the user can left-click on that field and a list of the detected crossing breaklines will appear. The user can then left-click on each detected crossing breakline and its x and y coordinates will be displayed at the bottom of the dialog box. The user may choose to report or draw all the detected potential problems using the **Report All** and **Draw All** buttons, or may highlight to select a single potential problem. If the user selects a single problem, the buttons change to **Report**

One and Draw One.

The user can left-click on the **Zoom** button to inspect the suspected problem; this will zoom to the relevant area of the drawing and place an arrow that points to the suspected problem on the drawing. Left clicking on the **Zoom In** button will allow closer inspection of the area of concern, and the **Zoom Out** button will return the user to the previous view.

For example, if only one entity was selected as a problem of concern, the "Report . ." button will indicate Report One and left-clicking on the **Report One** button brings up the Carlson Edit: c:\scad2005\USER\scadrprt.tmp dialog box that displays a Crossing Breaklines Report with the x and y coordinates of the problem, and the elevation difference between the crossing breaklines. Left clicking on the **Draw One** button draws the selected potential problem breakline on the drawing and brings up an AutoCAD dialog box that tells the user that one feature has been processed.

When the user closes the Error Log dialog box by left-clicking on the X in the upper right corner or on the **Done** button, the Error Log dialog box disappears and the contoured GeoFluv™ design remains on the screen. The user can then choose to edit the drawing using tools in the *Natural Regrade* menu, or any other Carlson menu tools, or can reopen the Design GeoFluv Regrade dockable dialog box to edit any of the GeoFluv™ settings there.

Pulldown Menu Location: Natural Regrade

Keyboard Command: gfContour

Prerequisite: Design geofluv

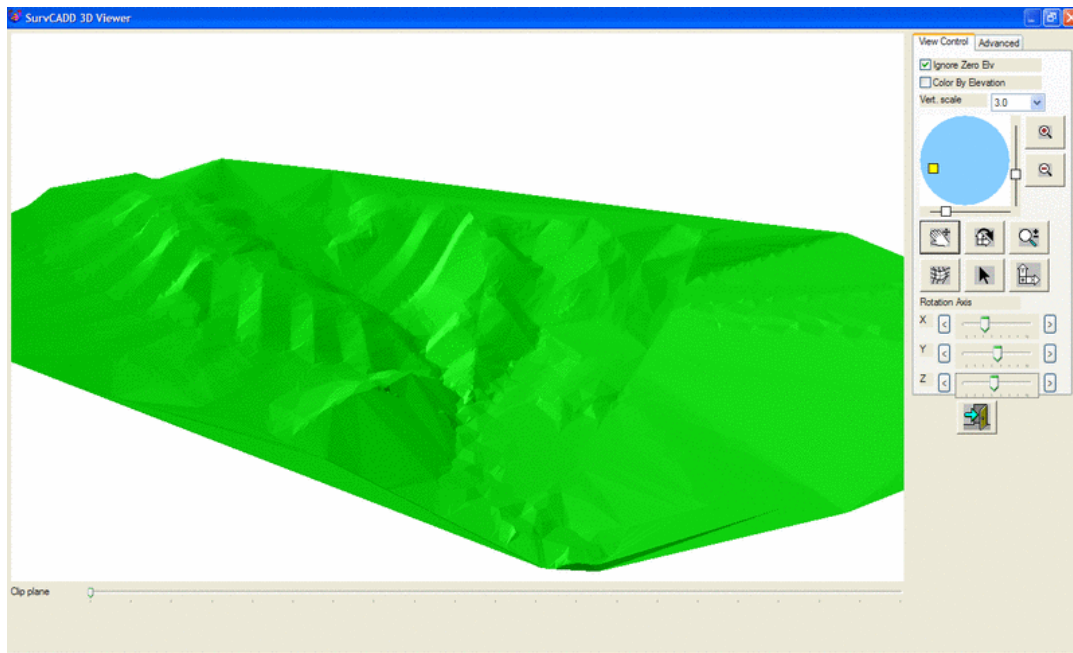
3D GeoFluv Contour Viewer

This Natural Regrade menu gives the user single-click access to the *3D GeoFluv Contour Viewer* that displays, in 3D, the surface created from the contours of a GeoFluv™ design. Viewing the contours is more representative of how the landform will look after grading than the surface produced by the *3D GeoFluv Surface Viewer*. The *3D GeoFluv Surface Viewer* shows you the surface based on the GeoFluv™ design, before using Triangulate and Contour. This resulting surface tends to be angular and faceted, and it is also the surface used for volume calculations. The *3D GeoFluv Contour Viewer* shows you the surface based on the GeoFluv™ design after using Triangulate and Contour, and therefore has more and smaller triangles, lending a smoother appearance to the 3D image.

Note that these commands use linework from particular layers in the drawing. This means that any editing the user makes to the linework in these layers will be reflected in the resulting 3D image.

The prerequisite to using the 3D GeoFluv Contour Viewer is that the contours must already exist. To create contours, the user can select the Triangulate and Contour option after clicking on the Draw Design Surface button on the Output tab of the Design GeoFluv Regrade command. Alternatively, the user can use the Draw GeoFluv Contours command on the Natural Regrade drop-down menu, which will create the contours based on the GeoFluv™ design that exists in particular layers in the drawing.

Clicking on the 3D GeoFluv Contour Viewer command first shows a dialog box asking the user to verify the layers in the drawing that the contours are in. There is also an option for saving the surface created from the contours into a TIN file, which can then be used with other tools in other *Carlson Civil / Survey* modules. Clicking on the OK button causes the contours in the selected layers to be "triangulated" into a solid surface, or TIN, and then opens the Carlson Software 3D Viewer with the GeoFluv™ design surface displayed.



The user has many options for viewing the image available in the View Control tab dialog box on the right side of the 3D Viewer screen. The 3D project view can be rotated on the x, y, and z axes using the sliding **Rotation Axis** controls to the right of the view. By left-clicking and dragging the slider control below the image, the user can clip off forward portions of the view. The viewer has the ability to add vertical exaggeration to aid inspection of lower-relief areas by left-clicking on the arrow to the right of the "**Vert. scale**" edit box and selecting a factor from the dropdown menu. The viewer can also change the position of the sun on the project to evaluate sunny and shady areas throughout the day, a very useful tool for identifying optimal areas for different plantings. The user can either left-click on the 'sun' (**yellow box in the blue circle**) and drag it across the 'sky' from the west (the blue circle is the sky) or move the slider buttons surrounding the blue circle. The 3D viewer has the ability to **color the view by elevation** layers by a toggle setting at the top of the dialog box. These views are helpful for construction to help workers visualize how the final project surface can be built in a series of lifts, for example by using GPS-guided truck dumping.

The toggle buttons on the dialog box below the blue and yellow 'sun and sky' indicator control:

Pan The user holds down the left-click button and moves the mouse across the drawing to pan the drawing in the 3D Viewer.

Rotation The 3D project view can be rotated on the x, y, and z axes by left-clicking the rotation toggle button and then holding down the left-click button while moving the mouse on the drawing.

Dynamic Zoom The user can zoom in and out of the 3D view by holding down the left-click button and moving the mouse up and down on the drawing.

Shading The user can toggle to fill with shading (color) between the TIN file's triangular edges to give the drawing a solid surface appearance.

Average Elevation The user can toggle to have an arrow appear on the drawing surface that indicates the average elevation at the point at which the arrow points.

Reset to Plan The user can left-click on this button to cancel all rotation settings and return to plan view.

The Carlson Software 3D Viewer's Advanced tab has options to block model objects, shade the view, export the view image, and save the view that are fully described in the General Commands/View Commands documentation.

The user can **exit the 3D Viewer** by left-clicking on the X at the upper right or the door at the bottom of the dialog box.

Pulldown Menu Location: Natural Regrade

Keyboard Command: gfViewC

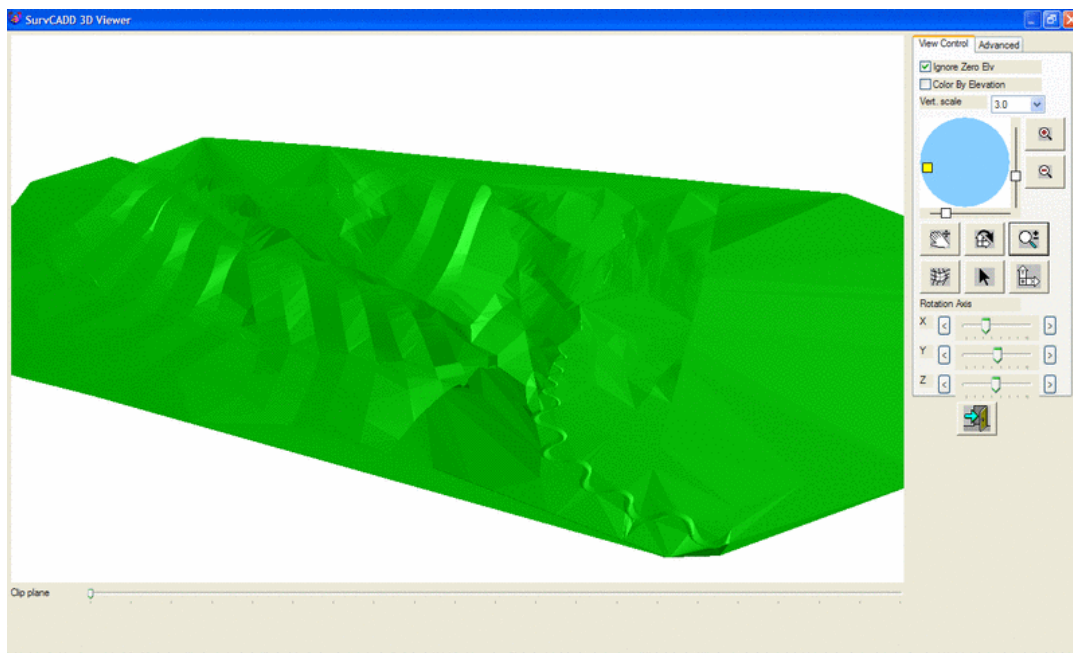
3D GeoFluv Surface Viewer

This *Natural Regrade* command gives the user single-click access to a 3D GeoFluv™ surface viewer that is based on the GeoFluv™ design. The 3D GeoFluv Surface Viewer displays, in 3D, the surface created from the linework that was drawn by the Design GeoFluv Regrade command. This resulting surface is the same one that is used for volume calculations. Alternatively, the 3D GeoFluv Contour Viewer displays, in 3D, the surface created from the contours of a GeoFluv™ design which has more and smaller triangles lending a smoother appearance to the 3D image.

Note that these commands use linework from particular layers in the drawing. This means that any editing the user makes to the linework in these layers will be reflected in the resulting 3D image.

The prerequisite to using the 3D GeoFluv Surface Viewer is that the linework for a GeoFluv™ design must already exist in the drawing. To create the GeoFluv™ design in the drawing, the user can click on the Draw Design Surface button on the Output tab of the Design GeoFluv Regrade command.

Clicking on the 3D GeoFluv Surface Viewer command causes the 3D Surface Viewer dialog box to appear on the screen.



The user has many options for viewing the image available in the View Control tab dialog box on the right side of the 3D Viewer screen. The 3D project view can be rotated on the x, y, and z axes using the sliding **Rotation Axis** controls to the right of the view. By left-clicking and dragging the slider control below the image, the user can clip off forward portions of the view. The viewer has the ability to add vertical exaggeration to aid inspection of lower-relief areas by left-clicking on the arrow to the right of the "**Vert. scale**" edit box and selecting a factor from the dropdown menu. The viewer can also change the position of the sun on the project to evaluate sunny and shady areas throughout the day, a very useful tool for identifying optimal areas for different plantings. The user can either left-click on the 'sun' (**yellow box in the blue circle**) and drag it across the 'sky' from the west (the blue circle is the sky) or move the slider buttons surrounding the blue circle. The 3D viewer has the ability to **color the view by elevation** layers by a toggle setting at the top of the dialog box. These views are helpful for construction to help workers visualize how the final project surface can be built in a series of lifts, for example by using GPS-guided truck dumping.

The toggle buttons on the dialog box below the blue and yellow 'sun and sky' indicator control:

Pan The user holds down the left-click button and moves the mouse across the drawing to pan the drawing in the 3D Viewer.

Rotation The 3D project view can be rotated on the x, y, and z axes by left-clicking the rotation toggle button and then holding down the left-click button while moving the mouse on the drawing.

Dynamic Zoom The user can zoom in and out of the 3D view by holding down the left-click button and moving the mouse up and down on the drawing.

Shading The user can toggle to fill with shading (color) between the TIN file's triangular edges to give the drawing a solid surface appearance.

Average Elevation The user can toggle to have an arrow appear on the drawing surface that indicates the average elevation at the point at which the arrow points.

Reset to Plan The user can left-click on this button to cancel all rotation settings and return to plan view.

The Carlson Software 3D Viewer's Advanced tab has options to block model objects, shade the view, export the view image, and save the view that are fully described in the General Commands/View Commands documentation.

The user can **exit the 3D Viewer** by left-clicking on the X at the upper right or the door at the bottom of the dialog box.

Pulldown Menu Location: Natural Regrade

Keyboard Command: gfView3D

Prerequisite: Design geofluv

Calculate GeoFluv Volume

This *Natural Regrade* drop-down menu command is used to calculate the cut and fill volume difference between the GeoFluvTM design surface in the drawing and another surface, the Comparison Surface. The Comparison Surface is specified on the Output tab. Possible sources for the Comparison Surface may be pre-disturbed, such as the natural landsurface or an earlier reclamation design, or may be disturbed, such as a mine pit or construction site.

If the valley bottom or GeoFluv Boundary polylines have been moved, or any setting in the GeoFluv project has been changed that can affect volumes, then the design surface in the drawing will be out-of-date. The "Draw Design Surface" button must be used to create a new surface in the drawing to reflect the changes.

When the user clicks on the command, a dialog box appears with the cut and fill required to transform the first surface into the current design surface (layers GF_Channels and GF_Ridges). The first surface is the Comparison Surface. The cut and fill are calculated wherever the two surfaces overlap unless an inclusion polyline is given in which case the cut and fill are calculated within the inclusion polyline.

The results are displayed in cubic yards in the English system of units and cubic meters in the metric system.

Pulldown Menu Location: Natural Regrade

Keyboard Command: gfVolume

Prerequisite: Design geofluv

Cut/Fill Centroids

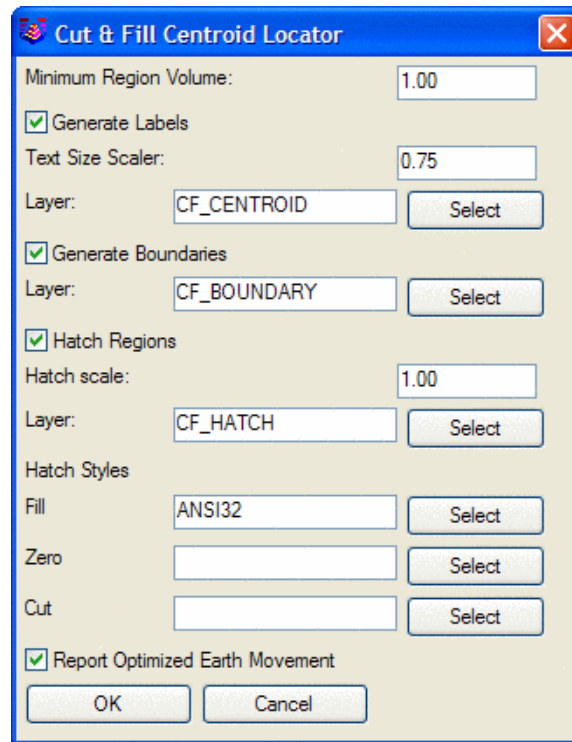
This *Natural Regrade* dropdown menu command is designed to show the amounts and locations of cut and fill that are required to transform the Comparison Surface (typically the disturbed surface) into the design surface. The Comparison Surface is specified in the current GeoFluvTM project on the Output tab. The design surface for this command is the result of combining the GeoFluvTM design in the layers in the drawing **within** the GeoFluv Boundary plus the Surface for Elevations (specified on the Setup tab) **outside** the GeoFluv Boundary.

After comparing these two surfaces, this command identifies the centers of earth-material volumes that need to be moved and the centers of voids that need to be filled to create the GeoFluv™ design. The command includes the option to identify optimal straight-line material movement paths.

Note that this command uses linework from particular layers in the drawing. This means that any editing the user makes to the linework in these layers will be reflected in the results.

The prerequisites for using the Cut Fill Centroids command is that the linework for a GeoFluv™ design must already exist in the drawing and the Comparison Surface must be set. To create the GeoFluv™ design in the drawing, the user can click on the Draw Design Surface button on the Output tab of the Design GeoFluv Regrade command. The Comparison Surface file can be entered by clicking on the Comparison Surface button on the Design GeoFluv Regrade Output tab.

Clicking on the Cut Fill Centroid command causes the Cut & Fill Centroid Locator dialog box to appear, if the prerequisites have been met. The Cut & Fill Centroid Locator dialog box gives the user several options.



Minimum Region Volume allows the user to type a minimum cut / fill centroid region volume for the calculations into an edit box.

Generate Labels is a toggle setting that allows the user to have the centroid region number, cut or fill, and volume value labeled on the drawing next to a crosshair that indicates the centroid coordinates.

Text Size Scaler: Allows the user to type a scale factor into an edit box to enlarge or reduce the size of label text. This command is inactive when Generate Labels is toggled off.

Layer: Allows the user to specify the layer on which the labels will be drawn by either typing them into an edit box or by left-clicking on the Select button which produces a dropdown list of existing layers from which to choose. The user can highlight a layer name on the list and click OK at the bottom of the dropdown menu to select a layer.

Generate Boundaries is a toggle setting that allows the user to specify if *Natural Regrade* shall create cut / fill boundaries and, if so, to what layer the boundaries should be saved. If toggled off, the edit window is inactive. When toggled on, the user may accept the default boundary layer name, type a different boundary layer name into the edit box, or left-click on the Select button to choose from the list of layers.

Hatch Regions is a toggle setting that allows the user to have the centroid regions covered with hatching to make it easier to discriminate from other parts of the drawing.

Hatch Scale: Allows the user to type a scale factor into an edit box to enlarge or reduce the size of hatching. If the user specifies a scaler that causes the hatch spacing to be too dense or the dash size too small, this error will be reported on the Command Line. [Tip: Before drawing a different scale, erase the previous iteration using Edit/Erase/Erase by layer.] This command is inactive when Hatch Regions is toggled off.

Layer: Allows the user to specify the layer on which the hatching will be drawn by either typing the layer name into an edit box or by left-clicking on the Select button which produces a dropdown list of existing layers from which to choose. The user can highlight a layer name on the list and click OK at the bottom of the dropdown menu to select a layer.

Hatch Styles: Allows the user to specify hatch styles to differentiate among Fill, Zero elevations between Fill and Cut, and Cut material by either typing them into an edit box or by left-clicking on the Select button which produces a dropdown list of existing layers from which to choose. The user can highlight a layer name on the list and click OK at the bottom of the dropdown menu to select a layer.

Report Optimized Earth Movement is a toggle setting that allows the user to specify if *Natural Regrade* shall not only calculate and identify the cut and fill centroids, but also determine the shortest straight-line haul distances to distribute the material from cut to fill regions.

```

Cut & Fill Centroid Report

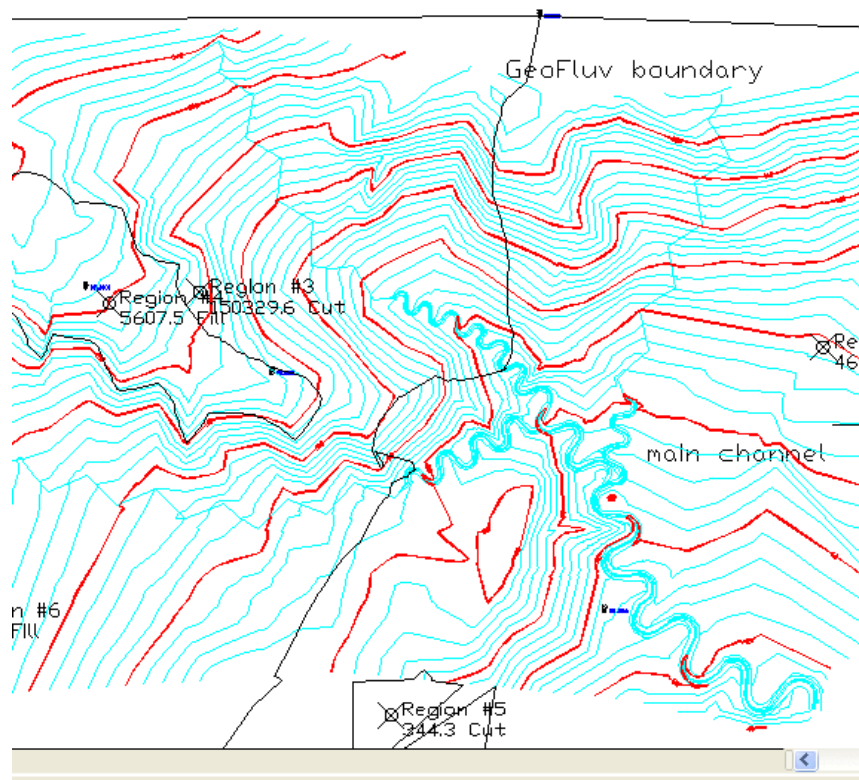
Original Ground: C:\SCAD2005\DATA\REGRADE_EXIST.FLT
Design Surface:  \sa4.2.tin

Region  Volume(C.Y.)  Northing  Easting
1       1103732.3  Cut   -361.15   1137.23
2       463494.4  Fill    697.46   1407.06
3       150329.6  Cut    763.14    660.70
4        5607.5  Fill    750.09    552.55
5        344.3   Cut    257.18    890.03
6         33.5   Fill    375.83    356.83

Earth Movement Report:
From Region  To Region  Volume(C.Y.)  Distance
1           2           312820.5      1092.45
1           4           5607.5        1255.67
1           6            33.5         1073.38
3           2          150329.6      749.24
5           2            344.3         679.09
External    2          785270.9
Total Internal Volume * Distance: 461685727.13
Total External Volume: 785270.89

```

When the user is satisfied with the settings on the Cut & Fill Centroid Locator dialog box, left-clicking on the OK button at the bottom will cause the centroid command selections to be executed and the Cut & Fill Centroid Report to appear on the screen. This report lists the three-dimensional surfaces that were compared to make the calculations, the material and void centroid regions that were identified, the volume of material or void in each region, whether the region identifies a cut or fill area, and the northing and easting of each centroid.



If the user toggled on Report Optimized Earth Movement, the report will be appended with the Earth Movement Report which lists the volume in cubic yards of material that has to be removed from region x to region y and the straight-line haul distance for the material movement, when such a solution can be found.

Pulldown Menu Location: Natural Regrade

Keyboard Command: gfCentroids

Prerequisite: Design geofluv

GeoFluv Channel Cross-Section Report

The *Natural Regrade* dropdown menu's GeoFluv Channel Cross-Section Report gives the user single click access to the detailed-by-station GeoFluv™ channel cross section Report that the user can create using the Channel's tab in the Design GeoFluv Regrade dockable dialog box when not actively working in the design. A user can select any GeoFluv™-designed channel in a drawing without any other files and generate this report.

Natural Regrade Summary Report on Channel "Carlson Creek R1"1/14/2005 18:
(If it were to be drawn using the Draw Design Surface command.)
*GeoFluv(TM) approach is used for channels steeper than -0.04 in lieu of 1

Cross-section reports are done every 50.00(ft.).
Stations are measured along the centerline, starting from the headwaters.
Stations are only reported when there is a real cross-section
formed by the four polylines of the built channel.
The length of the centerline is 1035.84(ft.).
Left and right are from the point of view of looking downstream.

station (ft.): 500.00
slope at station: -0.04
flood prone width (ft.): 4.74
flood prone depth (ft.): 0.48
flood prone area (sq.ft.): 1.34
bankfull width (ft.): 2.38
bankfull depth (ft.): 0.19
bankfull area (sq.ft.): 0.31
bottom width (ft.): 0.84
Shields shear stress, bankfull width (lbs/sq.ft.): 0.21
Shields shear stress, flood prone width (lbs/sq.ft.): 0.76
right side slope (%): 32.82
left side slope (%): 19.78

station (ft.): 550.00
slope at station: -0.04

Refer to the Channels tab's Report button description for details of the report.

Command prompt:

Select any channel polyline created by GeoFluv. Select objects:

Pulldown Menu Location: Natural Regrade

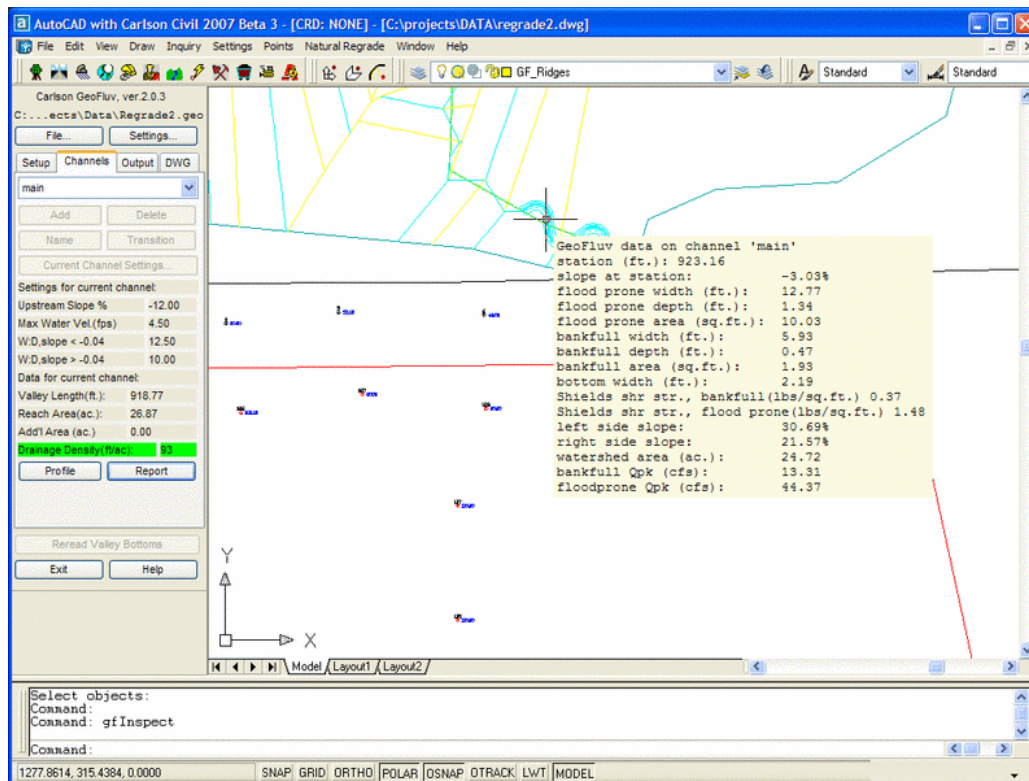
Keyboard Command: gfReport

Prerequisite: Design geofluv

GeoFluv Channel Inspector

This command allows the user to obtain detailed design information from the *Natural Regrade* drawing by passing the cursor over the point of interest (on a polyline in the drawing).

When the user left-clicks once on the GeoFluv Channel Inspector command in the *Natural Regrade* dropdown menu, a check mark appears to the left of the command signaling the user that the command is toggled on. Then when the user passes the cursor over any polyline in the drawing, detailed GeoFluv design information for that point on the polyline appears on the screen next to the cursor. The command remains active until the user left-clicks on the command in the dropdown menu to toggle it off.



The GeoFluv Channel Inspector works on ridge, valley, and contour polylines also, but obviously does not present channel design information for those polylines. This convenient and powerful feature allows the user to inspect designs without having to refer back and forth from a database to the drawing, but instead read directly from the drawing. Note that the information displayed is not an average for the entire line, but is specific to location on the line and can vary even in very small distances.

There are some limitations to be aware of. First, to inspect the zig-zag A-channel, the GeoFluv project that was used to create that channel must currently be open in the Design Geofluv Regrade tool. Secondly, using almost any command, including Edit Longitudinal Profile and Auto Longitudinal Profile, to modify a channel polyline causes the extended entity data to be removed which prevents the GeoFluv Channel Inspector from having certain information that it needs.

Pulldown Menu Location: Natural Regrade

Keyboard Command: gfInspect

Prerequisite: Design geofluv

View Longitudinal Profile

This Natural Regrade dropdown menu command allows the user to view the longitudinal profile of any polyline in the GeoFluv design and obtain the elevation and slope at any point along the profile.

Command prompt:

Select objects

The user places the cursor over the polyline that they want to inspect and left-clicks. The profile viewer appears on the screen. The user can move the cursor along the profile and the station, elevation, and slope information are displayed along the bottom of the dialog box at the position indicated by the cursor. Simultaneously, an arrow moves along the channel in the drawing pointing to the location along the longitudinal profile that the cursor is covering on the profile viewer.

Radio buttons for Vertical Exaggeration settings aid the user in evaluating low relief profiles.

Radio buttons also allow the user to toggle back and forth between Zoom and Pan drag action for the mouse. When set to Zoom, holding the mouse left-click button down while moving the mouse up and down will cause the viewer

to zoom in and out on the longitudinal profile. When set to Pan, holding the mouse left-click button down while moving the mouse across the viewer will allow the user to pan around the longitudinal profile. Whether in Zoom mode or Pan mode, the middle mouse button can be held down to pan the profile in the display. Thus, if the user has a middle mouse button, staying in the Zoom mode and using the middle mouse button to pan is most efficient.

The Grid Ticks Only toggle allows the user to see a grid on the profile display or to see only tick marks on the axes.

The GeoFluv design made using Natural Regrade typically has many sub-watershed ridge and swale lines. The lines are 3D polylines and their profiles are generated from the user inputs (based on project area field work). The blue bar at the top of the View Longitudinal Profile pop-up dialog box will display the name of the line the user is viewing to supplement the visual dot-within-a-triangle that tracks along the line on the drawing.

The naming convention for the lines is to identify the sub-watershed, e.g., main L2R1 (note L and R are capitalized when identifying channels and their subwatersheds), followed by the sub-watershed ridge or swale line identification where lower case 'r' indicates a sub-watershed ridge line and a lower case 's' indicates a sub-watershed swale line. The sub-watershed ridge or swale line letter is followed by a number indicating its sequential order from the head of the sub-watershed. Using this convention, the 'main L2R1 r1' designation identifies the first sub-watershed ridge on the right valley wall (r1) of the sub-watershed that holds the first tributary entering from the right valley wall (R1) of the second tributary entering from the left valley wall (L2) of the main channel.

Pulldown Menu Location: Natural Regrade

Keyboard Command: gfViewPro

Prerequisite: Design geoflucv

Edit Longitudinal Profile

The Natural Regrade dropdown menu's Edit Longitudinal Profile command gives the user quick access to a powerful longitudinal profile editing tool that can change the entire longitudinal profile of a 3D polyline or just a portion of the profile.

Examples of tasks that this profile editor is well suited for is creating saddles on a long ridge line to further dissect topography and creating a 'hump' on the ridge profile where the user might want to leave excess material.

When the user left-clicks on Edit Longitudinal Profile, the command line directs the user to Select 3D Polyline and the command prompt reads, "Select objects:." The user moves the cursor to the 3D polyline that they wish to edit and left-clicks on the polyline. The "Edit Longitudinal Profile – Double Click to Adjust Profile" pop-up dialog box appears on the screen. The dialog box has a profile viewer similar to the View Longitudinal Profile command viewer. The user can move the cursor along the profile and the station, elevation, and slope information are displayed along the bottom of the dialog box at the position indicated by the cursor. Simultaneously, an arrow moves along the channel in the drawing locating the point along the longitudinal profile that the cursor is covering.

The dialog box gives the user toggle settings for the following options:

Adjust connecting linework, when toggled on (the default setting) by left-clicking on the box, will cause Natural Regrade to automatically change all connecting linework, e.g., connecting ridges, to smoothly fit the new longitudinal profile. To update contours to reflect the modified surface, the Draw GeoFluv Contours command can be used.

Grid Ticks Only toggle allows the user to see a grid on the profile display or to see only tick marks on the axes.

Radio buttons give the user the following options:

Vertical Exaggeration settings aid the user in evaluating low relief profiles.

Zoom and Pan drag action for the mouse - When set to Zoom, holding the mouse left-click button down while moving it up and down will cause the viewer to zoom in and out on the longitudinal profile. When set to Pan, holding the mouse left-click button down while moving it around the viewer will allow the user to pan around the longitudinal profile. Whether in Zoom mode or Pan mode, the middle mouse button can be held down to pan the profile in the display. Thus, if the user has a middle mouse button, staying in the Zoom mode and using the middle mouse button to pan is most efficient.

A slider button on the Blend 'x' % control allows the user to specify the percentage of the polyline to which they want to apply the profile change. As the user holds down the mouse left-click button on the slider button and moves the slider button left to right, the percentage of the line that will be affected by the edit is displayed above the slider.

When the user has specified the settings that they want to use for the edit, they move the cursor above or below the longitudinal profile in the display to the elevation that they want to raise or lower the profile at that point and double left-click on the mouse. The profile will raise at that point to the specified elevation and the line will blend from that elevation to the remaining profile over the distance specified using the "Blend 'x' %" slider control. The user can make multiple adjustments to the longitudinal profile in this fashion until the desired profile is achieved. The user then left-clicks on the OK button to apply the changes. Left clicking on the Cancel button will close the dialog box without applying the changes.

The user can toggle on the 'Original Line' option that will display the original profile in a different color to help the user see the effect of the proposed profile edits

The GeoFluv design made using Natural Regrade typically has many sub-watershed ridge and swale lines. The lines are 3D polylines and their profiles are generated from the user inputs (based on project area field work). The blue bar at the top of the Edit Longitudinal Profile pop-up dialog box will display the name of the line the user is viewing to supplement the visual dot-within-a-triangle that tracks along the line on the drawing. The Edit Longitudinal Profile command allows the user to make an irregular profile that can vary from a smooth convex to concave profile, for example when the user wants to make dips in a ridgeline profile to make saddle features. (Note: Because it can make irregular profiles it is not recommended for use on stream channels as knickpoints subject to erosion could be designed.)

The naming convention for the lines is to identify the sub-watershed, e.g., main L2R1 (note L and R are capitalized when identifying channels and their subwatersheds), followed by the sub-watershed ridge or swale line identification where lower case 'r' indicates a sub-watershed ridge line and a lower case 's' indicates a sub-watershed swale line. The sub-watershed ridge or swale line letter is followed by a number indicating its sequential order from the head of the sub-watershed. Using this convention, the 'main L2R1 r1' designation identifies the first sub-watershed ridge on the right valley wall (r1) of the sub-watershed that holds the first tributary entering from the right valley wall (R1) of the second tributary entering from the left valley wall (L2) of the main channel.

Command Prompt:

Select objects:

Pull-down Menu Location: Natural Regrade

Keyboard Command: gfEditPro

Prerequisite: Design geofluv

Auto Longitudinal Profile

The Natural Regrade dropdown menu's Auto Longitudinal Profile command gives the user quick access to a powerful longitudinal profile editing tool that can change the entire longitudinal profile of a 3D polyline or just a portion of the profile in a smooth curve by specifying starting and ending slopes.

When the user left-clicks on Auto Longitudinal Profile, the command line directs the user to Select 3D Polyline and the command prompt reads, "Select objects:". The user selects the 3D polyline that they wish to edit and left-clicks on it. The Auto Longitudinal Profile pop-up dialog box appears, which gives the user the following options.

Top Slope can be user-specified by typing a value into the edit box, or the default value, which is the present value for the selected 3D polyline, can be accepted. When the user exits the edit box, e.g., with Tab key, an entered slope value is applied to the profile and the user can inspect the revision on the profile viewer. The user can make multiple adjustments to the longitudinal profile in this fashion until the desired profile is achieved. This setting will be the slope at the upper end of the longitudinal profile.

Bottom Slope can be user-specified by typing a value into the edit box, or the default value, which is the present value for the selected 3D polyline, can be accepted. When the user exits the edit box, e.g., with Tab key, an entered slope value is applied to the profile and the user can inspect the revision on the profile viewer. The user can make multiple adjustments to the longitudinal profile in this fashion until the desired profile is achieved. This setting will be the slope at the lower end of the longitudinal profile.

Connect to Ridge allows the user to allow a longitudinal profile, e.g., on a ridgeline, to have an extended convex profile before beginning its concave longitudinal profile. When toggled off (the default setting with the "Convex curve length (ft.)" edit box inactivated), the GeoFluv approach will design a longitudinal profile between the upper and lower elevations of the 3D polyline by constructing a vertical curve using the specified bottom and top slope percents. When toggled on, the "Convex curve length (ft.)" edit box is activated and the user can type in a desired distance value that the upper end of the profile can remain convex before the vertical curve is applied using the specified Top Slope percentage. This feature can be used to place extra material in a hump, for example on a ridgeline, and still have the face of the material grade in a concave profile toward the valley bottom.

When Adjust connecting linework is toggled on (the default setting) by left-clicking on the box, Natural Regrade will automatically change all connecting linework, e.g., connecting ridges, to smoothly fit the new longitudinal profile. To update contours to reflect the modified surface, the Draw GeoFluv Contours command can be used.

The dialog box has a profile viewer similar to the Edit Longitudinal Profile command viewer. The user can move the cursor along the profile and the station, elevation, and slope information are displayed along the bottom of the dialog box at the position indicated by the cursor. Simultaneously, an arrow moves along the channel in the drawing pointing to the point along the longitudinal profile that the cursor is covering.

Radio buttons for Vertical Exaggeration settings aid the user in evaluating low relief profiles.

Radio buttons also allow the user to toggle back and forth between Zoom and Pan drag action for the mouse. When set to Zoom, holding the mouse left-click button down while moving it up and down will cause the viewer to zoom in and out on the longitudinal profile. When set to Pan, holding the mouse left-click button down while moving it around the viewer will allow the user to pan around the longitudinal profile. Whether in Zoom mode or Pan mode, the middle mouse button can be held down to pan the profile in the display. Thus, if the user has a middle mouse button, staying in the Zoom mode and using the middle mouse button to pan is most efficient.

Grid Ticks Only toggle allows the user to see a grid on the profile display or to see only tick marks on the axes.

The user can toggle on the 'Original Line' option that will display the original profile in a different color to help the user see the effect of the proposed profile edits.

The GeoFluv design made using Natural Regrade typically has many sub-watershed ridge and swale lines. The lines are 3D polylines and their profiles are generated from the user inputs (based on project area field work). The blue bar at the top of the Auto Longitudinal Profile pop-up dialog box will display the name of the line the user is viewing to supplement the visual dot-within-a-triangle that tracks along the line on the drawing. The Auto Longitudinal Profile command allows the user to make a smooth profile between the upper and lower line elevations by varying the upper (Top Slope) and lower (Bottom Slope) profile-starting slope values. The user sets either the Top Slope or Bottom Slope value in the appropriate edit box and then clicks on the other box to update the slope profile. (Note: Because it can make profiles based on slope values other than the user's field-determined upstream and downstream values, it is not recommended for use on stream channels as knickpoints subject to erosion could be designed.)

The naming convention for the lines is to identify the sub-watershed, e.g., main L2R1 (note L and R are capitalized when identifying channels and their sub-watersheds), followed by the sub-watershed ridge or swale line identification where lower case 'r' indicates a sub-watershed ridge line and a lower case 's' indicates a sub-watershed swale line. The sub-watershed ridge or swale line letter is followed by a number indicating its sequential order from the head of the sub-watershed. Using this convention, the 'main L2R1 r1' designation identifies the first sub-watershed ridge on the right valley wall (r1) of the sub-watershed that holds the first tributary entering from the right valley wall (R1) of the second tributary entering from the left valley wall (L2) of the main channel.

When the user has specified the settings that they want to use for the edit, they then left-click on the OK button to apply the changes. The new vertical curve will be applied to the GeoFluv design.

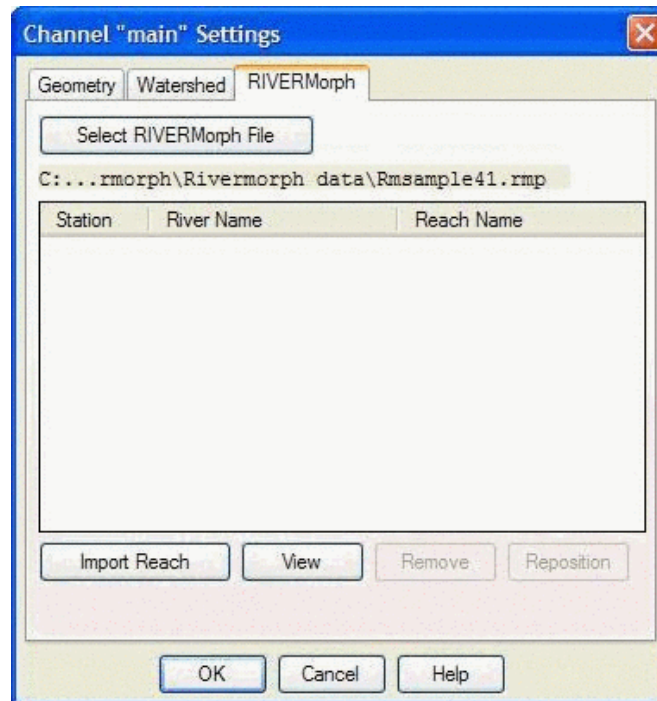
Pulldown Menu Location: Natural Regrade

Keyboard Command: gfAutoPro

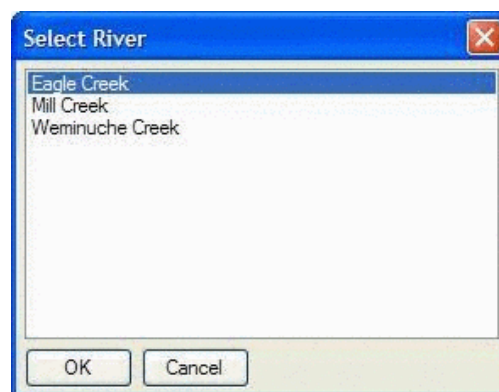
Prerequisite: Design geofluv

RIVERMorph tab

The RIVERMorph tab provides for importation of stream reach design files from the 2-dimensional RIVERMorph program into *Natural Regrade* to produce a 3-dimensional design. The resulting 3-dimensional design is in a .tin format that can be directly exported to machine-control programs like Carlson Grade that guide heavy equipment operators to build the design without the need for additional field survey and staking work.



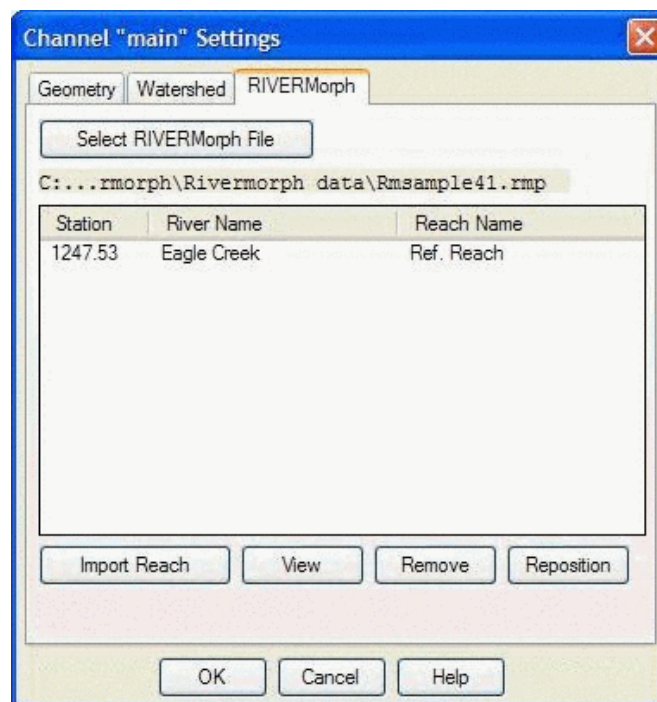
The Select RIVERMorph File button accesses the Open RIVERMorph Project - (rmp) file selection dialog box. The user can browse through RIVERMorph files to select the file to import, or select the file from a list of recently used files. When the user highlights the desired file and clicks the Open button, the file name appears below the Select RIVERMorph File button and the user is ready to begin importing the desired RIVERMorph reach.



When the user clicks on the Import Reach button, the Select River dialog box pops up; the user highlights the desired stream file and clicks OK to select the stream.



The Select Reach dialog box then pops up and the user highlights the desired reach and clicks OK. *Natural Regrade* will advise the user by a pop-up dialog if there is a mismatch in slopes or, if the slope range is valid, will display the Place RIVERMorph Reach pop-up dialog box.



The Place RIVERMorph Reach dialog box gives the user options for *Natural Regrade* to place the reach at appropriate locations based on design .tin elevations, or at user-specified points of known elevation. If the user selects a point with a slope mismatch, *Natural Regrade* will place the reach at the nearest matching point.

When the reach has been placed, it will be displayed in the reach list on the RIVERMorph tab in the Channel "(name)" Settings dialog box. The user can highlight any reach on the list and click on the View button to get a summary of the reach details. The Remove button allows the user to select a reach from the list and delete it from the current GeoFluv project. The Reposition button allows the user to revise the RIVERMorph reach location in the GeoFluv project.

When the user clicks the OK button at the bottom of the RIVERMorph tab dialog box, the RIVERMorph reach design data are imported into the GeoFluv design input data in *Natural Regrade*. The user can use the Preview

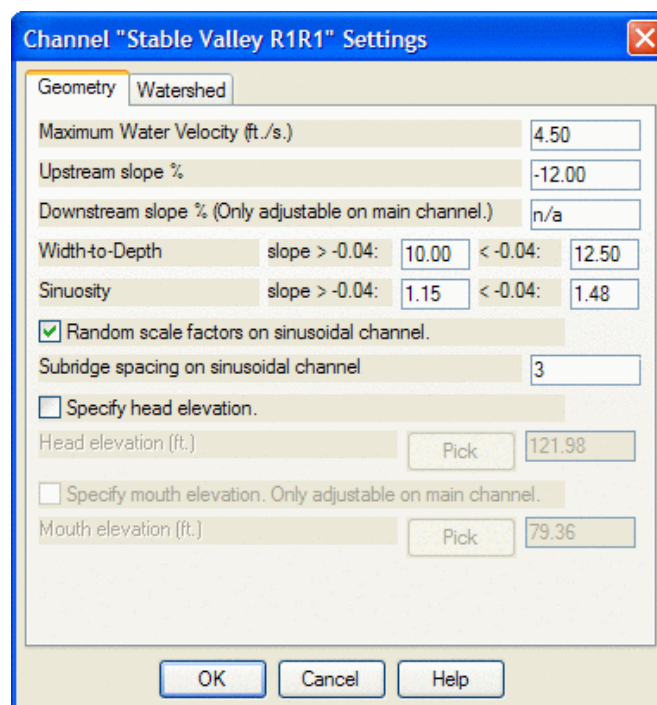
button on Output tab to verify the reach location. When the user clicks the Draw Design Surface button on the Output tab, the RIVERMorph data are made into a 3-dimensional design in the GeoFluv project.

Settings

This button allows the user to specify settings that will vary the channel discharge and the related channel geometry and upland ridge and subridge morphology specific to the subwatershed active in the Channels tab current channel name box. The settings are organized on two tabs, Geometry and Watershed. The Geometry tab has settings for maximum velocity, upstream slope, downstream slope, width to depth ratio, sinuosity, random scale factors on sinusoidal channel, subridge spacing on sinusoidal channel, and channel head and mouth elevation. The Watershed tab has settings for runoff coefficient when using the Rational Runoff Method (the default method), or to allow input of discharge computed by an alternate method, and to add runoff from contiguous land areas.

Command Prompt: (blank, dialog box appears)

Left-clicking on the "Settings" button brings up the "Channel 'xxxx' Settings" dialog box that gives the user the options shown below. The optional settings made in the "Channel 'xxxx' Settings" dialog box will apply only to the Channel 'xxxx' subwatershed. The blue subject bar at the top of the dialog box displays the name of the channel's subwatershed to which the Settings will apply. The user will select a different channel in the "Current Channel" window of the "Channels" tab and then left-click on "Settings" to make these changes to other channels and their subwatersheds, e.g., 'Channel yyyy', 'Channel zzzz,' etc. After specifying the settings in the dialog box, the user can apply them by left-clicking the "OK" button at the bottom of the dialog box.



Maximum Water Velocity (ft./s.): The user can specify a maximum water velocity for the channel by typing the desired value into the edit box. Velocity is inversely related to channel cross-sectional area for a given discharge according to the relationship $Q/a=v$, where Q is discharge (cubic feet per second), a is area (square feet), and v is velocity (feet per second).

Upstream slope %: The user can specify the upstream slope for the channel using this edit box. This feature can be used to vary the channel's longitudinal profile that will join to a mouth slope dictated by the receiving channel slope at their confluence. It can also be used to tie into the upstream slope when the headwaters of the channel are at the GeoFluv Boundary and join with an upstream channel slope draining "Additional watershed area."

Downstream slope % (Only adjustable on main channel.): The user can specify the mouth slope for the main channel at the GeoFluv Boundary to join smoothly to the downstream channel slope by typing the desired slope into the edit box. If the Channel's tab Settings dialog box is open for any tributary to the main channel, the edit box will read "n/a."

Width-to-Depth, slope > -0.04: xx.xx , < -0.04: xx.xx: The user can specify width-to-depth ratios for channels with slopes greater and less than -0.04 by typing the desired width-to-depth ratio into the edit box. The default values are 10.00:1 for channels with greater than -0.04 slope and 12.5:1 for channels with less than -0.04 slope.

Sinuosity, slope > -0.04: xx.xx , < -0.04: xx.xx: The user can specify sinuosity for channels with slopes greater and less than -0.04 by typing the desired sinuosity into the edit box. The default values are 1.15 for channels with greater than -0.04 slope and 1.48 for channels with less than -0.04 slope.

Random scale factors on sinusoidal channel: The meander pattern of the idealized draft valley bottom channels (<-0.04) will be determined by mathematical constants and thus will be very uniform, changing (enlarging) as a function of flow (related to discharge) and valley bottom orientation. Checking the 'Random scale factors on sinusoidal channel' box will randomly vary the constant values, within their acceptable ranges for stable channels, such that radius of curvature, meander length, and meander belt width vary. This random variation produces a more natural appearance for the channel and related upland landforms.

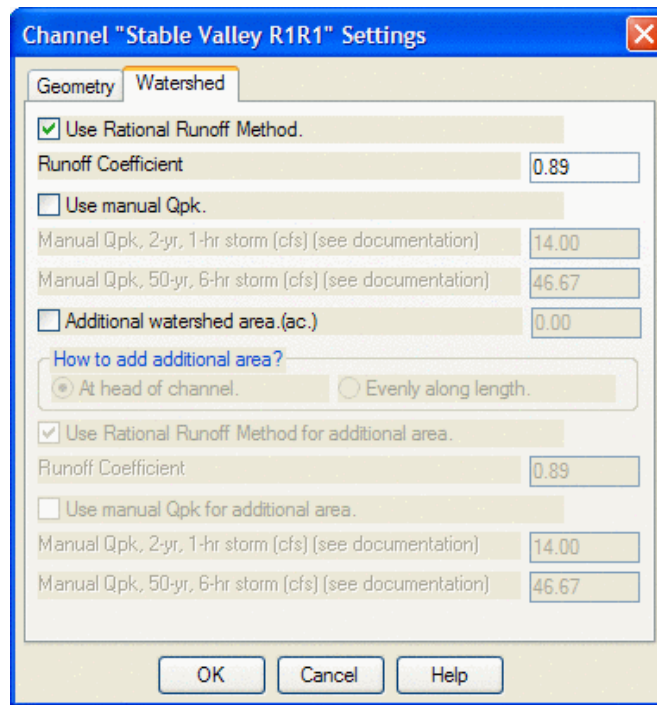
Subridge spacing on sinusoidal channel: This setting applies to channels with slopes <-0.04. The lower-gradient channels, with slopes <-0.04, may have an adjacent floodplain (or terrace) area and the uplands landform may begin some distance from the channel banks. The user can use this setting to create some of this open floodplain or terrace area by increasing the spacing between subridges. A subridge spacing setting of 3, for example, will create a subridge on every third meander bend of the channel with an opening for the floor of the subridge valley between these subridges.

Note: The user must select odd-number spacing; specifying even number spacing will result in all subridges and subridge valleys on opposite sides of the valley. Even spacing can be made with manual SurvCADD editing. The user can also manually add or delete subridges, or vary subridge longitudinal profiles using *Natural Regrade's* longitudinal profile editors, to introduce more variation to the draft GeoFluvTM landform.

Specify head elevation: The user can specify the head elevation for any channel, rather than accepting an elevation that is automatically determined from the Pre-disturbance file specified in the Settings tab. The user checks the box to select this option and then proceeds in one of two ways. The user can type a desired headwater elevation into the Specify Head Elevation field. Alternately, the user can left-click on the Pick button and then identify a (COGO) point of the desired elevation on the drawing. To use the Pick method, the user left-clicks the cursor near the desired point and then, by moving the cursor diagonally, creates a box around the point. The user left-clicks again to define the opposite corner of the box surrounding the desired point and the point elevation is entered into the Specify Head Elevation field.

Specify mouth elevation: The user can (and should) specify the mouth elevation for the main channel only. This setting becomes inactive on the tributary channels because their mouth elevation is controlled by the main channel's longitudinal profile. The procedures for setting the elevation are the same as in Specify Head Elevation above.

[Note: The user should specify the mouth elevation of the main channel in the GeoFluvTM project area because this elevation and the channel slope immediately downstream of this point **may be the most critical variables** for assuring a stable landform design. The elevations that *Natural Regrade* interpolates from the 'Pre-disturbed surface' specified in the Settings tab are appropriate for creating and comparing draft design alternatives, but a channel mouth elevation interpolated from a map surface can vary from the actual elevation on the order of feet. A channel will be expected to adjust to elevation and slope inaccuracies by erosion.]



Use Rational Runoff Method: This is the default setting for calculating runoff to the GeoFluvTM channels in *Natural Regrade* and is the setting that will be used when the box is checked. The Rational Runoff Method calculates a peak discharge using the formula $Qpk = CIA$, where C is the runoff coefficient, I is the rainfall intensity, and A is the acreage. The user enters the appropriate runoff coefficient for the area within the GeoFluvTM boundary in the Runoff Coefficient field and *Natural Regrade* does all the related calculations.

Use manual Qpk: The user can choose to input a peak discharge value calculated by some other method should he wish by checking the 'Use Manual Qpk' option. When the user checks this box, the runoff coefficient field in the Use Rational Runoff Method setting (and use of that method) becomes disabled. The user then types in the peak discharges that he wants to use for the two storm events.

[Note: The GeoFluvTM approach uses the 2-yr, 1-hour storm event to calculate bankfull discharge and the 50-yr, 6-hr event to calculate a flood-prone discharge. Reclamation landforms constructed using the GeoFluvTM approach that use these inputs have been stable in a very harsh and erosive high-altitude desert environment through extreme storm events. Using other input values may give unsatisfactory results.]

Additional Watershed Area: This setting allows the user to incorporate runoff from contiguous lands into the GeoFluv Boundary. When the user checks the Additional Watershed Area box, the fields below become active and offer a choice of how the additional runoff will enter the GeoFluv Boundary. If the head of the GeoFluvTM channel is downstream of the Additional Watershed Area, as when joining to an upstream channel reach, the user should select the "At head of channel" option. The GeoFluvTM channel's headwater dimensions will then be sized to accommodate the runoff from the area above the channel headwaters within the GeoFluv Boundary and the Additional Watershed Area upstream of that. If the Additional Watershed Area is subparallel to the GeoFluvTM channel, checking "Evenly along length" will introduce the runoff from the Additional Watershed Area gradually along the GeoFluvTM channel reach and the channel dimensions will increase proportionately along the reach. The remainder of the settings are as described above in "Use Rational Method" and "Use manual Qpk."

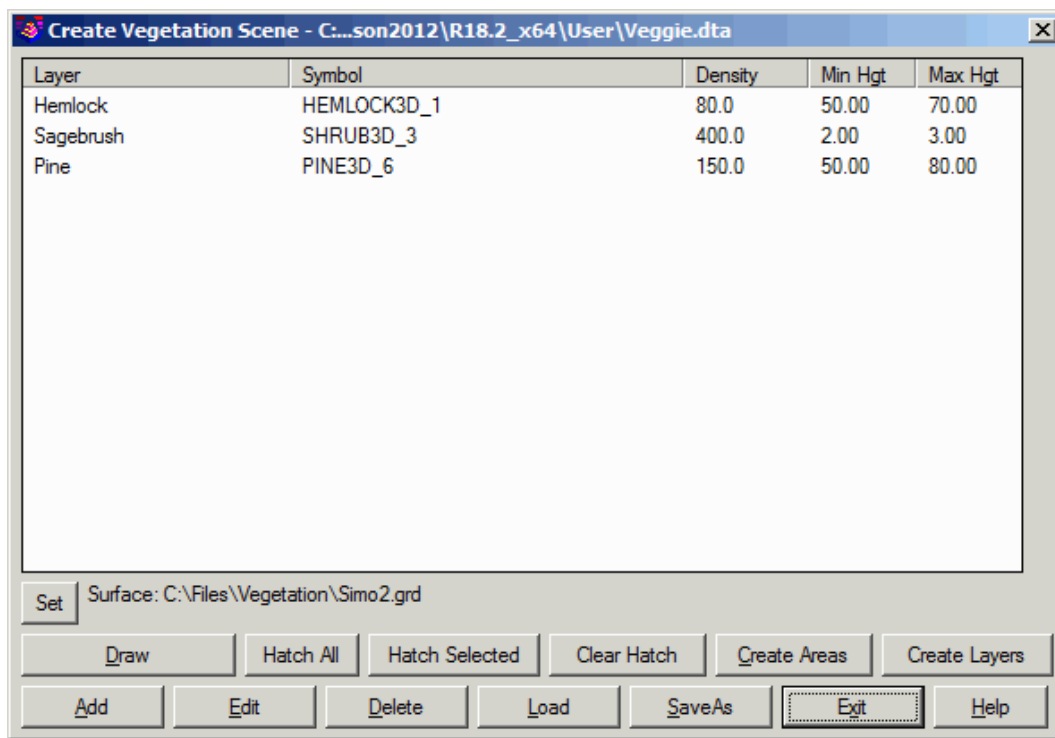
Reread Valley Bottoms

This command updates GeoFluv with information about changes to the valley bottom polylines (or GeoFluv Boundary polyline).

The valley bottom and GeoFluv Boundary 2D polylines are user inputs for fundamental values to the GeoFluv program. They are used to directly calculate valley length, subwatershed area, and drainage density, and affect discharge calculations and channel dimensions. All commands that need to use these input polylines always check the drawing first and use the current version of these polylines in the drawing. The primary use of this command is to quickly update the data displayed on the Setup tab, the Channels tab and the Output tab after a change has been made in the drawing to either the GeoFluv Boundary polyline or any valley bottom polyline.

Create Vegetation Scene

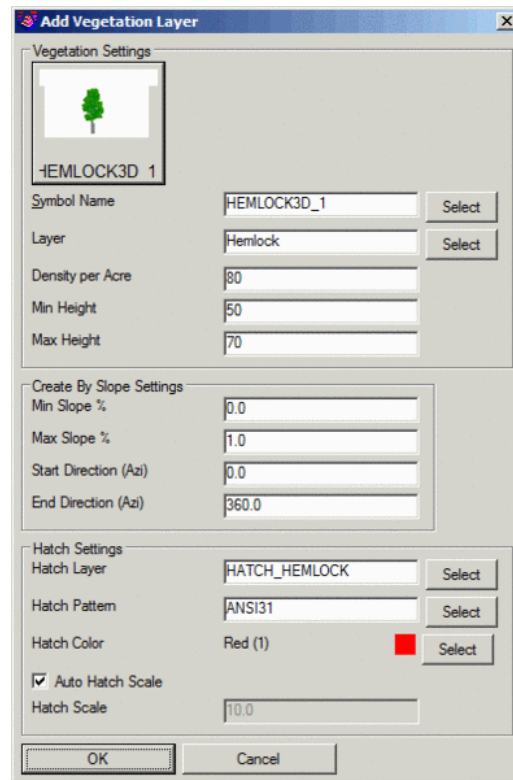
This command places 3D symbols for vegetation for visualization by commands such as 3D Viewer Window and Surface 3D Flyover. This command is layer based where closed polylines on different layers are used to different types of vegetation. The vegetation layers are defined in the dialog shown here. The list shows the layer name, symbol name and parameters for each vegetation type. Use the Add, Edit and Delete functions to manage the list of vegetation. The SaveAs and Load functions store and recall the vegetation definitions to a .veg file. The Draw function creates symbols in the drawing within closed polylines on the vegetation layers using the vegetation parameters for symbol name, size and density. The Hatch functions hatch the closed vegetation polylines as a way to visualize that the vegetation areas are correctly defined. The Create Areas function creates closed polylines on the vegetation layers using the vegetation parameters for slopes and the specified surface model. The Create Layers function creates the vegetation layers in the drawing if they don't already exist.



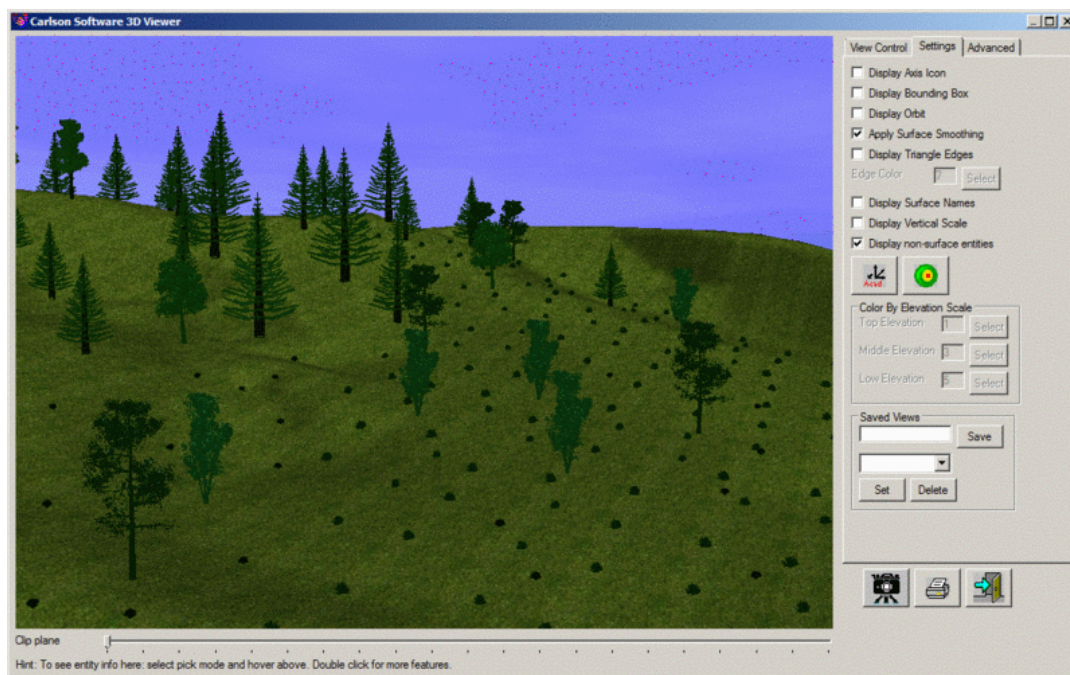
The Add and Edit functions define the vegetation parameters in the dialog shown here. A preview of the 3D symbol is shown along with the symbol name. To change the symbol, type in a name or pick the Select button to pick from the Symbol Library. There are several 3D symbols for vegetation included in the default install under the 3D Trees category in the symbol library. To add your own symbol to the library, run the Settings->Symbol Library command. The symbols should be created at unit height (height=1) so that the program can scale to the target height. The Layer is used both for the closed polyline for the vegetation area and for the vegetation symbols. Density Per Acre controls how many vegetation symbols to draw with the area. The Min/Max Heights define the range of heights for the vegetation symbols. The program will randomly size the symbols within this range.

The Create By Slope Settings apply to the Create Areas function. The Min/Max Slopes control the range of surface slopes that the vegetation exists. The Start/End Directions control which slope facing directions that the vegetation exists. The directions should be entered in clockwise order. For example, for vegetation that only occurs on south facing slopes, the Start Direction could be set to 90 and the End Direction set to 270.

The Hatch Settings apply to the Hatch functions. These hatch settings include the pattern, layer, color and scale. The Auto Hatch Scale option sets the scale to fit within the area.



When viewing with the 3D Viewer Window or Flyover, use the Apply Surface Smooth, Apply Texture and Display Sky options to improve the image.



Pulldown Menu Location: Natural Regrade

Keyboard Command: vegdef

Prerequisite: Closed polylines and surface file

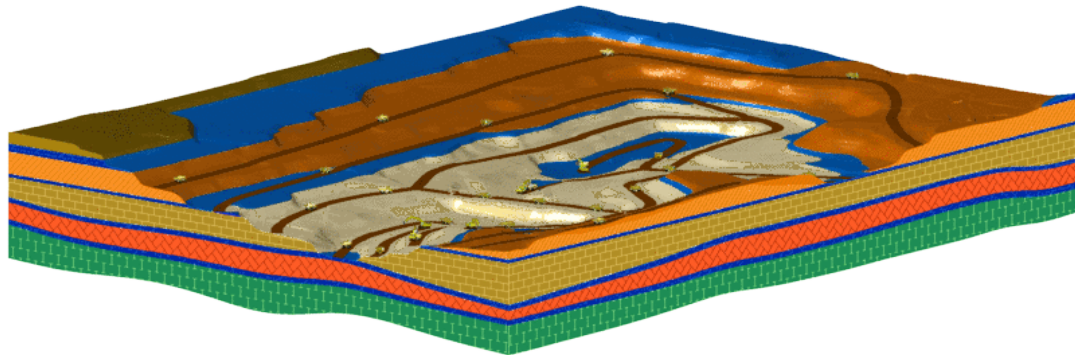


Geology Module

15

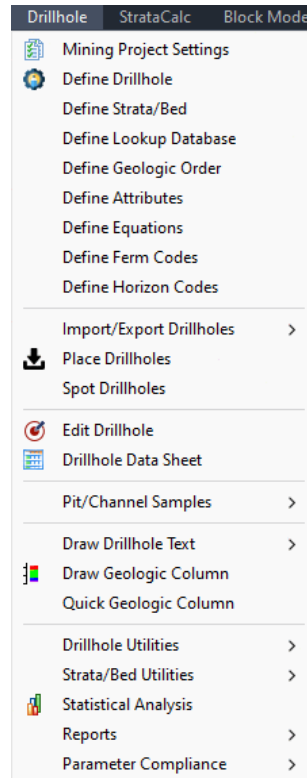
The Geology Module consists of various commands for defining the geologic deposit. Major features include

drillhole analysis and visualization, strata correlation, block modelling, cross section extraction, reserve calculation, and more! Commands are accessible from the Drillhole, StrataCalc, Block Model, and Grids pulldown menus.



Drillhole Menu

The Drillhole menu commands define, input, edit and report drillhole data.



Mining Project Settings

This command sets the current file names used for several Carlson definition files. For example, this command assigns the Drillhole Configuration file (.ch file) to be used for drillhole placement in other routines such as Place Drillhole and Drillhole Data Sheet. The Mining Project Manager lets you set up different groups of definitions. For example, if you work with two mines, then you can have two sets of strata definitions (strata1.sdf and strata2.sdf), two sets of attribute definitions (attrib1.atr and attrib2.atr), etc. When switching between the projects, you can load the .mpj file to instantly update all of your relevant settings.

Whenever a command is executed that creates one of the below files, it will be automatically added to the Mining Project Manager. However, you can also specify the file names directly from this command.

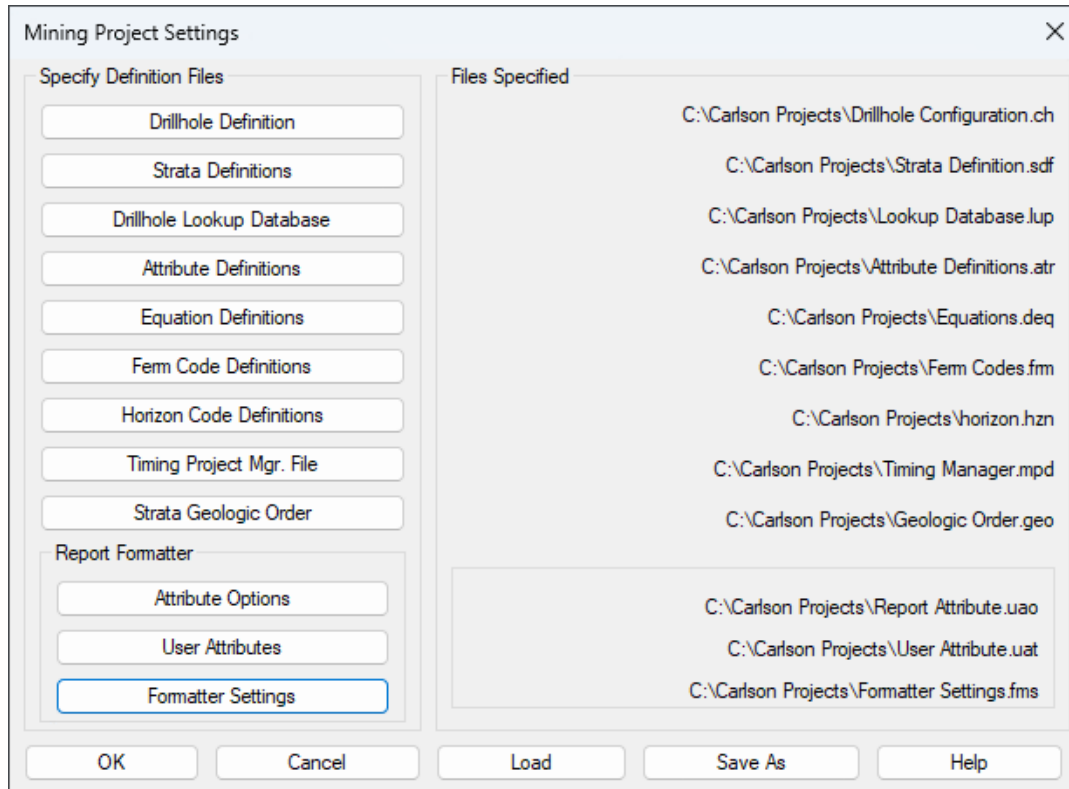
All the file names in Mining Project Manager are saved with the drawing in the drawing .INI file. You can also save this set of file names to a mining project file (.mpj) by picking the Save button. The Load button will then recall the set of file names from the selected .MPJ file.

Following is a list of commands that will create the files shown in the Mining Project Manager. The extensions of the files are shown in the dialog window below.

1. Drillhole Definition: Define Drillhole
2. Strata Definitions: Define Strata/Bed
3. Drillhole Lookup Database: Define Lookup Database
4. Attribute Definitions: Define Attributes

5. Equation Definitions: Define Equations
6. Ferm Code Definitions: Define Ferm Codes
7. Horizon Code Definitions: Define Horizon Codes
8. Timing Project Mgr. File: Underground Timing, Surface Equipment Timing, Solids Timing

9. Strata Geologic Order: Define Geologic Order
10. Attribute Options: In the Report Formatter, Attribute Options
11. User Attributes: In the Report Formatter, User Attributes.
12. Formatter Settings: In the Report Formatter, Export

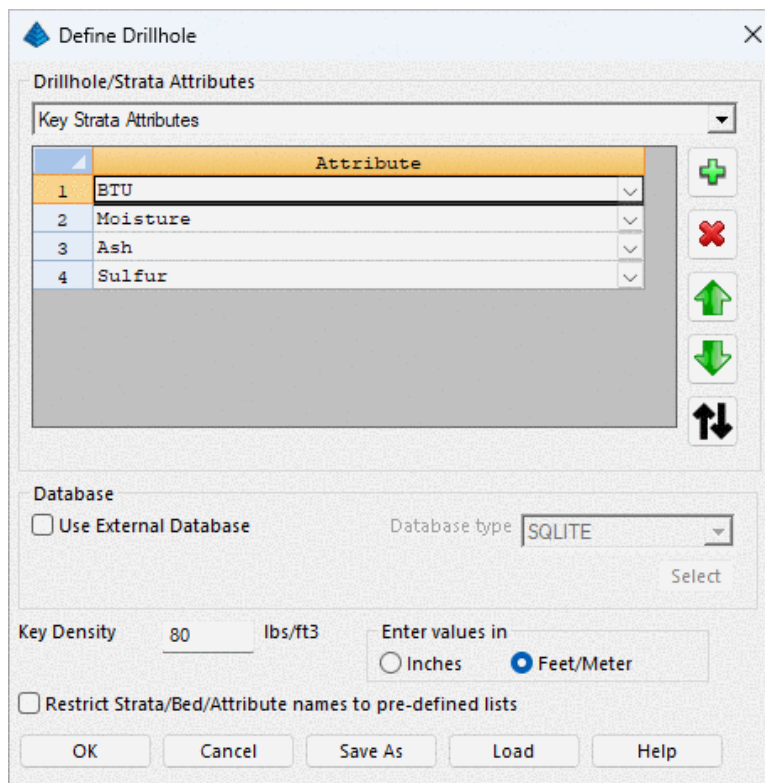


Pulldown Menu Location: Drillhole

Keyboard Command: mproject

Define Drillhole

Define Drillhole is a prerequisite to creating, importing, and processing drillholes. This is a configuration file containing settings for manual drillhole entry, data storage, predefined attributes and the default key density. There is no geologic data stored in this .CH file. The dialog for this command is shown below.



Drillhole/Strata Attributes: This dropdown menu displays the attributes that have been defined in the spreadsheet below this option. Any attributes defined in this spreadsheet will be applied to all drillholes/strata, regardless if the drillholes have a value associated with the attribute. Attributes may be relate to the drillhole itself (e.g. date drilled, drilling company), or to key/non-key strata within the drillhole(e.g. density, assay). You may also define attributes as combinations of other attributes by writing an equation. The equation must start with an equal sign (=) and can include standard mathematical operators (+ - * /). Three examples of attribute equations are given below.

=LBS*SULF/(10⁶*BTU)

=MOIST/LBS

=BTU/1000

Attributes may be added, removed, and reordered with the use of the green icons to the right of the spreadsheet. Attributes may be sorted alphabetically with the use of the sort button. If an attribute has already been defined with the Define Attributes command, it will be available in the dropdown menu to the right of each attribute name. Drillhole attributes may also be defined in the Define Lookup Database command. Strata Attributes may also be defined in the Define Attributes command. However, it is not necessary to predefine drillhole or strata attributes - they may simply be imported directly with the drillholes or added to the drillholes manually with the Edit Drillhole command.

Use External Database: Drillholes may be stored in one of two methods. The default method is leave this option disabled. In this case, the drillholes will be stored directly in the drawing file as extended entity data within the CAD Dictionary. This method is recommended for most projects as there is no requirement to maintain an external file. However, if your project contains thousands of drillholes, the drawing may become exceedingly large and sluggish. In this case, you may wish to enable this option, as the drillhole data will then be stored in a separate file and referenced when needed. Any changes made to the drillholes in the drawing will update the external database, and vice-versa. When enabled, you must select a **Database Type** to set the format of the external database. The four types of databases are:

- **SQLITE** - Data will be stored as a .cdb (Carlson Database) or an .sqlite file. The SQLITE format is open-source and can be viewed/maintained with a variety of free programs. Common web browsers even offer extensions for viewing these files.
- **MDB** - Data will be stored as a .mdb (Microsoft Database - used with older versions of Microsoft Access). Please note that Microsoft now offers a new .accdb format which replaces the .mdb format. As a result, the MDB format is not fully supported and may not function properly on 64-bit computers. If your drillhole data is already stored in this format, it is recommended to save it as an .accdb file within Microsoft Access.
- **ACCDB** - Data will be stored as an .accdb (Microsoft Access Database) file.
- **ODBC** - Data will be stored in one of a variety of other formats such as Excel files or customized database formats.

After selecting a database format, you will need to click the **Select** button to create a new database file or select an existing one. It is important to note that you will still need to import the drillholes after creating this file. If you need to import drillhole data directly from a database file, you can do so through the Drillhole Import command.

Key Density: This field is the default density used to calculate the weight of key strata. The units are lbs/ft³ or kg/m³, depending on the drawing unit system. In addition to this option, there are other ways to specify the density of strata. When calculating reserves, you can select which method to use. By default, if the density is not defined anywhere else, the value set in this dialog will be used.

1. Define Drillhole
2. Define Strata
3. DENSITY grid in a Geologic Model
4. Define Grade Parameters (sets density of each block within a block model)

Restrict Strata/Bed/Attribute Names to Predefined Lists: This option will not import or use any names that don't match up with the predefined names set here, in the Drillhole/Strata Attributes. This allows you to easily filter out drillhole data that is not needed.

Prompts

Select Existing/New Drillhole Configuration File: Selection Dialog Box

Select Drillhole Configuration File Selection Dialog Box

Specify a file name in which to store the drillhole definition.

Define Drillhole Dialog

Drop-Down Menu Location: Drillhole

Keyboard Command: chdef

Define Strata/Bed

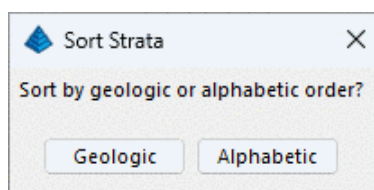
Define Strata serves several purposes. It is an optional settings/configuration file that defines strata and bed hatch patterns, Key status, density, conformance modeling settings, and bed splitting settings. Without the strata definitions predefined, StrataCalc commands will use defaults.

Strata Definitions: C:\Desktop\Current\Simple Geology\Settings\Strata Def.sdf

Strata/Bed Name	Full Name	Key	Color	Hatch	Density	Layer	Draw Legend
1 C0_KEY	Coal 0	<input checked="" type="checkbox"/>		SOLID	Set 85.00	Coal 0	Set <input checked="" type="checkbox"/>
2 C1_IB	Coal 0 Interburden	<input checked="" type="checkbox"/>		ANSI38	Set 150.00	Coal 0 Interburden	Set <input checked="" type="checkbox"/>
3 C1_KEY	Coal 1	<input type="checkbox"/>		SOLID	Set 82.00	Coal 1	Set <input checked="" type="checkbox"/>
4 C2_IB	Coal 2 Interburden	<input type="checkbox"/>		ANSI38	Set 155.00	Coal 2 Interburden	Set <input checked="" type="checkbox"/>
5 C2_KEY	Coal 2	<input type="checkbox"/>		SOLID	Set 83.00	Coal 2	Set <input checked="" type="checkbox"/>
6 COAL	Coal	<input type="checkbox"/>		SOLID	Set 80.00	Coal	Set <input checked="" type="checkbox"/>
7 C_KEY	C Coal	<input type="checkbox"/>		SOLID	Set 80.00	C Coal	Set <input checked="" type="checkbox"/>
8 DOL	Dolomite	<input type="checkbox"/>		DOLMIT	Set 150.00	Dolomite	Set <input checked="" type="checkbox"/>
9 GRN	Granite	<input type="checkbox"/>		GRANITE	Set 155.00	Granite	Set <input checked="" type="checkbox"/>
10 LI	Lignite	<input type="checkbox"/>		ANSI31	Set BY_DRILLHOLE	Lignite	Set <input checked="" type="checkbox"/>
11 LS	Limestone	<input type="checkbox"/>		MSBLOCKC	Set BY_DRILLHOLE	Limestone	Set <input checked="" type="checkbox"/>
12 MDST	Mudstone	<input type="checkbox"/>	256	MUDST	Set BY_DRILLHOLE	Mudstone	Set <input checked="" type="checkbox"/>
13 OB	Overburden	<input type="checkbox"/>	256	ANSI31	Set BY_DRILLHOLE	Overburden	Set <input checked="" type="checkbox"/>
14 SH	Shale	<input type="checkbox"/>	256	ANSI31	Set BY_DRILLHOLE	Shale	Set <input checked="" type="checkbox"/>

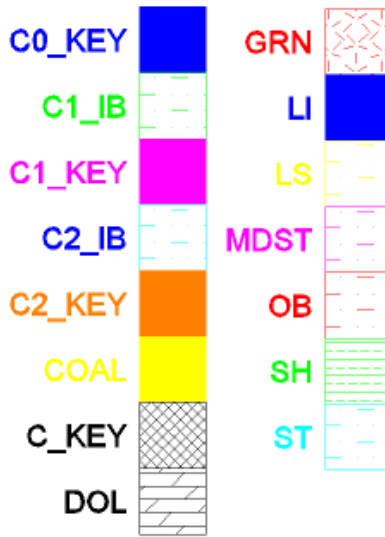
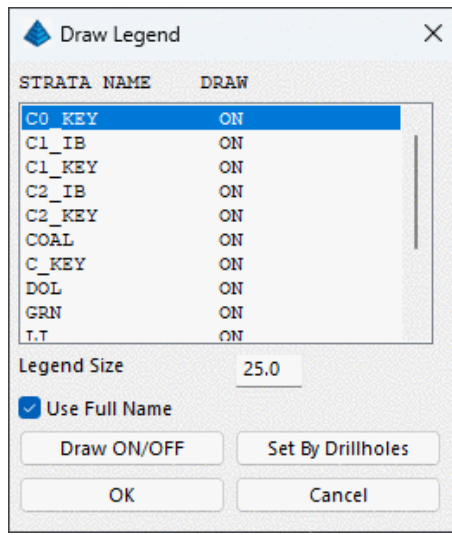
Buttons: Edit, Edit Multiple, Add, Remove, Move Up, Move Down, Sort, Draw Legend, Column Options, Read Drillholes, Read Model, Clear, Load, Save, Save As, Help, Exit

Define Strata is a dialog based spreadsheet editor for strata definitions. The first window shows a list of all of the currently defined strata. If no strata are yet defined, this table will be empty. This command is used to define both Strata, such as COAL and Beds, such as C.KEY. To add a new strata definition, click on the Add button. This brings up the Strata Definition dialog. To edit an existing strata definition, highlight the corresponding line in the table and then click on Edit button which also brings up the Strata Definition dialog. To edit the same property for multiple strata such as the Hatch Scale, highlight multiple rows in the spreadsheet by picking with the Ctrl and/or Shift keys, and then pick the Edit Multiple button. You can also use the standard Windows Copy and Paste functions to edit cells in the spreadsheet. To remove a definition, highlight the corresponding line and then click on the Remove button. The Sort button will sort the strata either by name alphabetically, or by following the Geologic Order file that is created with the Define Geologic Order command.

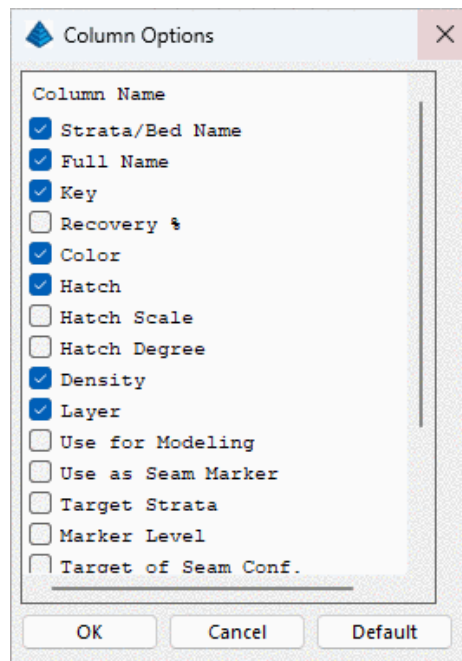


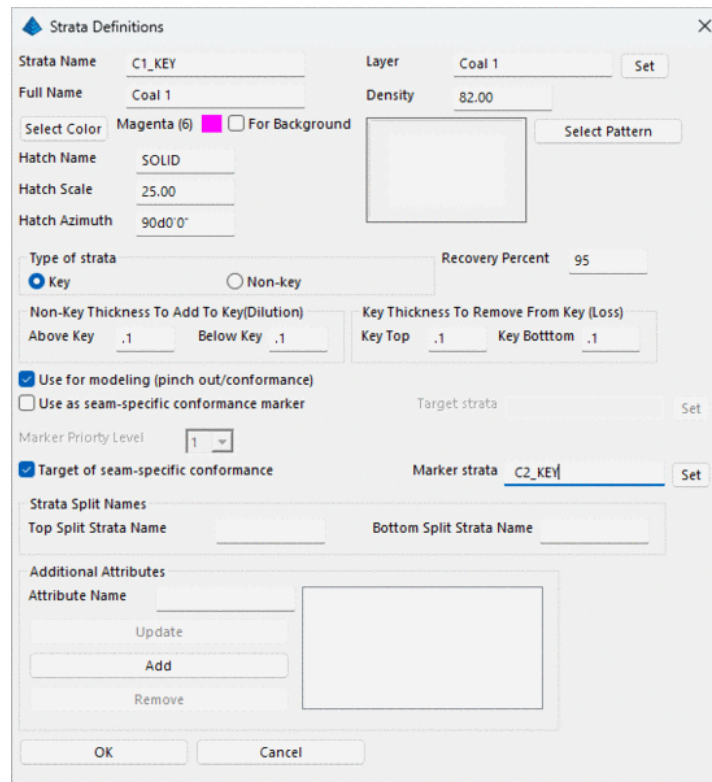
The Draw Legend button will draw a legend of the selected strata. The Read Drillholes button will read the selected drillholes and add a default definition for any strata names found in the drillholes that are not already in the table. The Read Model button will search the Geologic or Mining Model for strata names and put them on the list. The Save and Save As buttons exit Define Strata after saving all the changes in the current session to the strata definition file (.sdf file). The Exit button exits Define Strata without saving.

Draw Legend brings up the selection window to choose which items to display in the legend. Then the legend location is selected.



The Column Options button displays the window to select which columns appear in the main Define Strata Screen.





- **Strata Name:** This is the strata or bed name that is used to match up with the names in the drillhole. Enter in the strata names exactly as they appear in the drillhole. For bed names, there are four extensions or suffixes that must be added to the bed name to match up with the interval of the bed in the drillhole. They are as follows: `_OB`, `_KEY`, `_PARTING`, and `_BOTTOM`, but these options may be modified in Carlson Configure. These must also be entered below, in the target and marker strata windows, and in the strata split names when using bed names. The strata name may use wildcards. This allows fewer strata definitions. The strata list is automatically sorted to place the most general matches in the end of the list. This concept is illustrated in the following example:

Strata definitions: C1, C?, C*, *

Strata in drillhole: COAL, SAND, C2, C1, C12

Applied Matches:

COAL C* This is the least general match to the COAL. '*'=anything

SAND * This word begins with 'S' and all other keys begin with 'C'

C2 C? '?'=any one symbol, so C? is less general than C*

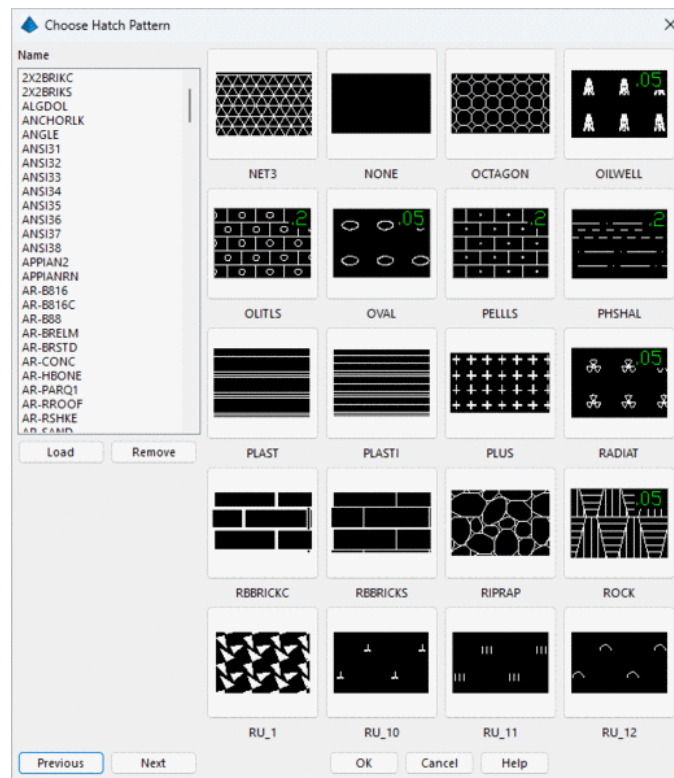
C1 C1

C12 C* C? does not match because of extra symbol on the end.

- **Full Name:** This is the name that will appear as an option in reports and other strata selection windows.
- **Layer:** This is the AutoCAD layer that will be created and used for this strata or bed.
- **Select Color:** This is the color that the solid or hatch pattern will be drawn in for the layer entered above.
- **For Background:** If this is checked, then the background will be a solid fill in the color specified. The hatch

pattern will appear on top of it in a black color.

- **Hatch Name:** This is the name of the selected hatch. A preview of it shows up next to the Select Pattern button. Clicking the Set button will open the dialog of available hatch patterns.
- **Hatch Scale:** This is the scale factor used to size the hatch pattern.
- **Hatch Azimuth:** To rotate a hatch pattern, enter in an azimuth. 90 is horizontal and the default.
- **Density:** Enter in the strata density in pounds per cubic foot or in kg/cubic meter. This is used in Mine Reserves. If no density is set in the strata definition, the program displays "BY_DRILLHOLE" which means that key-strata density of the drillhole will be used. This key-strata density is set by Define Drillhole and can be viewed or modified with Edit Drillhole.
- **Select Pattern:** This button brings up the predefined 127 geologic hatch patterns of Carlson. Each screen with 20 patterns is shown below. Just click on the one to select for each strata or bed.



- **Type of Strata:** The Key or NonKey status can be set here, by strata or bed, which is used by Place Drillholes to determine the type of the user-specified strata name. Also, while entering strata names in Place Drillholes, the strata definitions are checked to see that the strata names are already defined. If the strata name is undefined, the user can define it from there, leave it undefined, or re-enter the name. This check avoids typos and ensures that the strata names match since they should be consistent for the same strata across the drillholes.
- **Recovery Percent:** A recovery can be set for each strata or bed here. It is used in reserves and scheduling.
- **Non-Key Thickness To Add To Key (Dilution):** These values apply to the Use Strata Definitions option in the Surface Mine Reserves and Geologic to Mining Model commands.
- **Key Thickness To Remove From Key (Loss):** These values apply to the Use Strata Definitions option in the Surface Mine Reserves and Geologic to Mining Model commands.
- **Use for modeling (pinch out/conformance):** Turn on this option if the seam is to be used in modeling the drillholes for pinch out and conformance.

- **Use as seam-specific conformance marker:** Turn on this option if this is a "dominant" marker seam that other seams should conform to if needed.
- **Target strata:** Specify the "target" strata to conform to this marker bed. Enter the strata names, with spaces between the names. If they are beds, use the extensions `_TOP`, `_KEY`, `_PARTING` and `_BOTTOM`.
- **Select:** This will bring up a strata/bed list dialog box for easy selection.
- **Marker Priority Level:** This is the priority level for the marker bed. For example, the main marker bed A will have a priority of 1 for Seam C. Seam B is the next dominant, so seam B will have a priority level of 2 for seam C.
- **Target of seam-specific conformance:** This is defining the current seam as a target seam for a different marker seam. Meaning this is not the dominant seam, it will conform to another seam.
- **Marker strata:** Just as in Target Strata above, but the opposite applies; enter in the Marker seam for the current seam to conform to.
- **Strata Split Names (Top and Bottom):** Strata can be defined to split from parent into children strata for modeling. For example, if A splits into A1 and A2, they should be entered in the Top and Bottom Split windows. When modeling, the drillhole that has A will correlate to A1 and A2, with the parting pinching out as it models its way closer to the hole with A. To make sure it is working properly, the seam A should not appear on a list of strata to process. There will only be A1 and A2 for modeling. In case there are more splits, the program allows children strata to split again. For example, the strata definition for A2 can split into A2A and A2B. In Surface Mine Reserves, quantities for the children seams (ie. A1 and A2) are reported for areas that the split exists and quantities for the parent seam (ie. A) are reported for areas without a split. Also, Grid file utilities and other functions such as limit lines can be used to bring children strata "back" together to get a full parent seam.
- **Additional Attributes:** This function will define any attributes specific to that strata, which do not apply to other strata and therefore would be misplaced in the Define Drillhole. The list of attributes for particular strata will be combined from the attributes defined for define Drillhole (key or non-key attributes, depending on the strata type) and attributes defined in Define Strata. In addition the attributes specific for a strata in some drillhole may be added directly to that strata in Edit Drillhole or Drillhole Data Sheet. The equation attributes described in Define Drillhole may be used in this dialog as well.

Prompts

Strata Definitions Table

Strata Definition Edit Dialog

Strata Hatch Patterns

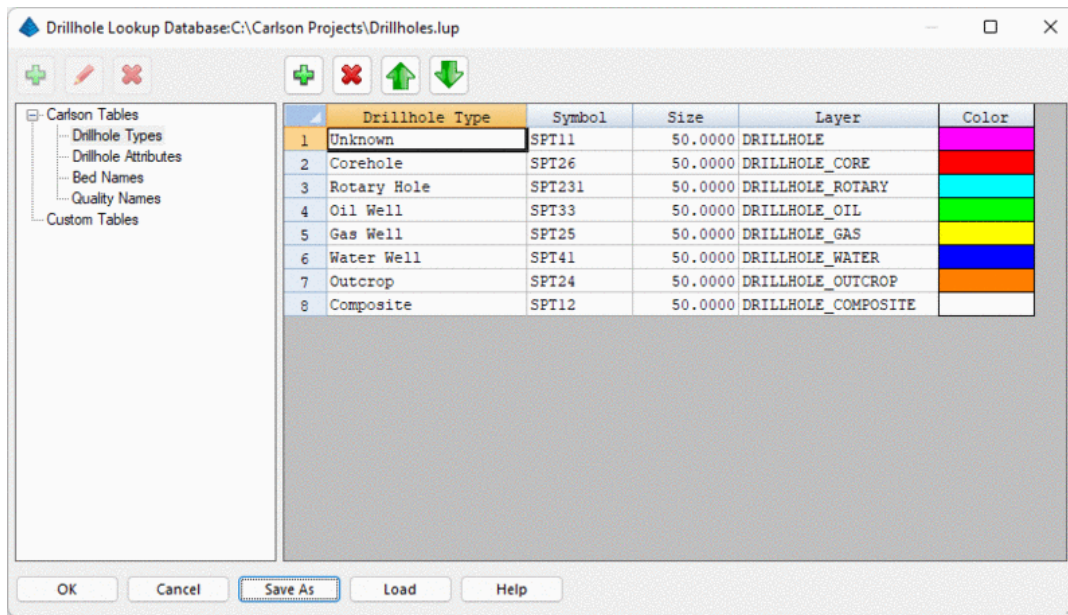
Pulldown Menu Location: Drillhole in Geology, StrataCalc in the Mining menu

Keyboard Command: sdef

Prerequisite: None

Define Lookup Database

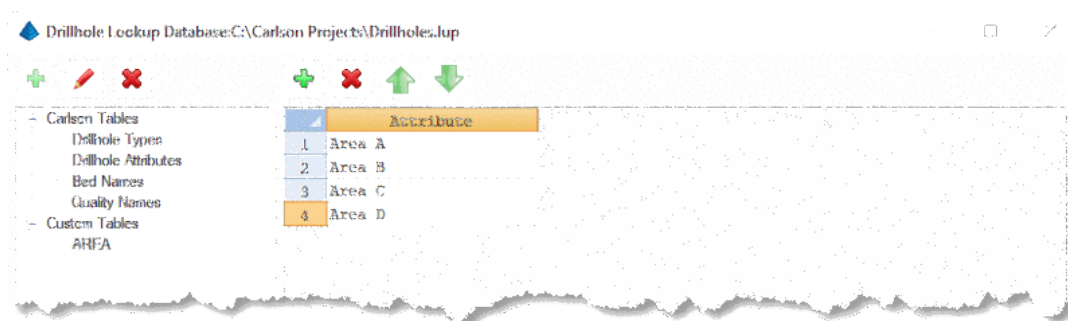
Define Lookup Database is a way to predefine drillhole fields such as drillhole type, attributes, descriptions and survey quality. Any other custom tables can be defined here for future queries and use.



Examples of the Carlson Tables are:

- **Drillhole Types:** This can be used to mark different types of drillholes, as used in the Import Drillhole command. Each can have a different layer, symbol and color for easy viewing on the map.
- **Drillhole Attributes:** Examples for this can be Drillers Name, Date Drilled, County, Mine Area. Drillhole Attributes differ from Strata Attributes in that they apply to the entire drillhole instead of a single strata.
- **Bed Names:** Bed Names can be predefined for easier selection in the Edit Drillhole command. Please note this is not required for importing Bed Names.
- **Quality Names:** This is simply a way to indicate how the coordinates of the drillhole were measured. When using the Import Drillhole command, you may specify an XY Quality and a Z quality for the drillhole to indicate how the Northing, Easting, and Surface Elevations were measured.

Custom Tables are added to represent any other fields that can have a predefined lookup fields, such as Mine Area, or Lab Company, so they can be easily selected from the list.



Prompts

Drop-Down Menu Location: Drillhole

Keyboard Command: dh_lookup

Files: \isp\corehole.arx, \isp\corehole.dcl, \isp\defcore.lsp

Define Geologic Order

This command is to assist the program when partial drillholes are encountered. While this is not a required routine, if many partial drillholes are encountered and the program reports that some strata are out of order, that may be an indication that this routine will help with the correlation. A reference list for strata order is created that all drillholes check to verify the order is correct.



Pulldown Menu Location: Drillhole

Keyboard Command: geo_order

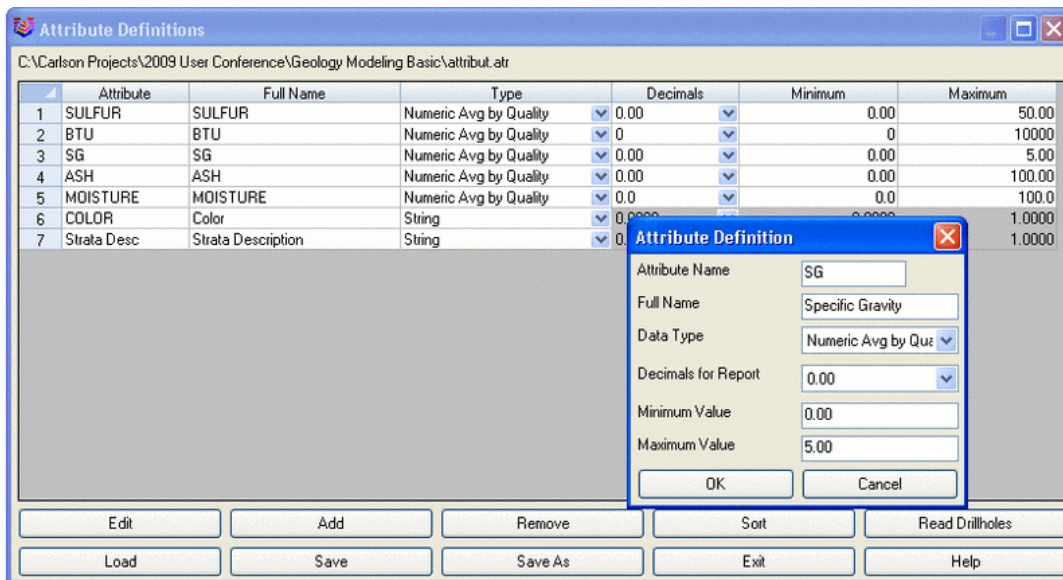
Define Attributes

This command defines the full name, value range and value type of attributes. This allows StrataCalc commands to identify invalid attributes that fall outside the minimum and maximum value range. Incorrect data entry is detected in *Place Drillhole* and *Drillhole Data Sheet*. Use the Attribute Validation Report command under Drillhole, to detect invalid attributes in the existing drillholes.

This routine also defines the data type as numeric or strings. Strings are for non-numerical attributes such as color name. String attributes need to be defined as such before importing the drillholes, otherwise the attribute values, such as green or red, will not import correctly.

For numeric attributes, there are settings to Average By Quantity or Average By Area which apply to *Surface Mine Reserves* for calculating the average attribute values. The default is to average by quantity which uses the strata quantities to calculate the weighted average. The average by area method uses the strata area as the weighting factor for the average. The average by quantity applies to attributes that you want to average by the strata volume such as BTU. For example, when the strata is thicker you pick up more quantities and the attribute value in these thicker areas should be weighted more. The average by area applies to attributes that count for the presence and not their volume. For example, for an attribute of thickness, you should average by area.

The Number of Decimal Places is used in the *Draw Drillhole Text* command and for the reports in *Surface Mine Reserves*. The attribute definitions are stored in a file with an .ATR extension.

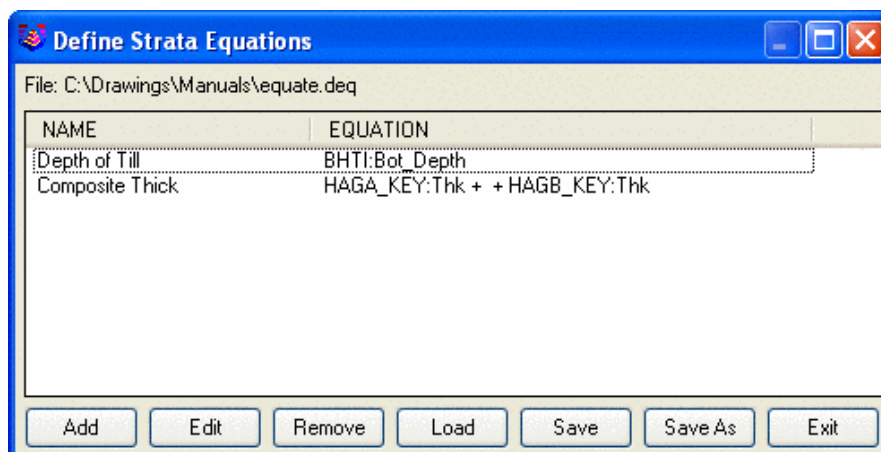


Pulldown Menu Location: Drillhole

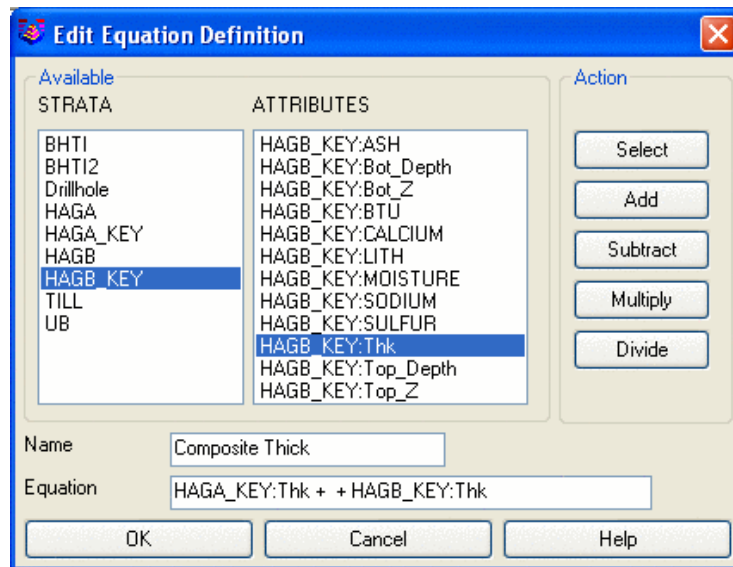
Keyboard Command: attrdef

Define Equations

This command defines equations using any existing strata attributes and data points as variables. These equations allow you to create new attributes based on a combination of strata data and values. The new attributes can then be used as data models in many routines such as *Make Strata Grid* or *Strata Isopach Maps*. This routine is useful for compositing or diluting thickness grids, or generating delivered quality grids in the initial modeling process. Any attribute that is attached to the drillhole attributes, such as water table elevation, or base of weathered zone (horizons that don't belong in the geologic column) can be put into an equation and then gridded to create the surface. After selecting the drillholes, the Define Strata Equations window appears. The selected drillholes are read to find all the available strata and attribute names for the equations. The Define Strata Equations dialog is for adding or removing equations, editing existing ones and loading and saving the equations to a .DEQ file.



User-defined drillhole attributes can now be modeled, such as water table, transgressive horizons or total depth of hole. The first step is to define a drillhole attribute in the drillhole definition file. After the holes have been assigned values for the drillhole attribute, they will appear on the list for Define Equations. If just that one value is selected and added, then it will appear on the list as an equation, capable of being isopached or gridded.



The window for the equation editing and selection has a list of all available equation variables. Equations can also contain values in addition to the drillhole variables. After defining the *.deq file, all routines that list the available strata attributes will have "EQUATIONS" at the top of the list. Selecting that will bring up a dialog window with a list of the user-defined equation names.

Prompts

Select the Drillholes to process.

Select objects: *pick drillholes to read strata and parameters from*

Define Strata Equations Dialog

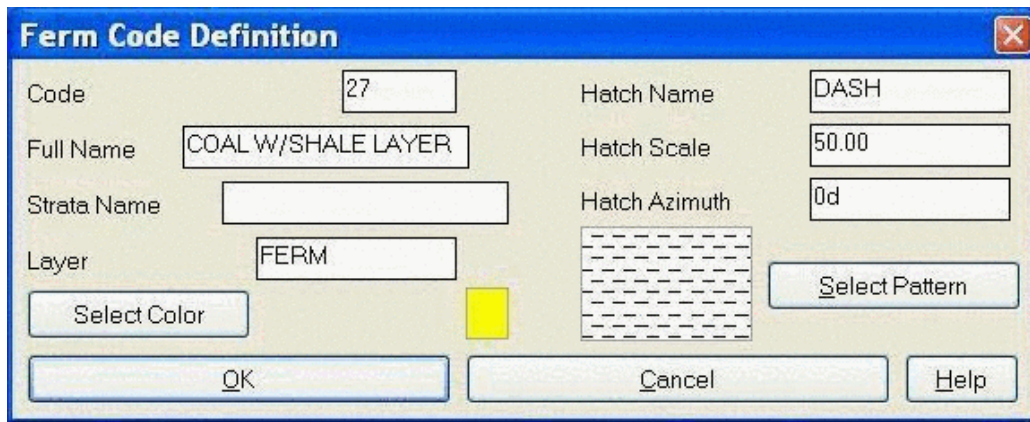
Pulldown Menu Location: Drillhole

Keyboard Command: chequeate

Define Ferm Codes

This command defines the table of Ferm codes. Each ferm code is assigned a full name and drawing options. These settings are user-definable and additional Ferm codes can be added. The drawing settings are used to hatch the strata in *Draw Geologic Column* by matching the strata Ferm code with the Ferm code definitions. Ferm codes are assigned to strata as a strata attribute. To use Ferm codes, each strata should have an attribute called "FERM". The FERM attribute needs to be defined in Define Attributes as a Non-Numeric attribute so that the Ferm code definitions are not limited to just numbers. A Non-Numeric Ferm attribute (a string) allows for Ferm codes containing letters such as 100A. The Ferm code definitions are stored in a file with a .FRM extension. If the window is empty on the initial screen, simply choose load and load the default FRM file from the Carlson DATA folder.

Ferm Codes are lithologic descriptions listed by numeric rock codes (found at <https://www.uky.edu/KGS/coal/coal-core-using-ferm-code.php>), as well as English names. The Ferm codes are derived from the core-logging manual, Cored Rocks of the Southern Appalachian Coal Fields, (<http://www.uky.edu/KGS/pubs/SappCoreBook/SAPP.html>) by J. C. Ferm and G. A. Weisenfluh from the University of Kentucky.



File: C:\scad2005\DATA\ferm.frm

Code	Description	Strata	Fill	Lauer	Color	Scale	Anale
328	DK GREY BURROWE		DASH	FERM	1	50.0	0d
329	DARK GREY SANDY		DASH	FERM	1	50.0	0d
500	SANDSTONE	SS	DOTS	FERM	3	50.0	0d
509	SANDSTONE /FOSS		DOTS	FERM	3	50.0	0d
541	GREY CROSS-BED		DASH	FERM	1	50.0	0d
543	GREY SS W/SHALE		DASH	FERM	1	50.0	0d
544	GREY MASSIVE SA		DASH	FERM	1	50.0	0d
546	CHURNED GREY SA		DASH	FERM	1	50.0	0d

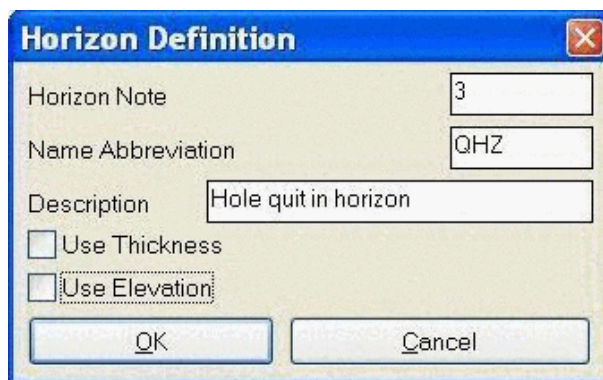
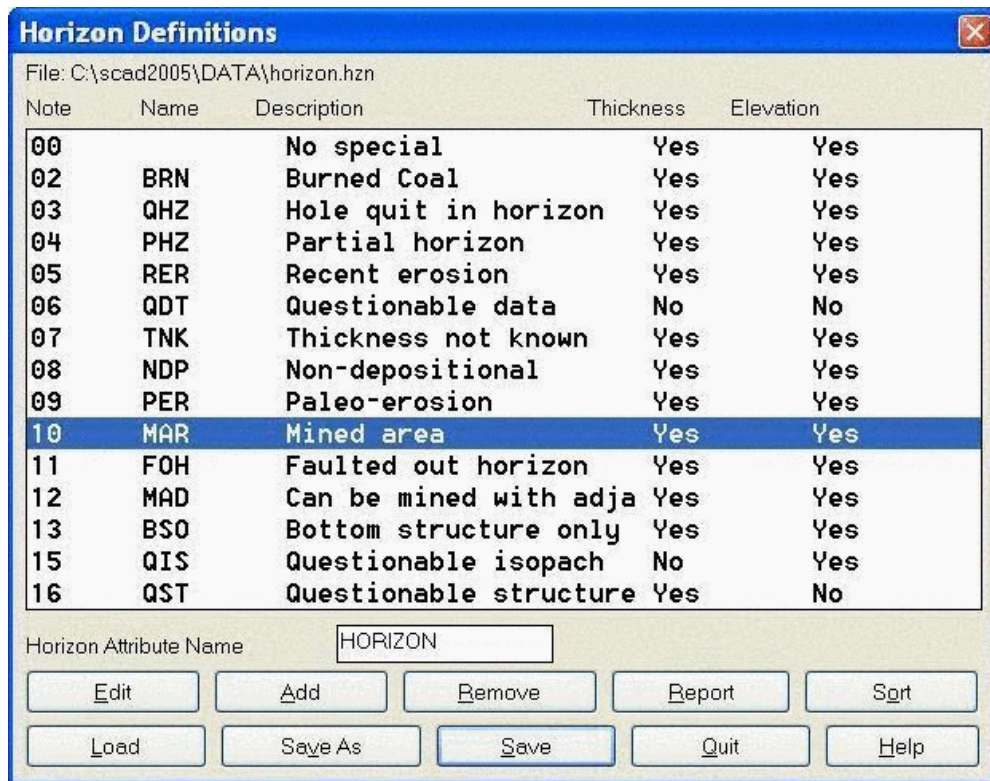
Buttons: Edit, Add, Remove, Sort, Assign, Report, Load, Save, SaveAs, Quit, Help

Pulldown Menu Location: Drillhole
Keyboard Command: fermdef

Define Horizon Codes

This command defines the table of horizon notes. Each horizon note is assigned a unique note number, name, description and processing options. These settings are user-definable and additional horizon notes can be added. The horizon definitions are stored in a file with a .HZN extension.

Horizon notes provide additional information about strata. To assign horizon notes to strata, the strata should have a horizon note attribute. The name of the horizon note attribute is defined in this command at the bottom of the dialog. By default this name is HORIZON, but may be changed. The horizon note from the definition file is matched with the strata by looking up the strata horizon note attribute value in the table. The attribute can be either a number to match by horizon note or a string to match with the horizon name. For example, a strata with an attribute HORIZON and a numeric value of 16 would use Questionable Structure from the table shown below. A strata with a HORIZON attribute of a string value QST would also use the Questionable Structure.



Besides using horizon notes as additional strata descriptors, horizon notes are also applied to strata processing in routines such as Strata Isopach Maps. When reading in the drillholes, if a strata has a horizon note with the Thickness option off, then this strata data will not be used for thickness modeling of this strata. Likewise if a strata has a horizon note with the Elevation option off, then this data point will not be used for elevation modeling of this strata. An application of horizon notes could be with coal section data where you have strata thickness value but the elevation is unknown. In this case, the strata in these drillholes could have a horizon note with the Thickness on and the Elevation off.

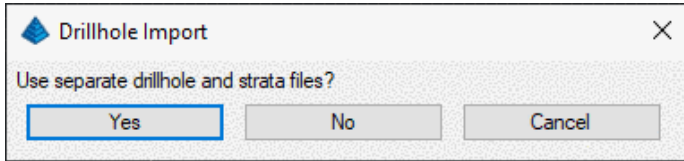
Pulldown Menu Location: Drillhole

Keyboard Command: hordef

Import Drillhole

This command imports drillholes into the drawing from a text file or database. There are many company-specific drillhole file formats available, but the Custom Import Formatter is flexible enough to handle almost any drillhole text file format. There are also two Carlson Standard Text formats and a Carlson Standard Database format that can be used to import from. The Mineral Drillholes format is similar to the Custom Import Formatter, but is intended for working with drillhole formats commonly associated with metal deposits. When the command is first executed, the below dialog will appear.

The Custom Import Formatter is the most flexible option. This option will allow you to select one or two files to define the drillhole locations and the strata information. If all both the collar and strata information are in a single file, choose "no" at this prompt. Note that if you choose "yes", the program will prompt you to select two files to import and will require you to configure two import dialogs.



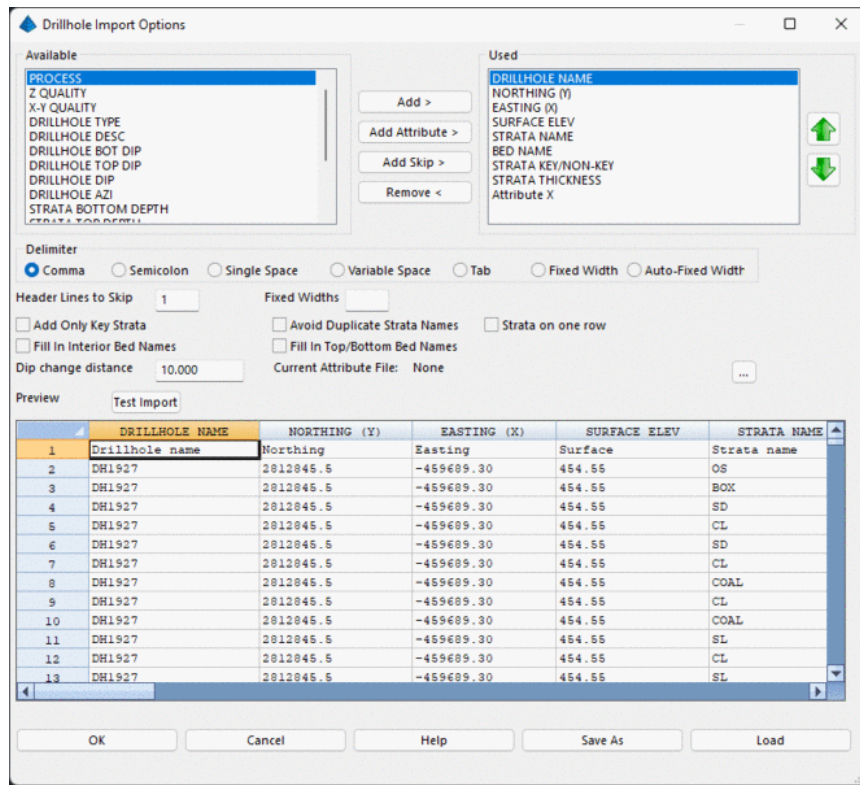
The text files may be in a .txt, .asc, .dat, or .csv file format. It is very common to store the drillhole data in a spreadsheet, then export the data to one of these formats. The text files may be comma delimited, single space delimited, tab delimited, fixed width, or Auto-Fixed width. An example set of drillhole data in a spreadsheet is shown below.

	A	B	C	D	E	F	G	H	I
1	Drillhole name	Northing	Easting	Surface	Strata name	Bed name	Key	Thick	Attribute X
2	DH1927	2812845.5	-459689.30	454.55	OS		NO	15.00	355
3	DH1927	2812845.5	-459689.30	454.55	BOX	BOX	YES	0.01	357
4	DH1927	2812845.5	-459689.30	454.55	SD		NO	11.09	152
5	DH1927	2812845.5	-459689.30	454.55	CL		NO	2.90	456
6	DH1927	2812845.5	-459689.30	454.55	SD		NO	2.00	150
7	DH1927	2812845.5	-459689.30	454.55	CL		NO	5.00	183
8	DH1927	2812845.5	-459689.30	454.55	COAL	MM	YES	3.00	21
9	DH1927	2812845.5	-459689.30	454.55	CL		NO	1.30	190

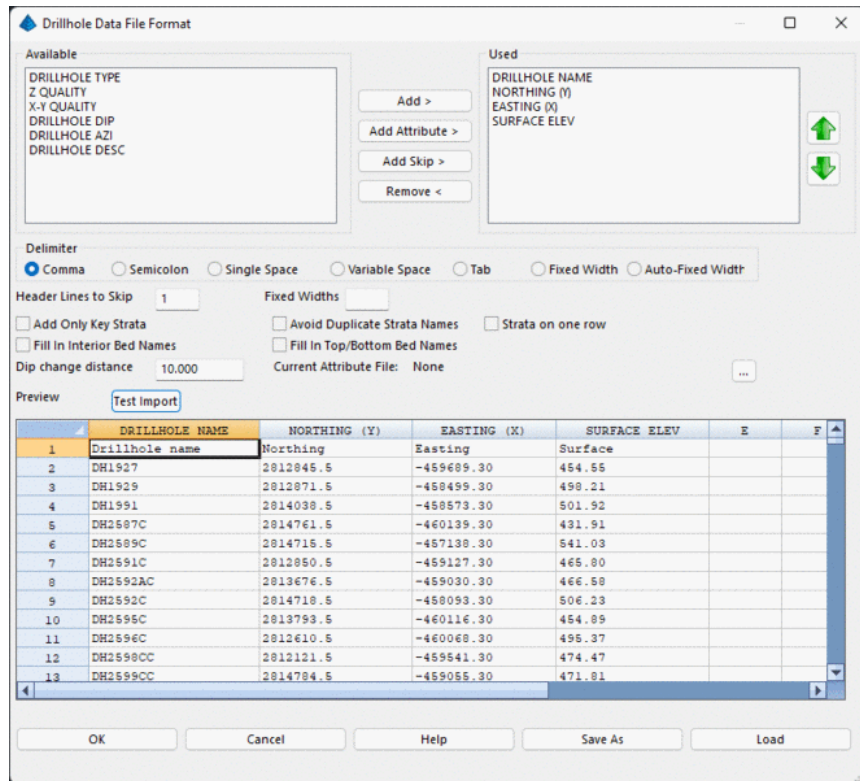
Here it is important to note that a single drillhole is normally expected to span multiple rows in the spreadsheet. Here each sample in the drillhole is represented on a new row. There is an option to import drillholes with each drillhole spanning a single row (with additional columns specified for each strata type). Note that when a drillhole spans multiple rows, data relating to the collar should not change down the rows (northing, easting, and surface elevation), whereas any data relating to the strata should change down the rows (strata name, thickness, elevation, etc.).

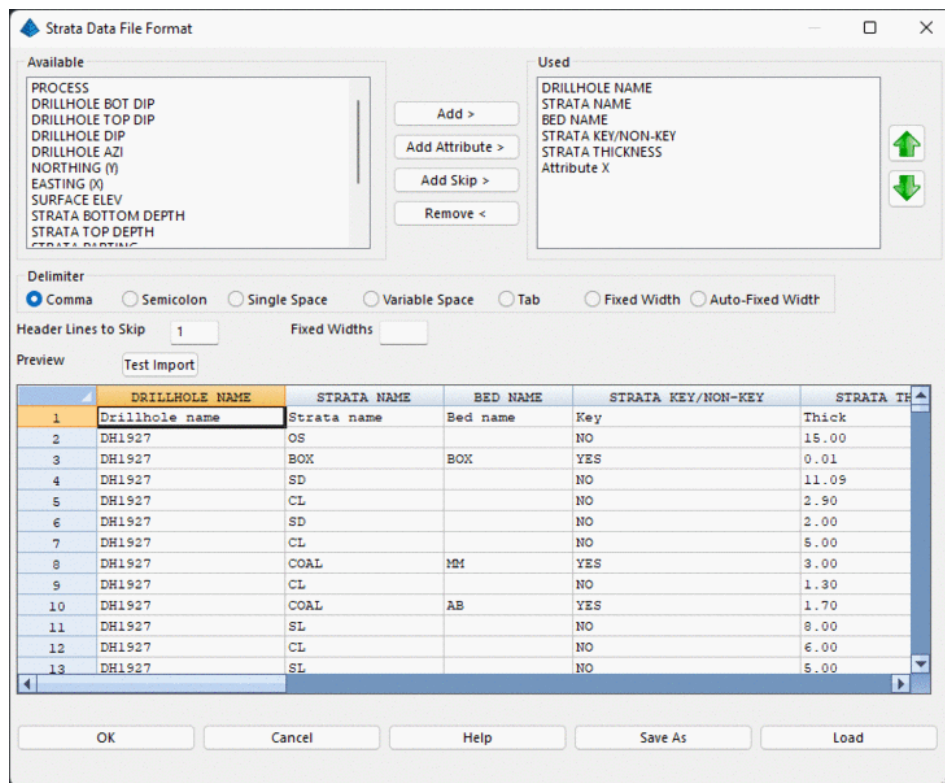
After selecting the file(s) to import, the below dialog will appear. Note that if all of the data is imported in a single file, the first dialog shown below will appear. If two files are specified (one for the collar data, one for the strata data, the following two dialogs will appear).

Single File Imported



Two Files Imported





This dialog is divided into two columns: Available and Used. The idea is to move relevant items from the Available column to the Used column in the order they appear in the drillhole file. Note that the names used in the Available and Used columns do not need to match the header names used in the drillhole file (Carlson always uses a predefined name of "DRILLHOLE NAME", but the header in your file can be something else such as "Drill ID" if you prefer). Not all items are needed, but the ordering of the items is very important. In the above example, the Drillhole Name is the first item in the Used column. You will notice that the Preview Window at the bottom of the dialog shows a spreadsheet-style preview of the file to be imported. Here you can see that the Drillhole Name is the first column of the data, and therefore it must be the first item in the Used list. The second item, Northing, is the second column in the data file. Items that are not properly ordered will result in drillhole data not importing properly.

Items may be moved between the columns by double-clicking them, or by using the **Add** and **Remove** buttons. Custom attributes (assays of the strata, date drilled, water table elevation, etc) may be defined by clicking the **Add Attribute** button. Note that when adding a custom attribute, you will need to specify the attribute as a Drillhole attribute (meaning it doesn't change with depth) or a strata attribute (meaning it will have a unique value for each strata sample in the drillhole). In the above example, Attribute X has been defined as a strata attribute at the bottom of the Used column. Any data that does not need to be imported may be omitted by placing a SKIP in the Used column. This is added by clicking the **Add Skip** button. Items in the used list may be reordered by clicking the green up/down arrows to the right of the list.

When importing drillhole data, you may import elevations, depths, or thicknesses of each sample. You may use any combination of top/bottom elevations, depth, and thickness to specify the sample thickness. Note that if a thickness or depth is specified, they are assumed to be along the length of the drillhole (very important if the holes are angled). If you only import thickness or depth, elevations will be automatically calculated on import (dip angles will be used when determining elevation)

Delimiter: This option specifies the delimiter symbol used to mark columns in the data. For the fixed width format, choose the Fixed Width toggle and then enter the column numbers separated by spaces in the edit box. For example, "8 15 24 32". The Auto-Fixed width will scan entire file first and detect columns if you have some fixed width format - it will detect where breaks between columns are.

Header Lines to Skip: This option allows you to specify how many rows are simply headers for the actual data. These header lines should not be imported as data.

Add Only Key Strata: This option will only import strata tagged as "key". Any gaps between key strata will be filled in with a generic nonkey layer name.

Avoid Duplicate Strata Names: This option will append a number to duplicate strata names within a drill-hole if these strata names do not have bed names. For example, if there are three SH strata names, then they would be imported as 'SH', 'SH2' and 'SH3'.

Strata on one row: This option applies to text files where the entire drillhole is on one row. Each strata is identified by a unique name which is combined with the strata field name. This allows you to have multiple strata value fields such as thickness and name on the same row. For example, consider two strata named COAL_A and COAL_B. When you click the Add button to add the Strata Name, a dialog appears for entering the strata identifier. In this example, you could enter COAL_A. Then click the Add button again for Strata Name and enter id as COAL_B. This creates two strata name fields called COAL_A:Strata Name and COAL_B:Strata Name. Without the Strata on one row option, you can only have one Strata Name per row.

Fill In Interior or Top/Bottom Bed Names: These options will set the bed name for strata that have no bed name to the first bed name found in a strata below the missing bed name strata. If no bed name is found in lower strata, then the program will look for a bed name in the higher strata. In this way, all the strata are assigned bed names. Otherwise only the strata with bed names from the import text file will have bed names in the drillholes.

Dip Change Distance: This value is only needed when the Drillhole Top Dip and the Drillhole Bottom Dip attributes are imported (most users don't use this option). With these two attributes, it is assumed that the drillhole dip angle is only measured at the top and the bottom of the drillhole, rather than on a regular interval down the drillhole. When these top/bottom dip values are imported, the program will distribute the change in dip angle down the hole on this interval. For example, if this value is set to 10, the program would calculate a new dip angle every 10 down the length of the hole. It is important to note that there are three methods that can be used to determine how the angle varies. These three options for the Dip Angle Method are described in Carlson Configure.

Current Attribute File: This option lists the current .atr file associated with the drawing. Clicking the ellipsis button will open the Define Attributes dialog. It is important to note that in order to import String attributes (anything other than a numeric value), you will need to pre-define the attribute as a string entity in the .atr file.

Test Import: This button will check that you have added necessary attributes to the Used column. If any of these attributes are left out, the program will fail to create the drillholes properly. It is important to note that this button does not check for all possible errors in the data; it only checks for the following conditions:

- Drillhole name must be specified
- Drillhole coordinates (X,Y,Z) must be specified
- Strata name must be specified
- Strata length must be specified (using some combination of top/bottom elevations, thickness, and from/to depths)
- Strata intervals cannot overlap

It is worth mentioning that three checks are automatically applied to the drillhole import.

- When both sample thickness and from/to depths are imported, the depths will take precedence over the thickness (depth intervals suggesting an interval of 5.00 feet vs. a thickness value suggesting an interval of 5.02

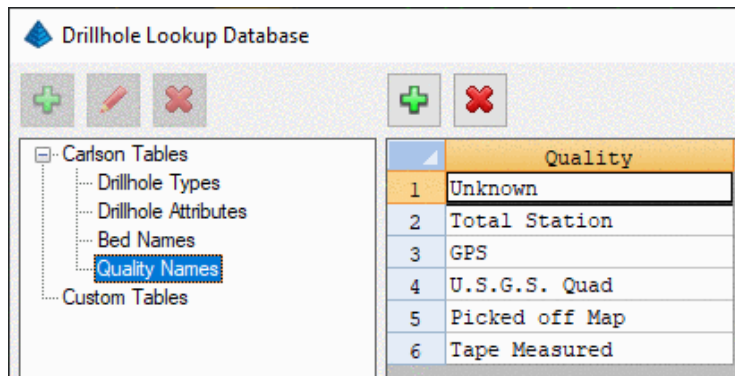
feet would result with an actual interval of 5.00 feet)

- When dip/azimuth values are specified for individual sample intervals as well as the entire drillhole, the sample intervals will take precedence

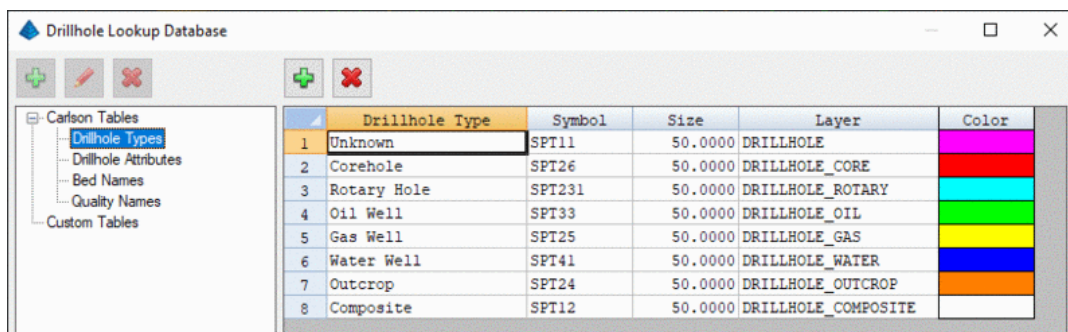
The **Load** and **Save** buttons allow you to save and recall the Custom Import Formatter settings to a settings file with a .IMPT file extension. Note that this file previously used an .IMP extension, but the actual format of the file has not changed. If you have a .IMP file that needs to be loaded, you may simply change the extension to be .IMPT.

Notes on specific attributes to be imported are listed below. Again, it is important to note that not all of these attributes are required, but the attributes that are required are marked with an asterisk (*). For each of these attributes, special characters such as exclamation points (!), pound signs (#), etc. are not recommended (though some are supported)

- *DRILLHOLE NAME - The identifier of the drillhole. Alphanumeric names with spaces are supported
- *NORTHING (Y) - Y coordinate for the drillhole collar
- *EASTING (X) - X coordinate for the drillhole collar
- *SURFACE ELEV - Elevation coordinate for the drillhole collar
- *STRATA NAME - Name of the strata type. These names may be abbreviations or full names (example: OB or Overburden are acceptable). It is common to use this name to identify the type of material (clay, coal, shale, etc.). If bed names are not used, strata names should not repeat down the length of the hole.
- BED NAME - Although not required for all projects, most users will need to specify bed names to handle repeating strata types. For example, if three beds of coal exist in the drillhole (or if each bed were sampled multiple times), the bed name is used to uniquely identify each bed. It is common to use the local name of the formation for the bed name (example: Pittsburgh Seam, Minnekahta Limestone, etc.)
- *STRATA KEY/NON-KEY - This identifies the strata as Key material that is potentially profitable for sale, or Nonkey material that will only ever be treated as waste. If the profitability of a strata is dependent on the quality/assay, it should be tagged as Key. When imported Key/Nonkey values, you may use "Yes" or "1" to identify Key strata. You may use "No" or "0" to identify Nonkey strata.
- *STRATA THICKNESS - Thickness of the sample down the length of the core.
- *STRATA BOTTOM DEPTH - Depth to the bottom of the sample measured from the collar elevation.
- *STRATA TOP DEPTH - Depth to the top of the sample measured from the collar elevation.
- *STRATA TOP ELEV - Top elevation of the sample.
- *STRATA BOTTOM ELEV - Bottom elevation of the sample.
- PROCESS - This sets the Processing option for the drillhole. This is assumed to be on when it is not specified. Note that any drillholes with this option turned off will not be used for any calculations.
- Z QUALITY - Identifies the method in which the collar elevation was measured. The predefined list may be configured in the Drillhole Lookup Database command. Note that when importing this data, an integer value should be used to specify the row on the predefined list. For example, a value of 2 relates to the second method on the predefined list (in the below example, this tells the program that the elevation was measured with a Total Station). If this is not specified, the program will default to the Unknown method.



- X-Y QUALITY - Same as the Z Quality, but specifies the method of measurement for the X and Y coordinates for the drillhole collar.
- DRILLHOLE TYPE - Specifies the type of drillhole, according to the Drillhole Types specified in the Drillhole Lookup Database command. When importing, an integer value should be used to specify the row on the predefined list. For example, a value of 3 tells the program that the drillholes is a Rotary Hole. If this is not specified, the program will default to the Unknown drillhole type.



- DRILLHOLE DESC - A text description of the drillhole.
- DRILLHOLE BOT DIP - The Drillhole Dip (degrees) measured at the top of the hole.
- DRILLHOLE TOP DIP - The Drillhole Dip (degrees) measured at the top of the hole.
- DRILLHOLE DIP - The Drillhole Dip (degrees) for the full length of the drillhole (note there is no variation in dip when this option is used). Note that the definition of 0 dip is defined in Carlson Configure > Mining Settings. By default, the program assumes that a dip value of zero is straight down.
- DRILLHOLE AZI - The Azimuth of the Drillhole Dip.
- STRATA AZI - The Azimuth of the Strata Dip (expected to vary with each strata sample)
- STRATA DIP - The Dip (degrees) of the drillhole at each strata sample (expected to vary with each sample).

It is important to note that if the dip/azimuth measurements are in a separate text file, this information can later be applied to the drillholes with the Import Dip/Azimuth command.

The Carlson Standard Text formats include a complete format that has all the drillhole data options and a simple format that contains the necessary fields. These Carlson format drillhole text files can be created with the Drillhole Export routine. Both formats are shown below. This standard format uses key-coded lines with comma separated entries. String entries are enclosed in single quotes. The first line of the file is a keyword VERSC13.2 to recognize the version of the data file.

The Carlson Standard Database option is the only format that is a database file and not a text file. This database format is an Access MDB or ACCDB file with several specific tables, listed below. When importing with this option, all of the below tables must be present in the database, even if they are empty. When importing from the Carlson database, you can filter by drillhole name, polyline area or query. To import all the drillholes, use the drillhole name option with a name of "*" for everything (this filter will be set by default). The polyline area option will only import drillholes within the selected closed polylines. The query option filters the drillholes by the specified SQL query using the drillhole database fields.

Table	Field	Data Type
DH_DESC	DESCID	Long Integer
	DRILLHOLEID	Integer
	NAME	Short Text
	VALUE	Short Text
DH_DIPAZI	DIPAZIID	Long Integer
	DRILLHOLEID	Integer
	ALONGCHDEPTH	Double
	DIP	Double
	AZI	Double
DH_STRATA	STRATAID	Long Integer
	DRILLHOLEID	Integer
	NAME	Short Text
	ELEVATION	Double
	KEY	Integer
	BED_NAME	Short Text
DRILLHOLE	DRILLHOLEID	Long Integer
	NAME	Short Text
	DESCRIPTION	Short Text
	NORTHING	Double
	EASTING	Double
	ELEVATION	Double
	KEY_DENSITY	Double
	PROCESS	Integer
	DH_TYPE	Integer
	XY_QUALITY	Integer

Z_QUALITY		Integer
MINERAL_DATA		Integer
DRILLHOLE_INTERVAL	ATTRINTERVALID	Long Integer
ATTRID		Integer
DRILLHOLEID		Integer
ATTR_NAME		Short Text
ATTR_FROM		Double
ATTR_TO		Double
ATTR_VALUE		Short Text
ATTR_TYPE		Short Text
STRATA_ATTR	ATTRID	Long Integer
STRATAID		Integer
NAME		Short Text
VALUE		Short Text
TYPE		Short Text

Other Specific Formats are hard-coded imports for data from specific mines or other software formats. Most of the time, the Custom Formatter is used, but if the holes are already in one of these formats, then they can be imported directly. If there is a format that doesn't follow a pattern that the Custom Formatter can use, then the import can be done by custom programming with these Other Formats.



DHDB is a format used by many mining companies. It is an Access file that is produced by the Highland Geocomputing company (<http://www.highlandgeocomp.com>). The ACCDB file contains 3 tables where the formats are shown here for Header, Lithology and Quality.

DHID	Northing	Easting	Collar	Total Depth	Hole Type	Township	Range	Meridian	Section	Qtr Sect	Qtr S
0001L	674,110.92	2,459,227.13	6,896.33	240.00							
0001N	673,403.30	2,460,280.06	6,867.42	260.00							
0002L	671,565.87	2,462,517.18	6,855.30	500.00							
0002N	672,691.51	2,461,522.65	6,863.71	260.00							
0002S	665,135.81	2,458,972.12	7,084.33	430.00							

Drill Hole ID	Easting	Northing	Collar	Named Ur	Bench	From	To	Thickness	Adj From	Adj To	Adj Thick	RQ	Hardne	Shac	Unit Col	GS
0001L	2,459,227.13	674,110.92	6,896.33			0.00	25.00	25.00	0.00	25.00	25.00					
0001L	2,459,227.13	674,110.92	6,896.33	BOX		25.00	240.00	215.00	25.00	240.00	215.00					
0001N	2,460,280.06	673,403.30	6,867.42			0.00	27.47	27.47	0.00	27.47	27.47					
0001N	2,460,280.06	673,403.30	6,867.42	BOX		27.47	27.47	0.00	27.47	27.47	0.00					

Drill Hole ID	Sample Num	Northing	Easting	Collar	seam	bed	Screen Size	Screened Pc	Sample Type	Sample Weig	Sample Date	Sampled By
0004L	0004L_01	671,635.89	2,461,702.16	6,882.31					Incremental			
0014L	0014L_01	670,215.86	2,461,827.15	6,872.30					Incremental			
0014L	0014L_02	670,215.86	2,461,827.15	6,872.30					Incremental			
0014L	0014L_03	670,215.86	2,461,827.15	6,872.30					Incremental			
0014L	0014L_04	670,215.86	2,461,827.15	6,872.30					Incremental			

The Import Mineral Drillholes option is very similar to the Custom Import Formatter, with a few exceptions:

- Dip and Azimuth must be specified
- Strata Names are not required, but a Rock Code must be specified
- Drillhole Length must be specified

Prompts

Select Drillhole Configuration File .ch file created by Define Drillhole. This dialog appears once. To change Configuration file use Mining Project Manager.

Choose Format Dialog

The prompting for other import formats may be different.

Pulldown Menu Location: Drillhole > Import/Export Drillholes

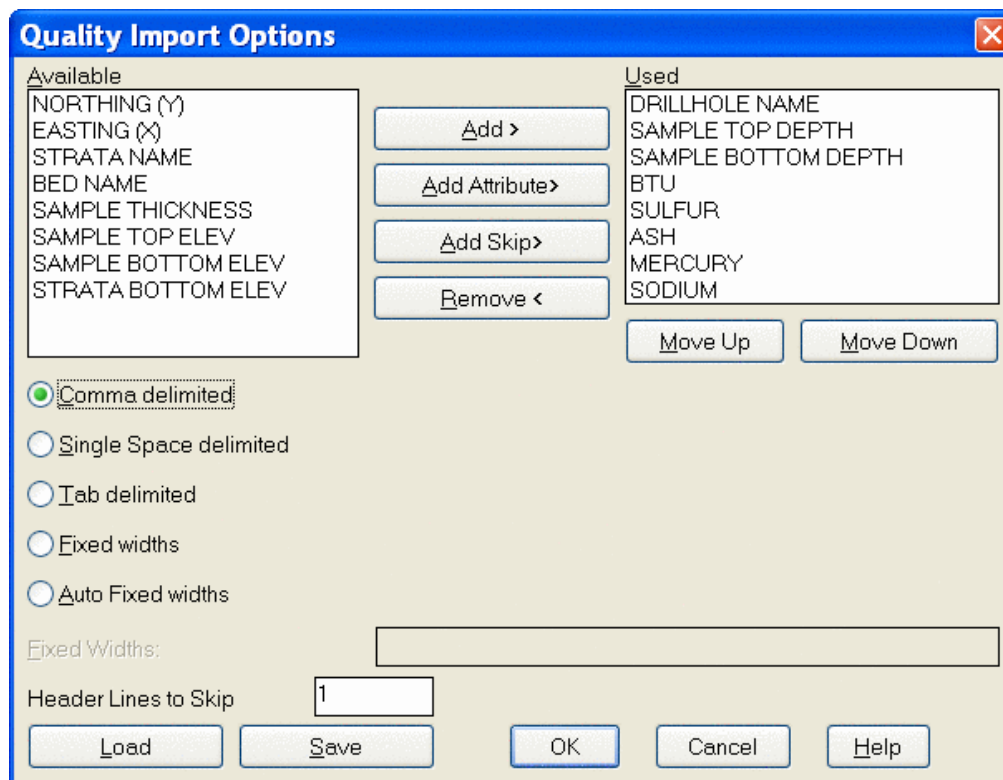
Keyboard Command: chimport

Import Qualities

Many times the lab analysis is received long after the structure data for new drillholes has been entered. This command imports attribute data from a text file into drillholes that are already existing in the drawing. Besides the attribute values, the text file must include the drillhole name or the drillhole northing and easting and a strata or bed name, from and to, or elevations. These values are used to locate where to assign the attribute values.

The order of the columns is set in the dialog shown below. To add one of the standard column types, highlight the name from the Available list and click Add. To add an attribute, click the Add Attribute button. Another dialog will appear for entering the attribute name.

To assign qualities to the drillhole by strata elevation, the program needs both the top and bottom elevation of the sample. This can be done by 2 elevations, by 2 depths or by elevation and thickness. If the sample doesn't match up with what is in the drillhole, it will split the strata and beds to match up with the sample interval from the quality file. To alleviate this, just use bed name instead of an interval to match up.



Prompts

Select the Drillholes to update.

Select objects: *pick the drillholes*

Choose Quality Text file to read

Added 30 quality values.

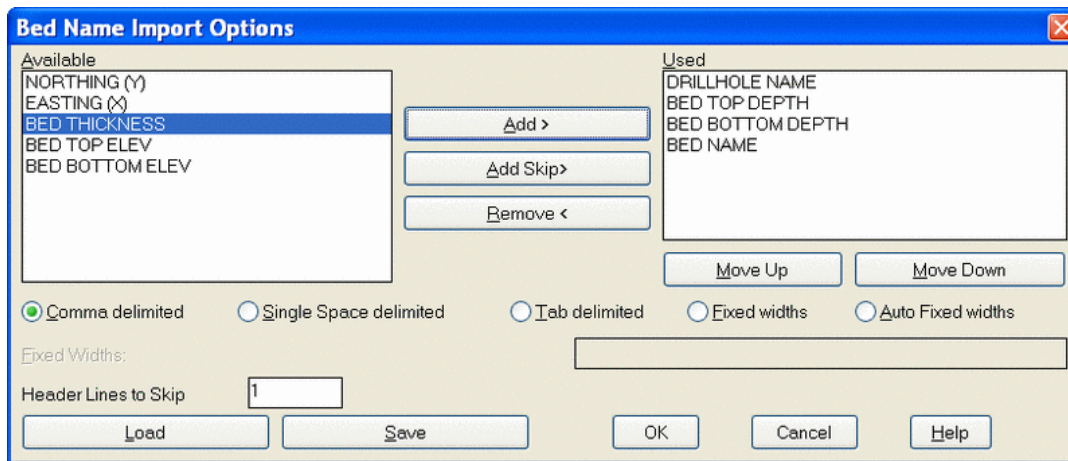
Pulldown Menu Location: Drillhole

Keyboard Command: chimport2

Import Bed Names

This command imports bed names from a text file into drillholes that are already existing in the drawing. Sometimes when importing drillhole information, there is a separate file for lithology. This will bring that file into the holes as a bed name. Besides the bed names, the text file must include the drillhole northing and easting and a strata identifier of thickness, depth, or elevation. These values are used to locate where to assign the bed names. The text file should have comma separated columns of data with each row containing all the data for the strata attributes.

The order of the columns is set in the dialog shown below. To add one of the standard column types, highlight the name from the Available list and click Add. To add an attribute, click the Add Attribute button. Another dialog will appear for entering the attribute name.



Prompts

Select the Drillholes to update.

Select objects: *Pick the drillholes*

Added 107 bed names.

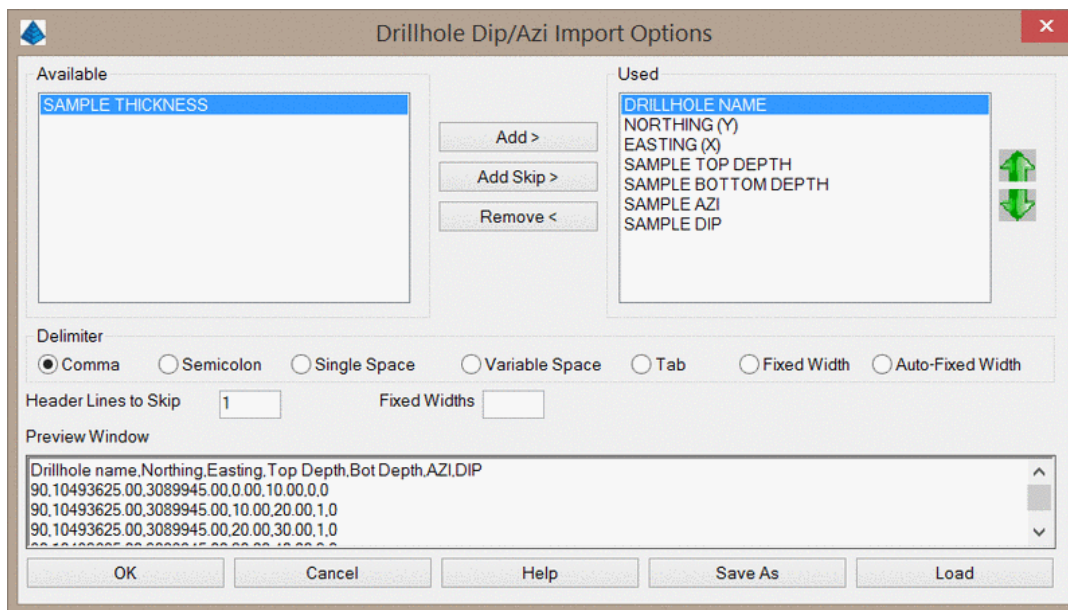
Pulldown Menu Location: Drillhole > Import/Export Drillholes

Keyboard Command: chimport3

Import Dip/Azimuth

This command imports dip and dip azimuth survey data that does not match the strata sample interval. For example, if a limestone core is quality sampled on 2' intervals, but the dip and azimuth readings are taken on a 5' interval, this command can be used to keep track of the drillhole dip without modifying the sample interval. This command will import the dip/azimuth information to each drillhole's Dip/Azimuth Table rather than applying the dip/azimuth values as new strata attributes.

When the command is executed, you will be prompted to select the drillholes to update. After selecting the drillholes, you will be prompted to select a text file containing the dip/azimuth data. The file must include the drillhole name or the drillhole Northing/Easting. After selecting the file, the below dialog will appear. Here you can specify which keyword matches each column in the text file. To add the dip/azi information, you must specify the top/bottom depth values or thickness values for each sample.



Pulldown Menu Location: Drillhole

Keyboard Command: chimport4

Coal Section to Drillhole

This command converts the strata data in coal sections into drillhole format. Coal sections are created in the Works pull-down menu of the Standard Mining Module and consist of strata names with thicknesses. Drillholes consist of a surface elevation and strata with top and bottom elevations. Since coal sections contain no real world elevation, the program needs two elevations to assign Z values to make the drillholes. The two elevations are the surface elevation and the top of section elevation. These elevations can be entered for each drillhole or calculated from a surface model defined in a grid file. The top of section elevation represents the top elevation of the first strata in the section. The top and bottom strata elevations for the drillhole are calculated using the top of section elevation and the thicknesses.

The program also prompts for the strata names to use in the drillhole in case the drillhole names differ from the coal sections names. For example, consider this coal section:

C-7
R-3
C-54

In order to avoid duplicate strata names in the drillholes, the two coal section strata named *C* could be named *Coal 1* and *Coal 2* for the drillholes.

Prompts

Drillhole Configuration File Dialog

Choose a drillhole configuration file (*.ch).

Coal Section Configuration File Dialog

Choose a coal section configuration file (*.sc).

Select coal sections.

Select objects: *pick coal section symbols*

Select Surface Grid

Optionally choose a grid file that models the surface elevation. The drillhole surface elevation will be derived from this grid file. Choose Cancel if there is no surface grid and the program will prompt for the surface elevations.

Select Top of Section Grid

Optionally choose a grid file that models the elevation of the top of the first strata in the coal section. Choose Cancel if there is no top of strata grid and the program will prompt for the coal section elevations.

Enter strata name for section coal <COAL>: *press Enter*

Is strata COAL key or non-key (<Non-key>/Key)? *Key*

Enter strata name for section rock <ROCK>: *press Enter*

Enter DrillHole name: *15-A*

Enter DrillHole description: *press Enter*

Pulldown Menu Location: Drillhole > Import/Export Drillholes

Keyboard Command: chconvert

Reassign Database File

This command will prompt for the new database file that contains the geologic data of the drillholes on screen.

Prompts

Select the Drillholes to reassign database file.

Select objects: Specify opposite corner: 216 found

Select objects:

Pulldown Menu Location: Drillhole > Import/Export Drillholes

Keyboard Command: chsetmdb

Audit Database File

This command will prompt for the database file to audit. The audit removes strata records that reference a drillhole that doesn't exist and removes attribute records that reference a strata that doesn't exist.

Pulldown Menu Location: Drillhole > Import/Export Drillholes

Keyboard Command: audit_cdb

Convert Drillholes to External Database/Convert Drillholes to Drawing Data

These commands perform a quick and easy two-way conversion from/to internal (DWG) or external (MDB) storage of drillhole data. This is used in converting the drillholes from one format to another. Before running either of these commands, the desired setting whether to use the external database or not must be set in the current drillhole configuration file (*.ch). There are no prompts, just processing time. Verify the conversion with an Edit Drillhole. The method of storage is listed on the screen.

Pulldown Menu Location: Drillhole > Import/Export Drillholes

Keyboard Command: chpts

Export Drillholes

This command exports drillhole data to one of three formats: Simple, Full, and Custom. The Simple export contains minimal data about the drillholes, while the full report contains all drillhole data (both will be written to a .txt or .csv file). The custom option will allow you to choose which drillhole data to export. For more information regarding the Custom option, see the Custom Drillhole Report command.

Prompts

Choose text file to create

Drillhole export file format (Simple/<Complete>)? *press Enter*

Select the Drillholes to export.

Select objects: *pick the drillholes*

Writing file c:\drawings\data\drillhole.txt

Pulldown Menu Location: Drillhole > Import/Export Drillholes

Keyboard Command: chexport

Drillholes to Points

This command creates points in the current coordinate file (.CRD) for the strata data from the selected drillholes. The current coordinate file can be set with the Set Coordinate File command under Points. The x,y position for the point comes from the drillhole position. The z value can be the strata elevation, thickness or attribute. The points can be drawn on the screen in addition to being stored to the CRD file. There is an option to create composite values from multiple strata. The drillhole name can be used as the point number or the description of the points. If the drillhole is not used as the point description, then there is a prompt to specify the point descriptions.

Prompts

Select drillholes and strata elevation polylines.

Select objects: *pick the drillholes*

Plot the points (<Yes>/No)? *N for No*

Create composite points (Yes/<No>)? *press Enter for No*

Use drillhole names for [Point#/Desc/<None>]? *press Enter for None*

Choose a Strata to Process dialog

Choose Value to Process dialog

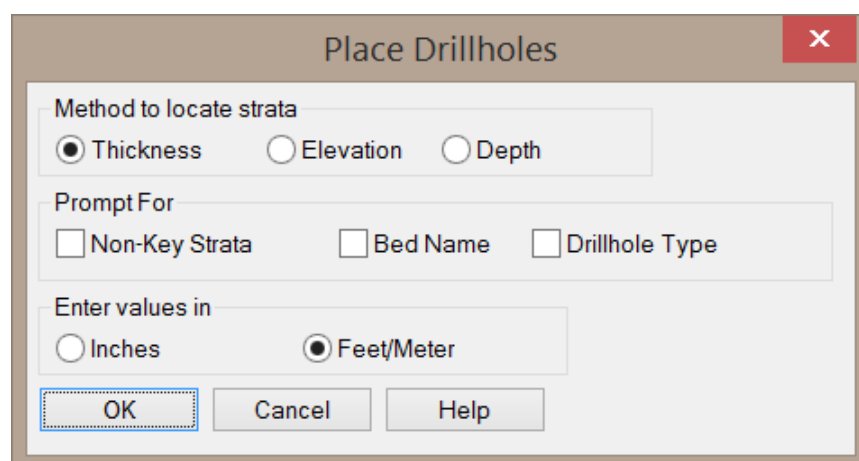
Description for points <C2_KEY_BTU>: *press Enter*

Pulldown Menu Location: Drillhole

Keyboard Command: chpts

Place Drillholes

This command generates drillholes in the drawing one at a time with a series of command line prompts. If the drillhole data is available in digital format, it is quicker to import the information with the Drillhole Import command. When the command is executed, the below dialog will appear.



Method to Locate Strata: This option determines which prompt you will receive to define the limits of each strata.

Prompt for: When these options are enabled, you will receive additional prompts when entering drillhole information.

Non-Key Strata: When this option is disabled, you will be prompted to enter information only for key strata. In this scenario, you will be prompted to enter the thickness and bottom depth/elevation of the key strata, with non-key strata filling in the gaps between the key strata. The non-key strata will be named automatically. For example, a key strata named "Strata A" will be overlain by a non-key strata named "Strata A.OB". When this option is enabled, you will be prompted to enter information for both key and non-key strata, and you will be able to assign a specific name to the non-key layers.

Bed Name: When this option is enabled, you will be prompted to enter a Bed Name for each strata. Bed names are used to group strata together.

Drillhole Type: When this option is enabled, you will be prompted to set a drillhole type. This controls appearance of the drillhole symbol, which can be configured in the Define Lookup Database command.

Enter Values in: This option sets the units of the length values being entered.

After clicking **OB**, you will receive a series of command line prompt. You will first specify the X and Y location of the drillhole followed by the surface elevation and drillhole description. If drillhole definition includes additional descriptions you will be prompted for values, these values are optional, so you may press Enter to continue. After entering this drillhole header information, a loop for entering the strata from top to bottom begins. If an undefined strata name is entered, there is an option to define that strata from there (see Define Strata for more on strata definitions). When all the strata are entered, press Enter to exit the loop. This draws the drillhole in the drawing as an INSERT entity in the current DRILLHOLE layer with all the data attached. From there, another drillhole can be entered or the routine can be exited by pressing Enter. At any time in this routine, entering 'Undo' backs up the prompting sequence.

Prompts

File Selection Dialog

Choose a drillhole configuration file.

Enter or Pick Drillhole x,y location (Enter to End): 5000,5000

Enter Drillhole surface elevation ('U' to Undo): 890.3

Enter Drillhole description ('Undo' to Undo): CH-11 This field is optional.

From here begins a loop to enter the strata in top to bottom order.

Enter strata name (Enter to End, 'Undo' to Undo): OV

Strata OV is a Non-Key strata. OV was found in the strata definitions as a Non-Key strata.

Enter bottom elevation of OV ('U' to Undo): 883.7

Now enter the Non-Key attribute values for the attributes defined by Define Drillhole.

Enter value for OV attribute ACID ('U' to Undo): 1

Enter value for OV attribute BASE ('U' to Undo): 2

Enter strata name (Enter to End, 'Undo' to Undo): C

Strata C is undefined? Define now (<Yes>/No)? No Defining strata C is optional.

Is C a Key or Non-Key strata (Key/<Non-Key>/Undo)? Key All must be set as either Key or Non-Key.

Enter bottom elevation of C ('U' to Undo): 879.7

Now enter the Key attribute values.

Enter value for C attribute BTU ('U' to Undo): 10000

Enter strata name (Enter to End, 'Undo' to Undo): *press Enter* if C is the last strata in the drillhole. Otherwise enter the next strata name.

Enter or Pick Drillhole x,y location (Enter to End): *press Enter to exit* Or specify another location for the next drillhole.

Pulldown Menu Location: Drillhole

Keyboard Command: chplace

Spot Drillholes

This command creates drillholes from existing drillholes and surface entities or from a predefined Geologic Model file. You supply the x,y coordinate for the new drillhole and the routine will calculate the surface elevation and the strata elevations and attribute values based on the other drillholes and surface entities or the grids in the Geologic Model. This feature is useful for providing drillers with an estimate of the geologic column drilling exploration holes.

Prompts

Select DrillHole Configuration File (.ch file created by Define Drillhole)

Make Grid File Dialog *choose a grid resolution*

Select Drillhole Type *choose a type*

Reading drillhole 42

Finding splits ...

Finding pinch out ...

Calculating seam stacking ...

Use drillhole surface elevations in surface model [Yes/<No>]? *N*

Ignore zero elevations [<Yes>/No]? *Y*

Reading points ... 5365

Ignored 306 points with zero elevation.

Ignored 770 duplicate points.

Inserting breaklines 5684 ...

Triangulating points ... 5365

Assigning grid values> 55000

Pass> 36 Null Z values left> 0

Choose modeling method [<Triangulation>/Inverse dist/Kriging/Polynomial/LeastSq]? *T*

Apply global trend to strata extrapolation [Yes/<No>]? *N*

Use Triangulation Subdivision [Yes/<No>]? *N*

Enter or pick drillhole location (ENTER to end):

Enter Drillhole name: Spot1

Enter Drillhole description: Spot

Triangulating points ... 42

Assigning grid values> 55000

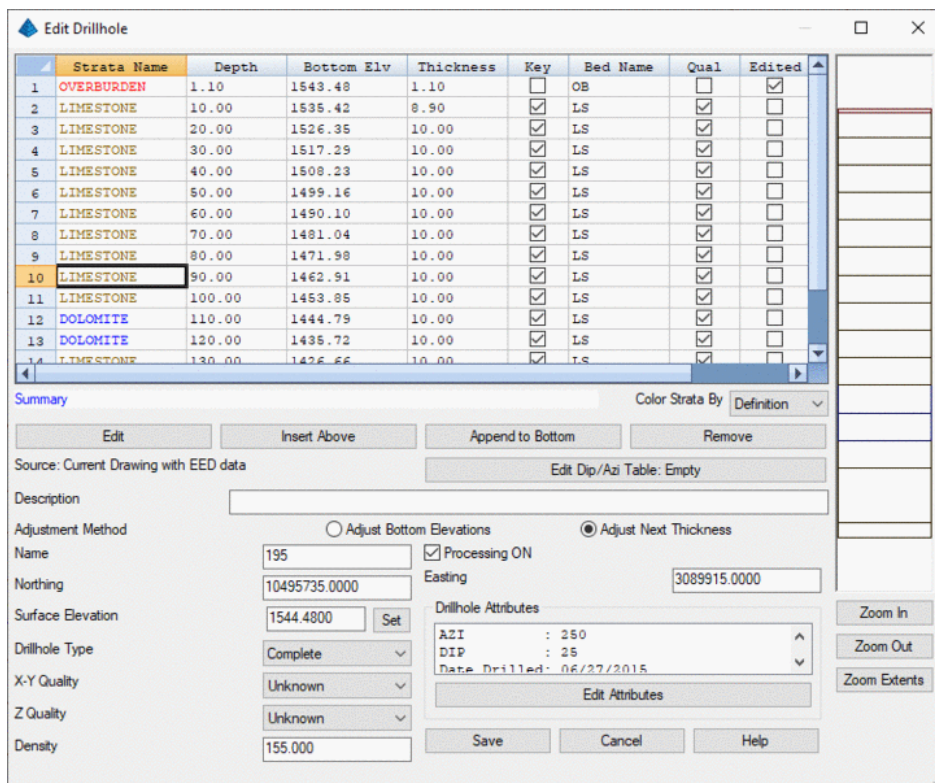
Enter or pick drillhole location (ENTER to end):

Pulldown Menu Location: Drillhole

Keyboard Command: chspot

Edit Drillhole

Edit Drillhole is a dialog based editor for viewing and editing drillhole data one hole at a time. The Edit Drillhole dialog lists the strata/bed layers, drillhole header information, and a graphical representation of the drillhole itself. The main dialog is shown below.



Each strata layer in the hole is listed as a separate row in the main table. These strata layers, along with the graphical representation of the hole, are color-coded according to the current strata definition file (from Drillhole Pulldown Menu > Define Strata) by default, but may also be colored according to a grade parameters file (from Block Model Pulldown Menu > Define Grade Parameters) by clicking the **Color Strata By** pulldown menu. When the Color Strata By option is set to 'Grade', you will be prompted for a Grade Parameters File, after which the strata layers will be colored according to the grade set in that file. An example is shown below, wherein the strata layers are colored according to their concentration of calcium oxide.

Strata Name	Depth	Bottom Elv	Thickness	Key	Bed Name	Qual	Edited
1 OVERBURDEN	1.00	1543.57	1.00	<input type="checkbox"/>	OB	<input type="checkbox"/>	<input checked="" type="checkbox"/>
2 LIMESTONE	10.00	1535.42	9.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
3 LIMESTONE	20.00	1526.35	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
4 LIMESTONE	30.00	1517.29	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
5 LIMESTONE	40.00	1508.23	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
6 LIMESTONE	50.00	1499.16	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
7 LIMESTONE	60.00	1490.10	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
8 LIMESTONE	70.00	1481.04	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
9 LIMESTONE	80.00	1471.98	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
10 LIMESTONE	90.00	1462.91	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
11 LIMESTONE	100.00	1453.85	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
12 DOLOMITE	110.00	1444.79	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
13 DOLOMITE	120.00	1435.72	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
14 LIMESTONE	130.00	1426.66	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
15 LIMESTONE	140.00	1418.53	10.00	<input checked="" type="checkbox"/>	LS	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Thick:40.000 Elev:1524.480 to 1484.480 Depth:20.000 to 60.000

Color Strata By Grade

Selecting multiple strata in the list (by clicking and dragging in the list or by holding CTRL while clicking strata) will display a summary of the selected strata in blue text below the list, as shown in the above image.

To edit a strata, highlight the strata in the list, and then click the **Edit** button, or simply double-click the strata in the list. Edits may also be made by single-clicking a value and typing in the new value. Note that when a strata or quality is manually edited, the **Edited** checkbox will be checked.

To insert new strata, highlight the strata below the desired location and click the **Insert Above** button.

To add a new strata to the end of the list click the **Append to Bottom** button.

Strata are removed by highlighting the strata and clicking the **Remove** button.

The **Adjustment Method** controls how strata will react to a change in thickness, elevation, or depth of another strata. If the **Adjust Bottom Elevations** option is selected, the thickness of all strata will be held constant as the change is applied. For example, if the thickness of the strata layer in row 1 is changed from 1 to 5, then the thicknesses of all strata below it will be held constant while the bottom elevation of each will decrease by 4. If the **Adjust Next Thickness** option is selected, elevations will be held constant while the thickness of the next strata is adjusted. For example, if the thickness of the strata layer in row 1 is changed from 1 to 5, then the thickness of the strata layer in row 2 will change from 9 to 5; no other strata layers will be affected.

When Strata are edited or added, the Edit Strata dialog appears as shown below. This dialog allows editing of information specific to the selected strata layer.

	Name	Value
1	CAO	30.750
2	MGD	21.470
3	SID2	0.084
4	AL2O3	0.033
5	FE2O3	0.056
6	S	0.008
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		

The left side of the Edit Strata dialog lists all strata in the hole. Selecting one of the strata from this list will allow you to edit that strata. The fields at the top of this dialog include the Strata Name, Bed Name, Strata Thickness, Bottom Elevation of the Strata, the Depth to the bottom elevation of the strata from the top of the hole, and the strata's key status (key or non-key). Each of these fields may be edited.

The **Select** button next to the Strata Name will allow you to assign the name by picking from a list of names the Strata Definition File. Similarly, the **Select** button next to the Bed Name field will let you select the bed name from the Bed Names Table in the Lookup Database file.

The table shown on this dialog lists all attributes specific to the selected strata. Attributes and their corresponding values may be entered/edited by simply clicking the field and typing.

The **Return** button will save any changes made to the strata and return to the main Edit Strata dialog.

The list of attributes may be sorted by clicking the Name or Value header at the top of the spreadsheet.

The right side of the Edit Drillhole dialog shows a graphical representation of the drillhole. Each strata layer will be colored according to the option selected in the **Color Strata By** pulldown menu. Hovering the mouse cursor over any strata layer will display the depth to the bottom of that strata layer from the top of the hole.

The window showing the graphical representation of the drillhole allows for zooming and panning to more easily inspect strata layers. Panning is done by holding down the mouse wheel while moving the mouse. Zooming is done by scrolling the mouse wheel, or by using the **Zoom In**, **Zoom Out**, or **Zoom Extents** buttons.

The Drillhole Header information is listed in the bottom half of the Edit Drillhole dialog. This includes the drillhole description, name, northing coordinate, easting coordinate, processing status, surface elevation, drillhole type, X-Y quality, Z quality, density, and drillhole attribute fields.

Above the drillhole header information is the drillhole **Source**. If the drillhole information is stored in the drawing itself (as is most common), the source will be 'Current Drawing with EED (Extended Entity Data) Data'. If the drillhole information is stored in an external database, the database directory will be listed as the source. Any changes made to the drillhole will update the EED/database after clicking **Save**.

The **Description** is a simple description of the hole for notes, comments, etc.

The **Name** is the drillhole identifier.

The **Processing ON** checkbox determines if the hole is used when making Strata Grid Files. If this box is unchecked, the drillhole information will not be used to make Strata Grid Files.

The **Northing** and **Easting** list the Y and X coordinates of the drillhole, respectively.

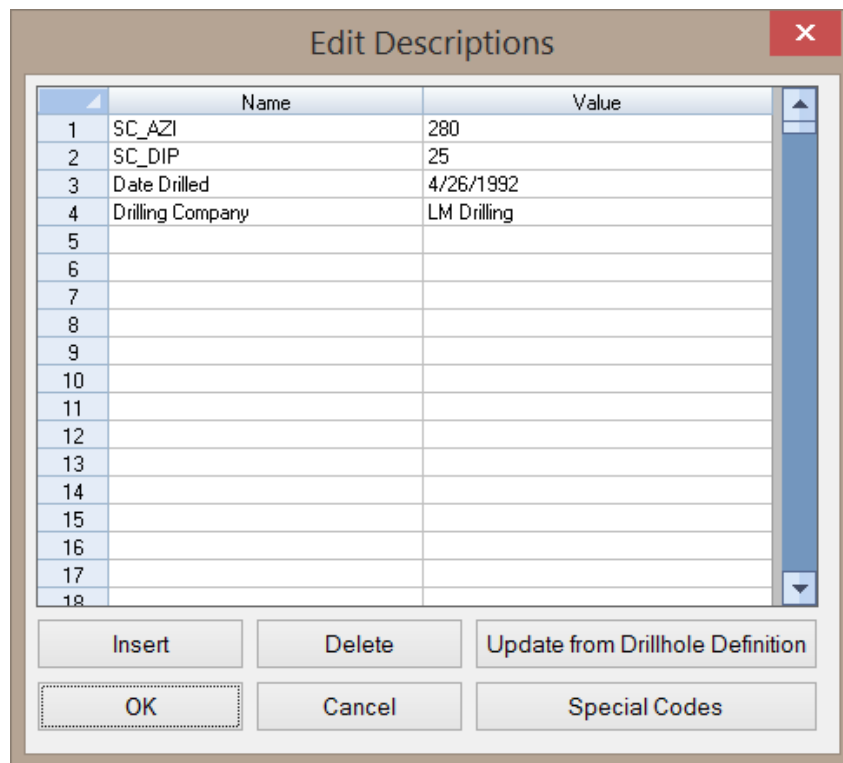
The **Surface Elevation** lists the elevation at the top of the hole. The **Set** button next to this field will set the hole elevation according to a .grd or .tin surface file.

The **Drillhole Type** will change the appearance of the hole according to the settings in the Drillhole Lookup Database command. These settings determine the drillhole, layer, symbol, color, etc.

The **X-Y Quality** and **Z Quality** are simple notes that tell how the drillhole X-Y coordinates and surface elevation were measured, respectively. Default options include Unknown, Total Station, GPS, USGS Quad, Picked off Map, and Tape Measured. Additional options may be added via the Drillhole Lookup Database command.

The **Density** sets the default density for strata in either lbs/ft³ or kg/m³ depending on the unit system selected under Settings Pulldown Menu > Drawing Setup. Note that density may be applied to individual strata/beds as user-defined attribute.

The **Attribute Fields** section lists attributes that apply to the drillhole itself rather than to specific strata, such as date drilled, driller's name, etc. Clicking the **Edit Attributes** button will open a new dialog, as shown below.



This dialog allows you to type in custom attributes that apply to the entire drillhole. Rows may be inserted or deleted with the **Insert** and **Delete** buttons. The **Update from Drillhole Definition** button will add any attributes listed in the drillhole definition file (from Drillhole Pulldown Menu > Define Drillhole).

Four optional, special attributes are also available, and may be set by clicking the **Special Codes** button. These attributes are:

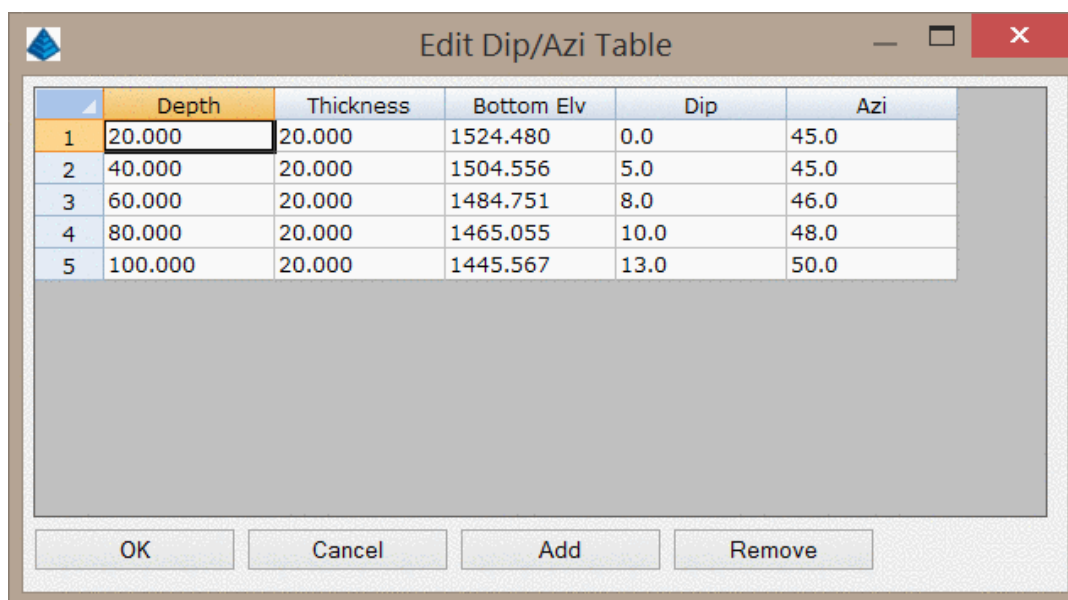
- SC_AZI or AZI: For dipping holes, this value sets the azimuth of the dip direction (0 is directed north, 90 is east, etc.)
- SC_DIP or DIP: For dipping holes, this value sets the angle (in degrees) of the hole. By default, a dip angle of 0 relates to a vertical hole, but this may be modified within Carlson Configure > Mining Settings.
- BELOW: This code is used when a strata is known to exist below the extents of the drillhole. The BELOW code is followed by the depth from the bottom of the hole to the top of the strata. The strata name is listed as the value for this code. For example, consider a strata, "Rock" that is known to exist 20' below the bottom of the hole. In this case, the attribute name is "BELOW20" and the value is "Rock".
- FAULTED: This code is used to skip pinchout and conformance for this drillhole.

Drillhole Dip and Azimuth is controlled by a hierarchy of options, listed below. If Option 1 is detected, the other two will be ignored. If Option 1 is not detected, then Option 2 will be used. If Option 2 is not detected, then Option 3 will be used. If none of these options are detected, the program will assume a vertical drillhole.

1. DIP/AZI Table
2. Strata DIP/AZI attributes
3. Drillhole DIP/AZI attributes

The DIP/AZI Table, shown below, allows the dip of the drillhole to vary on an interval that does not match the strata

samples. This information can be imported with the Import Dip/Azimuth command.



	Depth	Thickness	Bottom Elv	Dip	Azi
1	20.000	20.000	1524.480	0.0	45.0
2	40.000	20.000	1504.556	5.0	45.0
3	60.000	20.000	1484.751	8.0	46.0
4	80.000	20.000	1465.055	10.0	48.0
5	100.000	20.000	1445.567	13.0	50.0

Select Drillhole to edit [\langle Pick \rangle /Name]: pick a hole or type "N" and then type in a name. A double-click with the cross-hairs on a drillhole will also execute this command, but only on the edge of the symbol, not the center where the grip is.

Pulldown Menu Location: Drillhole

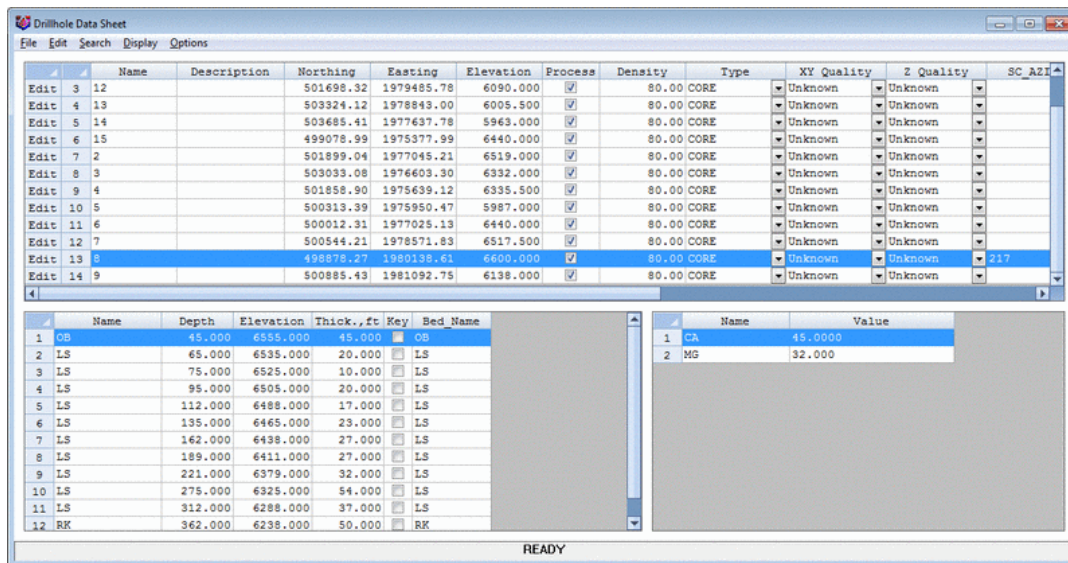
Keyboard Command: chedit

Drillhole Data Sheet

This function allows you to edit data multiple drillholes in a spreadsheet environment. The editor consists of three inter-linked spreadsheets: drillhole data, seam data and attribute data. The strata information corresponds to the currently selected drillhole in drillhole spreadsheet and in-turn the attribute spreadsheet shows data for the highlighted strata. The information changed in the editor will be saved into drawing or the external database on the exit. There is a prompt whether to save changes when exiting.

In any of the spreadsheets, additional entries may be created by moving down from the lowest line of the spreadsheet. The newly created entry needs to be filled in before another one may be created or before saving the spreadsheet. The *Del* key removes the highlighted line from spreadsheet. Drillholes deleted from spreadsheet do not get deleted from the drawing, but all changes made to the deleted drillhole are lost. The *Ins* key inserts an entry above the highlighted one, which is especially handy when you need to insert strata. The *Tab* key switches between spreadsheets. If an entry is invalid or a compulsory field has not been defined the cursor will stay in that field and a message will display in the bottom line of the editor. The incomplete entry should be finished or deleted before proceeding. Changes between Key and Non-key type of strata may change the set of strata attributes. Therefore the values of some attributes will be lost in the transition.

Using the Edit button next to each drillhole name brings up the Edit Drillhole screen, defined elsewhere in the Help manual.



The appearance of the editor may be modified by hiding unwanted columns using Display pulldown and rows may be hidden by selecting unwanted rows and selecting Edit-Hide. To show rows again select rows above and below and pick Edit-Show. The Options pulldown contains options for units selection and the method of elevation adjustment. These options are saved on the exit, so that next time the editor resets to these defaults.

Prompts

Select Drillhole Configuration File (this prompt appears once. Use Set Drillhole Definition to change current Drillhole Definition)

Select Drillholes: *pick Drillholes to load into editor.*

Pulldown Menu Location: Drillhole

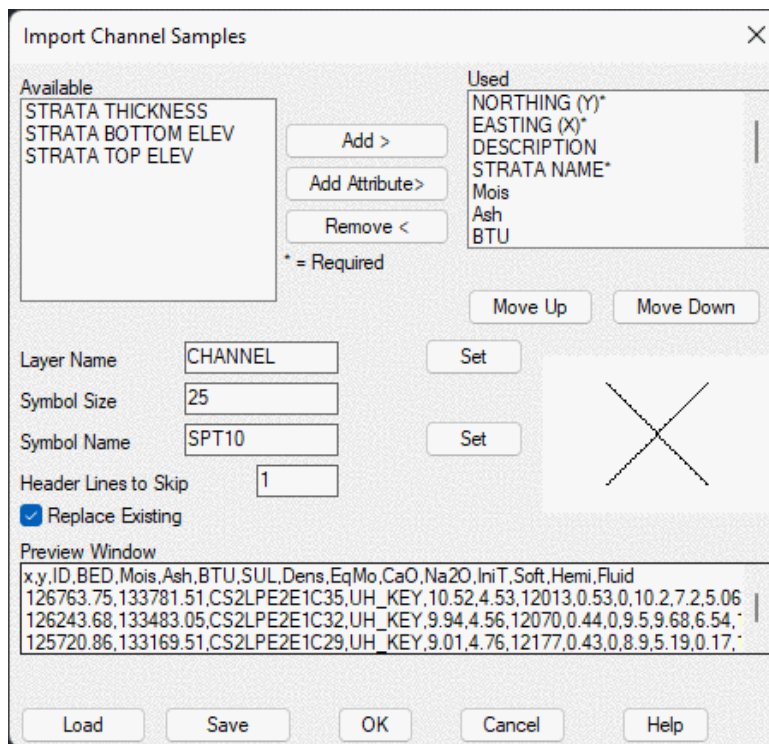
Keyboard Command: spread

Pit/Channel Samples - Import from Text File

This command creates channel samples from comma separated data in a text file. The command starts with a dialog to choose the symbol, symbol size and order of the data columns. Besides the attributes values, the text file must include the sample northing and easting and a strata identifier of name or elevation. The text file should have comma separated columns of data with each row containing all the data for the sample.

The order of the columns is set in the dialog shown below. To add one of the standard column types, highlight the name from the Available list and click Add. To add a user-defined attribute such as BTU, click the Add Attribute button. Another dialog will appear for entering the attribute name.

Channel Samples can be used as "partial" drillholes or spot samples. They are used in conjunction with the drillholes for modeling.



The option to **Replace Existing** will ensure that channel samples are not duplicated when imported. In order for a channel sample to be considered a duplicate, it must match all of the below criteria when compared to channel samples that currently exist in the drawing. If any

- Same X-Y coordinates
- Same Strata name
- Same Description
- Contains at least one of the same attribute (BTU, ASH, etc.)

Prompts

Import Channel Samples dialog

Select Text File dialog

Created 4 pit/channel samples.

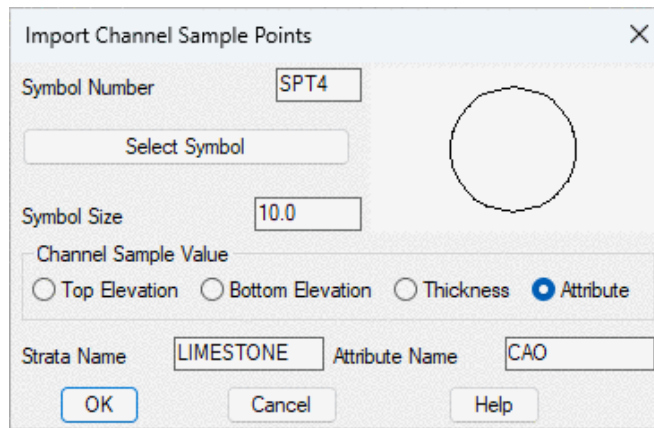
Pulldown Menu Location: Drillhole

Keyboard Command: chansam

Pit/Channel Samples - Import from CRD File

This command creates channel samples from points in a coordinate file. Channel samples are strata data points that supplement drillholes. Channel samples represent one strata value such as thickness for strata COAL1. The coordinate file consists of point number, x, y, z data.

The command starts with a dialog to choose the symbol, symbol size and strata attribute name. Then you specify the coordinate file and the range of points to make into channel samples. Similar to drillhole symbols, the channel samples are drawn as the chosen symbol in the CHANNEL layer with the data attached to the symbol. The channel samples can be used in modeling routines such as Strata Isopach Maps or Make Strata Grid Files, by selecting the channel sample symbols along with the drillholes.



Prompts

Import Channel Sample Points dialog

Point numbers to convert: *1-100 or all or **

Point numbers to convert: *press Enter*

Pulldown Menu Location: Drillhole

Keyboard Command: chansam1

Pit/Channel Samples - Import from Coal Section

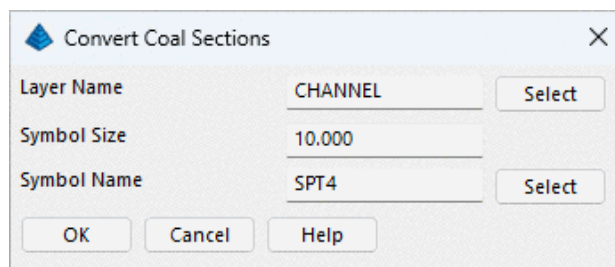
This command converts the strata data in coal sections into thickness channel samples. Coal sections are created in the Works pull-down menu of the Standard Mining Module and consist of strata names with thicknesses.

The program prompts for the strata names to use in the channel samples in case the strata names differ from the coal sections names. For example, consider this coal section:

C-7
R-3
C-54

In order to avoid duplicate strata names in the channel samples, the two coal section strata named *C* could be named *Coal1* and *Coal2* for the channel samples.

The program has settings for the layer, symbol block drawing name and symbol size for the channel sample symbols to create.



Prompts

Coal Section Configuration File Dialog

Choose a coal section configuration file (*.sc).

Select coal sections.

Select objects: *pick coal section symbols*

Create channel samples for section COAL [**<Yes>/No**]? *press Enter*

Enter strata name for section coal **<COAL>**: *press Enter*

Enter channel sample description **<COAL SECTION>**: *press Enter*

Pulldown Menu Location: Drillhole > Pit/Channel Samples

Keyboard Command: chconvert2

Export Pit/Channel Samples

This command creates a text file of sample data from the selected sample symbols in the drawing. The export file is in the format of Northing, Easting, Description, Strata Name, Attributes. This command is a way to backup the sample data. The export text file can be used to recreate sample symbols using the Import Samples routine.

Prompts

Export Text File to Write Specify a file name

Select samples to export.

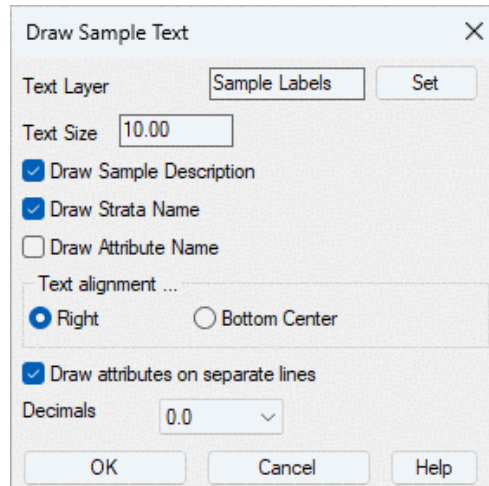
Select objects: *pick the sample symbols to export*

Pulldown Menu Location: Drillhole

Keyboard Command: chansam4

Draw Pit/Channel Sample Text

This command labels the sample data next to the sample symbol. The format and label options are set in the dialog. After selecting the samples to label, the program will read the attribute names from the symbols and prompt at the command line for each one whether to label the attribute.





```
Sample1  
SEAM2  
STRATA THICKNESS: 3.2  
STRATA BOTTOM ELEV: 4567.9  
CALCIUM: 92.1
```

Prompts

Draw Sample Text dialog

Select samples to label

Select objects: *pick the sample symbols to label*

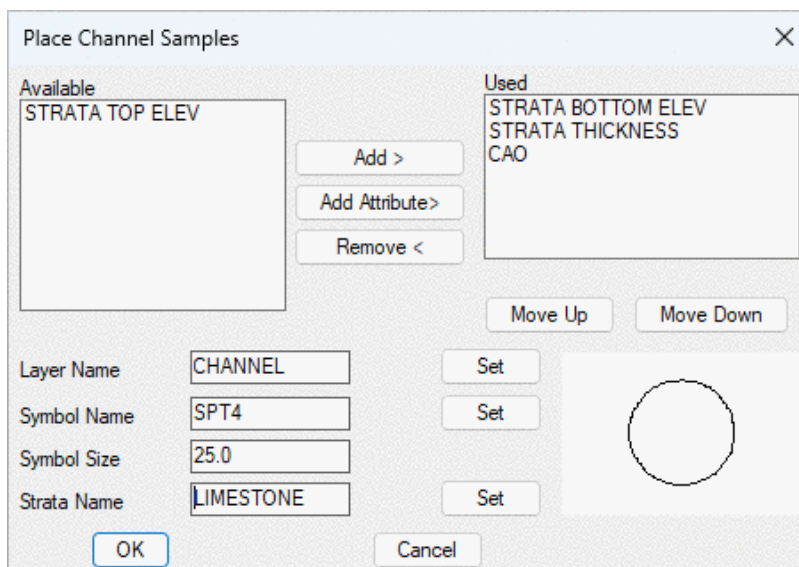
Label attribute BTU (<Yes>/No)? *press Enter for Yes*

Pulldown Menu Location: Drillhole

Keyboard Command: chantext

Place Pit/Channel Sample

This command creates channel samples at user-specified points and values. The command starts with a dialog to choose the symbol, symbol size, strata name and attribute names. The attribute names to assign are listed in order under the Used list. The Strata Thickness, Bottom Elevation and Top Elevation are the only pre-defined strata attributes. To add one of these attributes to the sample value list, highlight the attribute from the Available list and click Add. All other attributes are user-defined. To add a user-defined attribute, pick the Add Attribute button. Another dialog will appear for entering the attribute name. After filling out the dialog, pick OK. Then you pick the sample positions and enter the sample values at the prompting in the command line. Another option is to use the object snap and select something in the drawing with an elevation, such as a contour. The sample will then use that elevation, thickness or Z value for the sample value.



Prompts

Description <>:

Enter STRATA THICKNESS value: 3.2

Enter STRATA BOTTOM ELEV value: 4598.2

Enter CALCIUM value: 92.4

Pick channel sample location: *press Enter*

Pulldown Menu Location: Drillhole

Keyboard Command: chansam2

Edit Pit/Channel Sample

This command edits the strata name, attribute name or attribute value of a channel sample. To edit a channel sample, Either pick on the symbol after executing the command or simply "double-click" on any sample point. Any changes are saved back to the data attached to the channel sample symbol. Each sample must include a strata name, but all other attributes (including description, elevations, and thickness) are optional.

	Name	Value
1	CACO3	93.2
2		
3		
4		
5		
6		
7		
8		
9		

Prompts

Pick channel sample to edit: *pick a channel sample symbol*

Edit Channel Sample Points dialog

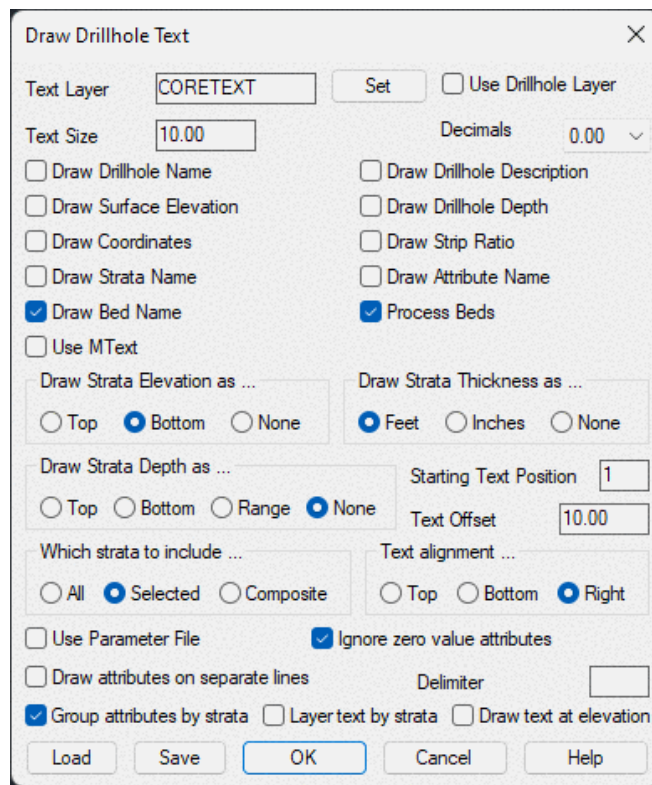
Pick channel sample to edit: *press Enter*

Pulldown Menu Location: Drillhole

Keyboard Command: chansam3

Standard Drillhole Text

This command draws text for the descriptions, surface elevation, strata names, bed names, thicknesses, attributes, and/or bottom elevations of the selected drillholes. Which values to draw is determined by the selections in the dialog shown below. When bed names are available they may be drawn next to the strata name.



Text Layer: This option controls the layer that will be used for the text labels. If the **Use Drillhole Layer** option is selected, text labels will be drawn on the same layer as the drillhole symbol. Note that strata layers may also be set according to the "Layer Text by Strata" option.

Text Size: This option controls the size of the text labels.

Decimals: This option controls the number of decimal places in the values. If the strata attributes are defined in Define Attributes, the number of decimal places to report will respect the attribute definition file.

Draw ____: These check boxes control which attributes of the drillhole will be labeled. These labels are self-explanatory, but a few notes on these options are provided below.

- **Coordinates:** Listed as X,Y
- **Strip Ratio:** calculated for the entire drillhole. The units for this ratio can be defined in Carlson Configure (Mining Settings). The density values used for determining stripping ratio will be pulled from the Strata Definition File (if these density values are not found, standard densities of 80 lbs/ft³ will be used for key material, and 150 lbs/ft³ for nonkey material).
- **Process Beds:** When this option is used, Bed naming conventions will be used in place of the strata names in the drillholes. More information on Bed naming conventions can be found in Carlson Configure (Mining Settings).

Use MText: This option will draw the label using an MTEXT entity instead of a standard TEXT entity.

Draw Strata Elevation/Thickness/Depth: These options control how the elevation/thickness/depth labels will be drawn.

Starting Text Position: This option sets the row to start the text labels. By increasing this value, you can add additional labels without overlapping previous labels. For example, if the drillholes already have two rows of labels, then use a starting text position of three.

Text Offset: This option sets the distance from the center of the drillhole symbol to draw the first text label. It is recommended that this value be set to half the diameter of the drillhole radius to prevent the labels from overlapping the symbol.

Which Strata to Include: This option determines which strata will be labeled. If the "Selected" option is chosen, you will be prompted to select the strata to label. If the "Composite" option is chosen, you will be prompted to select the strata to label, but all selected strata will be grouped as if they were a single strata.

Text Alignment: This option determines the placement of the labels relative to the drillhole symbol (right justified, center justified above the drillhole, center justified below the drillhole).

Use Parameter File: This option allows you to select a parameter file to filter the drillholes and draw text for only the drillholes that pass the filter. See Define Parameters for more on parameters.

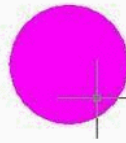
Ignore Zero Value Attributes: This option will prevent attributes with a value of zero from being labeled.

Draw Attributes on Separate Lines: This option will create a new line for each attribute of each strata, otherwise the attributes of a strata are grouped together on one line, but separated by the **Delimiter** field.

Group attributes by strata: This option will draw all the attributes for a strata together. Otherwise the attributes are grouped by attribute name such as having all sulfur attribute values drawn together.

Layer Text by Strata: When this option is enabled, the layers used for each label will be set according to the layers defined in the Strata Definition File.

Draw Text At Elevation: **This option** creates the labels at the surface elevation of the drillhole. Otherwise, the text is drawn at zero elevation.



2001-17
N: 1971581.7
E: 1458094.3
SURFACE ELEV: 4880.0
DEPTH: 171.0
STRIP: 1583.3
Coal/LS BOTTOM ELV:4742.0
THICKNESS:138.0
ASH:5.2
BTU:9350
MOISTURE:31.5
SUL:3.0
Sandstone/LS2 BOTTOM ELV:4724.0
THICKNESS:18.0
Coal/LS3 BOTTOM ELV:4709.0
THICKNESS:15.0
ASH:8.3
BTU:8175
MOISTURE:31.4
SUL:1.0

Prompts

Draw Drillhole Text Dialog

Select the DrillHoles to label.

Select objects: *pick the drillhole symbols*

Choose Key Attributes to Draw Optionally choose attributes to label.

Choose Non-Key Attributes to Draw

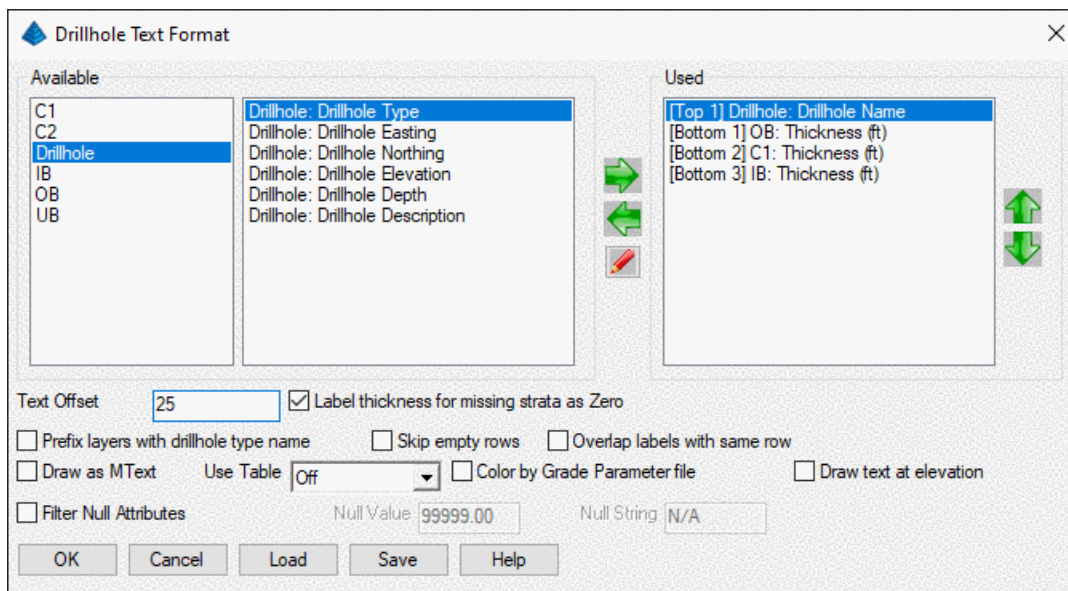
Pulldown Menu Location: Drillhole > Draw Drillhole Text

Keyboard Command: chtext

Drillhole Text Formatter

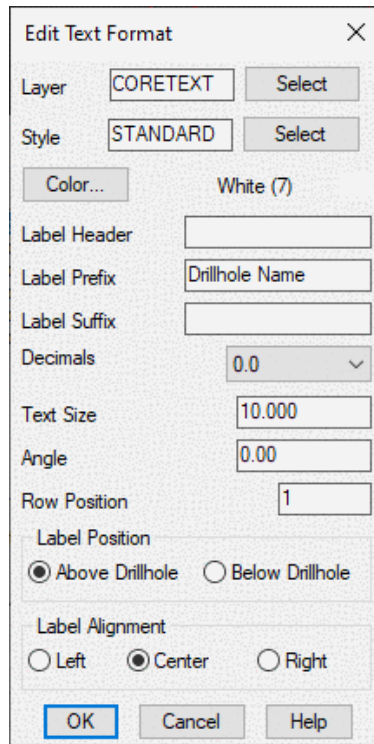
This routine allows for flexible labeling of any data contained in the drillholes- also known as "posting drillholes." After selecting which drillholes to label, the program prompts whether to process beds. If beds are processed, the strata will be composited by bed name into groups such as bed_top, bed_key, bed_parting, and bed_bottom for labeling. If beds are not processed, only the strata names will be used for labeling.

The Drillhole Text Format Options dialog, shown below, allows you to select the information to label. The full list of available information is shown on the left of the dialog, while the "Used" column on the right lists the information that will be labeled. The leftmost column lists the Drillhole, the Strata/Beds, and the Equations (if applicable). Selecting one of these groups will show the specific attributes that can be labeled in the middle column. These attributes can be added to the "Used Column" by double-clicking the attribute, or by highlighting the attribute and clicking the green right-arrow between the "Available" and "Used" columns.



- **Text Offset:** Sets the distance from the center of the drillhole to the text. To avoid overlapping the text with the drillhole symbol, it is recommended to set this value greater than or equal to the radius of the drillhole symbol.
- **Label thickness for missing strata as Zero:** This option will label missing strata with a thickness of zero. This option will also affect Equations in that null values will be treated as zeros. For example, if an Equation adds the thickness of Strata A and B, but a drillhole does not contain Strata B, the Equation will simply report the thickness of Strata A. When this option is disabled, Equations will not be evaluated when a strata is missing.
- **Prefix layers with drillhole type name:** This option prefixes the label layers with the drillhole type name. For example, if the Drillhole is tagged as Rotary hole and the labels are set to draw on layer "Label", this option would draw the labels on the layer "Rotary-Label".
- **Skip empty rows:** When no labels are drawn on a row, this option will slide the other labels up or down to fill in the missing row.
- **Overlap labels with same row:** This option will allow text labels to overlap if they are set to draw on the same row.
- **Draw as MTEXT:** Selecting this will group the text as MText (multiline text), otherwise it is separate standard text for each text item.
- **Use Table:** This option will use a table entity for labels on similar rows, or for labels representing repeating strata. For example, if 5 intervals of the same strata exist in the drillhole, the table entity will have 5 rows of data. Note that table entities may be drawn with or without grid lines.
- **Color By Grade Parameter File:** This option prompts for a Grade Parameter File to set the color of the labels based on the grade of each strata.
- **Draw text at elevation:** This will draw the labels at the same elevation as the drillhole collar.
- **Filter Null Attributes:** When enabled, this option will not draw labels for attributes with specific values/strings used as "null" markers.

When an attribute is added to the "Used" column, the below dialog will appear to control the settings of that attribute label. This dialog can also be accessed by double-clicking an attribute in the "Used" column or by highlighting an attribute and clicking the pen icon between the "Available" and "Used" Columns.



- **Layer:** Sets the layer for each label
- **Style:** Sets the text style
- **Color:** Sets the text color
- **Label Header:** This header text will be inserted on the row above the label. When drawing multiple labels for a strata type, this head can be used to group the labels together, as shown in the below image. In this example, the first Thickness label has been assigned a label header called "OB" while the second thickness label has been assigned a label header called "C1". It is important to note that the Elevations labels should not be assigned label headers.



- **Label Prefix:** Sets a prefix to the text. In the above image, the prefixes of "Bottom Elev: " and "Thick (ft)" have been assigned
- **Label Suffix:** Add a suffix to the text
- **Text Size:** Sets the size of text
- **Decimals:** Sets the number of decimals to display in the value
- **Angle:** Sets the rotation angle of the text
- **Row Position:** Sets the row position for the text. Row 1 will be closest to the drillhole symbol, Row 2 will be further away from the symbol, etc. The rows are counted either up or down from the symbol based on the **Label Position**.
- **Label Position:** Determines if the text is drawn above or below the drillhole symbol
- **Label Alignment:** This option aligns the text with the center of the drillhole symbol, or off to the side of the drillhole.

Select the Drillholes to label.

Select objects: *pick the drillholes*

Process beds [**<Yes>/No**]? *Press Enter for yes.*

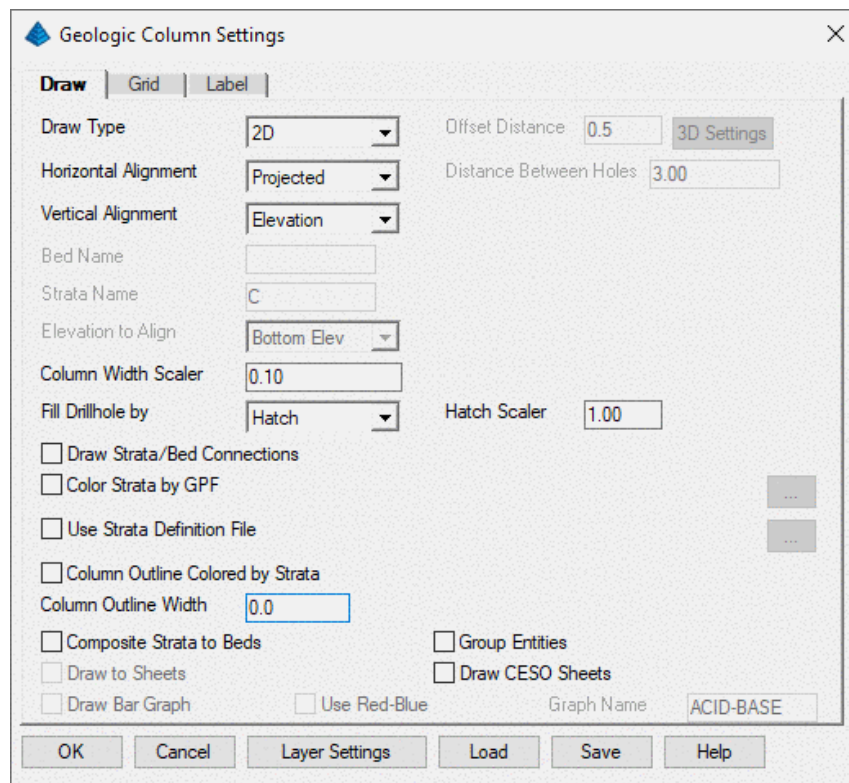
Drillhole Text Format Options dialog

Drop-Down Menu Location: Drillhole

Keyboard Command: chtext2

Draw Geologic Column

As the name suggests, this command draws geologic columns for selected drillholes. All of these settings may be saved to a geologic column settings file (.gcl). Notice that the dialog is divided into three tabs: Draw Settings, Grid Settings, and Label Settings.



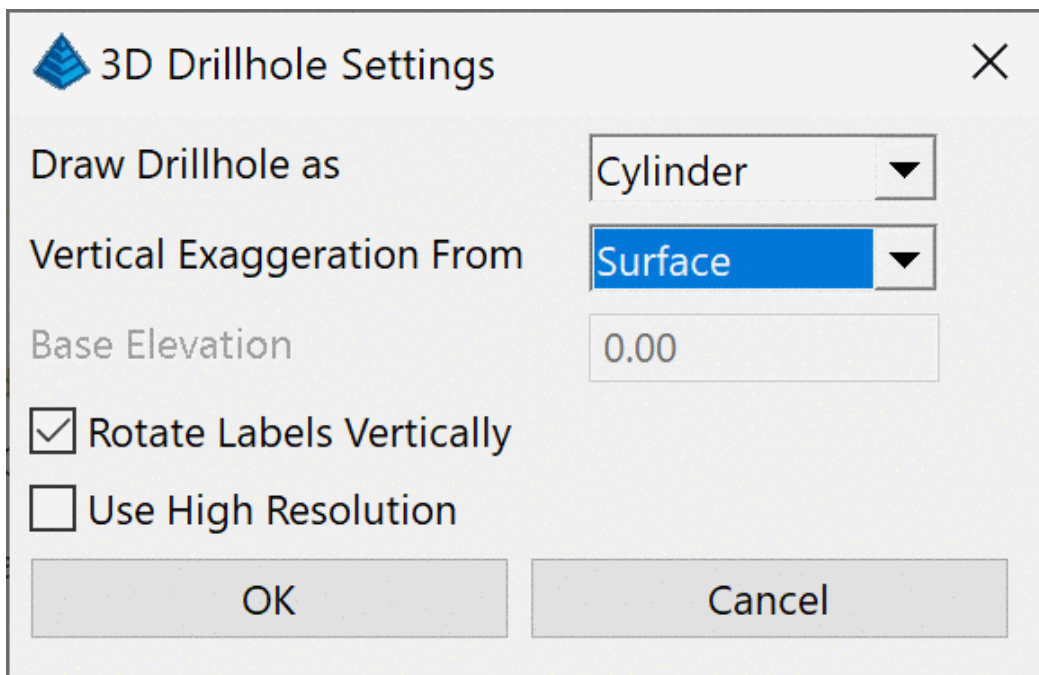
Draw Type: The geologic columns can be drawn in *3D*, *2D* or *Next to Drillhole*.

- The *3D* option will draw the columns in true 3D space.
- The *2D* option will draw a cross-sectional view of the columns in a user defined location.
- The *Next to Drillhole* option will draw a cross-sectional view of the columns next to the true drillhole locations.

Offset Distance This value is only applied when the Draw Type is set to the *Next to Drillhole* option. This value sets the distance between the drillhole symbol and the column. It is important to note that this distance is multiplied by the Horizontal Scale value on the Grid tab to set the actual offset distance.

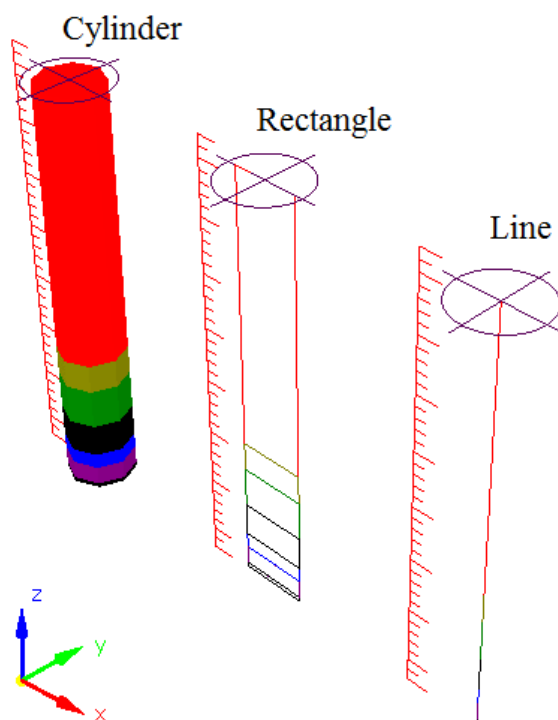
Distance Between Holes This value is only applied when the Draw Type is set to the *2D* option and the Horizontal Alignment is set to the *Line Up* option. For more details please see the description of the *Line Up* option below.

3D Settings: This button is only active when the Draw Type is set to the *3D* option. These settings control how the drillhole will be drawn in 3D space. When selected, the below dialog will appear.



Draw Drillhole as: This option controls how the column will be drawn. An example of each option is shown below as viewed in the 3D Viewer Window

- The *Cylinder* option will draw the column as 3D faces.
- The *Rectangle* option will draw the column as flat 3D polylines (this essentially draws a 2D column rotated into 3D space)
- The *Line* option will draw a single 3D polyline to represent each column.



Vertical Exaggeration From: This option controls how 3D columns will be drawn when vertical exaggeration is applied.

When the *Base Elevation* options is used, the column will be scaled, but the **Base Elevation** value (entered immediately below this option) will be held at the true elevation, even if the column does not intersect this elevation. When the *Surface* option is used, the column will be scaled, but the top of the hole will be held at the true elevation. For example, consider a drillhole with a collar elevation of 100 and a bottom elevation of 60 to be drawn with a 2X vertical exaggeration. If the *Surface* option is used, the top of the column will be drawn at elevation 100 and the bottom of the column will be drawn at elevation 20. If the *Base Elevation* option is used with a Base Elevation of 0, the top of the hole will be drawn at elevation 200 and the bottom of the hole will be drawn at elevation 120.

Rotate Labels Vertical: When selected, this option will draw all 3D text labels oriented in the vertical direction. When this option is not selected, 3D text labels will be oriented in plan view.

Use High Resolution: When selected, this option will draw the 3D drillhole in High Resolution.

Horizontal Alignment: This option is only available when the Draw Type is set to the *2D* option. Three methods are available for Horizontal Alignment.

- The *Individual* option will prompt you to select the location to draw each individual column.
 - The *Line Up* option will space the columns equal distances apart from one another, regardless of the true distance between the holes. The space between the columns is the product of the **Distance Between Holes** multiplied by the Horizontal Scale on the Grid tab.
 - The *Projected* option will space the columns by projecting them onto a cross-sectional alignment polyline.
- Both the *Line Up* and *Projected* options require that an alignment polyline be drawn through or near the columns of interest prior to the command. After clicking OK on the main dialog, you will be prompted to select this polyline, as well as the "maximum drillhole distance from alignment polyline" to filter out distant holes. Any drillholes outside of this range will not be drawn, even if they are selected.

Vertical Alignment: When Horizontal Alignment option is set to the *Line Up* or *Projected* options, the geologic columns may be aligned vertically in one of three ways.

- The *Elevation* option will draw the columns according to their true elevations.
- The *Surface* option will draw the columns so that all collar elevations are aligned. This allows a horizontal line to be drawn across the top of all the columns. With this option, an elevation axis will be drawn through each column for individual inspection.
- The *Strata* option will draw the columns so that a specific strata/bed is aligned across the columns. This would allow a horizontal line to be drawn through the top/bottom elevation of the strata/bed in all holes. With this option, an elevation axis will be drawn through each column for individual inspection. This option also requires a text string for either the **Strata Name** or **Bed Name** fields. The **Elevation to Align** dropdown list controls if the strata/bed will be aligned according to the top or the bottom elevation.

Column Width Scaler: This value, relative to the Horizontal Scale, determines the width of the geologic column. For example, a Column Width Scaler of 0.15 and a Horizontal Scale of 100 will produce a column with a width of 15 drawing units.

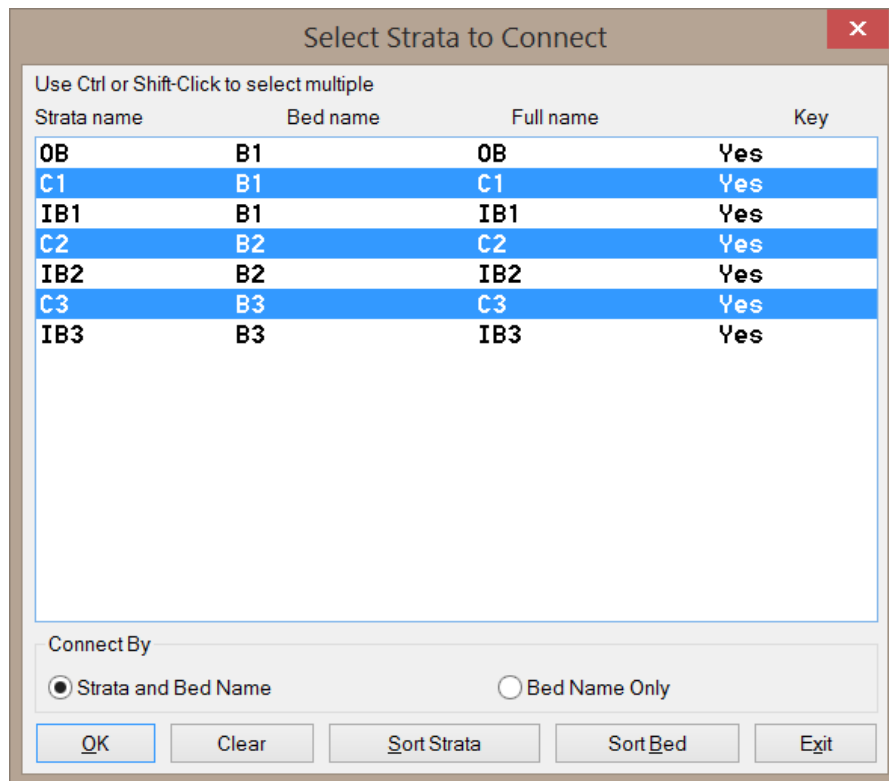
Fill Drillhole by: This option determines the hatch pattern for each strata.

- If the *Outline Only* option is selection, no hatch pattern will be drawn.
- If the *Hatch* option is selected, the strata will be filled with the hatch specified in the Strata Definition File. If a strata is not found in the Strata Definition File, the strata will be hatched with the ANSI31 pattern.
- If the *Solid Fill* option is selected, all strata will be hatch with the SOLID hatch pattern.

Hatch Scaler: This value is only used when the *Fill Drillhole by Hatch* option is selected. This value, relative to the hatch scales in the current Strata Definition File, determines the hatch scale of each strata's hatch pattern. For example, a Hatch Scaler of 5 and a strata with a defined hatch scale of 2 will produce a hatch pattern with a scale of

10. If a strata is not found in the Strata Definition File, the Hatch Scaler will be multiplied by a value of 5 to set the hatch scale of the strata.

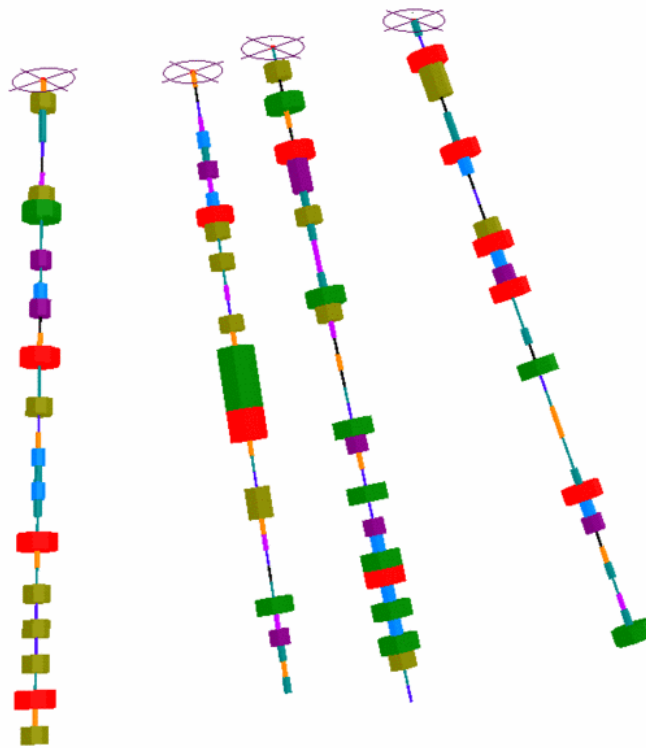
Draw Strata Connections: This option is not available when the *Align Horiz by Individual* option is selected. This option will draw polylines connecting similar strata between columns. Note that if a strata or bed pinches out between columns, this pinchout will not be drawn. After clicking OK on the dialog, you will be prompted for the strata to connect, as shown below.



Connect By: This option will determine if connections are drawn according to Strata Names or Bed Names. If the *Strata and Bed Name* option is selected, connections will be drawn for individual strata layers. If the *Bed Names Only* option is selected, connections will only be drawn for the top and bottom of each bed - individual strata connections will not be drawn.

Sort Strata: This button will sort the list alphabetically by Strata Name.

Sort Bed: This button will sort the list alphabetically by Bed Name. **Color Strata by GPF:** This option will color each strata according to a Grade Parameters File (GPF). You may select a GPF by clicking the ellipsis button to the right of the option. When the drillhole is drawn in 3D, the strata width will be scaled according to the *Geologic Column Scaler* defined for each grade in the Grade Parameters File. This allows for more visual interpretation of high-grade and low-grade sections of the column, as shown below.



Use Strata Definitions: This option allows the use of a Strata Definition File (which controls layering, colors, hatching, etc.) other than the one currently set in the Mining Project Settings. The ellipsis button will prompt you for this Strata Definition File.

Column Outline Colored by Strata: When this option is enabled, the outline of the drillhole will be colored the same as the hatch color. When this option is disabled, the drillhole will be outlined with a white/black line.

Column Outline Width: This value sets the thickness for the column outline.

Composite Strata to Beds: This option will composite strata together by Bed Name where applicable. In order to use this option, both the Strata Name and the Bed Name must be defined in the Strata Definition File.

Group Entities: This option will draw each column as a group of entities (not to be confused with a block entity).

Draw To Sheets: This option is only available if the Horizontal Alignment option is set to *Individual*. This will draw the individual drillholes on multiple sheets, breaking the page as necessary.

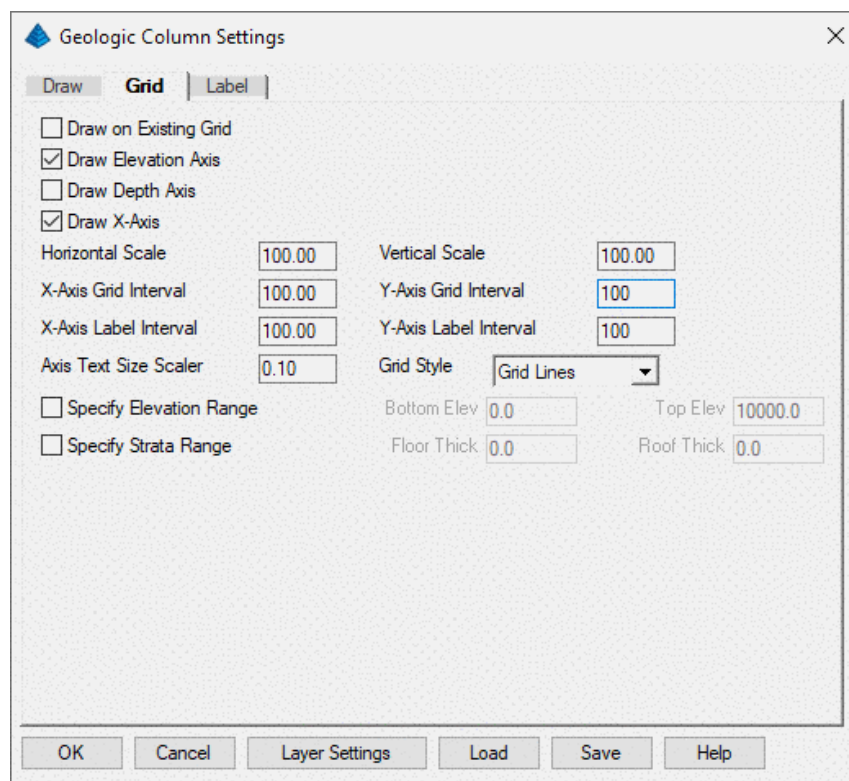
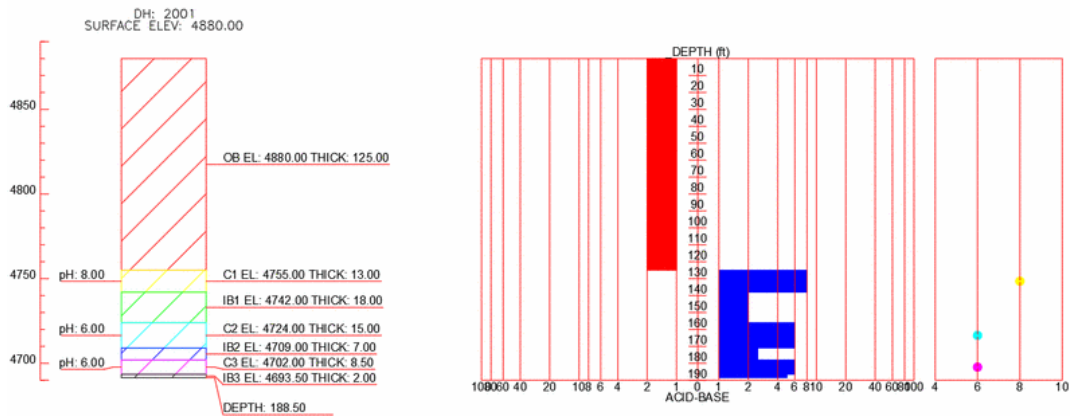
Draw CESO Sheets: This option is similar to the Draw to Sheets option, but is formatted specifically for a particular Carlson user. Unlike the Draw to Sheets option, this option will add header information to the sheets.

Draw Bar Graph: This option is only available when the Draw Type is set to the 2D option. An example Bar Graph as shown below. The graph is intended for acid-base accounting, but any drillhole attribute may be used. The graph is logarithmic, ranging in value from -100 to 100. Any values outside of this range will be drawn at the maximum value and a label will be drawn to indicate the true value. In addition to the Bar Graph, a Scatter Graph may also be drawn. This graph ranges in value from 4 to 10. Any values outside of this range will be drawn off the graph

Use Red-Blue: This option will draw the Bar Graph with only red or blue hatches when selected. Red hatches are used for negative values and blue hatches are used for positive values.

Graph Name: This text string is the title of Bar Graph, which will be drawn below the X-axis values. An example

of a geologic column with a bar graph is shown below.



Draw on Existing Grid: This option is only available when the Horizontal Alignment option is set to *Projected*. This option will draw the drillholes on a pre-existing cross section grid. When this option is selected, you will be prompted to select the bottom left corner of the section and enter the bottom elevation. This ensures the columns are drawn at the correct elevations on the section.

Draw Elevation Axis: This option will draw an elevation axis (vertical line) next to the columns.

Draw Depth Axis: This option will draw a depth axis (vertical line) next to the columns. When this axis is drawn next to a 3D drillhole, the axis will follow the dip of the hole rather than always being drawn as a vertical line.

Draw X-Axis: This option will draw a stationing axis (horizontal line) between drillholes when the Horizontal Alignment option is set to the *Line Up* or *Projected* methods.

Horizontal Scale: This value sets the overall drawing scale that many other options in the dialog reference for scaling. This allows you to quickly change the size the geologic columns with a single value.

Vertical Scale: This value, relative to the Horizontal Scale, determines the vertical exaggeration. For example, a Horizontal Scale of 50 and a Vertical Scale of 10 will produce a 5X vertical exaggeration.

X/Y-Axis Grid Interval: This value sets the grid line spacing for the X and Y axes. For example, a Y-Axis Grid Interval of 25 will draw a tick/grid line every 25 ft/m of elevation change.

X/Y-Axis Label Interval: This value sets the text intervals for the X and Y axes. For example, a Y-Axis Label Interval of 100 will draw an elevation/depth label every 100 ft/m of elevation change.

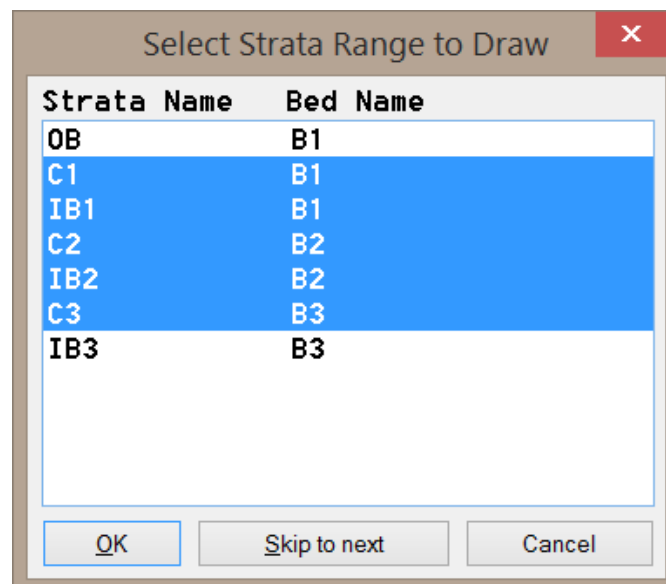
Axis Text Size Scaler: This value, relative to the Horizontal Scale, determines the text height of the elevation/depth/stationing labels. For example, an Axis Text Size Scaler of 0.15 and a Horizontal Scale of 100 will produce text labels with a height of 15 drawing units.

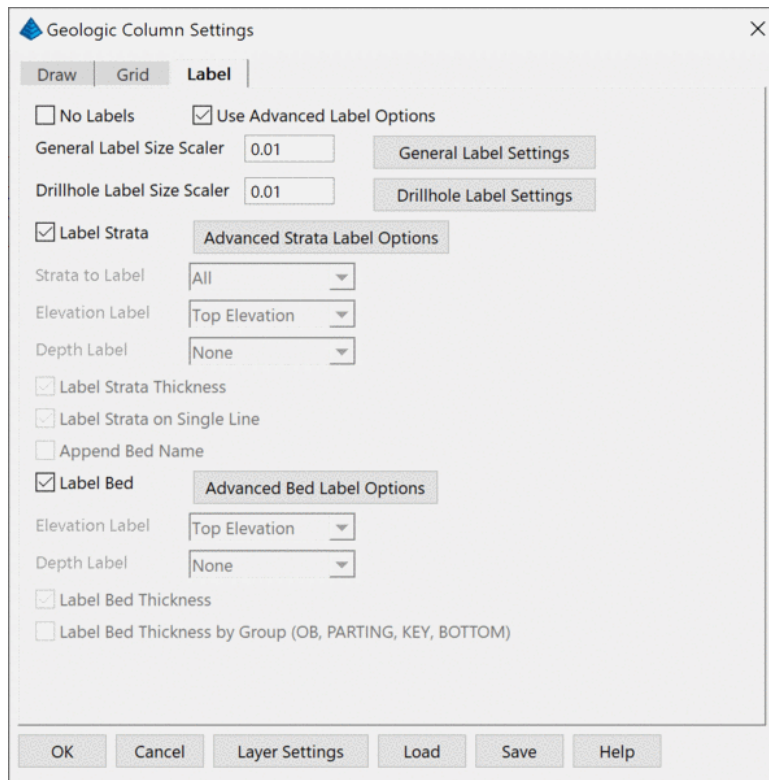
Grid Style: This option controls how the grid will be drawn.

- The *Grid Lines* option will draw a true grid with lines spanning the appropriate axes.
- The *Ticks Only* option will draw the grid axes, but instead of full lines spanning the axes, small tick marks will be placed along each axis.
- The *Ticks and Dots* option will draw tick marks along each axis, as well as points where true grid lines would intersect.

Specify Elevation Range: This option will draw the columns only between a range of elevations, set by the **Top Elev** and **Bottom Elev** values.

Specify Strata Range: This option will draw only a specific range of strata in the columns. After clicking OK on the dialog, you will be prompted for the strata to draw, as shown below. If not all strata are shown in this dialog, clicking *Skip to Next* will scan another hole for strata layers. Portions of strata above/below the selected strata can also be drawn according to the **Roof Thick** and **Floor Thick** values, respectively.





No Labels: This option will turn off labeling for the column and will also ghost out all labeling options in the dialog.

Use Advanced Label Options: This option will turn off all standard labeling options and will activate Advanced Strata and Bed Options.

General Label Size Scaler: This value, relative to the Horizontal Scale, set the text size labels for strata/bed/attribute labels.

General Label Settings: This button will open the below dialog, which controls many of the details of the column labels for strata/bed/attributes.

Draw Labels as: This option controls the type of text that is used for labels.

- The *DText* option will use a standard text entity for labels.
- The *MText* option will use MText (multi-line text) entities for labels.
- The *MLeader* option will use Multileader entities for labels, which are simply MText entities attached to a leader line.

Label Style: This text string sets the text style for the labels. The **Set** button may be used to select a pre-existing text style. Text styles may be edited via the `STYLE` command.

Use Full Strata Name: This option will use the Full Strata Names found in the Strata Definition File for strata labels.

Attributes on Single Line: This option will place all attributes for a single strata on one row rather than placing each attribute on an individual row. Not available if Advanced Label options are used.

Draw Label Leaders: This option will draw red leader lines from the column to the strata label and attribute labels. Labels will sometimes be shifted away from the column to avoid overlaps. Without these leader lines, the labels may be difficult to match up with the strata they relate to. Not available if Advanced Label options are used.

Draw Ferm Codes: This option will label each strata with its Ferm Code in addition to the true strata name (according to the Ferm Code definition file). If the **Ferm Label** option is set to *Code*, then the actual Ferm Code will be used. If the **Ferm Label** option is set to *Full Description*, then the full Ferm Code description will be listed rather than the Ferm Code itself.

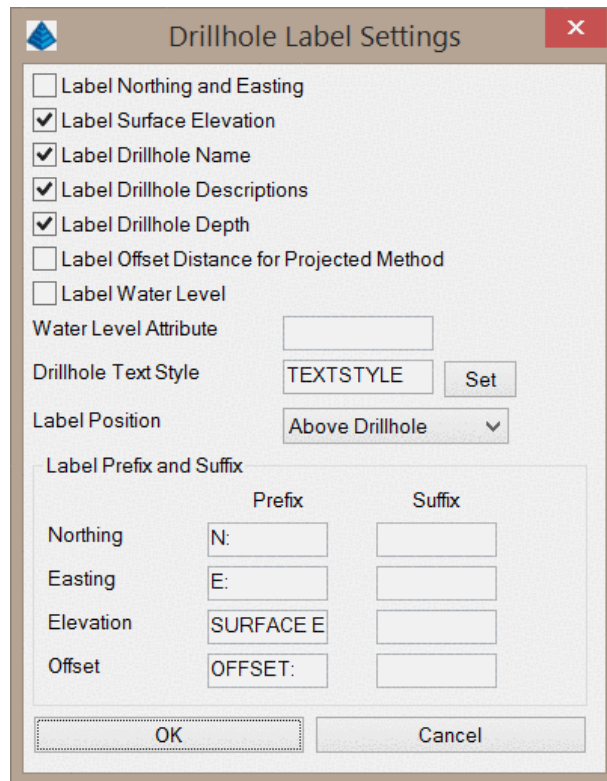
Label by Grade Parameters: This option will add the Grade Name to the label, according to a Grade Parameters File. If this option is set to *Left* the Grade Name will be listed with the attribute label. If the option is set to *Right*, the label will be listed with the strata label.

Elevation/Thickness/Depth Decimals: These options control the number of decimals to display for elevation, thickness, and depth labels, respectively. Not available if Advanced Label options are used.

Elevation/Thickness/Depth Prefix: These text strings set the prefix for elevation/thickness/depth labels, respectively. Not available if Advanced Label options are used.

General Label Size Scaler: This value, relative to the Horizontal Scale, set the text size labels for labels relating to the drillhole itself (such as the collar elevation, northing, easting, etc.)

Drillhole Label Settings: This button will open the below dialog, which controls many of the labels relating to the drillhole itself (rather than labels relating to strata within the drillhole).



Label Northing and Easting: This option will draw a label of the Northing and Easting coordinates for each column.

Label Surface Elevation: This option will draw a label of the Surface Elevation for each column.

Label Drillhole Name: This option will draw a label of the Drillhole Name for each column.

Label Drillhole Descriptions: This option will draw a label of the Drillhole Description for each column.

Label Drillhole Depth: This option will draw a label of the maximum depth for each column.

Label Offset For Projected Method: This option is only applied when the Horizontal Alignment is set to the *Projected* option. This option will draw a label indicating the perpendicular distance from the column to the alignment polyline.

Label Water Level: This option will draw a label of the water table elevation in the column. When this option is selected, the **Water Level Attribute** must match the name of the water level attribute defined in the drillhole. Note that the water level is an attribute of the drillhole itself; it should not be listed as an attribute of a particular strata.

Drillhole Text Style: This text string will set the text style for the drillhole label. The **Set** button may be used to select a pre-existing text style. Text styles may be edited via the STYLE command.

Label Position: This option determines where the drillhole label is drawn.

- If the *Above Drillhole* option is selected, the label will be center justified above the column.
- If the *On Side* option is selected, the label will be drawn to the right of the column.

Northing Prefix/Suffix: These two text strings will set the prefix and suffix for the label of the column Northing coordinate.

Easting Prefix/Suffix: These two text strings will set the prefix and suffix for the label of the column Easting coordinate.

Elevation Prefix/Suffix: These two text strings will set the prefix and suffix for the label of the column Surface Elevation.

Offset Prefix/Suffix: This option will only be applied when the Horizontal Alignment is set to the *Projected* option. These two text strings will set the prefix and suffix for the label of the column Offset from the fence alignment polyline.

Standard Strata/Bed Label Options

Label Strata: This option will draw labels for each strata found in the column. All strata label options will be ghosted when this option is not selected.

Strata to Label: This option will determine which strata are labeled.

- The *All* option will label all strata.
- The *Key-Only* option will only label key strata.
- The *Selected* option will prompt you to select the strata to label.
- The *Fit-Only* option will only draw a label if the strata is thick enough to place a label without having to adjust the label position to avoid overlaps.
- The *Bed-Only* option will only label strata with bed names.

Elevation Label: This option determines which elevation will be labeled for each strata.

- If the *None* option is selected, no elevation label will be drawn.
- If the *Top Elevation* option is selected, the top elevation of each strata will be labeled.
- If the *Bottom Elevation* option is selected, the bottom elevation of each strata will be labeled.

Depth Label: This option determines which depth will be labeled for each strata.

- If the *None* option is selected, no depth label will be drawn.
- If the *Top Depth* option is selected, the depth to the top of each strata will be labeled.
- If the *Bottom Depth* option is selected, the depth to the bottom of each strata will be labeled.

Label Strata Thickness: This option will draw a label for the thickness of each strata.

Label Strata On Single Line: This option will place all strata labels on a single row rather than placing each label on an individual row.

Append Bed Name: This option will label the Bed Name in addition to the Strata Name for each strata in the column.

Label Bed: This option will draw a label for the bed name of each strata. All bed label options will be ghosted when this option is not selected.

Bed Elevation Label: This option determines which elevation will be labeled for each bed.

- If the *None* option is selected, no elevation label will be drawn.
- If the *Top Elevation* option is selected, the top elevation of each bed will be labeled.
- If the *Bottom Elevation* option is selected, the bottom elevation of each bed will be labeled.

Bed Depth Label: This option determines which depth will be labeled for each bed.

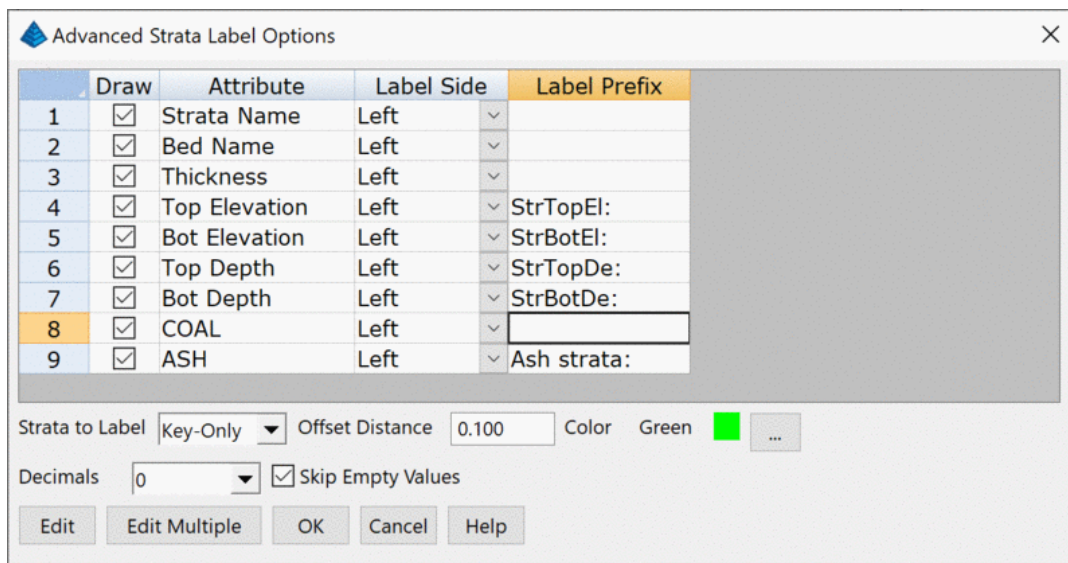
- If the *None* option is selected, no depth label will be drawn.
- If the *Top Depth* option is selected, the depth to the top of each bed will be labeled.
- If the *Bottom Depth* option is selected, the depth to the bottom of each bed will be labeled.

Label Bed Thickness: This option will draw a label for the thickness of each bed.

Label Bed Thickness by Group (OB, PARTING, KEY, BOTTOM): This option will draw a label for each of the four bed groups.

Advanced Strata/Bed Label Options

Advanced Strata Label Options are available when "Use Advanced Label Options" and "Label Strata" are checked.



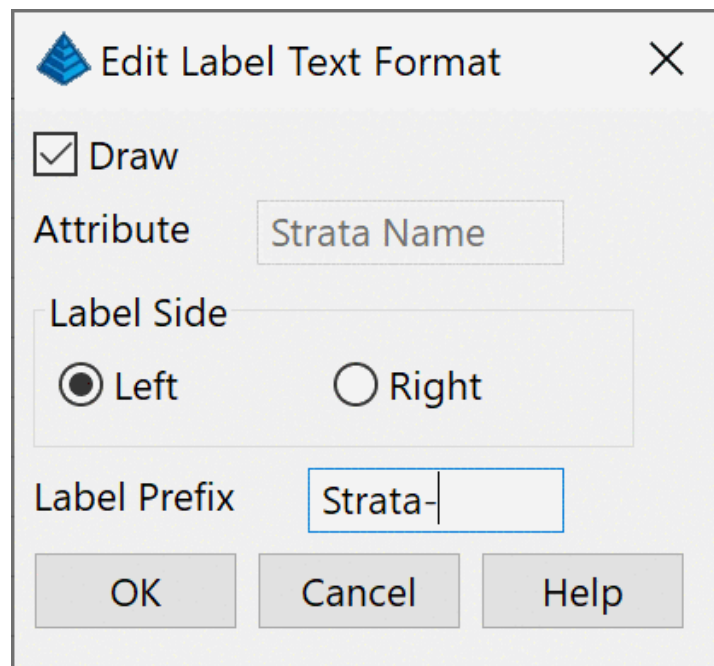
Draw determines which Strata Attribute will be labeled.

Attribute column shows all available Strata Attributes for labeling. Custom attributes such as ASH are read from the selected drillholes.

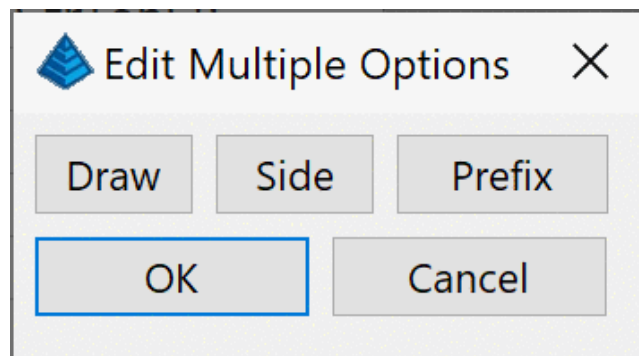
Label Side determines a label side relative to the geologic column for each Strata Attribute.

Label Prefix will set the prefix for the label.

Each Attribute Label can be edited in a separate window by picking the **Edit** button.



To edit label options for multiple Attributes such as the Draw, Side, Prefix highlight multiple rows in the spreadsheet by picking with the Ctrl and/or Shift keys, and then pick the **Edit Multiple** button.



Strata to Label: This option will determine which strata are labeled.

- The *All* option will label all strata.
- The *Key-Only* option will only label key strata.
- The *Selected* option will prompt you to select the strata to label.
- The *Fit-Only* option will only draw a label if the strata is thick enough to place a label without having to adjust the label position to avoid overlaps.
- The *Bed-Only* option will only label strata with bed names.

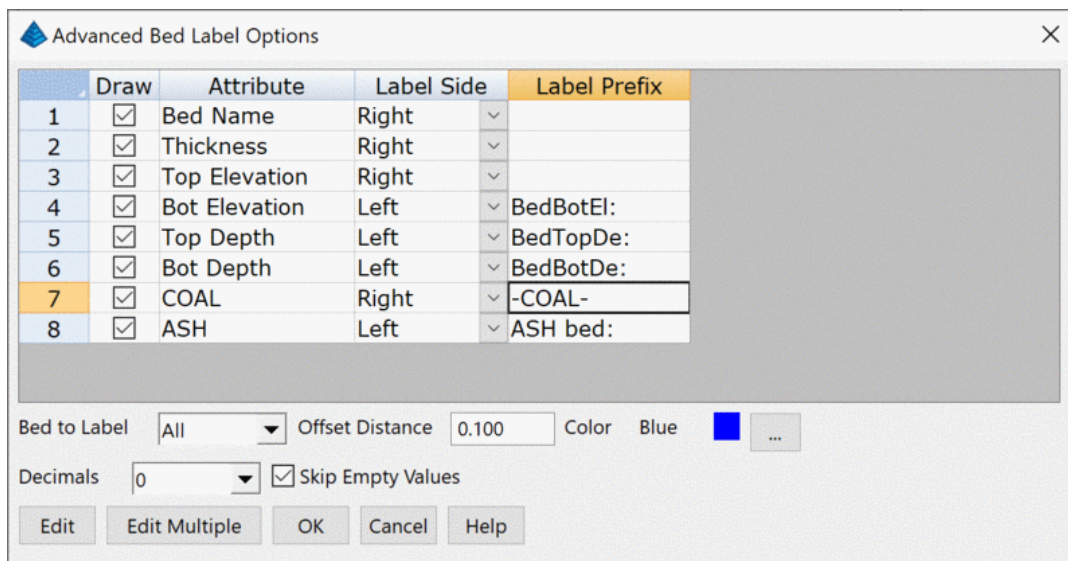
Offset Distance: This value, relative to the Horizontal Scale, set the distance between labels and the geologic column.

Color determines a color for all strata labels.

Decimals controls the number of decimals to display for all strata attributes.

Skip Empty Values will skip all attributes with no value.

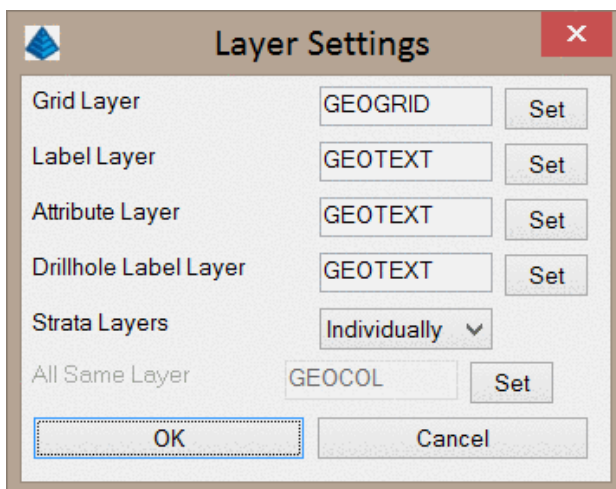
Advanced Bed Label Options are available when "Use Advanced Label Options" and "Label Bed" are checked.



All options described above for the Advanced Strata Label Options dialog are applicable to the Advanced Bed Label Options. The only difference is that **Bed to Label** has fewer options compared to **Strata to Label**:

- The *All* option will label all beds.
- The *Selected* option will prompt you to select the bed to label.
- The *Fit-Only* option will only draw a label if the bed is thick enough to place a label without having to adjust the label position to avoid overlaps.

Clicking the **Layer Settings** button will display the below dialog.



Grid Layer: This text string sets the CAD Layer of the grid linework. The **Set** button can be used to select a pre-existing CAD Layer.

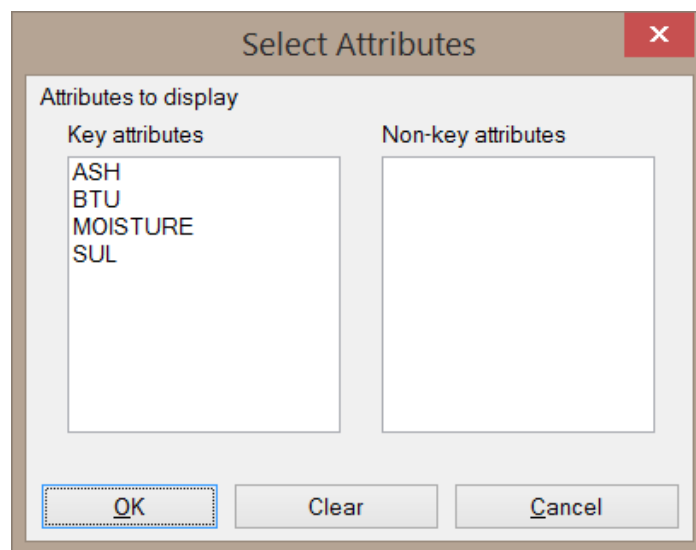
Label Layer: This text string sets the CAD Layer for most label entities, such as the elevation, depth, station, and strata. The **Set** button may be used to select a pre-existing CAD Layer.

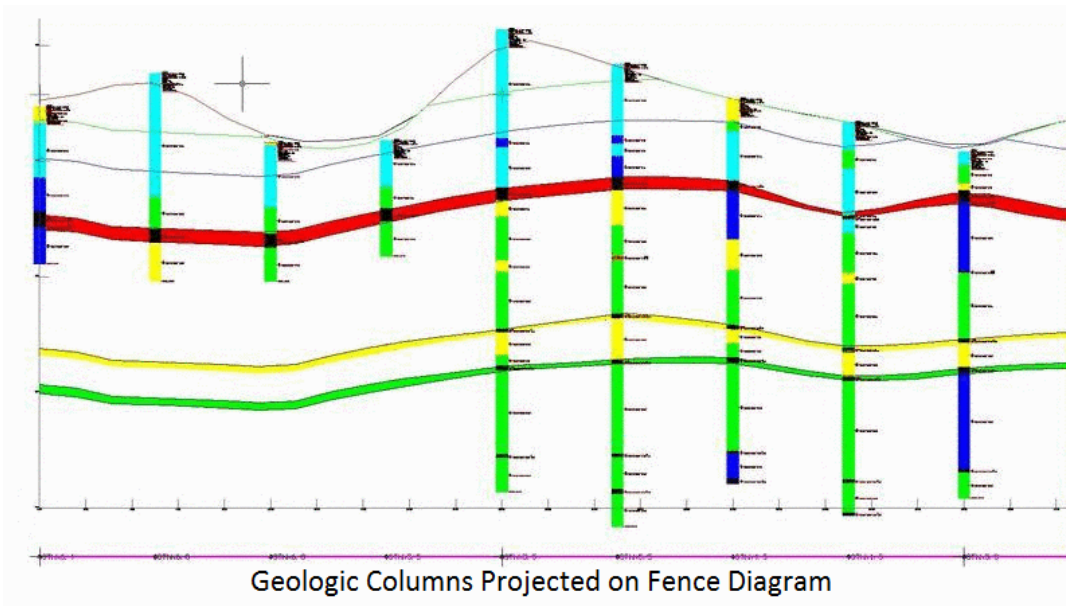
Attribute Layer: This text string sets the CAD Layer for attribute labels entities. The **Set** button may be used to select a pre-existing CAD Layer.

Drillhole Label Layer: This text string will set the CAD Layer for the drillhole label. The **Set** button may be used to select a pre-existing CAD Layer.

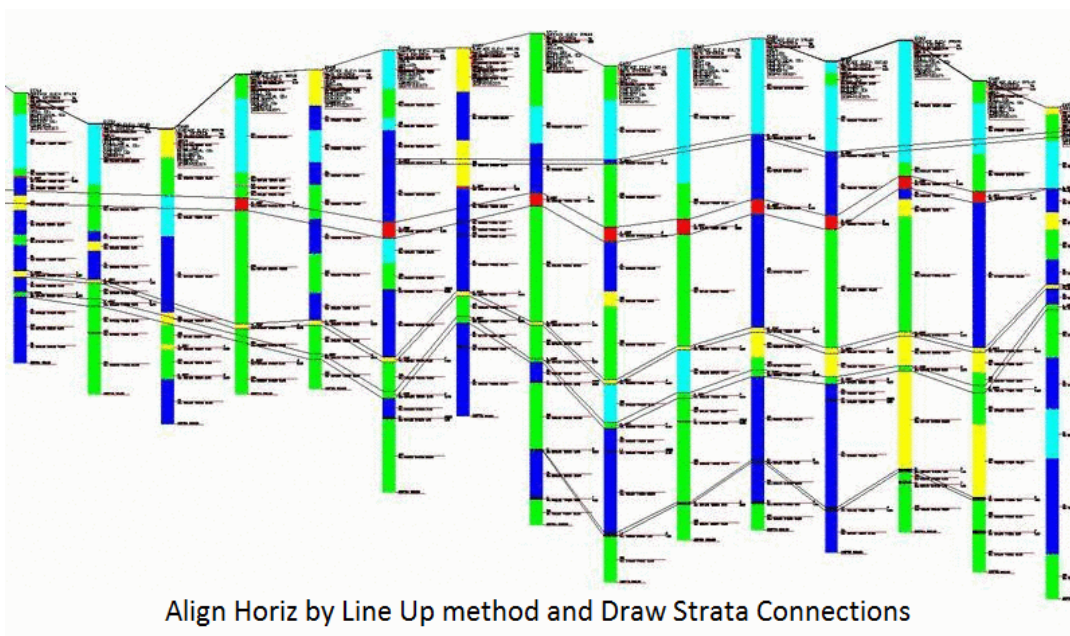
Strata Layers: This option will determine which CAD Layers are used for each strata in the column. If the *Individually* option is used, each strata will be drawn on the CAD Layer specified in the Strata Definition File. If the strata is not found in the Strata Definition File, each strata will be drawn on a CAD Layer matching the name of the strata. If the *Non-Key Same* option is used, all key strata will be drawn as if the *Individually* option were selected and all non-key strata will be drawn on the CAD Layer specified just to the right of this option. The ellipse button may be used to select a pre-existing CAD Layer. If the *All Same* option is selected, all strata will be drawn on the CAD Layer specified just to the right of this option. The ellipsis button may be used to select a pre-existing CAD Layer.

Select Attributes: If Standard Label Options are used, after clicking okay on the main dialog, you may be prompted for any additional settings in the command line. If you have chosen to draw strata/bed labels, you will see the below dialog, which allows you to choose which attributes to label. You may select multiple attributes by holding the CTRL or SHIFT keys while selecting attributes. To clear the selection of attributes, click the **Clear** button.

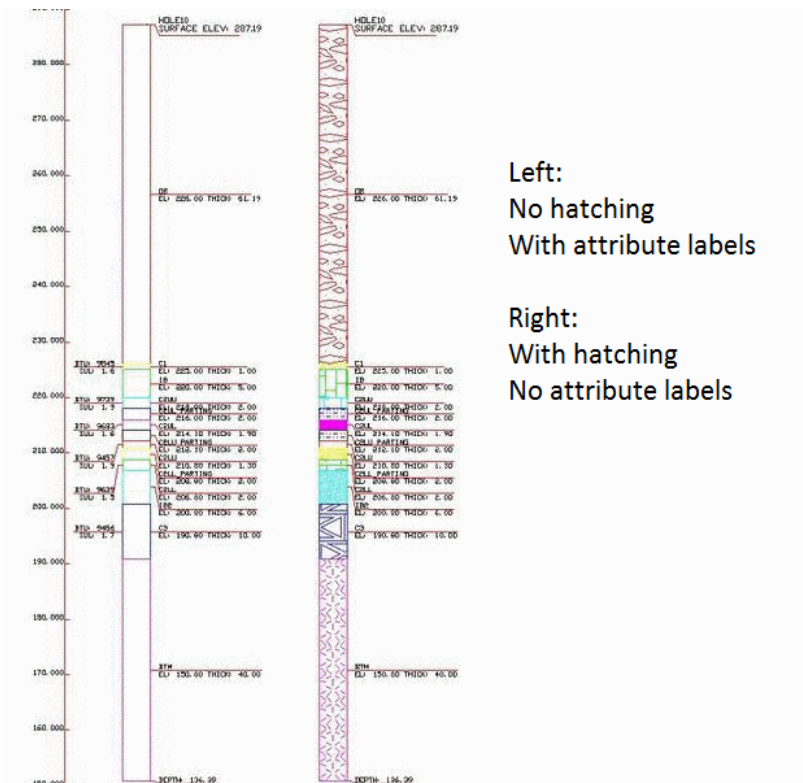




Geologic Columns Projected on Fence Diagram



Align Horiz by Line Up method and Draw Strata Connections



Left:
No hatching
With attribute labels

Right:
With hatching
No attribute labels

Prompts

Select drillholes for geologic column:

[FILter/?]/<Select entities>: select drillholes that will be used for the geologic column drawing

Geologic Column Settings dialog

Select Attributes to Draw dialog (if standard label options are used)

Pick location for geologic column: *pick or enter the bottom center point for the geologic column*

Pulldown Menu Location: Drillhole

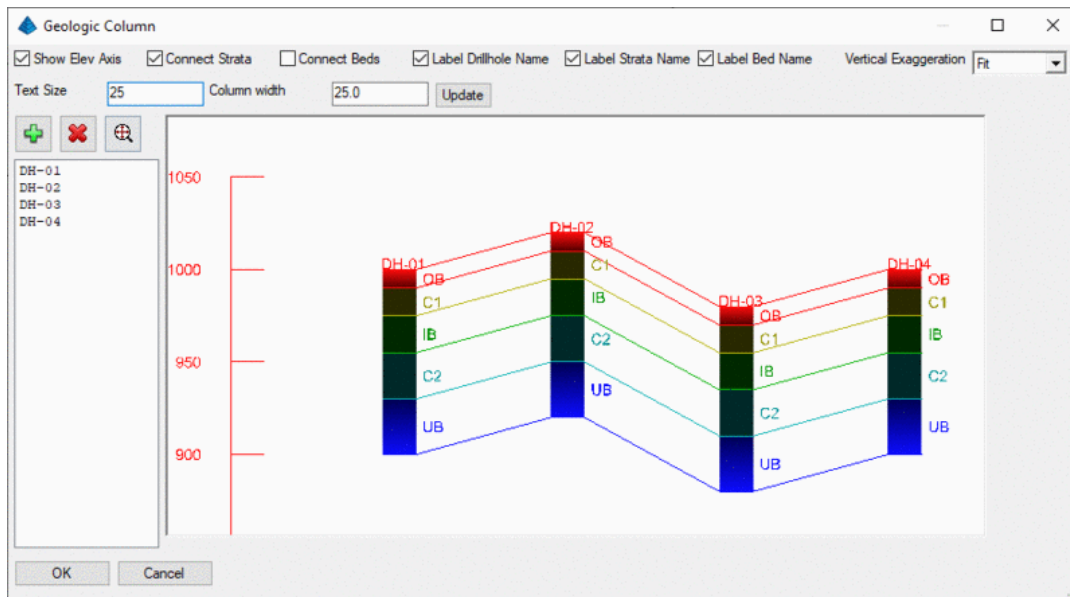
Keyboard Command: geocol

Quick Geologic Column

This command displays a section of drillholes as geological columns that are connected by a polyline fence line. This is an useful command to do quick views of correlations, depths and elevations of drillholes in a section, fence view. Before running this command, there needs to be a polyline drawn to the drillholes that are to be displayed. The seams colors are specified in the Define Strata command prior to running this command. There are options to Show Elevation Axis on the left, to Connect Strata and Beds, to Label Drillhole Name, and to Label Strata Names and Bed Names. The Vertical Exaggeration has an option to fit them to the window, or set the scale.

The green plus icon and the x icon may be used to add or remove drillholes from the view, respectively. The magnifying glass icon may be used to change the zoom of the window so that all drillholes are visible.

The **Text Size** and **Column Width** values will modify the appearance of the drillholes in the graphic view.



Prompts

Select Fence alignment polyline: (select the polyline drawn to the holes)

Maximum drillhole distance from alignment polyline <100.0>: (this is the distance it will search from the line to include the holes in the section)

Select Drillholes for geologic column. (select with a window or pick)

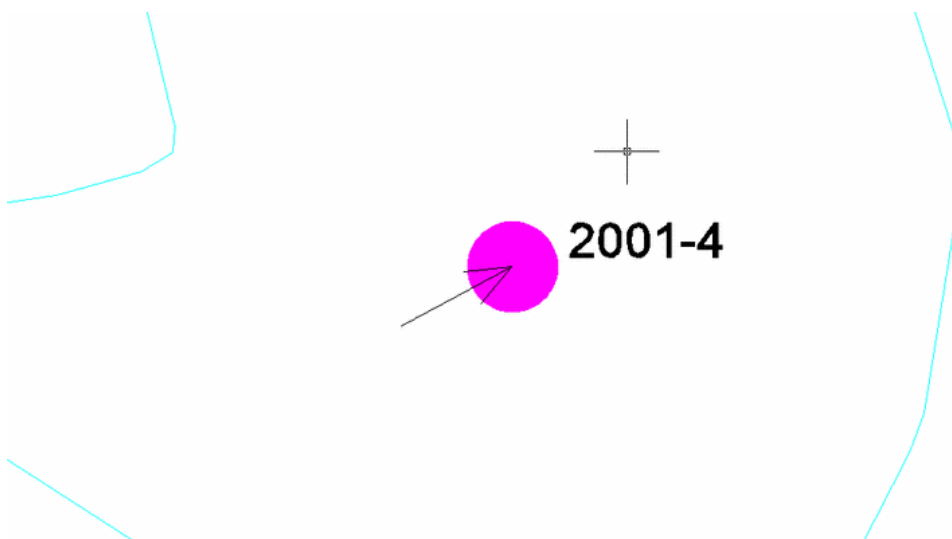
Select objects: Specify opposite corner: 6 found

Pulldown Menu Location: Drillhole

Keyboard Command: qgeocol

Find Drillhole

This command locates a drillhole by name. The screen is centered around the specified drillhole and a temporary arrow is drawn to it. Be sure that the drawing is not zoomed out too far, as the zoom is not modified, just the panning of the screen to center the drillhole.



Prompts

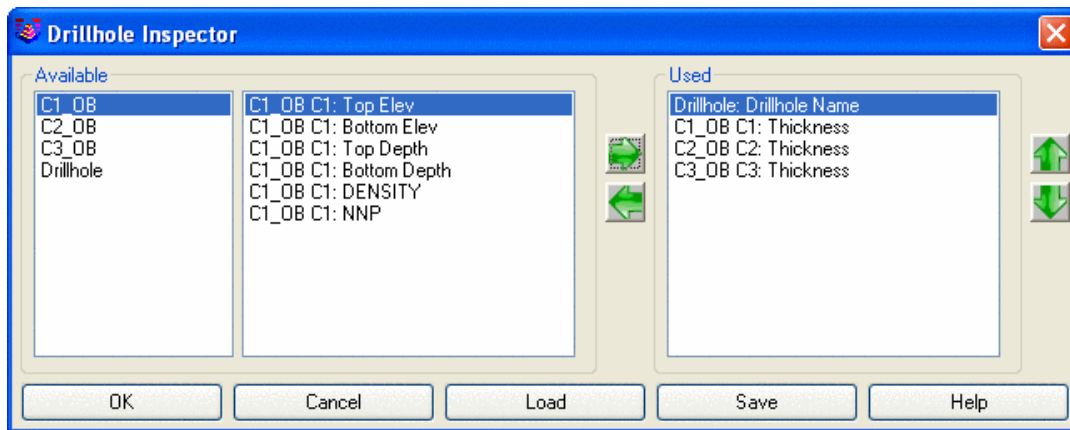
Enter drillhole name to find: 2001-4

Pulldown Menu Location: Drillhole

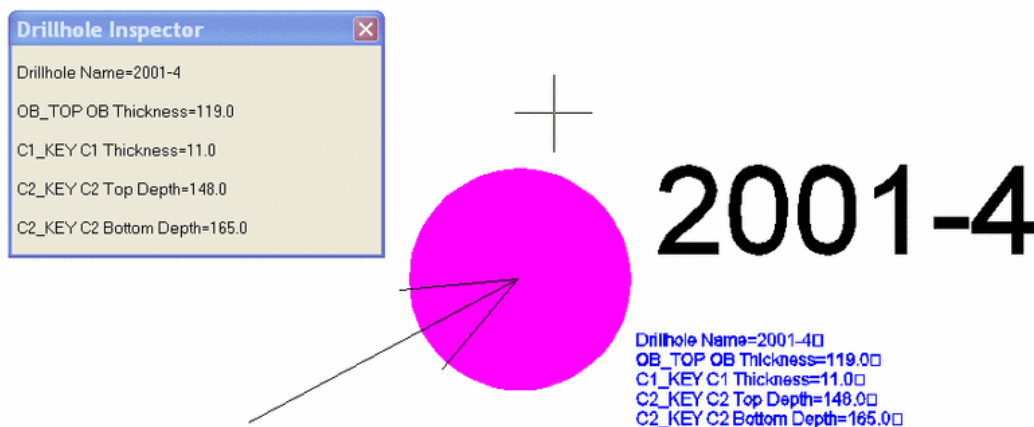
Keyboard Command: chfind

Drillhole Inspector

Another inspector command found in Carlson is the Drillhole Inspector. It analyzes the drillholes and displays, in the inspector window, the selected parameters. The first time this command is run, the inspector window (in the top left) is empty. The user must hit "O" for options to bring up the selection window. The possible parameters to be inspected are selected from the "available" window on the left and added to the "used" window on the right. This routine is very helpful in viewing drillhole data quickly as the cursor is moved across the drawing window, from hole to hole. A dashed arrow is shown in real-time to identify which hole is being inspected.



All drillhole information is available to the list for viewing. This includes such parameters as thickness, elevations, depth, and any user defined attributes. The "inspector window" in the top-left corner can be moved around the screen by grabbing the title bar at the top and dragging it. The values in the inspector window will change as the cursor is moved over each drillhole. Using the Label option, a label can be posted from available data in the drillhole to a specified insertion point, text size, and alignment.



Prompts

Process beds [Yes/<No>]? press Enter

Options/<Move pointer near drillhole>: Move cursor over drillholes. Type O to select drillhole options to view. Press Enter to exit.

Pulldown Menu Location: Drillhole

Keyboard Command: chinspect

Select Drillholes By Filter

This command creates a predefined selection set of drillholes by filtering them by any available field. The query is defined by either an And/Or. The field names are selected by the dropdown. These items are defined with the command Define Lookup Database, and also include all standard fields of a drillhole, such as name. Inclusion and Exclusion perimeters can also be used to aid in the filter. These are set in the upper right corner by selecting the perimeters. The SQL Query appears in the middle window. After choosing the Execute button, the drillholes that meet the criteria show up in the spreadsheet window below. To utilize these drillholes after the query, Save the query and hit OK. Then, at the command line in CAD, when prompted to "Select Objects" enter P (for Previous, or Predefined selection) and it will only use those holes for the operation.

AND/OR	FIELD NAME	OPERATOR	VALUE
- AND	NAME	LIKE	639*
- AND	DH_TYPE	=	CORE

SQL Query

```
SELECT * FROM DRILLHOLE WHERE  
[NAME] LIKE '639*' AND [DH_TYPE] = 'CORE'
```

NAME	DESCRIPTION	NORTHING	EASTING	ELEVATIO	KEY_DENS	PROCESS	DH_TYPE	XY_QUALI	Z_QUALIT
1 639260		639000.	3260000.	1400.9999	80.	1	0	0	0
2 639262		639000.	3262000.	1414.	80.	1	0	0	0
3 6392263		639230.	3263869.0	1449.	80.	1	0	0	0
4 6392615		639000.	3261500.	1409.	80.	1	0	0	0
5 6392625		639000.	3262500.	1419.	80.	1	0	0	0
6 6392645		639000.	3264500.	1461.	80.	1	0	0	0
7 6392655		639001.00	3265500.	1474.	80.	1	0	0	0
8 6392665		639001.00	3266500.	1466.	80.	1	0	0	0
9 6395261		639500.	3261000.0	1421.	80.	1	0	0	0
10 6395262		639500.	3262000.	1418.	80.	1	0	0	0
11 6395263		639500.	3263000.	1431.	80.	1	0	0	0
12 6395264		639400.00	3264000.0	1441.	80.	1	0	0	0

Prompts

Built selection of 22 drillholes.

To use type 'P' at Select objects: prompt.

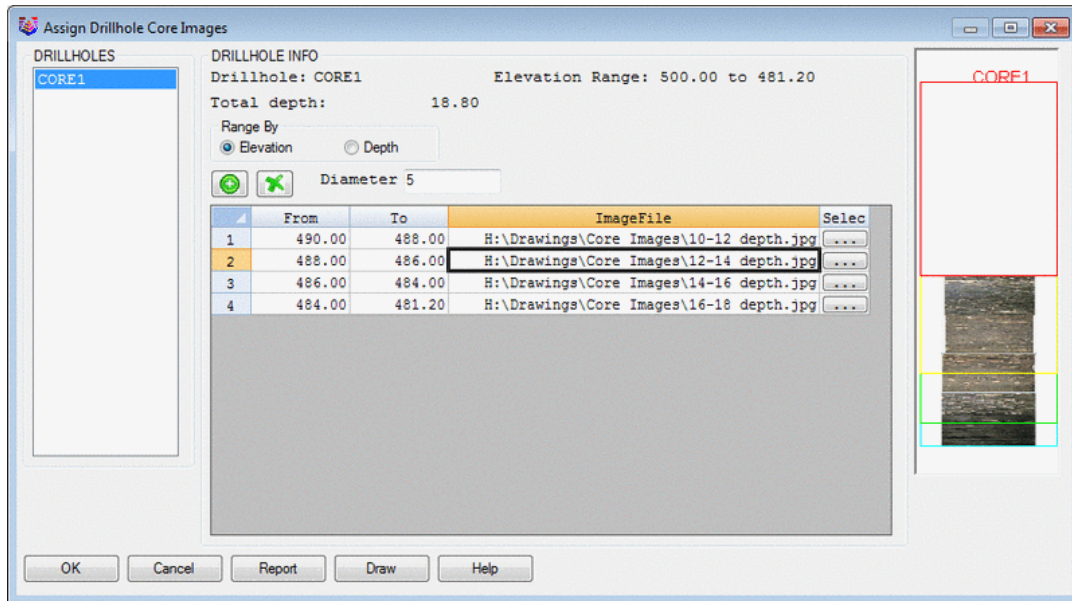
Command:

Pulldown Menu Location: Drillhole

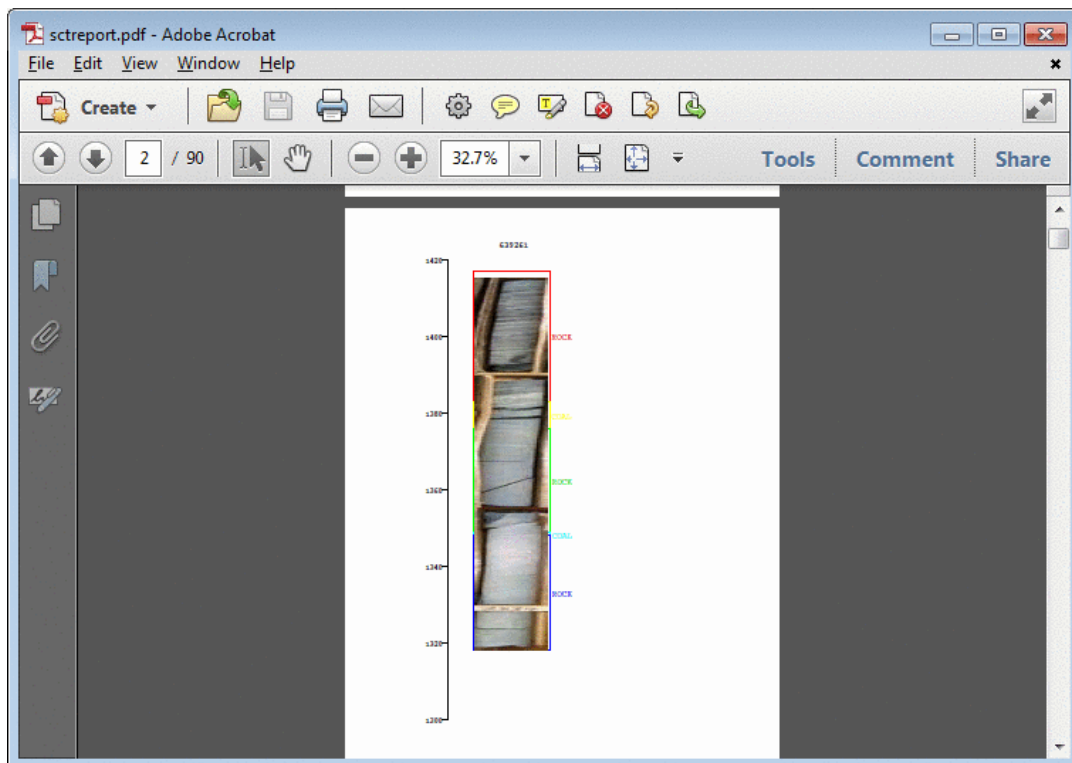
Keyboard Command:

Drillhole Core Images

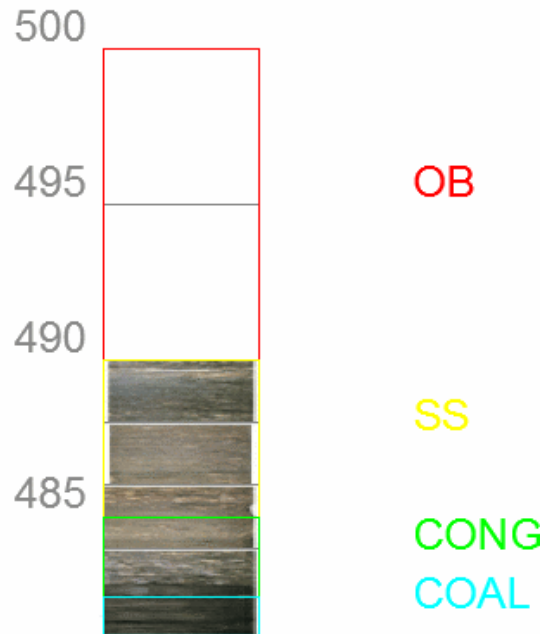
This routine uses images taken of cores and links them to depths in the holes for reporting and viewing in the drawing. The command will read both JPG and BMP files. In the assignment dialog the drillholes are listed on the left side. The highlighted hole shows up as a column on the right. The intervals for each photo are added to the middle section. Selecting the green + button adds a line in the window. The From and To Elev must be set for each image interval. For each line, use the Select button on the right to find the image file for each interval. As each is entered, the core image shows up on the right in the column for each interval. The Diameter controls how wide the column appears on the right.



Reports are generated of all the holes on the list with the Report button. An example appears here.



The Draw option will place the hole in CAD, and an example of that is shown here.



Prompts

Select drillholes to assign core images.

Select objects: *pick the drillholes*

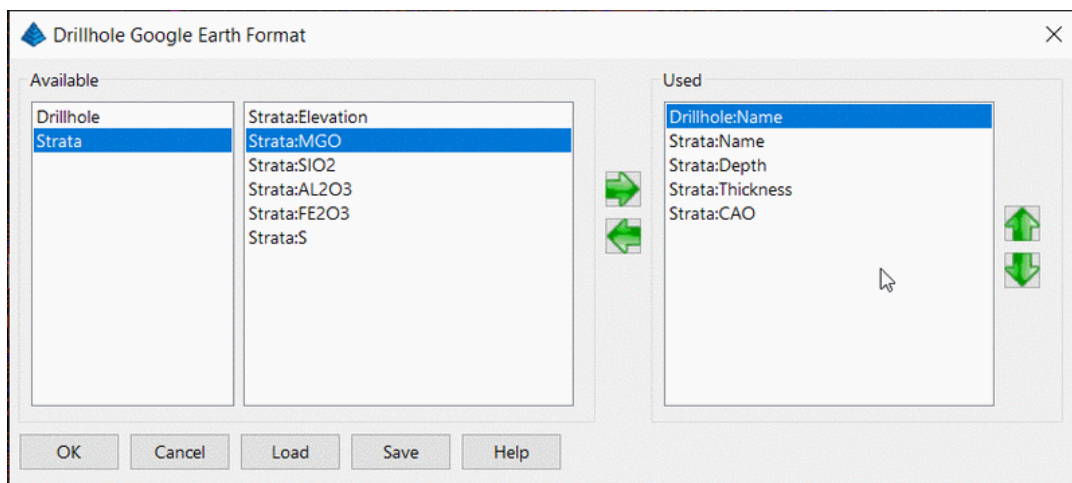
Drillhole Core Image Dialog

Drop-Down Menu Location: Drillhole, Drillhole Utilities

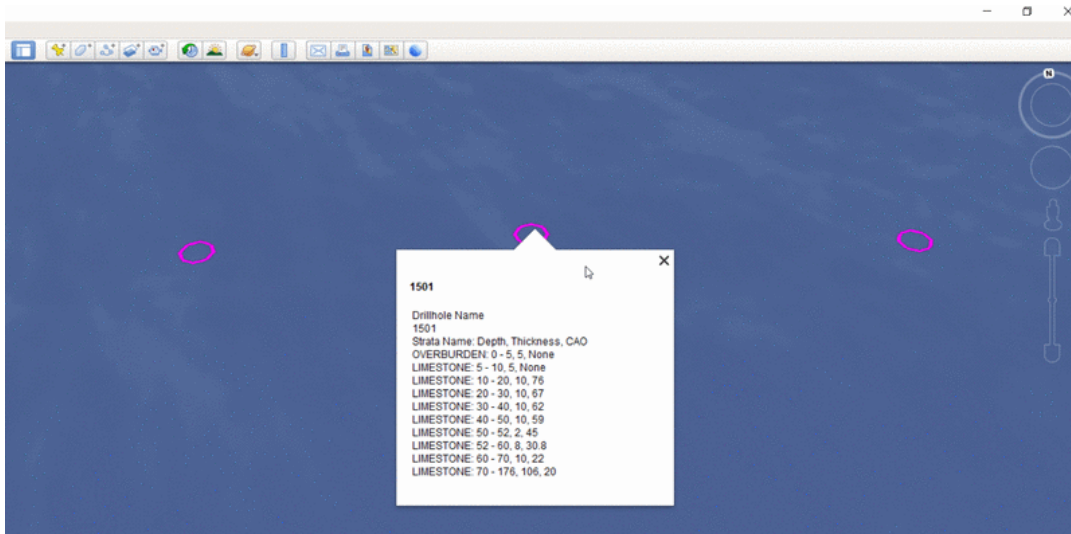
Keyboard Command: dhimage

Set Google Earth Drillhole Tags

This command sets selected fields of drillholes to be exported with the KML/KMZ data for display in Google Earth. Fields such as drillhole name, depth, thickness or any quality data may be tagged for export to Google Earth.



Select the fields from either the Drillhole or Strata section. Move them to the right in the order to be viewed in Google Earth. Then click OK. Once this command has been run, then the data is ready to export with the drillhole symbols to Google Earth. This is a separate command under **File > Export > Export Google Earth File**. Select the drillholes to export, and the symbols will appear georeferenced in Google Earth. The data can be verified with a pick on the drillhole symbol to review, and will appear similar to what is shown here.



Prompts

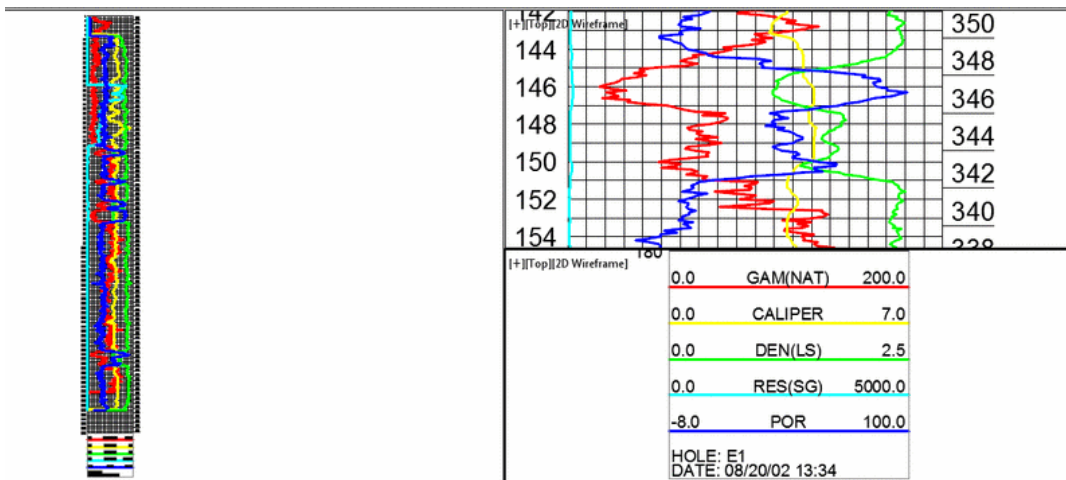
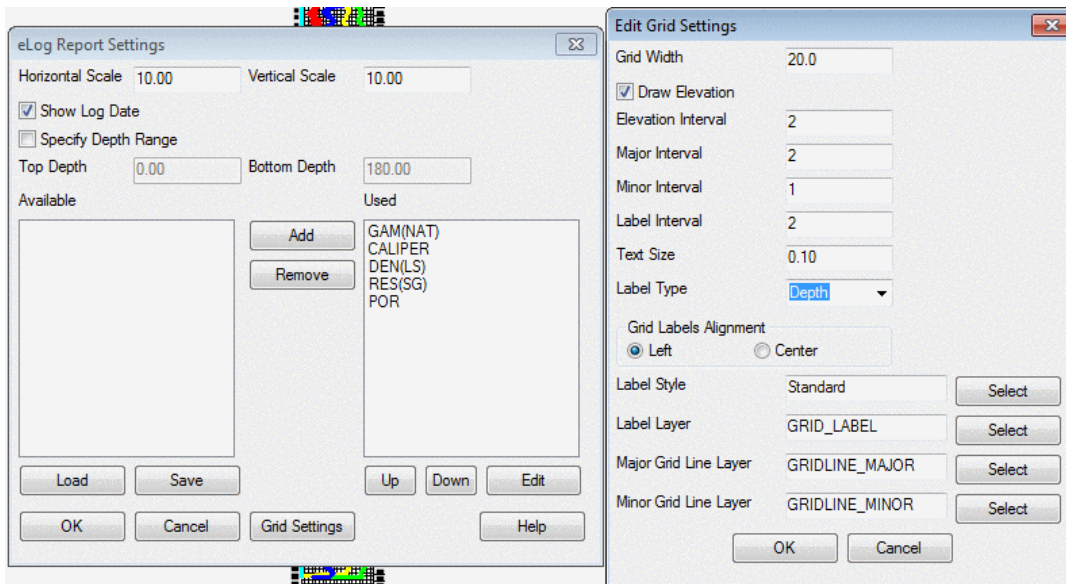
Select Drillholes, then dialog appears

Pull-down Menu Location: Drillhole > Drillhole Utilities

Keyboard Command: tagkmlch

Draw eLogs

This command takes a standard LAS file and draws the eLog data in CAD as linework, ready for analysis and plotting. It can be drawn next to an existing geologic column, or drawn by itself. LAS files are standard formats of electronic logs. The header data can vary, but the program reads through it and automatically find the Gamma, Caliper, Density, Resistivity and Por. The Automatic Mode draws the eLog next to existing geologic columns draw by the Draw Geologic Column command. For the Automatic Mode, there are settings for how far to draw the eLog to the left or right of the geologic columns, and for the folder with the LAS files. The program will match drillhole names in the drawing with the LAS file names in this folder.



Here is a sample LAS file format. The header data might be different for other files, but the program should recognize the columns either way.

Version Information

VERS. 2.0: CWLS LOG ASCII STANDARD -VERSION 2.0

WRAP. NO: ONE LINE PER DEPTH STEP

WELL INFORMATION BLOCK

#MNEM.UNIT DATA DESCRIPTION OF MNEMONIC

STRT.FT 0.400 :START DEPTH

STOP.FT 170.300 :STOP DEPTH

STEP.FT 0.100 :STEP UP_HOLE

NULL. -999.25 :NULL VALUE

COMP. TST :COMPANY

WELL. E1 :WELL

FLD . AREA Z :FIELD

LOC . :LOCATION

CNTY. MASON :COUNTY
 STAT. KENTUCKY :STATE
 SRVC. CARLSON :SERVICE COMPANY
 DATE. 08/20/02 :LOG DATE
 UWI . :UNIQUE WELL ID
 LIC . :LICENSE NUMBER
 Curve Information Block
 #MNEM.UNIT API CODE Curve Description
 #-----
 DEPT .FT 00 001 00 00 : 1 DEPTH
 GAM(NAT) .API-GR 00 310 00 00 : 2 GAM(NAT)
 CALIPER .INCH 00 280 00 00 : 3 CALIPER
 DEN(LS) .G/CC 00 350 00 00 : 4 DEN(LS)
 RES(SG) .OHM-M 00 220 00 00 : 5 RES(SG)
 POR .PERCENT 00 890 00 00 : 6 POR
 Parameter Information Block
 #MNEM.UNIT Information Description
 #-----
 FILE. PROCESSED :File Type
 FIID. 9039A :File Type Identifier
 VERS. 1.0 :System Version
 SER . 1 :System Serial Number
 TRUK. 0.4966 :Truck Calibration Number
 TOOL. 952 :Tool Serial Number
 TIME. 1334 :Time HrHrMinMin
 LAT . :Latitude
 LON . :Longitude
 LMF . GL :Log Measured From
 DMF . GL :Driller Measured From
 PD . GL :Permanent Data
 PDEV. :Elevation Permanent Data
 EKB .M :Elevation Kelly Bushing
 ELEV.DF :Elevation DF
 EGL .M 490.6 :Elevation Ground Level
 DRDP. 170 DW :Driller's Depth
 CASD. :Casing Diameter
 CASB. :Casing Bottom
 CASX. :Casing Type
 CAST. :Casing Thickness
 TNOC. :Time Circulation Stopped
 LOGU. 561 :Logging Unit
 RECB. B. MATAS :Recorded By
 OSR1. :Other Services
 OSR2. :Other Services
 OSR3. :Other Services
 BS .MM 4.75 :Bit Size
 MST . :Mean Surface Temperature
 TGRD. :Temperature Gradient
 MAGN. 0 :Magnetic Declination
 MDEN. 2.65 :Density Matrix
 MATR. SANDSTONE :Neutron Matrix
 DTMT. 54 :Delta T Matrix
 DTFL. :Delta T Fluid
 MUDES. :Mud Sample Source

MRS . :Mud Resistivity
 MTP . :Mud Temperature
 MFRS. :Resistivity Mud Filtrate
 MFTP. :Temperature Mud Filtrate
 MCRS. :Resistivity Mud Cake
 MCTP. :Temperature Mud Cake
 FTYP. WATER :Fluid Type
 FD .K/M3 :Fluid Density
 DFV .S :Fluid Viscosity
 FPH . :Fuild PH
 ELCO. 5000 :Electron Cutoff

Other Information

#MNEM.UNIT Information Description

A DEPTH GAM(NAT) CALIPER DEN(LS) RES(SG) POR
 0.400 -999999.0 -999999.0 -999999.00 -999999.0 -999999.0
 0.500 -999999.0 -999999.0 -999999.00 -999999.0 -999999.0
 0.600 26.3 0.0 0.00 0.0 0.0
 0.700 26.3 0.0 0.00 0.0 0.0
 0.800 46.1 0.0 0.00 0.0 0.0
 0.900 39.5 0.0 0.00 0.0 0.0
 1.000 23.0 0.0 0.00 0.0 0.0
 1.100 28.5 0.0 0.00 0.0 0.0
 1.200 55.9 0.0 0.00 0.0 0.0
 1.300 29.6 0.0 0.00 0.0 0.0
 1.400 36.2 0.0 0.00 0.0 0.0
 1.500 59.2 0.0 0.00 0.0 0.0
 1.600 42.8 0.0 0.00 0.0 0.0
 1.700 29.6 0.0 0.00 0.0 0.0
 1.800 32.9 0.0 0.00 0.0 0.0
 1.900 49.3 0.0 0.00 0.0 0.0
 2.000 36.2 0.0 0.00 0.0 0.0

Prompts

Add strata to top or bottom of drillhole [<Top>/Bottom]? *Top* (in this example)

Enter strata name: *Topsoil*

Enter bed name: *Top*

Status for strata [<Key>/Non-key]? *N*

Enter strata thickness: *1.0*

Select the Drillholes to process. Type ALL or select.

Updated 13 drillholes:

Pulldown Menu Location: Drillhole > Strata/Bed Utilities

Keyboard Command: addstrata

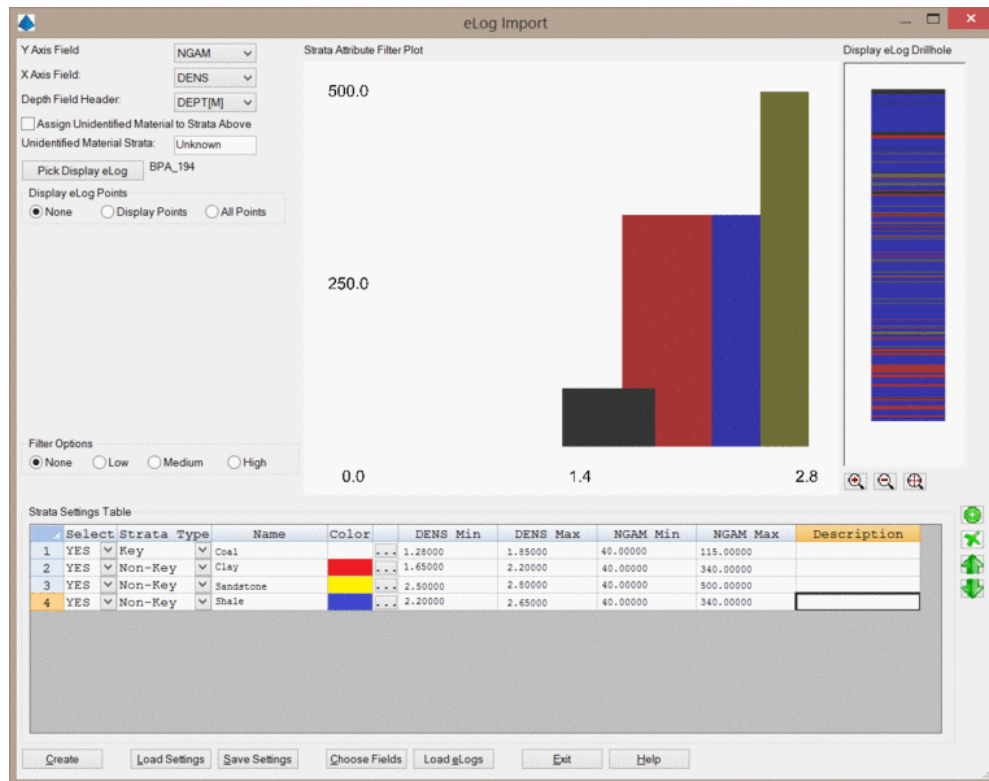
Prerequisite: Drillholes

Create Drillhole From eLog

This command will create drillholes from electronic logs of gamma ray and density readings that vary with depth (although other attributes may also be used for strata classification). These eLog files have an .las file extension,

which should not be confused with point cloud scans that sometimes use the same file extension. It is important to note that gamma/density surveys can vary greatly by equipment, and input settings will likely need to be calibrated to the equipment and project.

When the command is executed, the below dialog will appear.



To create a drillhole from an .las file, the following steps should be taken:

1. Click **Choose Fields** to setup the strata classification table
2. Populate the strata classification table
3. Click **Load eLog** to import .las files
4. Click **Create** to place the drillholes in the drawing

Y-Axis/X-Axis Field: These dropdown menus set the attributes to display on each axis of the strata classification chart shown in the middle of the dialog. The dropdown menus will only be populated after selecting fields with the **Choose Fields** button. These fields only affect the visualization of the strata classification table and do not affect how the strata are actually classified.

Depth Field Header: This dropdown menu sets the attribute to use for the Depth of the drillhole.

Assign Unidentified Material to Strata Above: This checkbox controls the classification of strata that do not fit into the strata classification table. When this option is enabled, unidentified strata will be assigned to the strata

above it. When this option is disabled, unidentified strata will be assigned a name according to the **Unidentifiable Strata Name** field.

Unidentifiable Strata Name: This field sets the name for strata that do not fit into the strata classification table.

Pick Display eLog: This button allows you to select an eLog file to use with the data preview. When multiple files are being imported at once, it can be useful to select a few different files to ensure the strata classification is applied correctly. When you click the Choose Fields button, the eLog file you use will be automatically selected for this option.

Display eLog Points: This option allows you to sample points on the graph area. Displaying these points allows you to quickly check where your eLog data fits into the strata classification you have defined.

The *None* option will not display data points.

The *Display Points* option will display points for the current eLog only (set via the **Pick Display eLog** button).

The *All* option will display points for all eLog files found in the **Load eLogs** list.

Filter Options: These options will filter the .las file to remove noise in the gamma/density readings. If the strata classification changes in an unrealistic manner, filtering the data may correct the problem.

The *None* option will not filter the input data.

The *Low*, *Medium*, and *High* options will filter the data more and more aggressively. When these filters are used, it is recommended to thoroughly check the output for accuracy.

The strata classification table is the spreadsheet at the bottom of the dialog. This table classifies each section of the drillhole to a strata according to min/max values of the density, gamma readings, and other attributes. This table should only be modified after selecting fields with the **Choose Fields** button. Strata may be added, removed, and sorted using the green icons to the right of the table. Here it is important to note that strata listed higher in the table will take precedence over strata on lower rows. In other words, if a section of the drillhole matches the criteria for multiple strata, it will be assigned to the first strata listed in the classification table. This concept is visualized in the classification chart. For example, in the above image, there is some overlap between the criteria for Coal and for Clay. Since Coal is listed at the top of the classification table, the black box in the chart representing Coal is drawn over the red box representing Clay.

Select: This column controls if the strata is used for classification. If this option is set to No, the respective strata will not appear in the drillhole.

Strata Type: This option allows you to define each strata layer as Key or Non-Key.

Name: This column sets the name for each strata.

Color: This column sets the color for each strata in the classification chart. This color is not used for any other purpose. Colors may be set by clicking the ellipse button to the right of this column.

Attribute Min/Max: These columns will be set according to the fields found in the .las file. Each column sets a bounding value for the strata. In the example shown in the above image, strata to be classified as Coal must have a density between 1.28 and 1.85 (note that these values are actually expressed as specific gravity) and a gamma reading between 40 and 115.

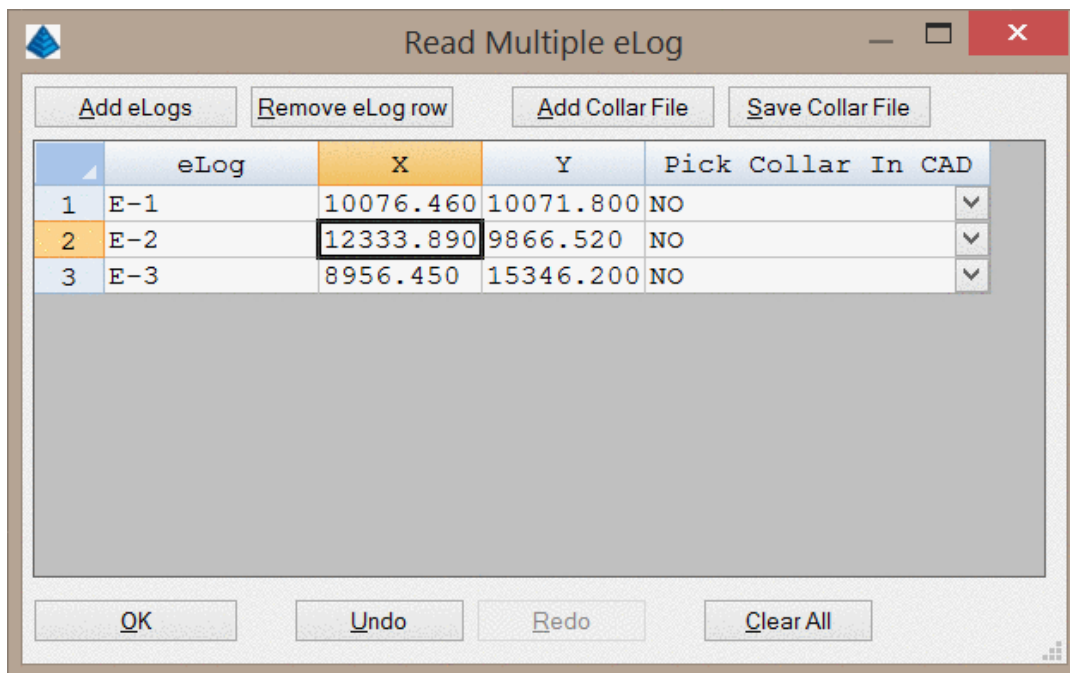
Description: This optional column can be used as a descriptor of the strata. This description will not be stored in the drillhole, however.

Create: This button will create a drillhole from the current .las file. When clicked, you will be prompted to select a location for the drillhole.

Load/Save Settings: These options will load/save settings to a .gd file.

Choose Fields: This button will prompt you for an .las file. The attribute headers found in the .las file will be added to the strata classification table.

Load eLog: This button will prompt you for one or more .las files. When clicked, the below dialog will appear.



Add eLogs: This button allows you to select .las files to add to the list.

Remove eLog row: This button will remove the currently selected eLog file from the list.

Add Collar File: This button allows you to import the coordinates of the eLogs from a text file. The format of this file should simply list the eLog name, the X coordinate, and the Y coordinate.

Save Collar File: This button allows you to save the current collar coordinates to an external text file.

Pick Collar in CAD: This option allows you to ignore the listed XY coordinates and allows you to instead pick the location of the collar for each drillhole in the CAD window.

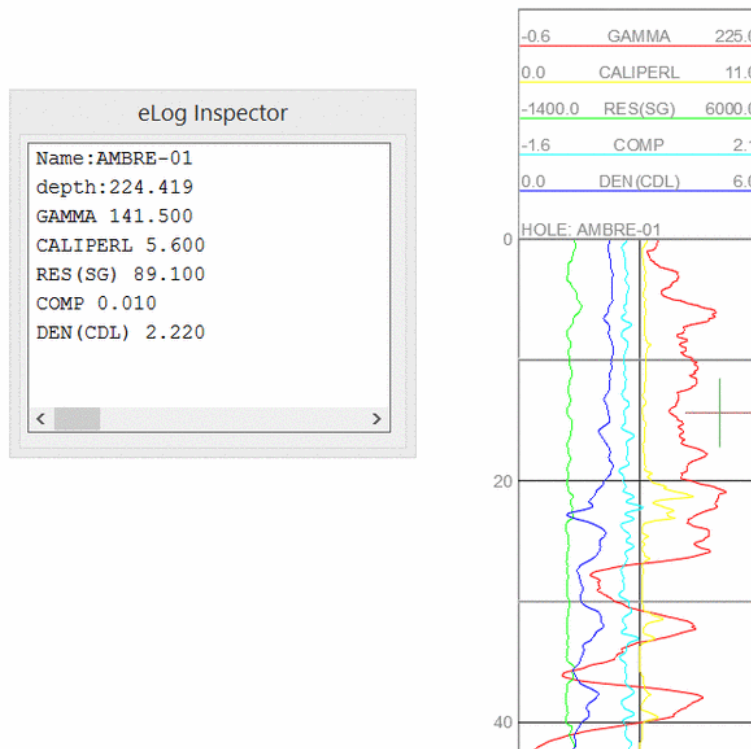
Pulldown Menu Location: Geology Module > Drillhole Pulldown Menu > Drillhole Utilities

Keyboard Command: eLog_import

Inspect eLog

This command is used to quickly display eLog attributes. This requires an eLog to have already been drawn on-screen via the Draw eLog command.

When active, a window will appear as shown below. As the cursor is moved over an eLog, the attributes of that eLog will be displayed in the window. Even if an attribute has not been drawn on the eLog, the attribute will still be displayed in the window.



Pulldown Menu Location: Drillhole > Drillhole Utilities

Keyboard Command: elog_inspect

Apply New Definition

This command allows you to update all drillholes after a change in the drillhole definition file (.CH) is made. The command Define Drillhole creates and modifies the .CH file.

Pulldown Menu Location: Drillhole, Drillhole Utilities

Keyboard Command: chredoall

Change Drillhole Symbol Number

This command will change the actual symbol of the drillhole on screen in the drawing from what was originally placed. The first window that appears is to select the symbol to change to. Click on the symbol desired with the cursor, select the holes to change and the holes are now changed.

Prompts

Symbol Selection Dialog *Choose a symbol*

Select the Drillholes to change symbol.

Select objects: *pick drillhole symbols*

Modified 3 drillhole symbols

Pulldown Menu Location: Drillhole

Keyboard Command: chsymbol2

Change Drillhole Symbol Size

This command changes the drillhole symbol size of the selected drillholes to the user-specified size. The diameter of the symbol will be changed to the entered drawing units. Another way to change all the symbols at once, is to change the size in Define Drillhole, then run Apply New Definition.

Prompts

DrillHole symbol size: *50*

Select the DrillHoles to resize.

Select objects: *pick drillhole symbols*

Changed 3 drillhole symbols

Pulldown Menu Location: Drillhole

Keyboard Command: chsymbol

Change Process On/Off Status

This command changes the processing status of the selected drillholes. If the status is off, then they can still appear on screen, but will be ignored for all processing. Drillholes are only used when the status is set to on.

Prompt

Select the Drillholes to set.

Select objects: Specify opposite corner: 42 found

Select objects:

Set processing status [<ON>/OFF]? *ON*

Modified 42 drillholes.

Pulldown Menu Location: Drillhole

Keyboard Command: chstatus

Change Drillhole Surface Z

This useful command will raise or lower the surface/collar elevation of the selected drillholes either by a surface model file or by an user entered differential. This is necessary when a series of holes have been drilled and surveyed from a local base elevation in the field. Once that bench mark elevation is determined, the selected holes in the drawing can be translated to the appropriate elevation. Older holes located before GPS surveys might not have the correct surface elevation. This will "drape" them onto a surface model file which can be a grid, TIN or FLT file.

Prompts

Select the Drillholes to process.

Select objects: all (to select the drillholes) 190 found

Set elevation by surface model or differential [<Model>/Differential]?

M for Model or Enter if in <>

Reading cell> 251316

Hold strata thickness [<Yes>/No]? y Yes will keep the same thickness of all strata, No will change the thickness of the first strata in the drillhole.

Modified 180 drillholes.

Skipped 10 drillholes already at elevation

OR:

Select the Drillholes to process.

Select objects: pick the drillholes

Enter elevation change: 150. Use a positive value to raise the elevation and a negative value to move down. This example will raise the elevations by 150.

Hold strata thickness [<Yes>/No]? press Enter for Yes If Yes, then the complete stratigraphic column will be moved up or down, keeping the thickness of each the same. If No, then the thickness of the first strata in the hole will be affected; thicker with a positive value as the collar elevation is raised, and thinner if negative.

Modified 15 drillholes.

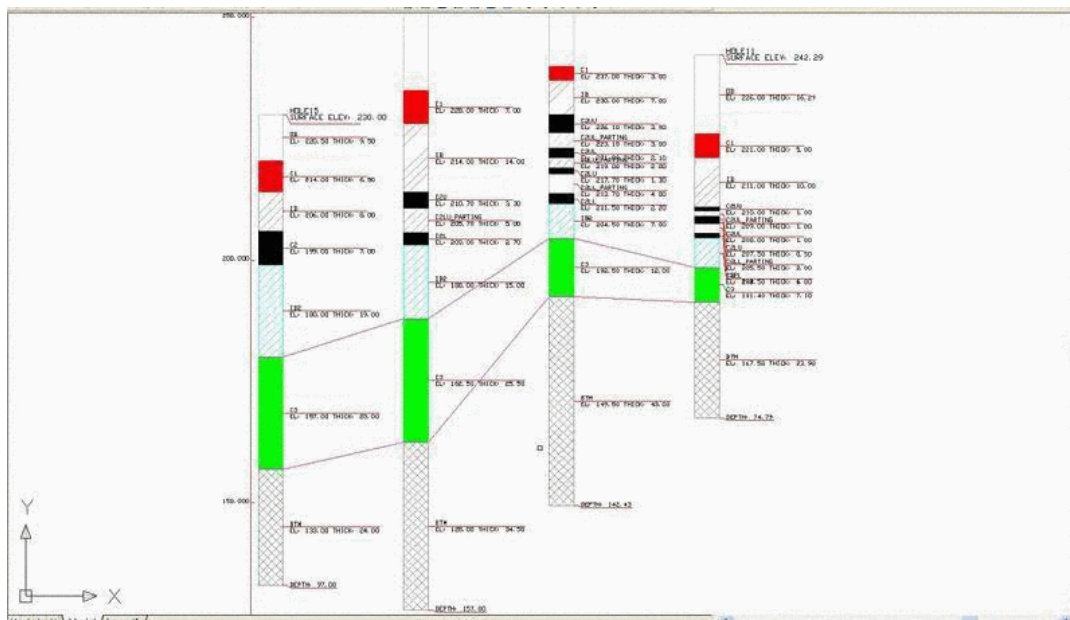
Pulldown Menu Location: Drillhole

Keyboard Command: chelev

Assign Strata Correlation

This command assigns strata and bed names to intervals in a geologic column view. This is a great tool for a geologist to do the on-screen correlating. Rows of geologic columns can be drawn, and correlated just by "picking" the intervals to connect with lines. Before starting this command, create geologic columns with the Draw Geologic Column command. The Fill option set to Outline Only can make it easier to pick the strata.

While viewing the geologic columns, pick the strata polylines to name on the left or right edge, or they can be selected with a window. Then a dialog appears where you can enter the strata name and an optional bed name. These names are saved back in the drillholes, or in the database. There is an option to connect the strata with the same names with lines. The one thing it does not do is update the text. The drillholes need to be drawn again to see that update.



Prompts

Connect strata with lines (<Yes>/<No>)? press Enter for Yes

Select the Strata to name.

Select objects: pick the strata polylines from the geologic column

Name Strata Dialog



Select the Strata to name.

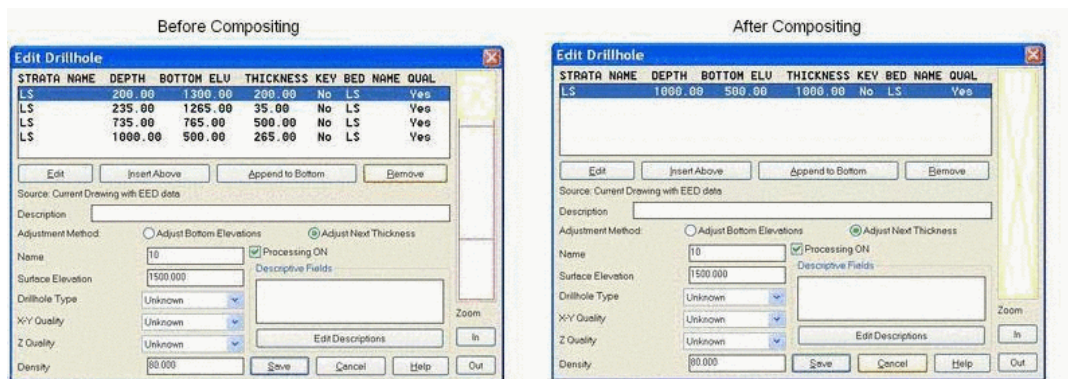
Select objects: press Enter to end

Pulldown Menu Location: Drillhole in Advanced Mining

Keyboard Command: nmstrata

Composite Duplicate Strata

This command will composite all duplicate strata names found within the selected drillholes. To be composited, the strata must have the same bed names, the same key/non-key status and they must be in sequential order without any other strata in between (no partings). There is an option for whether the strata must have the same name. Also, there is an option whether to composite only key or only non-key strata. Since Carlson doesn't model duplicate strata without bed names, this routine can be useful to composite a strata that duplicates itself. The strata qualities will be weight averaged by thickness, giving one value for each attribute.



Prompts

Select the Drillholes to process.

Select objects: select drillholes to composite duplicate strata

Strata to process [Key/Non-key/<Both>]? press Enter

Require strata name to match [<Yes>/<No>]? press Enter

Update 19 drillholes.

Pulldown Menu Location: Drillhole > Strata/Bed Utilities

Keyboard Command: dupstrata

Prerequisite: Drillholes

Composite Strata By Beds

This command composites strata within a drillhole based on bed name and key/non-key status. All the non-key strata with a matching bed name that are above the first key strata are composited and assigned the name `strata_top`. The non-key strata between the key strata are composited as `strata_parting`. The key strata with a matching bed name are composited as `strata_key`. The non-key strata below the last key are composited as `strata_bottom`. This compositing method is the same the StrataCalc uses for processing strata with bed names. The strata qualities will be weight averaged by thickness, giving one value for each attribute.

Before Composite Strata By Beds

Strata name	Bed name	Key	Thick	Bot	Elev
ROCK	C1	NO	67.00	1684.12	
ROCK	C1	NO	1.00	1683.12	
COAL	C1	YES	1.50	1681.62	
ROCK	C1	NO	0.25	1681.37	
COAL	C1	YES	0.25	1681.12	
COAL	C1	YES	2.00	1679.12	
ROCK	C1	NO	1.00	1678.12	

After Composite Strata By Beds

Strata name	Bed name	Key	Thick	Bot	Elev
C1.TOP	C1	NO	68.00	1683.12	
C1.PARTING	C1	NO	0.25	1682.87	
C1.KEY	C1	YES	3.75	1679.12	
C1.BOTTOM	C1	NO	1.00	1678.12	

Prompts

Select the Drillholes to composite.

Select objects: *pick the drillholes to process*

Pulldown Menu Location: Drillhole-> Strata/Bed Utilities

Keyboard Command: `compstrata`

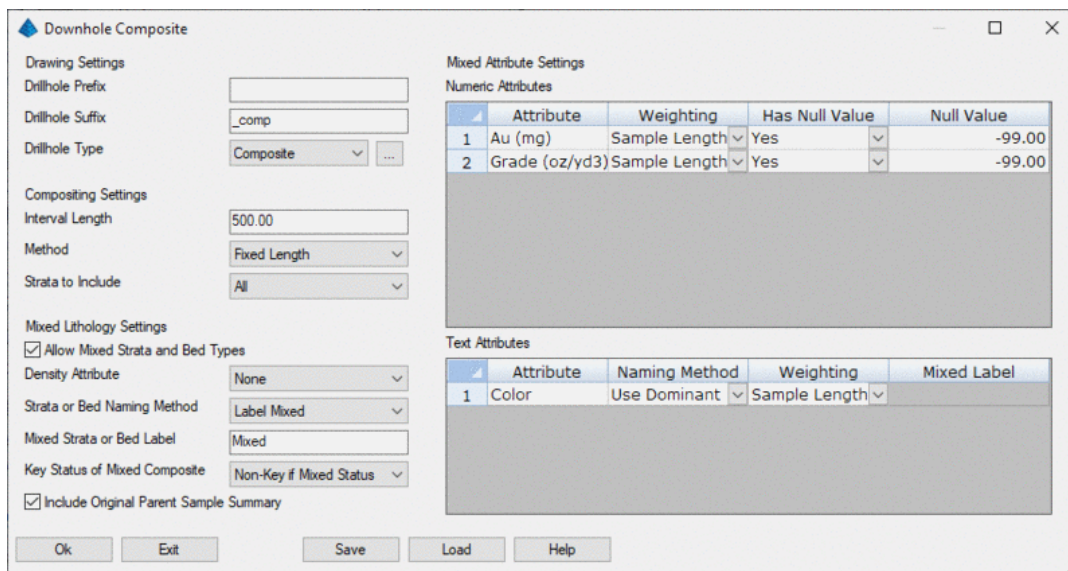
Prerequisite: Drillholes

Composite Strata By Interval

The purpose of compositing strata is to create regularized samples and control bias for geostatistical modeling. For example, one drilling campaign may have sampled strata on a 10' interval, while another drilling campaign sampled on a 3' interval. Using a mix of the two in modeling would lead to distinct zones of higher and lower grade variability purely due to the measurement length. Compositing the 3' to 10' would remove this measurement bias.

It is important to note that this command will create new drillhole entities rather than modifying the original drillholes. This allows you to run other commands with either the original drillholes or the composited drillholes to compare results.

When the command is first executed, you will be prompted to select the drillholes to modify. The below dialog will then appear.



Drillhole Prefix/Suffix: These settings will apply a prefix/suffix to the names of the composited drillholes.

Drillhole Type: This setting controls the Drillhole Type of the composited drillholes. It is important to note that this option will not allow you to select a Drillhole Type in use by the selected drillholes - you must select a new drillhole type. Drillhole Types may be modified by clicking the ellipsis button. More information regarding Drillhole Types is found in the Drillhole Lookup Database documentation.

Interval Length: This value sets the sample length for the new composites. If the **Method** is set to Fixed Length, the program will start at the top of the drillhole and create composites of this exact length moving downward. Any remaining length will be retained as a shorter sample. If the **Method** is set to Best Fit, the program will divide the drillhole into equal lengths that are as close to the specified **Interval Length** as possible.

Strata to Include: This option controls which Strata will be considered for compositing. When using the Selected option, you will be prompted to select the strata in a separate dialog.

Allow Mixed Strata and Bed Types: This option determines if compositing will occur across strata/bed boundaries. When enabled, other settings related to mixing strata/bed types will be made active.

Density Attribute: This option sets the density attribute used for attribute weighting. This option is only required when samples are attributes are weighted by Sample Weight (as opposed to Sample Length)

Strata or Bed Naming Method: This option determines the resulting strata/bed name when composites cross strata/bed boundaries.

The Use Dominant option will retain the name of the longest/heaviest sample contributing to the composite. For example if a 10' composite is created from 7' of Strata A and 3' of Strata B, the resulting composite strata name will be A.

The Concatenate option will concatenate the strata names contributing to composite. For example, if a 10' composite is created from 3' of Strata A, 3' of Strata B, and 4' of Strata C, the resulting composite strata name will be A.B.C.

The Label Mixed option will not retain any of the original strata names when more than one strata contributes to the composite, but will instead label the resulting strata according to the name entered for the **Mixed Strata or Bed Label**.

It is important to note that the Strata Names and Bed Names are considered separately. If a composite is created from 3 samples of Strata A with varying Bed Names, only the Bed Name will be subject to change. Alternatively, if a composite is created from 3 samples with 3 different Strata names but the same Bed Name, only the Strata Name will be subject to change.

Key Status of Mixed Composite: This option controls the key status of the resulting composite strata.

The Use Dominant option will retain the key status of the longest/heaviest strata contributing to the composite.

The Key if Mixed Status will set all Mixed composites as key.
 The Non-Key if Mixed Status will set all Mixed composites as non-key.

Include Original Parent Sample Summary: When enabled, the composited strata will contain additional strata attributes indicating the strata attribute values from the contributing strata.

Numeric Attributes: This spreadsheet will list all numeric strata attributes found in the selected drillholes. Strata attributes may be averaged by the **Weighting** column (Sample Length or Sample Weight). If the strata attributes contain a specific value for null values, this value may be specified in the **Null Value** column when the **Has Null Value** column is set to Yes.

Text Attributes: This spreadsheet will list all text strata attributes found in the selected drillholes. If the **Naming Method** is set to Dominant, only the value of the longest/heaviest contributing strata will be retained in the composite, based on the option selected in the **Weighting** column. If the **Naming Method** is set to Concatenate, the resulting text value will be a concatenation of the contributing text values. If the **Naming Method** is set to Label Mixed, the resulting text value will be set according to the **Mixed Label** column.

An example drillhole before and after compositing on a 5' Fixed Length is shown below.



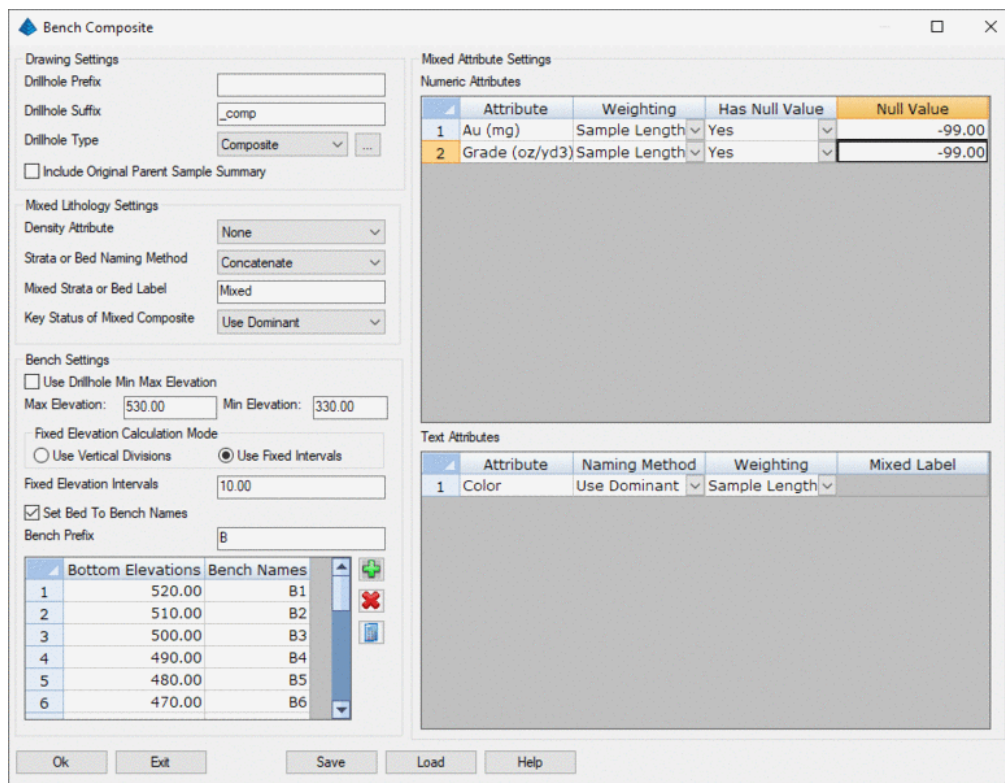
Pulldown Menu Location: Drillhole > Strata/Bed Utilities
Keyboard Command: compdownhole

Composite Strata by Elevation

The purpose of compositing strata is to create regularized samples and control bias for geostatistical modeling. For example, one drilling campaign may have sampled strata on a 10' interval, while another drilling campaign sampled on a 3' interval. Using a mix of the two in modeling would lead to distinct zones of higher and lower grade variability purely due to the measurement length. Compositing the 3' to 10' would remove this measurement bias.

It is important to note that this command will create new drillhole entities rather than modifying the original drillholes. This allows you to run other commands with either the original drillholes or the composited drillholes to compare results.

When the command is first executed, you will be prompted to select the drillholes to modify. The below dialog will then appear.



Drillhole Prefix/Suffix: These settings will apply a prefix/suffix to the names of the composited drillholes.

Drillhole Type: This setting controls the Drillhole Type of the composited drillholes. It is important to note that this option will not allow you to select a Drillhole Type in use by the selected drillholes - you must select a new drillhole type. Drillhole Types may be modified by clicking the ellipsis button. More information regarding Drillhole Types is found in the Drillhole Lookup Database documentation.

Include Original Parent Sample Summary: When enabled, the composited strata will contain additional strata attributes indicating the strata attribute values from the contributing strata.

Density Attribute: This option sets the density attribute used for attribute weighting. This option is only required when samples are attributes are weighted by Sample Weight (as opposed to Sample Length)

Strata or Bed Naming Method: This option determines the resulting strata/bed name when composites cross strata/bed boundaries.

The Use Dominant option will retain the name of the longest/heaviest sample contributing to the composite. For example if a 10' composite is created from 7' of Strata A and 3' of Strata B, the resulting composite strata name will be A.

The Concatenate option will concatenate the strata names contributing to composite. For example, if a 10' composite is created from 3' of Strata A, 3' of Strata B, and 4' of Strata C, the resulting composite strata name will be ABC.

The Label Mixed option will not retain any of the original strata names when more than one strata contributes to the composite, but will instead label the resulting strata according to the name entered for the **Mixed Strata or Bed Label**.

It is important to note that the Strata Names and Bed Names are considered separately. If a composite is created from 3 samples of Strata A with varying Bed Names, only the Bed Name will be subject to change. Alternatively, if a composite is created from 3 samples with 3 different Strata names but the same Bed Name, only the Strata Name will be subject to change.

It is also important to note that the **Set Bed To Bench Names** option will override any naming convention applied to the Bed Names.

Key Status of Mixed Composite: This option controls the key status of the resulting composite strata.

The Use Dominant option will retain the key status of the longest/heaviest strata contributing to the composite.

The Key if Mixed Status will set all Mixed composites as key.
 The Non-Key if Mixed Status will set all Mixed composites as non-key.

Use Drillhole Min/Max Elevation: When enabled, this option will automatically set the **Max Elevation** and **Min Elevation** values according to the max/min values detected from the selected option . When disabled, you will be able to set the **Max Elevation** and **Min Elevation** values manually.

Fixed Elevation Calculation Mode: This option controls how the Bottom Elevations will be automatically calculated. The Use Vertical Divisions option will allow you to specify the **Number of Vertical Divisions** to set between the **Max Elevation** and **Min Elevation**. The Use Fixed Intervals will allow you to enter the **Fixed Elevation Intervals** value to set the Bottom Elevations for each Interval. To apply each option, you may click the calculator icon to the right of the spreadsheet.

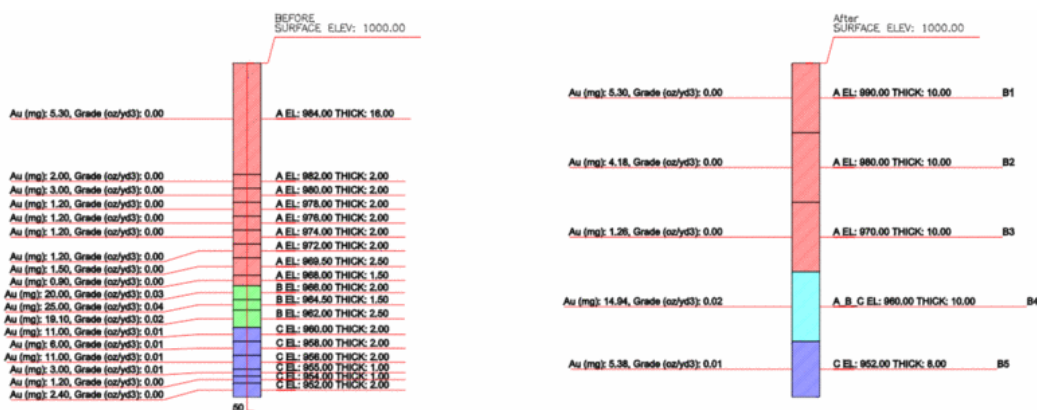
Set Bed to Bench Names: This option will overwrite all new bed names according to the **Bench Names** column of the spreadsheet. When intervals are automatically calculated by clicking the calculator icon, the bench names will be generated in increasing order from top to bottom. You may specify a **Bench Prefix** if desired.

The + and X icons next to the spreadsheet allow you to add or remove rows. This is useful if you need to add a Bottom Elevation that is not automatically calculated for you. It is important to note that Bottom Elevations must be listed in decreasing order from Top to Bottom. If this convention is not met, the out-of-order cells will be highlighted with a red fill.

Numeric Attributes: This spreadsheet will list all numeric strata attributes found in the selected drillholes. Strata attributes may be averaged by the **Weighting** column (Sample Length or Sample Weight). If the strata attributes contain a specific value for null values, this value may be specified in the **Null Value** column when the **Has Null Value** column is set to Yes.

Text Attributes: This spreadsheet will list all text strata attributes found in the selected drillholes. If the **Naming Method** is set to Dominant, only the value of the longest/heaviest contributing strata will be retained in the composite, based on the option selected in the **Weighting** column. If the **Naming Method** is set to Concatenate, the resulting text value will be a concatenation of the contributing text values. If the **Naming Method** is set to Label Mixed, the resulting text value will be set according to the **Mixed Label** column.

An example before and after compositing on 10' intervals from elevation 1000' to elevation 950' is shown below. Note that the strata names have been concatenated while the bed names have been set according to the bench name.



Pulldown Menu Location: Drillhole > Strata/Bed Utilities
Keyboard Command: compbench

Set Strata Key Status

This command processes a given set of drillholes, searches for the information from the selected drillholes and sets key status of each strata with the user-specified name to the required value. This is a much faster way to change key/non-key strata type than going through the Edit Drillhole or Drillhole Datasheet commands. The * wildcard is supported in this command for multiple selections. Bed names can be used as a filter.

Prompts

Select the Drillholes to process.

Select objects: Specify opposite corner: 7 found

Select objects:

Enter name of the strata to modify (wildcards supported): LS*

Enter bed name to limit the selection or press Enter for none:

Set status as [Key/<Non-key>]? K

Modified 7 drillholes.

Pulldown Menu Location: Drillhole, Strata/Bed Utilities

Keyboard Command: stratakey

Fill-in Missing Key as Zero

This command inserts zero thickness key strata when the strata is missing from the geological sequence in the drillhole. This change is permanent to the drillhole, so make sure to have a backup if this is not the desired change. This allows for more control in the pinch out modeling of the drillholes and thickness modeling.

Select the Drillholes to process.

Select objects: Specify opposite corner: 174 found

Modified 12 drillholes.

Pulldown Menu Location: Drillhole > Strata/Bed Utilities in Geology

Keyboard Command: fillstrata

Add Strata

This routine inserts an interval into the drillholes at either the very top or the very bottom of the selected holes. It is necessary to do this for certain types of geological modeling that have the top or bottom seam disappearing, and not all intervals in the hole have a bed name, such as in this lignite modeling. It "borrows" thickness from either the top or bottom interval, as in this example shown below in the ROCK strata. Some prefer to enter in a 0.1 interval and just call it TOP.

Edit Drillhole

STRATA NAME	DEPTH	BOTTOM ELU	THICKNESS	KEY	BED NAME	QUAL
ROCK	99.00	4818.17	99.00	No		No
COAL	102.00	4815.17	3.00	Yes	C1	No
ROCK	131.40	4785.77	29.40	No		No
COAL	142.40	4774.77	11.00	Yes	C2	No

Source: Current Drawing with EED data

Description:

Adjustment Method:
 Adjust Bottom Elevations
 Adjust Next Thickness

Name: Processing ON

Surface Elevation:

Drillhole Type:

X-Y Quality:

Z Quality:

Density:

Zoom:

Edit Drillhole

STRATA NAME	DEPTH	BOTTOM ELU	THICKNESS	KEY	BED NAME	QUAL
TOPSOIL	1.00	4916.17	1.00	No	TOP	No
ROCK	99.00	4818.17	98.00	No		No
COAL	102.00	4815.17	3.00	Yes	C1	No
ROCK	131.40	4785.77	29.40	No		No
COAL	142.40	4774.77	11.00	Yes	C2	No

Source: Current Drawing with EED data

Description:

Adjustment Method:
 Adjust Bottom Elevations
 Adjust Next Thickness

Name: Processing ON

Surface Elevation:

Drillhole Type:

X-Y Quality:

Z Quality:

Density:

Zoom:

It went from 99 to 98 thick, with the addition of 1 foot of topsoil. This will aid in the modeling of the seam C1 in holes where it does not exist, due to pinch or erosion. This routine should not be used in holes that are drilled in a valley, and the C1 seam is above the collar; it will be modeled with conformance.

Prompts

Add strata to top or bottom of drillhole [<Top>/Bottom]? *Top* (in this example)

Enter strata name: *Topsoil*

Enter bed name: *Top*

Status for strata [<Key>/Non-key]? *N*

Enter strata thickness: *1.0*

Select the Drillholes to process. Type ALL or select.

Select objects: Specify opposite corner: 22 found

9 were filtered out.

Select objects: *press Enter to accept*

Updated 13 drillholes:

Pulldown Menu Location: Drillhole > Strata/Bed Utilities

Keyboard Command: addstrata

Prerequisite: Drillholes

Delete Strata

This command allows you to remove certain strata in multiple drillholes in one step. If strata is removed the thickness of strata removed is added to strata below.

Prompts

Select the Drillholes to process.

Select objects: Specify opposite corner: 20 found

Select objects:

Enter name of the strata to modify (wildcards supported): *LS*

Enter bed name to limit the selection or press Enter for none:

Modified 20 drillholes.

Pulldown Menu Location: Drillhole > Strata/Bed Utilities

Keyboard Command: stratadel

Rename Strata

This command allows you to rename a certain strata in multiple drillholes in one step. It prompts for the old strata name and then the new name.

Prompts

Select the Drillholes to process.

Select objects: Specify opposite corner: 20 found

Select objects:

Enter name of the strata to modify (wildcards supported): *LS*

Enter bed name to limit the selection or press Enter for none:

Enter a new name for the strata: *LS1*

Modified 20 drillholes.

Pulldown Menu Location: Drillhole > Strata/Bed Utilities

Keyboard Command: strataren

Reset Invalid Attribute

This command will set the invalid attribute values in the selected drillholes to blank. The range of valid attribute values are set in the Define Attributes command. Any attributes found in the drillholes outside that range are deleted.

Prompts

Select the Drillholes to process.
Select objects: *select the drillholes*
Modified 309 drillholes.

Pulldown Menu Location: Drillhole > Strata/Bed Utilities
Keyboard Command: resetattr

Delete Attribute

This routine is a quick and easy way to remove the specified attribute from the drillholes. It will remove the attribute from all the strata in the selected drillholes.

Prompts

Select the Drillholes to process.
Select objects: *select drillholes*
Enter attribute name to delete: *BTU*
Modified 219 drillholes.

Pulldown Menu Location: Drillhole > Strata/Bed Utilities
Keyboard Command: delattr

Rename Attribute

The function of this command is to rename attributes of selected drillholes. This is the only way to globally rename the name of attributes within holes. Be sure to spell the attribute name precisely the way it appears in the holes.

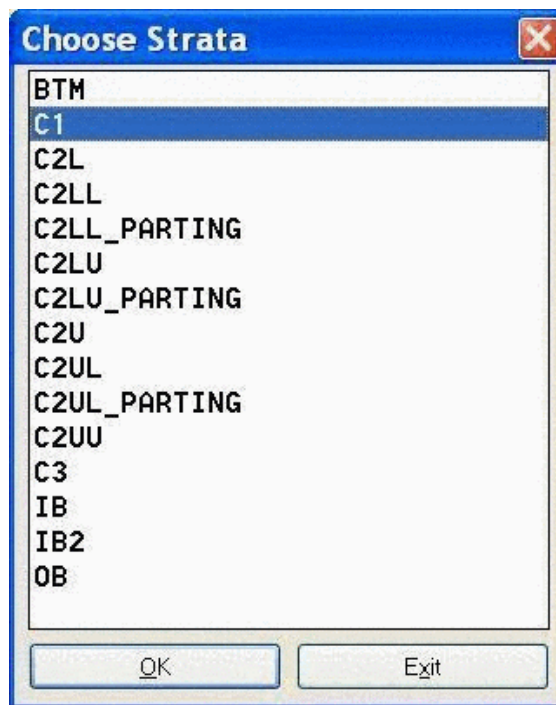
Prompts

Select the Drillholes to process.
Select objects: *select the drillholes*
Enter old attribute name to rename: *CA*
Enter new attribute name: *CAC03*
Modified 35 drillholes.

Pulldown Menu Location: Drillhole > Strata/Bed Utilities
Keyboard Command: renmattr

Assign Bed Names

This command assigns a bed name for each strata with picked names in a given set of drillholes. The strata list is generated based on drillholes selected enabling user to pick multiple strata to process. Hold the CTRL or SHIFT keys to select multiple strata for bed assignment.



Prompts

Select the DrillHoles for bed assignment:

Select objects: *Select Drillholes*

Enter name for the bed: *C1_BED*

Pulldown Menu Location: Drillhole > Strata/Bed Utilities

Keyboard Command: assignbed

Fill-in Bed Names

For some methods of processing the strata modeling routines, the drillhole set should have either no duplicate strata names or bed names assigned for each strata. When bed names are assigned to the key strata (or just some strata within drillhole), the blanks may be filled-in using this routine. The routine works from the top of the drillhole down until it finds the first strata with a non-empty bed name. That bed name is then applied to all strata with empty bed names above that strata. This procedure is then repeated to the bottom of the hole. For the strata below the last strata with bed name assigned, the last bed name is used.

Pulldown Menu Location: Drillhole > Strata/Bed Utilities

Keyboard Command: fillbed

Remove All Non-Key Bed Names

This command sets the bed name to blank for all non-key strata in the selected drillholes.

Pulldown Menu Location: Drillhole > Strata/Bed Utilities

Keyboard Command: remove_all_nk_bed

Prerequisite: Drillholes in drawing

Remove Bed Names After Last Key

For some methods of modeling, users prefer to remove or delete all of the bed names in a drillhole after the last Key interval. This command will do the entire selection set of drillholes at one time.

Prompts

Select the Drillholes for bed process.

Select objects: Specify opposite corner: 11 found

Select objects:

Updated 7 drillholes

Pulldown Menu Location: Drillhole > Strata/Bed Utilities

Keyboard Command: remove_last_nk_bed

Split Bed

For the selected drillholes, this command provides the ability to manipulate an existing bed at certain elevations or thickness. The bed can be split by changing the key/non-key status or by supplying a new bed name. With a new bed name, the bed will be split into one part with the old name and the other with the new name. The elevation at which to split the bed can be specified as a flat elevation or as a grid file. For the grid file, the split elevation is defined by the grid file at the drillhole location. The thickness method allows you to split off a specified thickness from the top or bottom of the bed or to add a specified thickness from the adjoining strata to the top or bottom of the bed. Besides changing the structure (thickness/elevation) of the bed, this routine will also split the bed qualities.

One application of this routine would be to use the thickness option to set the top 0.1 of the KEY bed to non-key to model loss at the top of the seam. Another application is to split a large KEY bed into two beds at an elevation grid to model mining the bed in two passes or benches.

Prompts

Split strata method [<Elevation>/Grid/Thickness]? *T for Thickness*

Select the Drillholes for bed split.

Select objects: *pick the drillhole symbols*

Enter name of the bed to split: *C2*

Rename bed or assign key/non-key status [<Name>/Status]? *press Enter for Name*

Enter new name for the thickness change: *C2B*

Split from top or bottom [<Top>/Bottom]? *press Enter for Top*

Add or subtract thickness from bed [<Add>/Subtract]? *S for Subtract*

Enter thickness (ft): *2*

By Elevation:

Split strata method [<Elevation>/Grid/Thickness]? *press Enter for Elevation*

Select the Drillholes for bed split.

Select objects: Specify opposite corner: 5 found

Select objects:

Enter name of the bed to split: *C1*

Rename bed or assign key/non-key status [<Name>/Status]? *N*

Enter new name for the upper part of the bed: *C1u*

Enter new name for the lower part of the bed: *C1L*

Enter a split elevation: *2510*

Pulldown Menu Location: Drillhole > Strata/Bed Utilities

Keyboard Command: splitbed

Rename Bed

This command replaces the old bed name with a new bed name in a selected set of drillholes.

Prompts

Select the Drillholes for bed renaming.

Select objects: Specify opposite corner: 42 found

Select objects:

Enter OLD name of the bed: C1

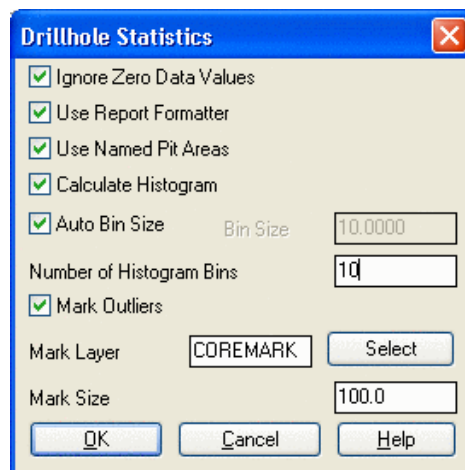
Enter NEW name for the bed: C-ONE

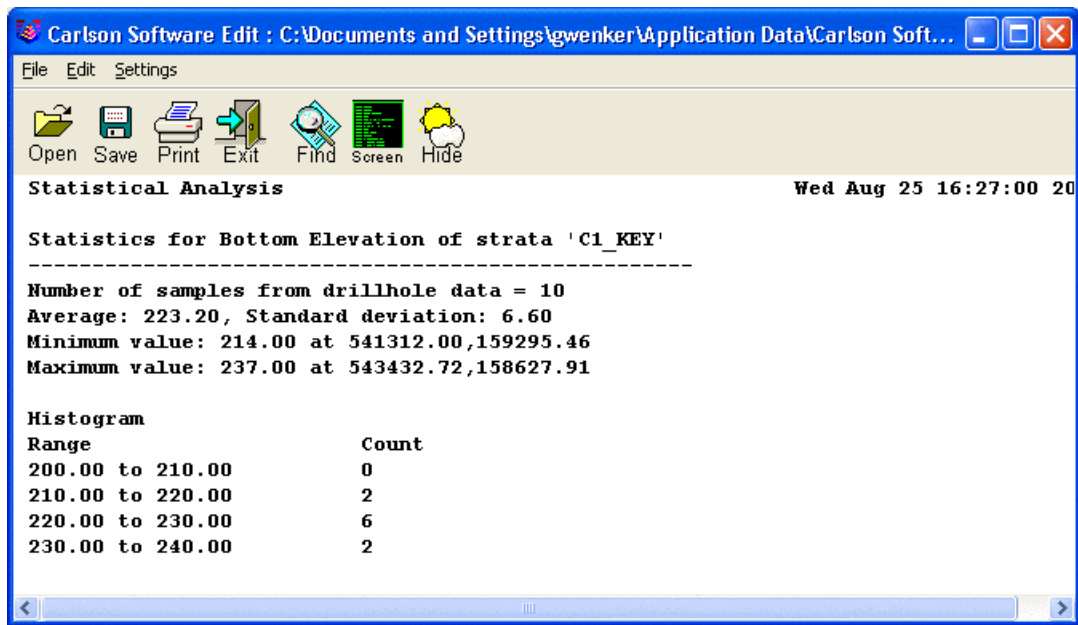
Pulldown Menu Location: Drillhole > Strata/Bed Utilities

Keyboard Command: renamedbed

Statistical Analysis

This command calculates the minimum, maximum, average, slope, and standard deviation of strata values in drillholes or from grids. These values can be the strata thickness, bottom elevation, or user-defined attributes such as BTU or GRADE. The statistics are calculated from either drillholes or a grid file. With drillholes, the statistics are calculated using the drillhole data points. With Geologic Model and grid files, the data points are the grid cells. Using a grid allows you to analyze an area by selecting inclusion and exclusion closed polylines of the area. Grid or drillholes both handle pit names for analyzing multiple areas at once. A grid file of a strata attribute can be created with the Make Strata Grid File or Autorun Strata Grids commands. The Statistical Analysis program starts with the option to use Grid, Geologic Model or Drillholes. With the Grid option, the grid file (.grd) to process can be selected individually in the standard file selection dialog. The Geologic Model option allows you to select multiple grid files from a Geologic Model File (.pre) that stores a series of grid files, as shown by highlighting multiple strata at one time. There are options for histogram and Bin size in the reports, as well as marking the outliers in the drawing for detailed review. The report can be viewed in the standard report viewer, or it can be turned on to use the report formatter.





Prompts

For drillhole statistics:

Calculate statistics from grid file or drillholes (Grid/Model/<Drillholes>)? *press Enter for drillholes*

Ignore zero values (<Yes>/No)? *press Enter*

Select the DrillHoles for report.

Select objects: *select the drillhole symbols*

Choose Strata to Process

Calculate THICKNESS statistics for strata X (<Yes>/No)? *No*

Calculate BOTTOM ELEVATION statistics for strata X (<Yes>/No)? *No*

Calculate BTU statistics for strata X (<Yes>/No)? *Yes*

Number of samples from drillhole data = 38 Ignored 34 zero values

Average: 10410.42, Standard deviation: 1371.25

Minimum value: 2305.00, Maximum value: 11147.00

Write report to file (Yes/<No>)? *Yes*

Enter the file name to write: *report.txt*

Write report to printer (Yes/<No>)? *Yes*

Make sure printer is on-line and connected to the printer port.

For grid and Geologic Model statistics:

Calculate statistics from grid file or drillholes (Grid/Model/<Drillholes>)? *Model*

Use named pit areas (Yes/<No>)? *press Enter for No*

Reading cell> 2989 Choose grid to process

Extrapolate grid to full grid size (Yes/<No>)? *press Enter Answer yes to calculate values outside the limits of the data to the full grid size.*

Select the Inclusion perimeter polylines or ENTER for none.

Select objects: *option to pick closed polylines*

Select the Exclusion perimeter polylines or ENTER for none.

Select objects: *option to pick closed polylines*

Number of samples from grid data = 612

Average: 10457.55, Standard deviation: 816.20

Minimum value: 3464.57, Maximum value: 11111.71

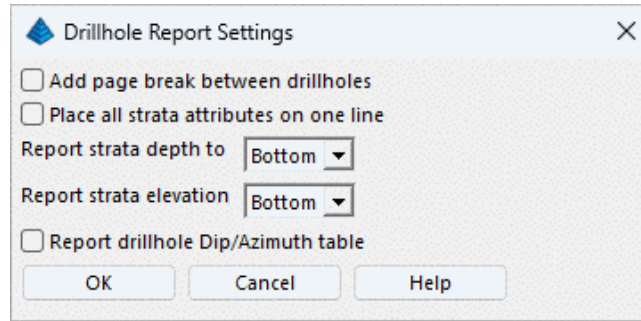
Pulldown Menu Location: StrataCalc

Keyboard Command: chstats

Prerequisite: Drillholes or grids

Standard Drillhole Report

Standard Drillhole Report retrieves data from the selected drillholes and generates a report in the Standard Report viewer that can be edited, printed and saved to file. For each drillhole, the drillhole location, description, strata names, and attributes are reported. When the command is executed, the below dialog will appear.

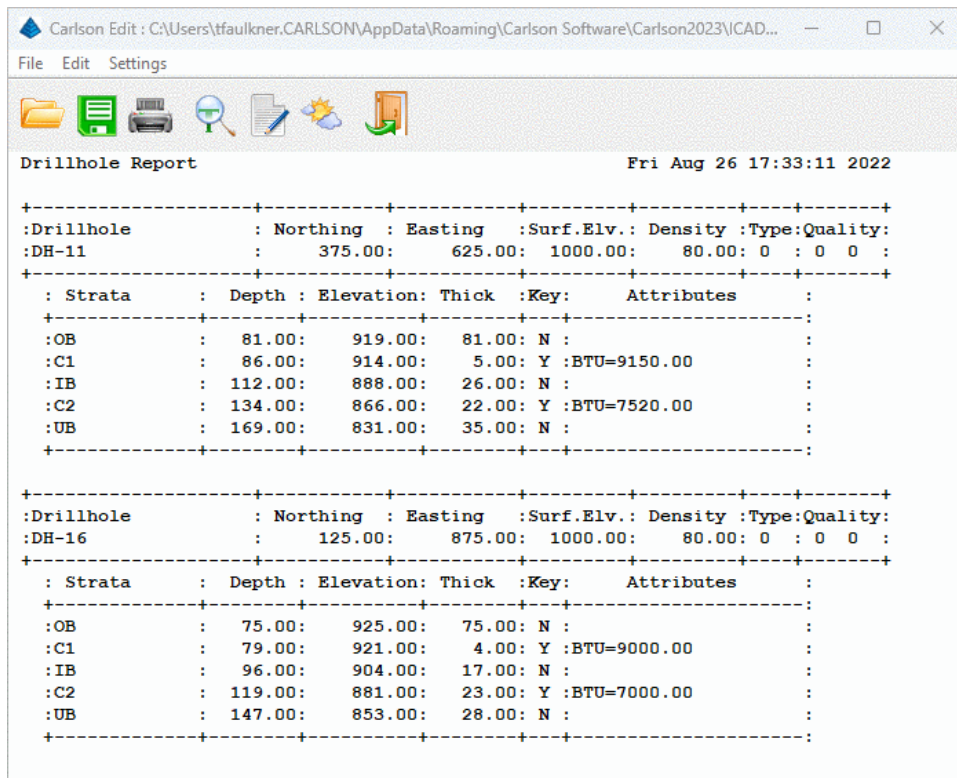


Add page break between drillholes: This option will add a significant spacer between the individual drillholes in the report.

Place all strata attributes on one row: This option will report all attributes on a single row. When disabled, attributes will appear on separate rows.

Report strata depth/elevation to: These options will control if the top or bottom depth/elevation is reported for each strata.

Report drillhole Dip/Azimuth table: This option will include the dip/azimuth table (dip and azimuth reported by varying depth) for the drillholes. Note that drillholes will not include this information by default.



Prompts

Select the DrillHoles for report.

Select objects: *select the drillhole symbols*

Add page break between drillholes [Yes/<No>]? *press Enter for No*

Place all strata attributes on one line [Yes/<No>]? *press Enter for No*

Report strata depth to [Top/<Bottom>]? *press Enter for Bottom*

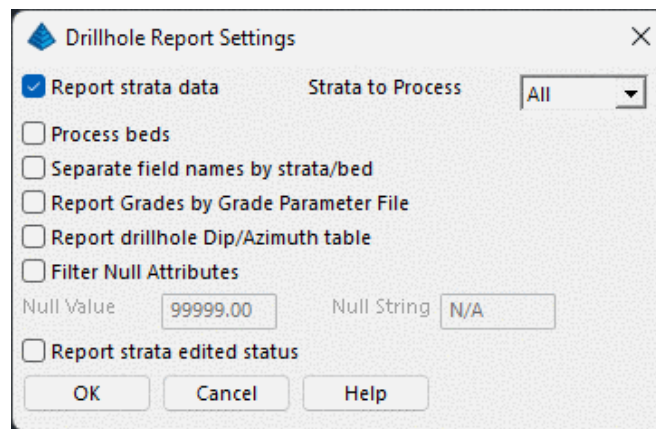
Report strata elevation of [Top/<Bottom>]? *press Enter for Bottom*

Pulldown Menu Location: Drillhole > Reports

Keyboard Command: chreport

Custom Drillhole Report

This command builds custom reports of the drillholes via the Report Formatter. The main difference between this command and Standard Drillhole Report is that this command loads all drillhole data into the Report Formatter. The Report Formatter allows you to choose which fields to report and their layout. The report data can be exported to Excel, to Access or into a comma-delimited file, or just saved or printed as shown.



Report strata data will include the strata values in the report. Otherwise only the drillhole header data will be included in the report.

Strata to Process determines if all strata or specific strata are included in the report.

Process beds will composite the strata by bed name. Otherwise the strata are reported individually.

Separate field names by strata/bed will add the strata name to each strata data field which allows you to control the report for the strata data separately by strata name. For example, if there were strata named "COAL1" and "COAL2", then you would have field names of "COAL1_THICK" and "COAL2_THICK". Otherwise, there would be only one strata thickness field named "THICK" which would report the thickness for both COAL1 and COAL2. When this option is enabled, all of the drillhole information will be reported on a single row (as opposed to using a new row for each strata layer).

Report Grades by Grade Parameter File will prompt for a GPF file to check and report the grade name for each strata.

Report drillhole Dip/Azimuth table will include the strata/dip table in the report. Note that this one of multiple options with which dip may be defined. If Dip is defined as a Strata attribute or a Drillhole attribute, this option will not impact the exporting of those attributes.

Filter Null Attributes will filter out values that match the **Null Value** or **Null String** option.

Report strata edited status will export an additional attribute that indicates if the drillhole value has been modified from its original value.

Prompts

Drillhole Report Settings dialog

Select the Drillholes for report.

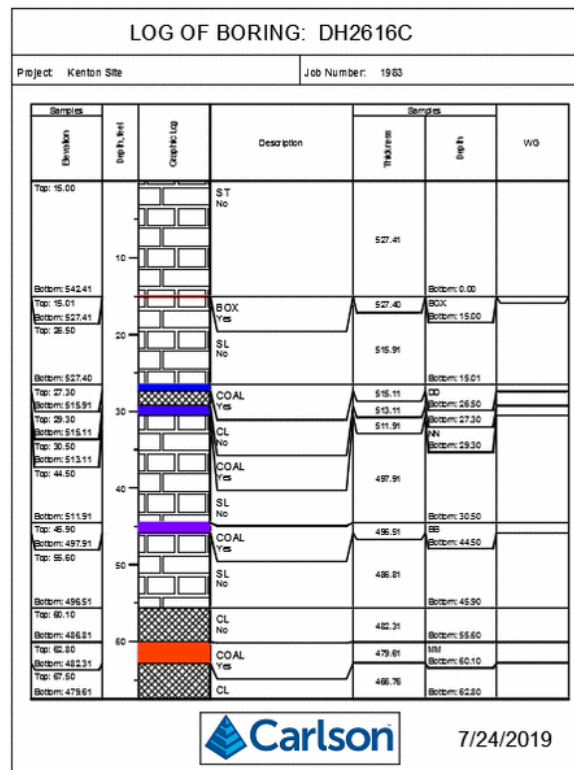
Select objects: *pick the drillholes*

Pulldown Menu Location: Drillhole > Reports

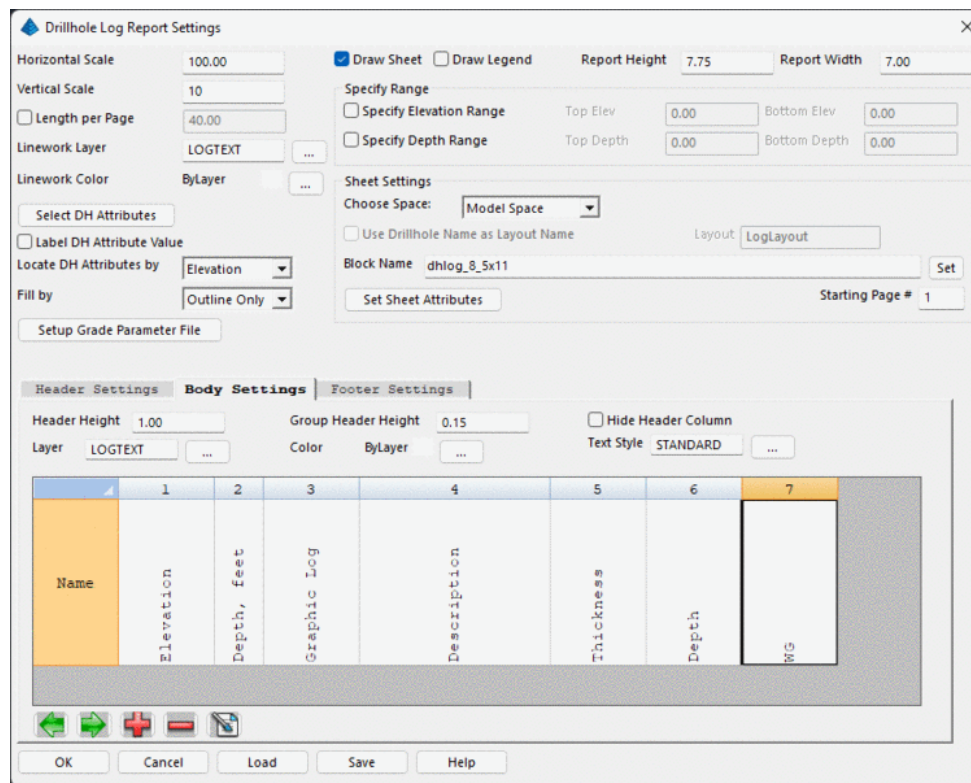
Keyboard Command: chreport2

Graphic Drillhole Report

This command creates a graphic drillhole log of the selected drillholes. It displays drillhole data such as depth, thickness, geological hatch patterns, and any attributes in the drillhole. An example report is shown below.

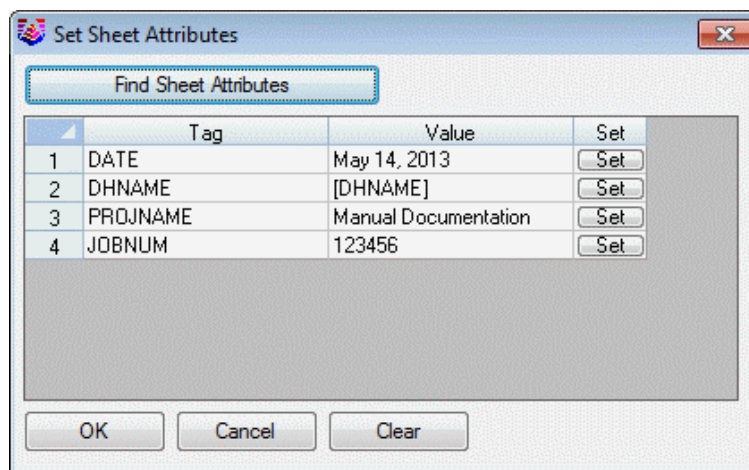


When the command is executed, you will be prompted to select the drillholes to report. This can be done by selecting drillholes on the screen, by entering the drillhole name, or by selecting the drillholes from a list. After selecting the drillhole to label, the below dialog will appear.

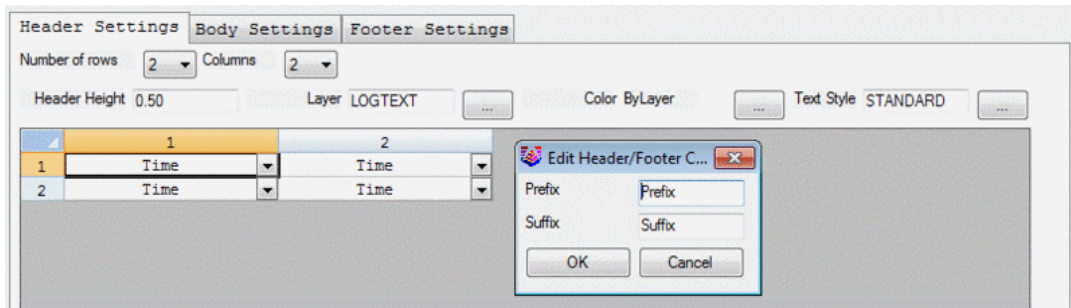


- **Horizontal Scale:** Sets the scale of the drawing. This usually matches the Horizontal Scale found in Drawing Setup, but not always.
- **Vertical Scale:** Controls the vertical exaggeration of the plot. If this value is less than the Horizontal Scale, then it will be vertically exaggerated. For example H100:V50 will be a 2:1, or 2X vertical exaggeration.
- **Length Per Page:** This option allows you to specify the length of the drillhole to display on the page. When enabled, the Vertical Scale will be automatically adjusted so that the desired hole length is displayed on the page.
- **Linework Layer:** This setting controls the layer that the linework is drawn for the boundaries, rows and columns.
- **Linework Color:** This sets the color of the linework.
- **Select DH Attributes:** The list of available drillhole attributes comes from the Define Lookup Database command from the Drillhole Attributes table. Besides defining the attribute names, the Lookup Database also defines the label size, color and layer for the attributes. The values for the attributes come from the drillholes under the Drillhole Attributes.
- **Label DH Attribute Value:** This option will add labels for the selected drillhole attributes.
- **Label DH Attributes by:** Controls whether to label the drillhole value on the Scale Bar and position the drillhole value as a depth or elevation.
- **Fill By:** This controls how to draw the strata in the graphics log. The graphic log can be filled with a hatch pattern, a solid fill, or can simply show the outline of the drill sample.
- **Setup Grade Parameter File:** This option uses a Grade Parameter File (.GPF) for the report. You can choose to color the graphic log or the data labels by the grade colors. There is also an option to label the grade name in the description field.
- **Draw Sheet:** This option draws the entire sheet for the report, including the title, project. Turning it off will turn off the Sheet Settings section, and it will just draw the main rows and columns of the log in the drawing.

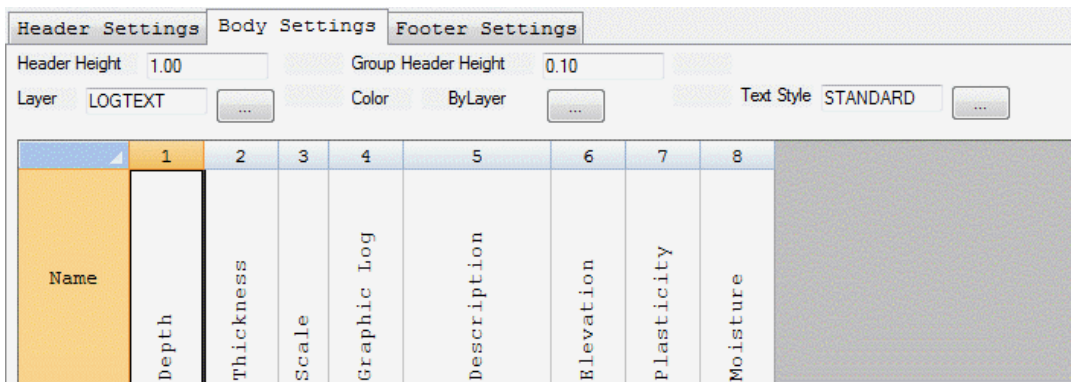
- **Draw Legend:** This option draws a legend of the strata names and hatch patterns.
- **Report Height/Width:** These parameters set the overall dimensions of the report. For use with the provided 8.5x11 sheet, these values should be set to 7.75 and 7.00, respectively.
- **Specify Elevation/Depth Range:** These options allow you to report only a portion of the drillhole, based on a range of elevations or depths. If both of these options are disabled, the entire drillhole will be reported.
- **Choose Space:** There are 3 options for drawing this report. Paper Space, Model Space and PDF. Paper Space will create a new Layout tab, using the name in the Layout Name window. Model Space will draw the plot in the working drawing, where picked. PDF will write the report out to its own PDF file.
- **Use Drillhole Name as Layout Name:** When using the Paper Space option, this option will use the drillhole name for the names of the Layout Tabs.
- **Layout Name:** When using the Paper Space option, this sets the name of the new Layout Tabs.
- **Block Name/Set:** This window sets the dwg drawing block of the sheet to use for the report. A template called dhlog_8_5x11.dwg is provided in the directory C:\Users\USERNAME\AppData\Roaming\Carlson Software\Carlson20XX\XX\SUP. This can be edited to customize how the block will appear in the report, with company logos, legends, etc.
- **Starting Page:** This sets the starting page of the reports for inclusion into existing documents.
- **Set Sheet Attributes:** This sets the topics for the header data in the report. For custom reports, set the project name and job number here. These are read from the Project and Job Number in the Drawing Setup command. They can also be overwritten here with a custom name. The Find Sheet Attributes searches the holes to populate this list of available items, such as Drillhole Attributes. To add custom drillhole attributes, an Attribute definition must be added to the sheet template. The tag of the Attribute Definition must match the tag shown in the below dialog. Once the Attribute Definition has been added, you may use this dialog to match the tag to a Drillhole Attribute. For example, you may add an Attribute Definition of "TD" to the sheet, which ties to the "Total Depth" Drillhole Attribute. To simplify this process, it is recommended to start with the provided dhlog_8_5x11.dwg file, copy and rename Attribute Definitions as desired, then saving the sheet template to a new file. Alternatively, Drillhole Attributes may be added to the report using the **Header/Footer Settings**, discussed below.



- **Header/Footer Settings:** These options prepare the Header format for the report. To add rows and columns, use the Number of rows and Columns pulldowns to set them. Then for each cell, drop them down to choose the field to display in the header. To set a Prefix or Suffix, choose the appropriate cell and choose the Cell Settings button.



- **Body Settings:** These options control how the main body of the report will appear. Rows and columns, header and group heights are set here.



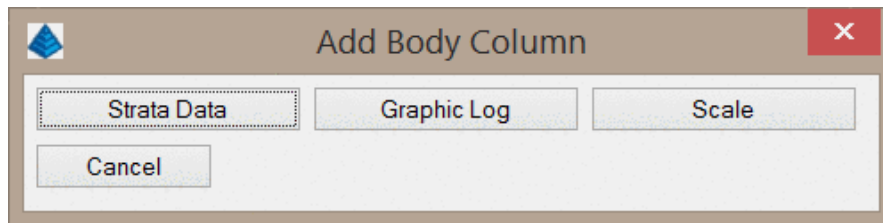
- **Header Height:** This value sets the height of the header label. For longer header names, this value should be increased to ensure the labels do not overlap the borders.
- **Group Header Height:** This values set the height of the Group Header name. Each column name can be assigned a Group Header name. When two adjacent columns share the same Group Header name, an additional label will be added to the Header name. In the below example, the Ash, BTU, and Sulfur all share a Group Header Name of "Quality".

Description	Depth	Quality		
		ASH	BTU	Sulfur

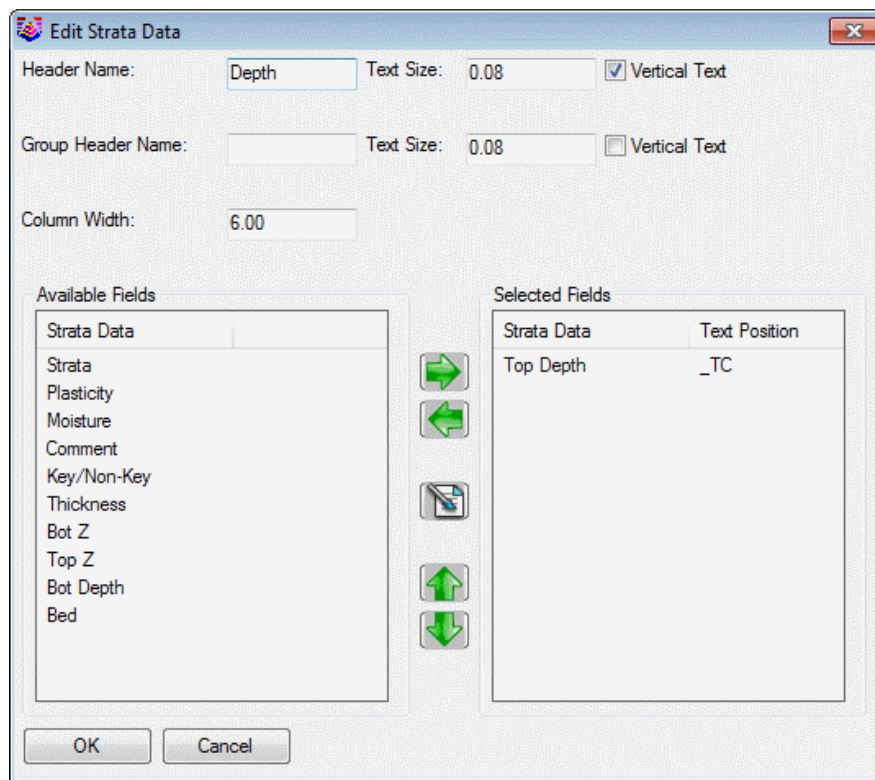
- **Hide Header Column:** When enabled, the gridlines for the column headers will not be drawn.
- **Layer:** This sets the layer for the report.
- **Color:** This sets the color of the report.
- **Text Style:** This sets the text style for the report.
- **Icons:** The arrow/plus/minus icons can be used to reorganize the order of the columns, or to add and remove columns. The pen-and-paper icon allows you to edit column properties.



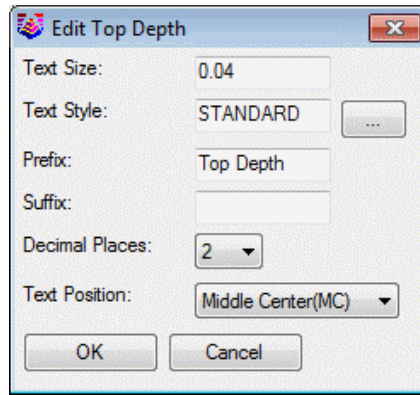
When a new column is added, the below dialog will appear. This allows you to add either Strata Date (thicknesses, qualities, etc.), a graphic log (visual representation of the drillhole), or a scale (to show elevation or depth).



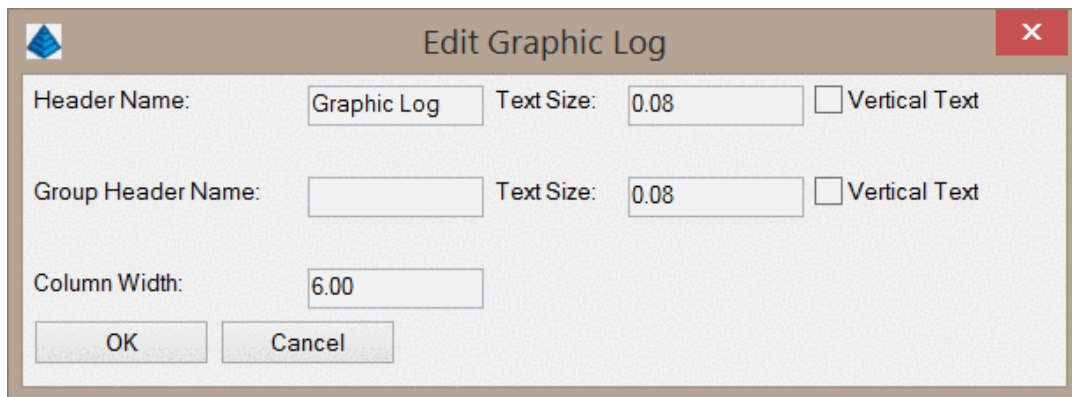
When Strata Data is selected, the below dialog will appear. Here you can add Header Names, Group Header Names, and the Fields to be included in the report. To add fields, move items from the available fields on the left to the right, under selected fields. The green arrows move them in all four directions. The middle, Edit button allows for editing the Selected Field. This is where the text size for the data in the columns is set, as well as the centering in the fields.



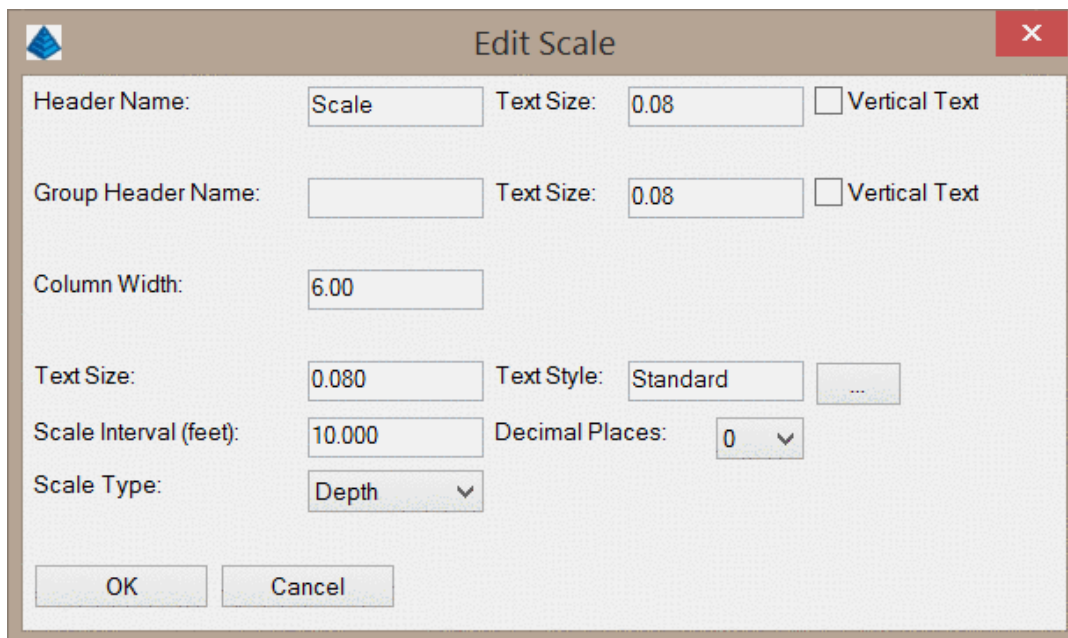
The middle Edit button allows for editing the Selected Field. This is where the text size for the data in the columns is set, along with the positioning, decimal places, etc.



When a Graphic Scale is added, the below dialog will appear. This is similar to adding Strata Data, but fewer settings are available.

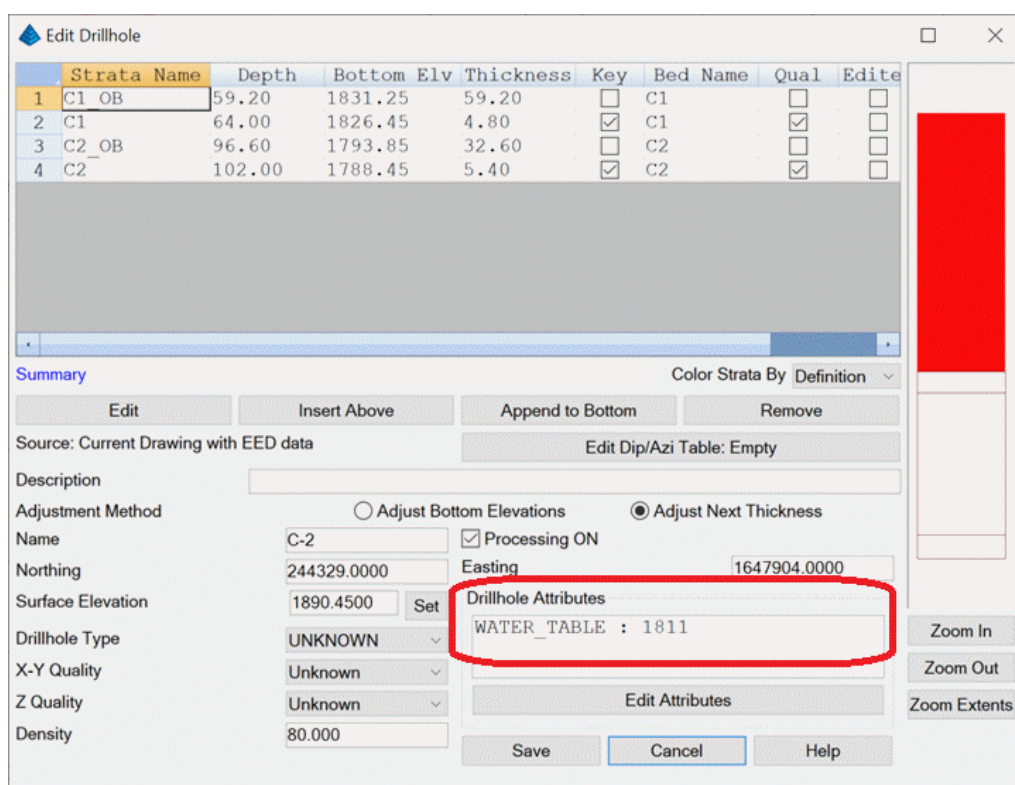


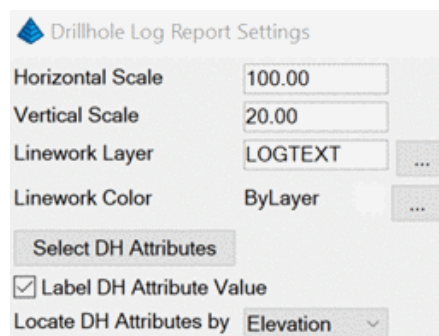
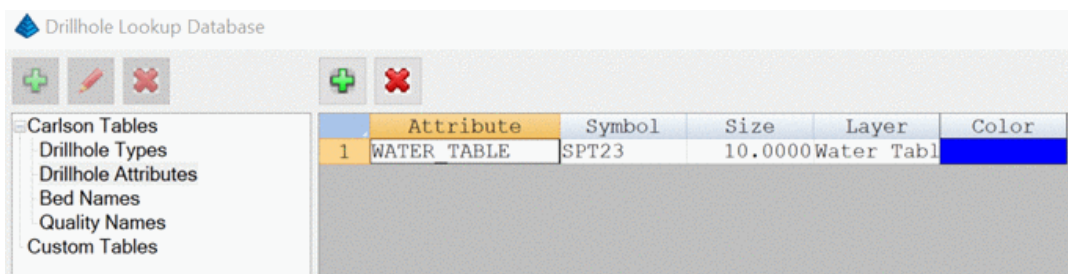
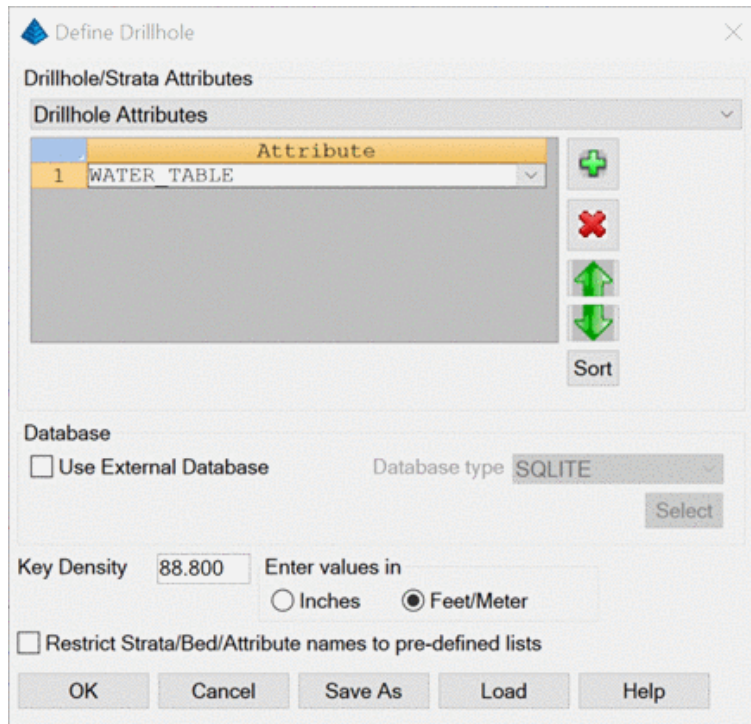
When a Scale is added, the below dialog will appear. Here you can specify the interval of the scale, the scale type, etc.



After all of the settings are defined, hitting OK will prompt for the location to draw the report. Based on the Choose Space setting, it will either go to Model Space, Paper Space Layout, or a PDF file.

To label the water table, the drillholes need an attribute named WATER_TABLE. Define Drillhole needs a drillhole attribute named WATER_TABLE and Define Lookup Database needs a drillhole attribute named WATER_TABLE with the symbol, size, color and layer to use for the report. Then in the dialog for Graphic Drillhole Report, pick on Select DH Attributes and select WATER_TABLE and turn on Label DH Attribute Value and Locate DH Attributes By.





Prompts

Select drillholes to report

Select objects: *pick the drillholes*

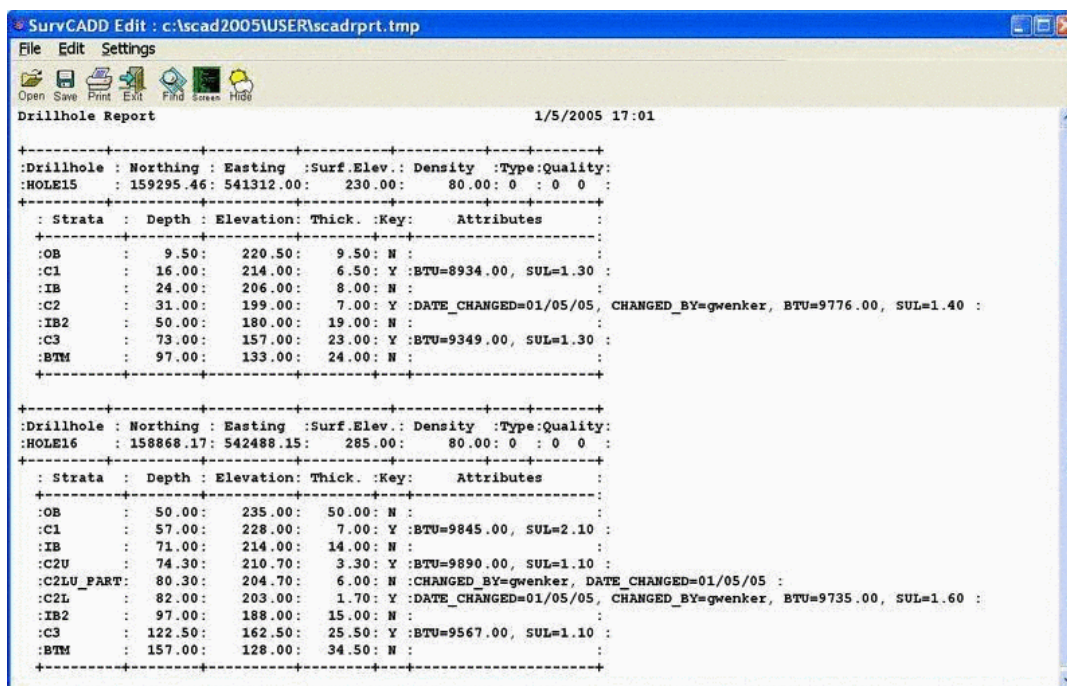
Pulldown Menu Location: Drillhole > Reports

Keyboard Command: chreport5

Prerequisite: Drillholes

Change History Report

This command creates a report of drillholes that have been modified during the specified time period. The program reads the change history from strata attribute variables called DATE_CHANGED and CHANGED_BY. The Edit Drillhole and Drillhole Data Sheet routines will update these changed attributes using the current date and the AutoCAD login name when any of the strata data is modified. For the report if a drillhole contains a changed strata, there is an option to either print all the strata for that drillhole or just the modified strata. To setup the DATE_CHANGED and CHANGED_BY attribute, run the Define Drillhole command and add these names to the Key strata attributes and Non-Key strata attributes.



SurvCADD Edit : c:\scad2005\USER\scadrprt.tmp

File Edit Settings

Open Save Print Exit Find Erase Help

Drillhole Report 1/5/2005 17:01

Drillhole	Northing	Easting	Surf. Elev.	Density	Type	Quality
HOLE15	159295.46	541312.00	230.00	80.00	0	0 0

Strata	Depth	Elevation	Thick.	Key	Attributes
:OB	9.50	220.50	9.50	N	
:C1	16.00	214.00	6.50	Y	:BTU=8934.00, SUL=1.30
:IB	24.00	206.00	8.00	N	
:C2	31.00	199.00	7.00	Y	:DATE_CHANGED=01/05/05, CHANGED_BY=gwenker, BTU=9776.00, SUL=1.40
:IB2	50.00	180.00	19.00	N	
:C3	73.00	157.00	23.00	Y	:BTU=9349.00, SUL=1.30
:BTM	97.00	133.00	24.00	N	

Drillhole	Northing	Easting	Surf. Elev.	Density	Type	Quality
HOLE16	158868.17	542488.15	285.00	80.00	0	0 0

Strata	Depth	Elevation	Thick.	Key	Attributes
:OB	50.00	235.00	50.00	N	
:C1	57.00	228.00	7.00	Y	:BTU=9845.00, SUL=2.10
:IB	71.00	214.00	14.00	N	
:C2U	74.30	210.70	3.30	Y	:BTU=9890.00, SUL=1.10
:C2LU_PART	80.30	204.70	6.00	N	:CHANGED_BY=gwenker, DATE_CHANGED=01/05/05
:C2L	82.00	203.00	1.70	Y	:DATE_CHANGED=01/05/05, CHANGED_BY=gwenker, BTU=9735.00, SUL=1.60
:IB2	97.00	188.00	15.00	N	
:C3	122.50	162.50	25.50	Y	:BTU=9567.00, SUL=1.10
:BTM	157.00	128.00	34.50	N	

Prompts

Select the DrillHoles for change history report.

Select objects: *pick the drillholes*

Enter Period Start Date or Enter for None (mm/dd/yy): 1/1/95

Enter Period End Date or Enter for None (mm/dd/yy): 9/9/06

Include only changed strata in affected drillholes (Yes/<No>)? Y

Pulldown Menu Location: Drillhole > Reports

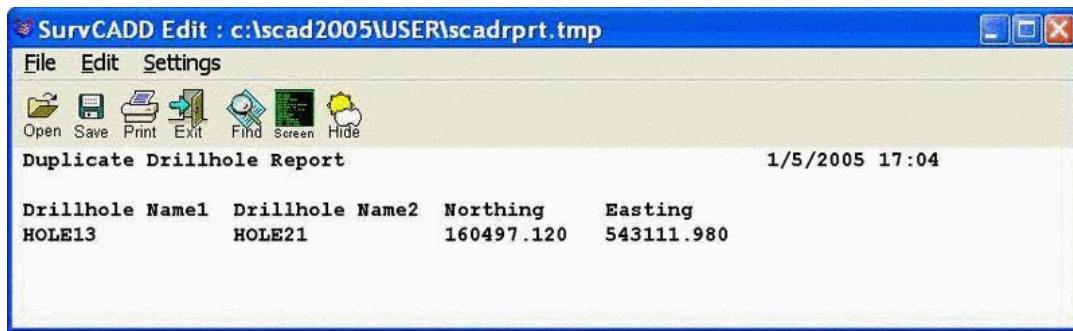
Keyboard Command: chhreport

Prerequisite: Drillholes

Duplicate Drillhole Report

This command reports drillholes located at the same easting/northing position. The holes can have different names. The report dialog displays the names, and locations of the problem holes. The command has an option to draw a circle, of specified size, around the holes that are found duplicate. The purpose of this routine is to help clean up the drillhole database by identifying duplicate position drillholes since the modeling can only process one drillhole at any one location. This routine only shows the duplicates and it is up to you to actually decide which duplicate

drillholes to remove or move. It does not report two drillholes that have the same name, but are at different locations.



Prompts

Draw circle around duplicate drillholes [Yes/<No>]? Y

Circle radius <100.0>: press Enter for 100

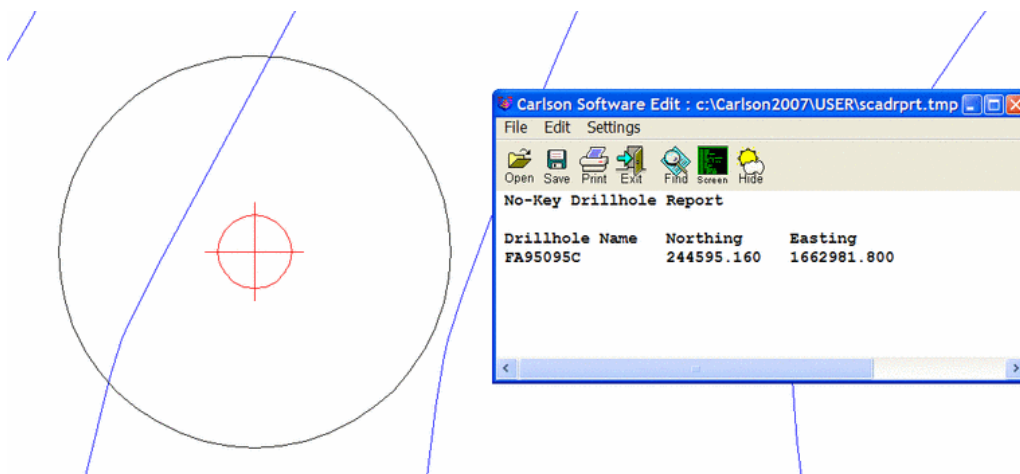
Pulldown Menu Location: Drillhole > Reports

Keyboard Command: dupch

Prerequisite: Drillholes

Drillholes Without Key Strata

This routine will report all selected drillholes that do not contain any Key strata. It also will give the option to draw a circle around those drillholes.



Prompts

Draw circle around no-key drillholes [Yes/<No>]? y

Circle radius <100.0>: 50

Drop-Down Menu Location: Drillhole

Keyboard Command: nokeych

Key Strata Missing Bed

This command reports any drillholes that have key strata with a blank bed name. There is an option to draw a circle around drillholes with missing bed names as a way to highlight them in the drawing.

Pulldown Menu Location: Drillhole > Reports

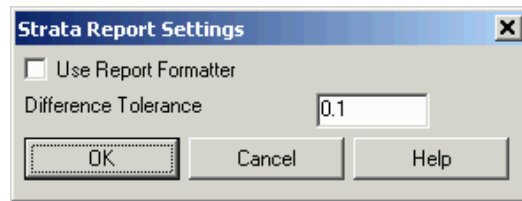
Keyboard Command: missbed

Prerequisite: Drillholes in drawing

Compare Strata To Model

This command compares drillhole data to grid models. Drillhole values that differ from the grid model by more than the specified Difference Tolerance are reported. The program can create a standard format report or the Use Report Formatter option can be used for custom reports or outputs to different formats like Excel. The purpose of this command is to check that the grid models match the source drillhole data.

After selecting the drillholes to process, you choose which strata process and then which value to process including thickness, elevation or attribute. Then select the grid model to compare with and the program generates the report.



Prompts

Strata Report Settings dialog

Select drillholes, channel samples and strata polylines.

Select objects: *pick the drillholes*

Compare Strata to Model

Model: c1_key.grd

Strata: C1_KEY

Attribute: Elevation

Drillhole Name Northing Easting Elevation Model Difference

C-3 244124.26 1647252.00 1726.12 1729.23 3.11

C-4 243829.00 1647651.00 1679.12 1679.34 0.22

C-2 244329.00 1647904.00 1826.45 1826.21 -0.24

12 drillholes within tolerance of 0.10

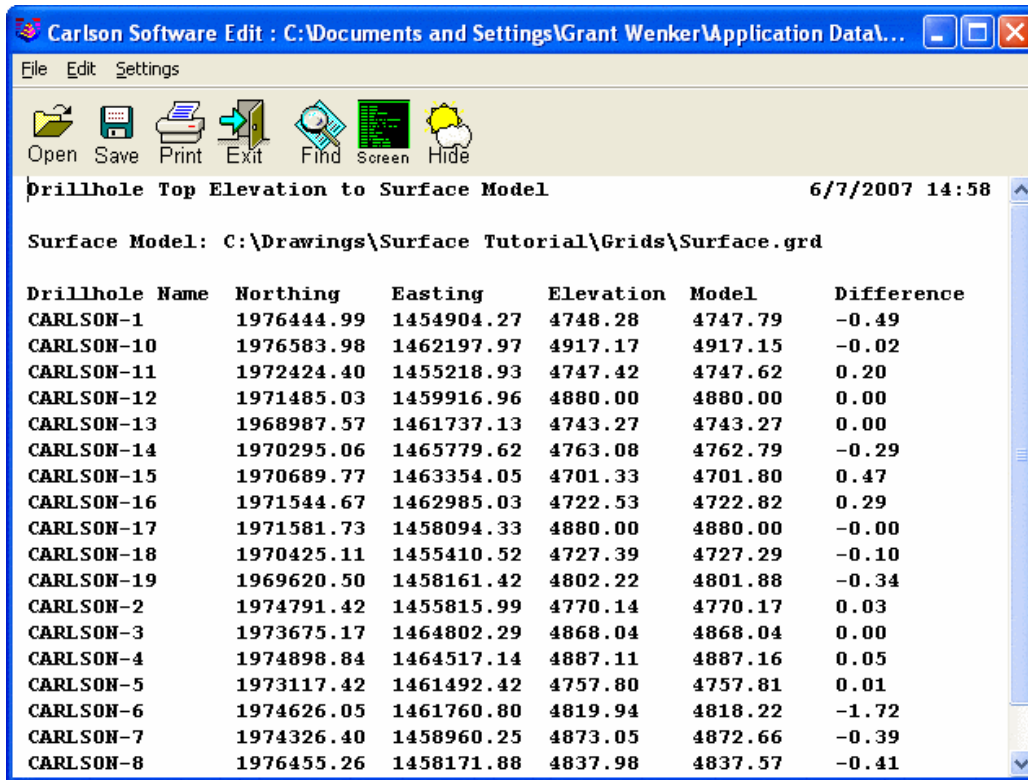
Pulldown Menu Location: Drillhole > Reports

Keyboard Command: chcutsheet

Prerequisite: Grid file and drillholes

Drillhole Top to Surface Model

This command reports the difference between the selected drillholes and a selected surface model. There is a tolerance that will not include any drillhole within this vertical distance from the surface model.



The screenshot shows a software window titled 'Carlson Software Edit : C:\Documents and Settings\Grant Wenker\Application Data\...'. The window contains a menu bar (File, Edit, Settings) and a toolbar with icons for Open, Save, Print, Exit, Find, Screen, and Hide. Below the toolbar, the window title is 'Drillhole Top Elevation to Surface Model' with a timestamp '6/7/2007 14:58'. The main content area displays a table with the following data:

Drillhole Name	Northing	Easting	Elevation	Model	Difference
CARLSON-1	1976444.99	1454904.27	4748.28	4747.79	-0.49
CARLSON-10	1976583.98	1462197.97	4917.17	4917.15	-0.02
CARLSON-11	1972424.40	1455218.93	4747.42	4747.62	0.20
CARLSON-12	1971485.03	1459916.96	4880.00	4880.00	0.00
CARLSON-13	1968987.57	1461737.13	4743.27	4743.27	0.00
CARLSON-14	1970295.06	1465779.62	4763.08	4762.79	-0.29
CARLSON-15	1970689.77	1463354.05	4701.33	4701.80	0.47
CARLSON-16	1971544.67	1462985.03	4722.53	4722.82	0.29
CARLSON-17	1971581.73	1458094.33	4880.00	4880.00	-0.00
CARLSON-18	1970425.11	1455410.52	4727.39	4727.29	-0.10
CARLSON-19	1969620.50	1458161.42	4802.22	4801.88	-0.34
CARLSON-2	1974791.42	1455815.99	4770.14	4770.17	0.03
CARLSON-3	1973675.17	1464802.29	4868.04	4868.04	0.00
CARLSON-4	1974898.84	1464517.14	4887.11	4887.16	0.05
CARLSON-5	1973117.42	1461492.42	4757.80	4757.81	0.01
CARLSON-6	1974626.05	1461760.80	4819.94	4818.22	-1.72
CARLSON-7	1974326.40	1458960.25	4873.05	4872.66	-0.39
CARLSON-8	1976455.26	1458171.88	4837.98	4837.57	-0.41

Prompts

Select the Drillholes for report.

Select objects: all *pick the drillholes*

Elevation Difference Tolerance <0.0>:

Use Report Formatter [Yes/<No>]? *No will use the standard report window*

Reading cell> 49410 Select the drillholes and strata polylines.

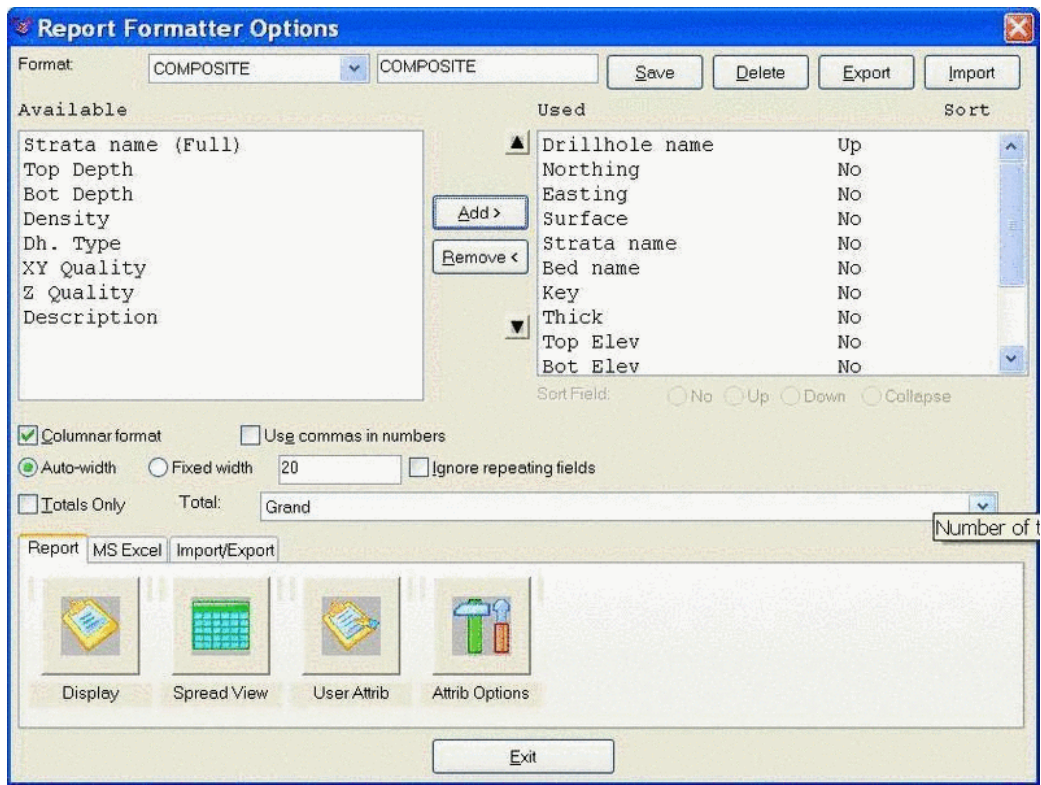
Pulldown Menu Location: Drillhole > Reports

Keyboard Command: chreport4

Prerequisite: Drillholes

Bed Composite Report

This command reports the individual strata qualities and the resulting composite qualities of drillholes. The strata are composited by bed name. It is useful for seams that are sampled extensively at many intervals, and even with different strata names. An example hole report is shown. Each individual strata and bed is broken out first, with the composite values following at the end of the report. The command uses the report formatter, so the output it gives is very customizable.



Edit : c:\scad2005\USER\scadrprt.tmp

File Edit Settings

Open Save Print Exit Find Screen

Composite Report

Drillhole name	Northing	Easting	Surface	Strata name	Bed name	Key	Thick	Top Elev	Bot Elev	RAW	ASH	RAW	SULFUR	RAW	BTU
5889C	417086.00	1771819.00	1410.00	USR_OB	U	NO	98.50	1410.00	1311.50						
5889C	417086.00	1771819.00	1410.00	USR	U	YES	1.18	1311.50	1310.32	9.530			4.470	13593.000	
5889C	417086.00	1771819.00	1410.00	US2_OB	U	NO	13.32	1310.32	1297.00						
5889C	417086.00	1771819.00	1410.00	US2	U	YES	0.80	1297.00	1296.20	13.860			1.520	12927.000	
5889C	417086.00	1771819.00	1410.00	US1_OB	U	NO	2.86	1296.20	1293.34						
5889C	417086.00	1771819.00	1410.00	US1	U	YES	1.85	1293.34	1291.49	8.230			0.720	13765.000	
5889C	417086.00	1771819.00	1410.00	US_OB	U	NO	2.29	1291.49	1289.20						
5889C	417086.00	1771819.00	1410.00	US	U	YES	0.10	1289.20	1289.10						
5889C	417086.00	1771819.00	1410.00	MSUU_OB	M	NO	15.10	1289.10	1274.00						
5889C	417086.00	1771819.00	1410.00	MSUU	M	YES	2.40	1274.00	1271.60	13.650			0.980	12833.000	
5889C	417086.00	1771819.00	1410.00	MSUL_OB	M	NO	1.71	1271.60	1269.89						
5889C	417086.00	1771819.00	1410.00	MSUL	M	YES	1.22	1269.89	1268.67	6.730			0.920	13884.000	
5889C	417086.00	1771819.00	1410.00	CB5_OB	C	NO	75.17	1268.67	1193.50						
5889C	417086.00	1771819.00	1410.00	CB5	C	YES	0.75	1193.50	1192.75	15.390			0.850	12609.000	
5889C	417086.00	1771819.00	1410.00	CB4_OB	C	NO	1.11	1192.75	1191.64						
5889C	417086.00	1771819.00	1410.00	CB4	C	YES	2.15	1191.64	1189.49	17.680			1.010	13218.000	
5889C	417086.00	1771819.00	1410.00	CB3_OB	C	NO	0.39	1189.49	1189.10						
5889C	417086.00	1771819.00	1410.00	CB3	C	YES	1.40	1189.10	1187.70	12.990			0.640	13144.000	
-----COMPOSITE-----															
5889C	417086.00	1771819.00	1410.00	U_TOP	U	NO	98.50	1410.00	1311.50						
5889C	417086.00	1771819.00	1410.00	U_PARTING	U	NO	18.47	1311.50	1293.03						
5889C	417086.00	1771819.00	1410.00	U_KEY	U	YES	3.93	1293.03	1289.10	9.807			2.042	13536.969	
5889C	417086.00	1771819.00	1410.00	M_TOP	M	NO	15.10	1289.10	1274.00						
5889C	417086.00	1771819.00	1410.00	M_PARTING	M	NO	1.71	1274.00	1272.29						
5889C	417086.00	1771819.00	1410.00	M_KEY	M	YES	3.62	1272.29	1268.67	11.318			0.960	13187.204	
5889C	417086.00	1771819.00	1410.00	C_TOP	C	NO	75.17	1268.67	1193.50						
5889C	417086.00	1771819.00	1410.00	C_PARTING	C	NO	1.50	1193.50	1192.00						
5889C	417086.00	1771819.00	1410.00	C_KEY	C	YES	4.30	1192.00	1187.70	15.754			0.862	13087.686	

Prompts

Select the Drillholes for report.

Select objects: *select the drillholes to process*

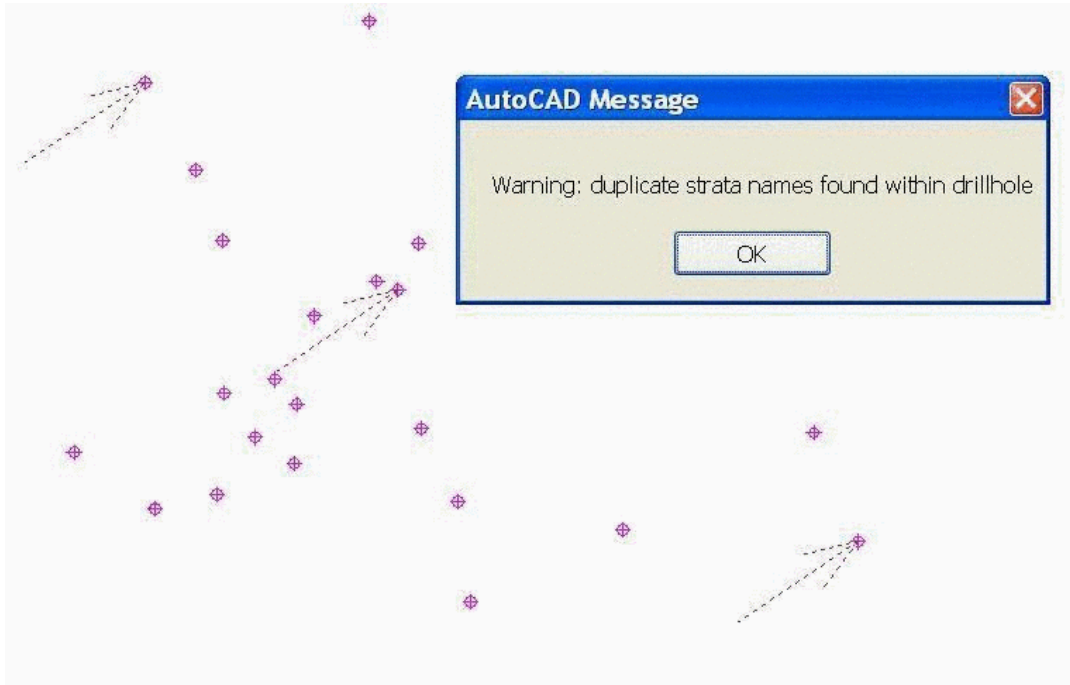
Pulldown Menu Location: Drillhole > Reports

Keyboard Command: chreport3

Prerequisite: Drillholes with quality data

Invalid Strata Report

Routines that process strata such as Fence Diagram or Make Strata Grid File require strata names to be in correct order and these routines refuse to continue if there is a problem with the strata. This command creates a report of drillholes that have strata that are out of order or have duplicate strata names. An arrow is drawn on the screen pointing to the drillholes that have the problem. If the screen is moved or zoomed, the arrows will disappear. This report can be used to identify the drillholes that need to be cleaned up.



SurvCADD Edit : C:\DRAWINGS\SURFACE TUTORIAL\GRIDS\sap_drillholes-Inva... 1/6/2005 09:02

File Edit Settings

Open Save Print Exit Find Screen Hide

Invalid Strata Report

Strata	Drillhole	Northing	Easting	Problem
MK	5989C	420322.000	1770461.000	Out of Order with MK_OB at HL241
MK	5989C	420322.000	1770461.000	Duplicate strata name
MK	5989C	420322.000	1770461.000	Duplicate strata name
USR	2289C	418600.000	1773613.000	Duplicate strata name
USR	2289C	418600.000	1773613.000	Duplicate strata name
CHL	HL90	421743.250	1768726.000	Duplicate strata name
CHL	HL90	421743.250	1768726.000	Duplicate strata name

Prompts

Select the drillholes and strata polylines.

Select objects: *pick the drillholes*

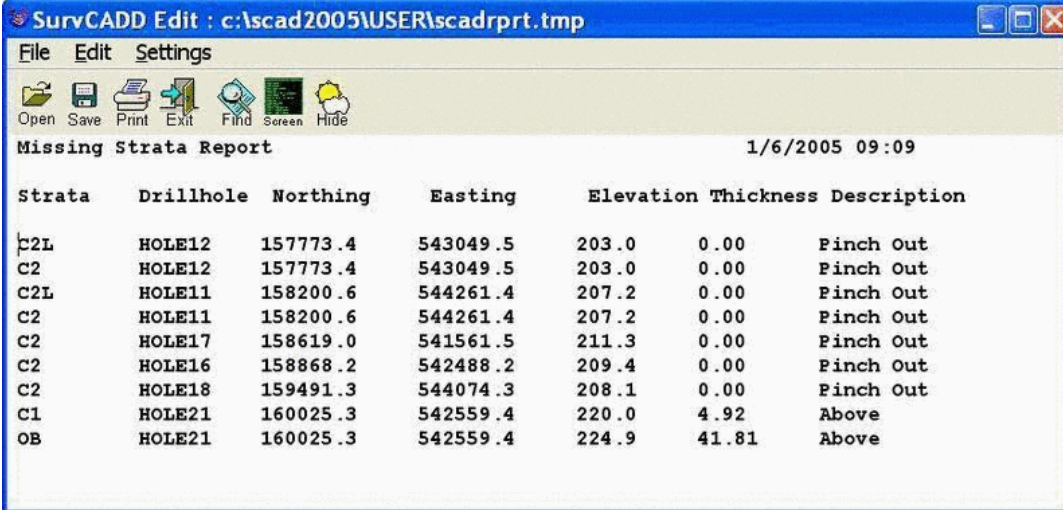
Pulldown Menu Location: Drillhole > Reports

Keyboard Command: badch

Prerequisite: Drillholes

Missing Strata Report

This command reports strata from the selected drillholes that are missing in the strata sequence. The strata sequence is determined by reading the strata from all the selected drillholes. When a strata is missing between other strata, this is reported as PINCH OUT. For example with strata A,B,C and a drillhole with A,C, then strata B will be reported as pinch out. If a strata is missing before the first strata in the drillhole, then this strata is reported as ABOVE. For example a drillhole with B,C would report strata A as above. Likewise a strata missing after the last strata in the drillhole is reported as BELOW. For example a drillhole with A,B would report strata C as below. In the report, the estimated bottom strata elevation and thickness for the missing strata is reported based on pinch out and conformance.



Strata	Drillhole	Northing	Easting	Elevation	Thickness	Description
C2L	HOLE12	157773.4	543049.5	203.0	0.00	Pinch Out
C2	HOLE12	157773.4	543049.5	203.0	0.00	Pinch Out
C2L	HOLE11	158200.6	544261.4	207.2	0.00	Pinch Out
C2	HOLE11	158200.6	544261.4	207.2	0.00	Pinch Out
C2	HOLE17	158619.0	541561.5	211.3	0.00	Pinch Out
C2	HOLE16	158868.2	542488.2	209.4	0.00	Pinch Out
C2	HOLE18	159491.3	544074.3	208.1	0.00	Pinch Out
C1	HOLE21	160025.3	542559.4	220.0	4.92	Above
OB	HOLE21	160025.3	542559.4	224.9	41.81	Above

Prompts

Report pinch out strata [**<Yes>/No**]? *Y*
Report strata above or below drillhole [**<Yes>/No**]? *Y*
Select drillholes, channel samples and strata polylines.
Select objects: Specify opposite corner: 13 found
Select objects:
Reading drillhole 13
Finding splits ...
Finding pinch out ...
Calculating seam stacking ...
Drillhole report
Pulldown Menu Location: Drillhole > Reports
Keyboard Command: missch
Related Commands: Place Drillholes

Strata Order and Count Report

This command reports all the strata existing in the selected drillholes, in order from top to bottom and the number of drillholes that contain each strata.

Strata	Count
UKR_OB	9
UKR	9
UK_OB	25
UK	25
MK_OB	72
MK	72
N5BU_OB	100
N5BU	100
N5BL_OB	116
N5BL	118
LT5B_OB	8
LT5B	8
USR_OB	224
USR	224
US3_OB	40
US3	40
US2_OB	188
US2	190
US1_OB	208
US1	208

Prompts

Select drillholes, channel samples and strata elevation polylines.

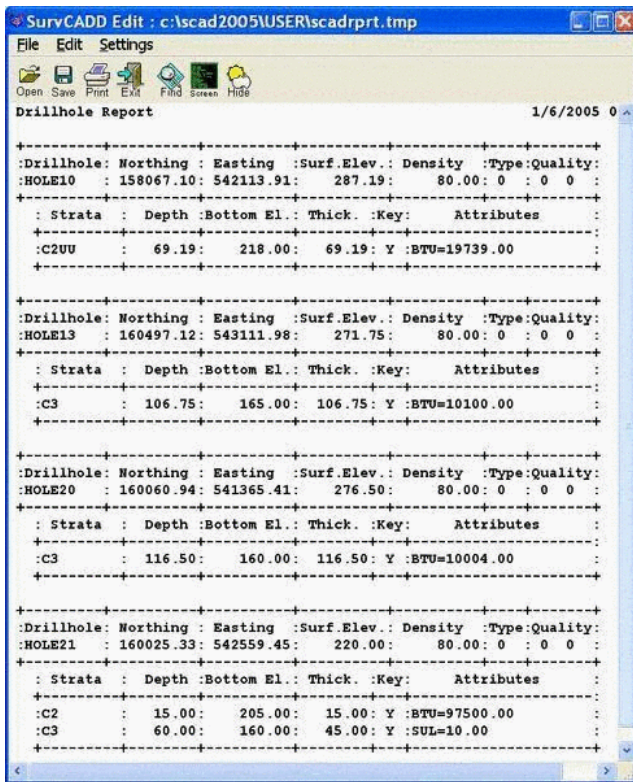
Select objects: *Select drillholes from the drawing*

Pulldown Menu Location: Drillhole > Reports

Keyboard Command: geo_order2

Attribute Validation Report

This command reads the attribute information from the selected drillholes and checks strata attribute values against the minimum and maximum values specified in the attribute definitions. The attribute definitions are created in the Define Attributes command. Any that fall outside the defined range are added to the report.



Attribute Validation report

Prompts

Select the Drillholes for validation report.

Select objects: Specify opposite corner: 13 found

Select objects:

Place all strata attributes on one line [Yes/<No>]? Y

Report strata depth to [Top/<Bottom>]?

Report strata elevation of [Top/<Bottom>]?

Pulldown Menu Location: Drillhole > Reports

Keyboard Command: checkattr

Prerequisite: Drillholes

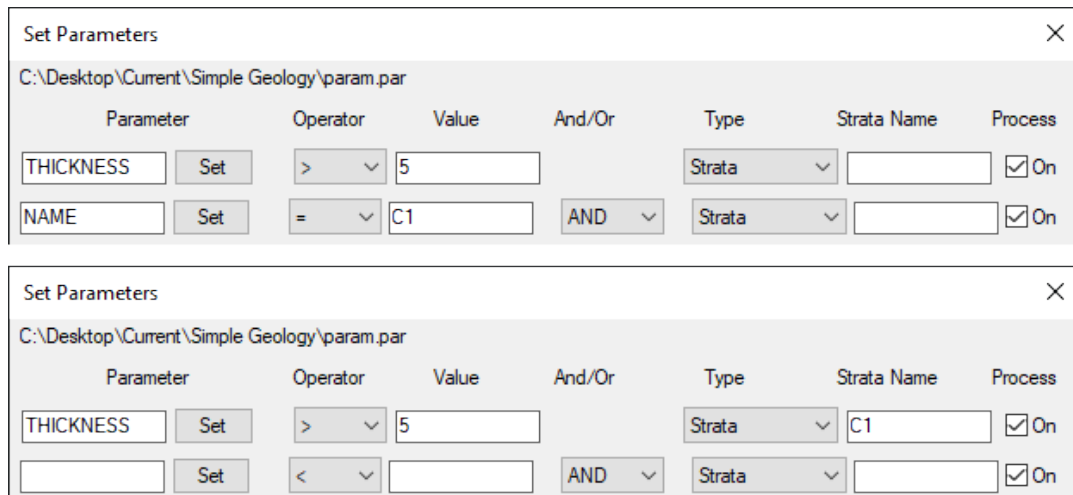
Define Parameters

This command prepares an attribute parameter definition file (.par file) that defines the attribute names along with an operator and value. The parameter definition files are filters that are applied in the Compliance Drillhole Report, Compliance Areas and Mark Compliance Drillhole commands, allowing you to report, select, and modify drillholes that match the criteria. When the command is executed, the below dialog will appear.

Notes on specific attributes are provided below. It is important to note that you may use asterisks (*) as wildcard characters. This command is case-sensitive.

Strata Parameters

- **Thickness:** Thickness of the sample in the drillhole
- **Depth:** Bottom depth of the sample in the drillhole
- **Top-Elev:** Top elevation of the sample in the drillhole
- **Bottom-Elev:** Bottom elevation of the sample in the drillhole
- **Name:** The name of the strata or the bed group. It is important to note that this can be used as an alternative to the Strata Name column of the main dialog. For example, the below configurations of tests will yield the same result. It is important to note that if you use this Name parameter rather than using the Strata Name column, the program will prompt you at the command line if the parameter tests must pass for all strata or if passing for a single strata will suffice.

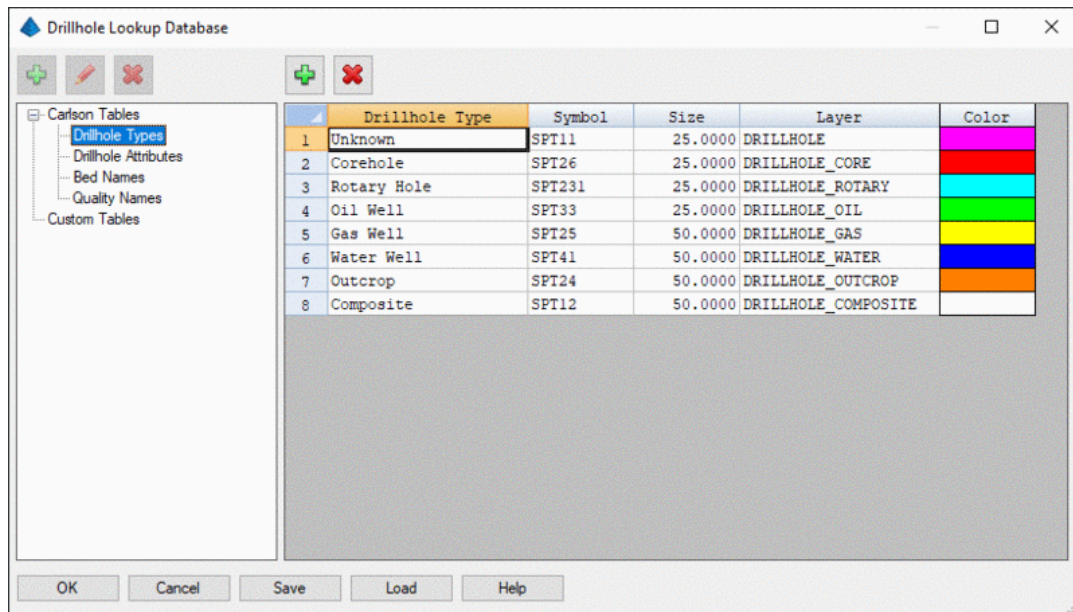


- **Bed:** The bed name of the strata. This will allow you to restrict the test to a strata name that might repeat to a specific bed. For example, if you have many samples with a strata name of "Coal" distributed across three different beds, you could add this BED parameter to restrict the tests to only check Bed "A".
- **Key:** The Key/Nonkey status of the strata. A value of 1 relates to a key strata, whereas a value of 0 relates to a nonkey strata.
- **Contain:** This is used to check for existence of a strata name in a drillhole. This allows you to check for the existence of a strata name. Note that this check is very similar to using a NAME parameter check.

Drillhole Parameters

- **Name:** The name of the drillhole. You may use asterisks (*) as wildcard characters to select all drillholes with a similar prefix/suffix

- **Elev:** The surface elevation of the drillhole.
- **Type:** The integer value relating to the type of the drillhole as defined in the Drillhole Lookup Database command. You should not refer to the drillhole type by name. For example, if you have defined drillhole Type 2 as a "Corehole", you should check for a value of "2" rather than the name "Corehole".



- **Desc:** The text description of the drillhole.
- **XY_Qual** and **Z_Qual:** The integer value relating to the XY or Z Quality (method of measurement of the drillhole coordinates) as defined in the Drillhole Lookup Database command. You should not refer to the Quality by name. For example, if you have defined the Quality of 3 as "GPS", you should check for a value of "3" rather than the name "GPS".

The Strata Name column is optional for assigning a strata name to a parameter. You only need to fill out the Strata Name column if you are analyzing multiple strata at once. For example, you could check THICKNESS on COAL2 strata name and THICKNESS with a different value on COAL3 strata name. If all the parameters apply to the same strata then when you run routines like Mark Compliance Drillholes, there is an option to test the entire drillhole or a specific strata. You can choose the specific strata option and then select a strata to process. Note: You do not fill in the strata name for the first or top parameter. The program will prompt you for the strata to process.

The And/Or field applies between the parameters in the current row and the row above. The Process column allows you to turn off processing for a parameter.

Pulldown Menu Location: Drillhole > Parameter Compliance

Keyboard Command: pardef

Compliance Drillhole Report

This command reports drillholes from the selected drillholes that meet all the qualities defined in a parameter file. Drillholes that don't meet the parameter file are not reported. There are three main prompts you may receive after

defining the parameters.

1) Process beds [Yes/<No>]?

This option determines if the program will read the strata names exactly as they exist in the drillhole, or if they will be regrouped by the bed names. This is very important if you are trying to test a parameter on a single strata sample, or a composite bed group. For example, suppose you have multiple thin samples of strata in a single bed, and you want to check for drillholes where the total bed thickness is greater than 5 ft. If you do not enter a YES for the Process Beds option, the program would only analyze the thickness of each individual strata in the bed, as opposed to checking the full bed thickness.

It is also important to note that when answering YES to this prompt, the strata names will not be considered, but rather, the regrouped bed name. For example if the strata name is "COAL", the Bed Name is "A", and the Key status is KEY, then the regrouped name would be "A_KEY". In this scenario, checking for the existence of COAL when you enter Yes to this prompt would yield no results. For more information on the regrouped names, see the Bed Name Suffixes section of Carlson Configure > Mining Settings.

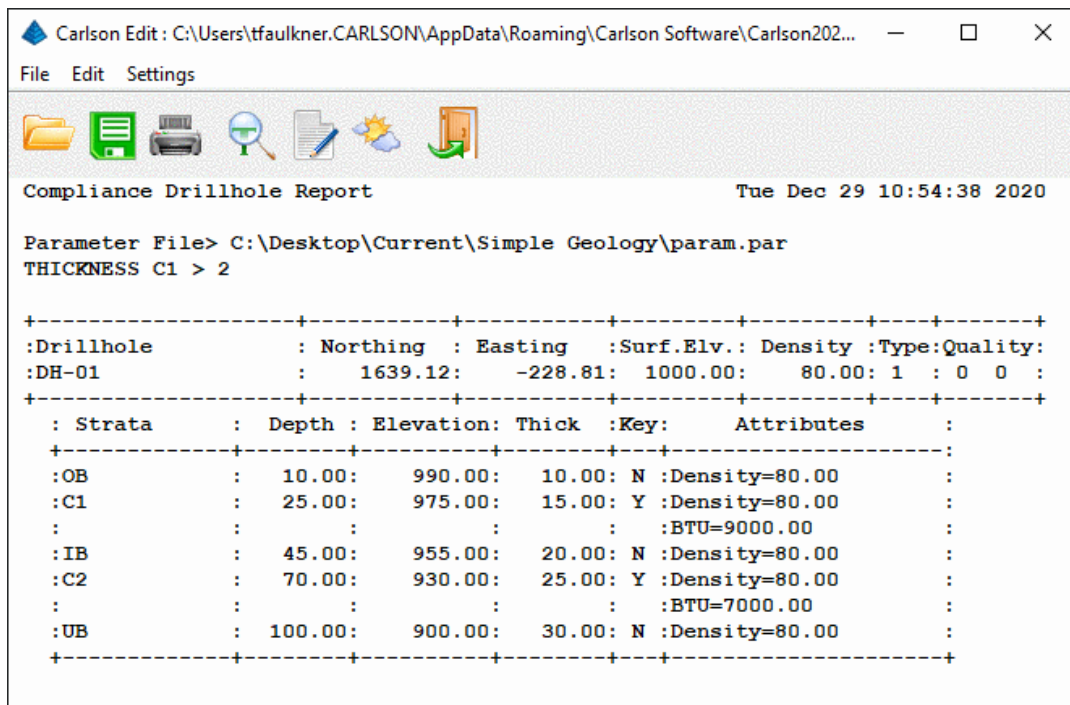
2) Analyze entire drillhole or specific strata [<Drillhole>/Strata]?

The entire drillhole method will include the drillhole in the report if any of the strata meet the parameter file. The specific strata method checks only the selected strata and includes the drillhole in the report if the strata meets the parameter file. When using the Strata option, a dialog will appear with the list of available strata layers.

3) Require all strata to match or any strata [All/<Any>]?

This option will only appear if a Strata Name has not been specified in the Parameter File. With this option, you can specify if all strata that are checked must pass the tests of the .par file, or if only one strata must pass the tests.

After these prompts, a text report will be displayed that shows a list of all of the drillholes passing the tests specified in the .par file. It is important to note that the entire drillhole will be reported, and not simply the strata that were tested.



Prompts

Parameter Dialog

Process beds [Yes/<No>]? *Y*

Analyze entire drillhole or specific strata [<Drillhole>/Strata]?

Require all strata to match or any strata [All/<Any>]?

Select the Drillholes to test.

Select objects: Specify opposite corner: 13 found

Compliance Drillhole report

Pulldown Menu Location: Drillhole > Parameter Compliance

Keyboard Command: charea3

Prerequisite: Drillholes

Compliance Areas from Grids

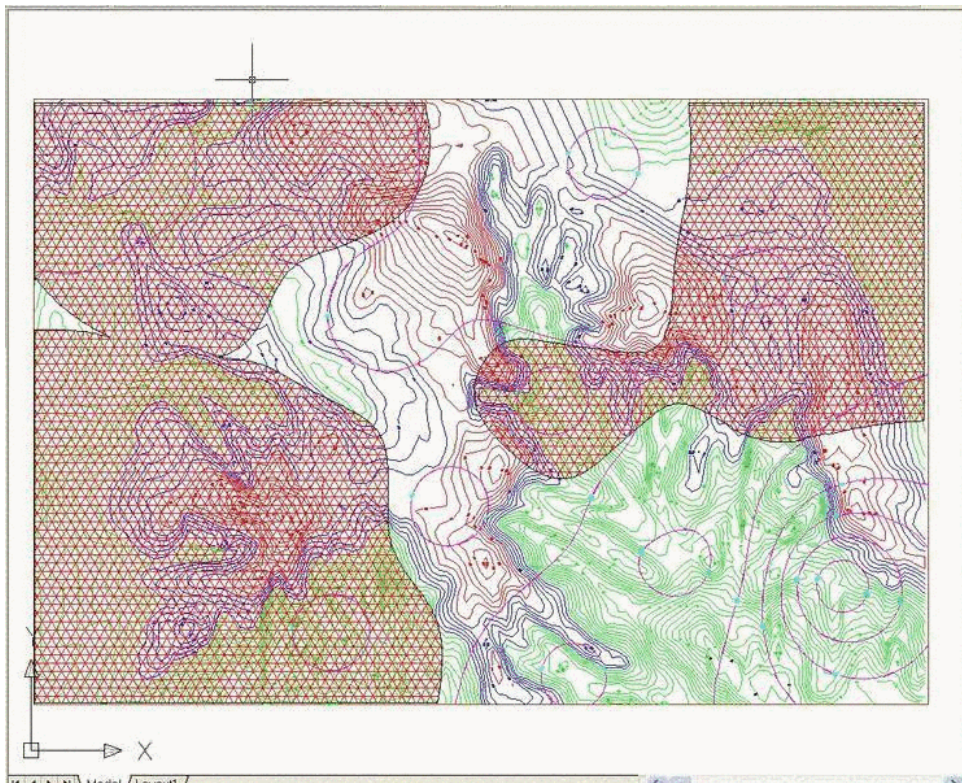
This command draws a closed polyline(s) and hatches the area(s) that meet the specified values for multiple grids. It is similar to the Parameter Compliance from Drillholes, but this analyzes grid files instead of drillholes.

The Set Parameters window is for selecting the grid files and specifying the values to use for compliance. The bottom portion is for choosing the layer, color, pattern and scale for the hatch. The area(s) that meet the criteria are then outlined and filled with the hatch pattern. If no inclusion/exclusion perimeter is selected, then the area of the extrapolated grid is hatched to the limits. As can be verified by the prompts, the command contours each of the grids for the values specified and then creates closed polygons where the values coincide.

GRID FILE	OPERATOR	VALUE	
SURFACE TUTORIAL\GRIDS\C1_KEY_BTU.GRD	>	8000	Select File
SURFACE TUTORIAL\GRIDS\C1_KEY_ASH.GRD	<	5	Select File
	<		Select File
	<		Select File
	<		Select File
	<		Select File
	<		Select File
	<		Select File
	<		Select File
	<		Select File
	<		Select File
	<		Select File
	<		Select File

Layer: GRD_AREA Select Color: ■ Pattern: ANSI31 Scale: 100.0

OK Clear Cancel



Prompts

Contouring elevation 8000.00

Inserted 686 contour vertices.

Reading file > c:\scad2005\USER\epattern.dta for pattern definitions...

Pre-processing grid cells ...

Processing cells ...

Reading cell > 194032

Contouring elevation 5.0

Inserted 875 contour vertices.

Reading file > c:\scad2005\USER\epattern.dta for pattern definitions...

Pre-processing grid cells ...

Processing cells ...

Pulldown Menu Location: Drillhole > Parameter Compliance

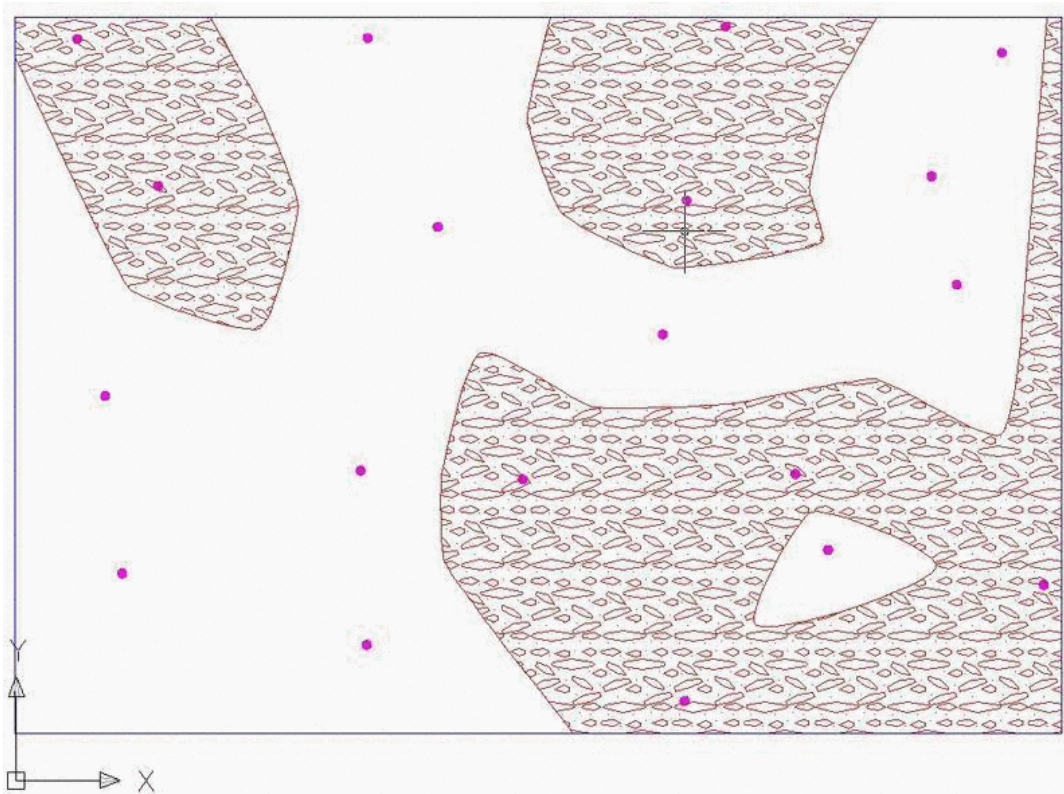
Keyboard Command: grdparam

Compliance Areas from Drillholes

This command draws a closed polyline(s) and hatches the area(s) that meet the specified values for parameters. It is similar to the Parameter Compliance from Grids, but this analyzes drillholes instead of grids.

The Set Parameters window is for selecting the parameters and values. In this example, two seams BTU are analyzed, so they are both entered in the Strata Name column.

Parameter	Operator	Value	And/Or	Type	Strata Name	Process
BTU	>	9200		Strata	C1_KEY	<input checked="" type="checkbox"/> On
BTU	>	8500	AND	Strata	C2_KEY	<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On
	<		AND	Strata		<input checked="" type="checkbox"/> On



Prompts

Use position from another file or pick grid position [**<Pick>/File**]:

Pick Lower Left grid corner **<1.45504e+006,1.96932e+006>**:

Pick Upper Right grid corner **<1.46634e+006,1.97717e+006>**:

Analyze entire drillhole or specific strata [**Drillhole/<Strata>**]? **S**

Select the Drillholes to test.

Select objects: all

19 found

Select objects:

Reading points... 18

Reading points... 0

Inserted 19 points.

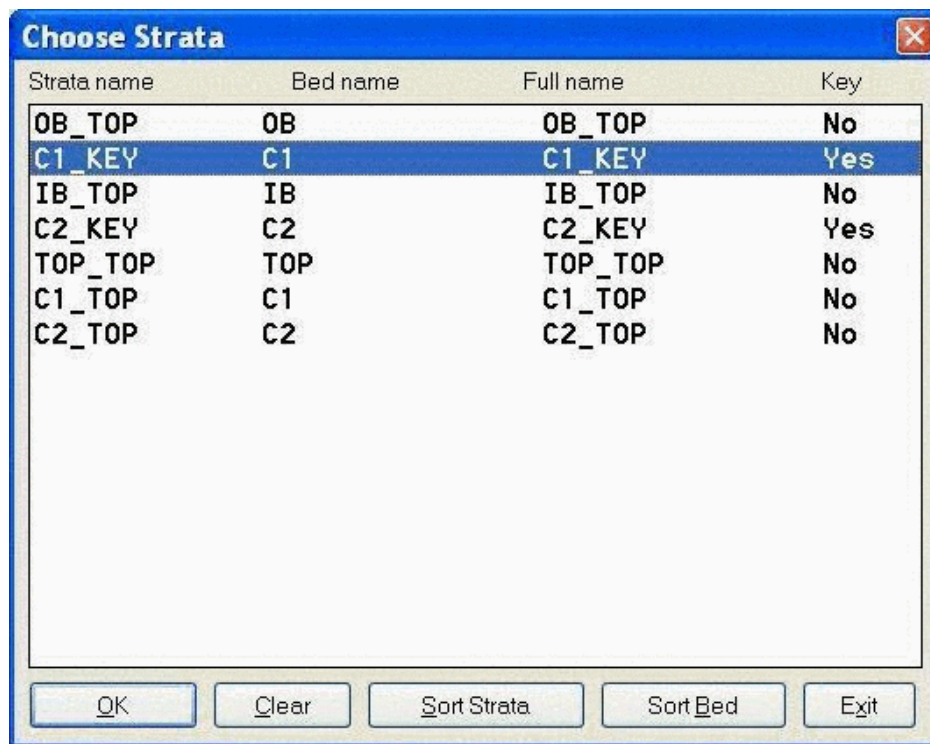
Pulldown Menu Location: Drillhole > Parameter Compliance

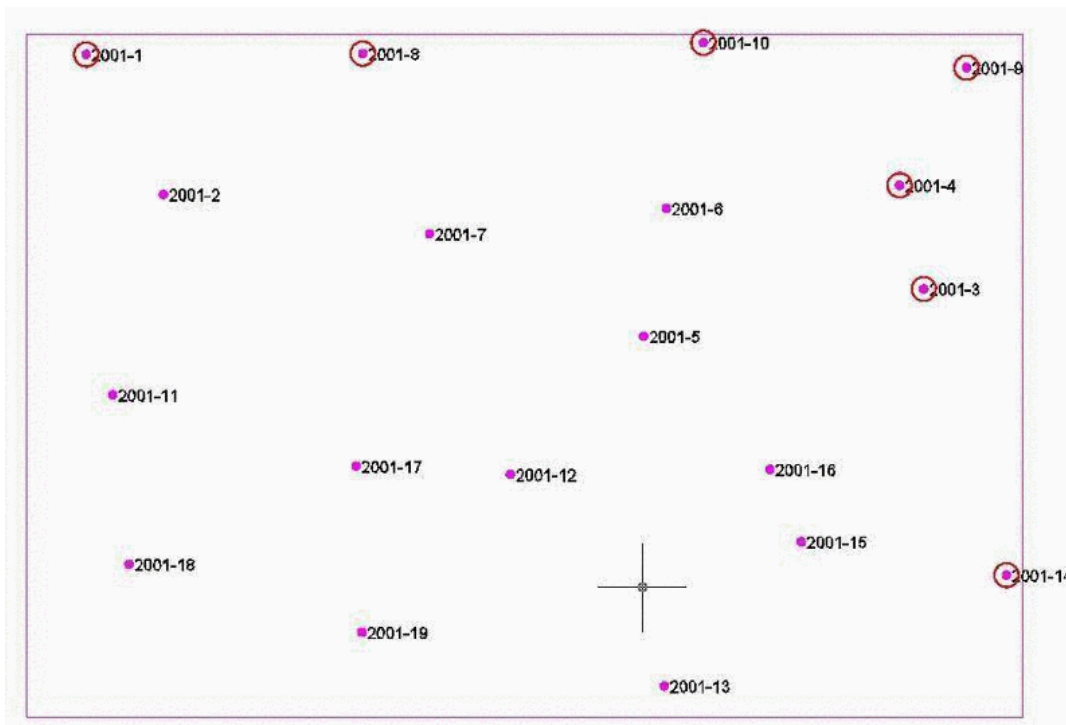
Keyboard Command: charea

Mark Compliance Drillholes

This command highlights drillholes that contain a strata that meets the specified parameters. The drillholes are highlighted by drawing a circle around them, changing the drillhole symbol, changing the drillhole type or changing the drillhole layer. There is also an option to create a selection set of the matching drillholes. Then you can use that selection set in any other command that prompts for drillholes by entering P for Previous at the Select Objects: prompt.

The parameters are set in the Define Parameters window. Be sure to spell the attributes exactly the same in the parameter list as in the drillholes. For more details about parameter definitions, see the Define Parameters command.





Prompts

Set Parameters dialog

Process beds [Yes/<No>]? *press Enter* This option composites the strata using their bed names into BED.TOP, BED.PARTING, BED.KEY and BED.BOTTOM. For example, you could have several quality samples of a seam broken into strata names CO with bed name A. The process beds option will composite all these samples into A.KEY. Otherwise the parameter filter will test each sample separately.

Analyze entire drillhole or specific strata [<Drillhole>/<Strata>]? *press Enter* The Strata option allows you to select which strata to test. Otherwise all the strata in the drillholes are tested. If Strata is chosen, the Choose Strata window appears for selecting.

Require all strata to match or any strata [All/<Any>]? *press Enter* The Any option will mark the drillhole if any of the strata match the parameter filter. The All option requires all the strata to match in order to mark the drillhole.

Select the Drillholes to test.

Select objects: *pick the drillhole symbols*

Drillhole value to change, create selection set or draw circle

[Layer/Symbol/Type/Selection/<Circle>]? *press Enter for Circle*

Mark layer name <COREMARK>: *press Enter for layer*

Mark size <25.0>: *100 Enter size.*

Marked 7 drillholes.

Pulldown Menu Location: Drillhole > Parameter Compliance

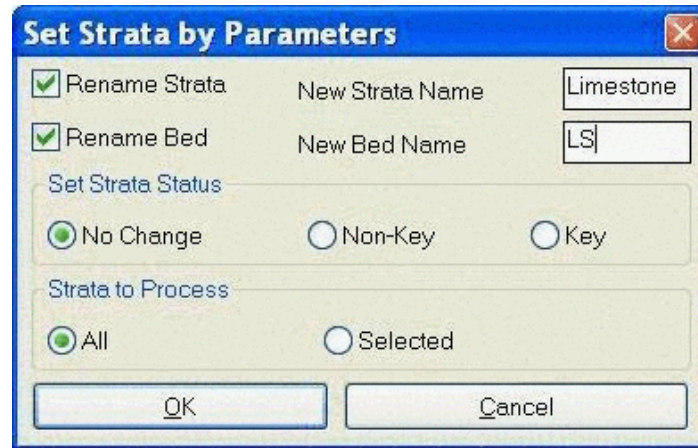
Keyboard Command: charea2

Prerequisite: Drillholes

Set Strata by Parameters

This command sets strata values of strata name, bed name or status (key/non-key) based on whether the strata passed the specified parameter filter. The parameter filter can test strata attributes such as thickness and qualities such as MGO. The change defined in the Set Strata by Parameters dialog will be apply to strata in the selected drillholes that passed the parameter filter. Also in the dialog, you can choose between processing all the strata or specific strata selected by strata name. For example, consider an ore seam that is made of several sequential strata with different quality samples of a strata attribute called GRADE. Set Strata by Parameters could be used to set the key/non-key

status of these strata based on whether the strata GRADE attribute is greater than 1.5%.



Prompts

Set Strata by Parameters dialog

Define Parameters dialog

Select the DrillHoles to process.

Select objects: *pick the drillholes*

Changed 42 strata in 20 drillholes.

Pulldown Menu Location: Drillhole > Parameter Compliance

Keyboard Command: strata_param

Split Bed by Parameters

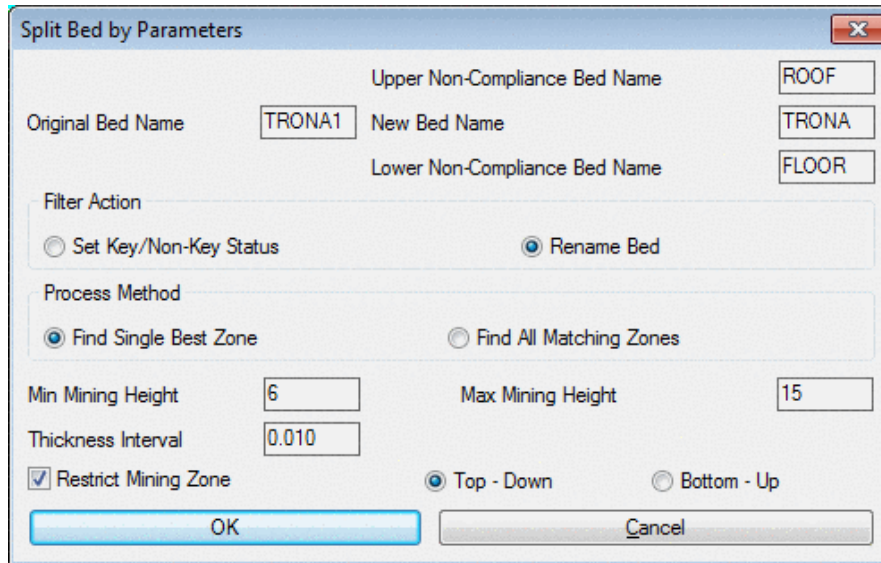
This command finds the largest portion of a bed such that the composited qualities meet the specified parameter filter. The bed can be split by key/non-key status or by name. With the status method, the bed portion that passes the filter is set key and the rest is set non-key. For the name option, you can specify the bed name for the portion that passes the filter and names for the portions that don't pass.

This command is different from Set Strata by Parameters because this command composites multiple strata to meet the filter instead of checking each strata individually against the filter. In Split Bed by Parameters, the composited portion that passes the filter can contain some strata that fail the filter so long as the total composite portion passes. For example, if the parameter filter is for strata attribute GRADE to be greater than 1.5, then final bed composite that passes could be made of one strata with GRADE of 1.6 plus part of another strata with GRADE 1.1 such that the composite GRADE equals 1.5. The strata that make up the bed can be divided into a part that is used in the composite and part that is discarded. For example in the above case, the strata with the GRADE of 1.1 could be originally 2 feet thick. Since this strata is below the target GRADE of 1.5, it could be that only 0.7 feet of this strata could be added to the composite without bringing the composite below 1.5. So the strata would be split at 0.7 feet with the remaining 1.3 feet not becoming part of the composite.

This command can also account for minimum and maximum mining heights to limit the composite bed thickness. If there is no portion that meets the filter at the minimum thickness, then the program will find the best portion. Likewise if the composite that meets the filter exceeds the maximum height, then the program will find the best portion with the maximum allowed thickness. The thickness interval resolution allows the intervals to be broken up into as small of units as required. The Find All Matching Zones will not stop at the single best zone, it will separate the beds into all of the matching zones.

Restrict Mining Zone is an option that defines where the program will start calculating from, either the top or bottom. For a surface mine, the Top Down would make the most sense, so that the most efficient method of mining

is designed. For underground, it is possible that the Bottom-Up option is preferred, so that extra material will be left in the roof for support.



Prompts

Split Bed by Parameters dialog

Define Parameters dialog

Select the DrillHoles to process.

Select objects: *pick the drillholes*

Changed 42 drillholes.

Pulldown Menu Location: Drillhole > Parameter Compliance

Keyboard Command: splitpar

Tag Drillholes for Processing

This command sets the processing status of drillholes based on specified parameters. The processing status is set as ON or OFF which is used by Stratacalc routines to determine whether to include the drillhole in processing. Drillholes have processing ON by default. The drillholes that do not match the specified parameters are tagged as processing OFF. The Reset Selected Drillholes option at the start is a way to set the processing status of the selected drillholes to ON and start fresh, with them all set to ON.

Prompts

Reset selected drillholes to processing on status (Yes/<No>)? *press Enter*

Define Parameters Dialog

Analyze entire drillhole or specific strata (<Drillhole>/Strata)? *press Enter*

Select the DrillHoles to test.

Select objects: *pick the drillholes*

Tagged 20 drillholes ON

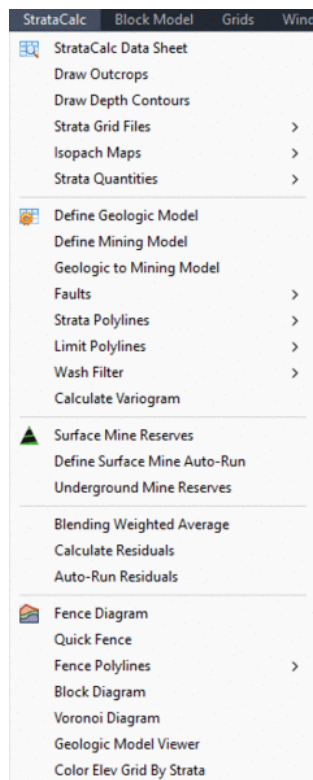
Tagged 6 drillholes OFF

Pulldown Menu Location: Drillhole > Parameter Compliance

Keyboard Command: charea4

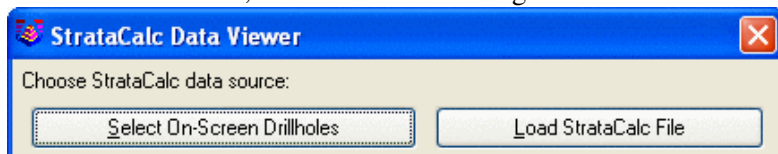
StrataCalc Menu

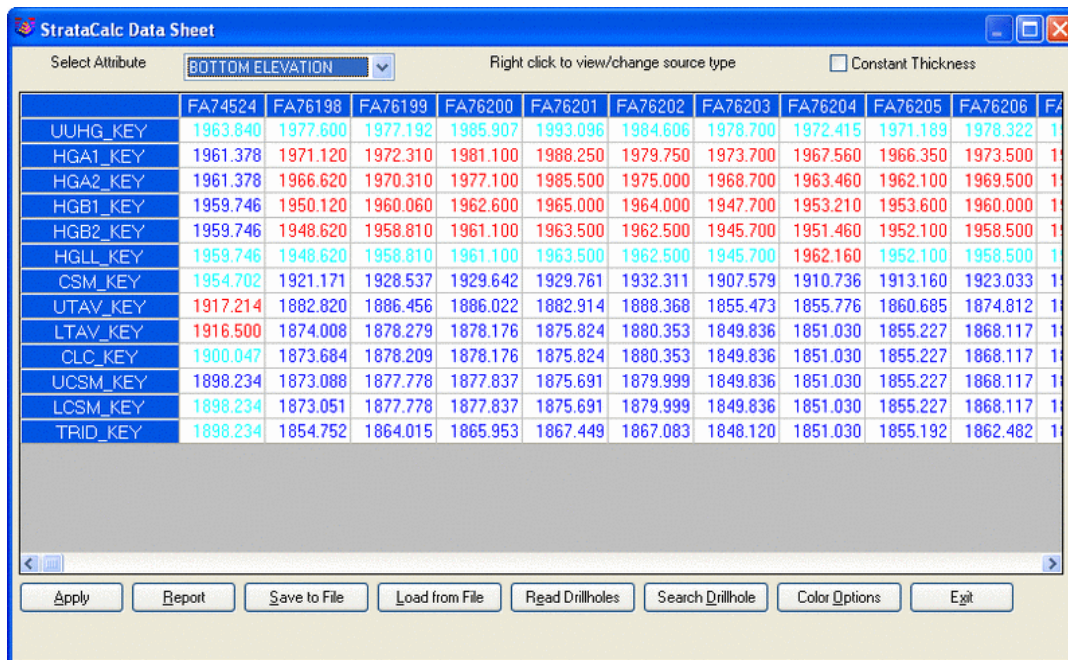
The StrataCalc pull-down menu has commands for creating and processing strata models.



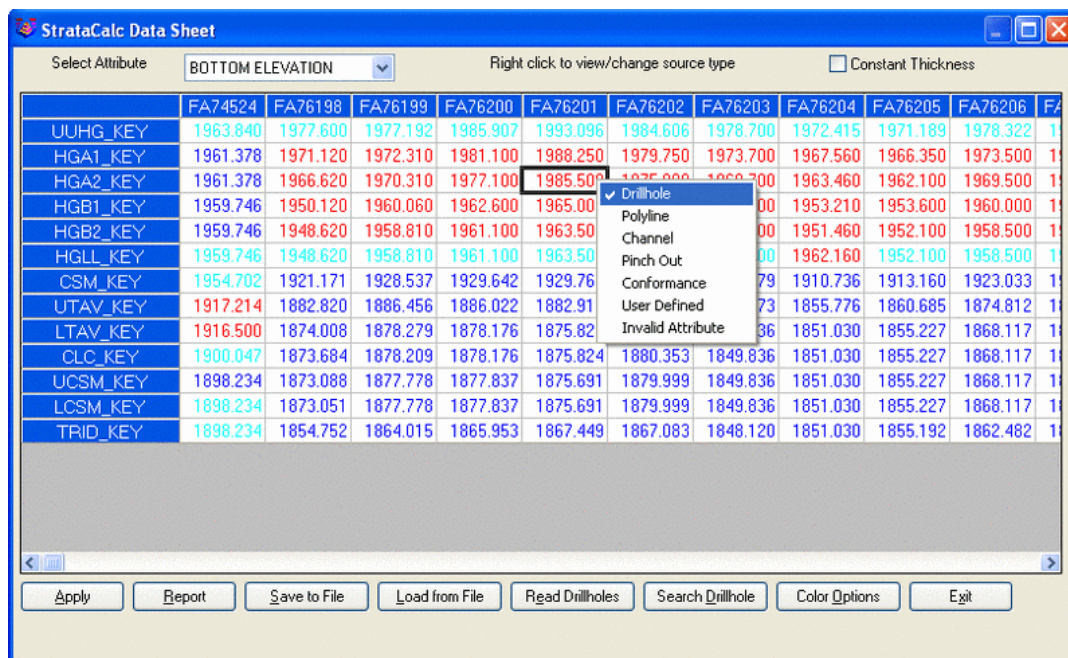
StrataCalc Data Sheet

This command displays geological data for viewing and editing of the processed values in a spreadsheet view. It could be called an interpolator, as it displays the values that the program interpolates and will use for modeling. Each value can be editing and "forced" or user defined values can be used instead of what is found in the drillhole, or modeled by the program. Each column represents a datapoint, such as a drillhole. The color of each text value represents the source of the data. This StrataCalc file (*.STC) will then be used for the modeling. Any time the program prompts "Select Drillholes", if none are selected and Enter is chosen, then the program will ask for the StrataCalc file. It will use this file and the values in it for the modeling. This is a great tool for controlling the geology. If a few drillholes are troublesome and not modeling correctly, the user can edit this file to what they want to use and create a more accurate model. The first window is to choose the source of the data, either the drillholes on screen, or to load an existing STC file. Then the Data Sheet comes up for viewing and editing.



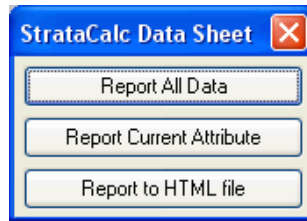


- **Select Attribute:** The dropdown controls the values to display and edit in the spreadsheet. It will always have Thickness, Bottom Elevation and Top Elevation. Any other user defined attributes will also be here for control.
- **Right click to view/change source type:** When placing the pointer on a cell and right clicking the mouse, the menu displays the possible sources of the value. The source used is the one with the check mark next to it. The source can be changed by picking a different one, and the color of the text in the cell would then change to represent that.



- **Constant Thickness:** If Constant Thickness toggle is ON, then when the user updates the top elevation, the thickness will be kept constant and the bottom elevation will be changed (bot elev = top elev - thick). If the bottom elevation is changed thickness will remain constant and top elevation be changed (top elev = bottom elev + thick). If the toggle is OFF, then if elevation is changed the thickness will also be updated accordingly. (thick = top elev = bot elev)
- **Apply:** This will apply the changes made to the spreadsheet the update the colors and values.

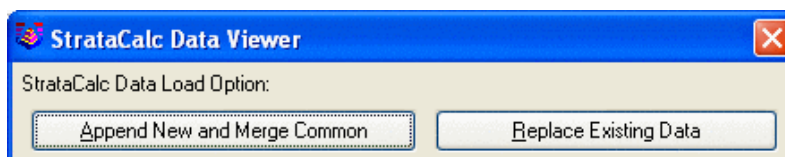
- **Report:** There are three options to see the values in a text report: Report All Data, Report Current Attribute, or Report to HTML. The HTML report takes the current spreadsheet view of the data and puts it into HTML format. The other two reports output to the report formatter.



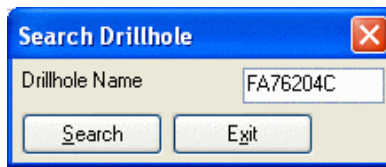
DRILLHOLE REPORT

Attribute	Name	Strata Name	Drillhome Name	Data Source	Attribute Value
THICKNESS	HGA1_KEY	FA97055C	Drillhole	Drillhole	4.50
THICKNESS	HGA1_KEY	FA97056C	Drillhole	Drillhole	4.70
THICKNESS	HGA1_KEY	FA97057C	Drillhole	Drillhole	5.15
THICKNESS	HGA1_KEY	FA97058C	Drillhole	Drillhole	4.80
THICKNESS	HGA1_KEY	FA97059	Drillhole	Drillhole	4.35
THICKNESS	HGA1_KEY	FA97060C	Drillhole	Drillhole	4.85
THICKNESS	HGA1_KEY	FA97061C	Drillhole	Drillhole	5.05
THICKNESS	HGA1_KEY	FA97062C	Drillhole	Drillhole	5.05
THICKNESS	HGA1_KEY	FA97063	Drillhole	Drillhole	4.45
THICKNESS	HGA1_KEY	FA97064C	Drillhole	Drillhole	5.00
THICKNESS	HGA1_KEY	FA97065	Drillhole	Drillhole	4.80
THICKNESS	HGA2_KEY	FA85P035	Drillhole	Drillhole	4.75
THICKNESS	HGA2_KEY	FA74524	Stack Conformance	Stack Conformance	2.46
THICKNESS	HGA2_KEY	FA76198	Drillhole	Drillhole	4.50
THICKNESS	HGA2_KEY	FA76199	Drillhole	Drillhole	2.00
THICKNESS	HGA2_KEY	FA76200	Drillhole	Drillhole	4.00
THICKNESS	HGA2_KEY	FA76201	Drillhole	Drillhole	2.75
THICKNESS	HGA2_KEY	FA76202	Drillhole	Drillhole	4.75
THICKNESS	HGA2_KEY	FA76203	Drillhole	Drillhole	5.00
THICKNESS	HGA2_KEY	FA76204C	Drillhole	Drillhole	4.10
THICKNESS	HGA2_KEY	FA76205	Drillhole	Drillhole	4.25
THICKNESS	HGA2_KEY	FA76206	Drillhole	Drillhole	4.00
THICKNESS	HGA2_KEY	FA76207	Drillhole	Drillhole	2.50
THICKNESS	HGA2_KEY	FA76208C	Drillhole	Drillhole	4.50
THICKNESS	HGA2_KEY	FA76209	Drillhole	Drillhole	5.00

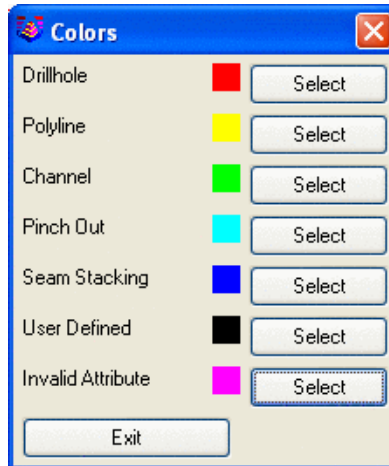
- **Save/Load to File:** This saves the changes to the STC file, or loads an existing STC file.
- **Read Drillholes:** Choosing this will bring up two options on how the drillholes should be loaded into the spreadsheet. They can either append and merge, or replace the holes if they already exist.



- **Search Drillhole:** To find a drillhole in the spreadsheet, chose this option and enter in the drillhole name. It will scroll to it and highlight the drillhole column of cells.



- **Color Options:** This is a customizable window to set which color the text will be displayed in the spreadsheet. There are seven main sources of the data: Drillhole, Strata Polyline, Channel Samples, Pinchout value, Seam Stacking value, User Defined, and an Invalid Attribute (out of the range of Define Attributes file).



Prompts

chinterp

Select drillholes, channel samples and strata polylines. (select the entities)

Select objects: Specify opposite corner: 294 found

Reading drillhole 294

Finding splits ...

Finding pinch out ...

Calculating seam stacking ...

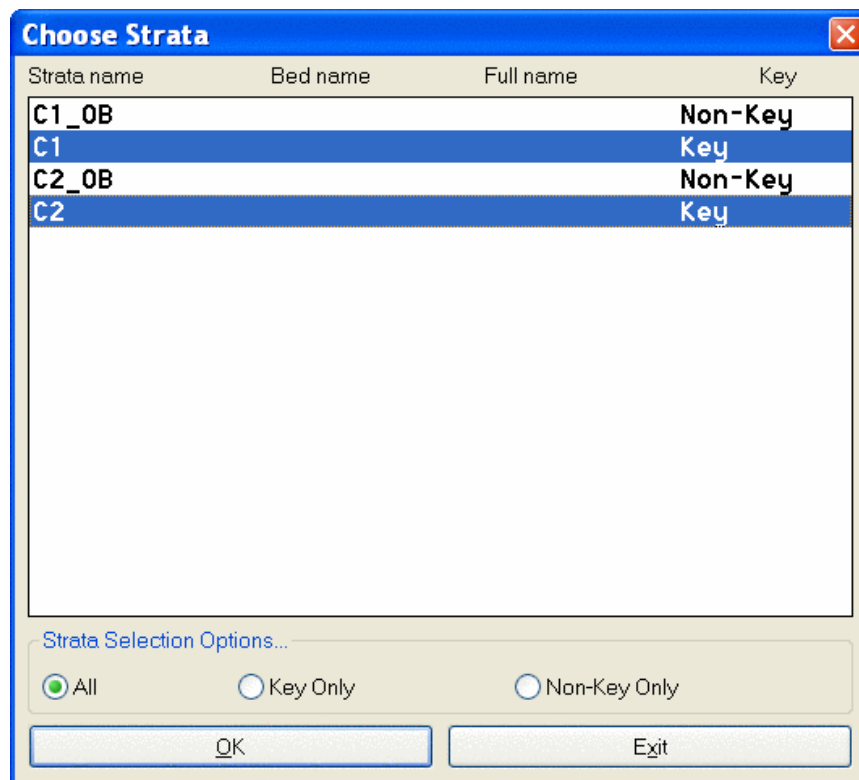
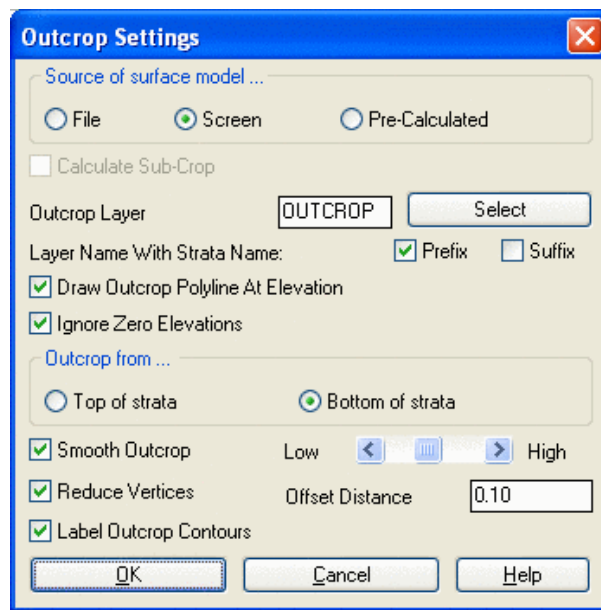
Processing only strata with beds.

Attribute Definitions missing (this is because the Define Attributes are empty)

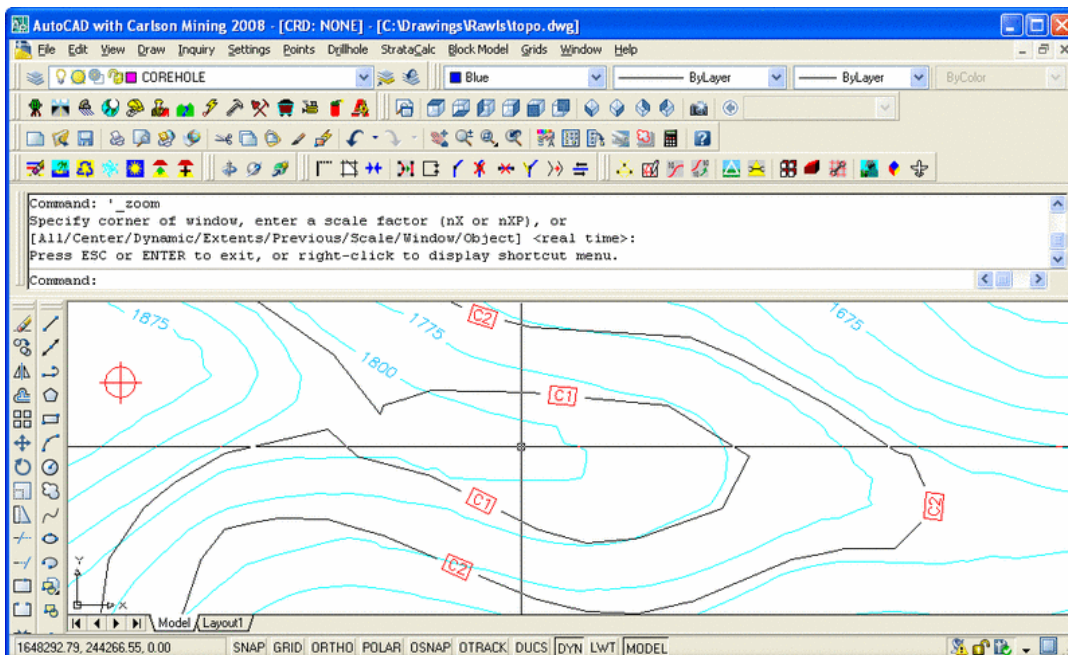
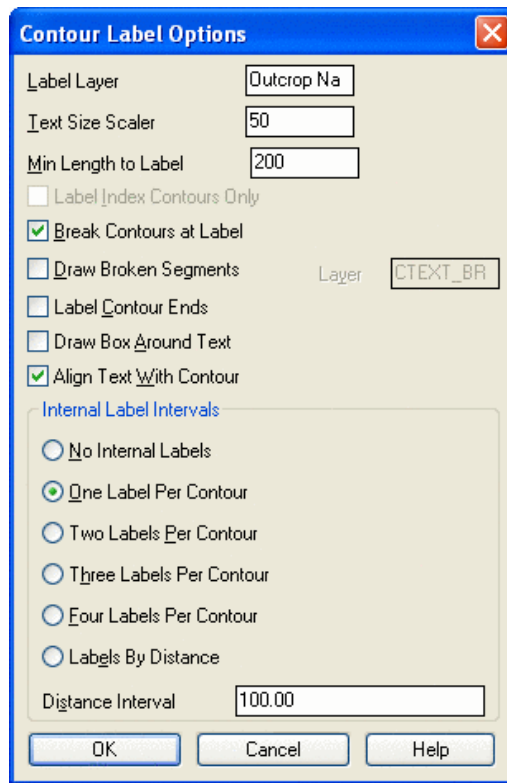
Keyboard Command: chinterp

Draw Outcrops

Draw Outcrops generates polylines that represent the outcrops of strata from the selected drillholes. An outcrop is where a strata comes up to the surface. An option from the Outcrop Settings dialog allows the outcrop line to be calculated from where the top of the strata hits the surface or from where the bottom hits. The Draw Outcrop Polyline At Elevation will draw the outcrops as 3D polylines on the surface topography. For example, on a slope, the top of strata outcrop will be higher than the bottom, and on the inside when viewed in plan view. When coming from drillholes, Draw Outcrops calculates the outcrop by creating a grid of the surface and a grid for the strata and then finding the intersection of the two grids. If the strata never breaks through the surface, no outcrop line will be created. The Geologic Model option reads the surface grid and strata grid from a Geologic Model file (.pre). This option will show a dialog with a list to choose from of the strata defined in the Geologic Model file. If the File option is used, then a Sub-Crop line can be drawn that will prompt for a thickness grid of the strata. The lines can be smoothed and vertices reduced with the settings shown below.



The option to Label Outcrop Contours will label the strata name in the contour line as contour text. The contour text settings window appears for the desired settings to be entered.



Prompts

Outcrop Settings dialog

Select surface entities & at least 3 drillholes

Select objects: Specify opposite corner: 43 found

Use drillhole surface elevations in surface model [Yes/<No>]? N

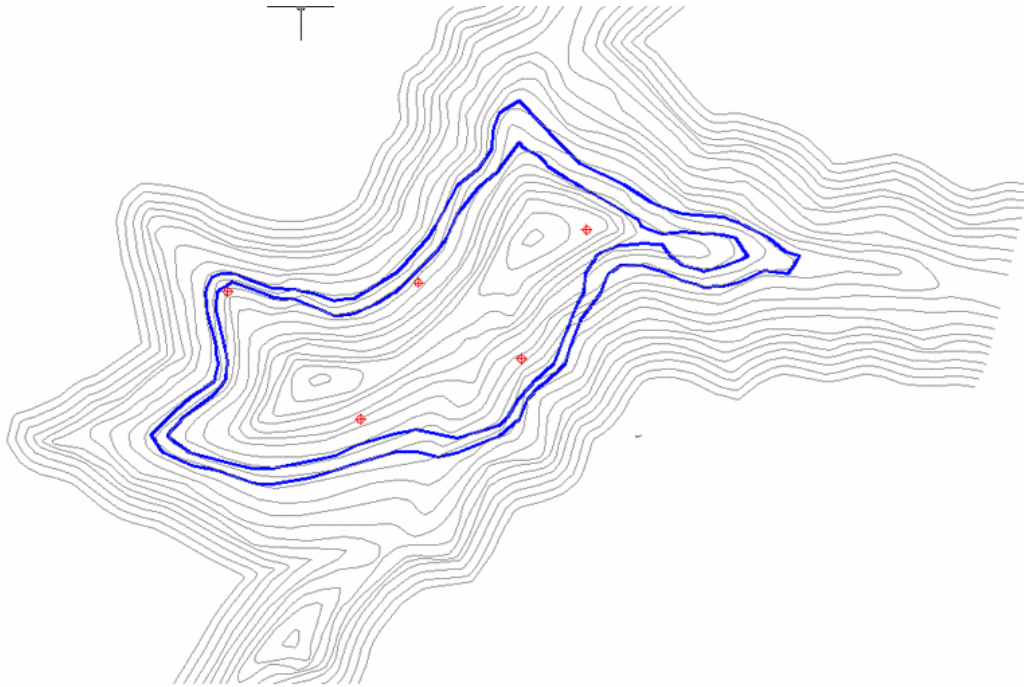
Reading points ... 3563

Ignored 5 duplicate points.

Output grids for strata and surface [Yes/<No>]? n

Choose modeling method [<Triangulation>/Inversedist/Kriging/Polynomial/LeastSq/ABOS]? i (or choose another method)

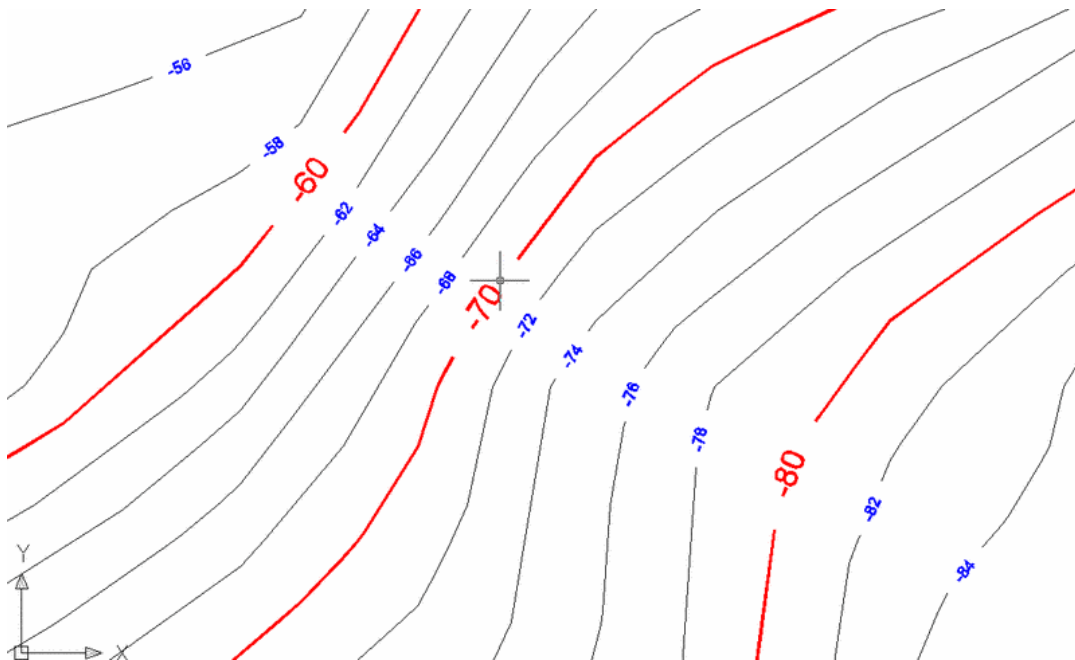
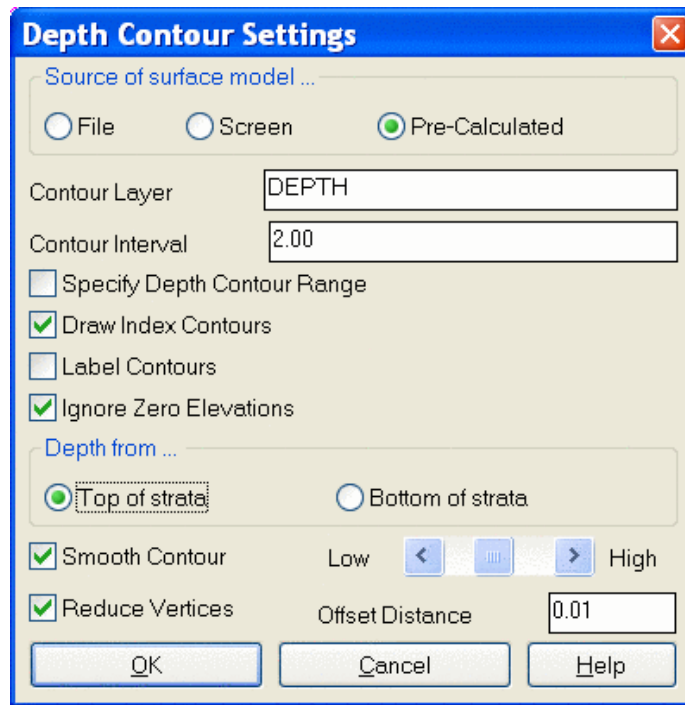
Use inverse distance to which power [First/<Second>/Third/Other]?
Use elliptical inverse distance [Yes/<No>]?
Calculating grid by inverse distances 2601...
Contouring elevation 0.0
Inserted 136 contour vertices.



Drop-Down Menu Location: StrataCalc
Keyboard Command: outcrop

Draw Depth Contours

This command creates contours for the depth between the surface and strata. The surface models for the ground and strata are modeled by grid if coming from the screen entities and drillholes. When using the Geologic or Mining Model file, the list of strata appears to select which strata to use. Specify Depth Contour Range allows for a starting and ending contour elevation. There is the option to contour the depth to either the top or the bottom of the strata. Options are at the bottom for smoothing and reducing the contour vertices. This is basically the thickness from the topography to the top or bottom of the strata. This grid could also be made in Grid File Utilities, but this is a quick way to get the contours without gridding first.



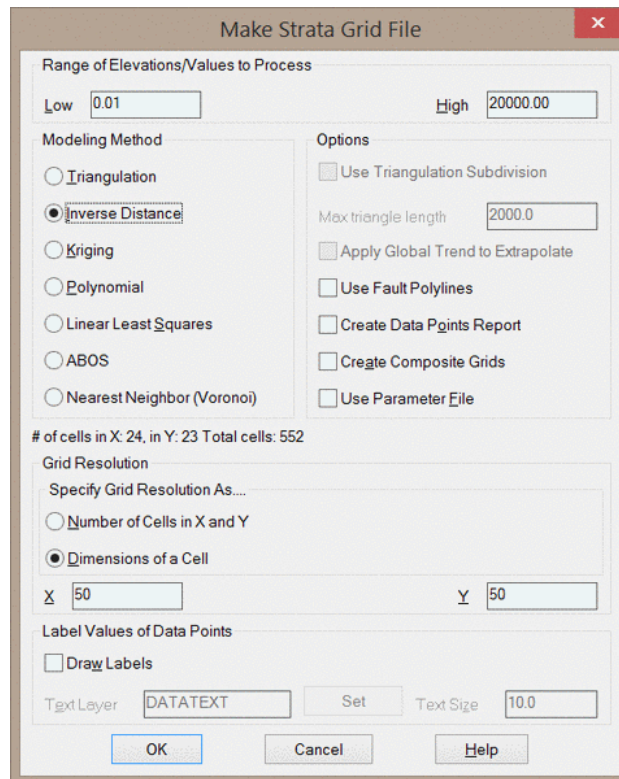
Pulldown Menu Location: StrataCalc in Advanced Mining and Mining
Keyboard Command: depth

Make Strata Grid Files

For each strata in the drillhole, this routine can generate 3D grid files of the strata thickness, top & bottom elevation, and attributes such as calcium, moisture, sulfur, and BTU. The grid files make up the geologic model in StrataCalc. A 3D grid file is a rectangular mesh of grid cells where each grid cell is the same size rectangle. The elevation or Z value of the four grid corners equals the value at those points. For example, consider a grid cell for a 3D grid mesh of strata thickness. The X and Y coordinates of a grid cell could be (0,0), (10,0), (10,15), and (0,15). The z values, which might be 4.5, 4.7, 4.8, and 4.6, represent the strata thickness at the four X and Y coordinates.

Make Strata Grid Files reads the strata data from the drillholes. The strata data is correlated and processed for beds, pinch out and conformance as specified in Carlson Configure. The strata data points are then used with the selected modeling method to calculate the grid. There is an option to use a parameter file (made with Define Parameters) to filter the strata data points. For example, when creating a thickness grid for the overburden of 6_COAL, you could have a parameter filter of THICKNESS 6_COAL > 1.0. This filter would make the program use only drillholes with 6_COAL greater than 1.0 when calculating the 6_COAL overburden grid. See the Define Parameter File command for more description on parameters.

The routine starts by prompting for the location of the grid files to create. The location can be specified by picking the lower left and upper right corners or by selecting an existing grid file which sets the new grid location and resolution to match the selected existing grid file. Next there is the Make Strata Grid File dialog for choosing the grid cell resolution, modeling method and other parameters.

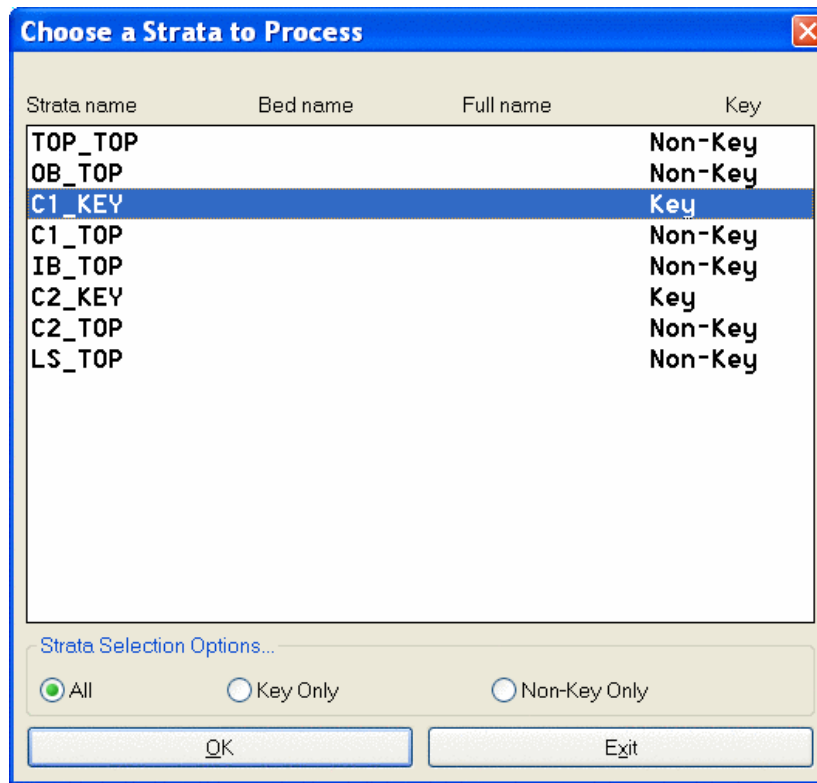


- **Range of Elevations/Values to Process:** Only values that fall within the range specified by the Low and High will be used to make the grid. Values that fall outside this range will be ignored.
- **Modeling Method:** There are multiple modeling methods available for geologic gridding, with many settings for each method. Each method is described in more detail below.
- **Use Triangulation Subdivision:** When modeling with triangulation, this option will divide each triangle into 3 or more smaller triangles for smoother modeling.
- **Max Triangle Length:** This value sets the maximum distance between data points to model when using the triangulation method. If data points are further than this distance from one another, the grid will be modeled with null values.
- **Apply Global Trend to Extrapolate:** When using the triangulation modeling method, values will only be modeled between data points - no extrapolation will be applied. When this option is enabled, however, the grid will be extrapolated to the full extents of the grid file.
- **Use Fault Polylines:** This option will prompt you to select fault lines to apply shifts to elevation grids. For more information regarding faults, see the Edit Fault Line documentation.
- **Create Data Points Report:** The Create Data Points Report option is another way to check the data points

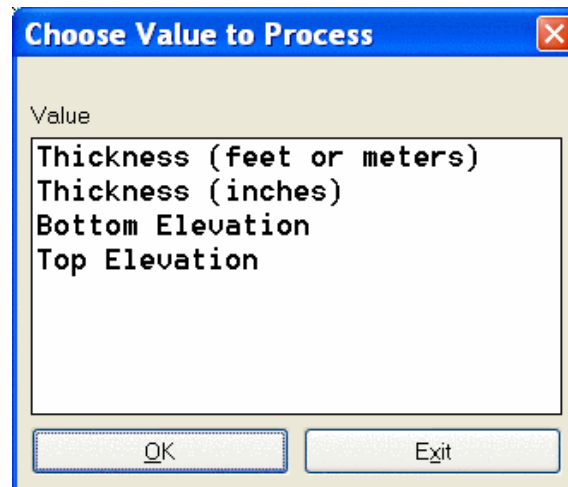
that are used in gridding. This option will create a report of the gridding data points. This report is displayed in the Standard Report viewer where it can be saved, printed or drawn on screen.

- **Create Composite Grids:** The Create Composite Grids option allows you to select multiple strata or beds to combine instead of gridding the strata individually. With compositing, the program shows the same strata selection dialog with the list of strata but you can now select multiple strata by picking from the strata list while holding down the Shift or Ctrl keys on the keyboard. For composites you can grid the top elevation which is the highest elevation at each drillhole for the composite, the bottom elevation which is the lowest composite elevation, the thickness which is the sum of the thicknesses for all the composite strata, or composited strata attributes.
- **Use Parameter File:** Turning this option on will use a parameter filter file made in the Define Parameters command.
- **Grid Resolution:** This option controls the resolution of the grid, that is, it determine the dimensions of each grid cell, as well as the total number of cells in the grid. As these options are updated, a preview of the cell dimension and number of cells will be displayed above this option. It is also important to note that there is a tradeoff between a higher grid resolution and processing time. It is recommended to keep the total number of cells at less than 1,000,000 per grid, but some computers may be able to process more data.
 - **Number of Cells in X and Y:** This option allows you set the total number of cells in the X and Y directions. For example, setting both of these values to 10 will create a grid will 100 cells. When using this option the dimensions of the cells will be automatically calculated based on the extents of the grid and the number of cells specified.
 - **Dimensions of a Cell:** This option allows you to set the dimensions of the cells. It is important to note that extents of the grid are fixed at this point, and that it may not be possible to use the exact dimensions specified here. However, the cells will be made to match these values as closely as possible.
- **Label Values of Data Points:** This option will draw a label next to each drillhole indicating what value was used at each drillhole for the gridding. The **Text Layer** and **Text Size** may be defined.. This option is a good way to visualize and check the data points that the program is using.

After the grid options dialog, the program prompts you to select the drillholes and fault lines. After reading in the selected drillholes, there is a Choose a Strata to Process dialog to choose the strata to process from the list. If Create Composite Grids was turned on, then multiple strata may be selected here. The All, Key Only and Non-Key Only are just viewing options to reduce the number of strata in the window if there are a lot, for ease of selection.



Once the strata is selected, another dialog shows the strata values and attributes that are available to grid. These will be thickness, elevations, equations, and any quality attributes that are in the drillholes. Choose a value to grid from the dialog.



Then the program will prompt for the grid file name to create (.GRD). Next the program will process the strata data points to create the grid and the results are stored in a user-specified file name. The grids created by Make Strata Grid File can be used as the geologic model for the Geologic or Mining Model file. Also the grids can be used in the grid application routines, in the Civil Design module, such as Plot 3D Grid File, Grid File Utilities and Elevation Difference.

Making Strata Grids with Fault Lines

There is an option to select fault lines in addition to the drillholes. The fault lines should be drawn as 3D polylines with elevations that equal the fault differential. The program will grid with all modeling methods using the fault lines for making strata elevation grids. The 3D fault polylines should be drawn such that the left side of the polyline, relative to the direction of the polyline, is the low side of the fault and the right side is the high side. As each grid corner elevation is calculated, the program checks each drillhole. If the drillhole is on the same side of the fault polyline as the grid corner, then no adjustment is made to the drillhole elevation data. Otherwise the drillhole is

projected onto the fault polyline and the polyline value at this point on the polyline is used to adjust the drillhole elevation. For example, if the fault polyline value was 5.0 and the grid corner was on the high side of fault while the drillhole was on the low side, then 5.0 would be *added* to the drillhole elevation for modeling at that grid corner. If the grid corner was on the low side and the drillhole was on the high side, then 5.0 would be *subtracted* from the drillhole elevation. Reverse Polyline is a good way to reverse the fault line if it is drawn in the wrong direction.

Triangulation Modeling Method

This method is straight triangulation between the drillholes. Triangulation calculates these values by interpolating on the plane defined by the three points in the triangle that encloses the point. Since triangulation only interpolates, it can only calculate values within the area of the data. Afterwards, an extrapolation routine can then fill in the rest of the grid. This extrapolation uses a safe method that tends to average out the data. There is an option to extrapolation to apply the global trend. This option finds the average slope and direction of the existing data and applies this slope to extrapolating.

Inverse Distance Modeling Method

Inverse distance calculates the grid values by assigning weights to the existing data. The grid values calculated by inverse distance are a weighted average of the existing data. Inverse distance will not carry trends and the calculated grid values will never be higher than the highest existing data point. Likewise the calculated grid values will never be lower than the lowest existing data point. The weights are proportional to the inverse of the distance between the point to be estimated and the existing data point. Closer points are weighted more than points farther away. The inverse distance can be calculated to first, second, or third power which are $(1/d)$, $(1/d^2)$, and $(1/d^3)$ respectively. The power can also be any user-specified number such as 2.5. The inverse distance estimate is a weighted average with the individual weights computed as an inverse power of distance as follows:

$$W_i = \frac{d_i^{-\text{power}}}{\sum d_i^{-\text{power}}} \quad I = 1 \dots \text{number of samples}$$

where W_i is the weight computed for each sample i , each d_i is the distance between the location being estimated and sample i , and $-\text{power}$ is the inverse distance weighting power.

In Configure under Mining there are several options for controlling inverse distance. The Inverse Distance Search Radius is used for calculating a value at a point such that only drillholes that are within this Search Radius will be used in the calculations. The Inverse Distance Max Samples value limits calculations to the nearest specified number of drillholes to the point. For example, the program will use the nearest 10 drillholes. Inverse distance can also be controlled by quadrants which are divided into northeast, southeast, southwest and northwest. The Min Quadrants setting will use at least this specified number of drillholes from each quadrant as long as there are drillholes in the quadrant within the Search Radius. For instance, a setting of Min Quadrants of one would make the program look for at least one drillhole from each quadrant. The Max Quadrants value limits the number of drillholes used from each quadrant. For example if Max Samples was set to 25 and Max Quadrants was 10, then the total samples would be 25 with no more than 10 of the closest ones from each quadrant.

Elliptical inverse distance modeling method is an option that appears any time the Inverse distance modeling method is chosen. Elliptical inverse distance modeling will produce oval shaped model "bulls eyes" that are aligned by the specified azimuth of anisotropy. The specified factor is used to increase the weight of data samples that line up more closely with the azimuth at the model point. Data samples that exactly line up with the azimuth apply the full factor. Data samples that are perpendicular with the azimuth apply zero extra weight. Data samples at an angle between the azimuth and perpendicular apply a proportional adjustment. The specified factor is adjusted by azimuth is then added with one and then multiplied by the data sample weight. For example, using a factor of 1 and a data sample that matches the azimuth to the model point, this sample will get weighted double; $(1 + 1) * \text{weight}$. When using a factor of 2 and a data sample that matches the azimuth, the sample will get weighted triple; $(1 + 2) * \text{weight}$. The prompts will appear:

Use inverse distance to which power [First/<Second>/Third/Other]? *Second*

Use elliptical inverse distance [Yes/<No>]? Yes

Enter azimuth of anisotropy: 45

Enter anisotropic factor: 1

With these values of azimuth 45 and factor of 1, data samples that are NE/SW of the model points will get double weight (1+1), data samples that are NW/SE will get no extra weight, and data samples at in between angles will get a proportional extra weight.

Kriging Modeling Method

Kriging estimates the grid values by figuring the relationship between all the existing data points and then assigning weights to this data. Kriging finds the best fit linear unbiased estimates for the given data and model. Kriging can carry trends within and beyond the limits of the data and can find new high and low values. You must supply a model that defines the spatial relationship of the data which can be difficult. In fact, Kriging is a very complicated subject and you will need to reference an outside source for a detailed description such as *An Introduction to Applied Geostatistics* by Isaaks and Srivastava. Carlson uses Ordinary Kriging. All the parameters for this Kriging are specified in the dialog shown.

Kriging Spatial Continuity Model

Kriging Model Type

Spherical

Exponential

Gaussian

Power

Linear

Nugget Effect: 0.05

Range (a parameter): 2000.00

Scale (c parameter): 40.00

Azimuth of Anisotropy: 0.00

Anisotropy Ratio: 1.00

Search Radius: 4000.00

Max Points to Use for Kriging (1-25): 10

OK Cancel Help

Polynomial Modeling Method

The polynomial method is based off of triangulation. The difference is that instead of directly interpolating within each triangle, the polynomial method creates smooth transitions by using a fifth degree polynomial function that accounts for neighboring triangles. Since polynomial needs adjoining triangles, when there are fewer than five data points, there will be fewer than four triangles and the polynomial method will revert to straight triangulation. The same extrapolation logic for triangulation applies to the polynomial method.

Linear Least Squares Modeling Method

The linear least squares method finds the least squares best fit plane at each grid corner. The least squares routine weights each data point by inverse distance so that closer points are weighted more than points farther away. So the best fit plane varies at different points on the surface. The linear least squares method extrapolates trends very well. A lower inverse distance factor (i.e. 1.0) will weigh the data points more equally which models the trends more globally (sometimes called "global dip"). Likewise a higher inverse distance factor (i.e. 3.0) will weigh the closer data points more heavily which models local trends strongly (sometimes called "local dip"). Least squares

will trend and allows for data points that are new highs and lows, that don't appear in the original drillhole/point data. It does produce very nice, smooth contours that honor the data points.

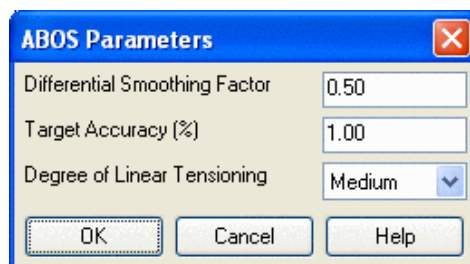
Approximation Base On Smoothing (ABOS) Modeling Method

ABOS is a method for modeling values of irregularly spaced points by using a continuous function with two independent variables. This method is developed and implemented by the developer of SurGe, Miroslav Dressler.

The ABOS method uses very simple mathematical tools - numerical tensioning and smoothing. The tensioning and smoothing are performed so that elements of matrix, which represents surface z-values at nodes of a regular rectangular grid, are repeatedly replaced by the weighted average of selected surrounding elements. The selection of elements involved into weighted average depends on the type of tensioning or smoothing.

Despite the fact that the mathematics of the ABOS method is simple, the resulting surface can be modified by a few parameters so that it is comparable with the surface created by sophisticated methods such as Kriging, Radial Basis Functions or Minimum Curvature.

Following dialog window shows ABOS parameters that need to be set in order to run ABOS inside the Carlson.



Differential Smoothing Factor:

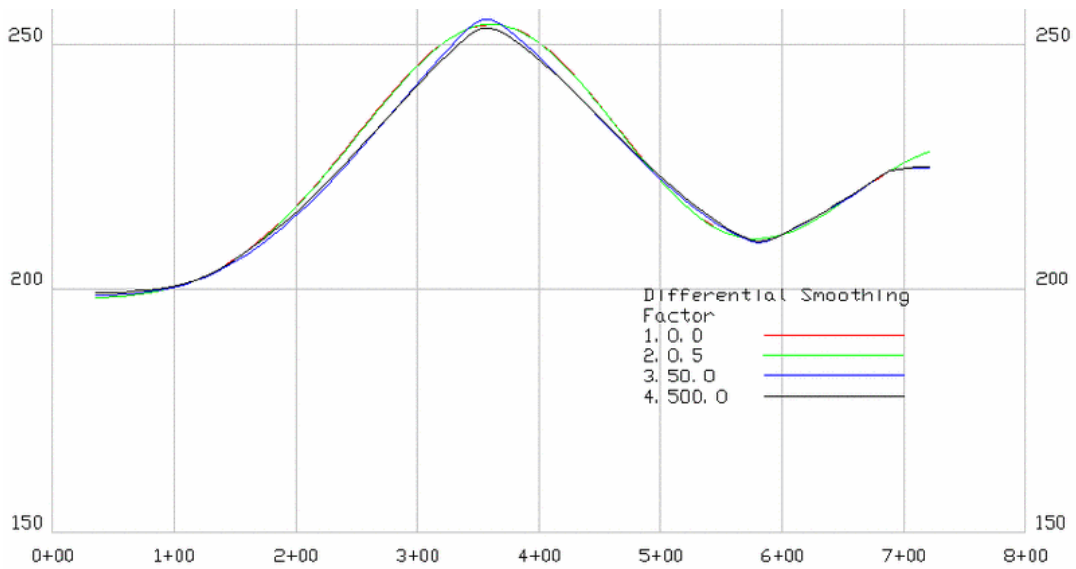
This parameter enables to control smoothness of generated surface. The larger value, the sharper interpolation is obtained. Typical values are:

0.00 - 0.30 ... for smooth interpolation

0.40 - 0.60 ... for normal interpolation (default value is 0.50)

0.70 - 1000.0 ... for sharp interpolation

Sharp / smooth model at local extremes can be improved by increasing the differential smoothing factor. The accuracy of generated surface decreases with increase in this factor. The zero value produces most smooth surface.



Target Accuracy:

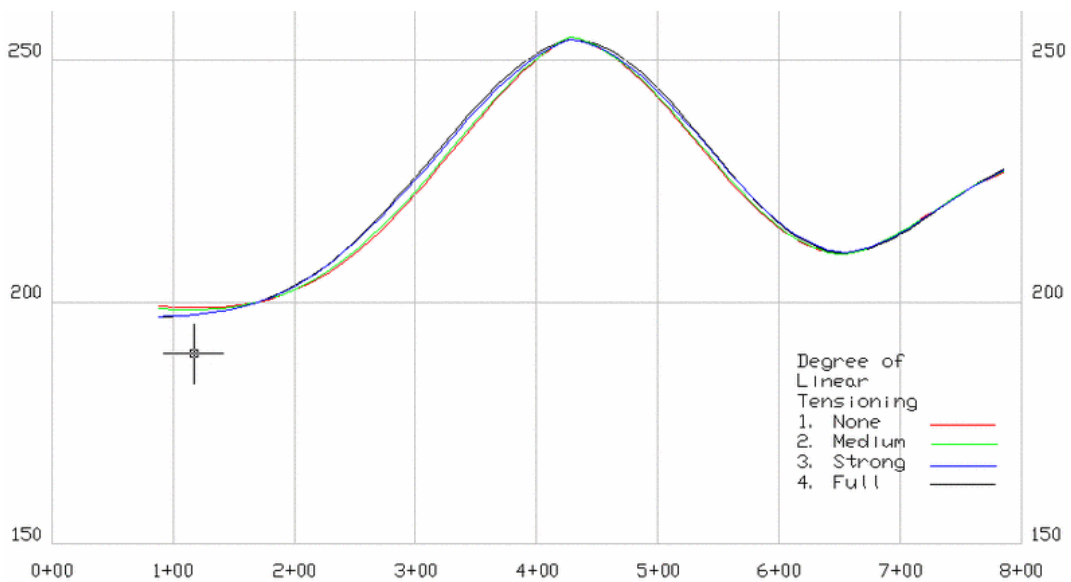
This parameter specifies how accurate the generated surface has to be. It represents the percentage value of average difference between z - coordinates of input points and generated surface from $(Z_{max} - Z_{min})$, where Z_{min} is the minimum value and Z_{max} is the maximum value of the z - coordinates of the input points. If the calculated accuracy $(Average\ DZ / (Z_{max} - Z_{min}) * 100)$ is less than the Target Accuracy, the process of interpolation stops and outputs the target grid.

Degree of Linear Tensioning:

This parameter enables to set the degree of linear tensioning. It can have four values:

1. None - for no linear tensioning
2. Medium - for medium linear tensioning
3. Strong - for strong linear tensioning
4. Full - for full linear tensioning

It is clear from the following figure of cross-section of a grid surface that the surface is more linear for higher degree of linear tensioning.



Nearest Neighbor (Voronoi) Modeling Method

The Nearest Neighbor method (also known as the Voronoi method) finds the nearest data point at each grid corner and assigns the grid corner to this data point. This method does not interpolate or extrapolate. Each grid corner is simply assigned the value of the nearest data point.

Prompts

Use position from another file or pick grid position (File/<Pick)? *press Enter* Using the position of an existing file copies the grid resolution and corner point locations to the new grid files. This is useful if you need to have grid files match exactly. Most of the time, grids should match position in the geologic model.

Pick Lower Left grid corner: *enter or pick a point*

Pick Upper Right grid corner: *enter or pick the second point to define the grid position*

Make Strata Grid File dialog box

Set the grid resolution and other options. A higher grid resolution increases the processing time. Also choose the Modeling Method in this dialog.

Select drillholes, channel samples and strata polylines.

Select objects: *select the drillhole symbols*

Select fault lines or Enter for none.

Select objects: *press Enter for none, or select the fault lines*

Choose Strata to Process dialog

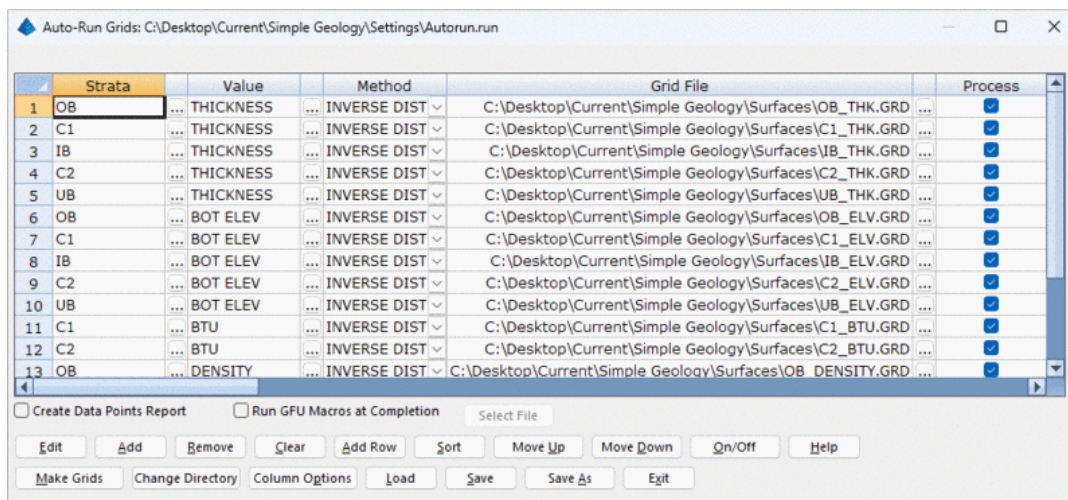
Choose Attribute to Process dialog

Pulldown Menu Location: StrataCalc

Keyboard Command: chgrid

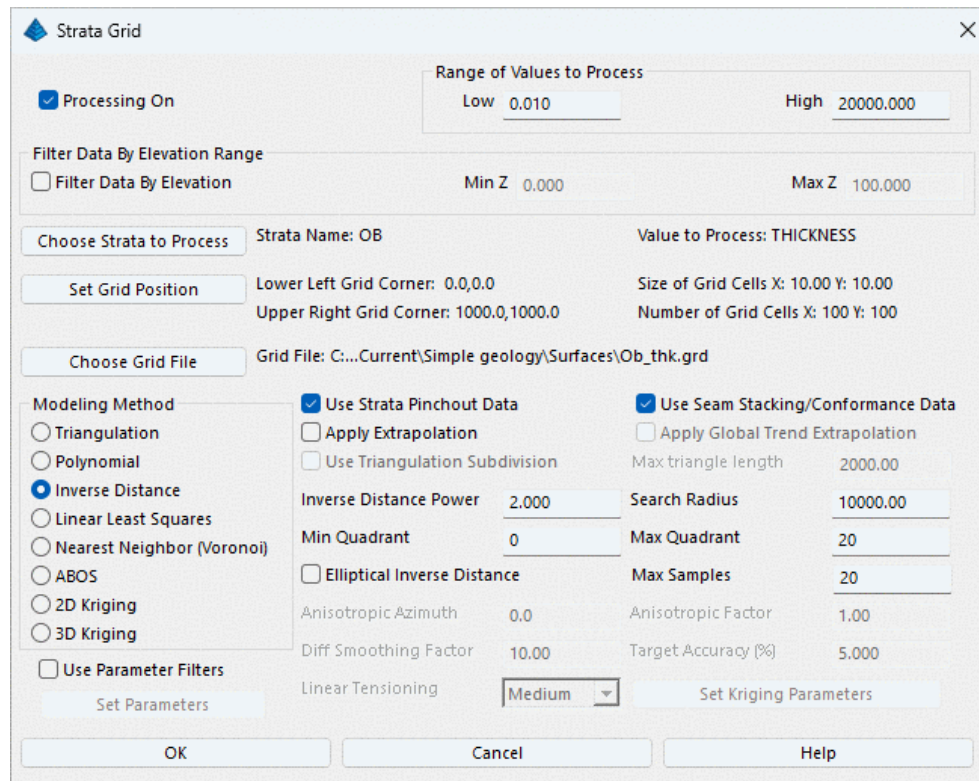
Define Strata Grids AutoRun

This command creates multiple strata grids from drillhole data at one time. This method is much faster than creating the grids one at a time with the Make Strata Grids command, especially when inputs change and the strata grids need to be recalculated quickly. Note that this command will create a .run (or a .runx) file, which can be used with the Run Strata Grids Auto-Run command rather than redefining the auto-run each time the grids need to be remade. Many of the settings in this command are the same as in the Make Strata Grids command, and will therefore not be discussed here. When the command is first executed, you will be prompted to select fault lines, drillholes, and inclusion/exclusion areas. It is important to note that if you select an inclusion area for this command, the program will only simply update grids inside the boundary - the entire grid will not be recreated. To ensure that the grids are populated with the latest available data, an inclusion boundary should not be selected.



This dialog lists the strata grids to be made with each grid listed on a separate row. Changes to each grid can be made in the spreadsheet or with the use of the Edit button.

Edit/Add: These buttons will open the below dialog which modifies parameters for the grid file. Recall that many of the below settings are discussed in the description of the Make Strata Grids command, and are not detailed here.



Processing On: This checkbox controls if the grid file will be created when the **Make Grids** button is clicked. If the grids take a long time to process and only some grids need to be processed, you can toggle this option to avoid making every grid file.

Range of Elevations/Values to Output: Only values that fall within the range specified by the **Low** and **High** fields will be used to make the grid. Values that fall outside this range will be ignored.

Filter Data By Elevation: This option will filter data points outside of the **Min Z** and **Max Z** range. Note that this is not the same as the Range of elevations/values to output. This option would allow you to limit the calculation to ignore the value because of its elevation, even though the attribute value may be valid. For example, suppose an attribute has a value of 10 and the sample exists at elevation 300. You could set the **Max Z** to a value of 250, which would prevent this attribute from being used.

Choose Strata to Process: This button will allow you to select the strata/bed and attribute to model in the grid file. The primary difference between selecting strata in the Make Strata Grids command and selecting strata here is that this command allows you to select multiple strata/beds at the same time. However, you may only create one attribute for the selected strata. For example, if you select five strata to model, you will not be able to select the elevation and the thickness attribute at the same time. Instead, you will have to define the five elevation grids and then repeat the process to define the five thickness grids. Also, when multiple strata are selected, you will only be able to select attributes that those strata have in common.

When selecting multiple strata, you will also be prompted if the strata should be composited or if separate grids

should be made for each strata layer. When strata attributes are composited into a single grid file, the attribute will be weighted by the thickness of the strata. Attributes that require averaging based on the weight of the material should be composited using Grid File Utilities.

After selecting the strata and attributes to model, you will be returned to the above dialog to set additional parameters for the grid(s). The attribute to be modeled and the first strata selected will be shown to the left of the Choose Strata to Process button.

Set Grid Position: This button will set the dimensions and extents of the grid cells. After setting the grid position once, all other grids in the autorun file will default to use the same position and cell sizes.

Choose Grid File: This button will set the name of the grid file. The program will automatically name grids with the following format: "Strata_Attribute.grd". For example, a file named "Overburden_Elv.grd" represents the bottom elevation of the Overburden strata. The below attribute names are used by default:

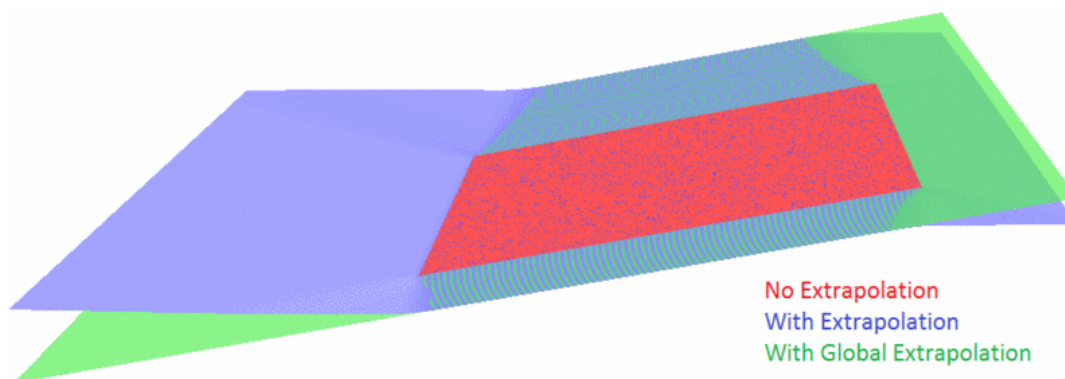
1. _ELV - this suffix is used for bottom elevation grids
2. _EL2 - this suffix is used for top elevation grids
3. _THK - this suffix is used for thickness grids
4. User-defined attributes will list the full attribute name in the grid file name

Use Pinchout Data/Use Seam Stacking/Conformance Data: These checkboxes control if pinchout and conformance logic will be applied to the grid file. However, this logic will only be applied if it is also enabled under Settings Pulldown Menu > Carlson Configure > Mining Settings. This allows you to quickly enable and disable pinchout and conformance data for select grids.

Modeling Method: Most of these methods are discussed in the section of the help manual corresponding to the Make Strata Grids command. However, the 3D kriging option is discussed in the Make Block Model command.

Apply Extrapolation: This option will extrapolate the grids where null values exist due to an inadequate search radius or inherent limitations by the modeling method. When a value is extrapolated, the data point at the edge of calculated values will be carried into the uncalculated area. This prevents continuation of slopes in the grids, as shown below.

Use Global Extrapolation: This option extrapolates the grids where null values exist, but unlike the Apply Extrapolation option, this option will search for localized slopes in the grid file and attempt to carry the slopes into the uncalculated area. This is shown in the below image.

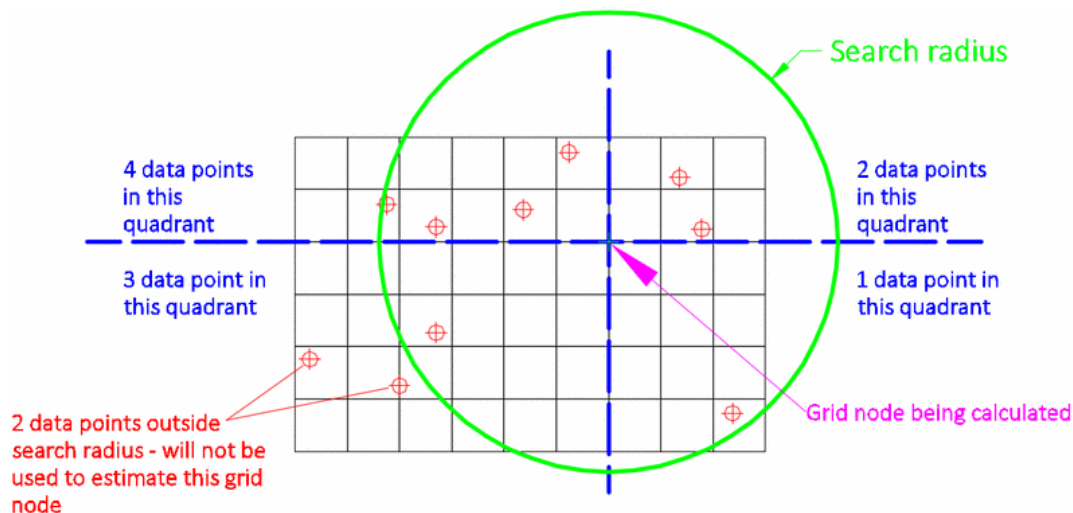


Max Triangle Length: This value sets the maximum triangle length for the Triangulation and Polynomial modeling methods.

Search Radius: This value sets the maximum search radius for data points. For example, if this value is set to 10,000, any data points beyond 10,000 ft/meters from a given grid node will not be used to estimate the value of that grid node. This concept is shown in the below image.

Max Samples: This value sets the maximum number of data points to use to estimate the value at a grid node. If this value is set to 20, only the 20 data points nearest the grid node being calculated will be used.

Min/Max Quadrant: When the program calculates the value of a grid node, the area around the grid node is divided into four quadrants, as shown below. These Min/Max Quadrant values set the minimum/maximum number of data points to use from each quadrant. This concept is shown in the below image. If the Min Quadrant value is set to 5, a value will only be assigned to the grid node if at least 5 data points are found within all four quadrants. If the Max Quadrant value is set to 20, only the nearest 20 data points in each quadrant will be used for estimation.



Use Parameter Filter: This option will filter data points according to parameters as set in the Define Parameters command. The parameters may be set by clicking the **Set Parameters** button.

Remove: This button will remove the currently selected row from the spreadsheet.

Clear: This button will clear all data from the spreadsheet.

Add Row: This button will create a blank row in the spreadsheet. This can be useful when slightly modifying a row, as you can select an entire row, copy the contents with CTRL+C, paste the contents into the blank row with CTRL+V, and then make the necessary modifications.

Sort: This button will sort the grids alphabetically by strata name. Note that the order in which the grids are listed does not actually have an effect on the grids.

Move Up/Down: These buttons will move the currently selected row up/down in the list.

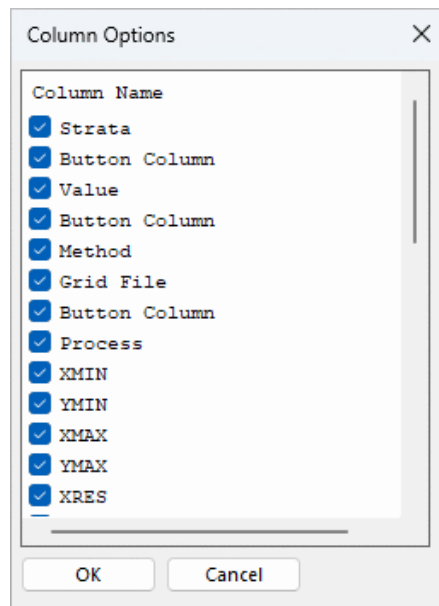
On/Off: This button will toggle the Processing status of the currently selected row(s). Note that the Processing

status controls if the grid is actually created when the Make Grids button is clicked.

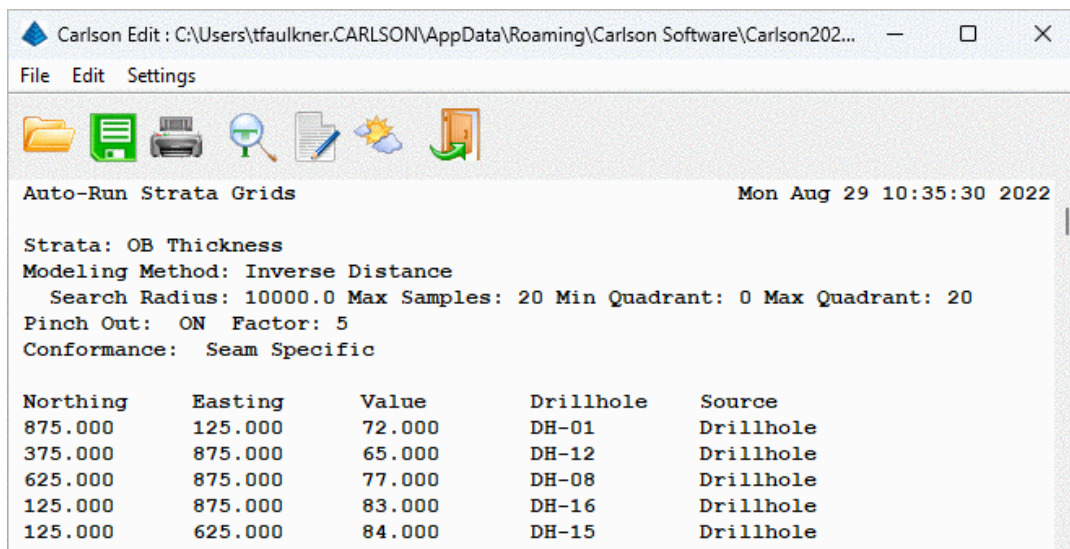
Make Grids: This button will create the grids listed in the spreadsheet.

Change Directory: This button will allow you to set the folder location for the grid files.

Column Options: This button will open the below dialog, which controls which columns are shown in the spreadsheet.



Create Data Points Report: When enabled, this option will output a text report of the data points that were used to create each grid file, as shown below.



Run GFU Macros at Completion: This option will run a Grid File Utilities macro (.gfu file) after making the grids. This file can be made with the Grid File Utilities command. This allows you to quickly customize grid files after they have been generated. When this option is enabled, you will need to click the **Select File** button to select the .gfu file.

Prompts

Select drillholes and strata elevation polylines.

Select objects: *select the drillhole symbols*

Auto-Run Grid Files dialog

Pulldown Menu Location: StrataCalc in Geology

Keyboard Command: chgrid2

Run Strata Grids AutoRun

This command will run the predefined RUN file for making strata grid files. This allows for easier definition of the Autorun macro, without having to wait for the modeling and drillhole processing time.

The Define Strata Auto-Run is where you build the macro file containing the grids to create. It does not have to create them at that time, it just prepares the file to be run at a later time. It is optional to select the drillholes at the beginning of the Define routine. Just hitting Enter will bring up the input dialog.

Run Strata Grids Auto-Run takes the predefined *.RUN file (from Define Strata Grids Auto-Run) and processes it, creating the grid files. As mentioned above, with the previous version in very large datasets, it sometimes took several hours after defining the Auto-Run, to actually select the Make Grids button. Now with this split into 2 routines, you can simply load up the predefined *.RUN file, select Make Grids and leave it over lunch, or overnight, while the modeling calculates the new grids.

Prompts

Select the RUN file dialog

Select drillholes and strata elevation polylines.

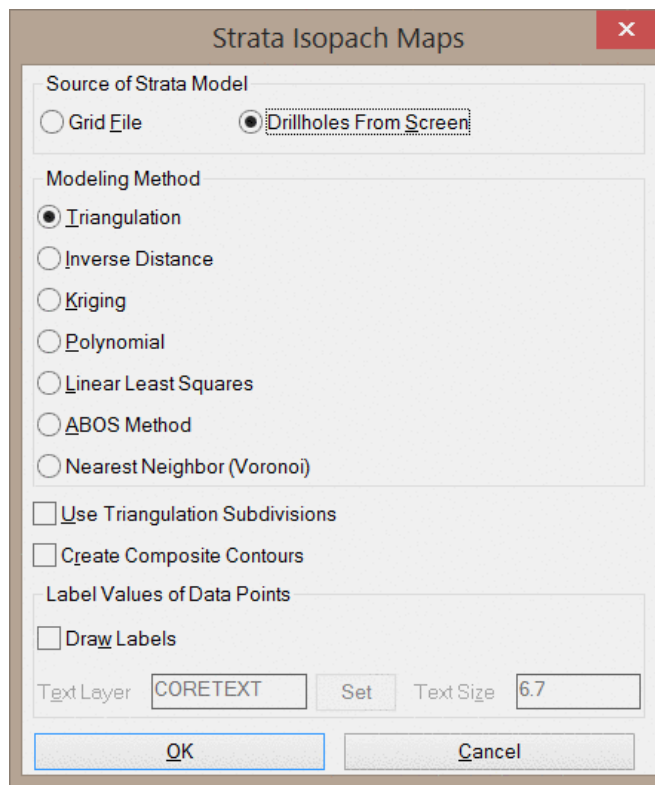
Select objects: *select the drillhole symbols*

Pulldown Menu Location: StrataCalc

Keyboard Command: chgrid3

Strata Isopach Maps

This command will create contours of various strata attributes, either directly from the drillhole data or from existing grid files. When the command is executed, the below dialog will appear.



Source of Strata Model: This option determines the source of the contour information.

The **Grid File** option will use an existing grid file to create the contours. When this option is used, the command is no different than the Contour from Grid File command.

The **Drillhole from Screen** option will use the drillholes in the drawing to create a temporary grid file and contour from that grid. When this option is used, the prompts will be similar to that of the Make Strata Grid Files command, followed by the Contour from Grid File command.

See the above links for explanations of each routine.

Prompts

Select drillholes, channel samples and strata polylines.

Select objects: *pick the drillholes*

Reading drillhole 19

Finding splits ...

Finding pinch out ...

Calculating seam stacking ...

Select the Inclusion perimeter polylines or ENTER for none.

Select the Exclusion perimeter polylines or ENTER for none.

Create composite contours [Yes/<No>]? n

Triangulating points ... 19

Assigning grid values> 33800

Inserted 336 contour vertices.

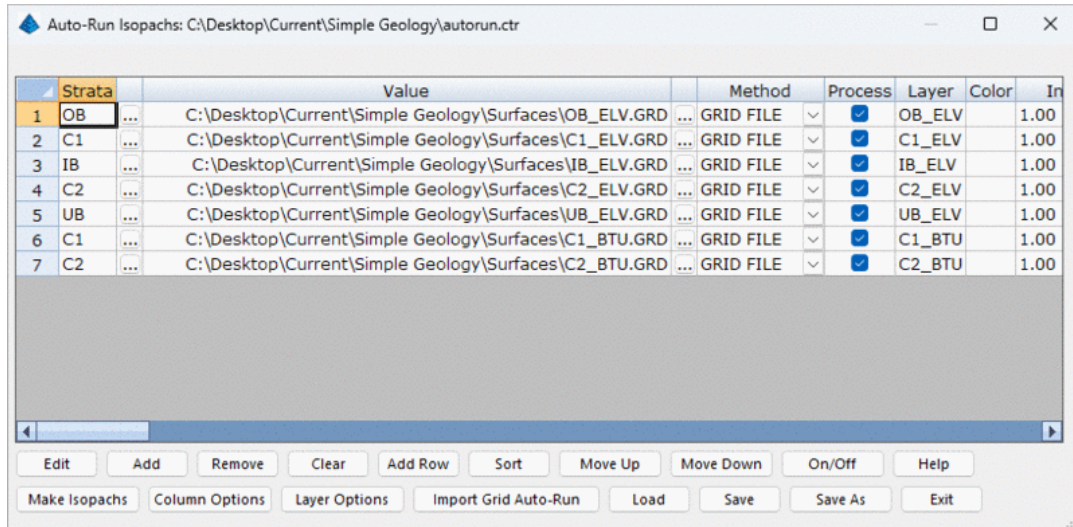
Pulldown Menu Location: StrataCalc

Keyboard Command: chiso

Prerequisite: Drillholes

Define Strata Isopach AutoRun

This command will create a list of contour maps to create. Contours may be extracted from existing grid files, or they can be generated directly from drillhole data. When the command is executed, you will be prompted to select drillholes (you do not have to select drillholes if you would prefer to use existing grids) and inclusion/exclusion perimeters. After this, the below dialog will appear.



Edit: Edits an existing entry in the list.

Add: Adds a new entry to the list.

Remove: Removes an entry from the list.

Clear: Removes all entries from the list.

Add Row: Adds a blank row to the list

Sort: Sort the list in alphabetical order by Strata Name.

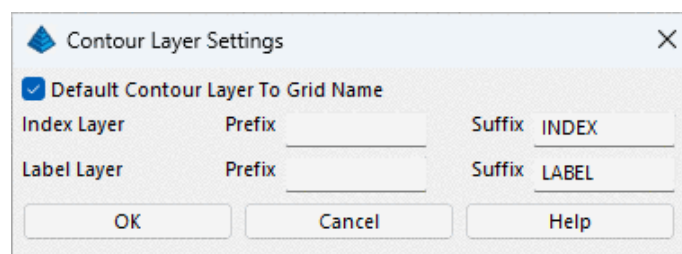
Move Up/Down: Moves the selected entry up/down in the list

On/Off: Changes the Processing status of the entry. When turned off, the entry will not be calculated.

Make Isopachs: Creates the contour maps.

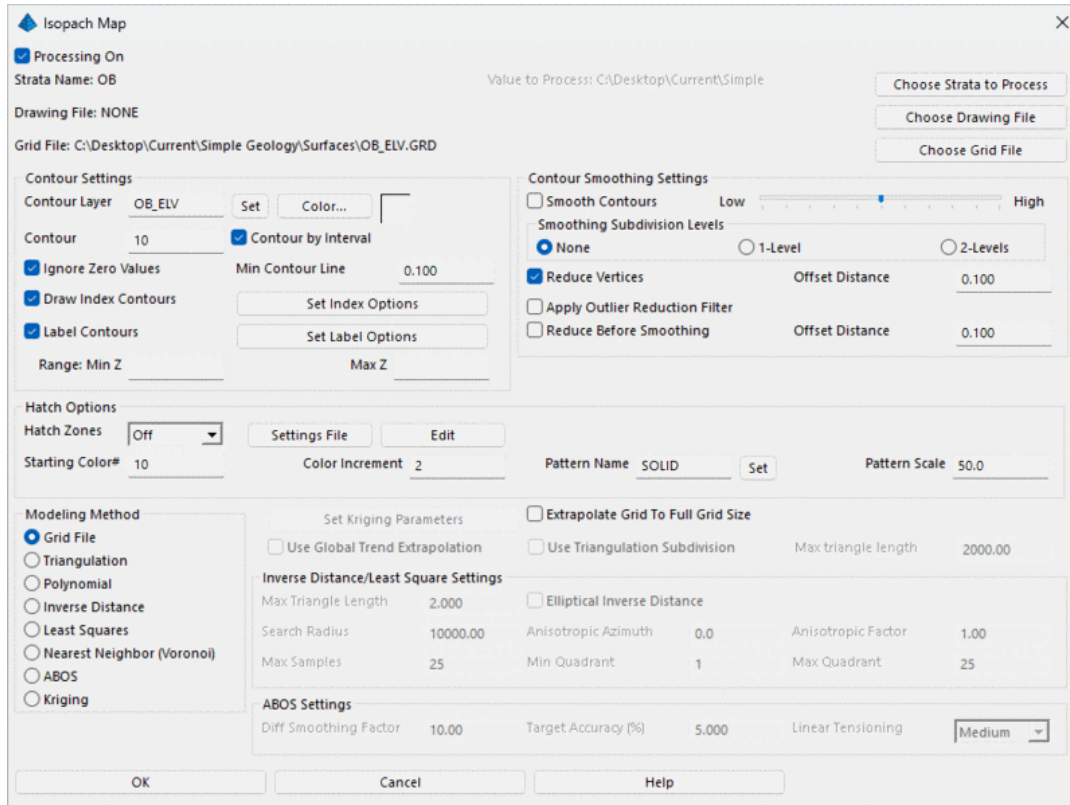
Column Options: Allows you to toggle the visibility of each column in the main dialog.

Layer Options: Opens the below dialog to control automatic layer naming. With this dialog, you can automatically set the contour layer to match the grid/strata names, as well as set prefixes/suffixes for the index contours and labels.



Import Grid Auto-Run: This option will load in a list of grids created from the Define Strata Grids Auto-Run command. When loading one of these files, it is important to note that generic settings will be applied to each grid in the list. It will be important to edit each entry to use appropriate contour intervals, colors, etc.

When you add/edit an entry, the below dialog will appear.



- **Processing on:** This options controls if the contour map will be generated or not.
- **Strata Name:** This will display the strata name if isopaching from drillholes. If isopaching a grid, then this will be blank. You may select a new strata with the Choose Strata to Process button.
- **Drawing File:** This option allows you to export the contour map to a new drawing file by clicking the Choose Drawing File button. If not specified, the contour map will be drawn in the current drawing.
- **Grid File:** This option will specify the grid file to extract the contours from. This option will only be enabled if the **Modeling Method** is set to Grid File.

The Contour Settings, Contour Smoothing Settings, and Hatch Options are described in the Contour from Grid File command.

The Modeling Method and associated settings are described in the Define Strata Grids AutoRun command.

Prompts

Select drillholes, channel samples and strata polylines or ENTER for None. Select Enter if isopaching only grid

files.

Select objects: *all*

61 found

Select objects:

Reading drillhole 194

Finding splits ...

Finding pinch out ...

Calculating seam stacking ...

Select the Inclusion perimeter polylines or ENTER for none.

Select the Exclusion perimeter polylines or ENTER for none.

Pulldown Menu Location: StrataCalc

Keyboard command: chiso2

Prerequisite: Drillholes or grid files

Run Isopach Maps AutoRun

This command will execute the predefined Auto-Run Isopach CTR file. It will prompt to select the file, inclusion/exclusion boundaries and the drillholes, then go right into generating the isopachs. It skips the Define Auto-Run step.

Prompts

Select Inclusion polyline (Enter for none):

Select Exclusion polyline (Enter for none):

Select drillholes, channel samples and strata polylines or ENTER for None.

Select objects: *all*

61 found

Select objects:

Reading drillhole 195

Finding splits ...

Finding pinch out ...

Calculating seam stacking ...

Triangulating points ... 195

Assigning grid values > 2600

Contouring elevation 4922.0000000

Inserted 35695 contour vertices.

Reading cell > 194032

Pulldown Menu Location: StrataCalc

Keyboard Command: chiso3

Prerequisite: The CTR file from Define Isopach Maps Auto-Run

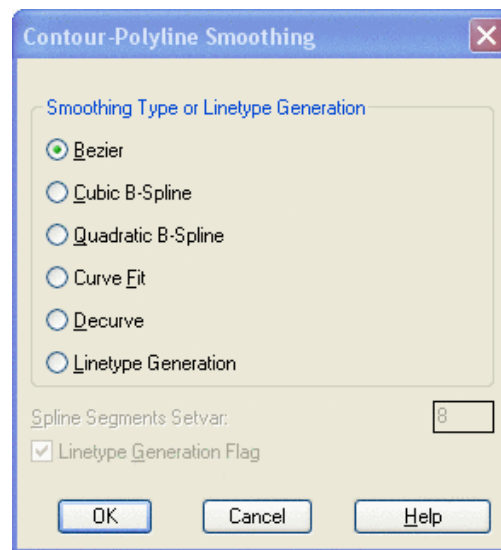
Smooth Contours

This command has options for applying smoothing to polylines. Select the radio button for the smoothing option you want to apply. If you use Quadratic B-Spline type smoothing or Cubic B-Spline type smoothing, the *Spline Segments* AutoCAD system variable is relevant. The Curve Fit option provides the least smoothing, and the Cubic B-Spline option applies the most. Another effective way of smoothing is by creating the contours from rectangular meshes using various grid resolutions. Increase the smoothing by lowering the grid resolution and decrease by raising the grid resolution. The Bezier option provides an incremental type of smoothing. The Linetype Generation option turns on the *Ltype Gen* flag for the selected polylines. For more information on this option and the spline smoothing options, look up the *PEDIT* command in the AutoCAD Reference Manual. After selecting the *OK*

button the routine will prompt for needed values.

Bezier smoothing is also embedded in many of the routines that create contours. Bezier smoothing applies the Bezier smoothing algorithm to polylines. This smoothing technique has two advantages over Spline or Curve Fit smoothing. One is that a Bezier smoothed polyline will pass through all of the vertices in the original polyline, while a Spline smoothed polyline only curves towards the original vertices and can pull away from vertices at sharp corners. Hitting all the original vertices can be an important feature in contour maps for maintaining the exact location of the contours. Another benefit of Bezier smoothing is the ability to control the looping and vertex factors. A higher looping factor increases the curving effect. Use this setting with some care, as too high a looping factor may cause nearby contour lines to cross after the smoothing has been applied.

Vertex reduction can also be applied along with the smoothing. This avoids having to create smoothed polylines with numerous vertices and then having to reduce these vertices in a second step. Be sure not to make the cutoff offset for reduction too high or you can negate or even reverse the smoothing effect. One disadvantage to Bezier smoothing is that it cannot be decurved like the other smoothing techniques.



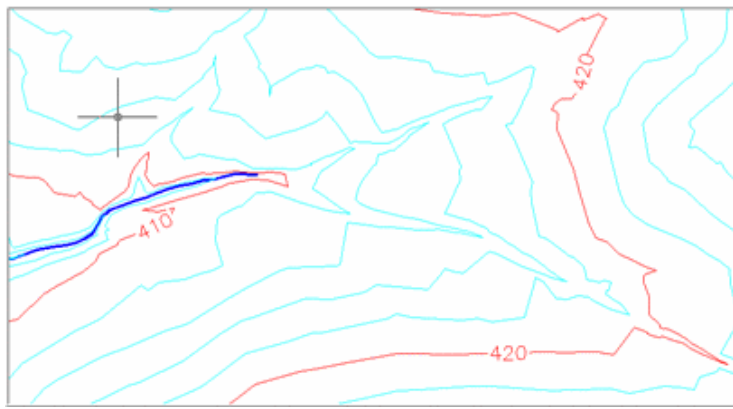
Prompts

Enter the looping factor (1-10) <5>: *press Enter* This determines the extent of curving. 1- least curvy, 10 - most curvy.

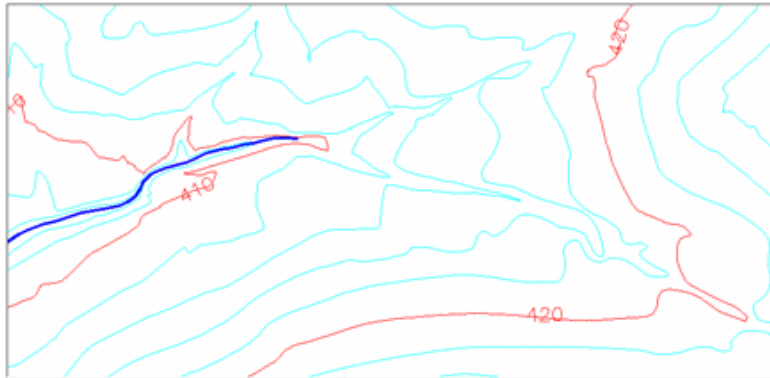
Enter the offset cutoff <0.05>: *press Enter* This value is the maximum shift distance for vertices reduction. A higher value removes more vertices.

Select polylines to smooth.

Select objects: *pick polylines*



Before Smoothing



After Smoothing

Pulldown Menu Location: Surface >> Modify Contours

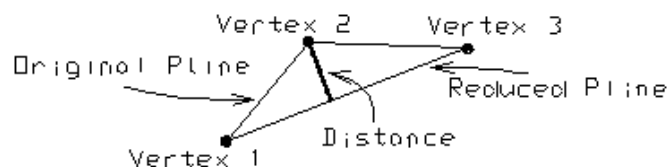
Keyboard Command: Smooth

Prerequisite: Create Contour lines to smooth

Reduce Contour Vertices

Contouring and smoothing often creates an explosion in file size due to the many vertices it adds to the individual contour polylines. Fortunately, many of these vertices are very close together, some of which can be removed with no visible effect on the contour polylines themselves. *Reduce Contours Vertices* can reduce the total number of vertices up to 90%. This has the benefits of a smaller drawing file, faster drawing loading, and faster regens.

This command removes vertices in a polyline that are within a user specified offset cutoff. The algorithm looks at three vertices at a time, and calculates the distance between the second point and the line from the first to the third point. If this distance is less than the user specified cutoff, the second point is removed. In theory, reducing the polyline vertices should not shift the polyline more than the user's cutoff distance. The default for this cutoff is one tenth of a foot. Increasing the cutoff will remove more vertices while decreasing it will more closely preserve the original contour line. When combining vertex reduction with smoothing, it is suggested to smooth before reducing, although it can be done the other way around.



Prompts

Enter the offset cutoff <0.1>: .3

Select polylines to reduce. *select polylines*

Select objects: *press Enter* to conclude selection

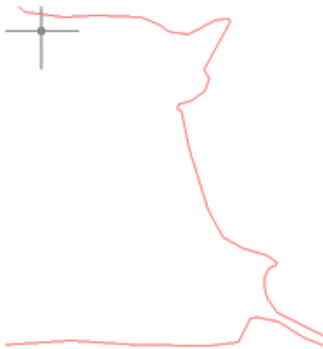
Processed polylines: 1

Total number of vertices: 1125

Number of vertices removed: 939



Before Vertex Reduction



After Vertex Reduction

Pulldown Menu Location: Surface >> Modify Contours

Keyboard Command: reduce

Prerequisite: Polylines (contours) with vertices to reduce

Edit Contours

This command revises a segment of a contour polyline. Begin by picking a point on the contour where you want to start editing. Then pick new points for the polyline. When finished picking new points, press Enter and then pick a point on the contour to connect with the new points. The polyline segment between the start and end points is then replaced with the new points.

If there is a triangulation file associated with the contours, then the command prompts for whether to update the triangulation surface file to match the contour edits. When this option is used, data points are added to the triangulation surface along the edited contour segment to make the triangulation surface match the contour line. Existing triangulation source data is retained. So the updated triangulation is the combination of the original source data and the additional points from Edit Contours. One way to get a triangulation surface associated with the contours is to use the Triangulate & Contour command with both Write Triangulation File and Draw Contours

options active.

Note: If the triangulation association is not used, then this routine has no effect on the actual triangulation or grid surface model file that the contours may have been drawn from. It only revises the drawn contour or polyline on the screen. If the contours are later regenerated from this file, the edits will be discarded.

Prompts

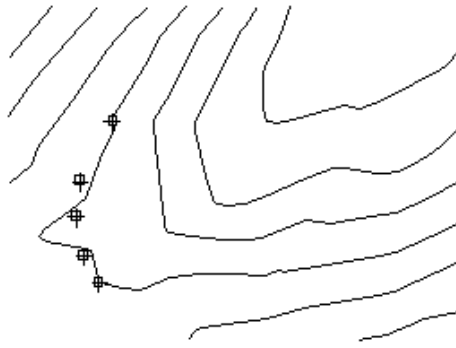
Select contour to edit: *pick the contour polyline at the place to start editing*

Pick intermediate point (Enter to End): *pick a point*

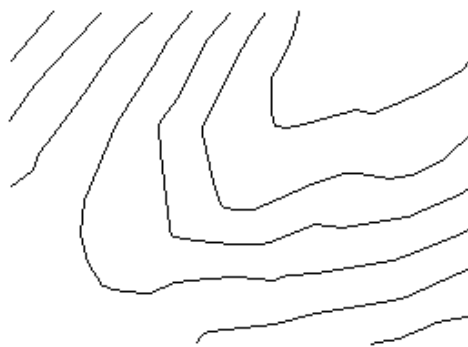
Pick intermediate point ('U' to Undo, Enter to End): *pick a point*

Pick intermediate point ('U' to Undo, Enter to End): *press Enter*

Pick reconnection point on contour: *pick the contour polyline at the place to join*



Edit this contour by picking new points



Contour with segment replaced with new points

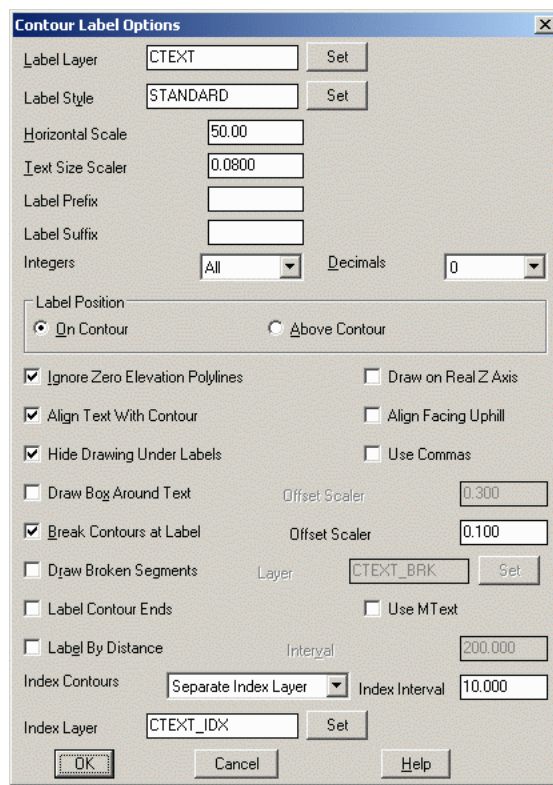
Pulldown Menu Location: Surface >> Modify Contours

Keyboard Command: editctr

Prerequisite: Polylines with elevation (contour polylines)

Contour Elevation Label

This command can be used to simultaneously create elevation labels on a group of contour polylines at elevation. First the command starts with a dialog with the label options. Then to place the labels, pick two points crossing the contour polylines at the desired label location. The program will find all the contour polylines that intersect the picked line (defined by the two picked points) and will place labels at the intersection point of each contour. A second crossing line can be initiated immediately, so multiple areas can be quickly labeled while remaining in the command. Alternatively, you can type P for Polyline at the Command prompt and select a polyline. Then the program finds all the intersections between the selected polyline and the contours and places labels at these intersections. The actual "z" elevation of the contour line determines the label value.



Label Layer specifies layer name for the contour labels that will be created.

Label Style specifies the text style to be used for labels.

Horizontal Scale is used in conjunction with the Text Size Scaler to determine unit height of the contour labels.

Text Size Scaler is a scaler that will be multiplied by the horizontal scale to set the actual text height of the labels in AutoCAD units.

Integers controls how many digits to label to the left of the decimal. For example, if all contours are in the 5000's, then setting for three digits would label the 5280 contour as 280.

Decimals sets the decimal precision for the labels to be created.

Label Position determines the label position in relation to the contour polyline.

- **On Contour** centers the label on the contour line.
- **Above Contour** places the label above the contour line. If this option is used, the options for Break Contours at Label and Draw Broken Segments become inactive.

Ignore Zero Elevation Polylines enables the routine to filter out all entities with an elevation of zero.

Hide Drawing Under Labels activates a text wipeout feature that will create the appearance of trimmed segments at the contour label, even though the contour line is still fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels, yet the contour lines themselves remain contiguous. This feature will also hide other entities that are in the immediate vicinity of the contour label.

Align Facing Uphill makes the label parallel to the contour and flips the label so that it reads facing uphill. Otherwise, the labels are made to face up relative to the current screen view. When this option is on, the program prompts for a triangulation surface file that should match the surface the contours represent.

Use Commas adds a comma into the labels for the thousands place such as "5,000" instead of "5000".

When **Align Text with Contour** is checked, contour elevation labels will be rotated to align with their respective contour lines.

When **Break Contours at Label** is checked, the contour lines will be broken and trimmed at the label location for label visibility.

When **Draw Broken Segments** is checked, segments of contours that are broken out for label visibility will be redrawn as independent segments. Specify the layer for these broken segments in the box to the right of this toggle.

Label Contour Ends creates labels off the ends of the contours.

Label By Distance places the labels by distance along the contour. The user is not prompted for screen picks of contour crossing when this option is used.

- **Interval** sets the distance interval to be used between labels on each contour.

When **Draw Box Around Text** is checked, a rectangle will be drawn around the elevation labels. The Offset Scaler controls the size of the rectangle.

The **Draw On Real Z Axis** chooses between creating the text entities at the elevations of the contours or at zero elevation.

The **Use MText** chooses between creating MText and DText label entities.

Index Contours: Label All will label both index and intermediate contours with the same settings. Label Index Only labels only the index contours. Separate Index Layer will label both index and intermediate contours with the index labels on a different layer.

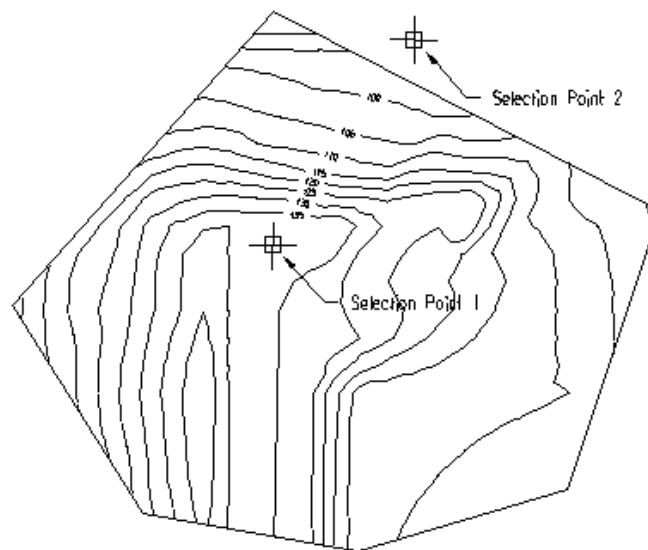
Prompts

Contour Label Options Dialog Opens Select the desired options and press OK.

Define a line which slices the contours at the desired label locations.

Pick 1st point (P-Polyline, Enter to end): *pick a point*

Pick 2nd point: *pick a point*



By selecting two points the contour lines that cross the line defined by the two points are labeled.

Pulldown Menu Location: Surface >> Contour Labels

Keyboard Command: gclabel

Prerequisite: polylines with elevation (contour polylines)

Move Label Along Contour

This command slides an existing contour label along a contour, maintaining its alignment with the contour. After moving the label, you can type F for Flip at the Command prompt to rotate the label orientation by 180. The label must have originally been created with the *Break Contours at Label* option *Off*. If the option to *Hide Drawing Under Labels* was used when the label was created, the wipeout will move with the label when using this command.

In addition to moving a label, an existing label can be copied and placed at a new position along the contour by using the Copy option at the first prompt.

Prompts

Copy/<Select contour label to move>: *Pick label*

Pick new contour label position: *Move mouse to relocate label*

Flip last/<Select contour label to move (Enter to end)>: *press Enter*

Pulldown Menu Location: Surface >> Contour Labels

Keyboard Command: move_ctr_label

Prerequisite: generated contour labels

Flip Contour Labels-Text

This command individually rotates each of the selected text entities by 180 degrees.

Pulldown Menu Location: Surface >> Contour Labels

Keyboard Command: fliptext

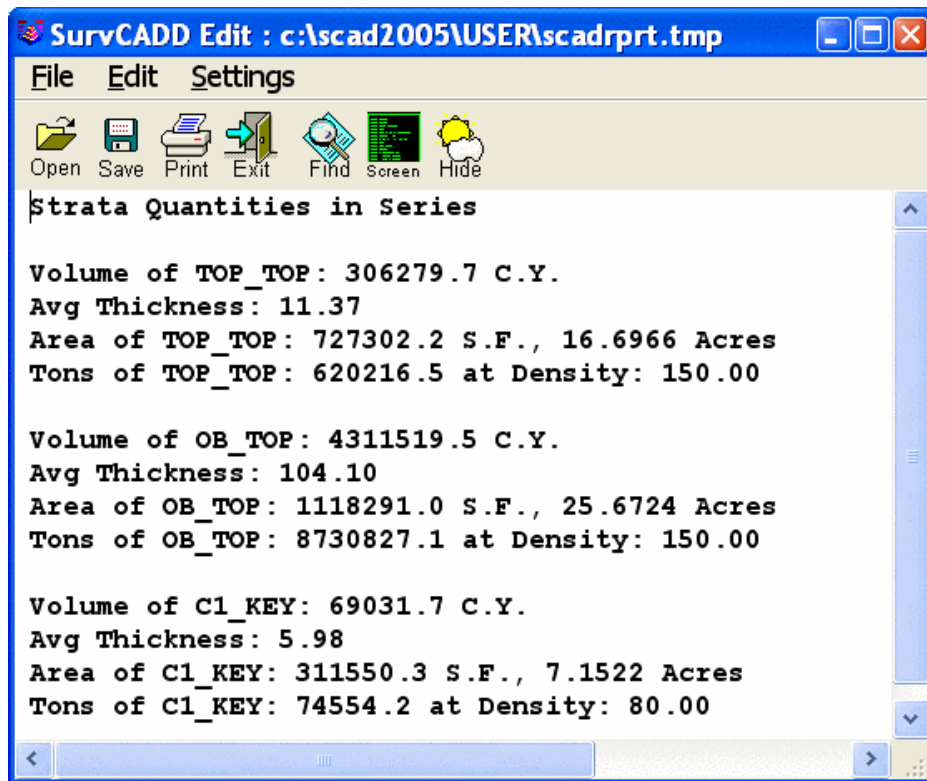
Prerequisite: Text labels on contours

Strata Quantities in Series

For each strata in the drillholes, this command can calculate the volume, tonnage, and average attribute values within a specified area. The possible attributes depend on the configuration set in Define Drillhole. For example, attributes might be moisture, sulfur, BTU, etc. After reading in the selected drillholes, Strata Quantities cycles through all the possible quantities for every strata from top to bottom. Only the desired quantities are calculated and the result is printed out in volume, tons and quality.

Strata Quantities calculates from a 3D grid of the values. The grid resolution is specified in a grid resolution dialog box. A higher resolution yields more accurate results, but slows down the routine.

The area for gathering quantities defaults to the limits of the selected surface entities and drillholes. To control the calculation area, multiple closed polylines for areas to include and/or exclude can be selected. Also, in the first selection for surface entities and drillholes, any selected polylines in the PILLARS layer will be made into exclusion areas and polylines in the PERIM layer will be made into inclusion areas. This is useful for getting quantities from drillholes among pillars and perimeters.



Prompts

Source of surface model [File/<Screen>]? *F*
 Select drillholes
 Select objects: Specify opposite corner: 19 found
 Select objects:
 Reading drillhole 19
 Finding splits ...
 Finding pinch out ...
 Calculating seam stacking ...
 Reading cell> 194032
 Pass> 7 Null Z values left> 0
 Select the Inclusion perimeter polylines or ENTER for none.
 Select objects: 1 found
 Select objects:
 Select the Exclusion perimeter polylines or ENTER for none.
 Select objects:
 Choose modeling method [<Triangulation>/Inverse dist/Kriging/Polynomial/LeastSq/ABOS]?
 Apply global trend to strata extrapolation [Yes/<No>]? *N*
 Use Triangulation Subdivision [Yes/<No>]? *N*
 Pre-processing grid cells ...
 Ignore zero attributes [<Yes>/No]?
 Triangulating points ... 19
 Assigning grid values> 194000
 Pass> 148 Null Z values left> 0
 Calculate quantities from strata TOP_TOP values [<Yes>/No]? *Y*
 Processing cells ...
 Volume of TOP_TOP: 306279.7 C.Y.
 Avg Thickness: 11.37
 Area of TOP_TOP: 727302.2 S.F., 16.6966 Acres

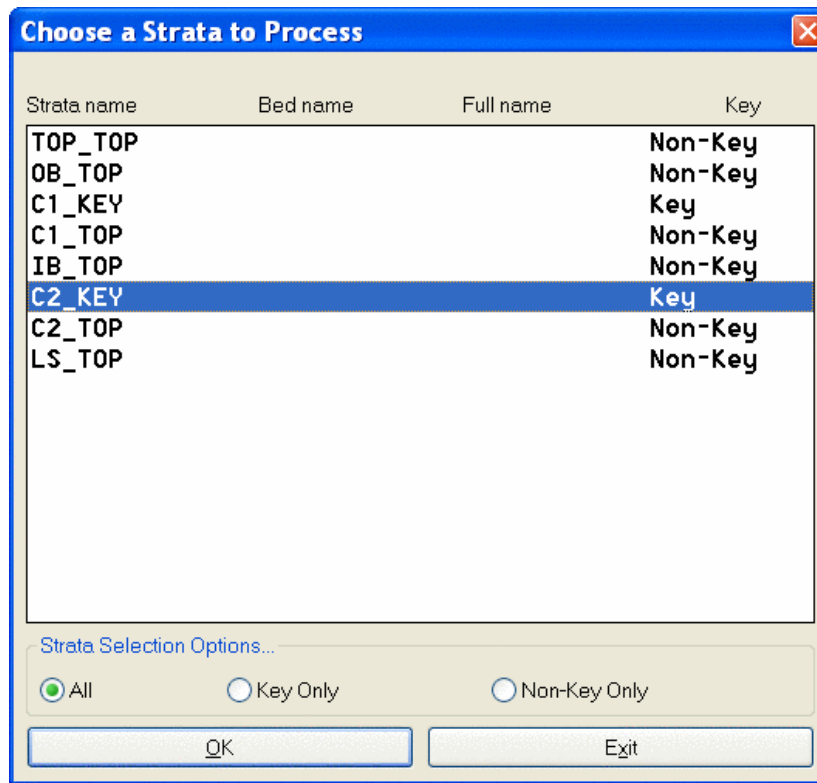
Enter the density of strata TOP_TOP in lbs/ft³ <80.000>: 150
Tons of TOP_TOP: 620216.5 at Density: 150.00
Calculate qualities from strata TOP_TOP values [<Yes>/No]? N
Triangulating points ... 19
Assigning grid values> 194000
Pass> 148 Null Z values left> 0
Calculate quantities from strata OB_TOP values [<Yes>/No]? Y
Processing cells ...
Volume of OB_TOP: 4311519.5 C.Y.
Avg Thickness: 104.10
Area of OB_TOP: 1118291.0 S.F., 25.6724 Acres
Enter the density of strata OB_TOP in lbs/ft³ <80.000>: 150
Pulldown Menu Location: StrataCalc
Keyboard Command: chquan

Selected Strata Quantities

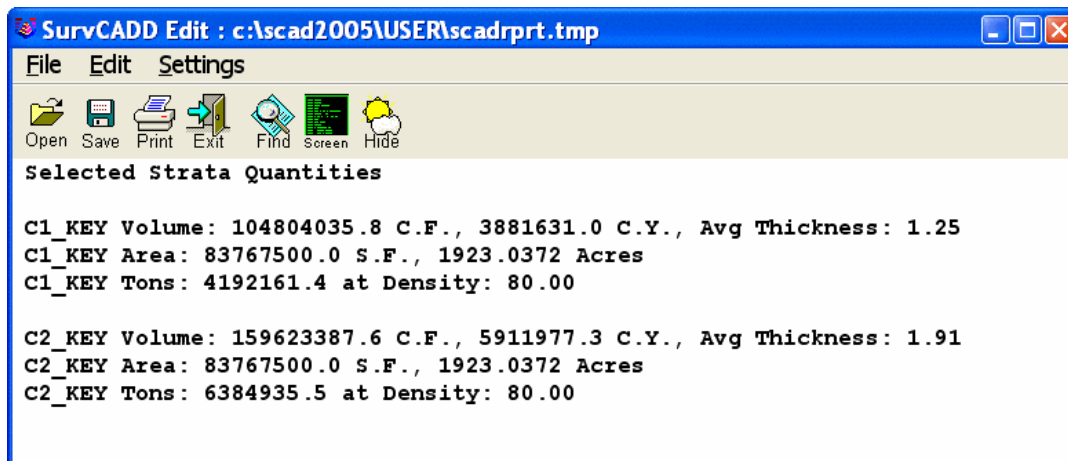
For each strata in the drillholes, this command can calculate the volume, tonnage, and average attribute values within a specified area. The possible attributes depend on the configuration set in Define Drillhole. For example, attributes might be moisture, sulfur, BTU, etc.

The area for gathering quantities defaults to the limits of the selected surface entities and drillholes. To control the calculation area, multiple closed polylines for areas to include and/or exclude can be selected. Also, in the first selection of drillholes, any selected polylines in the PILLARS layer will automatically be made into exclusion areas and polylines in the PERIM layer will be made into inclusion areas. This is useful for getting quantities from drillholes among pillars and perimeters.

This command is similar to the Strata Quantities in Series command. The difference is that this command does not process the strata in sequence from top to bottom. Also this command does not model the ground surface. Instead Selected Strata Quantities allows you to select the strata to process from the list of available strata. The advantage is that when there are many strata, you don't have to cycle through all the top strata to get down to a specific strata. Since the ground surface is not calculated, the disadvantage of this command is that the outcrop for the selected strata is not automatically calculated. The volume for the selected strata is calculated using the strata thickness and the calculation area. To limit the volume calculations to be inside the outcrop, draw the outcrop as a closed polyline which can then be used as an inclusion perimeter.



Another advantage of this command versus Strata Quantities in Series is that composite quantities can be calculated. Composite quantities allow you to select multiple strata and add their thickness together for the volume report. Also the weighted average of the attributes is calculated.



Prompts

Select drillholes, channel samples and strata polylines.

Select objects:

Reading drillhole 19

Finding splits ...

Finding pinch out ...

Calculating seam stacking ...

Select the Inclusion perimeter polylines or ENTER for none.

Select objects:

Select the Exclusion perimeter polylines or ENTER for none.

Select objects:

Choose modeling method [<Triangulation>/Inverse dist/Kriging/Polynomial/LeastSq/ABOS]? /

Use inverse distance to which power [First/<Second>/Third/Other]?

Use elliptical inverse distance [Yes/<No>]?

Pre-processing grid cells ...

Ignore zero attributes [<Yes>/No]?

Create composite quantities [Yes/<No>]?

Calculating grid by inverse distances 33880...

Processing cells ...

Enter strata C1_KEY density in lbs/ft³ <80.000>:

C1_KEY Volume: 104804035.8 C.F., 3881631.0 C.Y., Avg Thickness: 1.25

C1_KEY Area: 83767500.0 S.F., 1923.0372 Acres

C1_KEY Tons: 4192161.4 at Density: 80.00

Calculate qualities from strata C1_KEY values [<Yes>/No]? Y

Press ENTER to continue.

Calculating grid by inverse distances 33880...

Processing cells ...

Enter strata C2_KEY density in lbs/ft³ <80.000>:

C2_KEY Volume: 159623387.6 C.F., 5911977.3 C.Y., Avg Thickness: 1.91

C2_KEY Area: 83767500.0 S.F., 1923.0372 Acres

C2_KEY Tons: 6384935.5 at Density: 80.00

Calculate qualities from strata C2_KEY values [<Yes>/No]? Y

Pulldown Menu Location: StrataCalc

Keyboard Command: chquan2

AutoRun Strata Quantities

This command stores a list of strata to process, the value to process (volume or attribute) and the process method. It is the Auto-Run macro of the Selected Strata Quantities command. By clicking the Calculate Quantities button, the quantities for each item in the list can be calculated automatically without specifying all these options again. This routine allows you to save many steps when recalculating volumes for updated drillholes or different inclusion areas. These processing settings are stored in a file with a .VOL file extension.

Strata Quantities

Processing On Ignore Zero Values

Strata Name: C1_TOP

Value to Process: VOLUME

Choose Strata to Process

Modeling Method

Triangulation Inverse Distance Kriging Polynomial LeastSq ABOS

Use Triangulation Subdivision Use Global Trend Extrapolation

Inverse Distance Power: 2.000

Set Kriging Parameters

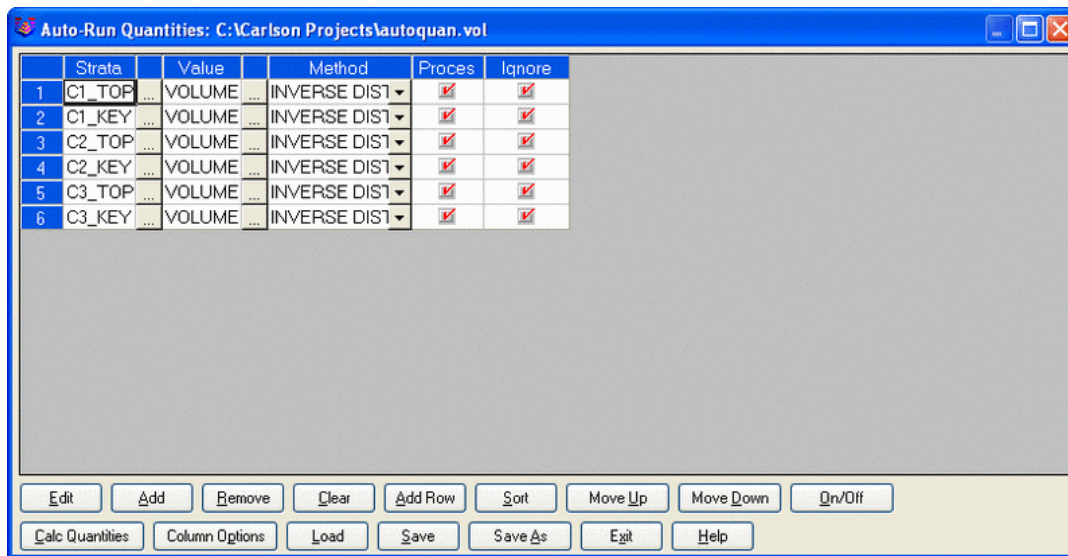
Elliptical Inverse Distance

Anisotropic Azimuth: 0.0 Factor: 1.00

Search Radius: 10000.0 Max Samples: 20 Min Quadrant: 0 Max Quadrant: 20

Diff Smoothing Factor: 10.00 Target Accuracy (%): 5.00 Linear Tensioning: Medium

OK Cancel Help



Prompts

Select drillholes, channel samples and strata polylines.

Select objects: Specify opposite corner: 48 found

Select objects:

Reading drillhole 42

Finding splits ...

Finding pinch out ...

Calculating seam stacking ...

Calculate VOLUME for strata [<Yes>/No]? Y

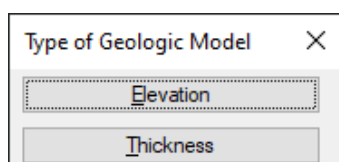
Auto-Run Quantities dialog

Pulldown Menu Location: StrataCalc

Keyboard Command: chquan3

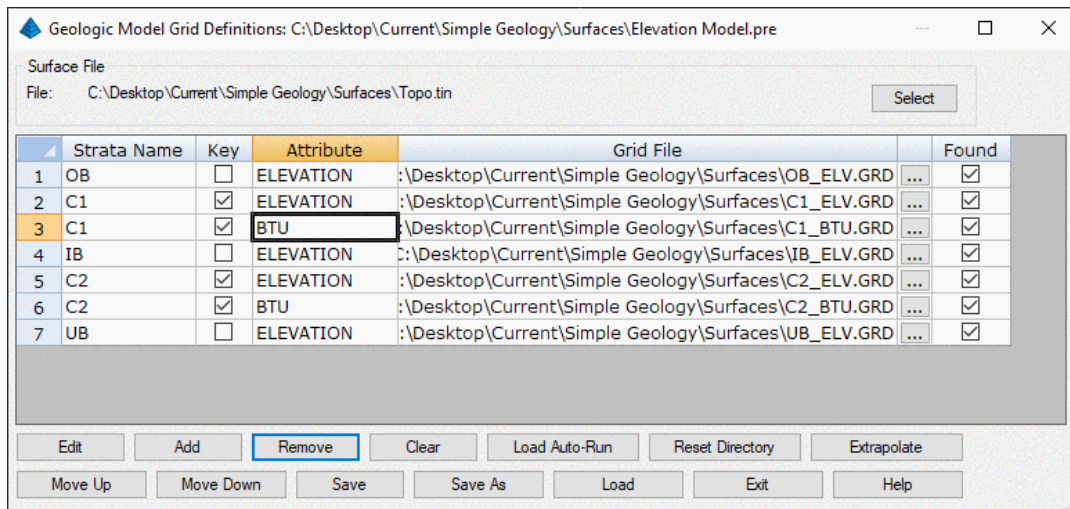
Define Geologic Model

This command assigns grid files to strata names in sequence. This list of grid files represents the geologic model of the strata. The Make Strata Grid File or Auto-Run Strata Grids commands can be used to create the grids before running this routine. The Geologic Model grid assignments are stored in a user-specified file name with the .pre file extension. There are two types of Geologic Model files: elevation and thickness. When creating a new Geologic Model file, you have the choice between elevation and thickness. Once the Geologic Model file is created, the type cannot be changed.

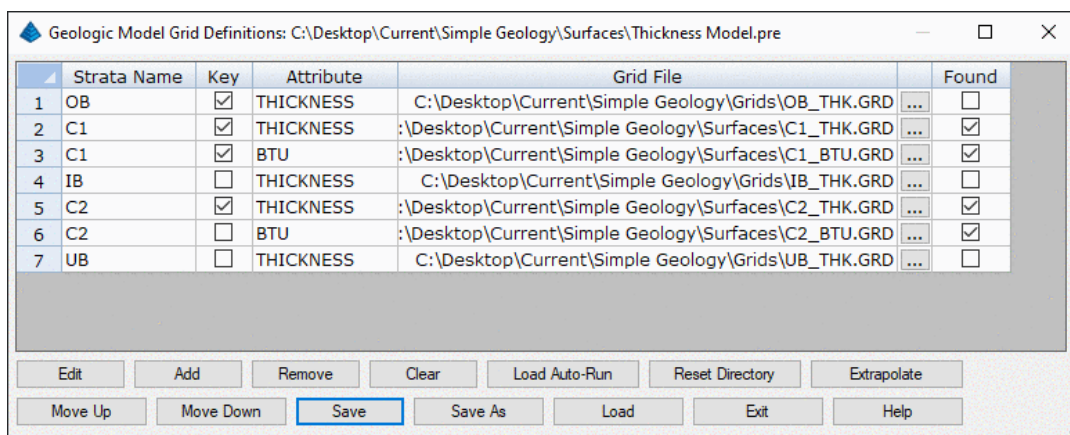


For elevation type Geologic Model files, the grid files define the bottom elevation surface model for the corresponding strata. A grid file for the ground surface model is also assigned in this routine. The surface file can be either a grid or a TIN file, but all other surfaces in the model must be grids. When the Geologic Model file is applied in StrataCalc volume calculations, the strata thickness is determined by comparing the current strata bottom elevation grid with previous upper strata or surface grid. Shown below is an elevation example. Note that these surfaces use a top-down dominance, meaning that if an upper surface (such as the topo) cuts down below a lower grid, the

lower grid will cease to exist in that area. In this way, you can model outcrops without having to modify the strata elevation grids to match the topo.



With thickness type Geologic Model files, the grid files represent the thickness for the corresponding strata. A surface grid file is not specified, and a flat surface is assumed. Without the ground surface elevation grid, the StrataCalc routines cannot find outcrops or adjust the strata thickness for variations on the surface. So the thickness grids that you supply the Geologic Model file must account for the outcrops and ground surface variations. To account for outcrops, you can set the thickness grids to zero in outcrop areas by using a closed polyline for the outcrop areas and running Grid File Utilities-Set Value with the Use Inclusion Areas option. A thickness model will not draw proper Fence Diagrams. To account for surface variations, you can make a grid of the ground surface and a bottom elevation grid of the top strata overburden. Then use Grid File Utilities to subtract the top strata elevation grid from the ground surface. The resulting difference grid is the top strata thickness. Shown below is a thickness example.



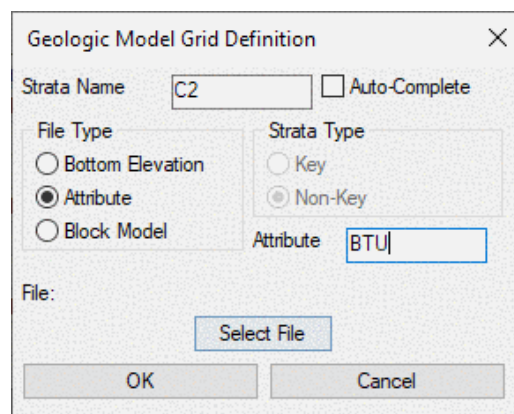
All these grid files should have the same location and resolution, though they can vary for some commands, such as Surface Mine Reserves. When grids have differing sizes/resolutions, the grids will be automatically resized to a common size/location, which can increase processing time. The resolution of the topo grid will be retained when these corrections are applied, effectively allowing one to model the topography with a higher resolution than the strata grids.

The model grids are specified in the dialog shown below. The surface grid is defined in the top of the dialog. To

assign the surface grid, pick the Select Grid File button in the upper right. Next there is a list of the strata names and the corresponding grid files. To add a strata grid, click the **Add** button. This brings up the below dialog where you enter the strata name and grid file. If you have multiple grids related to a single strata type (such as an elevation and an attribute), the Strata Name for all of those grids should use the exact same Strata Name.

Besides elevation and thickness grids, attribute grids such as BTU can be assigned to the strata names. In the Geologic Model Grid Definition dialog, click on the Attribute toggle and enter the Attribute Name. The strata in this list should be listed in top to bottom order. If a strata is out of order, highlight the strata in the list and click the **Move Up** or **Move Down** button.

The **Auto-Complete** option looks for patterns in the existing grids to fill out grids for another strata. For Auto-Complete, the strata name format must have an underscore with the name on the left and the type on the right such as "C1.KEY" with "C1" as the name and "KEY" as the type. For example, if the list of grids already has grids for strata C1.KEY of C1.bottom.grd and C1.BTU.grd, then when adding a new strata named C2.KEY, the Auto-Complete will look for C2.bottom.grd and C2.BTU.grd and automatically add these to the list.



The **Load Auto-Run** button will prompt you to select an Auto-Run Grid Definition File (.run). The program will then read the strata names from this file and build the Geologic Model Definition list. The **Reset Directory** button allows you to reassign the grid files directory in case the grid files have move since the Geologic Model file was created. Select the grid file at the top of the window, usually the first grid on the list. The surface grid will have to be redirected separately. The **Extrapolate** button will extrapolate and save all the grids to remove NULL values. This option will save time in StrataCalc routines that use the Geologic Model file and extrapolate the grids on-the-fly each time. There is an option to extrapolate by merging or flattening (pancaking) elevation grids with upper strata. The "pancaking" is an option using the Extrapolate function. It will assign the Nulls in a grid the elevation of the strata above. This change is saved to the .GRD file. The other option will just extrapolate the grid elevation out to the grid limits.

The purpose of this routine is to allow you to process stored grids in routines such as Surface or Underground Reserves and Fence Diagram. These commands have a Geologic Model option under the modeling methods. Otherwise these routines will read selected drillholes and calculate the strata models on-the-fly. The advantage to the Geologic Model is that you have more control over each of the geologic model grids. After creating the strata grid file, there are many routines that can be used to analyze and modify the grids to make sure that the grids model the strata the way that you want, and even create a mining model from modifying the geologic grids. For example, you can use Plot 3D Grid File, Grid File Utilities and Contour from Grid in the Civil Design module.

Finally, for users with the Ore Module that have created block models, they may be added into the Geologic Model for use with the Fence Diagram and Surface Reserves commands. There is an option, next to Elevation or Attribute for Block Model. The procedure here is to build the Geologic Model, interval by interval from the top down as usual. Then add in the block model for each interval, or strata that you have. It is just like the other attributes, in that it must have the same name as the strata it is referring to. There are options on the Fence Diagram and Surface Reserves screens to use the hatch fence by block model and to break out quantities by attributes.

Pulldown Menu Location: StrataCalc

Keyboard Command: precalc

Define Mining Model

This command creates a Mining Model file (.mmd) which stores a sequence of strata elevation grids along with strata attribute grids and a ground surface grid. The Define Mining Model works the same as the Define Geologic Model. See the Define Geologic Model section of the manual for a description of the functions.

The difference with the Mining Model is that the strata elevation grids are already for any adjustments including recovery, minimum key thickness to mine, minimum depth to mine, minimum separatable parting thickness and non-key thickness to add to the key. The Mining Model can be used in commands like Surface Mine Reserves and the program can directly process the strata elevation grids without having to apply those adjustments as it needs to for a Geologic Model.

Pull down Menu Location: StrataCalc

Keyboard Command: defmine

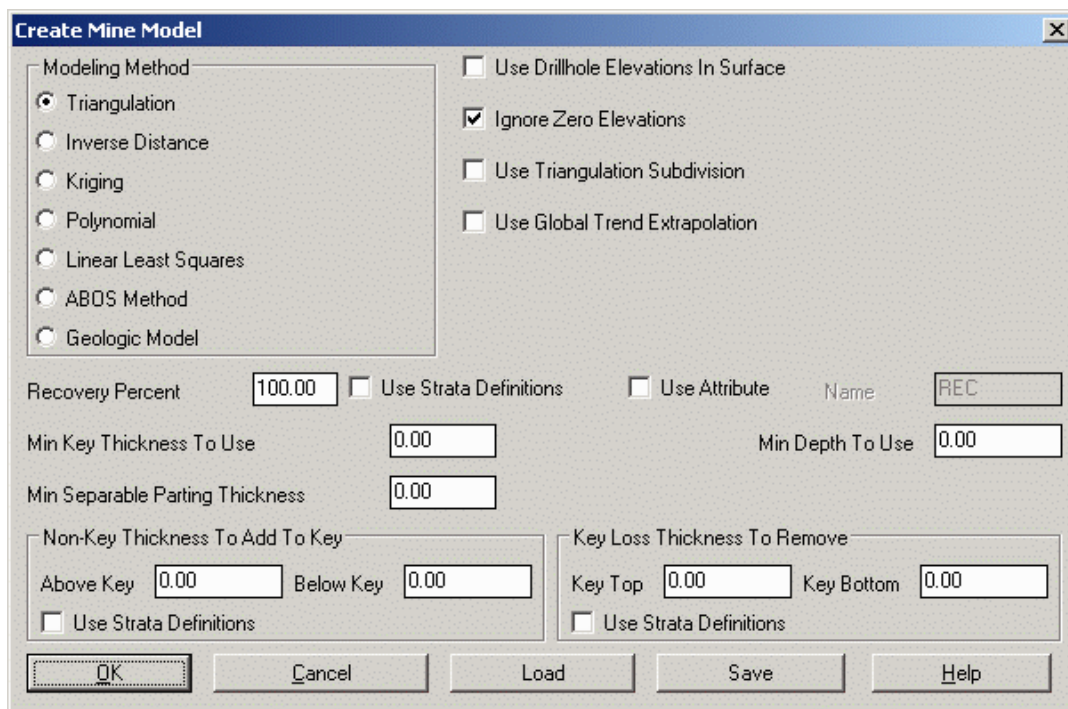
Prerequisite: grid files

Geologic To Mining Model

This command creates a Mining Model file (.mmd) by modeling drillholes or processing a Geologic Model. The Mining Model adjusts the strata elevations for recovery, minimum key thickness to mine, minimum depth to mine, minimum separatable parting thickness, key loss thickness to remove and non-key thickness to add to the key.

The input of the strata grids and the adjustments to create the Mining Model are a similar process to the Surface Mine Reserves command. See the Surface Mine Reserves section of the manual for a description of the input strata grids and the adjustments.

There are a couple key advantages of applying the strata elevation adjustments and saving them to the Mining Model. One purpose is to save time when processing the model in Surface Mine Reserves because the program does not need to take time to apply the adjustments each time the model is run. Another advantage is the mining model can be inspected with various grid tools such as Grid Inspector to verify that the adjustments are applied correctly.



Pull down Menu Location: StrataCalc

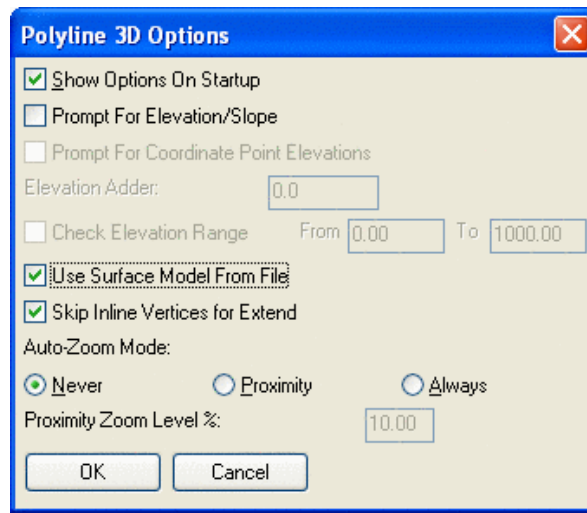
Keyboard Command: minemod

Prerequisite: Drillholes or Geologic Model file

Draw Fault Line

This command creates a 3D polyline that represents a fault line which is used by *Fence Diagram* in the intersection mode, or in the Make Strata Grid File command. This command is very similar to Carlson Draw > 3D Polyline command. In addition to elevation for each vertex of the polyline this command prompts for fault shift. This shift value is the amount of elevation difference from left to right across the fault line in the direction of the line is drawn. Note that a positive shift amount means that the right side of the fault line is lower than the left. Used is also prompted for the dip angle at the beginning, dip angle is same throughout the length of fault.

The fault line defines a fault surface starting at the elevations of the fault line and extending up and down by the dip angle. When a strata surface crosses the fault surface, the strata will shift by the amount of the shift value of the fault line at that point.



Prompts

Enter dip angle (degrees) <0.00>: 90

Reading cell> 89823

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: *pick a point*

Z: 492.29

Enter fault shift <0.00>: 35

[Arc/Distance/Close/Follow/Undo/<Pick point or point numbers>]: *pick a point*

Z: 481.30, Hz dist: 1408.86, Slope dist: 1408.90, Slope: -0.8% Ratio: -128.2:1

Enter fault shift <35.00>: *press Enter*

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: *pick a point*

Z: 456.90, Hz dist: 1114.75, Slope dist: 1115.02, Slope: -2.2% Ratio: -45.7:1

Enter fault shift <35.00>: 40

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: *pick a point*

Z: 456.02, Hz dist: 750.48, Slope dist: 750.48, Slope: -0.1% Ratio: -855.4:1

Enter fault shift <40.00>: 45

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: *press Enter*

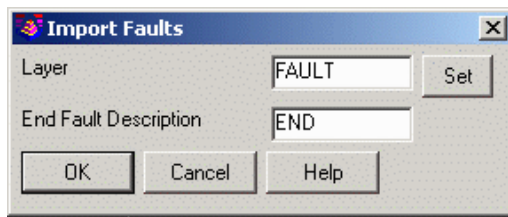
Pulldown Menu Location: StrataCalc

Keyboard Command: fault

Import Fault Line

This command creates fault polylines by reading data from a text file. Each row in the file is for a fault vertex. The data must include the X, Y and Z position plus the fault shift amount and dip angle. There is also an optional Description field which can be used when there are multiple faults in the same file. Use the End Fault Description as the description in the text file on the last point in the fault to identify where the fault ends and the next fault will start.

After selecting the text file to import, there is a dialog to define the format of the text file followed by a dialog for the End Fault Description and layer for the fault polylines to create.



Prompts

Text File to Load

Text File Import Options dialog

Import Faults dialog

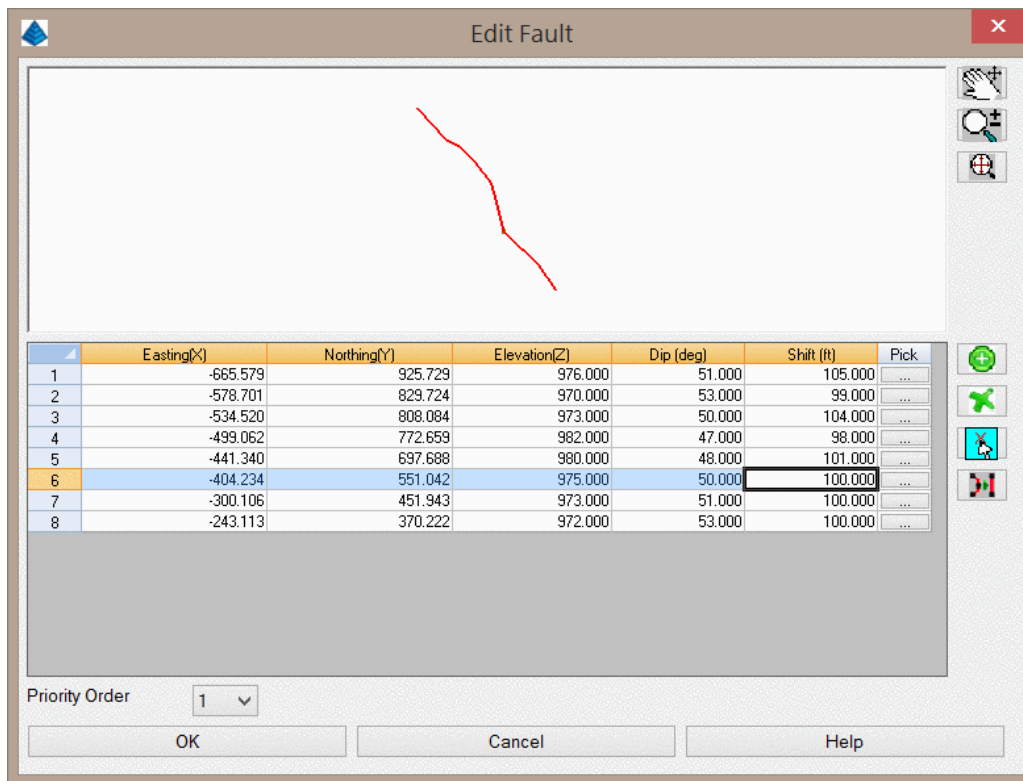
Prerequisite: Text file

Pulldown Menu Location: StrataCalc > Faults

Keyboard Command: import_fault

Edit Fault Line

This command is used to edit tagged fence polylines. When the command is executed, the below dialog will appear.



In this dialog, each vertex of the fault polyline is listed on a separate row. The coordinates, dip angle, and vertical shift associated with each vertex are also listed. A preview of the fault polyline is shown in the graphic window

with a highlight on the currently selected vertex.

Easting/Northing columns: These values list the X/Y coordinates of the fault polyline vertex.

Elevation(Z) column: These values list the elevation of the fault at the vertex. The elevation from row 6 in the above is set to 975. Notice that this is shown in the below diagram with the fault passing through this elevation.

Dip(deg) column: These values list the dip angle of the fault at the vertex. The dip angle is measured from the horizontal direction. The dip angle from row 6 in the above example is set to 50 degrees. This is shown in the below diagram.

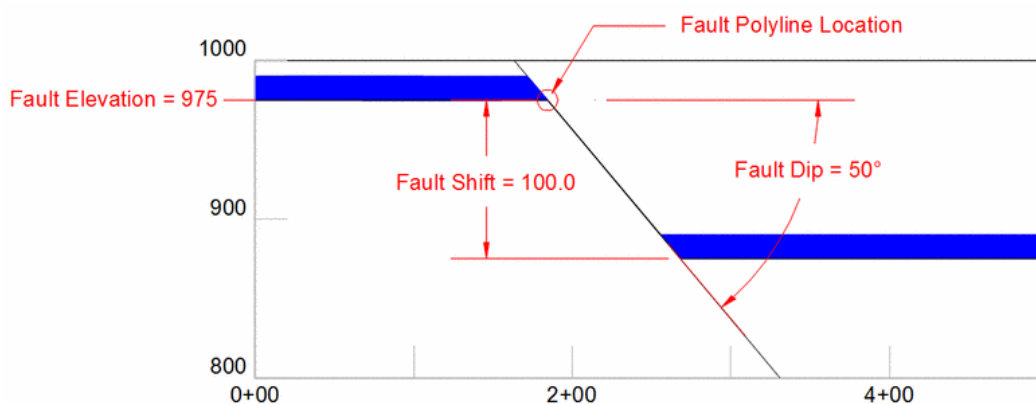
Shift column: These values list the vertical shift of the fault. When looking down the direction of the fault polyline, this value is the vertical shift when crossing from the left to the right side of the fault. Note that a positive shift amount means that the right side of the fault is lower than the left. The fault shift in the above example is set to 100'. This is shown in the below diagram.

Pick: This button will allow you to screen-pick a new X-Y location for the fault vertex.

Reduce Vertices: This button removes extra points from the fault without moving the fault position more than the specified cutoff tolerance.

Interpolate: This button interpolates shift values. This way you can fill out shifts at a few key points and have this Interpolate function fill in the rest of the shift values.

Priority Order: This option controls the order that the faults are processing. Priority 1 faults will be processed first, with lower priority faults being processed after.



The green + button to the right of the spreadsheet view will add a new vertex to the fault polyline.

The green X button will remove the currently selected vertex from the fault polyline.

The cursor button will move an the currently selected vertex to a new X-Y location.

The last button in the list will reduce the number of polyline vertices according to a offset cutoff amount. When used, this function will remove any vertices that will allow the polyline to move by less than the offset cutoff. Using the default value of 0.1 will prevent the fault polyline from moving more than 0.1 ft (or meters).

Prompts

Select fault polyline to edit: *Select fault polyline*

Pick new location: *pick a point near vertex to be edited*

Select fault polyline to edit (Enter to end):

Pick vertex to add to fault: *pick a point near vertex to add*

Enter Elevation <4786.99>: *enter elevation of surface topography*

Enter dip angle <90.0>: *enter the dip of the fault*

Enter fault shift (throw) <50.0>:

Pick vertex to add to fault (enter to end):

Pick new location:

Select fault polyline to edit (Enter to end): *pick another fault to edit*

Pull down Menu Location: StrataCalc

Keyboard Command: edit_fault

Prerequisite: fault lines drawn using Draw Fault Line

Calculate Fault Shift

This command assigns fault shifts to polylines. The polylines must be drawn ahead of time and they should follow the fault zones. After selecting the polylines to process, the program prompts for drillholes to process. Then select a strata to model. The fault shift is calculated by modeling the strata twice for each polyline. One model excludes the drillholes on the left of the polyline and the other model excludes the drillholes on the right. The modeling uses the inverse distance method. Then the elevation difference between the two models is used to assign the fault shifts along the polyline.

Prompts

Select fault lines.

Select objects: *pick polylines to assign fault shift to*

Select drillholes, channel samples and strata polylines.

Select objects: *pick entities to model*

Choose Strata To Process dialog

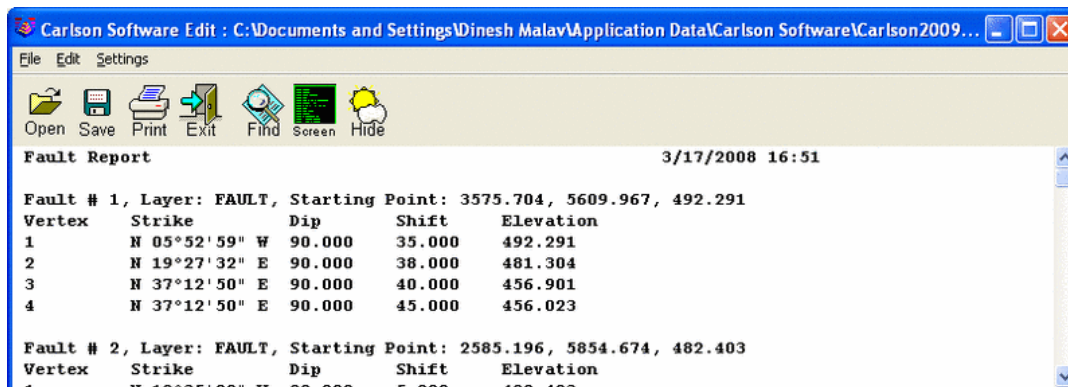
Pulldown Menu Location: StrataCalc

Keyboard Command: calc_fault_dz

Prerequisite: Polylines along the fault zones and drillholes

Report Fault Lines

This command is used to report information of each vertex of the fault lines drawn using Draw Fault Line Command.



Prompts

Select fault polylines to report:

Select objects: *all*

8 found

Select objects: *press Enter*

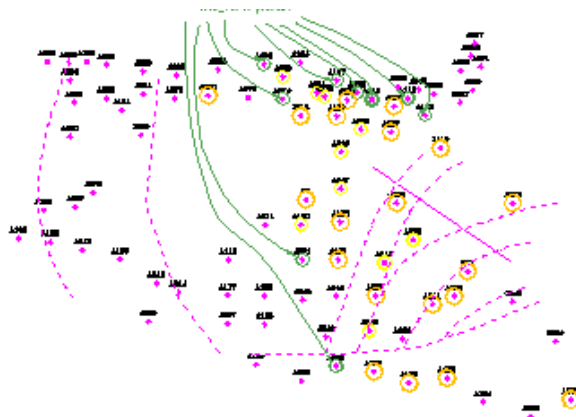
Pull down Menu Location: StrataCalc

Keyboard Command: report_faults

Prerequisite: fault lines drawn using Draw Fault Line

Highlight Fault Lines

This command is used to highlight fault lines drawn using Draw Fault Line Command.



Prompts

Select fault polylines to highlight:

Select objects: *all*

8 found

Select objects: *press Enter*

Pull down Menu Location: StrataCalc

Keyboard Command: highlight_faults

Prerequisite: fault lines drawn using Draw Fault Line

Identify Fault Polylines

This command is used to identify fault lines drawn using Draw Fault Line Command.

Prompts

Select fault polyline to identify: *select a fault polyline*

Vertex Strike Dip Shift Elevation

1 N 05°52'59" W 90.000 35.000 492.291

2 N 19°27'32" E 90.000 38.000 481.304

3 N 37°12'50" E 90.000 40.000 456.901

4 N 37°12'50" E 90.000 45.000 456.023

Select fault polyline to identify: *press Enter*

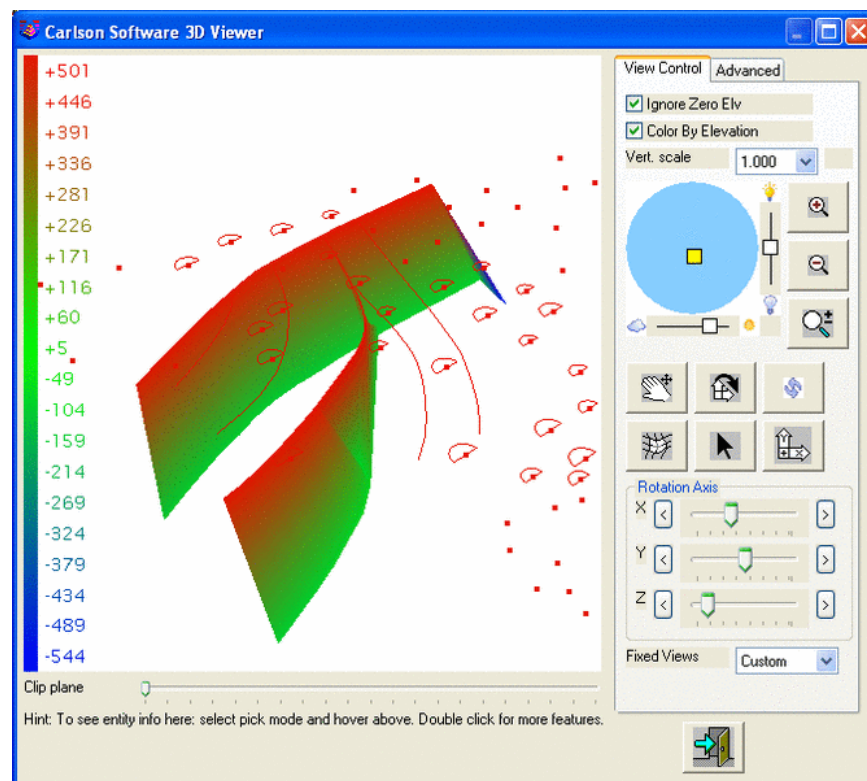
Pull down Menu Location: StrataCalc

Keyboard Command: id_faults

Prerequisite: fault lines drawn using Draw Fault Line

Draw Fault Surface

This command draws fault surfaces as 3D faces at dip angles from fault lines drawn using Draw Fault Line Command.



Prompts

Enter vertical depth to draw fault up to: 600

Select fault polylines to draw: all

2 found

Select objects: press Enter

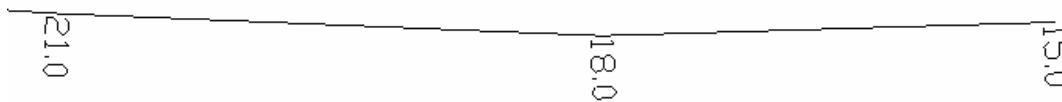
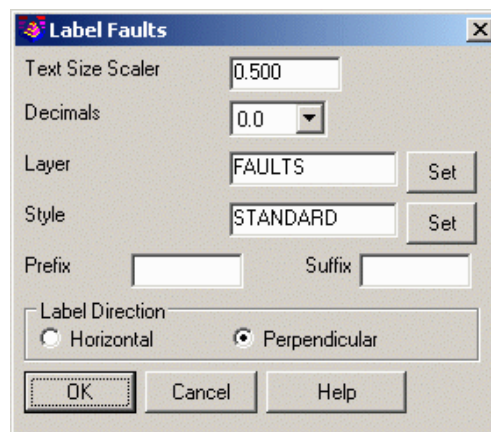
Pull down Menu Location: StrataCalc

Keyboard Command: draw_fault

Prerequisite: fault lines drawn using Draw Fault Line

Draw Fault Labels

This command labels the fault shift amounts for each point in the fault polyline. The dialog has settings for the label size, decimals, layer, style, prefix, suffix and orientation.



Prompts

Label Faults dialog

Select fault polylines to label.

Select objects: pick fault polyline

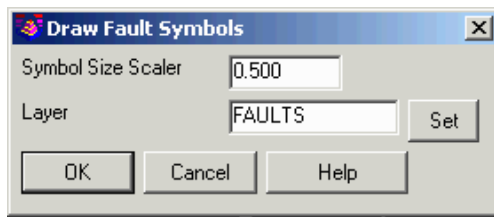
Prerequisite: Fault polyline

Pulldown Menu Location: StrataCalc > Faults

Keyboard Command: label_faults

Draw Fault Symbols

This command draws +/- symbols along fault lines. The + is on the upside and - on the downside of the fault. The program prompts for the Layer for the symbols and the Symbol Size Scaler which is multiplied by the current drawing horizontal scale to set the symbol size. Then the program prompts to select the fault polyline to label. Then the program draws the symbols at points picked along the fault.



Prompts

Draw Fault Symbols dialog

Select fault polyline for symbols: *pick a fault polyline*

Pick point for symbol: *pick a point along the fault*

Pick point for symbol (Enter to end): *press Enter*

Select fault polyline for symbols (Enter to end): *press Enter*

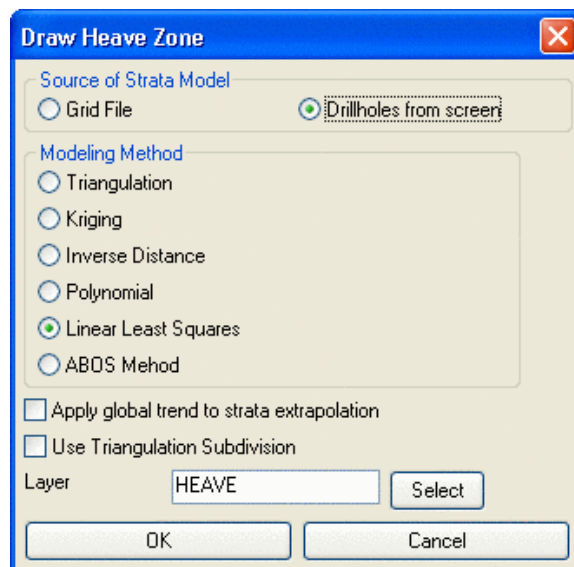
Prerequisite: Fault polyline

Pulldown Menu Location: StrataCalc > Faults

Keyboard Command: fault_sym

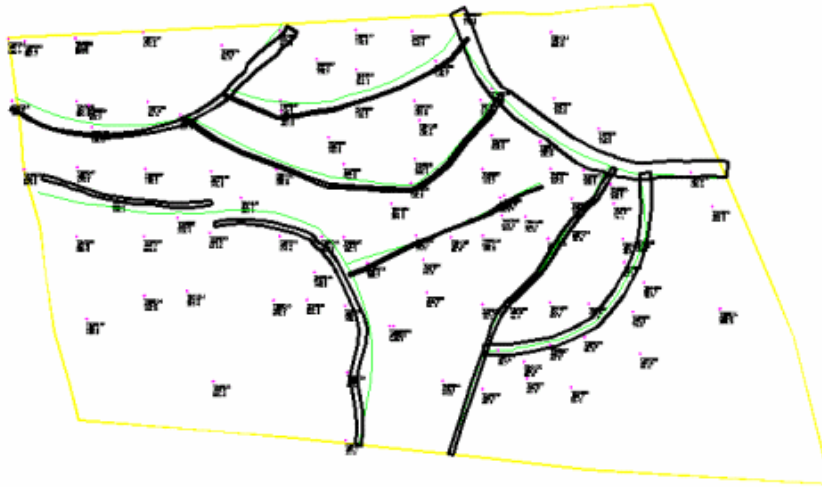
Draw Heave Zones

This command creates closed polylines that represent the fault heave zones where the material is broken up and not mineable. These polylines can then be used as exclusion areas in routine like Strata Isopach maps to avoid contours in those areas. They can also be tagged as Strata Limit Polylinies for exclusion for volumes and to delete the area in a Fence Diagram.



The command will use either an existing elevation grid file, or it will model from drillholes on screen. If there are multiple seams that are faulted, each one will have a slightly different area along the fault. Shown here is a result of drawing the heave zones. Most zones will run parallel to the faults, with the distance away from the fault based on

the displacement or shift of the fault.



Prompts

Here are the prompts when using the drillholes.

Command: draw_heave_zones

Select fault polylines to process:

Select objects: Specify opposite corner: 9 found

Select drillholes, channel samples and strata polylines.

Select objects: all

101 found

Reading drillhole 101

Select the strata limit polylines or ENTER for none.

Finding splits ...

Processing only strata with beds.

Use position from another file or pick grid position [<Pick>/File]?

Pick Lower Left grid corner:

Pick Upper Right grid corner:

Weighting factor to which power [First/<Second>/Third/Other]?

Strata Name: V_KEY

Calculating grid by linear least squares 2601...

Pulldown Menu Location: StrataCalc

Keyboard Command: draw_heave_zones

Prerequisite: Drillholes

Tag Heave Zones

This command tags polylines such as fault lines to indicate heave zones which are used by the Surface Mine Reserve routine to skip calculating the min and max thickness along these heave zone lines.

Prompts

Select heave zone polylines.

Select objects: *pick fault lines*

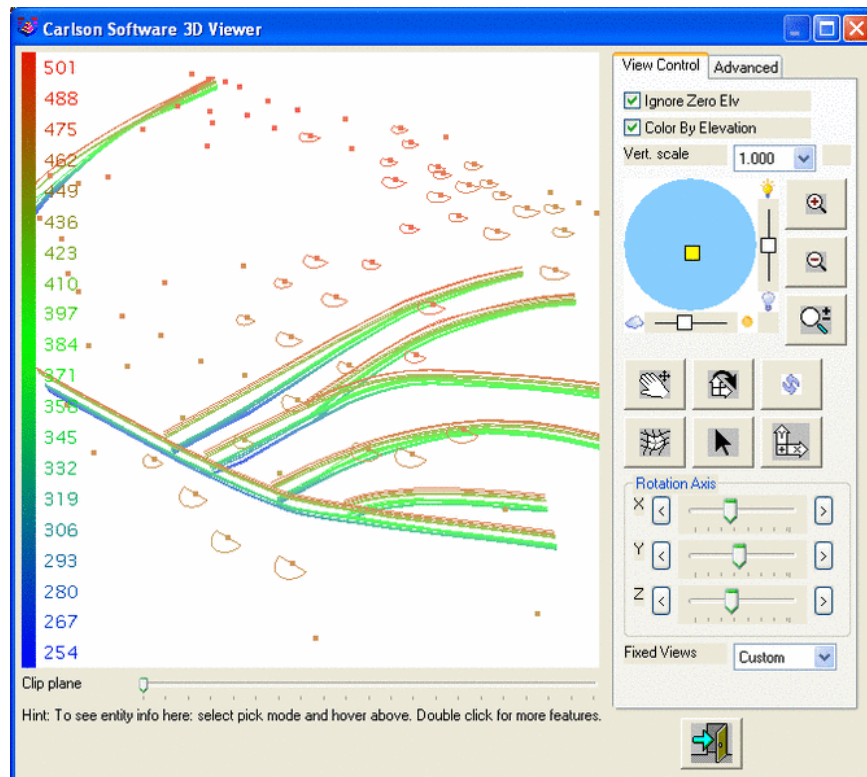
Pulldown Menu Location: StrataCalc

Keyboard Command: tag_heave

Prerequisite: linework

Create Strata Polylines at Faults

This command creates strata elevation polylines at the intersection of fault surfaces with strata models.



Prompts

Select fault polylines to process:

Select objects: *all*

7 found

Select objects: *press Enter*

Select drillholes, channel samples and strata polylines.

Select objects: *all*

93 found

Select objects: *press Enter*

Reading drillhole 93

Finding splits ...

Use position from another file or pick grid position [<Pick>/File]? P

Pick Lower Left grid corner: *pick lower left corner*

Pick Upper Right grid corner: *pick upper right corner to include faults*

Choose modeling method [<Triangulation>/Inverse dist/Kriging/Polynomial/LeastSq/ABOS]? I

Use inverse distance to which power [First/<Second>/Third/Other]? press Enter

Use elliptical inverse distance [Yes/<No>]? press Enter

Pull down Menu Location: StrataCalc

Keyboard Command: DRAW_STRATA_FAULTS

Prerequisite: fault lines drawn using Draw Fault Line and drillholes

Trim Isopach Within Fault Zone

This command trims isopach polylines in fault zones which applies when the isopachs are unreliable within the fault zones. The program prompts to pick one isopach polyline which is used to get the layer name of the isopach polyline. Then the program automatically selects all the polylines on that layer for processing. Next the program prompts for the fault zone width. The program automatically searches the drawing for all the fault polylines and trims the isopach polylines in areas within this fault zone width of the fault polylines.

Prompts

Select sample of isopach polyline to trim: *pick a polyline*

Fault zone width <10.0>: *press Enter*

Prerequisite: Fault polyline and isopach polylines

Pulldown Menu Location: StrataCalc > Faults

Keyboard Command: trim_fault

Apply Faults to Grid

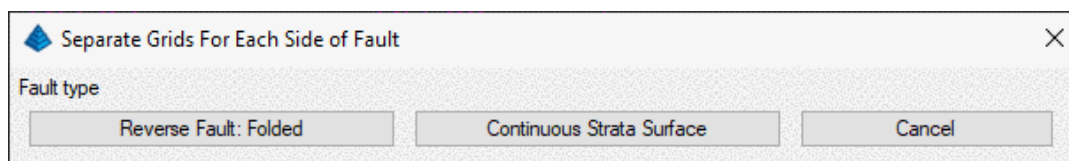
This command is used to offset the grid values of an existing strata elevation grid file to adjust for faults. The program saves a new grid file with the values offset based on the fault lines drawn with the Draw Fault Line command. There are two methods. The Separate Grids For Each Side of Fault method applies for both regular faults and reverse faults where the strata are folded over top of themselves within the fault zone. In both cases, the strata can be modeled separately for each side of the fault. The Adjust Grid Across Fault applies when the strata can be modeled with the same grid for the whole site. This method adjusts the grid in the fault zone.



Before running Apply Faults to Grid for this method, setup the faults and strata grids as follows:

1. Draw a fault polyline with the dip angle and elevation difference set using the Draw Fault Line command.
2. Draw a closed polyline around the area on the high side of the fault.
3. Create strata grids like usual. Make separate strata grids for the high and low sides.
4. Create a surface topo grid (only necessary for reverse faults)

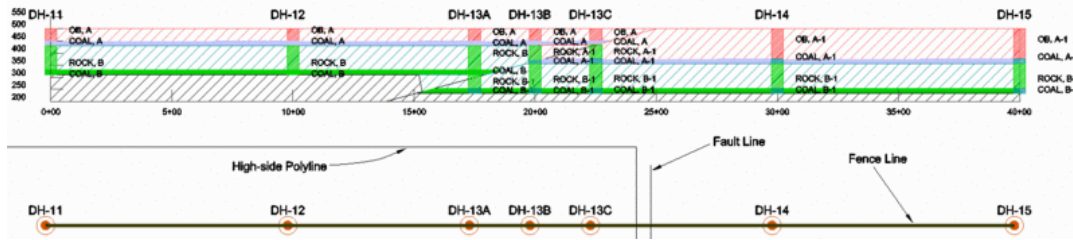
When using this method, you will be prompted with the below dialog



Reverse Fault: Folded

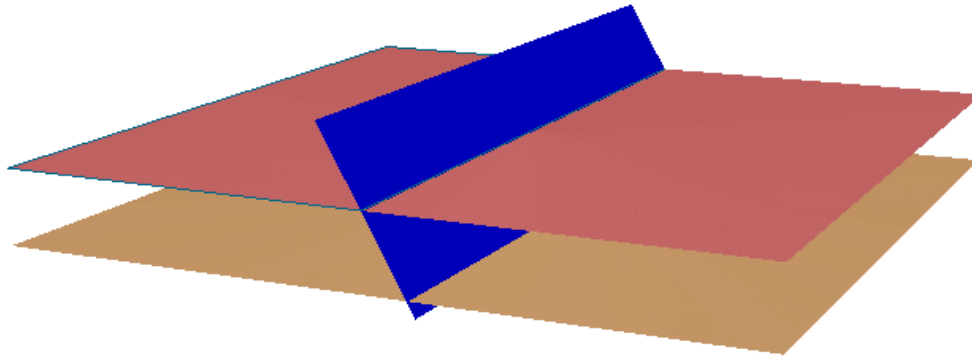
With this method, the program prompts for the fault polyline and high side polyline from steps 1,2 and the grids from steps 3,4. The processing for the high side strata grids is slightly different than the low side because the way

the program goes about embedding the fault surface to the strata surfaces. For the high side, you specify one strata grid at a time. For the low side, you specify the top and bottom strata grids (ie A1_TOP.grd and A1_KEY.grd).

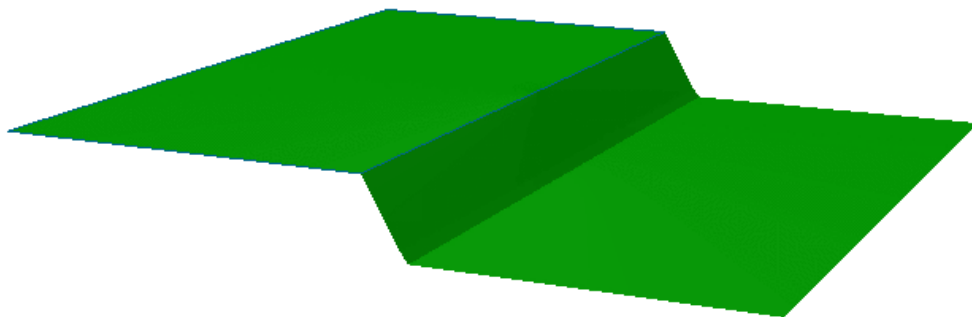


Continuous Strata Surface

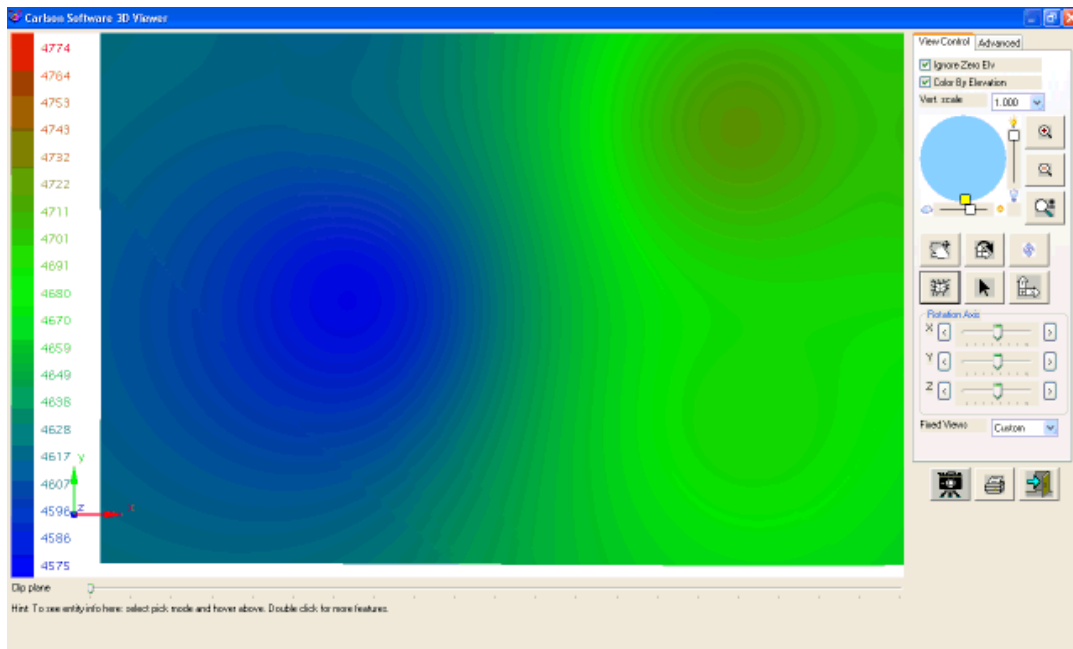
With this method the program prompts for the fault polyline, the high side inclusion polyline, and the high/low-side grids (no topo is needed). This will join the two grids and the fault surface into a single grid file as shown below.



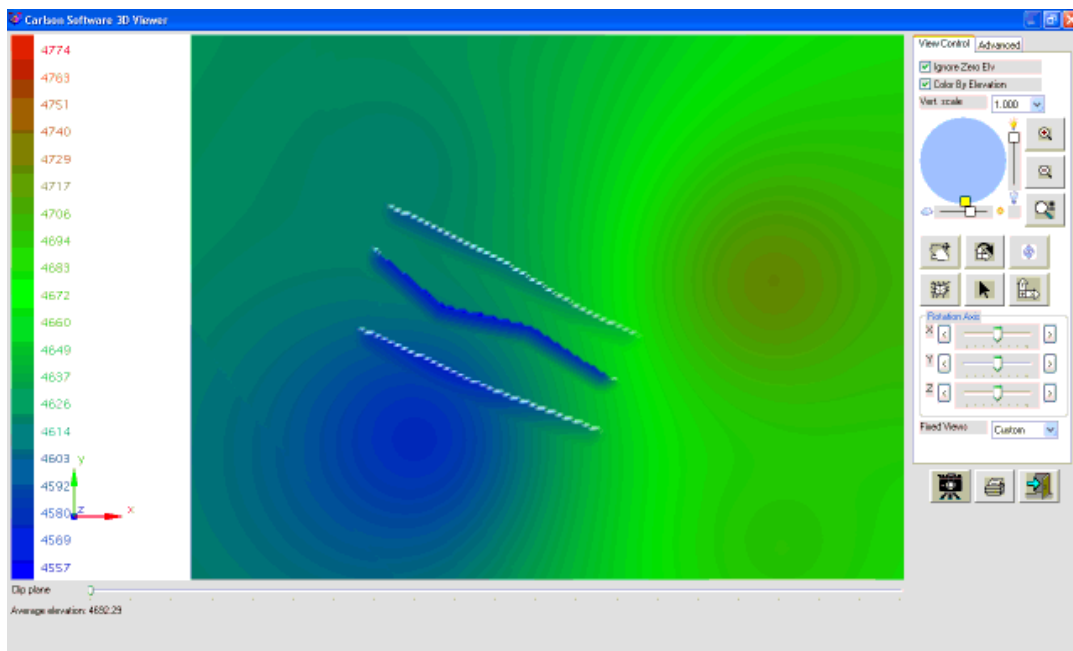
The above image shows the fault surface in blue, the high side of the grid in red, and the low side of the grid in orange. The below image shows how these three surfaces will be joined together.



The Fault Influence Distance is the offset distance away from the fault lines that will be used to transition from the original grid to the faulted values.



This is the grid before applying the faults



This is the grid after applying the faults

Prompts

Fault Influence distance <500.00>: 150 *Enter in distance to use.*

Select fault lines:

Select objects: *Select the fault lines*

Updating grid with faults 190000...

Pull down Menu Location: StrataCalc > Faults

Keyboard Command: apply_faults

Prerequisite: fault lines drawn using Draw Fault Line and a grid file

Remove Faults From Grid

This command creates a strata elevation grid file (.grd) with faulting removed by adjusting an existing strata elevation grid with faults and selecting fault lines from the drawing to process. The purpose of this routine is to check that the fault lines are defined correctly. When the fault lines are in the right places with the right shift amounts, then the resulting grid model from this routine should be smooth without shifting from faulting. The Fault Influence Distance is how far to the sides of the fault lines that the program will adjust the grid.

Prompts

Grid File to Process *Choose input strata elevation grid*

Fault Influence distance <500.00>: 100

Select fault lines.

Select objects: *pick fault lines*

Output Grid To Write *Choose strata elevation grid file name to create*

Pulldown Menu Location: StrataCalc

Keyboard Command: remove_faults

Prerequisite: strata elevation grid and faults

Make Unfaulted Strata Grid

This command creates a strata elevation grid file (.grd) from selected drillholes and faults in the same way as the Make Strata Grid File command. The difference with this command is that it applies at fault adjustments in reverse. The purpose of this routine is to check that the fault lines are defined correctly. When the fault lines are in the right places with the right shift amounts, then the resulting grid model from this routine should be smooth without shifting from faulting.

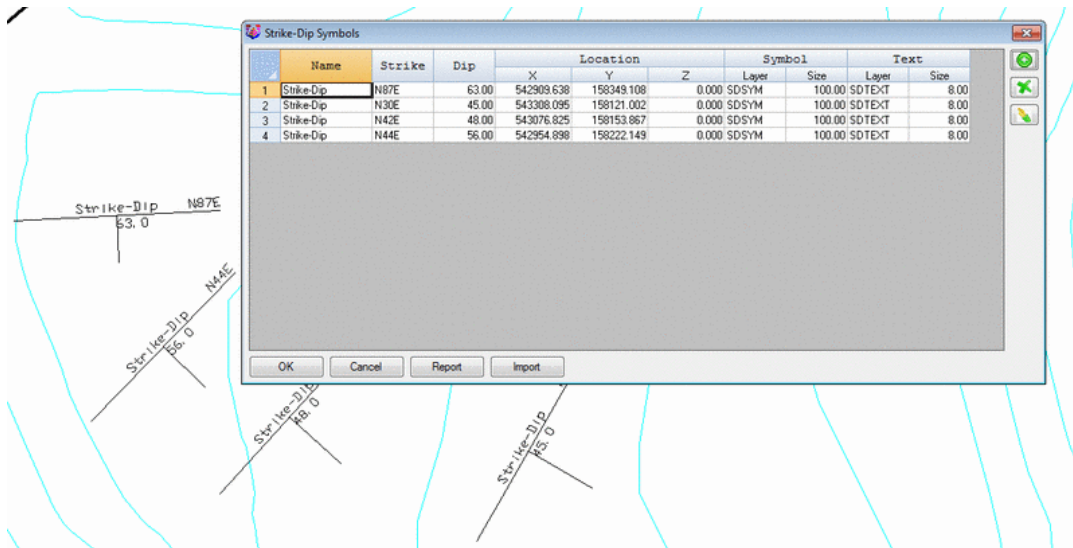
Pulldown Menu Location: StrataCalc

Keyboard Command: unfault_grid

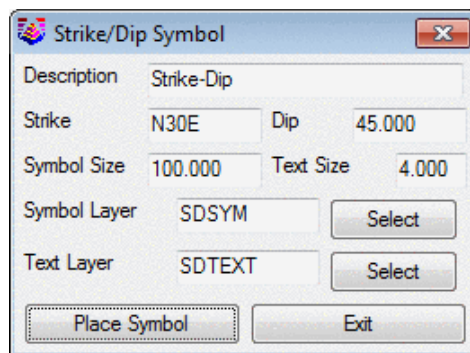
Prerequisite: drillholes and faults

Strike-Dip Symbol Editor

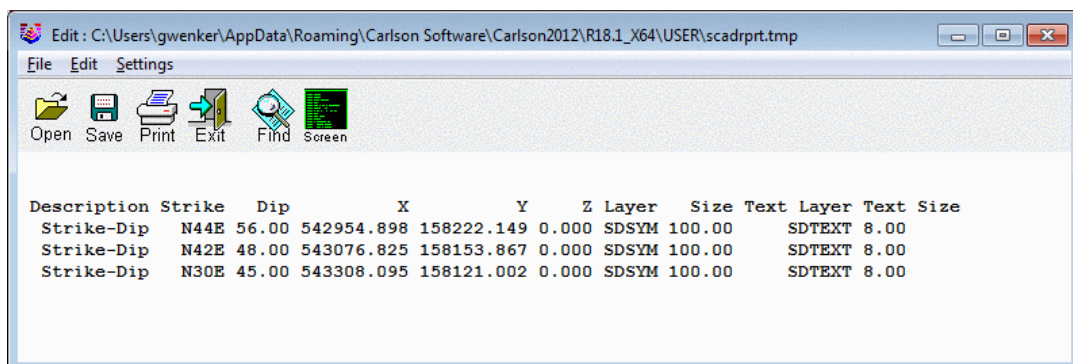
This command works in an editor where the strike and dip symbols can be added, deleted or edited in a spreadsheet type editor. There are options for reporting and importing from an external text file to bring the symbols in.



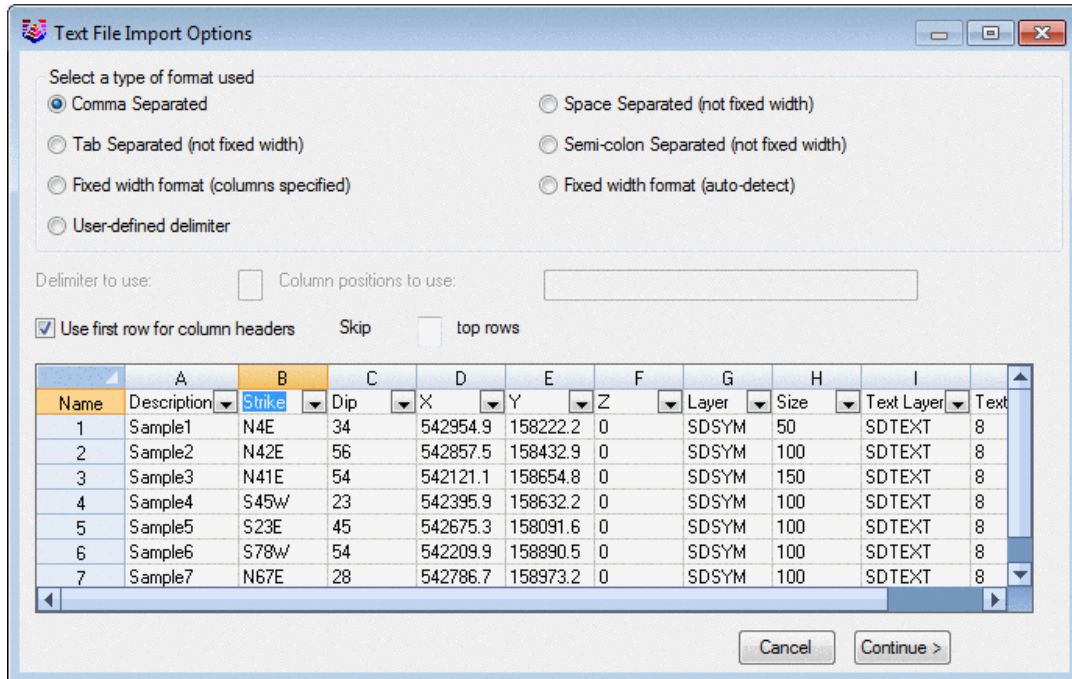
- **Add Symbol (+):** This button brings up the Strike/Dip Symbol window where a new symbol can be edited. There are defaults in the window that can be modified with the data. There are options to label a description, enter the strike and dip, and symbol and text size. The symbol and text layers can be entered here, or picked from a list of layers with the Select button. The Place Symbol button allows for screen picking of the new location for the symbol.



- **Delete Symbol (X):** This button deletes the highlighted row of the selected symbol.
- **Edit Symbol:** This button is similar to the Add button, but the highlighted row symbol appears in the window, allowing for changes that are done in the drawing when the Exit is selected.
- **Report:** This option will bring up the Report Formatter where the symbols are reported. The following data can be reported, such as X, Y, Z, layer, text size, and others.



- **Import:** This option will import the symbols into the drawing from a text file of the formats shown. It is automatic if a header line is used to define the columns, it will find the names and import the data. If there isn't a header line, then each column will have to be defined prior to importing, with the dropdown arrows.



Prompt

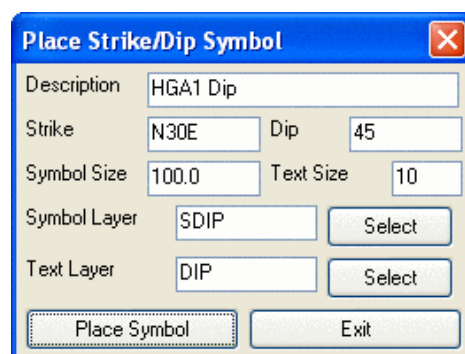
Select all strike-dip symbols.

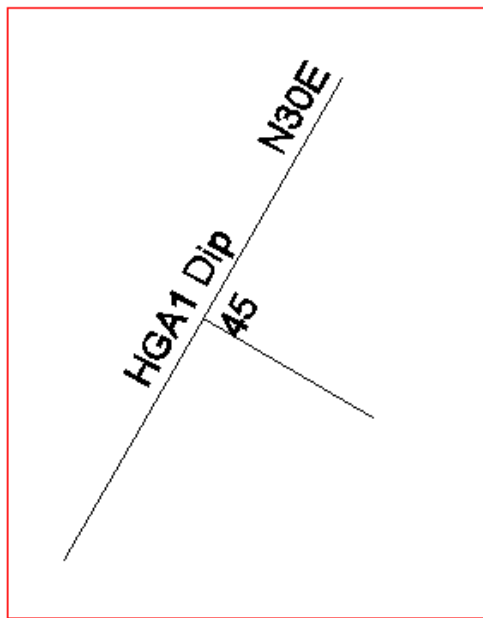
Select objects: Specify opposite corner: 20 found, 4 groups. (If there are no symbols in the drawing already, just hit "Enter" and the empty window will appear where new symbols can be added or imported).

Keyboard Command: EDIT_STRIKEDIP

Draw Strike-Dip Symbol

This command draws the standard strike-dip symbol based on user defined input. It has an option for entering and labeling a description. If no description is desired, then leave it blank. The strike and dip entered in the window will orientate the symbol placement. There are settings for symbol and text size, as well as layers.





Keyboard Command: `strikedip`

Tag Strata Polylines

This command attaches a strata name to a 3D polyline that represents the bottom elevation, top elevation or the thickness of that strata. When modeling the drillholes, each point in these strata polylines is used as an elevation or a thickness data point for the assigned strata. For example, a 3D polyline for the outcrop of a strata could be tagged with the strata name and used in processing for additional model enhancement. If bed names are used, be sure to include the `_TOP`, `_KEY`, `_PARTING` or `_BOTTOM` extensions to the bed name.

Prompts

Strata name: `Seam1`

Type of strata polyline [`<Elevation>/Thickness`]? *press Enter for Elevation*

Elevation reference [`<Bottom>/Top`]? *press Enter for Bottom*

Select strata bottom elevation polylines.

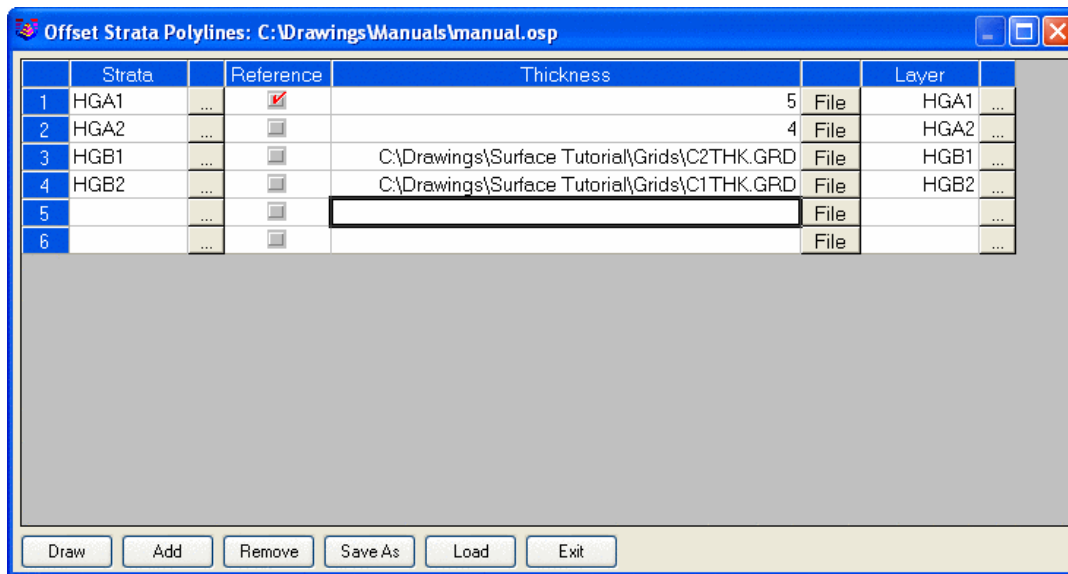
Select objects: 1 found

Pulldown Menu Location: Drillhole

Keyboard Command: `stratatag`

Offset Strata Polylines

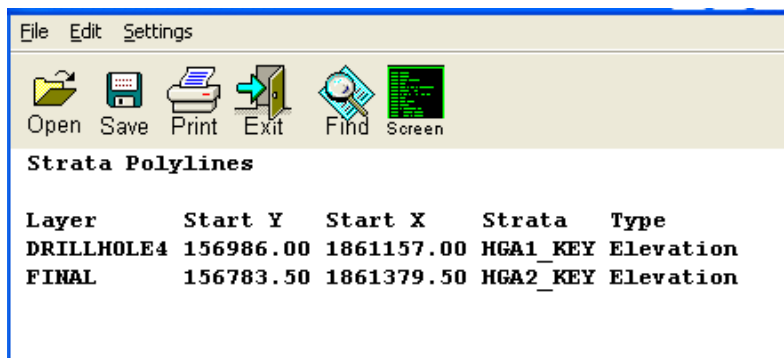
This command creates additional strata elevation polylines offset by the strata thickness from a reference/starting strata polyline for elevation. This is useful for manually creating faults where all the seams are offset along the elevation strata polylines. The first prompt is to open or create an offset strata polylines file (*.OSP). The first column is the strata/bed name attached to the strata polyline. The button next to them brings up the list of defined strata. A Reference polyline is one that already exists and this is the one that will be offset from. The thickness to offset the new polylines can either be entered as a value (+/-) or as a thickness grid file. The layer is the layer that the reference polyline is on, and the layer it will draw the new polylines on. The resulting new strata polylines can be used for more accurate modeling elevations.



Keyboard Command: offset_strata_pl

Report Strata Polyline

This command generates a report using the Report Formatter containing all the Strata Polyline automatically found in the drawing. It reports the layer, XY of the start and end of the polyline, the tagged strata name and the type (elevation or thickness).



Keyboard Command: report_strata_pline

Highlight Strata Polyline

This command simply highlights the named strata polylines in the drawing. A regen of the screen will unselect the polylines.

Prompts

Select the polylines to check.

Select objects: Specify opposite corner: 5 found

Highlighted 3 Strata Polyline. Skipped 2 other polyline.

Pulldown Menu Location: Drillhole in Geology

Keyboard Command: highlight_strata_pl

Identify Strata Polylines

This command simply allows you to pick a polyline and report the strata name, if any, attached to this polyline. These are tagged for use in modeling the bottom elevation or thickness in addition to drillholes.

Prompts

Select strata polyline to identify:

Strata Name: SEAM2 Type: Thickness

Select strata polyline to identify:

Strata Name: SEAM1 Type: Elevation

Select strata polyline to identify: *pick one that is not tagged*

No strata name assigned to polyline.

Select strata polyline to identify:

Pulldown Menu Location: Drillhole

Keyboard Command: strataid

Untag Strata Polylines

This command removes the strata polyline information and returns the selected polylines to the original status: regular polylines.

Prompts

Select strata polylines to have strata tags removed.

Select objects: 1 found

Pulldown menu location: Drillhole

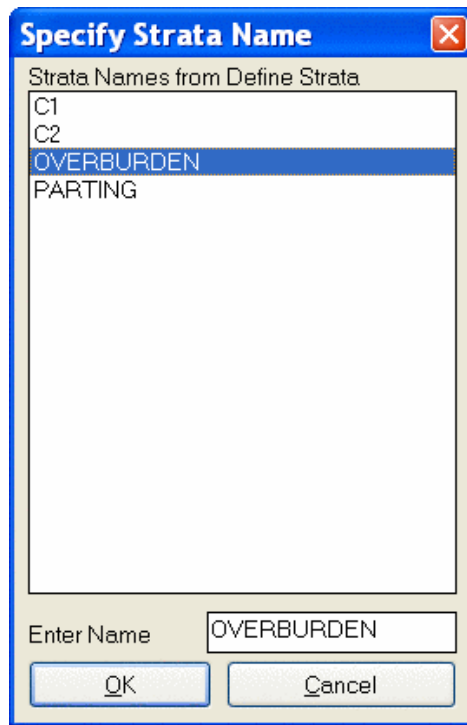
Keyboard command: stratauntag

Prerequisite: Named strata polylines

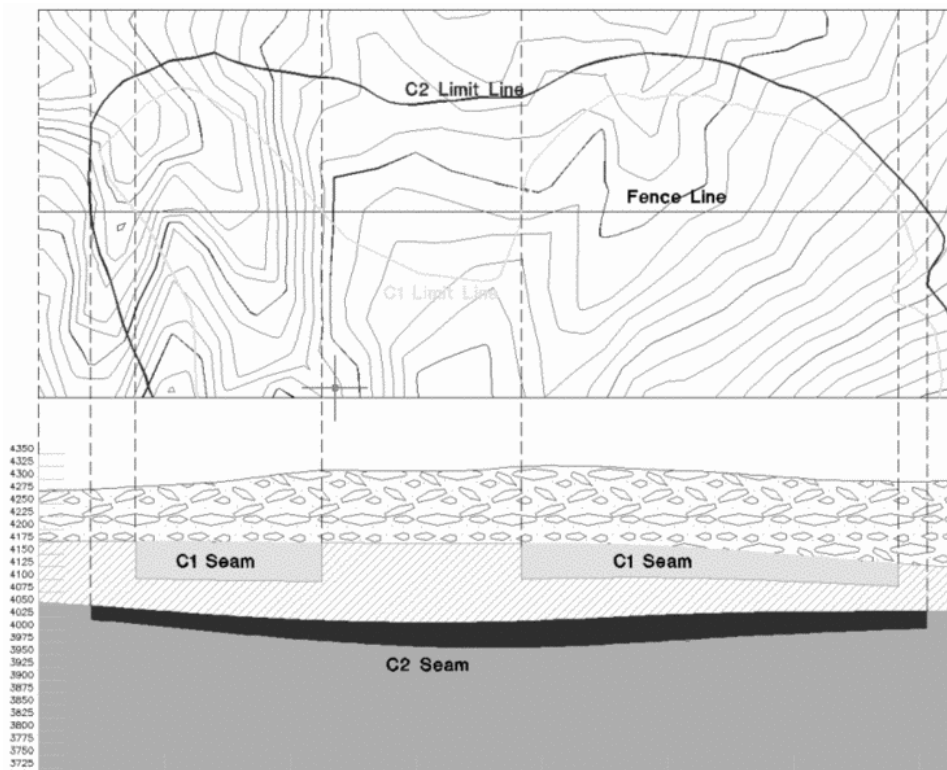
Name Limit Polylines

This command assigns strata names to identify polylines as inclusion or exclusion areas for the strata. Strata limits are used to model discontinuities such as glacier washout or volcanic strata. Strata that gradually trend to pinch out should not use strata limits. The pinch out should be modeled using just the drillholes. When assigning the names for strata in the Geologic Model file, be sure they are spelled exactly the same here, so they are applied correctly. Also, when using Limit Polylines with bed names in the drillholes, be sure to add the `_TOP`, `_KEY`, `_PARTING`, OR `_BOTTOM` to make sure the limits are applied to each bed interval.

Strata limit polylines are used in the Stratacalc routines to indicate where the strata exists. The inclusion and exclusion polylines are used for trimming the contours in Strata Isopach Maps and used to limit volumes in routines like Surface Mine Reserves. Also in areas outside the strata limits, the strata thickness is set to zero and 10000 is added to the strata elevation. This forces the strata up to clip out to the strata above. Also drillholes that are outside the strata limits are ignored for processing that strata. There is a choice to assign all strata names to the limit lines, or Specific strata. If Specific is chosen, then the following dialog appears.

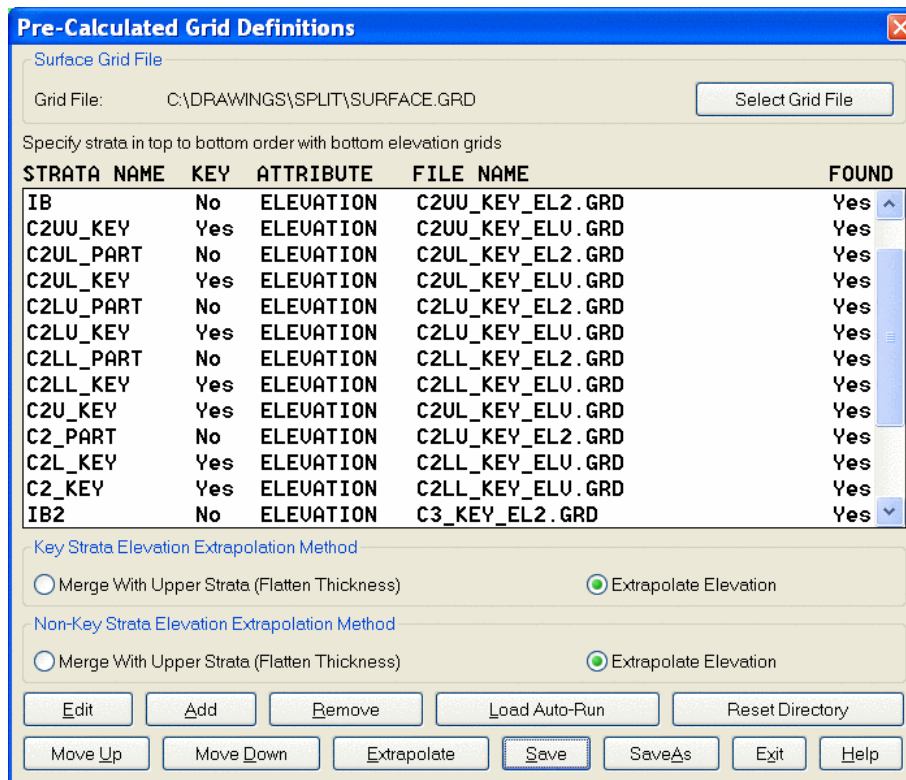


When drawing a Fence Diagram and using Limit Polylines, the strata cutoff occurs exactly at limit the polylines. Shown here is an example of a Fence Diagram, plotted with the plan view for verification. Notice how the limit lines crop the seams in section view precisely at the limit lines. The plan view was placed on top of the fence diagram just with basic AutoCAD drafting tools. Dashed polylines were then drawn with ortho on to illustrate the crops honoring the limit lines exactly. For all StrataCalc routines that use limit polylines, there is a command line message to report when limit polylines are applied.

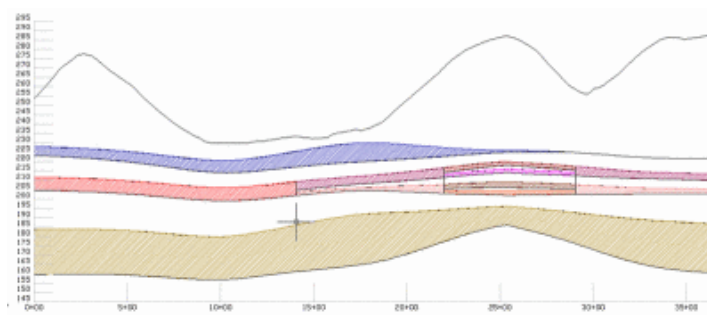


Another use of Limit Lines is to control the transition from a full seam to where it splits. Limit lines must be drawn for one or both instances. The full seam and the two split seams can both appear in the Geologic Model file for volumes and Fence diagram. An example Geologic Model is shown here. Notice the C2_KEY splits into C2L_KEY

and C2U_KEY. They also split. All of these can exist in the Geologic Model, as long as there are limit lines on screen to control where they appear.



The result is a Fence Diagram as shown here, where the limit lines control where the full seam, and its splits appear.



Prompts

Enter strata name to apply (<All>/Name)? UB for strata UB

Select Inclusion perimeter polylines.

Select objects: *pick a closed polyline*

Select Exclusion perimeter polylines.

Select objects: *press Enter*

Pulldown Menu Location: StrataCalc

Keyboard Command: minelmt

Report Limit Polylines

This command reports all Strata Limit Polylines that are in the drawing. The report contains the layers, area, XY of the polyline endpoints, and whether it is an inclusion or exclusion limit polyline.

Layer	Area SF	Area Acres	Start Y	Start X	Inclusion	Exclusion
GE-CROP-ISL-A	109324.1	2.51	161058.36	1866080.60	HAGA_KEY HGA1_KEY HGA2_KEY	
GE-CROP-ISL-A	75936781.3	1743.27	161018.30	1864279.75	HAGA_KEY HGA1_KEY HGA2_KEY	
GE-CROP-ISL-B	85324281.9	1958.78	161293.49	1864599.23	HAGB_KEY HGB1_KEY HGB2_KEY	
GE-OLDMINES-ISL	613930.3	14.09	158416.39	1865103.60		HAGA_KEY HGA1_KEY HGA2_KEY
GE-OLDMINES-ISL	870671.8	19.99	157284.92	1865677.49		HAGA_KEY HGA1_KEY HGA2_KEY
---- Grand Total	162854989.3	3738.64	799071.47	9325740.67		

Keyboard Command: report_strata_limit

Highlight Limit Polyline

This command simply highlights the named strata limit polylines in the drawing. A regen of the screen will unselect the polylines.

Prompts

Command: highlight_limit_pl

Select the polylines to check. (*select all the polylines of interest*)

Select objects: Specify opposite corner: **10 found, 2 groups, 12 total**

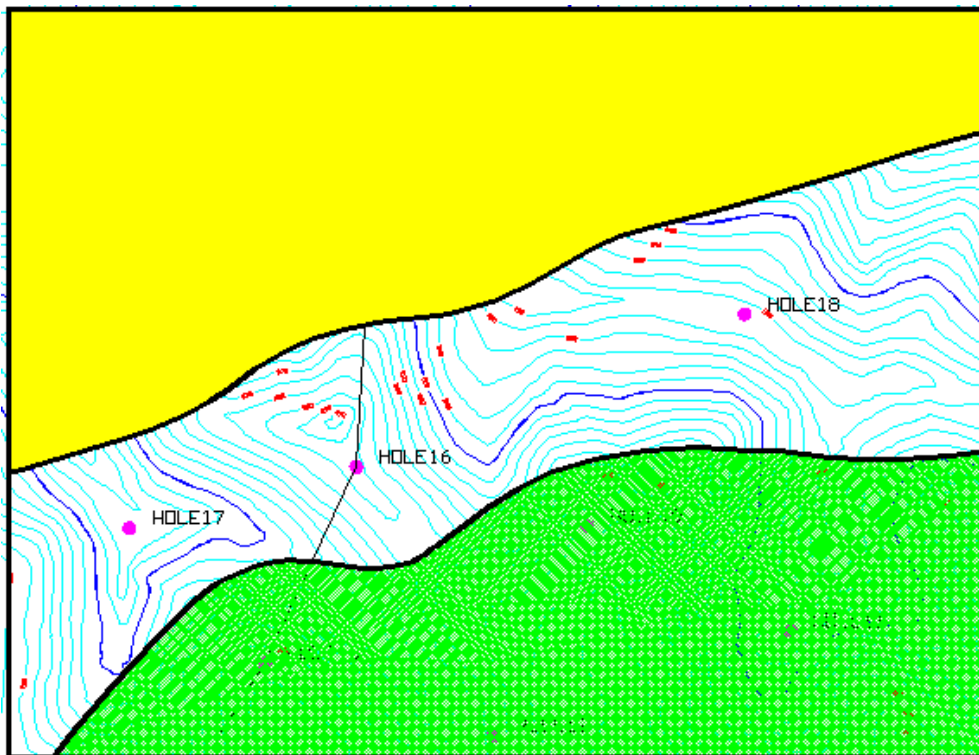
Highlighted 2 Limit Polyline. Skipped 10 other polyline.

Pulldown Menu Location: StrataCalc in Geology

Keyboard Command: highlight_limit_pl

Identify Limit Polyline

This command reports the strata names assigned to a polyline for strata limits. There is the option to select one and view its name, to pick inside and have it identify which lines enclose that point, or to search the drawing and automatically highlight and report all that it finds. If the hatch option is used, then the hatch window appears where the hatch pattern can be selected for each limit polyline.



Prompts

Pick on a polyline, pick inside or search drawing [**<Pick>/Inside/Search**]: *i*

Pick inside limit polyline to identify:

Hatch Current, select Next or Exit [**<Hatch>/Exit**]: *H to hatch*

or

Pick on a polyline, pick inside or search drawing [**<Pick>/Inside/Search**]:

Select mining limit polyline:

Current: Inclusion: C2U C2L IB1

Hatch Current or Exit [**<Hatch>/Exit**]: *e to exit*

Pull-down Menu Location: StrataCalc in Geology

Keyboard Command: minelmtl

UnTag Limit Polylines

This command removes the strata limit assignments from the selected polylines.

Prompts

Select limit polylines to have limit data removed.

Select objects: *pick the limit polylines*

Pull-down Menu Location: StrataCalc

Keyboard Command: rmlimit

Define Washing Curve

Coal washing is accomplished by one of two major processes, by density separation or by froth flotation. This density curve will be applied to all the attributes, and save to a StrataCalc Datasheet STC file, so that the selected float level density of attributes can be gridded and saved in the geologic model. Define Washing Curve is a prerequisite for Apply Wash to define the wash curve for the desired density interval, attributes, and their values. The data is entered here to define the minimum and maximum float levels, the interval for the density. The total mass of the sample is shown in the total material field. All attributes are custom entered based on what is in the sample report. This algorithm assigns each strata/bed interval value to a washing file, saved out as a *.WASH file.

Units	Float	Mass	Inherent	Ash	Volatile	Fixed	Calorific	Total
	Kg/m3	Kg	%(ad)	%(ad)	%(ad)	%(ad)	MJ/kg	%(ad)
Total Material		1.2800	2.2203	34.3625	20.2891	43.1281	19.6049	0.9196
Float 1	1.5000	0.2200	2.5000	15.1000	32.2000	50.2000	27.5000	1.7800
Float 2	1.5500	0.1100	2.2000	23.2000	25.0000	49.6000	24.9000	0.0670
Float 3	1.6000	0.0900	2.2000	26.5000	21.0000	50.3000	22.5600	0.7600
Float 4	1.6500	0.0800	2.3000	28.5000	19.4000	49.8000	20.7700	0.3800
Float 5	1.7000	0.1600	2.4000	32.8000	17.7000	47.1000	20.0100	0.4400
Float 6	1.7500	0.2000	2.3000	38.3000	16.5000	42.9000	17.6100	0.5500
Float 7	1.8000	0.1300	2.1000	42.2000	15.9000	39.8000	16.3500	0.3800
Sink	1.8000	0.2900	1.9000	51.9000	15.5000	30.7000	12.9800	1.5500

The Units are custom defined by the user and can be lbs/ft3 or Kg/m3. The mass can be custom defined to Kg, pounds, or anything the user wants. The Float Density is a specific gravity or relative density. The Sink is the material that is higher than the density, and sinks to the bottom. The attributes in the columns are the values of each, when washed to the density, removing any heavy waste rock.

Data Mode: The Data Mode setting determines the way the weight and attributes are entered for each density. Fractional shows just that percentage for each density, while Cumulative displays the totals increase as the density increases. This must be set to match the source data.

Material Source: Selecting a drillhole graphically is optional, but is useful in going through each drillhole's strata/bed entry and finding the best fit between wash file raw material, and drillhole strata/bed values. Preselecting the drillhole will limit the scope of the best-fit search within that drillhole. If the drill hole is not selected, each strata/bed interval will be checked against every wash file to find the best fit. This is why preselecting the Source Drillhole is an optional input but does give the potential of speeding up the process. The output summary from Apply Wash shows the wash file and drillhole strata/bed match up, and the associated fitting error, a lower number indicates closer match between raw and drillhole values.

Testing Source: These fields are used to identify the Lab Name, ID# and Date of the sample. This is saved in the WASH file for later reporting and review.

Add Float Level: This adds a new row to insert a new density.

Add Attribute: This adds a new column for a new attribute. The name will be incremental, such as "Attribute5".

Edit Attribute Title: This option will rename the newly added attribute to the desired name.

Delete Float: This removes the selected row of the selected density.

Delete Attribute: This removes the selected attribute column.

Clear Data: This option deletes all the rows and columns of float levels and attributes for starting over.

Save/SaveAs/Load: This command creates a *.WASH file that is then used for Apply Wash. Each seam sample is saved as its own wash file. The format is comma delimited and is shown here. This format could also be edited or created in Excel, if the user prefers.

Source Drillhole,DH-01CPY

Data Mode,0,0 : Fractional 1 : Cumulative

Row Names,Float Density,Mass,Inherent Moisture,Ash Content,Volatile Matter,Fixed Carbon,Calorific Value,Total Sulphur

Unit Type,Kg/m3,Kg,%(ad),%(ad),%(ad),%(ad),MJ/kg,%(ad)

Total Material,1.6832,1.2800,2.2203,34.3625,20.2891,43.1281,19.6049,0.9196

Float 1,1.5000,0.2200,2.5000,15.1000,32.2000,50.2000,27.5000,1.7800

Float 2,1.5500,0.1100,2.2000,23.2000,25.0000,49.6000,24.9000,0.0670

Float 3,1.6000,0.0900,2.2000,26.5000,21.0000,50.3000,22.5600,0.7600

Float 4,1.6500,0.0800,2.3000,28.5000,19.4000,49.8000,20.7700,0.3800

Float 5,1.7000,0.1600,2.4000,32.8000,17.7000,47.1000,20.0100,0.4400

Float 6,1.7500,0.2000,2.3000,38.3000,16.5000,42.9000,17.6100,0.5500

Float 7,1.8000,0.1300,2.1000,42.2000,15.9000,39.8000,16.3500,0.3800

Sink,1.8000,0.2900,1.9000,51.9000,15.5000,30.7000,12.9800,1.5500

Shown here in Excel:

Row Names	Float Density	Mass	Inherent Moisture	Ash Content	Volatile Matter	Fixed Carbon	Calorific Value	Total Sulphur
Unit Type	Kg/m3	Kg	%(ad)	%(ad)	%(ad)	%(ad)	MJ/kg	%(ad)
Total Material	1.6832	1.28	2.2203	34.3625	20.2891	43.1281	19.6049	0.9196
Float 1	1.5	0.22	2.5	15.1	32.2	50.2	27.5	1.78
Float 2	1.55	0.11	2.2	23.2	25	49.6	24.9	0.067
Float 3	1.6	0.09	2.2	26.5	21	50.3	22.56	0.76
Float 4	1.65	0.08	2.3	28.5	19.4	49.8	20.77	0.38
Float 5	1.7	0.16	2.4	32.8	17.7	47.1	20.01	0.44
Float 6	1.75	0.2	2.3	38.3	16.5	42.9	17.61	0.55
Float 7	1.8	0.13	2.1	42.2	15.9	39.8	16.35	0.38
Sink	1.8	0.29	1.9	51.9	15.5	30.7	12.98	1.55

Prompts

Command: C:DEFINEWASH

Define Washing Curve Dialog

Drop-Down Menu Location: StrataCalc ->Wash Filter->Define Washing Curve

Keyboard Command: definewash

Apply Wash

This command takes the wash curve files and runs them through the washing process to report the results either at a specified density, or with certain targets defined to calculate the density to wash the product to. These results can be saved as a *.STC Stratacalc file that may then be used with Make Strata Grid or AutoRun Strata Grids to make grids of these washed attribute values, and add them to the geologic model for mine quantities and reserves.

Attribute	Raw Avg
1 ASH CONTENT	28.83
2 CALORIFIC VALUE	21.43
3 INHERENT MOISTURE	1.86
4 TOTAL SULPHUR	.80

Attribute	Equality	Value	Units
1 Calorific Value	>	24.0000	MJ/kg
2 Calorific Value	<	30.0000	MJ/kg

Attribute	Equality	Value	Units
1 Calorific Value	=	25.0000	MJ/kg

Attribute	Equality	Value	Units
1 Total Sulphur	<	50.0000	(ad)

Lithology Status	Count
Missing Attributes	18
Failing Targets	0
Passing Raw Targets	2
Passing Processing Targets	5

Attribute	Raw Avg
1 ASH CONTENT	15.00
2 CALORIFIC VALUE	26.00
3 INHERENT MOISTURE	1.00
4 TOTAL SULPHUR	.50

Attribute	Float	Sink
1 ASH CONTENT	21.01	42.81
2 CALORIFIC VALUE	25.00	16.19
3 INHERENT MOISTURE	2.33	2.12
4 TOTAL SULPHUR	1.00	.87

Attribute	Float	Sink
1 ASH CONTENT	19.29	42.81
2 CALORIFIC VALUE	25.29	16.19
3 INHERENT MOISTURE	1.95	2.12
4 TOTAL SULPHUR	.86	.87

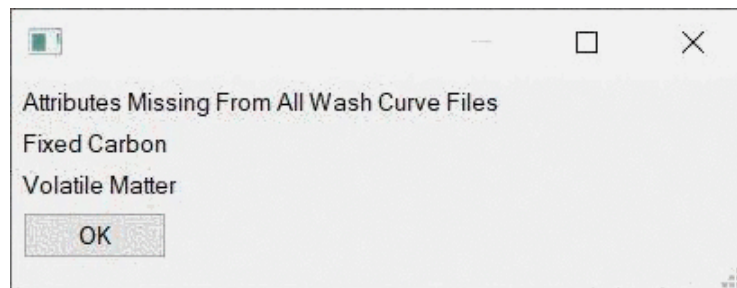
	Target	Active
Min Density	1.510	1.550
Max Density	1.790	1.750
Number of Float Levels	2	5
Number of Files	4	3

Wash File Settings

- **Min / Max Density:** The density range is entered here. Target is what the user wants for low and high, and Active is what is found in the data.
- **Number of Float Levels:** These are the steps or intervals of the density ranges, such as 1.5, 1.55, 1.6, 1.65 etc. There must be at least two float rows, so that a line can be interpreted between the two points on the

graph/curve.

- **Number of Files:** This shows the number of WASH files that were selected and read in. Target shows the actual files that were selected, Active shows the number that contain usable, complete data.
- **Target and Active:** The Target column displays what the user has defined, the Active displays what the program can use, that has complete data.
- **Select:** This button brings up the file selection dialog to select the *.WASH files that were created with Define Washing Curve command. One or multiple files may be selected with CTRL or SHIFT.
- **Missing:** If more than one WASH file is selected, then this Missing button is active. This displays any attributes that are missing from the samples or drillholes. These samples will not be used in the wash analysis, as all attributes must be in all sample points for the proper results. If some attributes are missing, the window appears as follows.

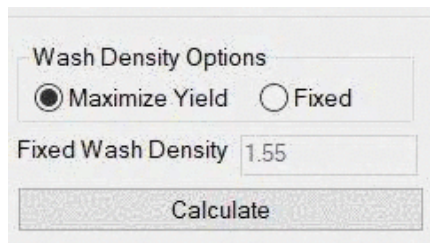


Input Lithology		
	Attribute	Raw Avg
1	ASH CONTENT	28.83
2	CALORIFIC VALUE	21.43
3	INHERENT MOISTURE	1.86
4	TOTAL SULPHUR	.80

Select
Loaded 25

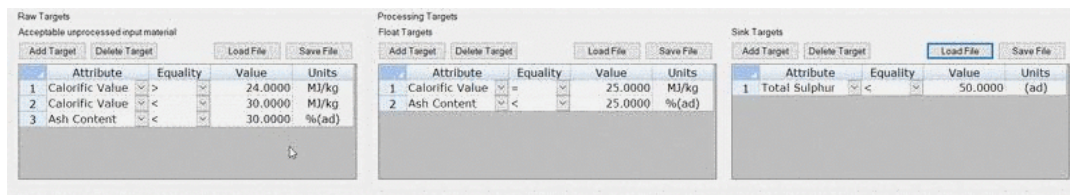
Input Lithology

- **Attribute:** These are the attributes found in the drillhole, stratacalc file, and wash file to be used for the washing results.
- **Raw Average:** This displays the raw average values of all the samples from all the drillholes and lithology intervals.
- **Select:** Even though drillholes were selected at the start of the command, this option allows for adding or reselecting different drillholes for the analysis. The Loaded number refers to the total number of intervals that are used for the analysis, this could be more than the total drillholes.



Wash Density Options

- **Maximize Yield / Fixed:** These two options determine the calculation method. Maximize Yield will require the attribute Targets sections filled out, and the program will calculate and report the density to meet the desired attribute targets. The Fixed Density method will calculate and report the attribute results if the material is to be washed to this specific density.
- **Fixed Wash Density:** This density is what will be washed to with the Fixed option set above.
- **Calculate:** After selecting the density options, and if the targets are filled out for Maximize Yield, then selecting this button will display the results in the bottom sections, and will be ready to report the results.

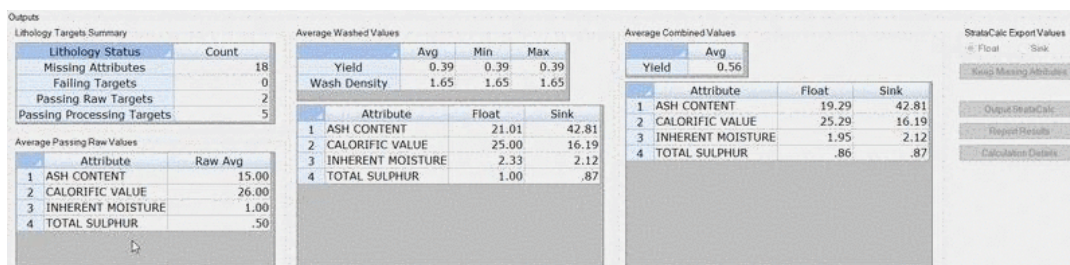


The targets are only used when the Maximize Yield option in the Wash Density Options is used. All three target fields have an Add and Delete Target for the attribute rows. Each target can be saved out as a separate file, with a TRGT file extension.

Raw Targets: The raw targets are first analyzed to see if these conditions are met using the raw material without any washing. If the material passes and meets these targets, then there is no need to wash, and the material is not processed and will be used as is, or raw. The unprocessed material is acceptable and will be used without washing.

Processing Targets - Float Targets: These attribute targets are the quality of material that is desired from the result of washing. Entering the minimum or maximum values of the attributes determines what density the material needs to be washed to for float.

Processing Targets - Sink Targets: This is the target of the sink material, usually waste rock. If the attribute target is used here (not required if targets are set for float), then both the float and sink targets must be met for the material to be washed and pass.



Lithology Targets Summary: The total number in the count should add up to the total number of Loaded Input Lithology samples.

- **Missing Attributes:** Displays the number of samples that were missing at least one of the attributes, so the wash analysis could not be performed on it.
- **Failing Attributes:** Displays the number of samples that failed to meet the attribute targets, so washing was not applied to them.
- **Passing Raw Targets:** Displays the number of samples that met the attribute targets without even needing to be washed.
- **Passing Processing Targets:** Displays the number of samples that did meet the attribute targets and were properly washed and output to the results for modeling.

Average Passing Raw Values: If the Raw material attributes meet and pass the targets, this window displays the average values in raw, unwashed state.

Average Washed Values

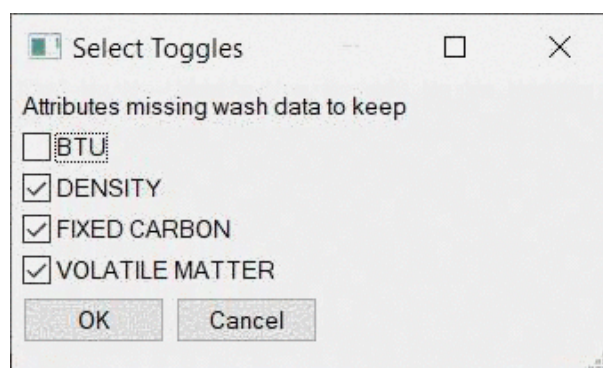
- **Yield:** This displays the percentage of total material meeting and passing the attribute targets when washed to the Wash Density.
- **Wash Density:** This is the density either calculated based on the desired attribute targets, or it is the user entered density to determine the attribute values at that density.
- **Attribute Float / Sink:** These are the average values of all the washed, float and sink material when the material is washed to the specified density meeting the targets.

Average Combined Values

- **Yield:** This reports the total percent yield of both the Raw and Washed (passing Float) of the material.
- **Attribute Float / Sink:** This displays the combined average of both the Raw and Washed (Float) material attributes.

StrataCalc Export Values: This is to select whether to save the attributes out as the Float or Sink values.

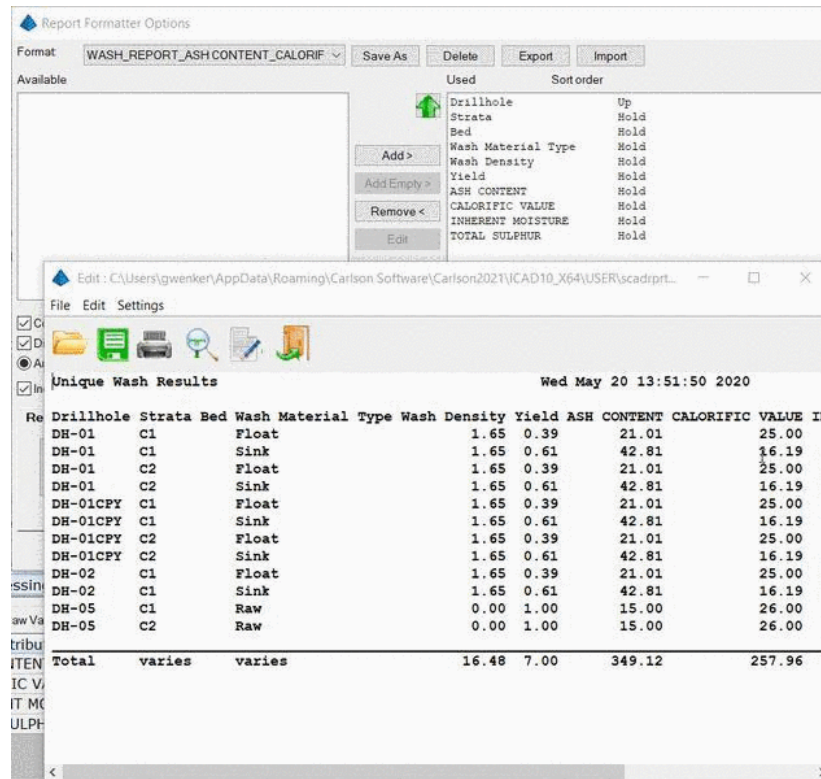
Keep Missing Attributes: Selecting this option gives the option to keep and report any attributes that exist in the drillholes but were not defined or used for any washing analysis. Deselecting any will not include the attribute in the output.



Output StratCalc: In order to make a geologic model of the washed attributes, the values at the selected /

calculated density need to be saved out to a StrataCalc STC file. This file can be viewed and edited with the separate command StrataCalc Datasheet. When the user is prompted to select drillholes or StrataCalc file, this STC file can be the source of the datapoints for the modeling.

Report Results: The Report Formatter will contain the calculated results and allow for custom viewing in many formats such as TXT, HTML or Excel. The washed Float and Sink attribute values for each drillhole, seam and sample point are displayed in the report. The Yield of the Float and Sink totals 1.0. Raw quality values are also included in this report that did not require washing.



Calculation Details: This report shows all the calculation details, attributes used and missing or not used, targets, and the calculations used. This report is a good reference to use as a backup or document the exact washed attribute calculation. Each drillhole and sample interval is listed, including the WASH file that is used for the calculations.

Save Settings/Load Settings: This command creates a *.AWASH file saves all the entered settings for this command.

Prompts

Command: C:APPLYWASH

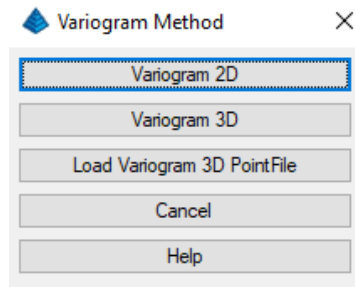
Apply Wash Dialog

Drop-Down Menu Location: StrataCalc ->Wash Filter->Apply Wash

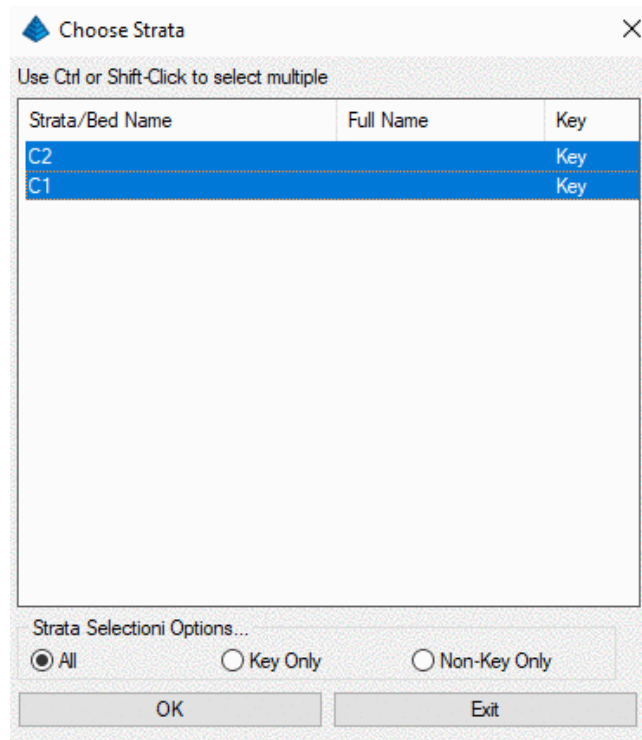
Keyboard Command: applywash

Calculate Variogram

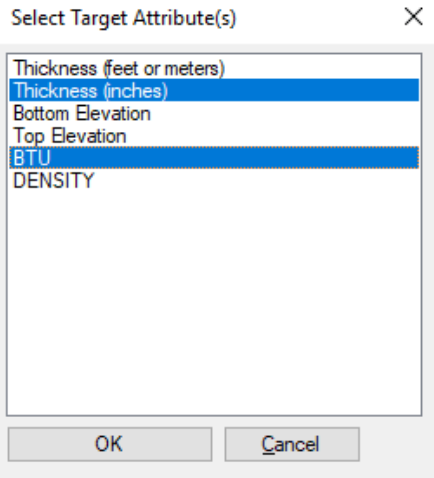
This command takes the selected drillholes and calculates the variogram data, outputting the Nugget, Sill and Range. These parameters are entered into the Carlson Kriging dialogs for gridding and block modeling. Variograms may be 2D or 3D. When the command is first activated, you will be prompted to select the variogram type, as shown below. Selecting Load Variogram3D point file will skip the drillhole, Strata/Bed, and attribute selection steps, and load a "DAT" point file directly and go into the 3D variogram dialogue.



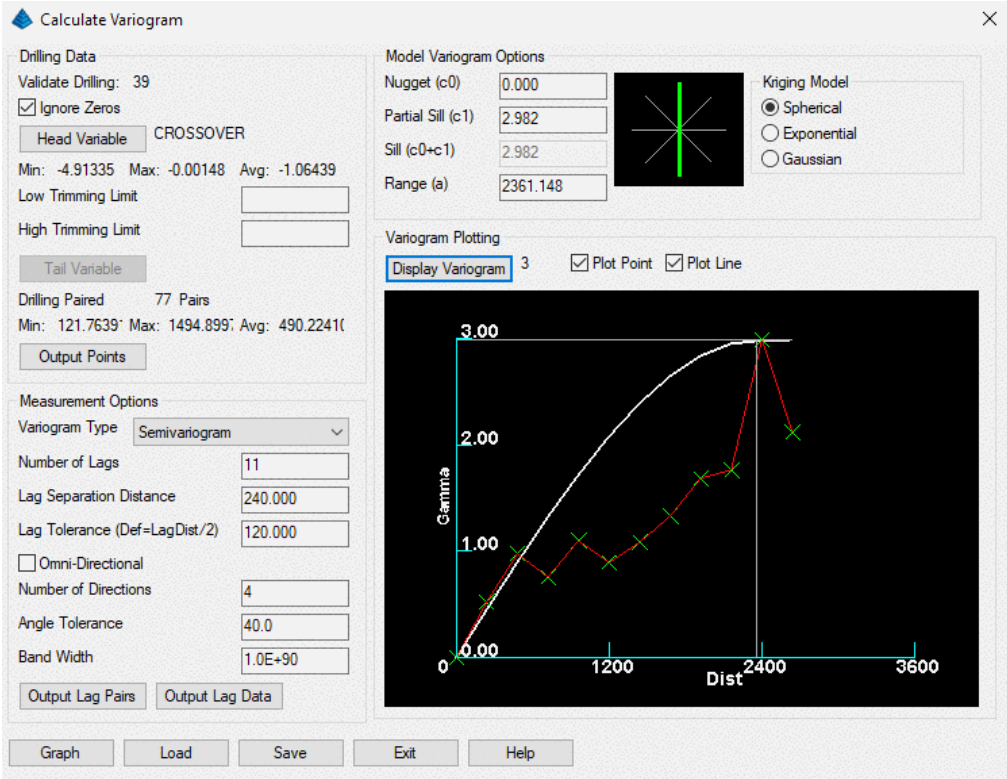
After selecting the variogram type, you will be prompted to select the drillholes, channel samples, and strata polylines to use for variogram calculation. You will then be prompted to select a strata to process, as shown below.



After selecting the strata to process, you will be prompted to select a value or values to process, as shown below.



The 2D variogram window is shown below.



Drilling Data:

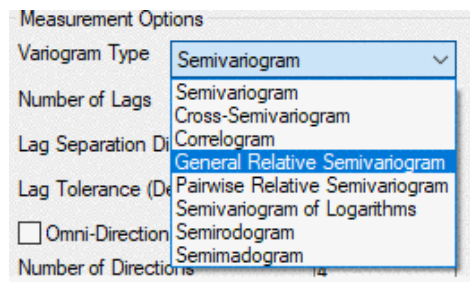
This section covers the base data points and attributes which are being used in the analysis.

- **Head Variable:** Lists the variable that is analyzed.
- **Tail Variable:** Lists secondary the variable that is analyzed for Cross Semivariogram and Correlogram.
- **Ignore Zeros:** Ignores any zero values in the drilling.
- **Validate Drilling:** Lists the number of holes used.
- **Min/Max/Average:** Lists the minimum, maximum and average values of the sample points.
- **Low and High Trimming Limit:** Trimming limits will provide a range selection to process. A value of 0 trimming will delete null values.
- **Drilling Paired Dist.:** This shows the number of found pairs in the drillholes.
- **Min/Max/Avg:** This is the distance between all sampled pairs of drillholes.
- **Output Points:** Outputs a ".csv" file of the points in the analysis for verification and further analysis..

Measurement Options:

This bottom left section of the dialogue covers the several options for the measured variogram

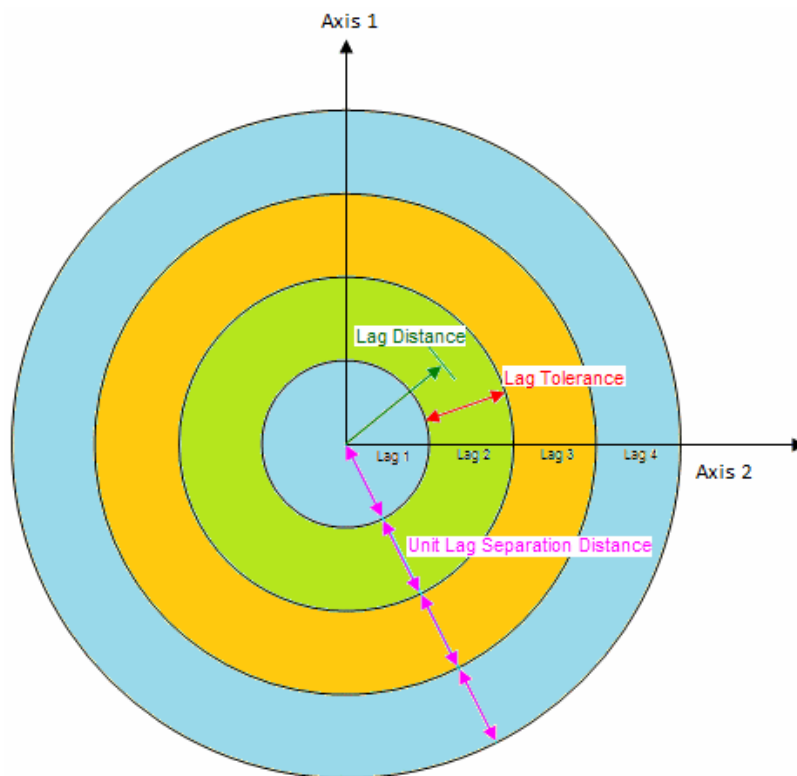
The type of Variogram Type button is a dropdown with the following options.

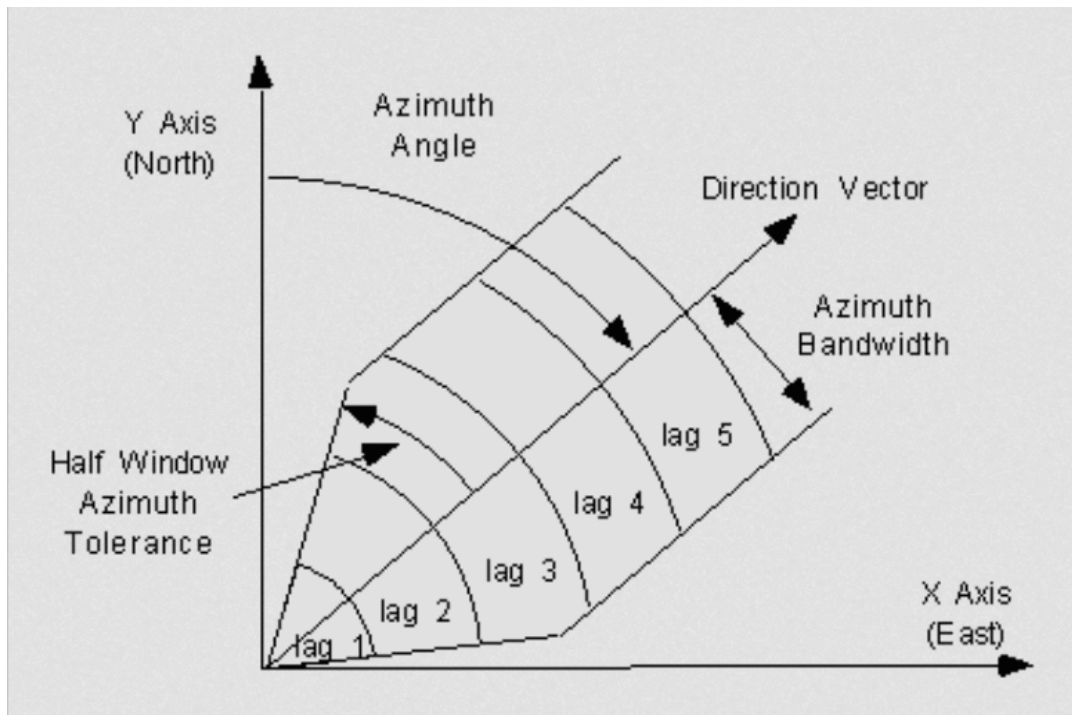


- Semivariogram
- Cross Semivariogram
- Correlogram
- General Relative Semivariogram
- Parwise Relative Semivariogram
- Semivariogram of Logarithms
- Semirodogram
- Seminadogram

The Cross Semivariogram and Correlogram calculate the relationship between two different attribute types and distance. To calculate them a tail variable will have to be selected.

Variogram Measurement Attributes:



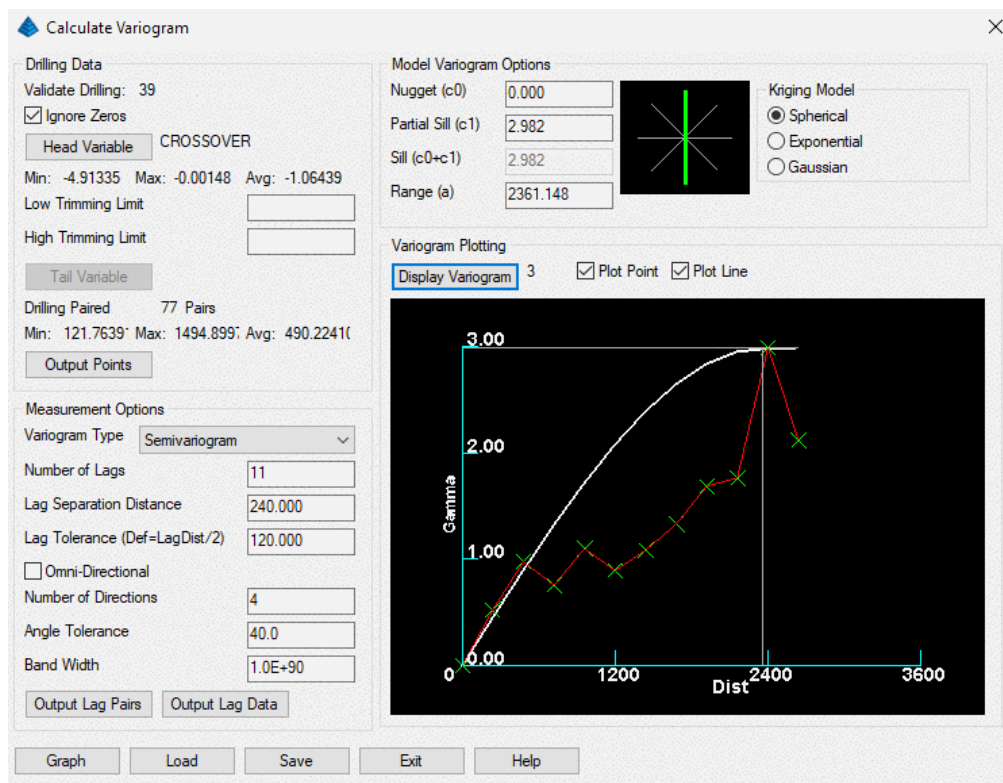


- **Number of Lags:** Set the number of lags, or pairs to model.
- **Lag Separation Distance:** Distance between the pairs. The distance between the lags should be set to the minimum drilling paired distance.
- **Lag Tolerance:** Lag Tolerance is Lag Separation Distance divided by 2.
- **Omni-Directional:** Omni-Directional will look in multiple circular groups for pairs. If it is off, then it will look just directional for the pairs.
- **Number of Directions:** Directions it will look for pairs
- **Angle Tolerance:** angle to stay under for looking for pairs.
- **Band Width:** How wide to look for the pairs.
- **Output Lag Pairs:**
- **Load/Save:** Saves the variogram data as a VIO file that can be loaded later.

Pressing Graph will show the plot as a red line and the points as green X. They can be turned on or off to display in the window. The curve must then be fit to the red line manually with left mouse by double clicking to locate the values.

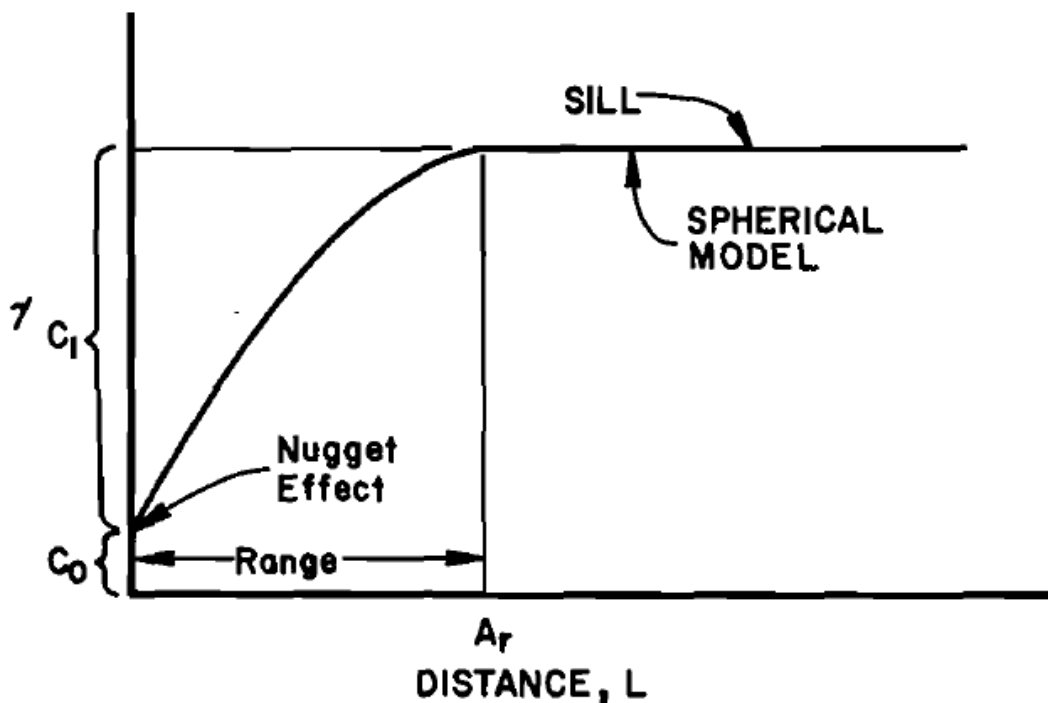
Fitting:

Omni-Directional: Click Exponential option beside big circle on the right to begin. Left click left mouse and drag on the plot to set Range, Sill and Nugget or enter them in the boxes to update the curve. To move the Nugget, left click on the far left side, at the beginning of the curve.

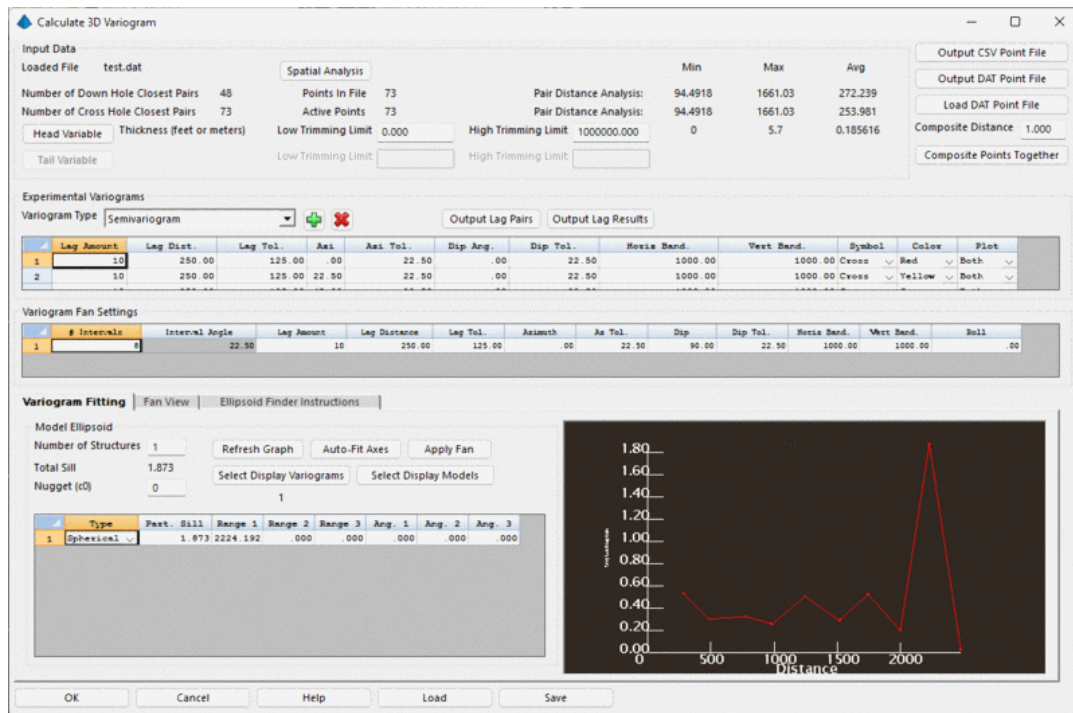


Directional Trend: The variogram plots redraw by clicking on the lines on the large directional window at the top, with the green line, so the 4 angles would be: 0 Azimuth, 135 Azimuth, 90 azimuth and 45 azimuth. The resulting values are displayed above for geostatistical modeling.

The nugget, sill and range are labeled here, for a spherical model:



The 3D variogram is shown below. It calculates a 3D ellipsoid of anisotropy which is used in 3d Kriging by following the steps in "Ellipsoid Finder Instructions Tab".



Input Data:

This section of the dialog deals with the input data, summarizing, and manipulating it.

- **Loaded File:** Informs user the file which has been loaded or whether using the selected data from CAD.
- **Points in File:** The number of data points in the file. Each point does not hold all of the selected attributes.
- **Active Points:** The number of data points which have both the head and tail variable.
- **Closest Pairs:** Gives data on each points closest neighbor. This information helps select the lag distance and decide whether compositing is necessary
- **Spatial Analysis:** This button will scan the sample data and generate summary values which are used to selected values for the Experimental Variogram.
- **Low Trimming Limit:** This value sets the minimum attribute value to consider for the variogram.
- **Maximum Trimming Limit:** This value sets the maximum attribute value to consider for the variogram.
- **Head Variable:** This is the active value for all variogram types, and it can be changed
- **Tail Variable:** This second variable used in Correlogram and Cross Semivariogram analysis

The following items are utility for saving, loading, and manipulating the input data points

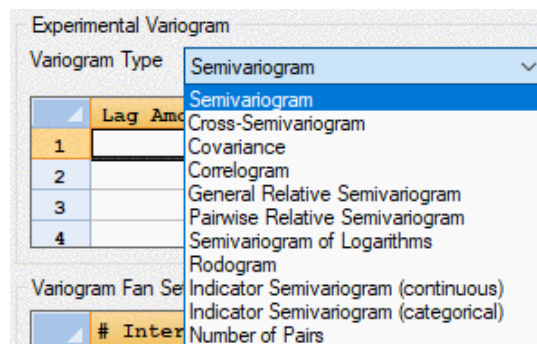
- **Output CSV Point File:** Gives data on each points closest neighbor. This information helps select the lag distance and decide whether compositing is necessary

- **Output DAT Point File:** This button will scan the sample data and generate summary values which are used to selected values for the Experimental Variogram.
- **Load DAT Point File:** This will load a dat file into the model. Useful for saving work from point compositing. Only selected Head and Tail variables will be saved
- **Composite Distance:**The distance which points should be combined together. When compositing points values are averaged together. Compositing is done by selecting "Composite points Together" button.

Experimental Variogram:

This section deals with specifying the settings which to measure directional variograms and how they should be displayed.

- **Variogram Type:** The Variogram types include all of the options of 2D as well as including a plot of the number of pairs. This isn't a variogram type, but instead a shows the number of pairs used to determine the value of each lag, which makes validating data easier.



- **Output Lag Pairs:** This will output a csv file with a list of each lag pair and their associated directional variogram information and the data values.
- **Output Lag Results:** This will output a csv file with the resulting values for each lag bin which can be used to store the results of past experiments.
- **Variogram Table:** Each row represents a measured variogram with controllable direction and direction measurement tolerances. How each variogram is displayed can also be controlled with Symbol, Color, and Plot options.

Variogram Fan Settings:

This section deals with laying out a set of variograms in a plane. The Azimuth and Dip define the first direction of the plane normal and the roll describes the spin of the variogram fan. It is suggested to try apply the fan with Azimuth at 0, Dip at 90, and Roll at 0 and then vary the Roll values to see the changes in experimental variograms.

Model Ellipsoid items deal with fitting a model variogram, and the three directional variograms needed for a 3d Kriging.

Experimental Variograms are not automatically updated for each change in the Variogram Fan Settings. This can allow for custom settings per Experimental Variogram. Select "Apply Fan" to apply to create a set of Experimental Variograms according the the Variogram Fan settings.

The Variogram Fan Settings and Experimental variogram directions are automatically set during the Ellipsoid finder steps, so no further adjustments beyond lag distance settings or direction interval are recommended.

Variogram Fitting:

This section deals with displaying the results of Experimental Variograms and creating the model variograms which are used by 3D Kriging.

The display parameters can refresh for the graph to show adjustments in experimental variogram and adjust which experimental variograms are being displayed.

The Model ellipsoid section deals with setting the values of the modeled variogram and each item is described below

- **Select Display Models:** Specify which or none of the model variograms are being displayed, only Experimental Variograms 1, 2, 3 can be modeled and they are associated with the Major, Minor, and Intermediate Axis of Anisotropy respectively
- **Auto Fit Axes:** This will automatically set the nugget, sill and ranges for all 3 axes according to a single structure model. This uses a max variography point fitting technique, final judgement and adjustment according to statistical judgement is recommended.
- **Number of Structures:** The number of unique variography regions. Each structure is associated with one row of the Model Ellipsoid table
- **Total Sill:** The sum total of each structure's sill and the nugget effect
- **Nugget Effect:** The expected variation at zero distance, a stand in for the inherent variation of the rock mass
- **Range:** The distance at which the model variogram reaches the specified sill value
- **Angle:** These describe the orientation of the Model Ellipsoid. Angle 1 describes the Azimuth, Angle 2 describes the dip, and Angle3 describes the roll. These values do not affect the plot of the model ellipsoid, but the Kriging analysis. It is not suggest to change them once set by the Ellipsoid Finder

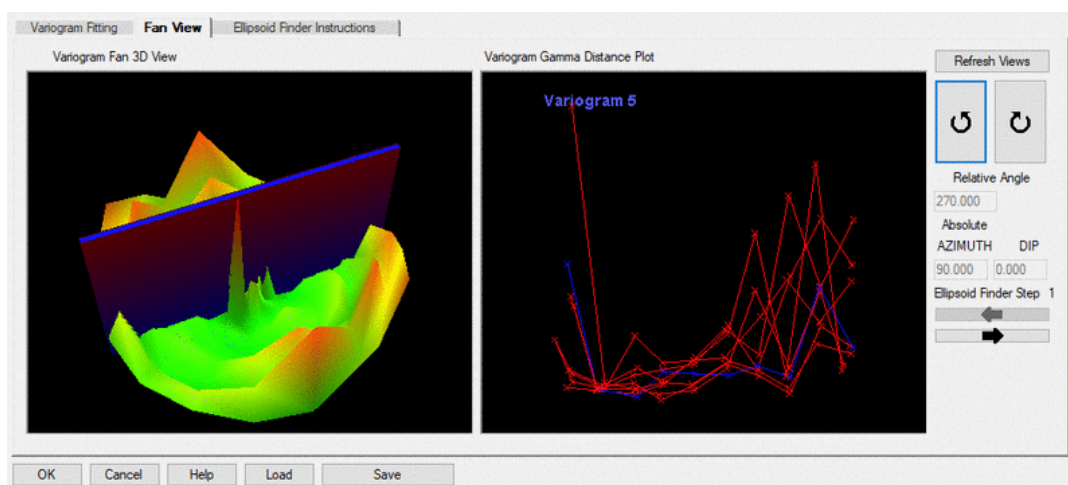
Fan View:

This section deals with visualizing the variography in 3D and 2D plot forms to assist in the selection of minimum or maximum directional variance. Following the steps

Fan View:

This section deals with visualizing the variography and selecting the minimum or maximum directional variance for the Ellipsoid Finder. The Variogram Fan 3D view shows the variography in the z axis, along the variogram fan the X, Y axis. The adjustable plane is the indicator of the selected direction. The "Variogram Gamma Distance Plot" is the same plot which is in Variogram Fitting but with all of the variograms showing. The selected variogram is labeled and shown in blue.

Selecting the rotational arrows switches the target Experimental variogram. Selecting the left right arrows applies the selected variogram direction, updates the Variogram Fan and Experimental Variograms.



Ellipsoid Finder Instructions:

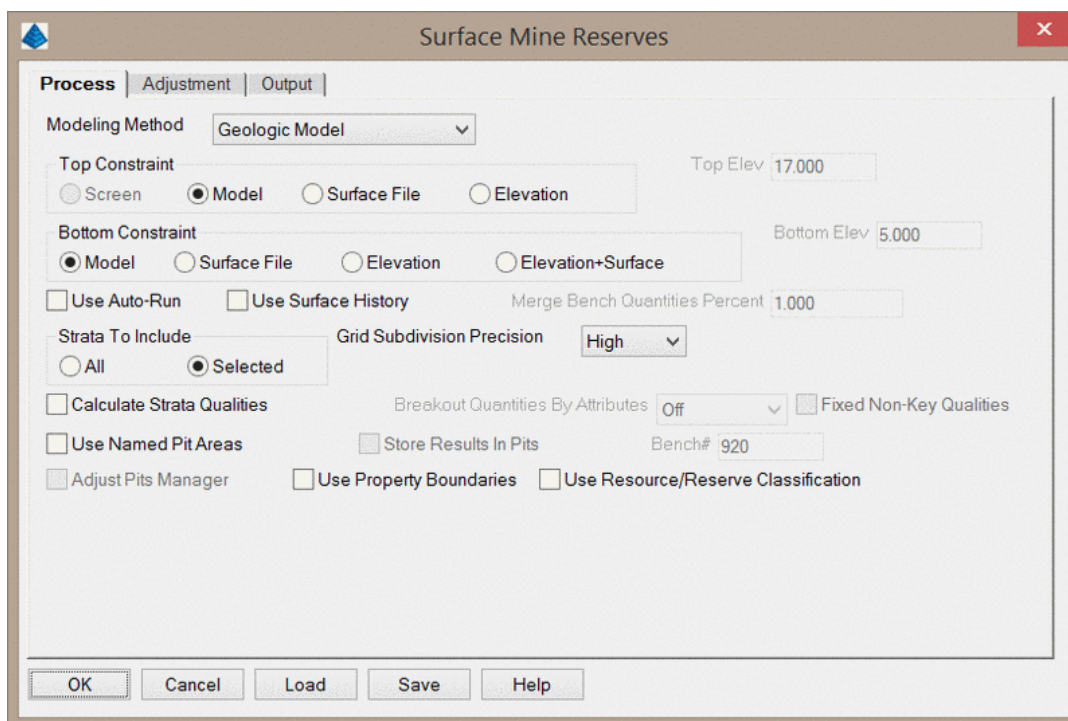
This section gives instructions and tips on calculating the Ellipsoid of Anisotropy. Selecting the left right arrows applies the selected variogram direction. The prompted minimum and maximum directions of variance must be selected in the "Fan View" tab. Once all the directions are found the Sill and Range values must be selected in the "Variogram Fitting" tab. Selecting the "Save" button will save the Modeled ellipsoid of anisotropy and "Load" will bring back previously saved

Pulldown Menu Location: StrataCalc

Keyboard Command: vgramrun

Surface Mine Reserves

This command calculates quantities and qualities directly from drillholes or from predefined Geologic/Mining Models. Additional outputs include stripping ratio contours, composite thickness grids, and more. The dialog is divided into three tabs - Process, Adjustment, and Output - as shown below. It is important to note that if an option is grayed out (not editable) it will not affect the calculations. Some options cannot be used together, and so the unnecessary options will be grayed out to avoid confusion.



Modeling Method: This dropdown menu sets the modeling method to be used for calculations.

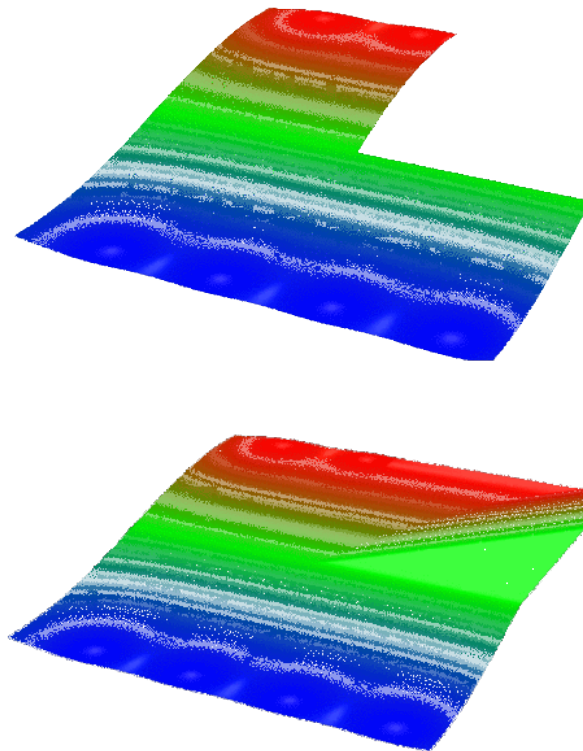
The first seven options (Triangulation, Inverse Distance, Kriging, Polynomial, Linear Least Squares, ABOS Method, and Nearest Neighbor) will calculate the reserves from drillholes, strata polylines, and pit/channel samples using the selected modeling method. This option will create grids for calculations, but the grids will not be saved. For more information, see the section of the help manual corresponding to the Make Strata Grids command.

The Block Model option will also calculate the reserves from drillholes, strata polylines, and pit/channel samples by first creating a block model. This block model will not be saved.

The Geologic Model and Mining Model options will calculate the reserves from pre-calculated grid files stored in a .pre or .mmd file. It is highly recommended that you use either a Geologic Model or Mining Model for calculating reserves, as the other options will need to generate grids on-the-fly every time the command is run. Using a Geologic/Mining Model will greatly increase calculation time and reproducibility of the results.

The only difference between a Geologic Model and a Mining Model is the file extension (however, a Mining Model must be based on elevation grids whereas a Geologic Model may be based on elevations or thickness grids). The distinction is made purely for organizational purposes in that the Geologic Model can be used to represent the true geology while the Mining Model can be used to represent the strata as they will be mined. For example, some strata layers may need to be composited in the Mining Model. For more information on creating Geologic/Mining Models, see the sections of the help manual corresponding to the Define Geologic Model, Define Mining Model, and Geologic to Mining Model commands.

Here it is important to note that this command will temporarily extrapolate grid files in the Geologic/Mining Model. This replaces all null values with the nearest known data point. These changes are not saved to the .grd file itself however. When grids are extrapolated, you will receive a notification of which grids were temporarily modified for the calculation. This can increase the amount of time required to complete the calculation, and for this reason, it is recommended that you manually extrapolate grid files using either Grid File Utilities, or the Extrapolate function in Define Geologic Model. When the program automatically extrapolates surface, it will hold the values at the edge of the grid and extrapolate horizontally. An example of a surface file with null values, and the resulting grid that would be temporarily created for this calculation is shown below.



Top Constraint: This option sets the upper limit for the reserve calculation. All calculations will be based on a layering of strata grids. This upper limit will omit any strata information that may exist above it. Four options are available for use:

The Screen option will prompt you for 3D entities (usually elevation contours) to create a grid file. This grid file will not be saved, however.

The Model option will use the surface file specified in the Geologic/Mining Model.

The Surface File option will prompt you for an existing grid or TIN file.

The Elevation option will use a flat elevation as set in the **Top Elev** field.

Bottom Constraint: This option sets the lower limit for the reserve calculation. This lower limit will omit any strata information that may exist below it. Four options are available for use:

The Strata Model option will use the bottom-most elevation grid file specified in the Geologic/Mining Model.

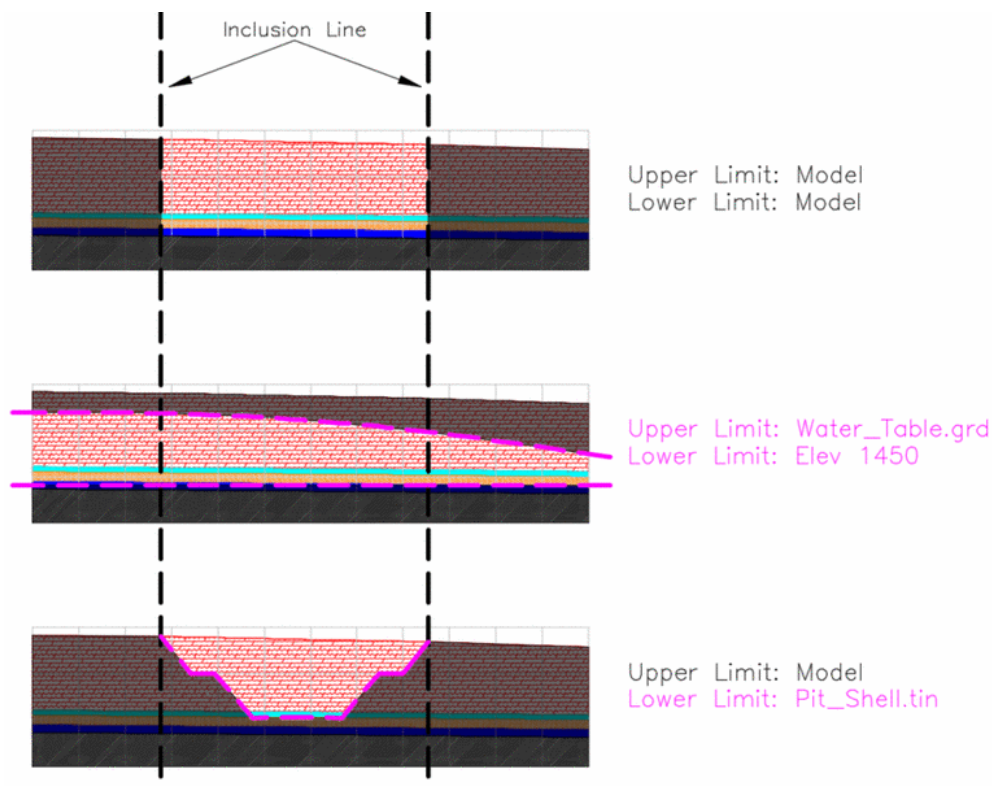
The Surface File option will prompt you for an existing grid or TIN file. This is most commonly used when calculating the reserves within a pit shell.

The Elevation option will use a flat elevation as set in the **Bottom Elev** field.

The Elevation+Surface option will allow you to define both an elevation and a surface file to limit the calculation. For example, you can constraint the calculation with a pit shell surface, but also with a bench elevation.

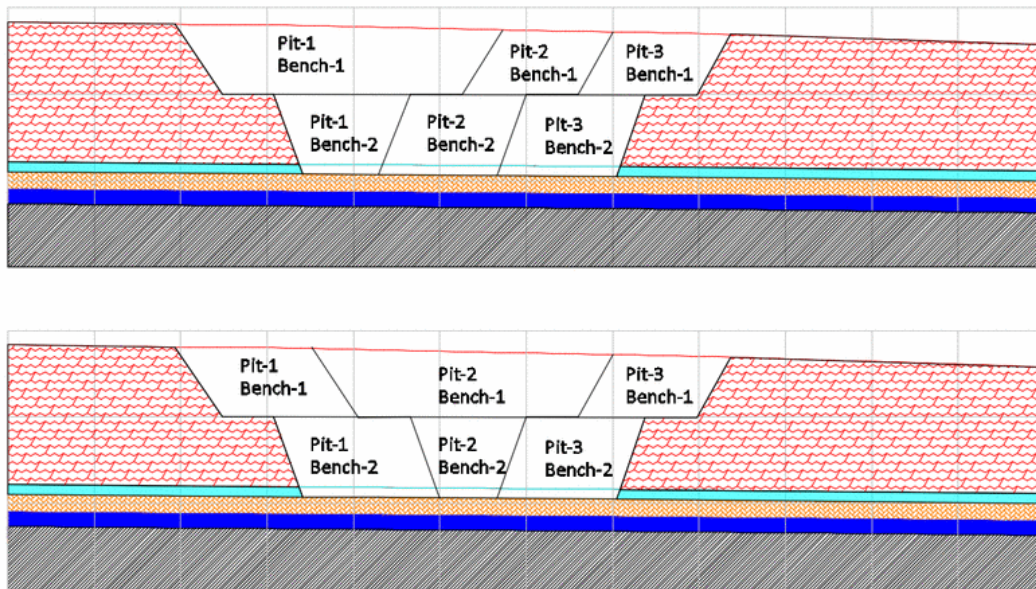
Top Elev and Bottom Elev: These fields will set the elevation limits for the reserve calculation.

Whenever selecting the upper and lower limits of the reserve calculation, it is important to visualize the options you are selecting. The below illustrations show some of the ways that the upper and lower limits can be used to constrain the calculation.



Use Auto-Run: This option will allow you to divide the reserves into benches according to a Surface Mine Reserves Autorun file (.sma file). Dividing reserves into separate benches is useful when using the Store Results in Pits option as it allows for more selective mining with the Surface Equipment Timing command. If this option is not used, results can still be stored in pits, but all data will be stored on a single bench. For more information, see the section of the help manual corresponding to the Define Surface Mine Auto-Run command.

Use Surface History: This option will calculate reserves according to a Surface History file (.gsq file). This file is essentially a list of grid or TIN files and can be created manually or with the Design Bench Pit command. When this option is used, reserves will be divided according to the volumes between the surface files (i.e. the first volume is calculated between the first and second surface file, the second volume is calculated between the second and third surface file, etc.). The below images show an example of how the reserves will be divided into three pits and two benches using a surface history file. The ordering of the surfaces in the Surface History file will have a significant impact on the calculations as it determines the cut angles to be used. For example, the first image shown below cuts out Pit 1 first, whereas the second image cuts out Pit 2 first. For more information on the Surface History file, see the section of the help manual corresponding to the View 3D Surface History command.



Merge Bench Quantities Percent: This option is only available when using a Surface History File. When active, this value is used as a tolerance for reporting volumes. For example, if this value is set to 1%, any volumes less than 1% of the total bench volume will be merged into the previous bench.

Strata to Include: This option determines which strata layers will be calculated for the Report. If the Selected option is chosen, you will be prompted with a dialog for which strata to include. If you are not using the Auto-Run option, the Selected option can be useful for storing specific strata layers onto a bench.

Grid Subdivision Precision: This option controls how grid cells will be subdivided along angled borders for more precise calculations. For the most accurate results, use the High option. Using the Low option will decrease accuracy in exchange for improved calculation time. For grid volume calculations, the program subdivides grid cells that cross a boundary. This setting controls how many times the cell is subdivided. Subdividing more increases the volume accuracy but takes longer. The default for this setting is High. It only makes sense to set it to Low when you're in a hurry and you've got a huge model and a lots of pits. Otherwise, the routine should be fast anyway that you might as well use High and get full precision.

Calculate Strata Qualities: This option controls if strata qualities will be calculated. These are the user defined attributes such as BTU, Ash, Moisture, etc. Non-user defined attributes such as thickness, area, etc. will always be calculated. When qualities are merged for reporting, by default they will be weight averaged according to the strata volume and density. Here it is important to note that the weight averaging of attributes will not account for null values. For example, if Strata A has a Moisture attribute of 0.3, but Strata B does not have a Moisture attribute

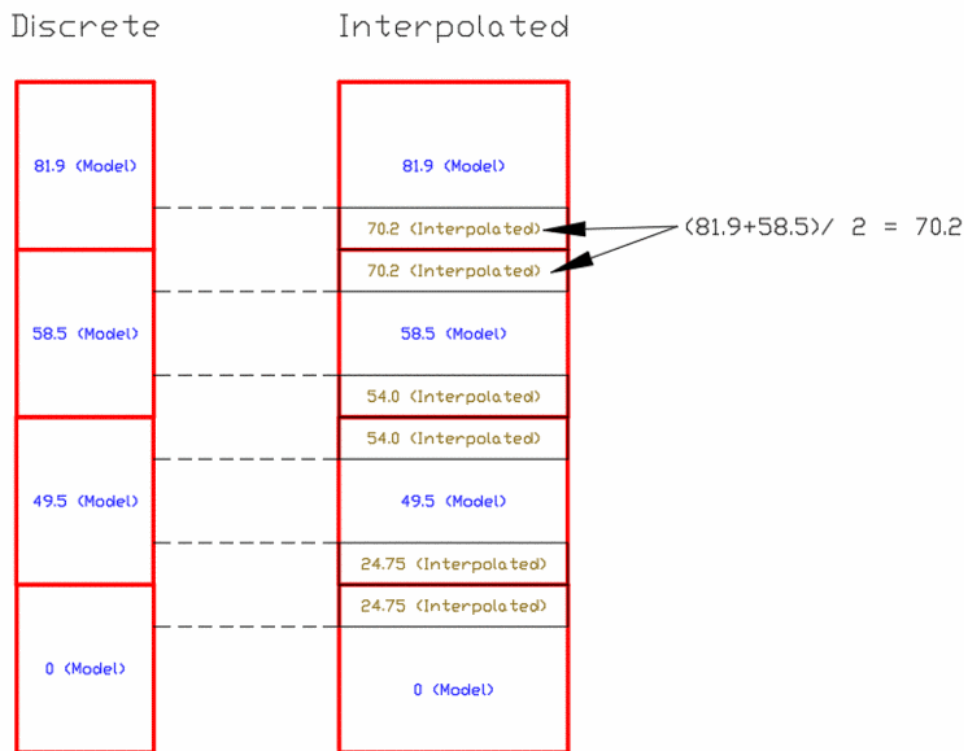
defined, then the combined Moisture content for the two stratum will be 0.3. To properly weight attributes, it is necessary to define an attribute value for each strata. Besides reporting the averaged quality, the program also reports the minimum and maximum quality values. When qualities have not been defined for all strata, you may find the Fixed Non-Key Qualities option useful.

Breakout Quantities by Attributes: This option can only be used when a .blk file is used (either generated on-the-fly from drillholes or when it is stored in a Geologic/Mining Model). When enabled, you will be prompted for a Grade Parameter File (.gpf file) which defines the various grades of the material. This option will report the quantities of material according to this file. For example, if the .gpf file relating to a limestone project defines Grade A material as having 90-100% Calcium, Grade B as having 80-90% Calcium, etc., then the report will calculate the tonnage, volume, and average Calcium content for each grade individually.

The Off option will not breakout quantities by attribute. You will not be prompted for a .gpf file with this option. The Interpolated option will actually interpolate block values between data points, thus allowing for variation of grade within each block.

The Discrete option will treat block values as discrete for the entire block. This prevents any variation of grade within individual blocks.

A comparison of the Discrete and Interpolated options is shown below. Notice that in the Discrete method, each red block is homogeneous. There is no variation of quality within the block. For the interpolated option, however, each red block is subdivided. Half of the block retains the modeled value, while the upper and lower fourth of each block is a calculated average.



Fixed Non-Key Qualities: This option will prompt you to enter a fixed value for each strata attribute found in the drillholes or in the Geologic/Mining Model. These values will be used for all non-key strata. This option is useful for properly weighting attributes due to dilution with non-key material. For example if you have defined an attribute for Sulfur content in your key strata, but not your non-key strata, this option will allow you to quickly assign a Sulfur value to the non-key material for attribute weighting. Using this option for proper attribute weighting is faster than modifying the drillholes or Geologic/Mining Model with these attributes. Here it is important to note that if a non-key strata attribute has been defined in the Geologic/Mining Model, this option will overwrite that value.

Use Named Pit Areas: This option will restrict the calculation to use only closed polylines that have been tagged as pits. Polylines may be tagged as pits using one of the various pit tagging commands in the Surface Mining Module > Boundary Pulldown Menu. These commands include Name Pit Polylines, Assign Pit Names By Layer, Pit by Interior Point, Pit by Interior Text, Pit Matrix Layout, Pit Layout by Advance, Pit Layout by Width, and Pit Layout by Rate. When this option is used, reserves will be split into their respective pits. When this option is not used, you may still use pit polylines (or other, untagged polylines) as inclusion areas, but the reserves will not be divided by pit.

Store Results in Pits: This option is only available when the Use Named Pit Areas option is used. This option will store the total non-key volume, key volume, key tons and all quality attributes in the pit polyline as extended entity data. These quantities and attributes can then be used by the Surface Equipment Timing command. Besides the quantity and attribute values, a Bench number is also stored with the quantities for sequencing each bench. Here it is useful to note that all the material stored on a single bench must be mined together. If two types of material are to be mined separately, they should be placed on separate benches with the Use Auto-Run option or by repeating the Surface Mine Reserves command and incrementing the Bench Number while selecting specific strata.

Rather than storing discrete values in the pits, the non-key volume, key volume, and key tonnage may be stored in the pits as grid files. If the Output Thickness Grids option on the Output tab is used, these three values will be stored in the pits as grids rather than discrete values. Note that you may also store grids in the pits with the Surface Mining Module > Boundary Pulldown Menu > Pit Timing Quantities > Assign Timing Grids command. When this command is run, the grids will overwrite the discrete values.

Bench #: This value will set the bench number for storing results in pit polylines. Note that any reserves calculated will be stored together on this bench number. Note that benches must be integer values - alphanumeric bench numbers are not supported.

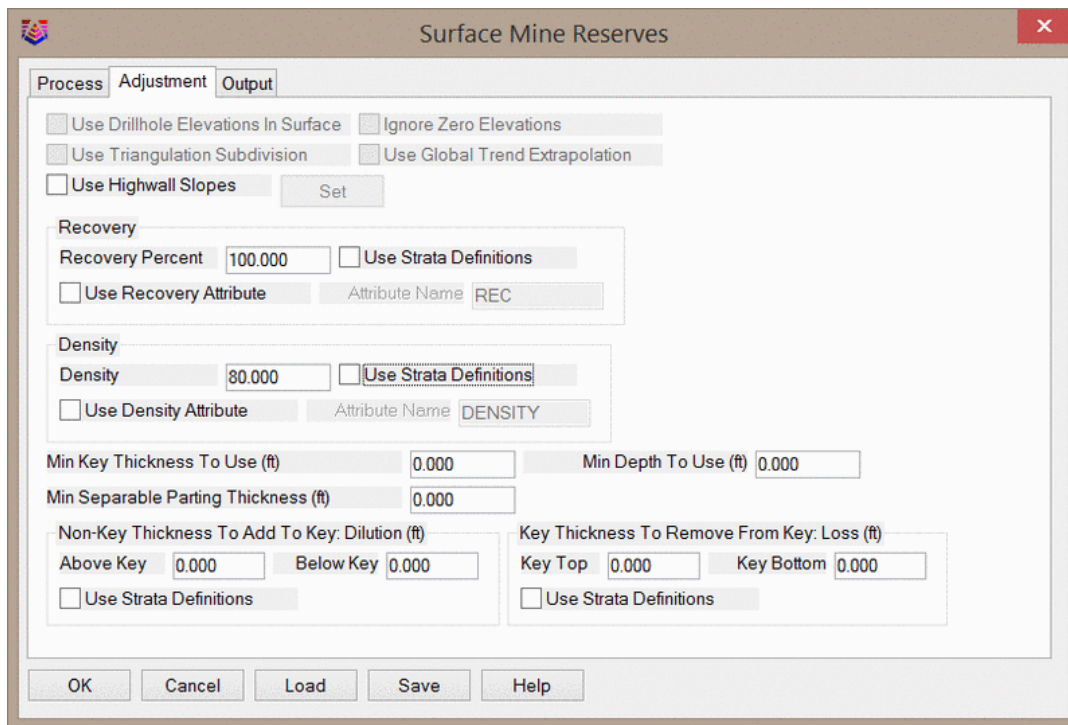
Adjust Pits Manager: This option will allow you to adjust the pit boundaries after the calculation is complete, then reprocess to check the updated results. This can be useful when trying to target a specific volume/tonnage to store in a pit. When this option is enabled, the below Pits Manager dialog will be docked to the bottom of the screen. You can then modify the pit polylines with standard CAD commands (move, grip edit, etc.). Clicking the Process button will then rerun the calculation and display the results. Clicking the Report button will send the results to the Report Formatter. Clicking the Exit button will close the Pits Manager without generating a report.

Process	Pit	Strata	Pit Area S.F.	Total C.Y.	COAL Tons	BTU
Report	Pit 1	C1_TOP	410462.8	1242447.5		
	Pit 1	C1_KEY	410462.8	101102.0	120109.2	12007.713
	Pit 1	C2_TOP	410462.8	325767.2		
	Pit 1	C2_KEY	410462.8	91401.2	108584.6	11492.317
Exit	Pit 2	C1_TOP	233272.6	961928.5		

Use Property Boundaries: This option will divide the reserves according to Property Boundaries, which are closed polylines that have been tagged using the Assign Property Names or Property Names by Text command (available in Surface Mining Module > Boundary Pulldown Menu and Underground Mining Module > Property Pulldown Menu). As long as an inclusion polyline is selected, these property boundaries will be automatically selected when the calculation is performed, even if the property boundaries are on a frozen layer. If an area lies outside of a tagged property boundary, the reported field for Owner will simply say "Unknown" rather than using one of the tagged owner names. It is important to note that "island" property boundaries are supported - for example, if Property A is completely inside of Property B, the program will not double-count the volume/tonnage in Property A.

Use Reserve Classification: This option will divide the reserves according to a Reserve Classification file (.rsv

file) into Measured, Indicated, Inferred, and Hypothetical categories. This file can be generated with the Geology Module > Grids Pulldown Menu > Reserve Classification command. When this option is used, the Report Formatter will include a new attribute called "Reserve Class."



Use Drillhole Elevations in Surface: When calculating the reserves directly from drillholes, this option determines if the drillhole collar elevations will be used to generate the topographic surface.

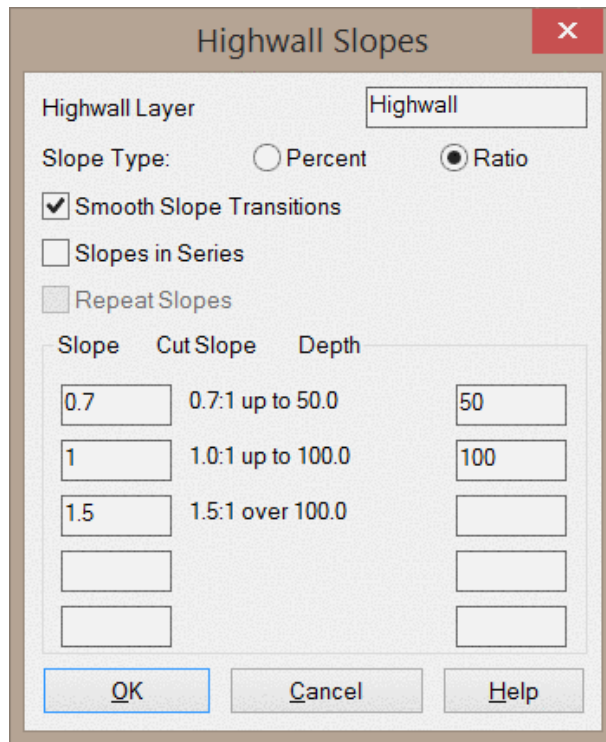
Ignore Zero Elevations: When calculating reserves directly from drillholes, this option will determine if entities at zero elevation will be used to generate the topographic surface. Generally, this option is useful in that it allows you to broadly select entities in the drawing without manually filtering out zero elevation entities.

Use Triangulation Subdivision: This option only applies when using the Triangulation or Polynomial modeling methods to calculate reserves directly from drillholes. When enabled, this option will subdivide triangles to create smoother surfaces.

Use Global Trend Extrapolation: This option only applies when using the Triangulation or Polynomial modeling methods to calculate reserves directly from drillholes. When enabled, this option finds the average slope and direction of the interpolated data and applies this slope to extrapolate to the full extents of the grid.

Use Highwall Slopes: This option will apply highwall slopes to the inclusion polylines to simulate a more realistically shaped pit. This option is limited in that the inclusion polylines selected for reserve calculation must represent the toe of the highwall. This option will project highwalls upwards until it reaches the topography. This option does not account for benches, and so more complex calculations will need to be made using a Surface History File generated from the Design Bench Pit command.

Set (Slopes): This button will open the dialog shown below, which controls the highwall slopes to use.



Highwall Layer: This field sets the layer to use for the highwall linework. This linework will be drawn as a 3D polyline where the highwall projection meets the surface.

Slope Type: This option controls if the highwall slope is expressed as a percent or as a ratio.

Smooth Slope Transitions: This option will gradually transition between slope angles as depth increases around the pit. In the above example, the pit will have an overall slope ratio of 1.0 where the cut depth is between 50 and 100', and a slope ratio of 1.5 for cut depths greater than 100'. In areas where the cut depth is exactly 100', this option will smoothly transition from a slope ratio of 1.0 to 1.5 rather than abruptly changing the slope angle.

Slope in Series: This option will use all slope angles specified in the dialog. Using the above dialog as an example, the cut angle from 0-50 feet (or meters) will always be a 0.7 ratio. After this first 50 feet (or meters) of cut, the slope will be a 1.0 ratio for the next 50 feet (or meters). Any remaining cut will use a 1.5 ratio.

Repeat Slopes: This option will repeat the slopes with specified depths until the projection meets the surface topography. In the above example, the program will repeat cut ratios of 0.7 and 1.0 until it reaches the surface. Since no cut depth has been specified for the 1.5 cut ratio, this slope angle will not be used with this option.

Recovery Percent: This value will set the overall recovery percent for key strata. This is applied after adjustments for non-key dilution and key loss. The volume of material that is not recovered will be reported as non-key material.

Use Strata Definitions: This option will use the recovery percent specified in the current Strata Definition file. This allows you to set a recovery for each strata individually.

Use Recovery Attribute: This option will set the recovery percent according to an attribute defined in the drillholes or in the Geologic/Mining Model. This allows you to use a grid file to set the recovery percent, thus allowing the recovery of each strata to vary.

Attribute Name: This field sets the recovery attribute name to search for in the drillholes or Geologic/Mining Model. This attribute name must exactly match the attribute in the drillholes/model, or else the recovery percent will not be applied.

Density: This value sets the density to use for all strata. Density is always expressed in lbs/ft³ or kg/m³.

Use Strata Definitions: This option will use the density specified in the current Strata Definition file. This allows you to set a density for each strata individually.

Use Density Attribute: This option will set the density according to an attribute in the drillholes or in the Geologic/Mining Model. This allows you to use a grid file to set the density, thus allowing for a varying density within each individual strata.

Attribute Name: This field sets the attribute name to use for density in the drillholes or Geologic/Mining Model. This attribute name must exactly match the attribute in the drillholes/model, or else the density will not be applied.

Min Key Thickness to Use: This value sets a minimum key thickness for reporting. Any areas where the key strata is less thick than this value, the strata will be reported as non-key discard material.

Min Depth to Use: This value sets a minimum depth of cover for key strata for reporting. Any areas of key strata that have less than the specified thickness of cover will be reported as non-key discard material. An example use of this option is accounting for oxidized coal that is (or is close to) outcropping.

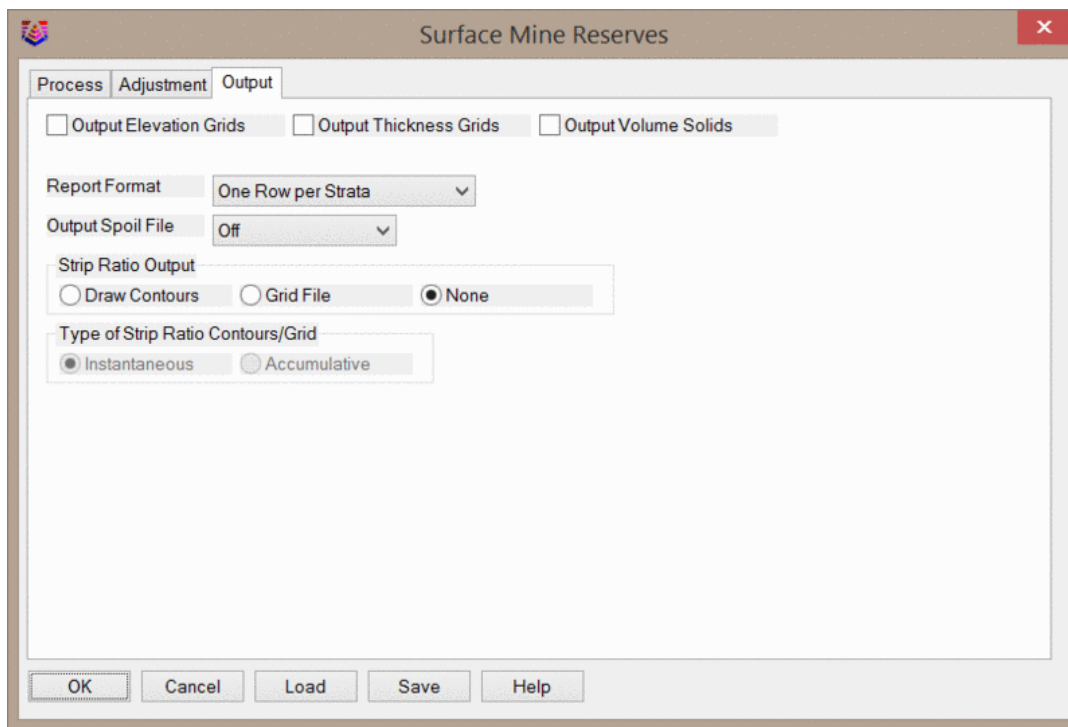
Min Separable Parting Thickness: This value will be used as a tolerance for calculating volume of non-key material between key strata based on thickness. For example, if this value is set to 0.5, any non-key material sandwiched between key strata that is less than 0.5 feet (or meters) thick will be reported as key material. Areas where the non-key strata is greater than 0.5 thick will not be affected.

Non-Key Thickness To Add To Key (Dilution): Above Key/Below Key: These values set the amount of non-key thickness above/below each key strata that will be mined with the key strata. This diluted non-key material will not be mixed with the key strata, but will instead be reported as another key strata layer. The final report, however, can be formatted to account for the dilution of strata qualities. Note that dilution will be accounted for determining the Stripping Ratio in the final report and in grid/contour output.

Use Strata Definitions: This option will use the Dilution values set in the Strata Definition File. This allows you to set the Dilution for each strata individually.

Key Thickness To Remove From Key (Loss): Above Key/Below Key: These values specify the amount of key thickness above/below the key strata that will be subtracted from the key. This amount will be subtracted from the key quantities and reported as non-key material. Note that loss will be accounted for determining the Stripping Ratio in the final report and in grid/contour output.

Use Strata Definitions: This option will use the Loss values set in the Strata Definition File. This allows you to set the Loss for each strata individually.



Output Elevation Grids: This option will create grid files for the bottom elevation of each strata. Each grid will need to be named separately, and so it is recommended to instead use the Strata Grids Autorun to make these grids quickly.

Output Thickness Grids: This option will create composite grid files for the total key thickness, non-key thickness, and key tonnage (expressed in tons/sq ft or tonnes/sq m). Each grid file will need to be named separately. Note that when the option to Store Results in Pits is used, these grids will be referenced in the pit polylines. When this option is used, you will also be prompted to Divide Bench by Thickness. This division will create three more grid files that represent the divided thickness.

Output Volume Solids: This option will create solids of the mined bench according to the pit names. These solids will be automatically saved in a PIT_MODEL folder as .mdl files. This PIT_MODEL folder will be automatically created in the current project folder.

Report Format: This option controls the format of the report.

The One Row per Strata option will report each strata layer on a separate row. This default option is the most commonly used report format.

The All Strata on Same Row option will report all strata layers on a single row. This creates many extra columns for the report, but allows for easier comparison of pit volumes.

The Group Key/Non-Key Pairs option will place overburden strata on the same row as the key strata. This option searches for similar naming in the strata names. For example, the strata COAL_KEY and COAL_OB will be grouped together due to the similarity of the strata names.

Output Spoil File: This option allows you to create a Spoil Source File (.spo file) for use with the Spoil Placement Timing command. This file includes the volume and centroid location of waste material to be placed in spoil piles.

The Off option will not create a Spoil Source File.

The Non-Key Only will only include non-key strata in the Spoil Source File.

The All Strata option will include all strata in the Spoil Source File.

Strip Ratio Output: This option allows you to output stripping ratio as contours or a grid file. The units of stripping ratio will default to (yd³ waste)/(ton key) or (m³ waste)/(tonne key), but different units may be selected in Carlson Configure.

The None option will not output the stripping ratio as contours or a grid file. Note that when this option is used, the stripping ratio will still be calculated in the final report.

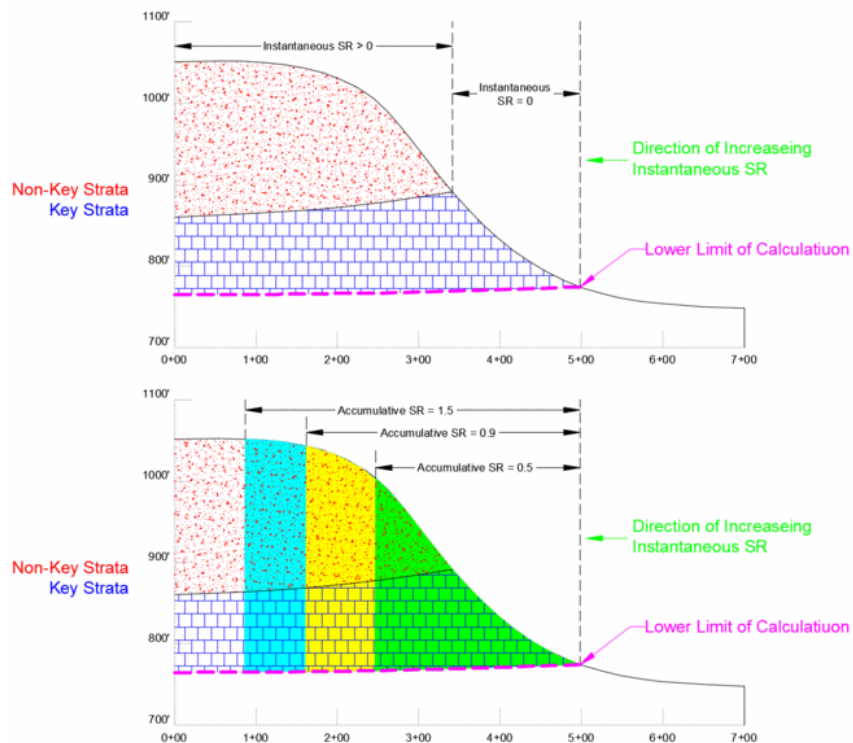
The Draw Contours option will output contour lines of the stripping ratio. This option will use settings similar to those used in the Contour from Grid File command.

The Grid File option will create a grid file that represents the stripping ratio.

Type of Strip Ratio Contours/Grid: This option controls how the stripping ratio is expressed when it is output to contours or a grid file

The Instantaneous option will calculate the stripping ratio for each individual grid node. This stripping ratio will account for all non-key and all key material within the calculation limits - any non-key material existing below the last key strata will also be used for the calculation of the stripping ratio. When the reserves are divided into benches, the stripping ratio will be reported by-bench.

The Accumulative option is intended for use in situations where the key strata outcrops and is relatively flat. The below images illustrate the way this form of stripping ratio is calculated based on direction. The program will first determine the lowest Instantaneous stripping ratio in the calculation area. The program will then determine in which direction the stripping ratio increases the least. If mining were to start in the area of lowest stripping ratio and then move one grid node in the direction of least increasing stripping ratio, the Accumulative stripping ratio would be based on all key and non-key strata between the start point and the next grid node location. In other words, imagine that mining starts at the outcrop of the key material. As mining progresses into the hillside, the accumulative stripping ratio is the overall stripping ratio of all material mined up to the current position.



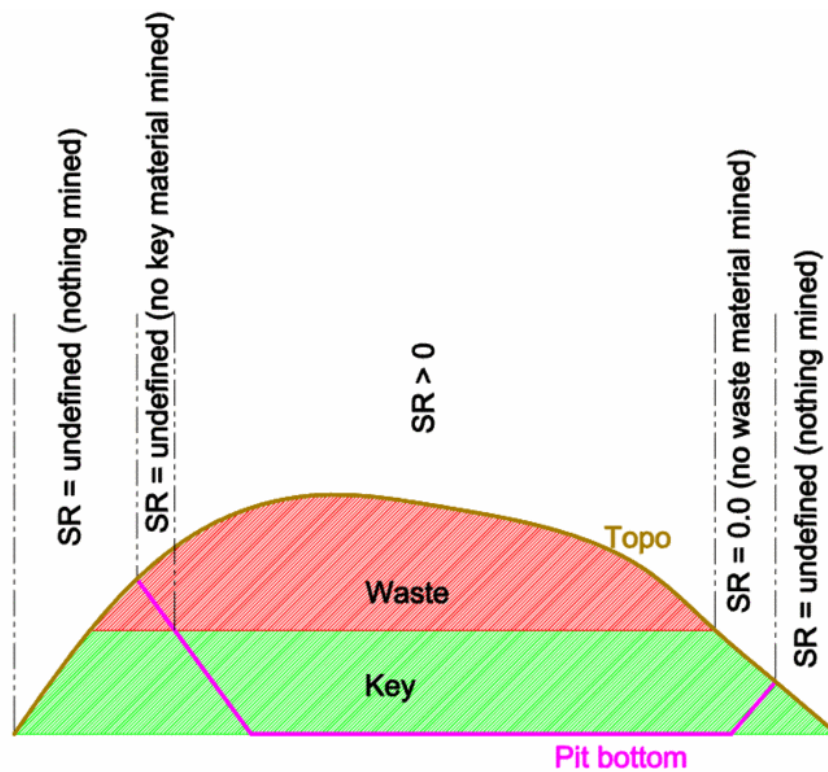
There are some situations in which a stripping ratio cannot be calculated, as shown in the below illustration.

For areas in which neither waste or key material are included, the stripping ratio will be undefined - if a stripping ratio grid is created, it will be null in this area.

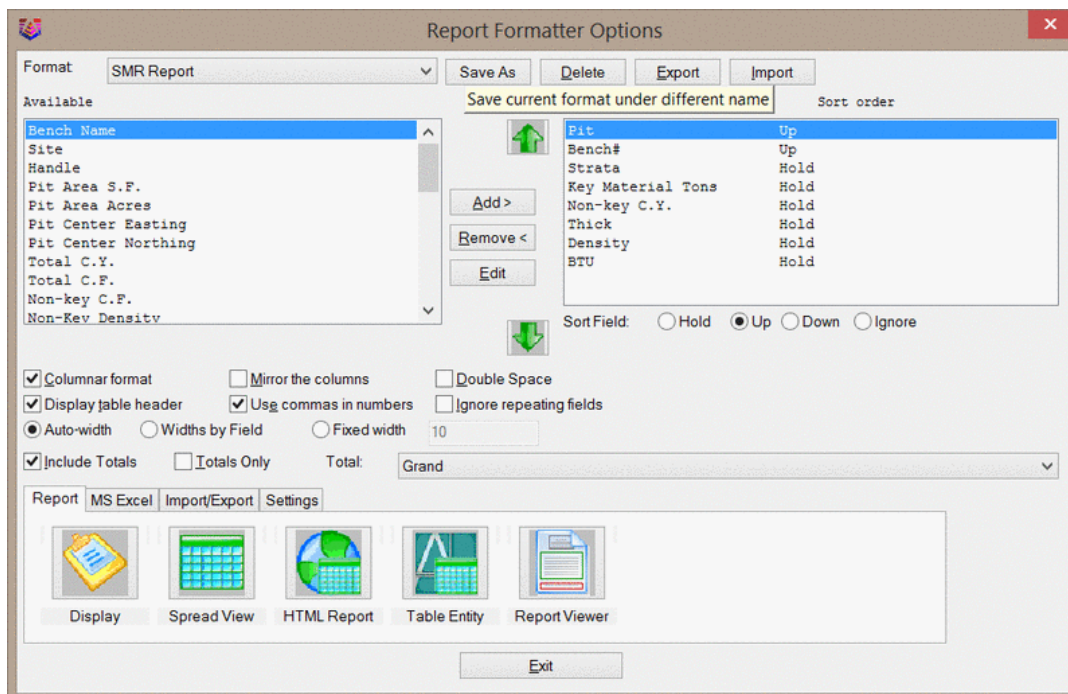
For areas in which only waste material is included, the stripping ratio will also be undefined- if a stripping ratio grid is created, it will be null in this area.

For areas in which no waste material is included, the stripping ratio will be zero.

For areas in which waste material is included, but only a very small portion of key material is included, the stripping ratio can approach an undefined value, which may appear as spikes in the stripping ratio grid file. This is normal for areas with a very small amount of key material. However, the stripping ratio output grid will not exceed a value of 1000. Spikes in the stripping ratio grid can be further filtered with Grid File Utilities.



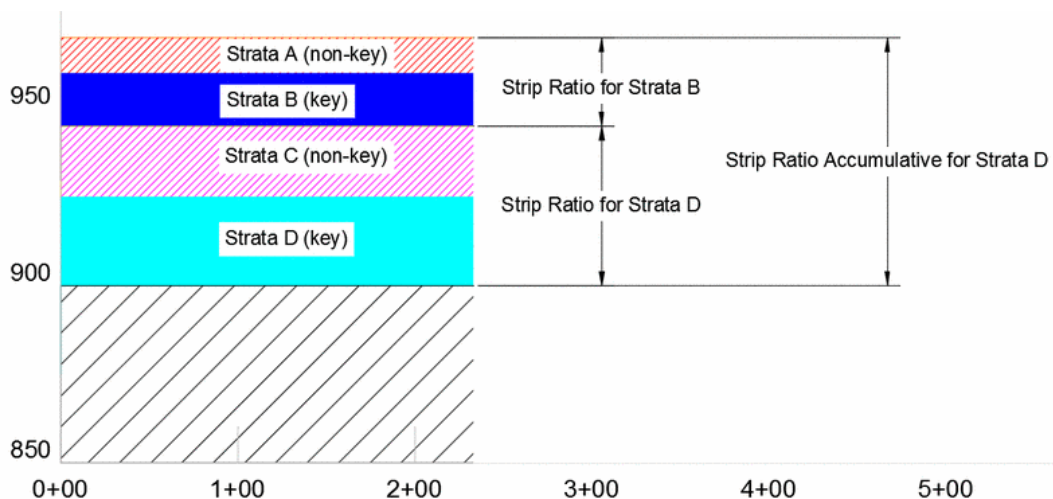
The Report Formatter displays all information calculated with this command, as shown below. For general information on using the Report Formatter, see the corresponding of the help manual.



Here it is important to note that when a parent seam has been configured to split into two child seams according to the strata definition file, the program will automatically report the splits. For example, if seam A splits into B and C, the program will report tonnage for seam A where it exists as a parent as well as seams B and C where they exist as child seams.

As an additional means of dividing reserves, Cut Sets defined by the GIS Module > GIS Tools Pulldown Menu > Polygon Processor command will be considered when the reserves are calculated. If a Cut Set has been defined for wetland areas with a name of "Wetlands", an additional reporting field will be available in the report format called PP_Wetland. This will show the amount of strata inside the wetland area. Any strata outside the wetland area will be reported separately.

The report formatter will include two forms of stripping ratio: "Strip Ratio" and "Strip Ratio Accumulative". The latter term should not be confused with the Accumulative Stripping Ratio for the stripping ratio grid/contour output. Consider four strata layers: A, B, C, and D where stratum A and C are non-key while stratum B and D are key. When these four stratum are calculated on a single bench, the report will include a "Strip Ratio" and "Strip Ratio Accumulative" for stratum B and D. Both attributes reported for strata B report the stripping ratio as if only strata A and B were mined. The "Strip Ratio" for strata D will report the stripping ratio as if only strata C and D were mined. The "Strip Ratio Accumulative" reported for strata D will report the stripping ratio as if all strata above it were also mined (stratum A and B will be included). The below image is an illustration of this concept.



Miscellaneous note on block models: When the program detects a Geologic Model containing one or more Block Models (.blk files), the thickness of the strata containing the block model will be compared to the thickness of the block model itself. Recall that each strata in a Geologic Model is defined by the difference between two elevation grids (or a thickness grid). Also, a block model will be defined with upper and lower limits (also two elevation grids). If the strata thickness is greater than the block model thickness, the program will extend the block model thickness an extra half-block height on the top and bottom of the model.

Carlson block models will always have top/bottom layers of blocks with half the height of other blocks (for more information on this, see the section of the help manual corresponding to the Make Block Model command). To account for the full block size, surface mine reserves applies the above logic. The idea is that whenever the true strata thickness is thicker than the block model thickness, the extra top/bottom block height needs to be accounted for.

Two types of thickness values can be reported for each strata: Thick Geologic and Thick Mined. Thick Geologic is the full Geologic Thickness, as if no constraints were applied to the model. Thick Mined, however, reports the thickness will all constraints applied (such as a pit shell that only cuts through a portion of the strata). When reporting Pit Volumes, it is recommended to report Thick Mined, as it will reflect the thickness extracted only within the pit shell. It is worth mentioning that when average thickness is calculated, an area-based weight is applied rather than a volume-based weighting.

Two types of areas are included in the report: Area and Pit Area. The Pit Area is the area of the inclusion line used for the report. The Area, however, is not a measurement, but rather a calculation based on the volume and the full Geologic Thickness. The Area is calculated as Volume divided by Thick Geologic.

Prompts

Surface Mine Reserves dialog

Select surface entities and at least 3 drillholes. (Unless using a Geologic Model File PRE.)

Select objects: *select the drillhole symbols and surface entities.* Surface entities can include points, lines, and polylines.

Select the Inclusion perimeter polylines and ENTER for none:

Select objects: *select the polylines or named pit polylines.* The area within these polylines will be included in the calculations. They must be closed polylines.

Select the Exclusion perimeter polylines and ENTER for none:

Select objects: *select the polylines.* The area within these polylines will be excluded from the calculations. They must be closed polylines.

Make Grid File Set grid resolution

Triangulating points ... 49

Pass > 6 NULL Z values left > 0

Processing cell 2500 ...

Finished strata Y2

The above four steps are repeated for each strata.

Report Formatter

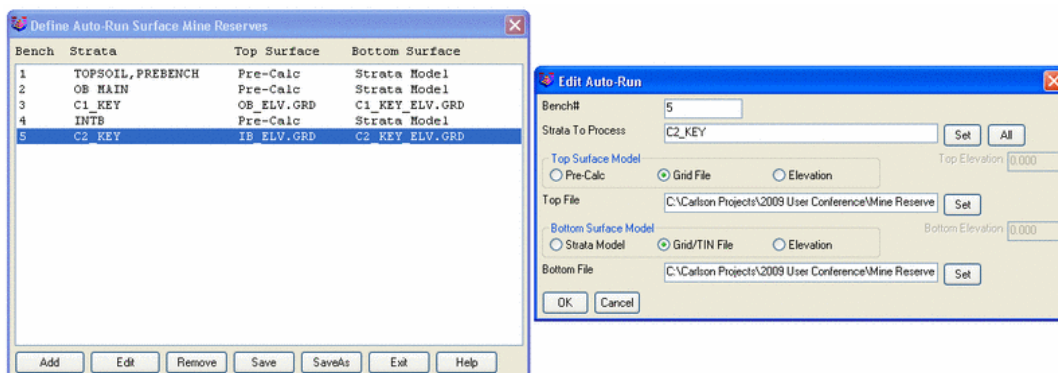
Pulldown Menu Location: Geology Module > StrataCalc and Surface Mining Module > Reserves/Timing

Keyboard Command: mtntop

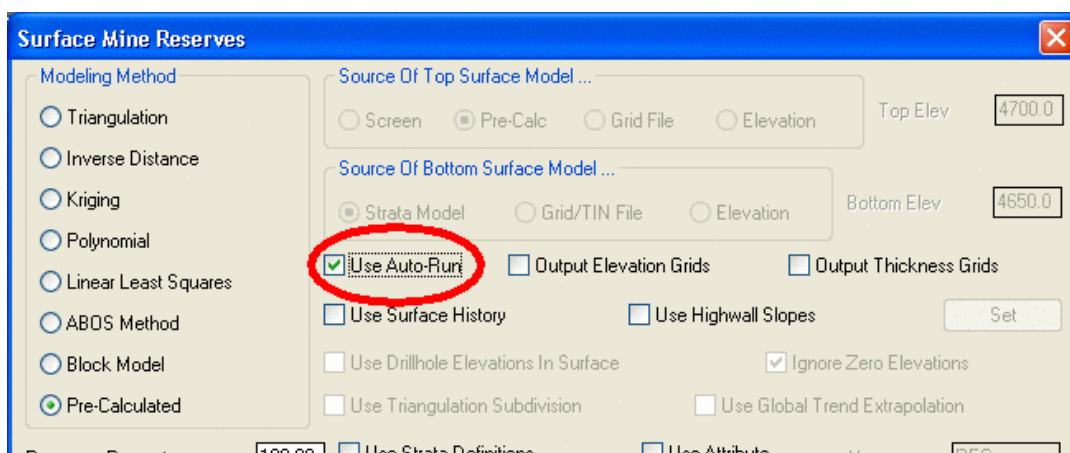
Define Surface Mine Auto Run

This command is the setup routine to create the autorun file for batch processing of multiple benches with one run of the Surface Mine Reserves command. Without this predefined autorun file, then the Surface Mine Reserves command needs to be run separately for each bench. With this Autorun file, all benches are predefined by the Geologic Model file, a top and bottom Grid or TIN, or an Elevation.

The steps necessary to create or edit the file are detailed below. Choosing Add or Edit will bring up the Edit Auto-Run window. This is where the Bench# is set. The Strata to Process can either be set to one, multiple, or all to define the benches. If coming out of the Geologic Model file, then the elevation grids defined in there will define the benches here. Alternately, the benches can be other grid or TIN surfaces, or flat elevations, and they will be intersected by the seams that appear in the Geologic Model file.



To utilize this Autorun file, turn on the option in the Surface Mine Reserves window for "Use Auto-Run" as circled below. Once this is on, the program will prompt to select the *.SMA file this command created.



Pull down Menu Location: StrataCalc

Keyboard Command: mntop_autorun

Prerequisite: Grids and PRE Calculated Grids file, named pits

Underground Mine Reserves

This command calculates quantities and qualities from drillholes or a predefined Geologic Model. This command is very similar to Surface Mine Reserves, but is simplified for underground application. There are many options for reserve calculation and they are detailed in order of appearance below.

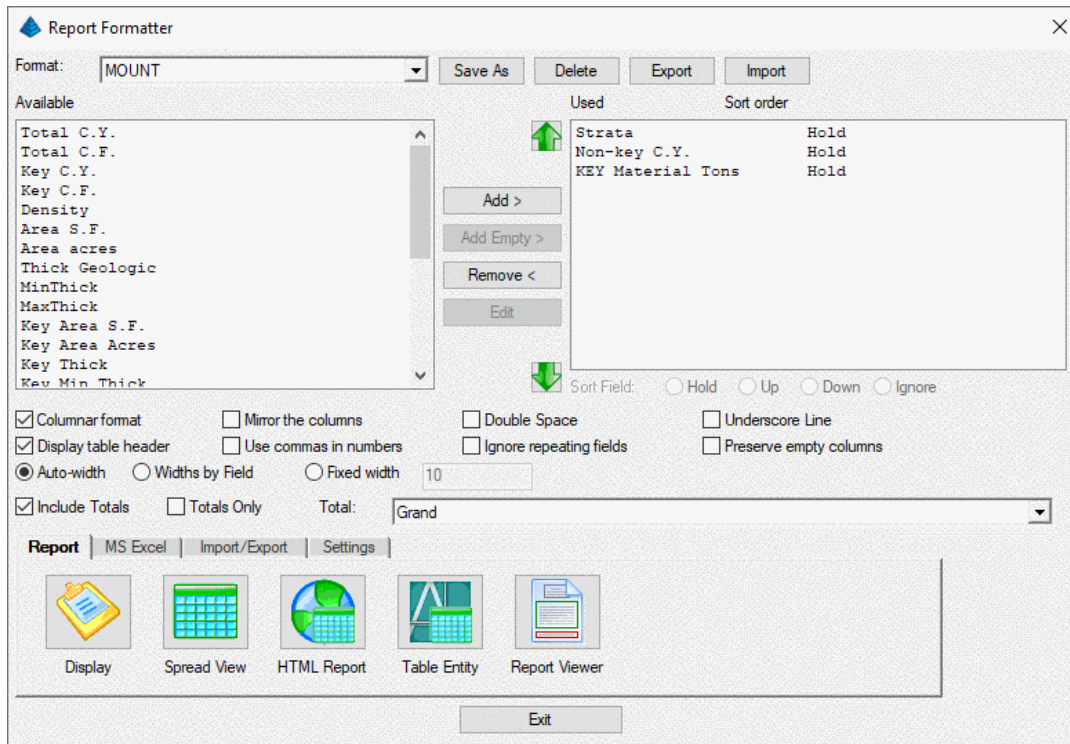
The screenshot shows the 'Underground Mine Reserves' dialog box. The 'Modeling Method' section has 'Triangulation' selected. The 'Recovery Percent' is set to 100.00. The 'Min Key Thickness to Use' and 'Min Separable Parting Thickness' are both set to 0.00. The 'Non-Key Thickness to Add to Key' section has 'Above Key' and 'Below Key' both set to 0.00. The 'Which strata to include ...' section has 'All' selected. The 'Use Attributes' checkbox is checked, and the 'Attribute Name' is set to 'REC'. The 'Specify Names' button is visible next to the 'Use Named Pit Areas' checkbox.

- **Modeling Method:** The reserves can be generated on-the-fly from the selected drillholes or read from stored grid files put into a Geologic Model file. On-the-fly calculation of reserves is generally NOT recommended, as this provides no method of visually inspecting the model for geologic accuracy. Instead, users are encouraged to first create a Geologic Model file and calculate reserves from the model. Nonetheless, on-the-fly calculations can be done for quick checks on estimated volume, but these results should not be reported unless the model has been verified visually with contours, cross sections, and visual inspection of the 3D grid files. Using the Triangulation, Inverse Distance, Kriging, Polynomial, Linear Least Squares, ABOS Method, or Nearest Neighbor option will calculate reserves on-the-fly. Using either the Geologic Model or Mining Model options will calculate reserves from an existing model.
- **Use Triangulation Subdivision:** This option only applies to the modeling methods of Triangulation and Polynomial. It subdivides the triangles to create smoother surfaces and ensure that contours do not cross.
- **Use Global Trend Extrapolation:** This also applies only to Triangulation and Polynomial. This option finds the average slope and direction of the existing data and applies this slope to extrapolating where there is not surface data.
- **Make Thickness Grids:** Turning this option on will create thickness grids of the selected or all strata. It will prompt to create an overburden thickness grid, a key thickness grid and a key tons grid (tons/sq ft. or m).
- **Output Grids for Each Strata:** Turning this option on will create grid files for the bottom of each strata found in the drillholes. However, it is recommended that strata grids are instead made with the Strata Grids Autorun command for easier review and repeat-ability of the grids.

- **Report Areas On One Row:** This option puts all the strata quantities and qualities for each inclusion polyline on one row. This format is best suited for only a few strata and for printing landscape on a page. If it is not selected, then each strata will appear on a separate row.
- **Use Property Boundaries:** Property boundaries can be used to break up the reserve by owner and property. The commands for laying out property boundaries are in the Boundary menu. Essentially, property boundaries are closed polylines with owner and property ID names. The property polylines do not need to be clipped with the calculation inclusion perimeter or pit polylines. The program will internally clip the properties with the calculation areas and report the amounts by property within each pit area. If a pit or inclusion polyline is not covered by a property, the property name used for these quantities is "unknown". When this option is enabled, the program will automatically detect the property boundaries.
- **Calculate Strata Qualities:** This toggle will report the average qualities for strata attributes such as BTU and sulfur. Otherwise the program skips calculating qualities to save time. Besides reporting the qualities for each strata individually, the program can also report the total averaged qualities. By default, the qualities are weight averaged by tons which are calculated by using the strata volume and density.
- **Recovery Percent:** This window defines the percentage of Key strata volume that was recovered in mining and to include in the volume and strip ratio reporting. The non-recovered key strata is added to the NonKey volume. The recovery percent in the Reserves dialog applies to all the Key strata. These values should be entered as percentages rather than decimals (e.g. "100" for total recovery as opposed to "1").
- **Use Strata Definitions (for Recovery):** When enabled, this option will check the current Strata Definition file for recovery values rather than using a constant recovery value for all key strata.
- **Use Attribute (for Recovery):** Another When enabled, this option will check the Geologic Model (or drillholes) for a recovery attribute grid. This allows the program to vary the recovery by area and strata rather than simply applying a constant value to the entire area. When enabled, the **Attribute Name** will need to perfectly match the recovery attribute found in the Geologic Model or the Drillholes (if the attribute in the model/drillholes is "REC", enter the exact same name in this text box).
- **Min Key Thickness To Use:** This option adds key strata to the nonkey strata above, in areas that have thickness less than the specified minimum thickness. Also areas with thickness less than the minimum are not counted in the reported strata area.
- **Min Separable Parting Thickness:** This setting will add the non-key parting quantities with the key quantities when the non-key parting thickness is less than the specified amount. For example, if the Min Separable Parting is set to 0.5, then a non-key strata between two key strata would be combined with the key in areas where the non-key thickness is less than 0.5 feet. In areas where the thickness is greater than 0.5, the non-key quantities are not adjusted. Combining the non-key quantities to the key will add to the total key tons and affect the strip ratio. If the non-key strata has qualities (i.e. ASH, Sulfur), then these non-key qualities will be composited by nonkey tons with the key tons for the portion of non-key that is less than the minimum parting. This will dilute the key qualities.
- **Non-Key Thickness To Add To Key: Above Key/Below Key:** These fields allow you to specify the amount of non-key thickness above and/or below the key strata that will be combined with the key (roof and floor dilution) This amount will be taken from the non-key quantities and added to the key. Similar to the Min Separable Parting Thickness, the non-key quantities will increase the total key tons which affects the strip ratio and key volume mined. Also any non-key strata qualities will be combined by thickness, weighted by NonKey density, and added to the key which dilutes the key qualities. For example, if you estimate that 0.25 feet on average of the overburden is taken with the coal, then you could set the Non-Key to Add to Key for Above to 0.25.
- **Which Strata To Include...:** This determines what will be calculated for the reserve. If All is chosen, then all strata in the drillholes or Geologic Model file will be calculated and reported. If Selected is chosen, then the next window to appear is where one or multiple strata can be selected. This can be used for multiple reserve runs, selecting different strata to represent each bench.

- **Use Named Pit Areas:** The area for calculating quantities defaults to the limits of the selected surface entities and drillholes if no inclusion perimeters are selected. To control the calculation area, multiple closed polylines for areas to include and/or exclude can be selected. An unnamed pit polyline will limit the area of calculations. Also, areas can be labeled with site and pit names (i.e. Site 1, Pits 101, 102, ...). Surface Mine Reserves will then calculate the strip ratios and volumes for each site and pit area. To use site and pit names, there are several commands for creating named pit polylines in the Boundary menu of Underground Mining. If this is selected, then it will look just for named pit polylines, ignoring anything else. The Specify Names button will name the polylines from inside this command if they are not already named using commands found under Boundary.

The Report Formatter Options dialog is the final step of the Underground Reserves routine. It is documented elsewhere in the manual for more details on its operation.



Prompts

Underground Mine Reserves dialog

Select surface entities and at least 3 drillholes. (Unless using a Geologic Model.)

Select objects: *select the drillhole symbols and surface entities* Surface entities can include points, lines, and polylines.

Select the Inclusion perimeter polylines and ENTER for none:

Select objects: *select the polylines or named pit polylines* The area within these polylines will be included in the calculations. They must be closed polylines.

Select the Exclusion perimeter polylines and ENTER for none:

Select objects: *select the polylines* The area within these polylines will be excluded from the calculations. They must be closed polylines.

Make Grid File Set grid resolution

Triangulating points ... 49

Pass > 6 NULL Z values left > 0

Processing cell 2500 ...

Finished strata Y2

The above four steps are repeated for each strata.

Report Formatter

Pulldown Menu Location: StrataCalc in Geology Module

Keyboard Command: reserves

Blending Weighted Average

This command calculates the weighted average for qualities resulting from mixing different materials together. The names of the qualities are user-defined. Up to four weighted qualities can be calculated at once. The quality names are specified in the first row of boxes. For each type of ore to include in the mix, fill out the row as shown below including the name, tons and value for each quality. The name is used in the report. The weighted average is based on the tons of each value. The report is presented in the standard report viewer which can print, file or draw the report.

Name	Tons	Quality 1	Quality 2	Quality 3	Quality 4
North Pit	500000	10000	5	4.1	29.5
South Pit	300000	8500	2.4	8	32.5
East Pit	600000	9000	4	7	31
West Pit	100000	11000	2	3.8	29

Name	Total Tons	BTU	Sulfur	Ash	Moisture
North Pi	500000.00	10000.00	5.00	4.10	29.50
South Pi	300000.00	8500.00	2.40	8.00	32.50
East Pit	600000.00	9000.00	4.00	7.00	31.00
West Pit	100000.00	11000.00	2.00	3.80	29.00
Weighted Average:		9366.67	3.88	6.02	30.67

Pulldown Menu Location: Misc > Blending Weighted Average

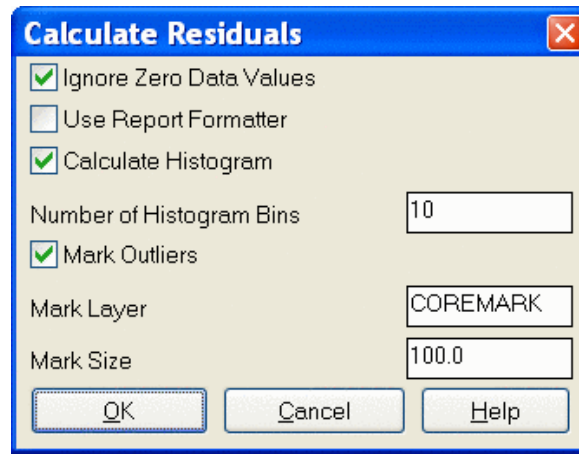
Keyboard Command: wgtqual

Calculate Residuals

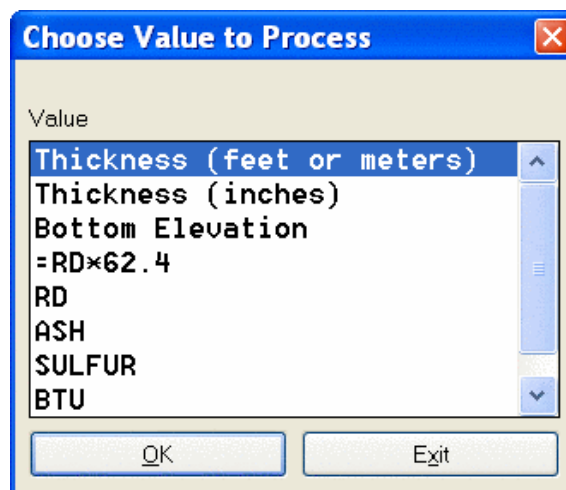
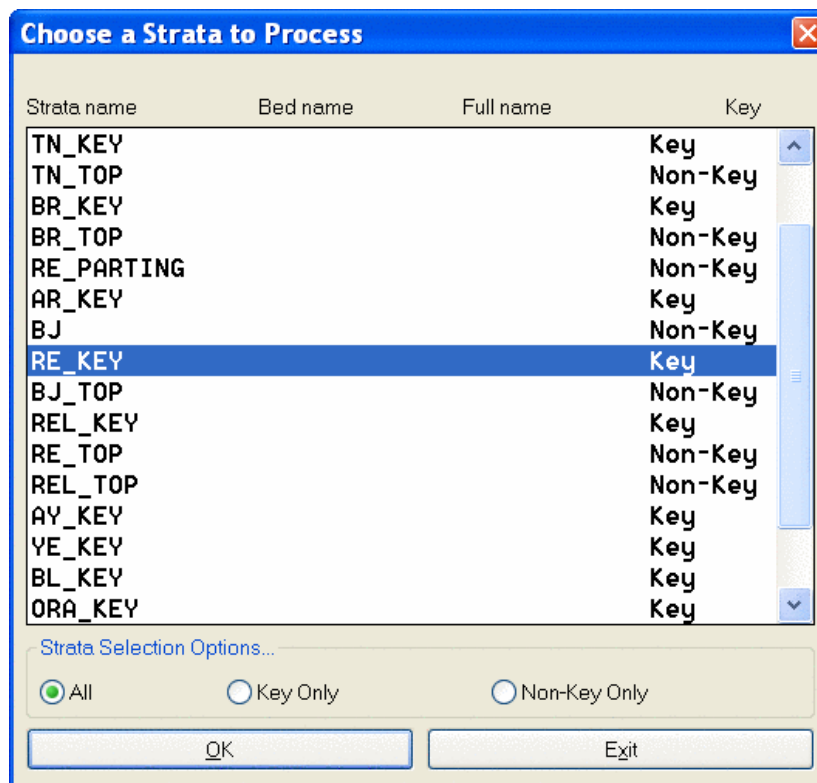
This command measures the accuracy of the strata grid model based on the residual at each drillhole data point. The residual is calculated by removing the data point and then comparing the grid model calculation at the point with the actual value. Residuals for any strata data value such as thickness, elevation or BTU can be calculated using any modeling method including triangulation, polynomial, Kriging, least squares and inverse distance.

Consider calculating residuals for thickness with five drillholes. For each drillhole, the drillhole is temporarily removed from the model and the thickness calculated at the drillhole position. For example, one drillhole could have a thickness of 5.0. When this drillhole is removed and the four remaining drillhole using inverse distance calculate the thickness at this point, the modeled value could be 5.25. The difference of 0.25 between the actual value 5.0 and the modeled value 5.25, is the residual.

The first dialog to appear contains some settings on the residual analysis.



- **Ignore Zero Data Values:** This will ignore any values of 0 that are found in the drillhole. They could be thickness, elevation or any quality attributes. Sometimes they should not be ignored, depending on what is being modeled.
- **Use Report Formatter:** Turn this option on to see the report with the Report Formatter. If it is off, then the standard report viewer is used.
- **Calculate Histogram:** The Histogram is a statistical graph of the distribution of a dataset. It shows the number of data samples within a series of value ranges.
- **Number of Histogram Bins:** The user can set the number of bins for the graph. For example, if a dataset of sulfur has values from 1.0 to 4.0 and the user asks for 3 bins, then the histogram will show the number of samples between 1.0 to 2.0, between 2.0 to 3.0 and between 3.0 to 4.0.
- **Mark Outliers:** The Mark Outliers option highlights data points that are more than 2 standard deviations from the data set average. This can be useful for flagging possible problem data points.
- **Mark Layer:** This is the layer of the circle mark drawn around the outlier drillholes.
- **Mark Size:** This is the size of the mark, or circle drawn around the outlier drillholes.



For triangulation and polynomial, the residual is not calculated for some points because removing these points shrinks the triangulation area and the point may fall outside the area. Also you need more than three points to calculate residuals with triangulation and polynomial methods.

Prompts

Select drillholes, channel samples and strata polylines.

Select objects: Specify opposite corner: 269 found

Select objects:

Reading drillhole 269

Choose modeling method [**<Triangulation>**/Inverse dist/Kriging/Polynomial/LeastSq/ABOS]? *l*

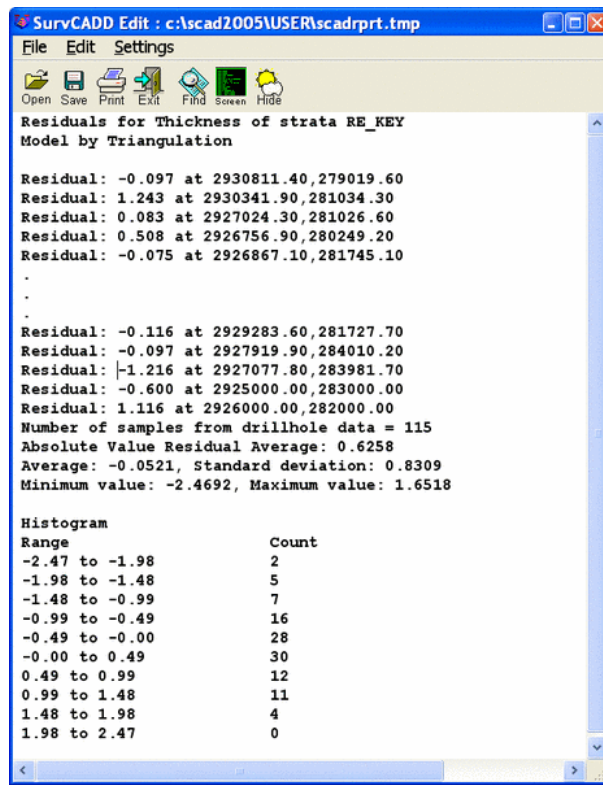
Use inverse distance to which power [**First**/**<Second>**/Third/Other]?

Use elliptical inverse distance [Yes/**<No>**]?

Calculating grid by inverse distances 4...

Try another modeling method (Yes/**<No>**)? *N*

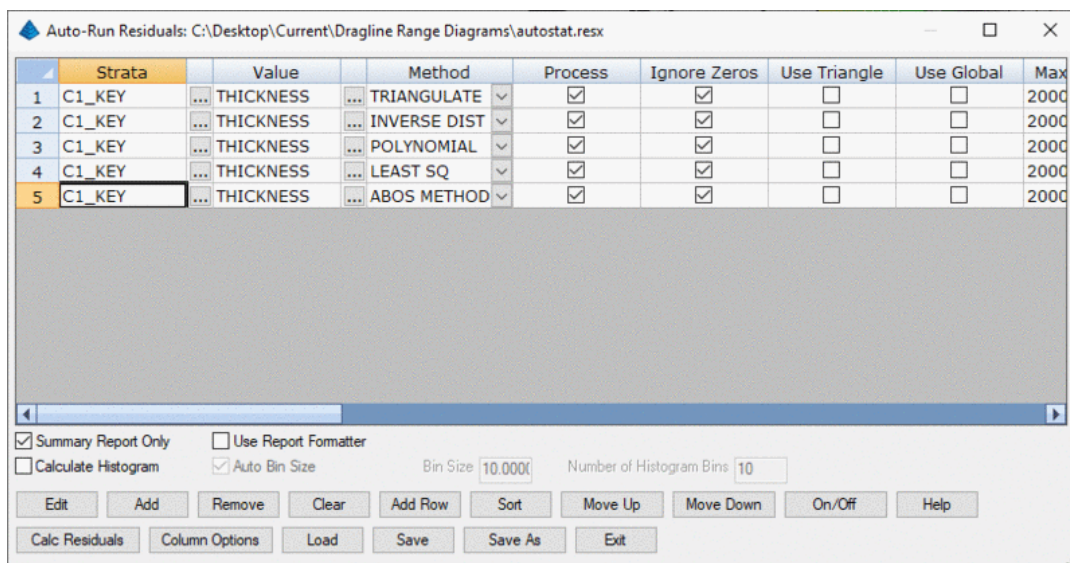
Calculate **BOTTOM ELEVATION** residuals for strata **9** (**<Yes>**/No)? *N*



Pulldown Menu Location: StrataCalc
Keyboard Command: residuals

AutoRun Residuals

This command automates running the Calculate Residuals command. For this command, you set up a list of strata names, values and modeling methods. Then all these methods can be run at the same time and the results displayed in a formatted report.



When the command is first executed, you will be prompted to select the drillholes, channel samples, and strata polylines that will be used for modeling. The above dialog will then appear.

Summary Report Only: Creates a simplified report rather than including the details on the residual that is calculated for each data point.

Use Report Formatter: Sends the report to the Report Formatter rather than using the predefined text format.

Calculate Histogram: Categorizes the residual values into histogram bins. The bins may be automatically set with the **Auto Bin Size** and **Number of Histogram Bins** options, or you may set an absolute **Bin Size**

Edit: Edits an existing entry in the list.

Add: Adds a new entry to the list.

Remove: Removes an entry from the list.

Clear: Removes all entries from the list.

Add Row: Adds a blank row to the list

Sort: Sort the list in alphabetical order by Strata Name.

Move Up/Down: Moves the selected entry up/down in the list

On/Off: Changes the Processing status of the entry. When turned off, the entry will not be calculated.

Calc Residuals: Calculates the residuals for each entry in the list, creating the final report.

Column Options: Allows you to toggle the visibility of each column in the main dialog.

When a new entry is added to the list, the below dialog will appear.

Auto-Run Residuals

Processing On Ignore Zero Values Output Residual Grid File

Choose Strata to Process Strata Name: Value to Process:

Choose Grid File

Modeling Method

Triangulation Use Triangulation Subdivision Use Global Trend Extrapolation

Inverse Distance Max triangle length 2000.00 Set Kriging Parameters

Kriging Inverse Distance Power 2.000 Search Radius 10000.00

Polynomial Min Quadrant 0 Max Quadrant 20

LeastSq Elliptical Inverse Distance Max Samples 20

ABOS Anisotropic Azimuth 0.0 Factor 1.00

Nearest Neighbor (Voronoi) Diff Smoothing Factor 10.00 Target Accuracy (%) 5.000

Linear Tensioning Medium

OK Cancel Help

The settings found in this dialog are very similar to those found in the Strata Grids Autorun command, and are not discussed in depth here. The **Output Residual Grid File** option will create a grid file from the residual values. This provides a visual means of inspecting the residual values rather than only viewing a text report. Note that when the residual grid is made, the Triangulation method will be used to connect the residual values. After picking the settings for the modeling method, you may click **OK** to add the entry to the list.

An example of the final report is shown below. Often, the most meaningful value is the Absolute Value Residual Average. A lower value indicates that the modeling method was able to very accurately predict the value in an unknown area, whereas a higher value indicates a less accurate modeling method. This is only one means of determining the applicability of the modeling method however. Some modeling methods may not be able to use all of the available data for calculating the residual value, which should be considered when picking a modeling method. The contours of the modeled surfaces should also be considered to ensure correct results. Every data set is different, and there is no universal "best" modeling method or settings that can be blindly applied to any data set.

File Edit Settings

Auto-Run Residuals Fri Oct 9 17:57:48 2020

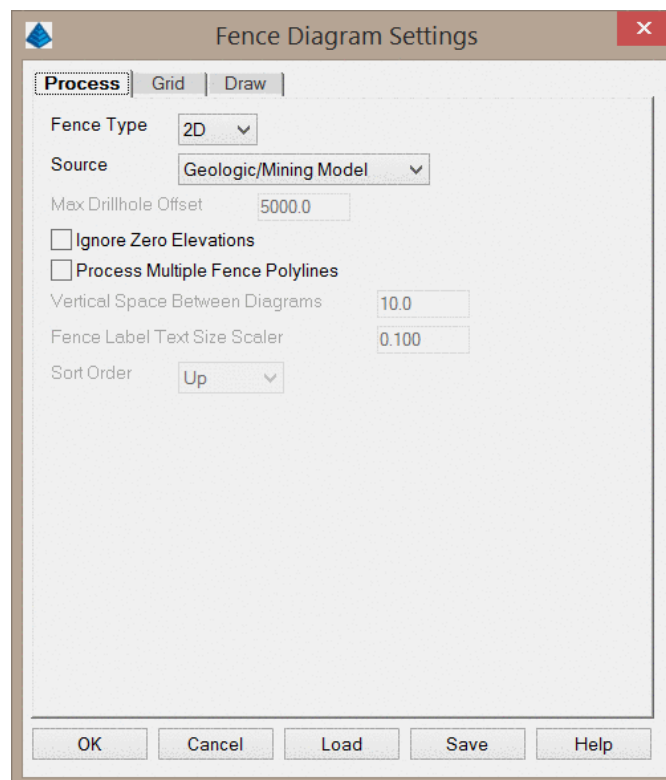
Strata Value	Model	Samples	Absolute Value	Residual	Average	Std Deviation	Avg	Min	Max
C1_KEY Thickness	Triangulation	2			3.40	1.47	3.40	1.93	4.87
C1_KEY Thickness	Inv Dist 2.00 Pwr	8			3.25	3.40	-0.14	-4.20	4.34
C1_KEY Thickness	Polynomial	2			3.01	1.36	3.01	1.65	4.37
C1_KEY Thickness	LeastSq	8			3.68	3.86	-1.18	-6.63	4.51
C1_KEY Thickness	ABOS	8			3.24	3.50	0.19	-4.86	5.43
Total									
C1_KEY Thickness varies									

Pulldown Menu Location: StrataCalc

Keyboard Command: autostat

Fence Diagram

Fence Diagram produces a profile of the surface and strata along the selected baseline polyline—a geological cross-section. Fence diagrams are a good way to verify that the geological model is correct—especially when the drillholes are plotted on them with Draw Geologic Column command. The first step is to draw a baseline polyline in plan view from which the fence diagram is created. This polyline can have more than two points. There are three tabs on this dialog, detailed below.



Fence Type: This option determines if the fence will be drawn in 2D at a user-specified location, or in 3D in real world coordinates.

Source: This option determines the source of the fence.

The *Intersection* method builds a model on-the-fly by triangulating the drillholes and then finding the intersection of the triangulation mesh and the fence polyline. In order to obtain data for the strata with the intersection method, the polyline must have at least one drillhole on each side. It is the only way to see a fault in a fence diagram when using the drillholes.

The *Grids from Drillholes* option builds a model on-the-fly. When this option is used, you will be prompted to select drillholes to build the model, as well as your modeling parameters. This option is only recommended for quick inspection of how the model will appear, but it is not generally recommended as some geologic modeling options cannot be applied with this method, such as accounting for faults.

The *Geologic/Mining Model* option is the most common method, as it pulls the geologic information from an existing Geologic or Mining model file (.pre or .mmd). This is the recommended method for drawing most fence diagrams.

Max Drillhole Offset: This option is only available when the Source is set to the *Grids from Drillholes* option. This value sets a search radius for drillholes. Any drillholes further than this distance from the polyline will not be used for building the geologic model.

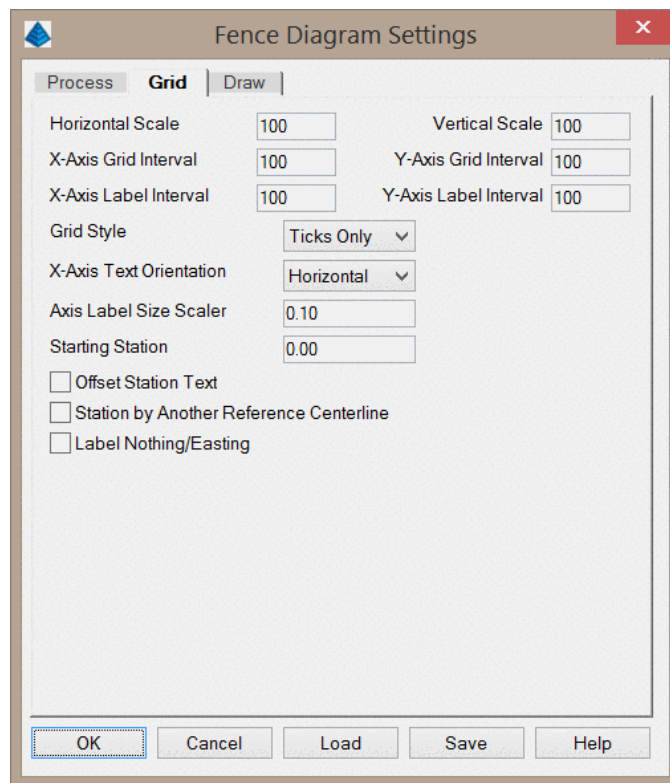
Ignore Zero Elevations: This option is only available when the Source is set to the *Grids from Drillholes* option. When creating a surface topo for the model, this option will disregard any entities at zero elevation.

Process Multiple Fence Polylines: This option allows you to draw multiple fence diagrams at the same time. Instead of picking a single fence alignment polyline, you can select multiple polylines that have been tagged using the Tag Fence Polylines command. Each of the fence diagrams will be labeled accordingly.

Vertical Spacer Between Diagrams: This value controls the spacing between fence diagrams when more than one are drawn at a time. This value is multiplied by the Horizontal Scale and the Fence Label Text Size Scaler to determine the true drawing distance between the diagrams.

Fence Label Text Size Scaler: This value controls the size of the fence labels when drawing more than one at a time (that is, the labels identifying Fence 1, Fence 2, etc.). This value multiplied by the Horizontal Scale sets the actual text size for the labels.

Sort Order: This option determines if the fences will be stacked upwards or downwards when more than one are drawn at a time.



Horizontal Scale: This value sets the overall drawing scale that many other options in the dialog reference for scaling. This allows you to quickly change the size the geologic columns with a single value.

Vertical Scale: This value, relative to the Horizontal Scale, determines the vertical exaggeration. For example, a Horizontal Scale of 50 and a Vertical Scale of 10 will produce a 5X vertical exaggeration.

X/Y-Axis Grid Interval: This value sets the grid line spacing for the X and Y axes. For example, a Y-Axis Grid Interval of 25 will draw a tick/grid line every 25 ft/m of elevation change.

X/Y-Axis Label Interval: This value sets the text intervals for the X and Y axes. For example, a Y-Axis Label Interval of 100 will draw an elevation/depth label every 100 ft/m of elevation change.

Grid Style: This option controls how the grid will be drawn.

The *Grid Lines* option will draw a true grid with lines spanning the appropriate axes.

The *Ticks Only* option will draw the grid axes, but instead of full lines spanning the axes, small tick marks will be placed along each axis.

X-Axis Text Orientation: This option controls the rotation of the stationing labels on the X-axis.

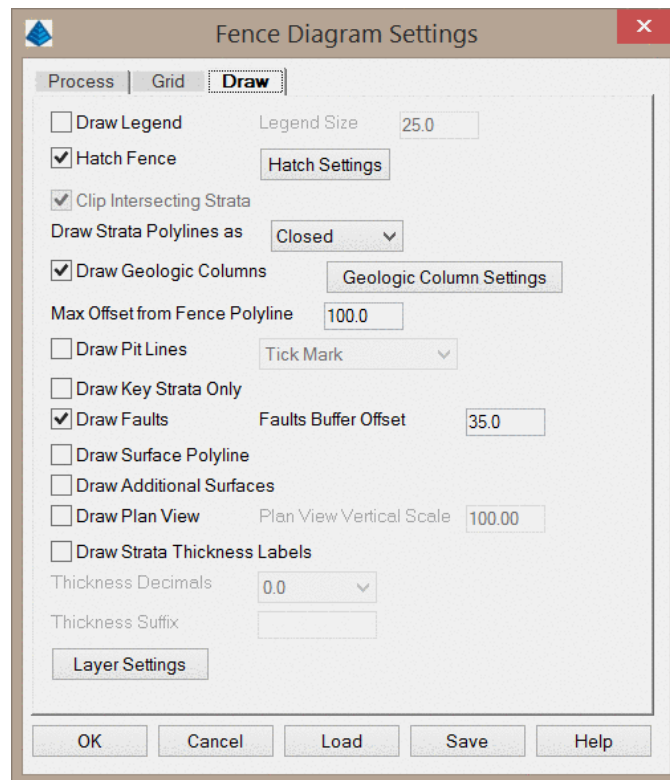
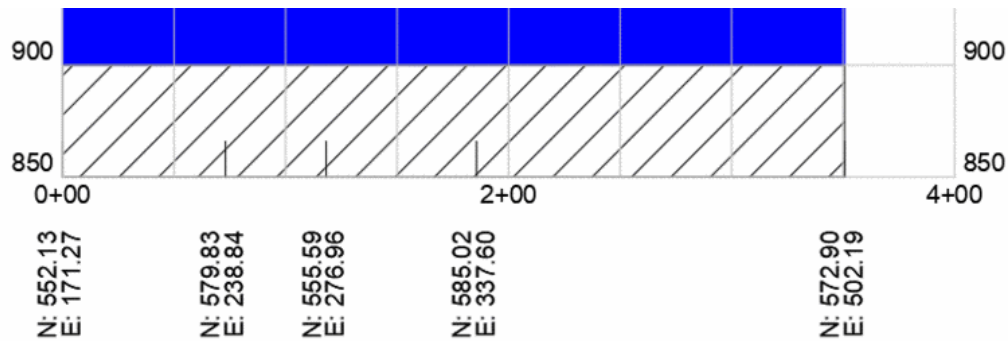
Axis Label Size Scaler: This value, relative to the Horizontal Scale, determines the text height of the elevation/depth/stationing labels. For example, an Axis Text Size Scaler of 0.15 and a Horizontal Scale of 100 will produce text labels with a height of 15 drawing units.

Starting Station: This is the station that is labeled at the beginning of the fence diagram horizontal X-axis grid. By default it is set to 0.

Offset Station Text: Offsets the station labels below the horizontal axis of the grid.

Station By Another Reference Centerline: This option will station the horizontal X-axis by a predefined centerline CL file. The centerline file will be prompted for after clicking OK.

Label Northing/Easting: This option labels the Northing and Easting coordinates along the bottom of the fence diagram at every vertex on the fence line that is drawn in plan view. An example is shown below.



Group Entities: This option creates a CAD Group from all the entities created by Fence Diagram. Then you can select the single group to get all the entities which makes commands like Move easier to move the fence diagram in the drawing.

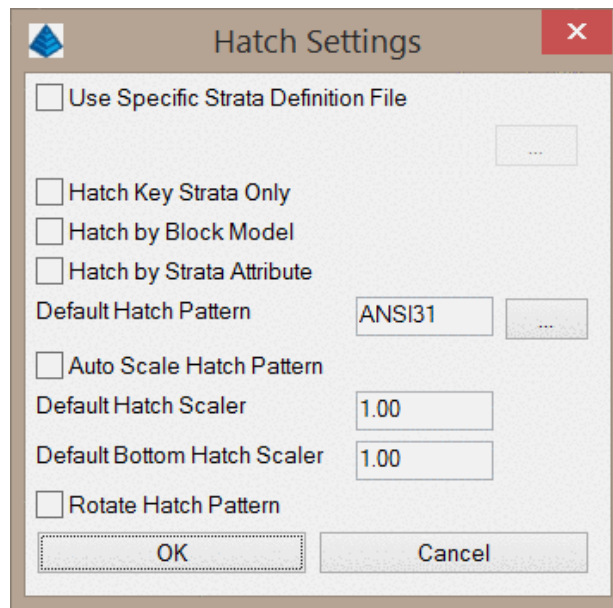
Draw Legend: This option draws a legend of the strata hatch patterns and colors. You will be prompted to pick the location for the legend after clicking OK.

Legend Size: This values sets the size of the legend.

Hatch Fence: This option will hatch in the strata layers in the fence. If this option is disabled, only polylines will be drawn. The strata hatch patterns are defined in the Define Strata command. If a strata exists that is not defined, it

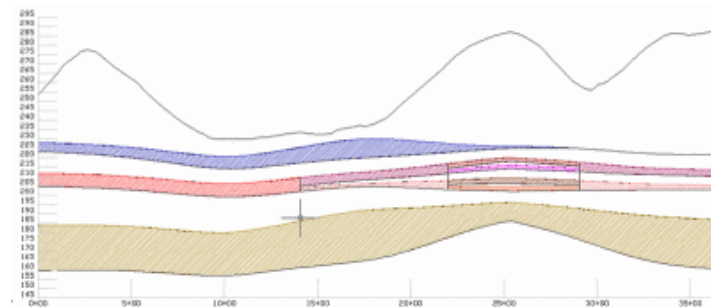
will use a default hatch pattern.

Hatch Settings: This button will display the below dialog.

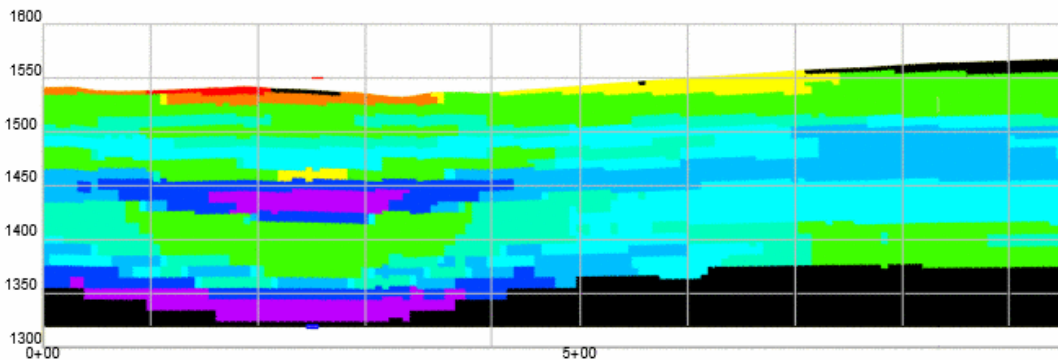


Use Specific Strata Definition File: This option will draw the strata hatch patterns using a strata definition file other than the current strata definition file. When enabled, you can select the strata definition file by clicking the ellipsis button.

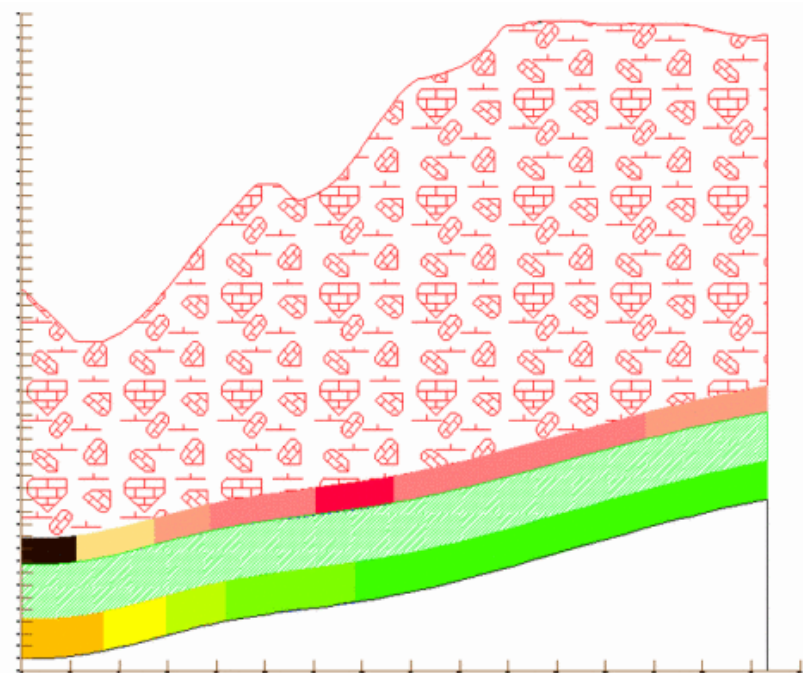
Hatch Key Strata Only: This option will just hatch Key strata that are drawn. The Non-Key strata will be left as polylines with no hatch inside. Shown here is an example that has just the Key strata hatched.



Hatch by Block Model: This option will hatch the seams using a Grade Parameter File. This option requires a block model to be stored in the Geologic Model to define the quality of the strata. An example is shown below.



Hatch by Strata Attribute: This option is similar to Hatch by Block Model, except there is no Grade Parameter File required. It will look at the quality attributes found in the Geologic Model, or in the drillholes, and bring up a list. You will be able to select an attribute to hatch by, and then a color palette will appear, allowing you to specify the colors and ranges for the quality zones. This will just change the hatch horizontally across the strata, not vertically as the Block Model does. Shown below is an example of coal seams colored by BTU.



Default Hatch Pattern: This option sets the hatch pattern for strata layers that are not defined in the Strata Definition file.

Auto Scale Hatch Pattern: This option will auto fit the hatching inside the strata perimeters.

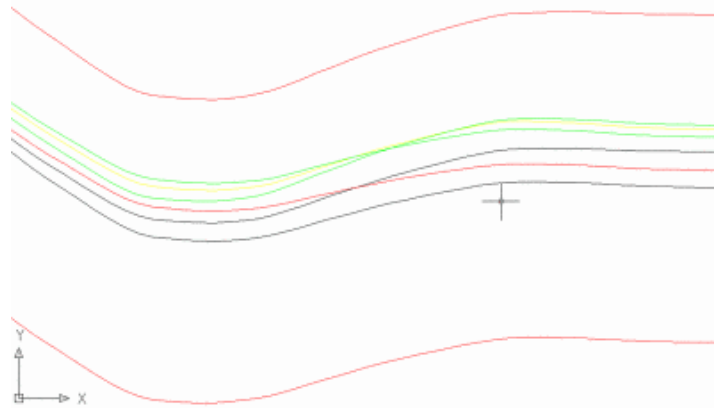
Default Hatch Scaler: This value sets the hatch pattern scale for default hatch patterns. The scale of the hatch pattern will equal this value multiplied by the Horizontal Scale.

Bottom Hatch Scaler: This value sets the hatch pattern scale for the portion of the fence diagram below the last strata layer. The scale of the hatch pattern will equal this value multiplied by the Horizontal Scale.

Rotate Hatch Pattern: This option rotates hatch patterns to best fit the angle of the strata. A hatch pattern can have only one rotation, so rotating will not be effective for a strata with multiple rotations. Rather than use this automated

method for setting the hatch rotation, the rotation for each hatch can also be set in the Strata Definition file.

Clip Intersecting Strata: This option will automatically clip strata layers that cross one another, with the upper grids clipping out the lower grids. This will display the strata layers as they will be interpreted for volume calculations. If this option is disabled, the strata layers will not clip one another. The below image shows two strata layers that clip one another. Note that this option cannot be used when hatch patterns are used.



Draw Strata Polylines as: This option determines if the strata layers are drawn as closed polylines, or as single, open polylines. It is recommended to use the *Single* option when drawing cross sections for use with the Range Diagram commands in the Surface Mining module.

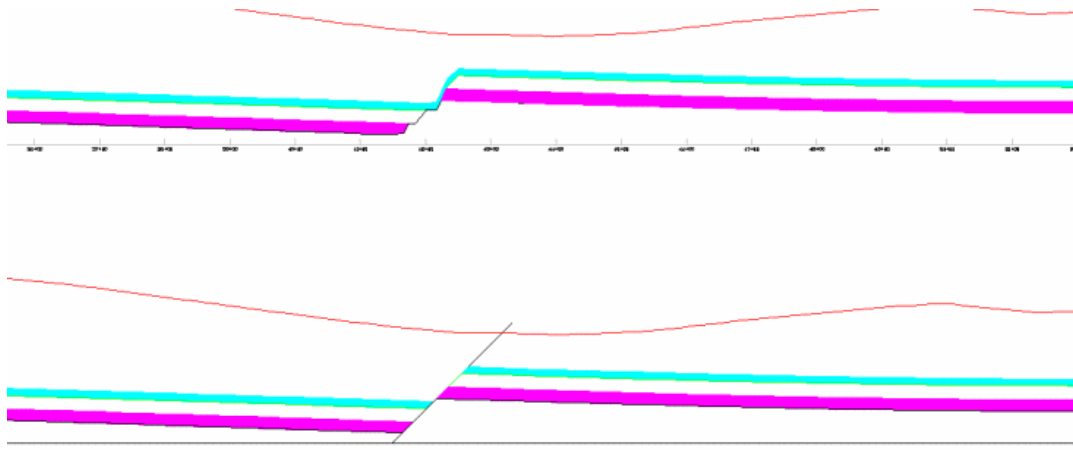
Draw Geologic Columns: This option will draw the drillholes on the Fence Diagram. When this option is enabled, you can click the Geologic Column Settings button to show settings associated with drawing the drillholes. These are the same settings included in the Draw Geologic Column command.

Max Offset from Fence Polyline: This option sets the search distance from the fence alignment polyline. Any drillholes beyond this distance will not be drawn on the fence diagram.

Draw Pit Lines: When this option is enabled, the program will prompt you to select pit polylines. These pit polylines will be labeled with a vertical line and labeled along the bottom of the fence diagram grid at the position where the fence plan view polyline crosses the pit polylines. This option is useful for dragline design commands such as Cut and Place. The dropdown list to the right of this option controls if the vertical line is set to a fixed height (Tick Mark option), or if the vertical line extends from the grid bottom to the bottom strata (To Bottom Strata option).

Draw Key Strata Only: This option will only draw the Key strata on the fence diagram. The Non-Key intervals found in between the Key will be drawn, but not be labeled individually.

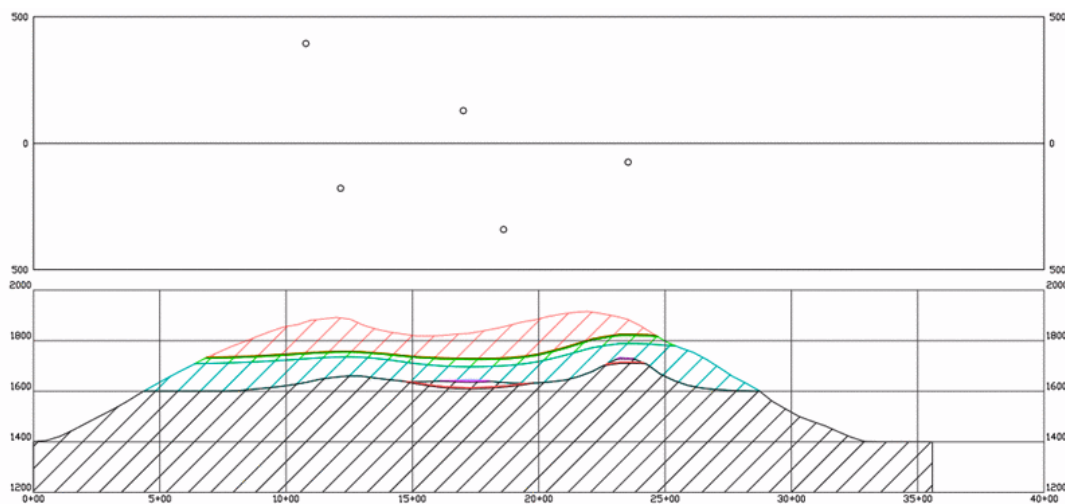
Draw Faults: This option will draw the fault line in the location and at the angle that it crosses the cross section. Although grid files may be modeled with a fault, this option ensures that the representation of the fault is sharp. The below image shows a fence diagram without this option (above) and with this option (below). The program uses the **Faults Buffer Offset** value to clip the strata. The fault line will be offset by this amount and the strata within the offset will be redrawn to remove the noise of the grid behavior near the fault. It is recommended to set the Faults Buffer Offset value to be at least 1.5 times the strata grid cell size.



Draw Surface Polyline: This option will draw a new polyline along the surface of the fence diagram. This is particularly helpful for show the true surface topography when there are outcrops, as this new polyline is guaranteed to run along the full length of the cross section. This is necessary for the Range Diagram commands found in the surface Mining module.

Draw Additional Surfaces: This setting will allow you to show additional surface files on the cross section that are not part of the geologic model. A prime example is the water table, which may traverse multiple strata types. When this option is used, you will be prompted to select the surfaces to draw after clicking OK.

Draw Plan View: This option will graph the drillhole locations above the fence diagram. For this graph, each drillhole is projected onto the baseline polyline to find the station and offset of the drillhole. This option is available when the Fence Diagram Source is set to Grids From Drillholes.

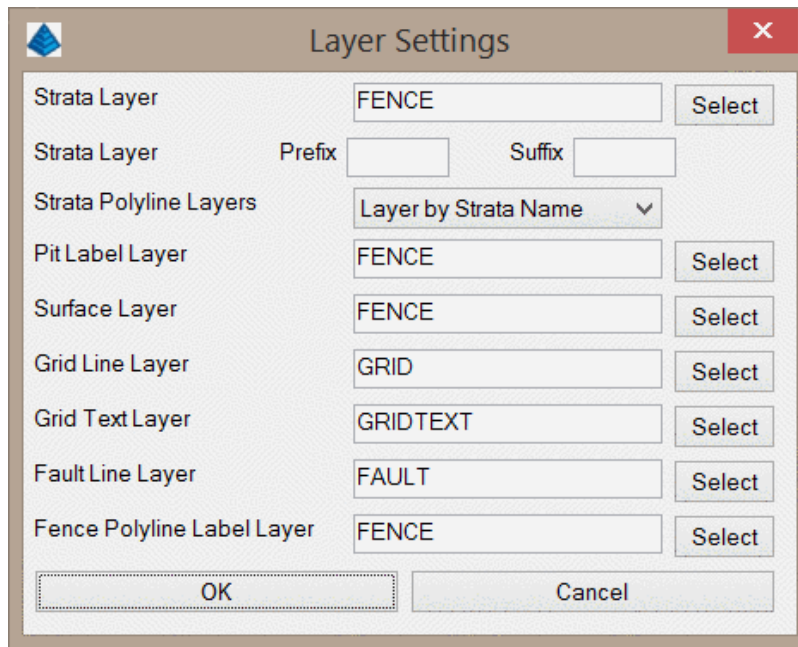


Plan View Vertical Scale: This value sets the scale factor for the Plan View drawn above the fence diagram.

Label Strata Thickness: This option adds a label for the strata thickness along the fence. When this option is used you may specify the number of decimals to display with the **Thickness Decimals** dropdown list, and you may also add a suffix to the labels with the **Thickness Suffix** text box.

Layer Settings: This button displays the below dialog to set the layers for the different portions of the dia-

gram.



The image shows a 'Layer Settings' dialog box with a title bar containing a blue diamond icon and a red close button. The dialog contains several rows of settings, each with a label, a text input field, and a 'Select' button. The settings are: 'Strata Layer' (FENCE), 'Strata Layer Prefix' (empty) and 'Suffix' (empty), 'Strata Polyline Layers' (Layer by Strata Name), 'Pit Label Layer' (FENCE), 'Surface Layer' (FENCE), 'Grid Line Layer' (GRID), 'Grid Text Layer' (GRIDTEXT), 'Fault Line Layer' (FAULT), and 'Fence Polyline Label Layer' (FENCE). At the bottom are 'OK' and 'Cancel' buttons.

Setting	Value	Action
Strata Layer	FENCE	Select
Strata Layer Prefix		
Strata Layer Suffix		
Strata Polyline Layers	Layer by Strata Name	
Pit Label Layer	FENCE	Select
Surface Layer	FENCE	Select
Grid Line Layer	GRID	Select
Grid Text Layer	GRIDTEXT	Select
Fault Line Layer	FAULT	Select
Fence Polyline Label Layer	FENCE	Select

Strata Layer: This value sets the layer the area of the diagram below the last strata layer (the portion of the cross section that has not been modeled as part of the geology). You may also set a **Prefix** and **Suffix** for this layer name.

Strata Polyline Layers: This option controls the layering for the strata polylines.

The *Layer by Strata Name* option will put each strata on a separate layer as defined in the strata definition file. If a strata definition file is not used, the layer for each strata will simply be the name of the strata.

The *All on Same Layer* option will put all strata polylines on the same layer, defined by the **Strata Layer** option.

The *Non-Key on Same Layer* option will place all of the Non-key strata on the same layer, while the Key strata will be placed on their own layers.

Hatch Block Model Layers: This settings allows adding the grade name to the hatch layer as a prefix or suffix.

Pit Label Layer: This option controls the layer for the Pit Labels that are drawn as a result of the **Draw Pit Lines** option.

Surface Layer: This option controls the layer of the surface polyline that is drawn as a result of the **Draw Surface Polyline** option.

Grid Line/Text Layer: These options control the layers of the grid lines and grid labels.

Fault Line Layer: This option controls the layer of the fault polyline drawn as a result of the **Draw Faults** option.

Additional Surface Layer: This option sets the for any additional surface polylines when the Draw Additional Surfaces option is used from the Draw tab.

Fence Polyline Label Layer: This option controls the layer of the labels for the fence name when multiple

fences are drawn at once (e.g. Fence 1, Fence 2, etc.)

Prompts

Fence Diagram Settings dialog box

Select polyline to pull fence diagram from: *pick the polyline*

Select surface entities & at least 3 drillholes.

Select objects: *Select the drillhole symbols and surface entities.* Surface entities can include points, lines, and polylines.

Reading points ... 139

Reading drillholes ...

Choose modeling method (<Triangulation>/Inverse dist/Kriging/Polynomial/LeastSq)? *Press Enter*

Bottom elevation of grid <1600.00>: *press Enter to accept default*

Top elevation of grid <2000.00>: *press Enter to accept default*

Pick the lower left corner for the fence diagram: *pick a point in a clear area of the drawing*

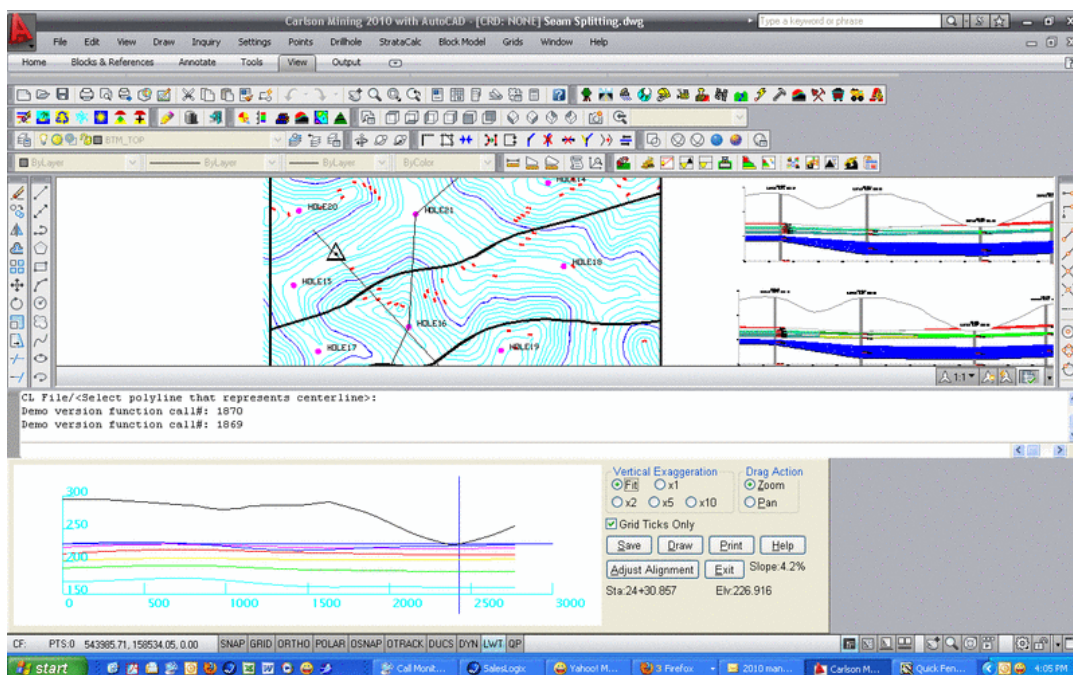
Pulldown Menu Location: StrataCalc

Keyboard Command: fence

Prerequisite: Drillholes or Geologic Model, and a fence alignment polyline

Quick Fence

Quick Fence is very similar to Quick Profile, in that an instant profile appears at the bottom of the screen, displaying the surfaces. This fence alignment can be set by picking points, selecting a centerline file, or selecting a polyline from the screen. A triangle appears in the plan view as the cross hairs are moved along the fence diagram below. This displays where along the fence line the cross hairs are. The slope of the surface, the station and elevation appear dynamically below the buttons. The Adjust Alignment function allows you to drag a horizontal alignment point and update the fence diagram in real-time for quick and efficient grid model verification. It is important to note that this command can only be used with Elevation-based geologic model files. Thickness-based geologic models cannot be used.



- **Vertical Exaggeration:** This usually starts at "Fit" to size the fence diagram into the window. The other

exaggerators, x1, x2, x5 and x10 can be selected to exaggerate it vertically.

- **Drag Action:** The fence image can be modified with either the zoom or pan actions.
- **Grid Ticks Only:** Turning off this box will extend the elevation and stationing tick marks all the way across the fence diagram.
- **Save:** This option will save the surface as a *.PRO profile
- **Draw:** Selecting this will put the fence diagram in the drawing with the Draw Profile command. The result will be similar to Draw Profile with multiple surfaces.
- **Print:** This will print the fence diagram to a PDF file.
- **Adjust Alignment:** This option allows for the fence line on screen to be moved. If the center of the polyline is selected, then the fence line can be moved in any direction, and the seams will dynamically update as the polyline is moved around. If one of the endpoints is selected, then the fence line can be rotated around the other endpoint, or extended to another length. This is a great tool to do a quick check of the geological grids with real-time graphic feedback on the model status.

Prompts

Select the Geologic Model File (*.PRE)

Pick starting point (CL-Centerline,P-Polyline): *p if the polyline is drawn on screen*

Polyline should have been drawn in direction of increasing stations.

CL File/<Select polyline that represents centerline>: *select the polyline*

Drop-Down Menu Location: under StrataCalc

Keyboard Command: qfence

Fence Polylines

There are four commands for managing fence polylines. The fence polylines are used by the Fence Diagram command as the horizontal alignment. You can run Fence Diagram without pre-defined fence polylines and pick the alignment polyline on-the-fly. One advantage with using fence polylines is that you can specify the alignment polyline once and then more easily select the fence polyline in Fence Diagram which is useful if you run Fence Diagram multiple times for the same alignment. Another advantage is that you can use fence polylines to run Fence Diagram on multiple alignments at the same time.

The fence polyline commands are **Tag**, **Untag**, **Label** and **Identify**.

The Tag Fence Polylines command prompts you to pick a polyline and give it a name. This polyline is then flagged as a fence polyline.

The Untag Fence Polylines command removes the tags from the selected polylines.

The Label Fence Polylines command creates a text label of the fence name in plan view at the start of the fence polyline. This label is created in the current layer. All the fence polylines in the drawing get labeled.

The Identify Fence Polylines command reports the name of the fence polyline. For Identify, you can either report the names for the individually picked polylines or have the program search the whole drawing and report the locations and names for any fence polylines.

Prompts

Tag Fence Polylines

Select fence alignment polyline: *pick a polyline*

Fence name <Fence 1>: *West Pit*

Select fence alignment polyline (Enter to end): *press Enter*

Identify Fence Polylines

Pick polylines to check or search drawing [<Pick>/Search]: *S for search*

Fence polyline West Pit at 1645793.98,244010.72, Layer: FENCE

Highlighted 1 fence polylines.

Label Fence Polylines

Text Size <10.00>: *press Enter*

Untag Fence Polylines

Select polylines to remove fence alignment name.

Select objects: *pick the polylines to process*

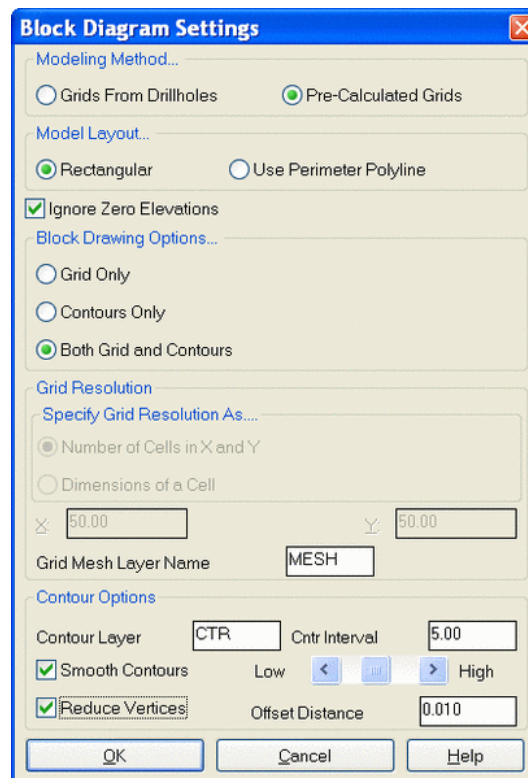
Pulldown Menu Location: StrataCalc

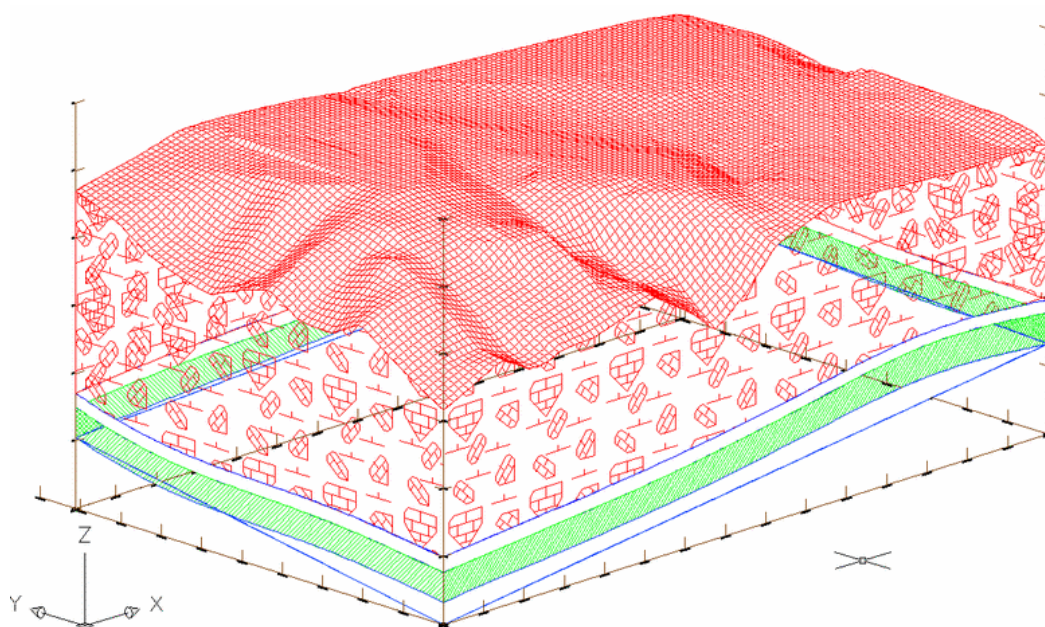
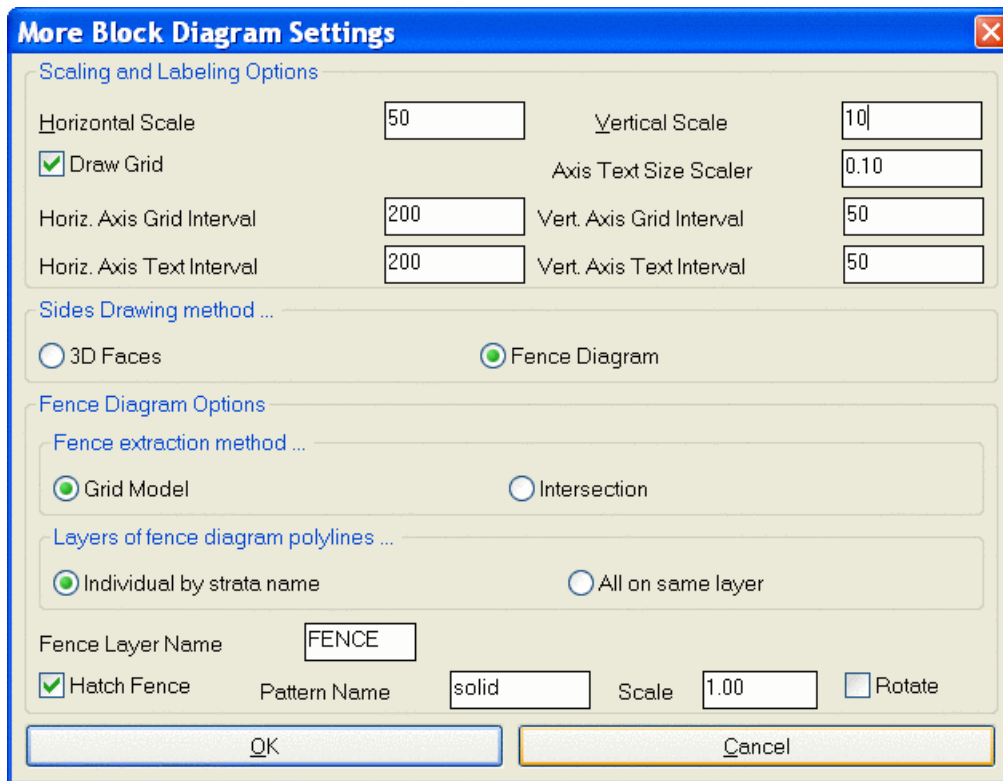
Keyboard Command: tag_fence_cl, untag_fence_cl, id_fence_cl, fence_cl_label

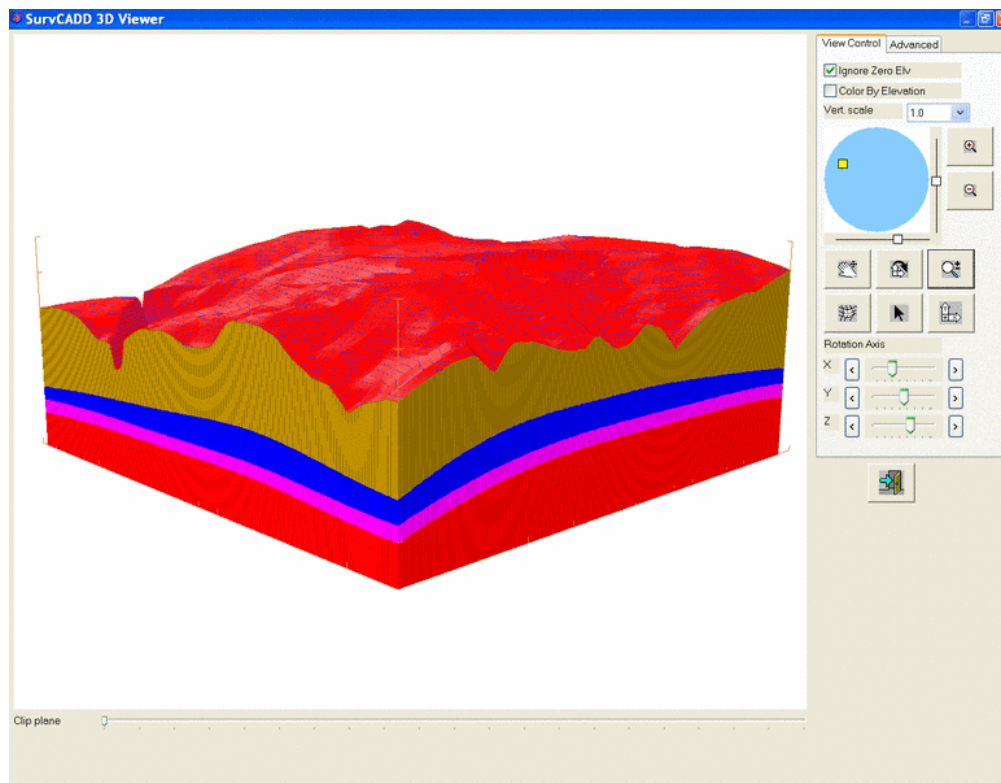
Prerequisite: None

Block Diagram

Block Diagram creates a 3D cut out of the site with contours and/or grid mesh on top and fence diagrams or solid faces on the sides. The options for gridding and contours are shown in the first dialog box. The second dialog shows the options for the fence diagrams and grid. The strata hatch patterns are defined in the Define Strata command. The default hatch pattern at the bottom of the dialog is used if there is no pattern defined for that strata. In order to process a strata for the fence diagrams, there must be at least one drillhole inside and at least one outside the block diagram box. This command is basically a 3D Fence Diagram, with grid faces or contours on top. All of these options are defined in Make 3D Grid File and Draw Fence Diagram.







Prompts

Pick or enter Lower Left block corner:

Pick or enter Upper Right block corner:

Reading cell> 194032

Pass> 7 Null Z values left> 0

Calling fence

Drawing strata OVERBURDEN

Drawing strata C1

Drawing strata PARTING

Drawing strata C2

Drawing grid text ...

Reading cell> 194032

Converting edges ...

Starting contour elevation <4610.000000>:

Ending contour elevation <4920.000000>:

Contouring elevation 4920.000000

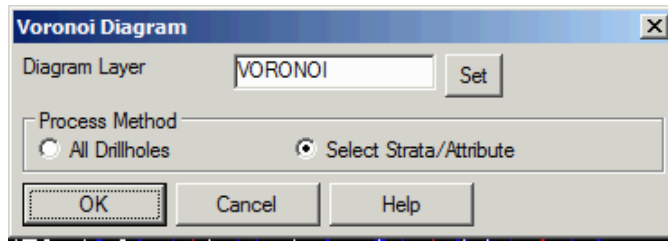
Pulldown Menu Location: StrataCalc

Keyboard Command: blockdia

Voronoi Diagram

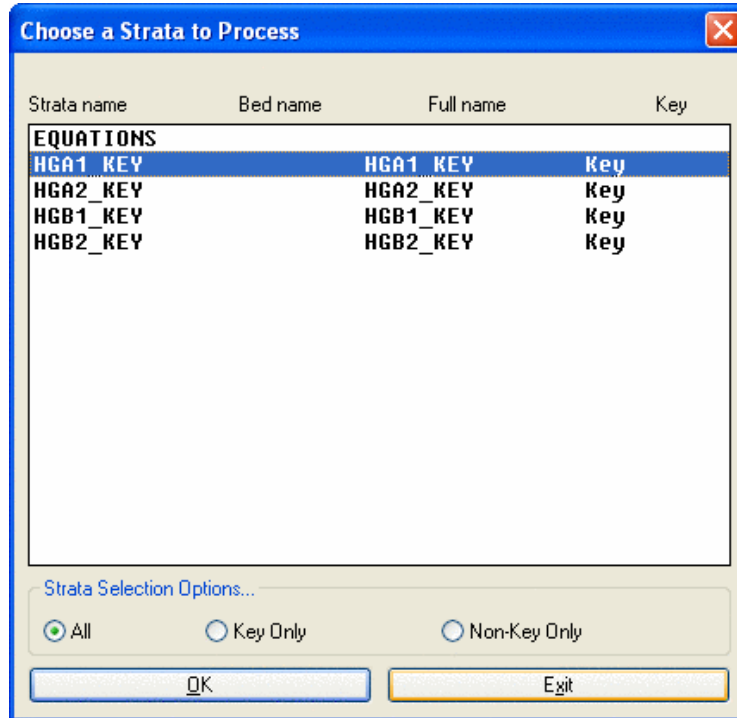
A Voronoi diagram is a special kind of decomposition of a space determined by distances to a specified discrete set of objects in the space, or by a discrete set of points or drillholes. It is useful here to divide the mine area up into polygon linework for the area of influence around drillholes. These are individual polyline segments, not closed polylines.

The first dialog prompts whether to use drillholes or simple CAD points for the input.

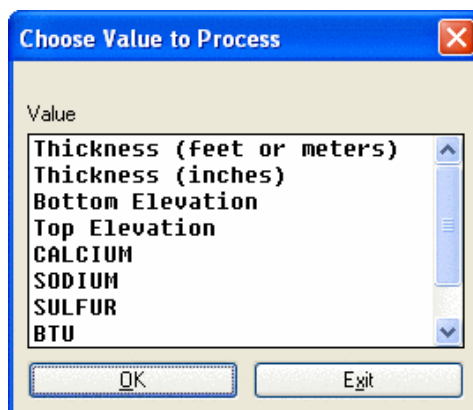


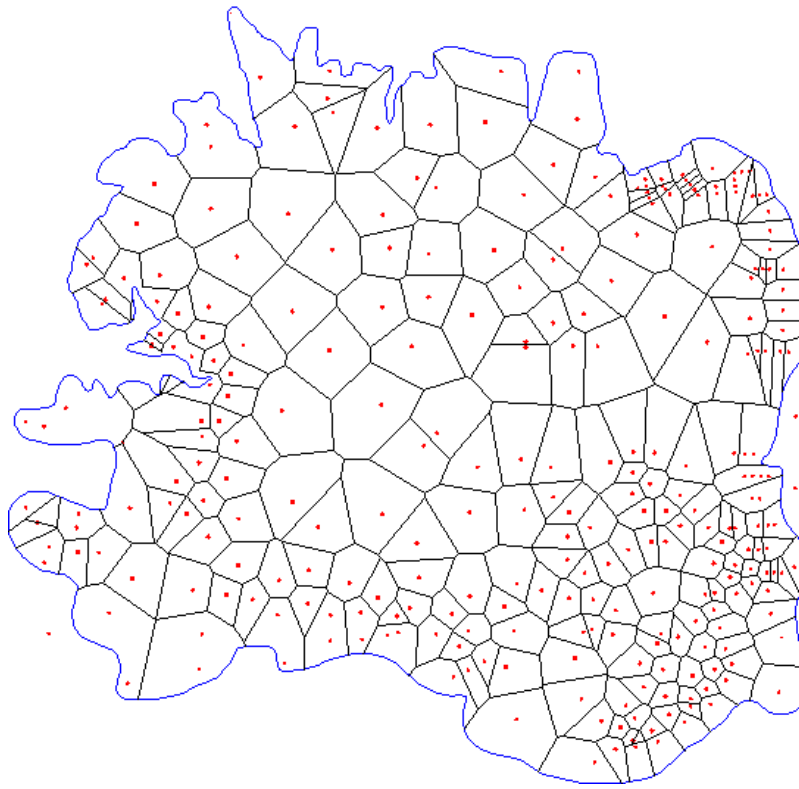
The next dialog prompts for the layer name for the diagram linework to create. The Process Method chooses between using all the drillholes at data points for the diagram or using only drillholes that have a certain strata and attribute.

When using Strata/Attribute, the next dialog asks for a strata to analyze.



The next dialog prompts for the value to process.





Prompts

Select drillholes, channel samples and strata polylines.

Select objects: Specify opposite corner: 437 found

Reading drillhole 437

Finding splits ...

Processing only strata with beds.

Select closed inclusion polyline: *There must be an inclusion polyline selected with this command.*

Drop-Down Menu Location: StrataCalc

Keyboard Command: CHVORONOI

Prerequisite: Drillholes

Geologic Model Viewer

This command loads the strata elevation models from a geologic model (PRE) into the 3D Viewer.

Pull down Menu Location: StrataCalc

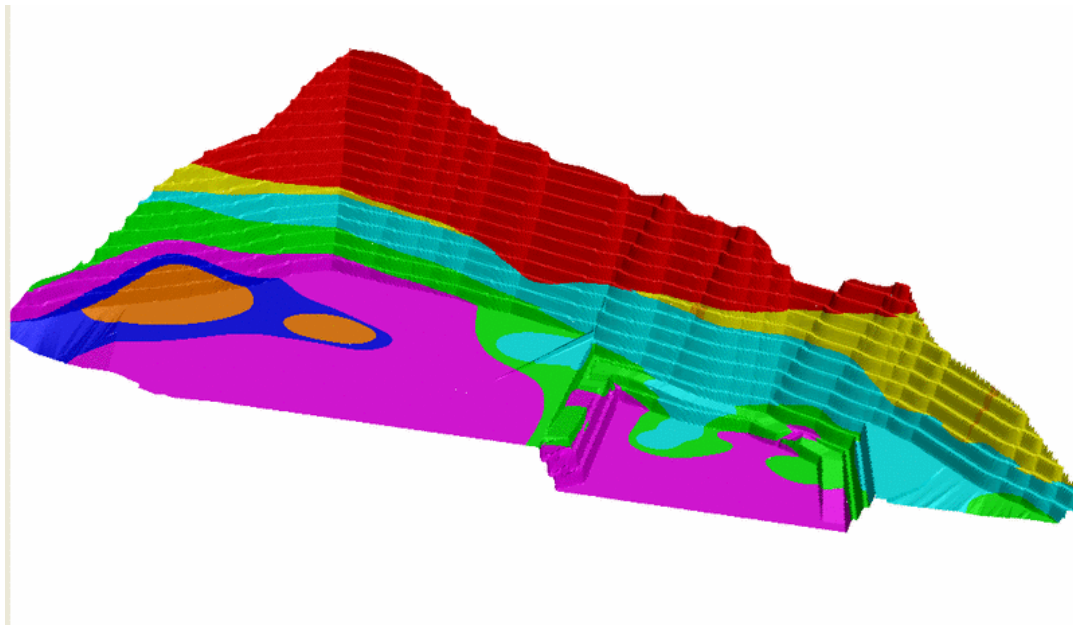
Keyboard Command: view_geology

Prerequisite: Geologic Model file

Color Elev Grid by Strata

This routine colors the 3D grid faces plotted in the drawing. It uses the colors defined by layers, or in the strata definition file and matches the names up with the intervals in the Geologic Model. It is very useful for locating the outcrop of strata, or displaying the subcrops of seams to a mining bench grid. This colored grid can then be viewed in 3D and shaded for full visual effect. Shown below is a final pit of a sand and gravel quarry. The grid has been colored by the seams defined in the Geologic Model file. There are options to set the subdivision tolerance. 0 will not subdivide the grid cells. A max of 5 will subdivide the grid into very small cells, for example, 50 x 50 grids will

subdivide down to 1.5 x 1.5. There is also an option to use the SDF file from Define Strata for the coloring and layers.



Prompts

Select the Geologic Model file

Reading cell> 21364

Select 3D Faces to Process... type all or window the faces

Select objects: Specify opposite corner: 18340 found

Use Layers from Strata Definition File [Yes/<No>]? y

Enter Max Subdivision Level (0-5)<3>: 0

Select 3D Faces to Process...

Select objects: Specify opposite corner: 5005 found

Select objects:

Draw Legend [Yes/<No>]? y

Enter Legend Size <25.0>:

Pick Legend position: *pick location for legend*

Select objects: Enter to accept

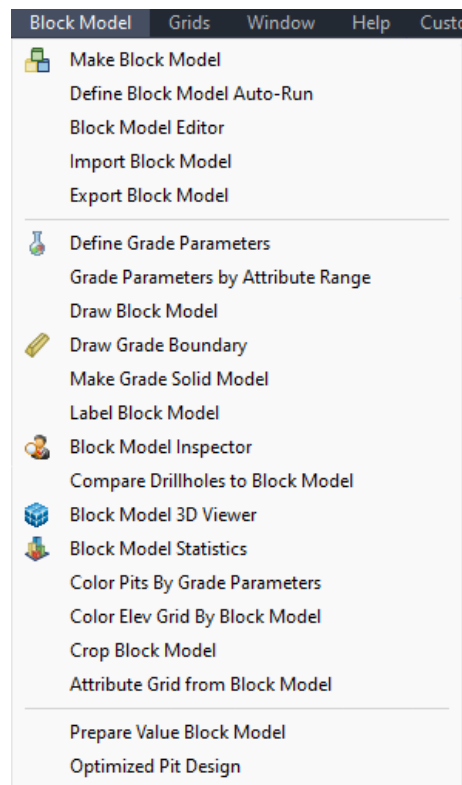
Keyboard Command: strata_color

Pull-down Menu Location: StrataCalc

Prerequisite: Need a 3D Grid plotted in the drawing and a Geologic Model containing grids that intersect the plotted grid.

Block Model Menu

The Block Model menu has commands for creating and viewing block models as well as pit optimization.

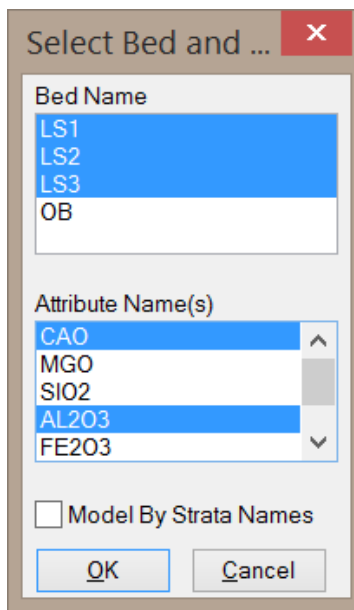


Make Block Model

This command will create a Block Model (.blk file) from drillholes and channel samples. A .blk file is simply a list of .grd files that model the variation of some attribute across a bed (e.g. calcium, magnesium, etc.) or across strata types. Because the block models are represented by a collection of grid files, there are some points that should be noted:

1. The number of blocks in a vertical column for any given X-Y location is constant throughout the model. This allows for variation in block height as the number of vertical divisions is held constant while the upper and lower extents of the block model may vary.
2. Sub-blocking is not currently supported.
3. All attributes must be represented with number. Alphanumeric attributes are not supported.
4. Block models are not a replacement for a Geologic Model, but rather a supplement to it. A Geologic Model may contain multiple block models to describe the variation of some quality of the strata/bed.

When the command is first executed, you will be prompted to select drillholes and channel samples to use. After selecting these entities, the below dialog will appear.



Bed Name: This list will only be populated by beds with attributes. Note that bed names represent a collection of strata. When a block model is created from a bed, there will be no distinction of strata types - all blocks will be treated as being a part of a single bed. In the above example, four beds with quality attributes have been detected: LS and OB. You may select multiple beds for the block model, but it is important to note that the block model will not keep track of the bed name in the blocks.

Attribute Name(s): This list shows the attributes detected within the selected bed. You may include multiple attributes in a single block model by clicking and dragging within the list or by holding the CTRL key while selecting attributes.

Model By Strata Names: This option will create the block model from selected strata names. This option will tag each block in the model as having a different Strata Index number. For example, Limestone blocks may be tagged with a 1, Clay blocks may be tagged with a 2, Overburden blocks may be tagged with a 3, etc. This allows you to distinguish between strata types in the block model rather than treating all blocks as a part of a single bed. You may still model the variation of some attribute throughout the model when this option is selected.

This option will prompt you to create a Grade Parameter File (.gpf file) that defines each Strata Index number. It is recommended to do this as it is the only place to keep track of the strata types in the block model.

This option will also prompt you to create a Geologic Model. This Geologic Model will be listed as having a single strata with a block model added to define the variation in the qualities. When modeling by strata names, the block model actually defines the various types of strata, so it is not necessary to define multiple strata layers in the Geologic Model.

After selecting the options on the above dialog, you will be prompted to select the horizontal extents of the block model by picking X-Y locations on the screen or by referencing the location of an existing grid file. After following these prompts and setting the horizontal extents of the block model, the below dialog will appear.

Specify Horizontal Resolution As...: This option determines how the size of the cells are defined. The Number of Cells in X and Y option will use the **X** and **Y** fields to set the number of cells in the horizontal plane. With this option, the dimensions of the cells will be calculated rather than manually set. The Dimensions of a Cell option will set the cell dimensions using the **X** and **Y** fields. If you opt to set the horizontal extents according to an existing grid file, this dialog will not allow you to set the size of the cells in the X and Y directions.

Vertical Position: This option will define the upper and lower bounds of the block model. The Fixed Elevations option will use flat elevations as specified by the **Bottom Z** and **Top Z** fields. The Follow Ore Model option will create grid files to define the upper and lower extents of the selected bed. With the Follow Ore Model option, the **Block Model Method** will determine how the grids are made.

Crop No Grade: This option is only available when using the Follow Ore Model option. When enabled, you will be prompted for a Grade Parameter File (.gpf file) which defines the various grades of material. When making the upper and lower limits of the block model, these grids will only extend to portions of the bed that have been defined within the Grade Parameter File. In other words, any blocks with values not defined in the Grade Parameter File will be cropped out.

Crop Block by Estimated Strata/Bed: This option is only used when the Vertical Position is set to Fixed Elevation. When this option is off, the blocks will be populated with values at all elevations between the Top and Bottom Z. When this option is on, only some blocks will be populated with block values. The program will create top and bottom elevation grids that follow the ore layer, and any blocks outside of this elevation range will not be populated with values. To ensure that all blocks are populated with values, this option should be disabled.

Number of Vertical Divisions: This option controls the number of blocks per x-y location in the model. If this value is set to 5, a total of 6 blocks will be present in each column of the block model. Here it is important to note that the height of each block will depend on the upper and lower limits of the model. If these limits are set at flat elevations, then the block height will not vary with location. If these limits follow the ore model, it is very likely that the block height will vary with location.

It is also important to note that the top and bottom block of each column will be half the height of the other blocks. This can be seen in the image at the bottom of this article. When calculating reserves with the Surface Mine Reserves command, the program can apply logic to account for the missing half-blocks at the top and bottom of the block model. For more information on this logic, see the section of the help manual corresponding to the Surface Mine Reserves command (subsection "Report Formatter and Miscellaneous Notes").

Block Model Method: This option determines which modeling method is used to create the block model.

1. The Nearest Neighbor method will use the nearest-neighbor method of assigning block values. No interpolation will be applied, but instead the block will take on the attribute value of the nearest sample point.
2. The Kriging method will assign block values based on a variogram. It is recommended to first calculate these variogram parameters using the Geology Module > StrataCalc Pulldown Menu > Calculate Variogram command. When this option is used, you will be prompted with the below dialog.

Minimum: This value sets the minimum attribute value to process when creating the block model.

Maximum: This value sets the maximum attribute value to process when creating the block model.

Min Points to Use: This value sets the minimum number of sample points to use when assigning values to each block in the model.

Max Points to Use: This value sets the maximum number of sample points to use when assigning values to each block in the model.

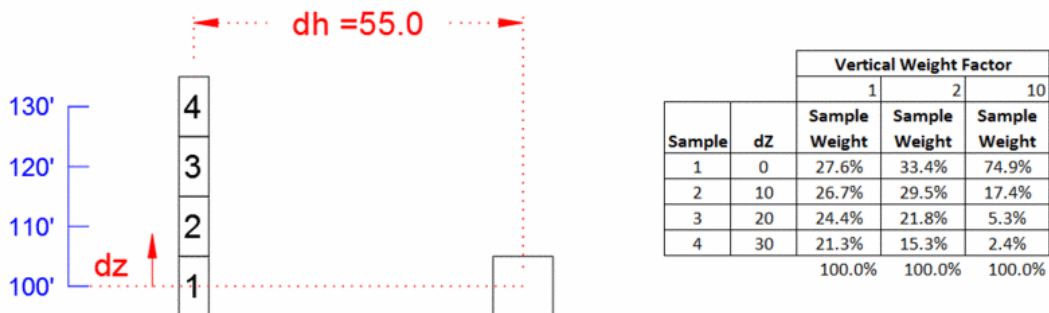
Search Radius X, Y, Z: These three values set the search radius of points to use when creating each block in the model. These values will determine the maximum distance of influence of each data point used to create the model.

On the right side of the dialog are the variogram parameters. These values may be loaded from a previously calculated variogram using the Load Model button. Detailed information on each of these parameters is provided in the section of the help manual relating to the Calculate Variogram command.

3. The Inverse Distance method will use inverse distance weighting to assign block values. When this option is used, you will be prompted to set the weighting factor and if you would like to use elliptical weighting.
4. The Prompt Each Attribute option will allow you to set the Block Model Method for each attribute in the block model. For example, you can use the Inverse Distance method to model one attribute the Kriging method to model another.

For details about each modeling method, see the section of the help manual corresponding to the Make Strata Grids command.

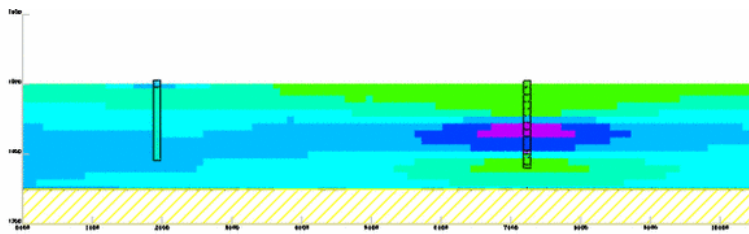
Vertical Weight Factor: When using the Inverse Distance method as the modeling method, the Vertical Weight Factor can be used to modify the degree of vertical trending in the deposit. It should be noted that this value is not the same as the distance weighting power. When this value is set to 1.0, the model will be calculated using normal inverse distance weighting. When this value is greater than 1.0, the vertical distances between the blocks and the drillhole samples will be exaggerated by the Vertical Weight Factor to allow for more vertical variation in the model. When this value is less than 1.0, vertical variation in the model will decrease. The below diagram and table demonstrates how various Vertical Weight Factors will impact the relative weight of each sample on the resulting block value. In this example, an inverse distance weighting power of 2 is used for all calculations.



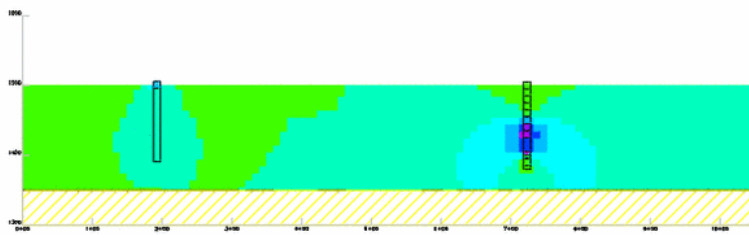
In the above example, there are 4 quality samples in the black drillhole. The example block lies 55' away from the drillhole at a centroid elevation of 100. Each of these 4 samples will be weighted to determine the actual value of the block. The table to the right shows how increasing the Vertical Weight Factor will impact the weight of each sample on the block value. Notice that as the Vertical Weight Factor increases, the weight of Sample 1 on the block value increases significantly, while the weight of Sample 4 on the block decreases significantly. This means that for a higher Vertical Weight Factor, a block's value will more closely resemble that of samples at elevations similar to the block.

The below images show how a block model can vary simply by modifying the Vertical Weighting Factor. Three values are used for comparison: 0.1, 1, and 10. Notice how the Vertical Weight Factor of 10 creates horizontal trending between the data points, whereas the other two model have decreasing degrees of horizontal trending. Note that when the Vertical Weight Factor is set to a low value of 0.1, there is almost no variation in the qualities in the vertical direction.

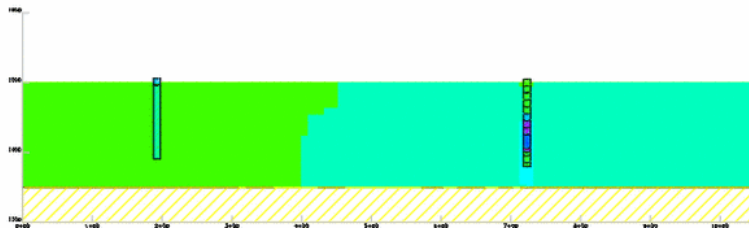
Vertical Weight
Factor = 10



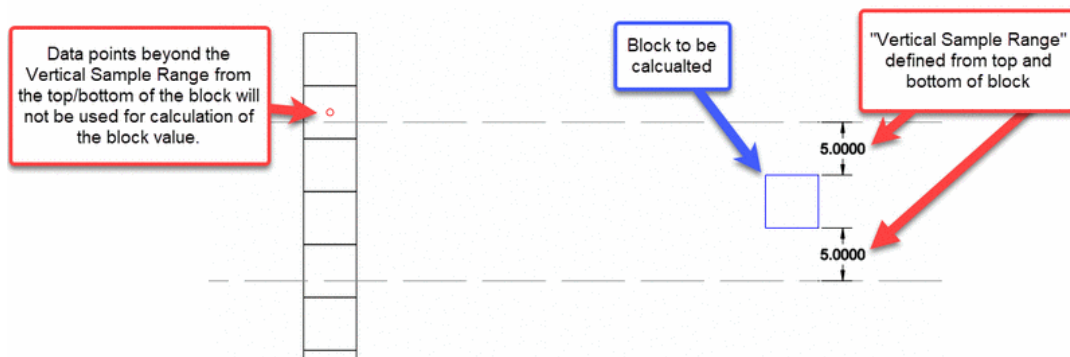
Vertical Weight
Factor = 1



Vertical Weight
Factor = 0.1



Vertical Sample Range: This option allows you to limit the vertical distance the program will search for data points when calculating block values. The below image is a visual explanation of this option. It is important to note that this Vertical Sample Range is checked as each individual block in the model is calculated. For each block level, the program finds sample points from the drillhole within the block level elevation range plus this Vertical Sample Range. When this setting is set to zero, the program creates a single composite data point at each drillhole for each block level.



Horizontal Search Radius: This value sets the search radius for the Inverse Distance modeling method. When the program is calculating the value of a block, any sample points further than this distance from the block centroid will be ignored.

After entering the additional prompts, you will be prompted to give the block model (.blk file) a name. All grids that are created in addition to the .blk file will be named similarly. For example, if the block model is named "Limestone", 5 vertical divisions have been set, and the block model represents the Calcium attribute, then the below files will be created.

Limestone.blk

Limestone-top.grd
 Limestone-bot.grd
 Limestone-Calcium0.grd
 Limestone-Calcium1.grd
 Limestone-Calcium2.grd
 Limestone-Calcium3.grd
 Limestone-Calcium4.grd
 Limestone-Calcium5.grd

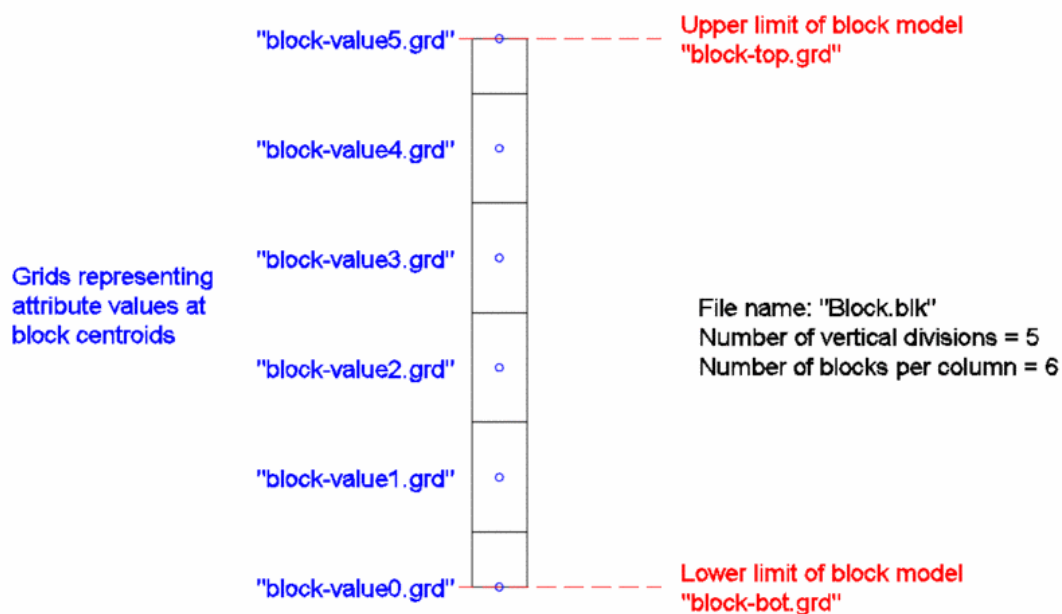
The "Limestone-top.grd" file represents the elevations of the upper limit of the block model.

The "Limestone-bot.grd" file represents the elevations of the lower limit of the block model.

The "Limestone-Calcium*.grd" (where * is a number) files represent the calcium content for each block level. Note that when 5 vertical divisions have been specified, 6 attribute grids will be created. The attribute grid ending with a "0" represents the calcium content for the lower-most block in each x-y location of the block model. The attribute grid ending with a "5" represents the calcium content for the 6th block from the bottom in each x-y location of the block model.

If more than one attribute is modeled, an additional set of grids will be created for each attribute using a similar naming convention.

The below image shows the side view of a column of blocks with grid files labeled in their respective positions.



Keyboard Command: BLKMODEL

Pull-down Menu Location: Geology Module > Block Model

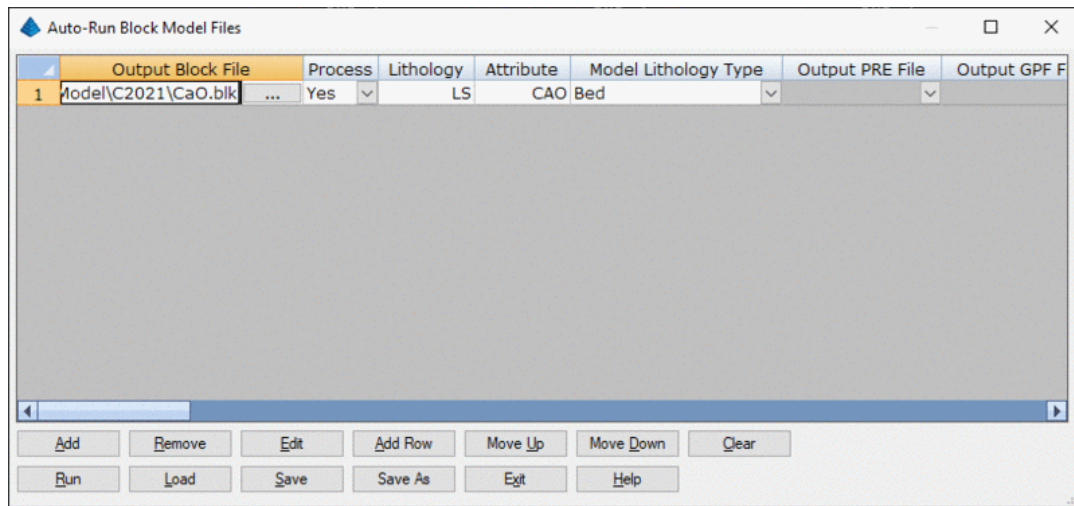
Prerequisite: Drillholes with a bed name, and variable quality values that can be vertically modeled.

Define Block Model Auto-Run

This command will allow you to define one or more block model files to create. This command is simple a macro-form of the Make Block Model command. The advantage of using an Auto-Run is that all settings are saved in a single file. This allows you to quickly recreate block models when input data files change. This also makes it possible to review the settings that were used to create a block model, ensuring new models are consistent with previous work.

When the command is executed, you will be prompted to select the drillhole to use for the calculation. After

selecting the holes, the below dialog will be displayed.



A new block model may be added to the list by clicking the **Add** button. When this button is selected, you will see many of the same settings found in the Make Block Model command. Each setting is explained below, but there are additional settings explained in the Make Block Model command documentation.

You may **Remove** or **Edit** files with the respective buttons. Individual settings may be modified directly in the spreadsheet of the main dialog.

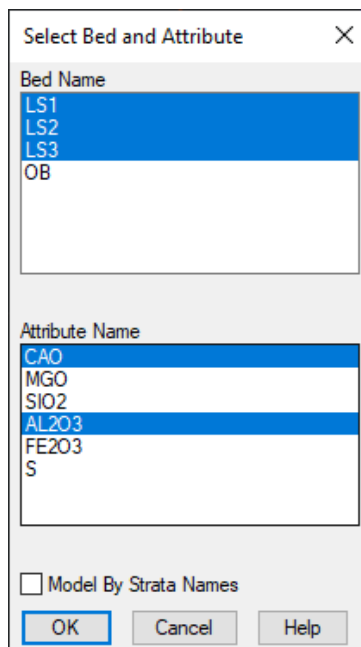
Clicking the **Add Row** button will copy the currently selected block model, but the settings should be reviewed to ensure each model has a unique file name.

Clicking the **Move Up/Down** buttons will move the selected block model up/down in the list. This will change the order in which the block models are calculated.

Clicking the **Clear** button will delete all rows from the spreadsheet.

Clicking the **Run** button will generate the block model(s) in the list. The block model files will not be generated prior to clicking this button. Note that any files that have the **Process** column set to "No" will not be created.

When a new block model is added to the list, you will first be prompted to set the filename. After this, the below dialog will appear.



Bed Name: This list will only be populated by beds with attributes. Note that bed names represent a collection of strata. When a block model is created from a bed, there will be no distinction of strata types unless the **Model by Strata Name** option is enabled. In the above example, four beds with quality attributes have been detected: OB and three distinct groups of LS. You may select multiple beds for the block model, but it is important to note that the block model will not keep track of the bed name in the blocks. Only data points from the selected bed will be used for block estimation. If more than one bed is selected, there will be no distinction between the source of the data (in the above example, the program will mix LS1, LS2, and LS3 data points).

Attribute Name(s): This list shows the attributes detected within the selected bed. You may include multiple attributes in a single block model by clicking and dragging within the list or by holding the CTRL key while selecting attributes.

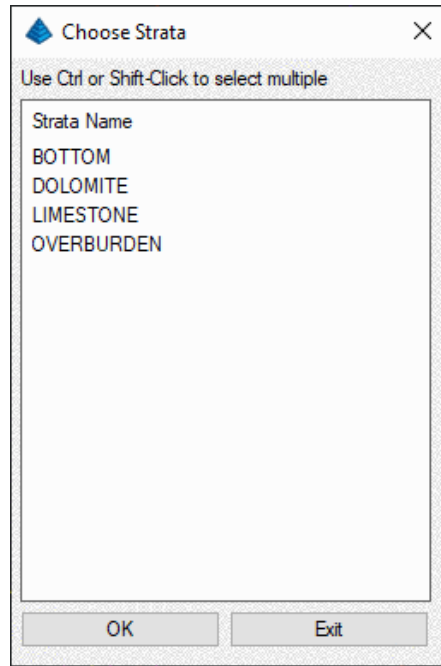
Model By Strata Names: This option will create the block model from selected strata names (strata will be selected on the following dialog). Note that Bed Names do not have to be listed in order to model by Strata Names. This option will tag each block in the model as having a different Strata Index number. For example, Limestone blocks may be tagged with a 1, Clay blocks may be tagged with a 2, Overburden blocks may be tagged with a 3, etc. This allows you to distinguish between strata types in the block model rather than simply grouping all blocks as part of the same bed. You may still model quality attributes in the model when this option is selected.

This option will automatically create a Grade Parameter File (.gpf file) that defines each Strata Index number. It is recommended to do this as it is the only place to keep track of the strata types in the block model. Note that the Grade Parameter File will have the same name as the Block Model, but will have a .gpf file extension.

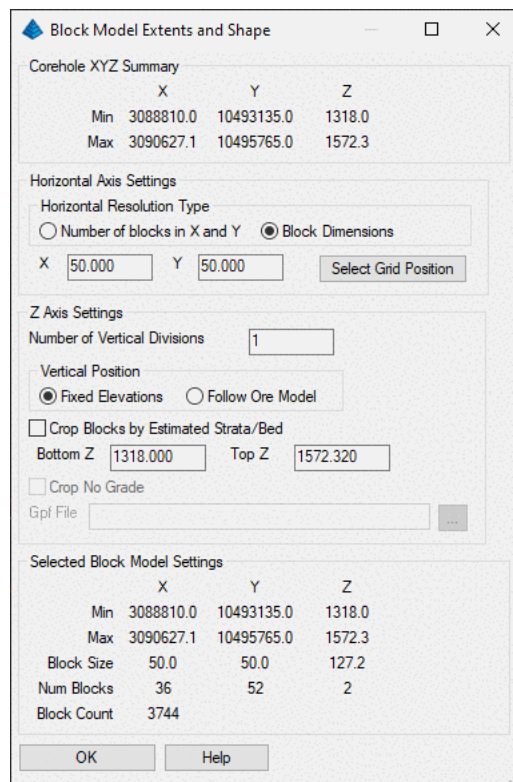
This option will also automatically create a Geologic Model. This Geologic Model will include a single layer with a block model added to define the variation in the qualities. When modeling by strata names, the block model actually defines the various types of strata, so it is not necessary to define multiple strata layers in the Geologic Model. Note that the Geologic Model will have the same name as the Block Model, but will have a .pre file extension.

When this option is enabled, the below dialog will appear after clicking OK. This dialog allows you to select the strata types to include in the block model. It is important to note that strata layers not selected will not

be considered when the block model is created. For this reason, it is recommended to select all strata types in this list.



After clicking OK, the below dialog will appear.



Corehole XYZ Summary: This table lists the minimum and maximum extents of the XYZ coordinates of

the data points to be used for creating the block model. Note that this is not the same as the actual extents of the block model.

Horizontal Resolution Type: This option determines how the horizontal size of the blocks are defined. The *Number of Block in X and Y* option will use the **X** and **Y** fields to set the number of blocks in the horizontal plane. With this option, the dimensions of the cells will be calculated rather than manually set. The *Block Dimensions* option will set the block dimensions using the **X** and **Y** fields as the actual dimensions.

The XY extents of the block model may be set by clicking the **Select Grid Position** button. When this button is clicked, you will be prompted to select the corner extents of the block model.

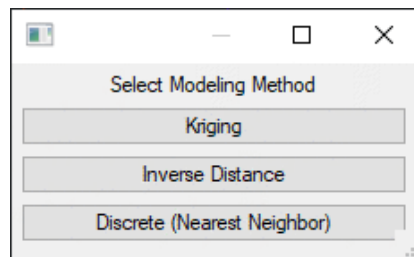
Vertical Position: This option will define the upper and lower bounds of the block model. The *Fixed Elevations* option will use flat elevations as specified by the **Bottom Z** and **Top Z** fields. The *Follow Ore Model* option will create grid files to define the upper and lower extents of the selected bed.

Crop Block by Estimated Strata/Bed: This option is only used when the Vertical Position is set to Fixed Elevation. When this option is off, the blocks will be populated with values at all elevations between the Top and Bottom Z. When this option is on, only some blocks will be populated with block values. The program will create top and bottom elevation grids that follow the ore layer, and any blocks outside of this elevation range will not be populated with values. To ensure that all blocks are populated with values, this option should be disabled.

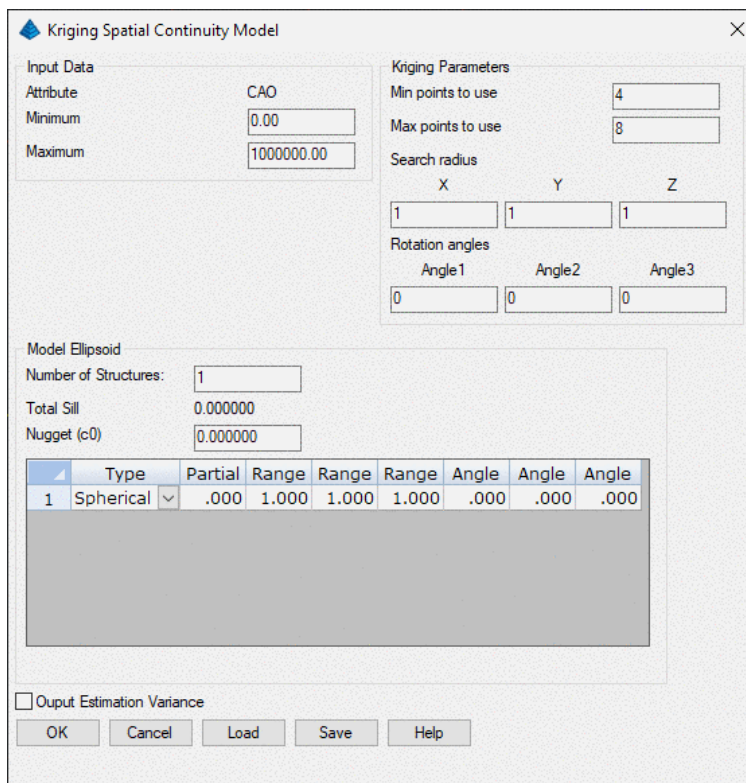
Crop No Grade: This option is only available when using the Follow Ore Model option. When enabled, you will be able to specify a Grade Parameter File (.gpf file) to define the various grades of material. When making the upper and lower limits of the block model, these grids will only extend to portions of the bed that have been defined within the Grade Parameter File. In other words, any blocks with values not defined in the Grade Parameter File will be cropped out.

Selected Block Model Settings: This table lists various information about the extent of the block model to be created. Note that changing settings such as the Top/Bottom Z will cause this table to update.

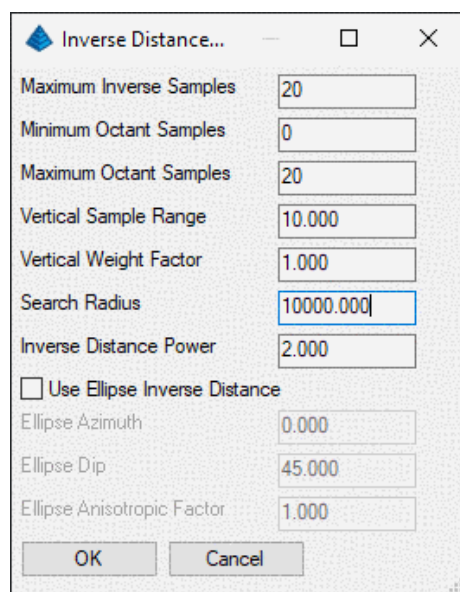
After clicking OK, the below dialog will appear.



1. The **Discrete (Nearest Neighbor)** method will use the nearest data point to of assign block values. No interpolation will be applied, but instead the block will take on the attribute value of the nearest sample point.
2. The **Kriging** method will assign block values based on a variogram. It is recommended to first calculate these variogram parameters using the Geology Module > StrataCalc Pulldown Menu > Calculate Variogram command. When this option is used, you will be prompted with the below dialog to set the kriging parameters.



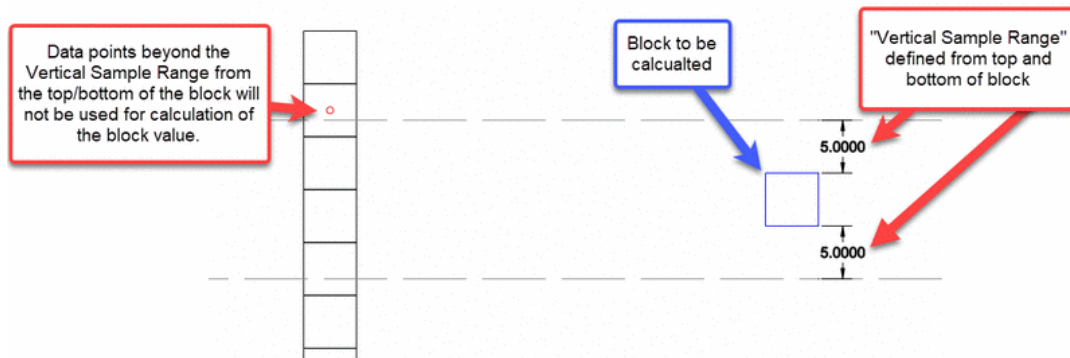
1. The **Inverse Distance** method will use inverse distance weighting to assign block values. When this option is used, the below dialog will be displayed.



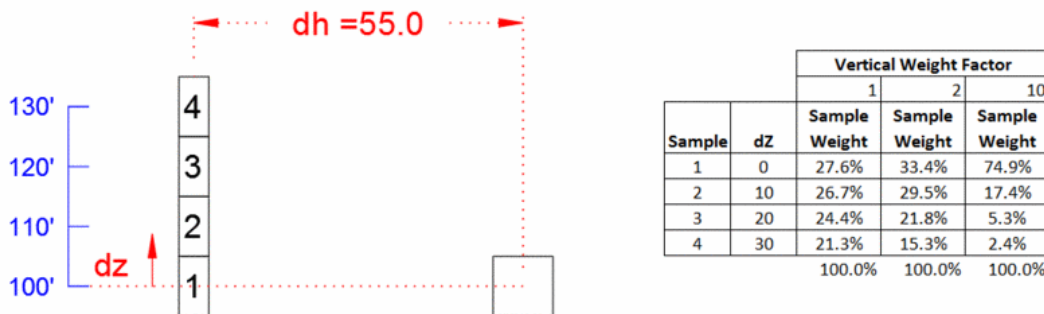
Maximum Inverse Samples: Sets the maximum number of data points to use when calculating the value of each block. The points nearest each block will be used for the calculation.

Minimum/Maximum Octant Samples: Sets the minimum/maximum number of data points to be used in each octant around the block to calculate the block value. If the program cannot detect the minimum number of data points in each octant, the block value will not be calculated.

Vertical Sample Range: This option allows you to limit the vertical distance the program will search for data points when calculating block values. The below image is a visual explanation of this option with a Vertical Sample Range of 5. It is important to note that this Vertical Sample Range is checked as each individual block in the model is calculated.



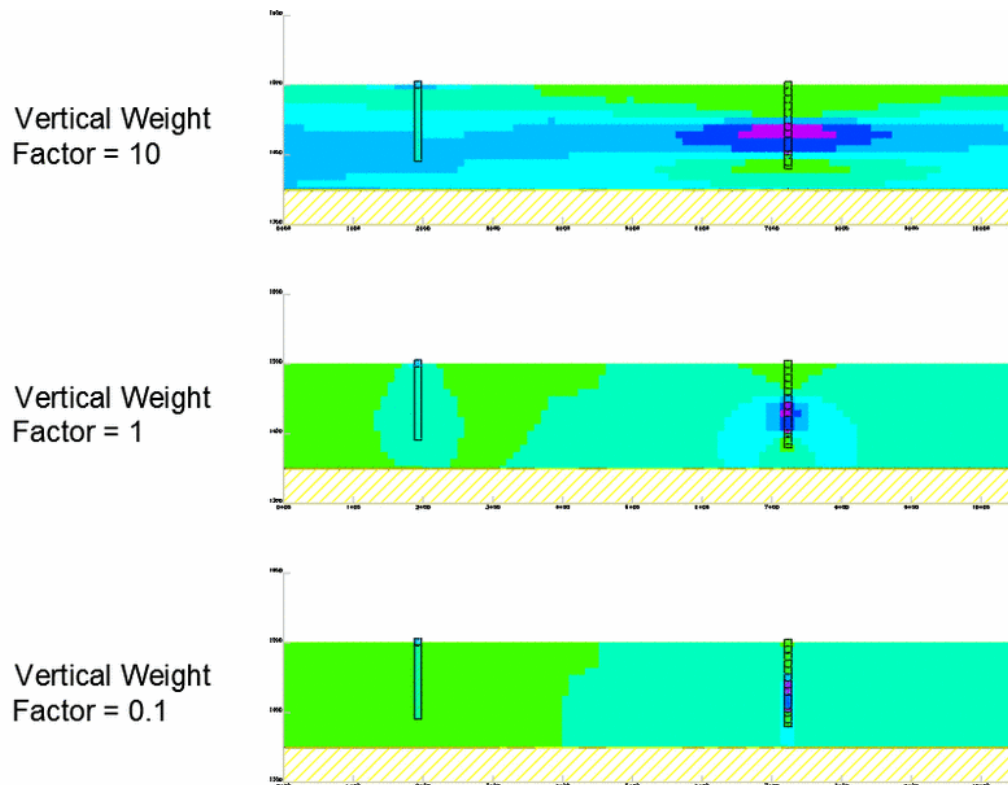
Vertical Weight Factor: When using the Inverse Distance method as the modeling method, the Vertical Weight Factor can be used to modify the degree of vertical trending in the deposit. It should be noted that this value is not the same as the distance weighting power. When this value is set to 1.0, the model will be calculated using normal inverse distance weighting. When this value is greater than 1.0, the vertical distances between the blocks and the drillhole samples will be exaggerated by the Vertical Weight Factor to allow for more vertical variation in the model. When this value is less than 1.0, vertical variation in the model will decrease. The below diagram and table demonstrates how various Vertical Weight Factors will impact the relative weight of each sample on the resulting block value. In this example, an inverse distance weighting power of 2 is used for all calculations.



In the above example, there are 4 quality samples in the black drillhole. The example block lies 55' away from the drillhole at a centroid elevation of 100. Each of these 4 samples will be weighted to determine the actual value of the block. The table to the right shows how increasing the Vertical Weight Factor will impact the weight of each sample on the block value. Notice that as the Vertical Weight Factor increases, the weight of Sample 1 on the block value increases significantly, while the weight of Sample 4 on the block decreases significantly. This means that for a higher Vertical Weight Factor, a block's value will more closely resemble that of samples at elevations similar to the block.

The below images show how a block model can vary simply by modifying the Vertical Weighting Factor. Three values are used for comparison: 0.1, 1, and 10. Notice how the Vertical Weight Factor of 10 creates horizontal

trending between the data points, whereas the other two model have decreasing degrees of horizontal trending. Note that when the Vertical Weight Factor is set to a low value of 0.1, there is almost no variation in the qualities in the vertical direction.



Search Radius: This value sets the search radius for the Inverse Distance modeling method. When the program is calculating the value of a block, any sample points further than this distance from the block centroid will be ignored.

Inverse Distance Power: Sets the weighting power for the inverse distance function. For more information on the Inverse Distance algorithm, see the Make Strata Grid File command documentation.

Use Ellipse Inverse Distance: This option will allow you to introduce a directional trend in the data. For example, if the data values tend to trend in a north-south direction (with more variation in the east-west direction), you may use an elliptical weighting factor.

Ellipse Azimuth: Sets the azimuth for the more heavily weighted data. For example, if data is expected to trend heavily in the north-south direction, an azimuth of zero should be used.

Ellipse Dip: Applies a dip to the anisotropic ellipse.

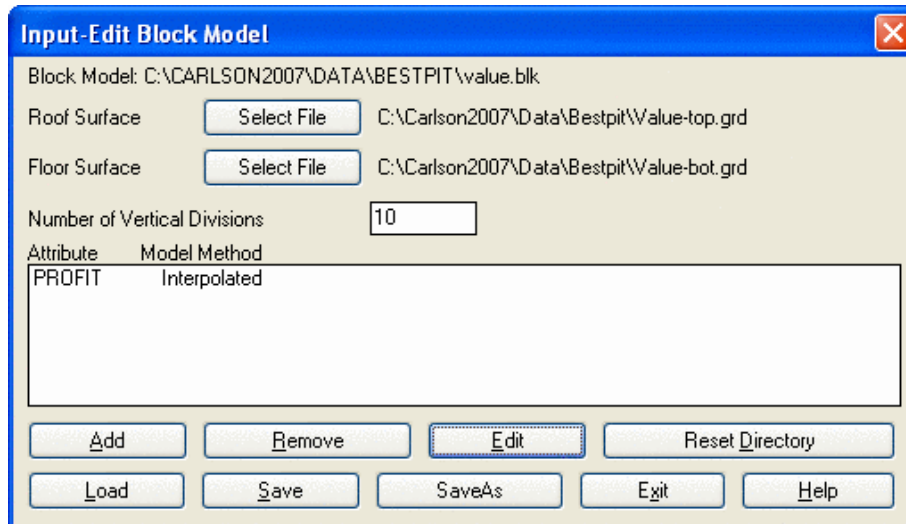
Ellipse Anisotropic Factor: The specified factor is used to increase the weight of data samples that line up more closely with the azimuth. Data samples that exactly line up with the azimuth apply additional weight, whereas data samples that are perpendicular with the azimuth apply zero extra weight. Data samples at an angle between the azimuth and perpendicular apply a proportional adjustment. The specified factor is adjusted by the azimuth and is then added with one and then multiplied by the data sample weight. For example, using an isotropic factor of 1 and a data sample that matches the azimuth to the model point, this sample will get double weighted; $(1 + 1) * \text{weight}$. When using a factor of 2 and a data sample that matches the azimuth, the sample will get triple weighted; $(1 + 2) * \text{weight}$.

Keyboard Command: BLKMODELAUTO

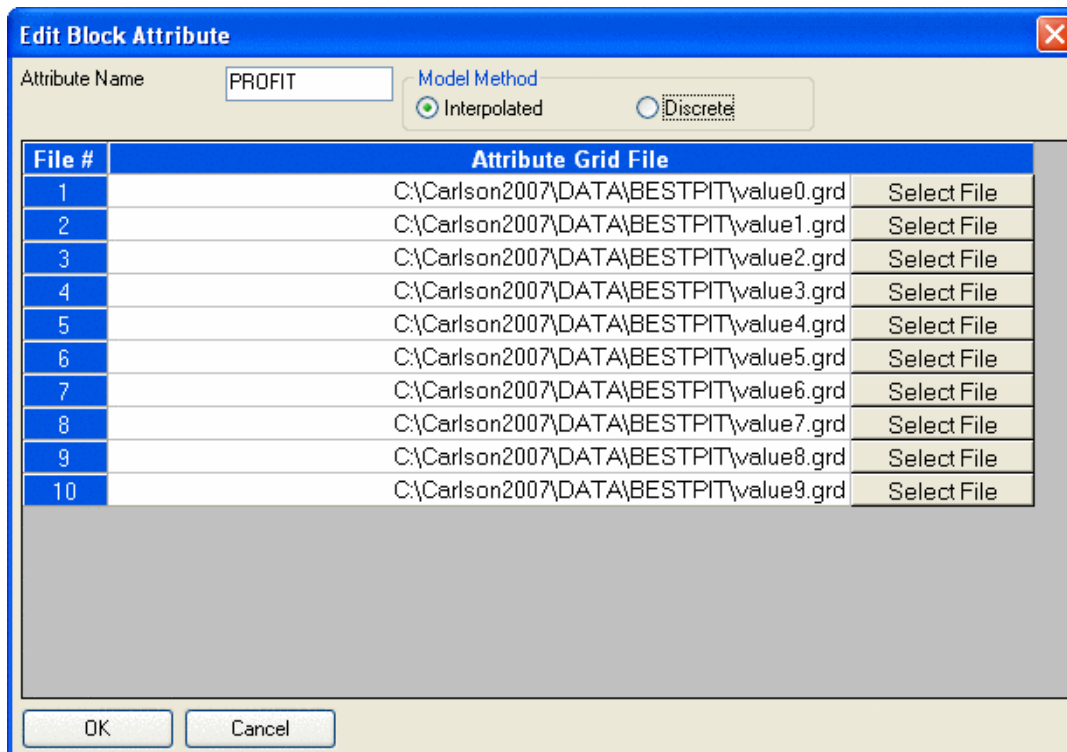
Pull-down Menu Location: Geology Module > Block Model

Block Model Editor

This command allows for creating the block model from pre-made grids, not using drillholes. It looks uses a grid file for the top and bottom of the ore zone. It then will reference at least one quality grid file for each vertical division. This is useful if the roof and floor grids already exist, and they will be used instead of grids made by other routines, such as Make Block Model. The dialogs are very easy to get around in.



Choosing the Add or Edit button will bring up the Edit Block Attribute window. The attribute name is entered at the top. The user can add up to 500 vertical divisions or grids. Save the block model as a BLK file for plotting and for use in Fence Diagram and Surface Reserves.



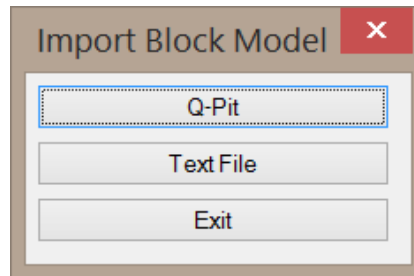
Pulldown Menu Location: Block Model

Keyboard Command: blkedit

Prerequisite: Grids of at least the roof and floor, and a quality grid

Import Block Model

This command imports block models from other programs into a Carlson .blk file. You can import from a Q-Pit file, or from a generic ASCII file from a variety of other programs such as Vulcan, Surpac, Leapfrog, etc. When you start the command, the below dialog will appear.



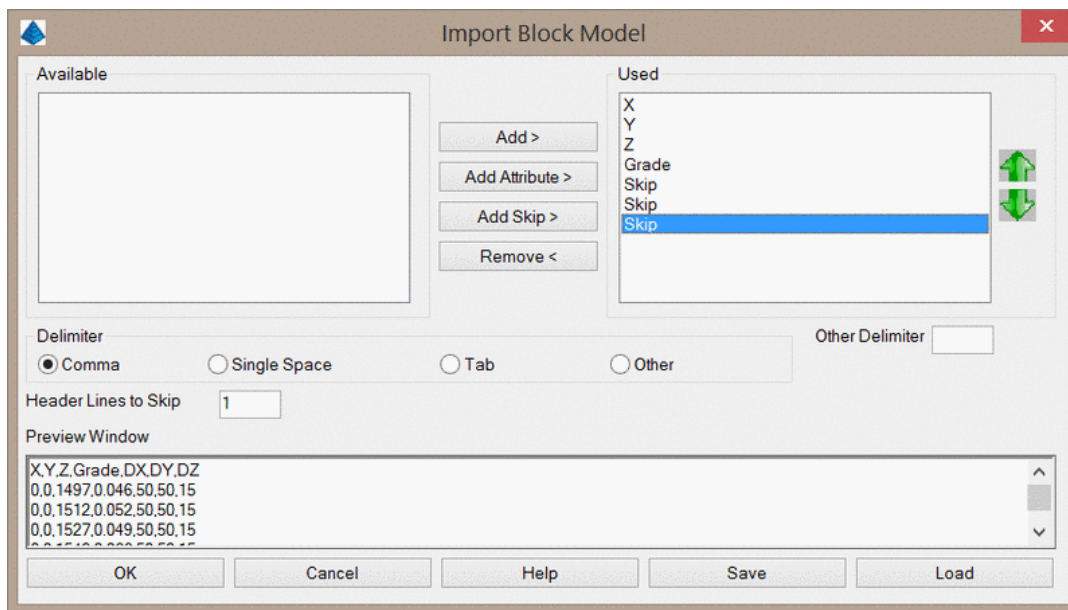
Q-Pit Block Model

This is an automated routine for the Q-Pit file format. After selecting the Q-Pit file, a Carlson .blk file will be created.

Text File

This method allows you to import a generic ASCII text file of the block model. The below dialog will appear, allowing you to define each attribute in the text file. The Available list will include the X,Y, and Z attributes (representing the X,Y, and Z coordinates of the centroid of each block), but you may add additional attributes by clicking the **Add Attribute** button. Only attributes in the Used list will be imported into the block model. You may move attributes between the Available and Used lists by double-clicking the attribute, or by using the **Add** and **Remove** buttons. Attributes should be ordered in this list as they appear in the ASCII file from left to right (the first column of the file should be the first row of the Used list, the second column of the file should be the second row of the Used list, etc.). The Preview at the bottom of the dialog shows the first few lines of the ASCII file to help guide you as you re-order the attributes in the Used list. You may skip over attributes in the file by clicking the **Add Skip** button and ordering the SKIP attribute appropriately.

The X,Y, and Z fields must be used to specify the centroid of each block. You must also specify at least one quality attribute. Other attributes such as dX, dY, dZ which are commonly included in block model exports are not required for this command.



Delimiter: This option specifies the text character that separates the columns of the file. This is most commonly a comma, but other delimiters may be used as well.

Header Lines to Skip: This option allows you specify the number of header lines in the text file. In the above example, there is one header line that must be skipped over.

Null Value: This option is a value in the text file to treat as a null or "no value". For example, 99999.

Please note that Carlson block models do not currently support sub-blocking, mega-blocking, or rotated block models. If you attempt to import a sub-blocked block model, the smallest block size will be applied to all blocks, which can drastically increase the number of blocks that are created in the .blk file. If you attempt to import a rotated block model, the block centroids will still be used, but the program will not be able to detect the true size of each block and will often create a large number of smaller blocks instead.

Pulldown Menu Location: Block Model

Keyboard Command: readblkm

Prerequisite: File to import

Export Block Model

This command writes a comma separated text file (CSV) of block model data. Each row in the file represents the XYZ coordinates of the block centroid, the sizes of the blocks along each axis, and the attribute values. An example is shown below.

	A	B	C	D	E	F	G
1	X	Y	Z	dX	dY	dZ	Grade
2	10000	10000	200	50	50	2	89
3	10000	10000	202	50	50	2	82
4	10000	10000	204	50	50	2	89
5	10000	10000	206	50	50	2	86

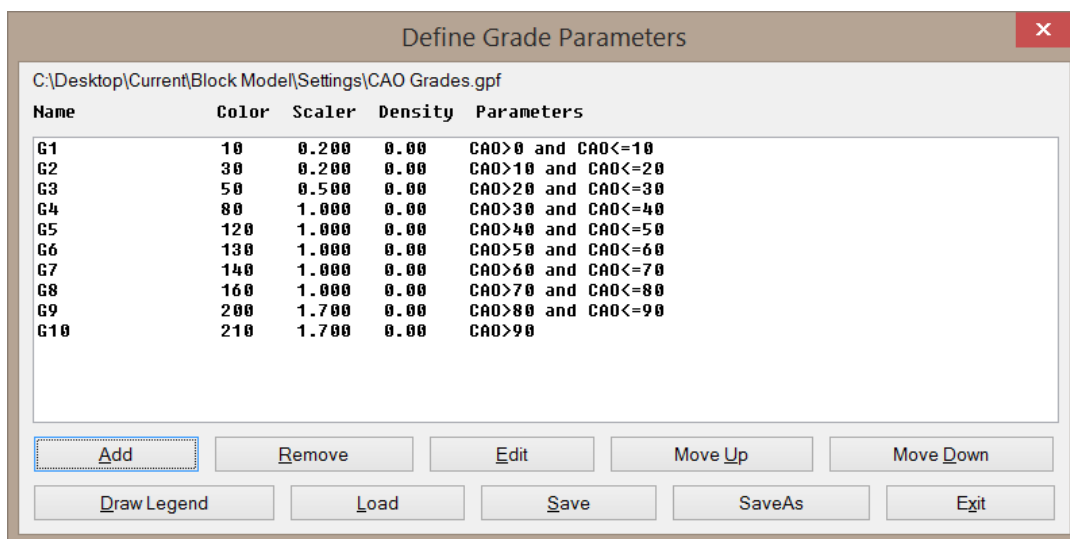
Pulldown Menu Location: Block Model

Keyboard Command: writeblkm

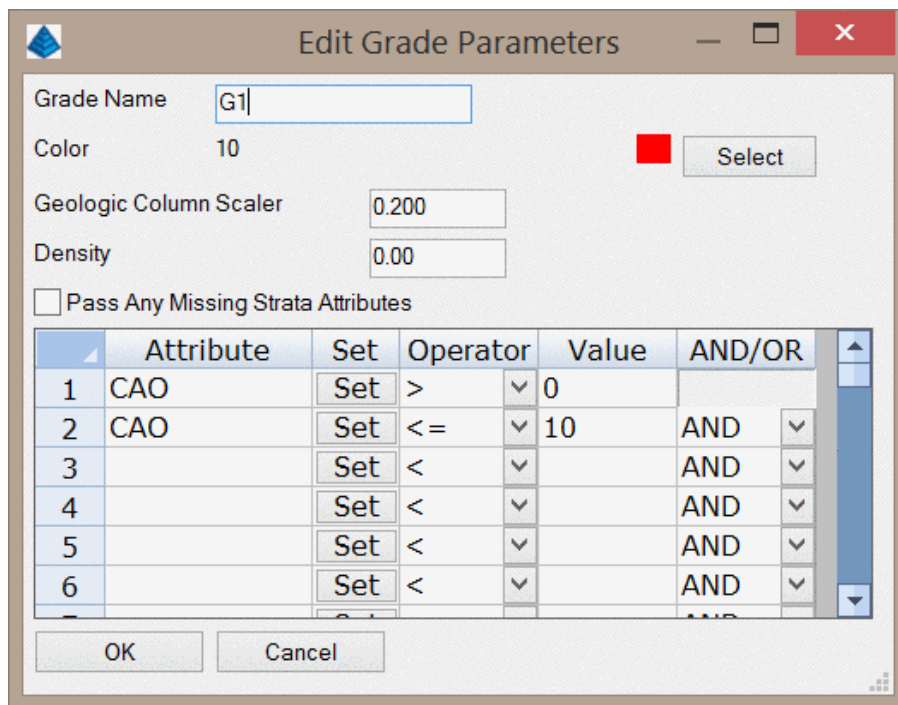
Prerequisite: File to import

Define Grade Parameters

This command creates the grade parameter definitions for use with block models. This assigns each block to a category based on the attribute values to be used for block model viewing and reserve reporting. When the command is executed, you will be prompted to create a new file or open an existing file. After creating/selecting a file, the below dialog will appear. Note that the below example has been populated with ten grade definitions based on CAO percentage.



Clicking **Add** or **Edit** will open the Edit Grade Parameters dialog (shown below).



Grade Name: This text field sets the name of grade. The Grade Name will be displayed in the Block Model 3D viewer and in reserve reports. Grade Names may be used more than once if needed, but it is important to note that duplicate grade names will be reported only once.

The Name of the grade may appear on the list more than once, if several options exist to create a certain grade.

Color: This option sets the display color of the blocks.

Geologic Column Scaler: This value controls the width of the Geologic Column in the Draw Geologic Column command. Changing this value allows you emphasize certain grades by making them smaller or larger. Increasing this value will increase the width of the geologic column.

Density: This value controls the density of the blocks when calculating tonnage with Surface Mine Reserves and Underground Mine Reserves.

Pass Any Missing Strata Attributes: This checkbox determines how missing block values are used for categorization. For example, suppose a grade is based on both CAO and MGO content. With this option enabled, blocks that are missing MGO values can still be considered for grade classification. With this option disabled, blocks without MGO values will not be considered for grade classification.

The spreadsheet in this dialog controls the definitions of the grade. Grades may be defined with up to 50 attribute "tests", where each row of the spreadsheet represents a different test. Each row must contain an attribute name, an operator, and a value. The attribute name must perfectly match the attribute names found in the block model, otherwise an error occur. Attribute names may be entered manually, or by clicking the **Set** button. It is important to note that the **Set** button will only pull attribute names from the Attribute Definition File. An Attribute Definition File is not required to create this file, but if no Attribute Definition File is found, there will be no attribute names to choose from.

There are six Operators available for each test, shown below. It is recommended that each attribute be defined with

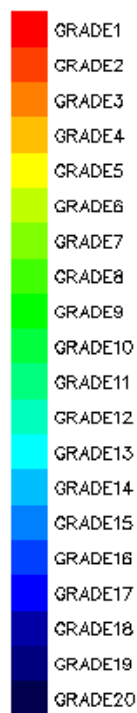
an upper and a lower bound when appropriate. For example, grades that are defined by CAO values between 20 to 30 percent should have a lower bound of 20 percent and an upper bound of 30 percent.

- "<" Less Than
- ">" Greater Than
- "<=" Less Than or Equal To
- ">=" Greater Than or Equal To
- "=" Equal To
- "Not =" Not Equal To

Tests may be compounded with AND or OR relationships. For example, a WASTE grade could be defined as failing to have a passing CAO value OR a passing MGO value.

The **Move Up/Down** buttons control the order of the grade definitions. When the program analyzes a block model, the first grade in the list will be checked first. Any blocks assigned to this grade will not be considered for any other grade classification. Thus, the ordering of the grades can be used to your advantage when hierarchical tests are required.

The **Draw Legend** button places a legend in the drawing as shown here where the grade name is displayed next to the color sample in the legend. The user chooses the legend layer and size. This is useful after color hatching a fence diagram. Drawing the legend next to the fence diagram is a good reference to see the quality of the ore.



Prompts

Pick location for legend: pick location on screen

Layer name for legend <LEGEND>: Enter to select

Size for legend <20.0>: Enter or change size

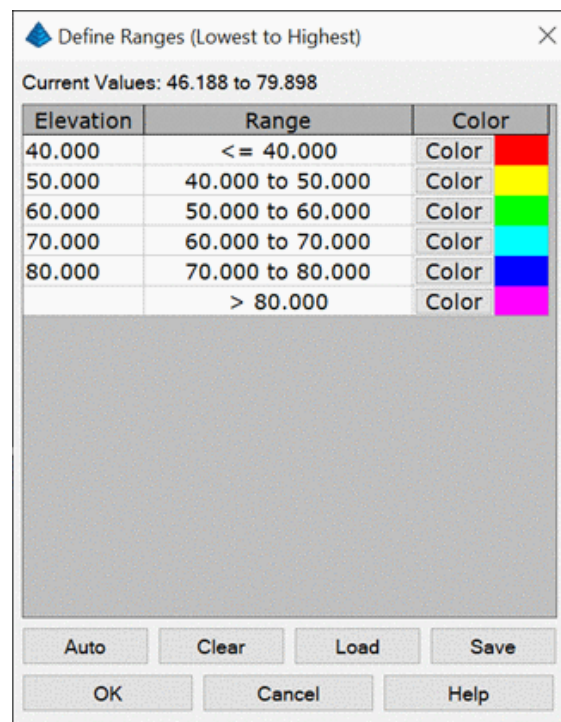
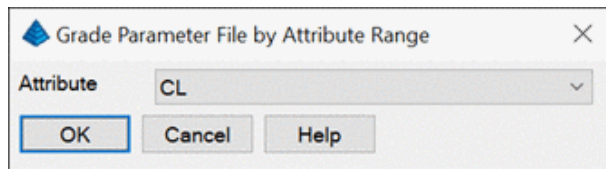
Keyboard Command: DEFATTRF

Pull-down Menu Location: Ore

Prerequisite: Need to run Make Block Model, or Block Model Editor first to get the BLK file. Then define the quality/grade parameter values and their color scheme.

Grade Parameters By Attribute Range

This command creates a grade parameter file for a range of attribute values stored in a block model. The command starts by selecting a block model to process. Then select an attribute from the block model. The program reads the values of the attribute in the block model and display the minimum and maximum values in the Define Ranges dialog. Enter the range values from lowest to highest. You can also use the Auto button to fill out the ranges and colors by an increment. After the Define Ranges dialog, the program prompts for the grade parameter file (GPF) to create. The grades are named by the range values. If you want to edit the grade names, run the Define Grade Parameters command



Keyboard Command: blk_attr_gpf

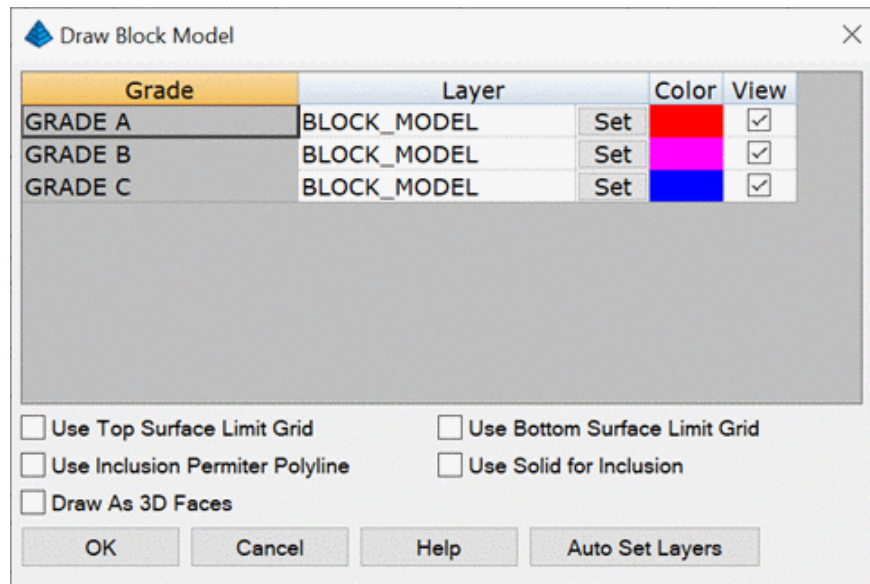
Pull-down Menu Location: Block Model

Prerequisite: Block model (blk)

Draw Block Model

This command draws a block model in the CAD drawing. Once in the CAD drawing, the blocks can be viewed with the 3D Viewer Window or by orbiting the drawing itself. When the command is executed, you will be prompted to select a .blk file and a grade parameter file (.gpf file - controls coloring of the blocks). After selecting these files, the

below dialog will appear.



Spreadsheet: In the spreadsheet you can customize the properties of the grades including the layer name, color and whether to draw.

Use Top/Bottom Surface Limit Grid: These toggles allow you select constraining elevation grids. Any blocks with a centroid above/below these elevation grids will not be drawn.

Use Inclusion Perimeter Polyline: When enabled, you will be prompted to select an inclusion polyline. Any blocks with a centroid outside this polyline will not be drawn.

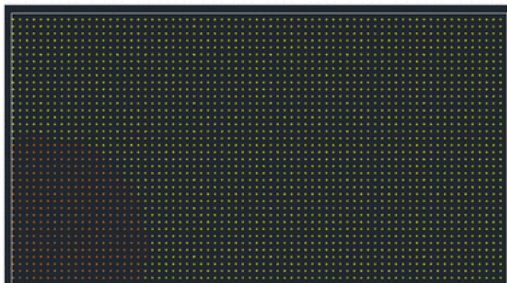
Use Solid for Inclusion: When enabled, you will be prompted to select a solid (.mdl file). Only blocks with centroids inside of this solid will be drawn.

Draw as 3D Faces: When enabled, all six sides of the blocks will be drawn as 3D faces. When this option is disabled, the blocks will be drawn as points. Drawing the blocks as points will reduce the demand on your computer's hardware, thus allowing for quicker camera movements.

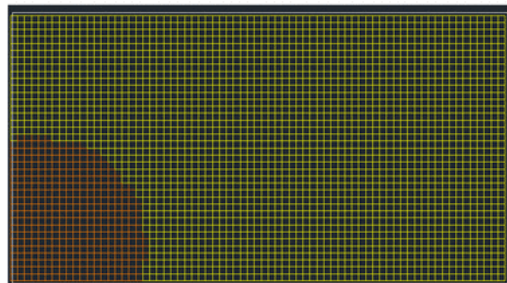
Auto Set Layers: This function sets the layer name by the grade names with a specified prefix and suffix.

After clicking **OK**, the block model will be drawn in CAD, as shown below.

Drawn as Points



Drawn as 3D Faces



Even though the blocks may be drawn as points, these points will be rendered as full blocks if loaded into the 3D Viewer Window. When the blocks are drawn as 3D faces, you may wish to modify the SHADEMODE variable to toggle between a wireframe view and a shaded view.

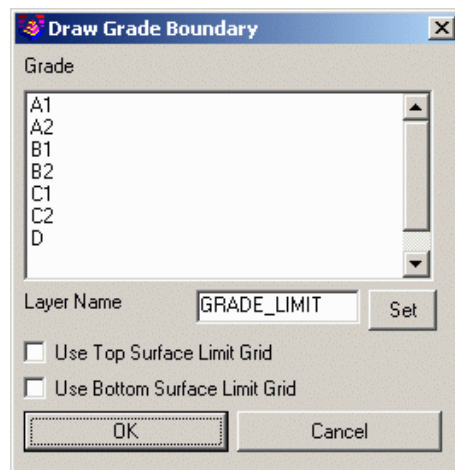
Keyboard Command: drawblkm

Pull-down Menu Location: Block Model

Prerequisite: Need a *.BLK file and a *.GPF file from Make Block Model, Block Model Editor and Define Grade Parameters.

Draw Grade Boundary

This command draws polylines along the boundary of areas of a block model that contain a specified grade. The grades are defined in a Grade Parameter file (GPF). After selecting the block model and grade parameter files, there is a dialog to choose the target grade to process and the layer for the boundary polylines to create. There are options to limit the top and bottom of the block model to check for the target grade. Any sample in a column of the block model that has the target grade and is within the top and bottom limits will make the location of that column count as an area for containing the target grade. So the grade boundaries mark areas that have at least some of the target grade while these areas might also have some other grades.



Prompts

Block Model To Process file selection

Grade Parameter File To Process file selection

Draw Grade Boundary dialog

Pulldown Menu Location: Block Model

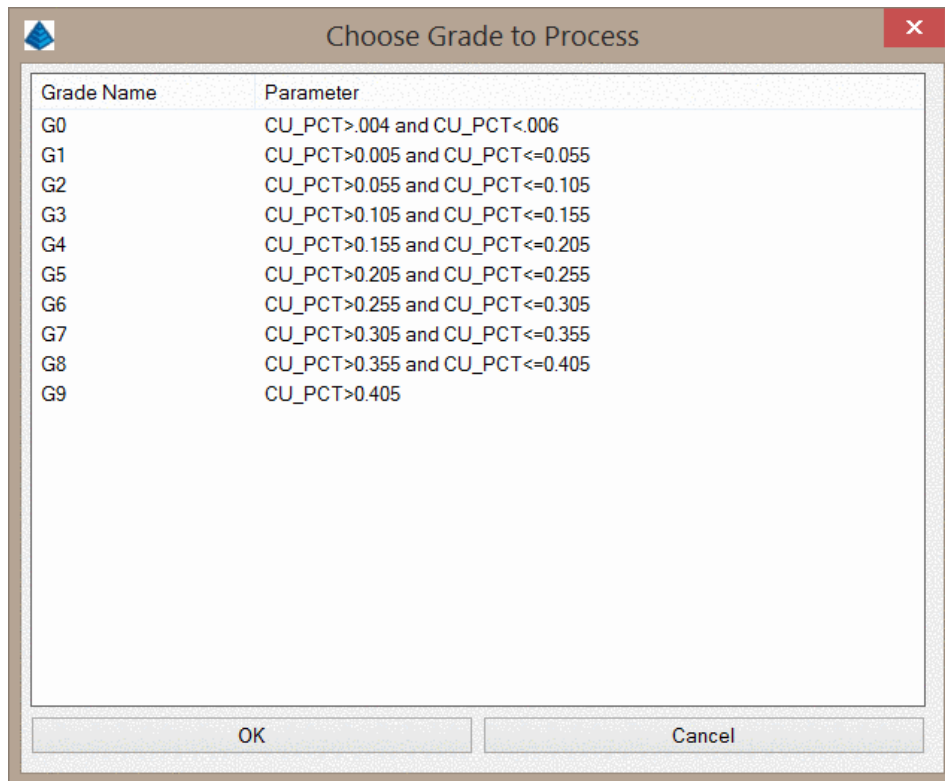
Keyboard Command: draw_grade_limit

Prerequisite: A BLK model file and a GPF grade parameter file

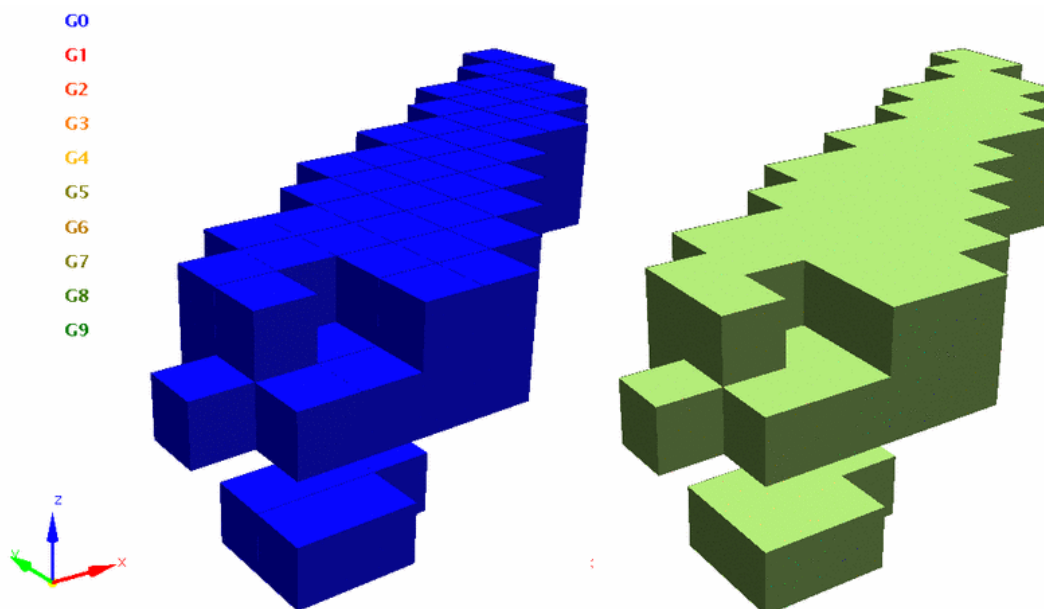
Make Grade Solid Model

This command will create solids (.mdl files) of blocks in a block model matching a specific grade. The inputs are simply a block model and a grade parameter file, and the output is a solid (.mdl file).

When the command executes, you will be prompted to select an existing block model (.blk file) and a grade parameter file (.gpf file). After selecting these files, the below dialog will appear, listing all of the grades.



Here you may select a grade to create a solid. After clicking OK, you will be prompted to specify an .mdl file to create. The below image shows an example Grade in a block model (left) and the corresponding solid that is created using this command (right). Notice how the original block model shows the individual blocks, while the solid does not.

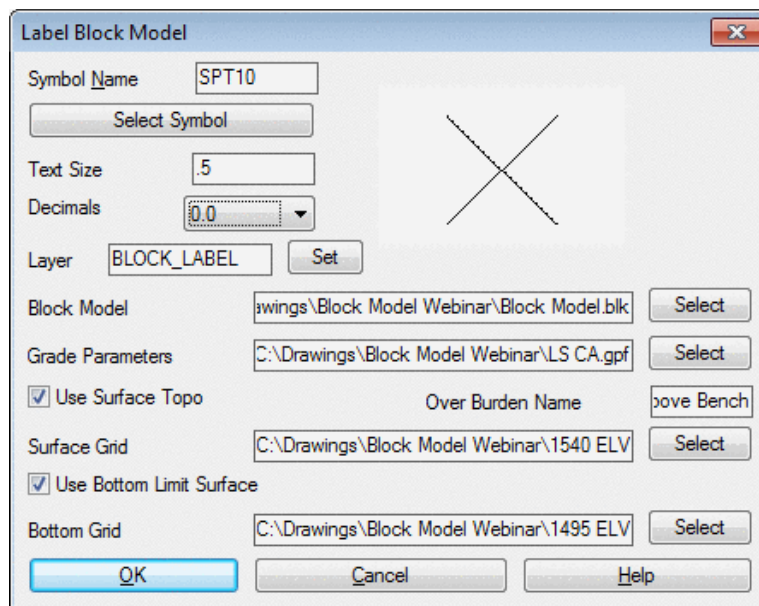


Block Model File
Grade Parameter File
Select Grade to Create Solid
Solid to Create

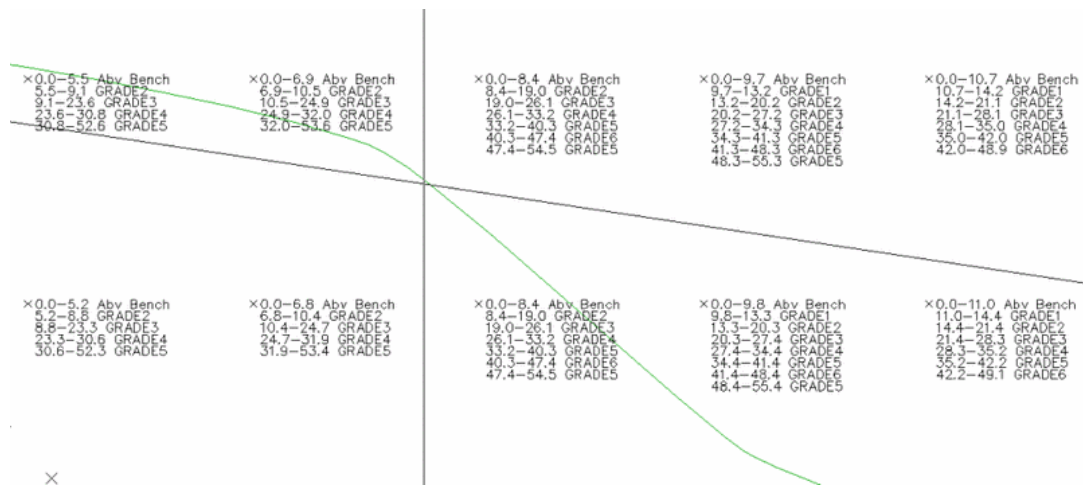
Pulldown Menu Location: Block Model
Keyboard Command: BLK_TO_SOLID
Prerequisite: Block Model and Grade Parameter File

Label Block Model

This command is an automatic way to label the grades of a block model between two surfaces. There are options for symbols, text size, decimal precision and the CAD layer it will be drawn on.



The Block Model and Grade Parameters are set first. Then there is the optional Surface Topo and Bottom Limit Surface to contain the labels to just that depth for the labels. This is useful if the blocks within a certain bench height from 1540 down to 1495 elevation are to be labeled. Once OK is selected, there are prompts to select an inclusion and exclusion perimeter. After that is an option to skip over block cells to label. This will create fewer labels based on how many are skipped over.



Prompts

Select the Inclusion perimeter polylines or ENTER for none.

Select objects: *pick polyline*

Select the Exclusion perimeter polylines or ENTER for none.

Select objects: *press Enter*

Number of cell to skip <2>: *press Enter*

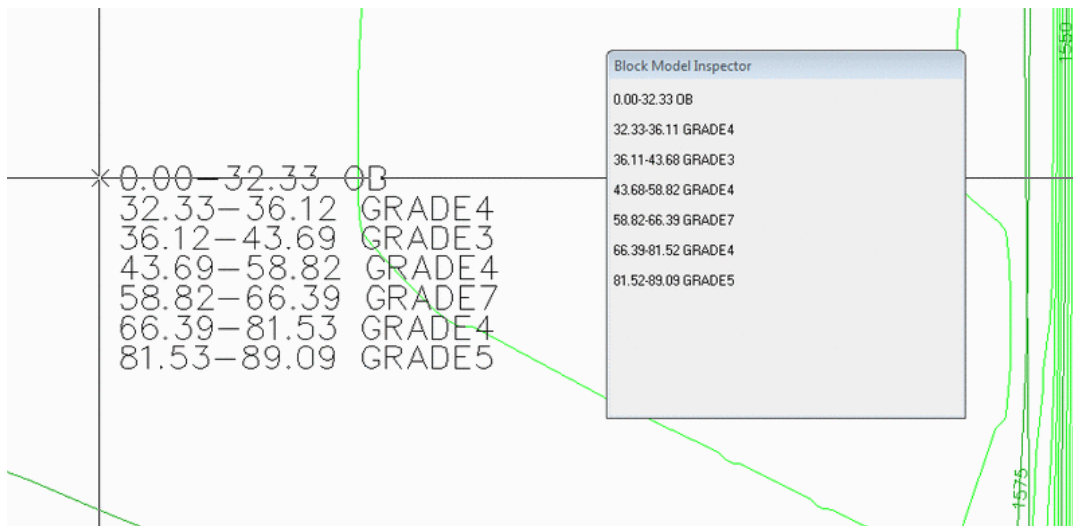
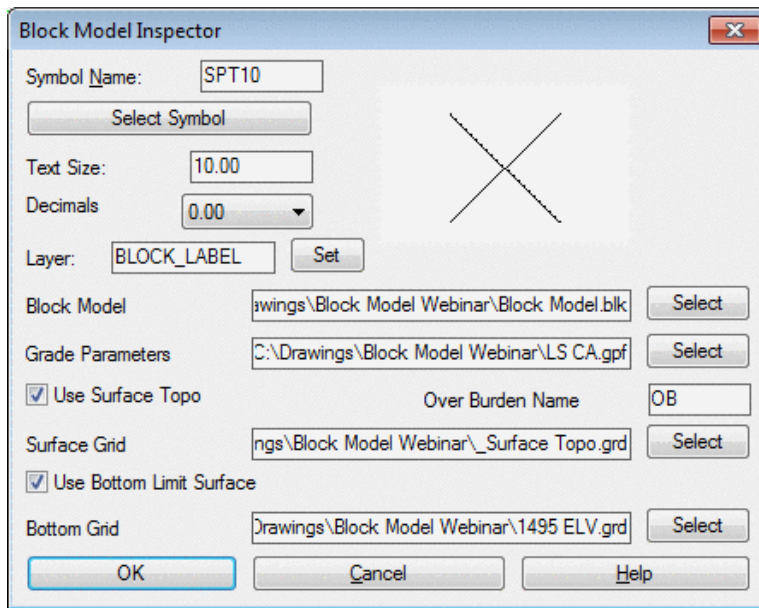
Pulldown Menu Location: Block Model

Keyboard Command: label.blkm

Prerequisite: Need a BLK model file and a GPF grade parameter file.

Block Model Inspector

This command displays the from and to depths, and the grade value as the cursor is moved across the block modeled area. The location can be labeled with a left mouse click to put the text in the drawing. This is a quick way to check the model and for labeling maps to give to the field crew showing the depth range and grade. If "Use Surface Topo" option is on, the depth is calculated from the "Surface Grid" and labeled as "Over Burden Name", such as OB. The Use Bottom Limit Surface will contain the model so that it only labels down to it, such as a flat bench elevation, or pit floor grid. If there exists a no-grade zone for certain elevation range this zone is reported as No-Grade zone.



Φ 0.00-17.70 OB
 17.70-18.17 GRADE_B
 18.17-19.11 GRADE_A
 19.11-21.46 GRADE_B

Prompts

Reading cell> 4828

Select symbols/<Enter or pick point>:

Pulldown Menu Location: Block Model

Keyboard Command: blkm_inspector

Prerequisite: Need a BLK model file and a GPF grade parameter file.

Compare Drillholes to Block Model

This command reports differences between strata attribute values from selected drillholes and attribute values from a block model. This command is a way to check how closely a block model matches the drillhole data. For each strata attribute value, the program compares to the block model at the strata position. The resulting values are reported using the Report Formatter.

Prompts

Block Model To Process file selection

Select drillholes.

Select objects: *pick the drillholes*

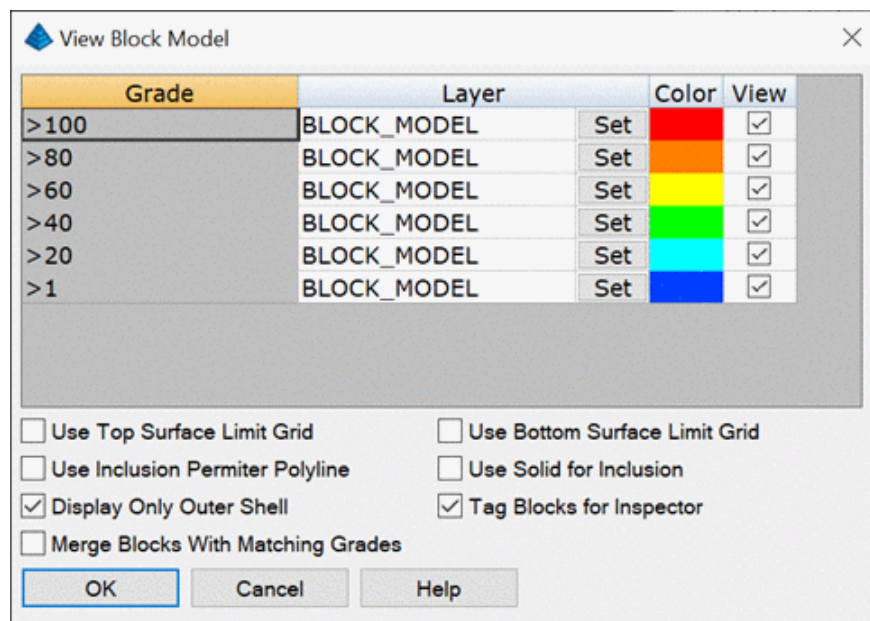
Pulldown Menu Location: Block Model

Keyboard Command: blkm_compare

Prerequisite: A BLK model file and drillholes

Block Model 3D Viewer

This command allows you to view a block model in the 3D viewer window directly from the .blk file. When the command is executed, you will be prompted to select a .blk file and a grade parameter file (.gpf file - controls coloring of the blocks). After selecting these files, the below dialog will appear.



Spreadsheet: In the spreadsheet you can customize the properties of the grades including the layer name, color and whether to view.

Use Top/Bottom Surface Limit Grid: These toggles allow you select constraining elevation grids. Any blocks with a centroid above/below these elevation grids will not be drawn in the 3D Viewer.

Use Inclusion Perimeter Polyline: When enabled, you will be prompted to select an inclusion polyline. Any blocks with a centroid outside this polyline will not be drawn in the 3D Viewer.

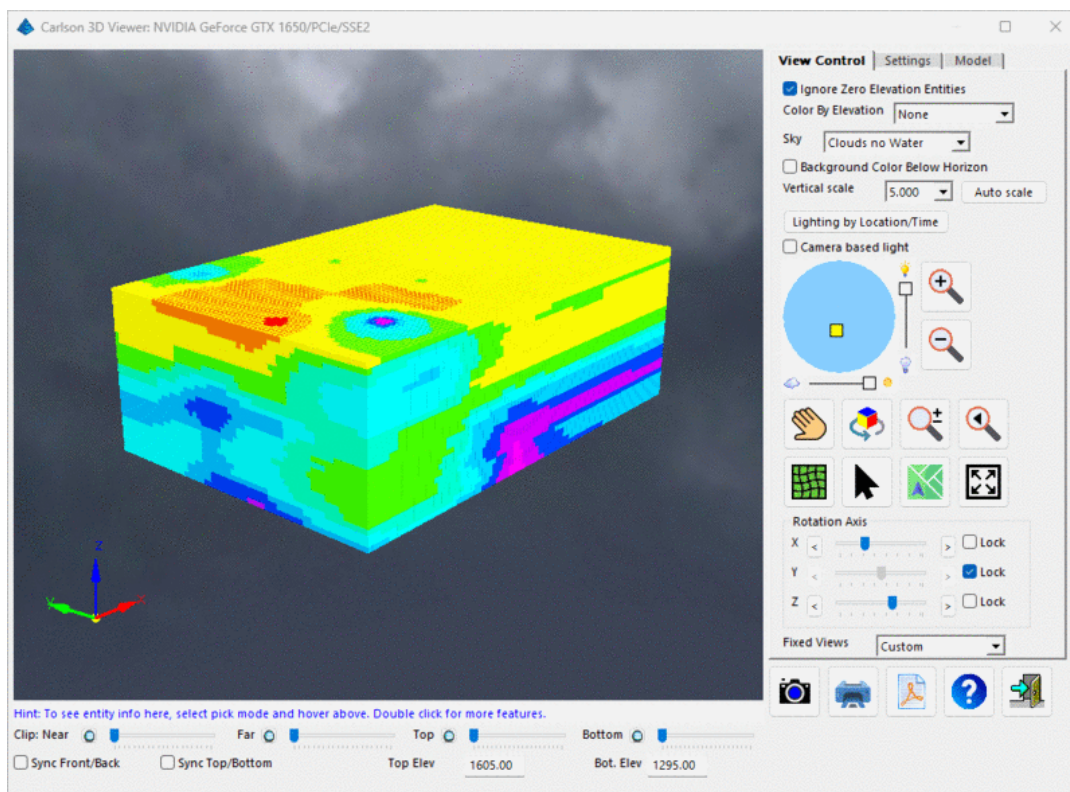
Use Solid for Inclusion: When enabled, you will be prompted to select a solid (.mdl file). Only blocks with centroids inside of this solid will be drawn.

Display Only Outer Shell: This option renders only the blocks that border a different grade or border the sides of the model. Otherwise, all the blocks are rendered.

Tag Blocks for Inspector: This option stores data of the block dimensions and grade in each block. Then in the 3D Viewer, you can pick the pointer mode and the viewer reports these values as you move the cursor over a block. Turning this option off speeds up loading.

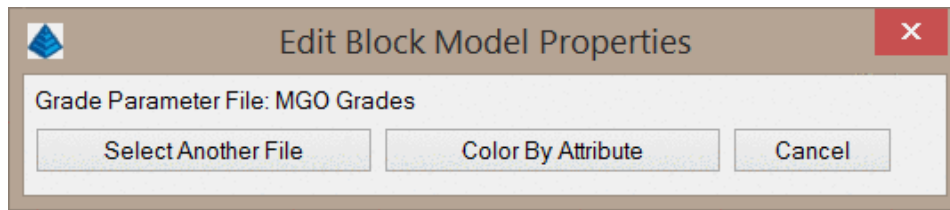
Merge Blocks With Matching Grades: This option combines vertically neighboring blocks with the same grade which speeds up processing and viewing a bit.

After selecting these settings and clicking OK, the 3D Viewer window will be displayed as shown below. All controls for this dialog are discussed in the 3D Viewer Window section of the help manual. However, one additional control is specific to this command. The **Auto Scale** button will modify the Vertical Scale to make block viewing easier. This is especially useful for blocks that cover a large area, but do not span a relatively large depth.



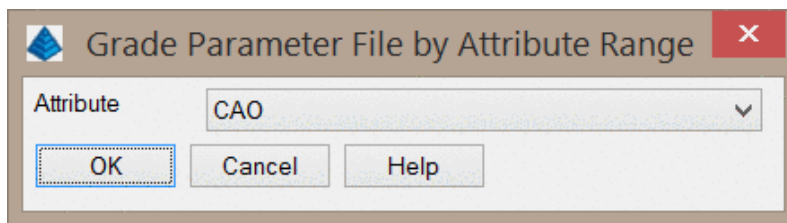
You may view additional block models at the same time by picking the Model tab, right-click on Block Models and pick Add Block Model. Also from the Model tab, you may add surface models or CAD entities to the scene.

You may change the coloring of the blocks by right-clicking "Block Model" on the Model tab. The below dialog will be displayed.

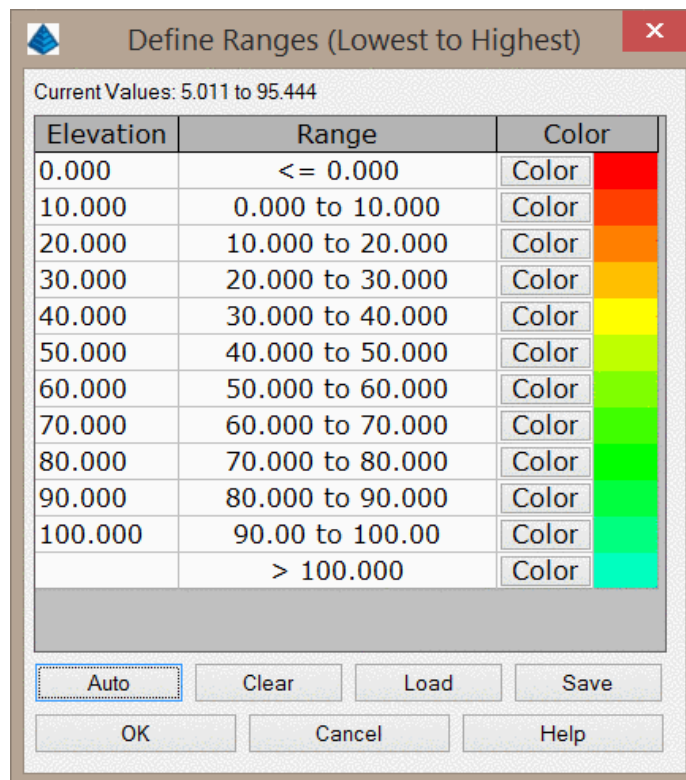


Select Another File: This option will prompt you to select a new Grade Parameter File to recolor the block model.

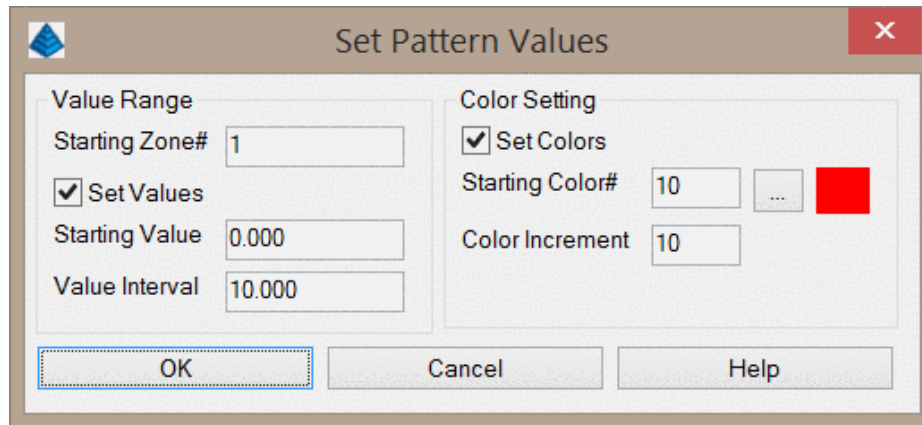
Color by Attribute: This option will allow you to recolor the blocks by defining the attribute values and colors on-the-fly. This is a simplified version of a Grade Parameter File, with the restriction that coloring can only be based on a single attribute. Clicking this option will display the below dialog, which will prompt you to select the attribute of interest.



The next dialog that appears is shown below. Here you can define the range of values and the color for that range.



The **Auto** button can be used to quickly create the ranges using a constant interval. This dialog is shown below.

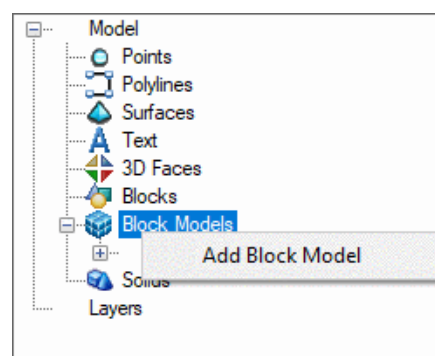


Starting Zone: This value sets the first row to modify. If several rows have already been defined and do not need to be modified, set this value to the next undefined row.

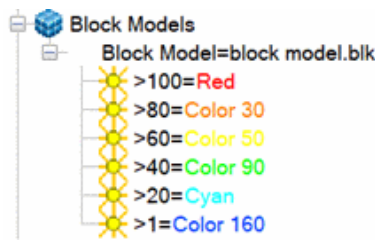
Set Values: This option will modify the values of the rows. If the values have already been set and do not need to be changed, this option should be disabled. When enabled, the Starting Row value will be set to the **Starting Value**. Each row after will increment by the **Value Increment**.

Set Colors: This option will modify the colors of the rows. If the colors have already been set and do not need to be changed, this option should be disabled. When enabled, the Starting Row color will be set to the **Starting Color #**. Each row after will increment by the **Color Increment**. The color values may be viewed by clicking the ellipse button to the right of the **Starting Color**.

It is worth noting that you can load more than one block model into the 3D Viewer. After loading the first block model, you can right-click on the Block Model branch to load a second model.



Grade View Controls: On the Model tab, you can list the block and all the grades. From this list, you can turn grades on/off for viewing by double-clicking on the grade name, or by right-clicking and choosing Thaw Layer or Freeze Layer. Also with right-click, you can set the grade as transparent.



Pulldown Menu Location: Block Model

Keyboard Command: cube_blk

Prerequisite: A BLK model file and a GPF grade parameter file.

Block Model Statistics

This command is used to report data value statistics on each grid or overall statistics within the block model. The user is prompted to select one of the reporting method, Report by Attribute/Quality, Report by Grade Parameters or Report by Levels/Grids. The reports are generated using the Report Formatter. There is an option to select an inclusion perimeter to limit the report to an area. Otherwise the report will cover the whole block model.

The Report By Attribute/Quality reports statistics for all the samples of each attribute in the block model. Here's an example:

Name	Samples	Min Value	Max Value	Avg Value	Variance	Std Dev
SIO2	143820	3.07	71.12	26.88	61.25	7.83
CAO	143820	4.03	53.31	34.78	47.13	6.86

The Report By Levels/Grids reports statistics for each attribute grid at each level in the block model. The attribute names in the report have the level sequence number as a suffix. The grid levels are numbered from 1 at the bottom and going up. Besides the statistics on the attribute values, the report also has statistics on the elevation and depth for each level. Here's an example portion of a report:

Name	Samples	Min Value	Max Value	Average Value	Std Dev	Avg Top Z	Avg Bot Z	Avg Depth
CAO[1]	1410	21.42	43.78	35.67	3.91	47.93	47.43	101.67
CAO[2]	1410	20.96	43.71	35.67	3.99	49.92	48.93	100.17
CAO[3]	1410	20.59	43.82	35.67	4.06	50.92	49.92	99.18
CAO[4]	1410	20.37	43.88	35.68	4.14	51.92	50.92	98.18

For Report By Grade Parameters the program prompts for a Grade Parameter File (gpf) and reports the number of blocks for each grade along with the min, max and average values of the attributes for each grade. With this report, besides using an inclusion perimeter to limit the blocks to report, you can also use a solid model (MDL) or top or bottom surface grids.

Grade	Samples	% of Total	GOLD Min	GOLD Max	GOLD Avg
>100	1	0.0	103.84	103.84	103.84
>80	1	0.0	80.48	80.48	80.48
>60	11	0.0	60.65	76.21	67.33
>40	89	0.2	40.26	59.21	48.11
>20	390	1.0	20.02	39.90	27.15

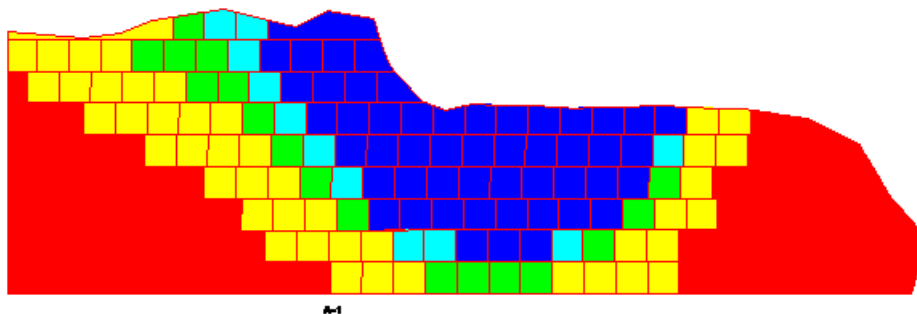
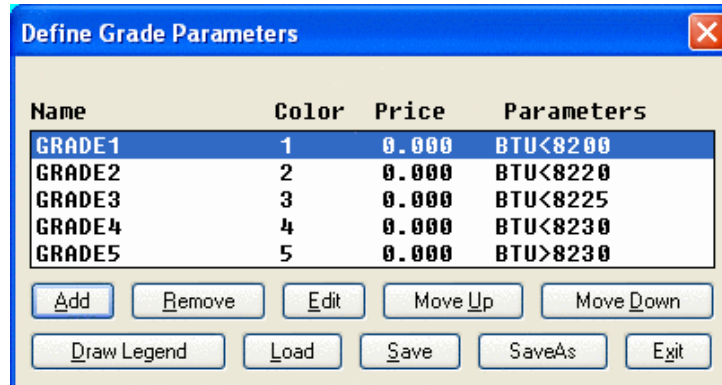
Pulldown Menu Location: Block Model

Keyboard Command: blkstats

Prerequisite: .BLK model file

Color Pits by Grade Parameters

This command colors pits by assigned grade color from grade parameters file. It could be useful in identifying the pits with different grades/qualities.



Prompts

Color Pits by Grade Parameters

Select all directioned pit polylines.

Select objects: *all*

202 found

Select objects: *press Enter*

Enter bench number to hatch (1-4): *4, enter bench number to be colored*

Pull-down Menu Location: Block Model

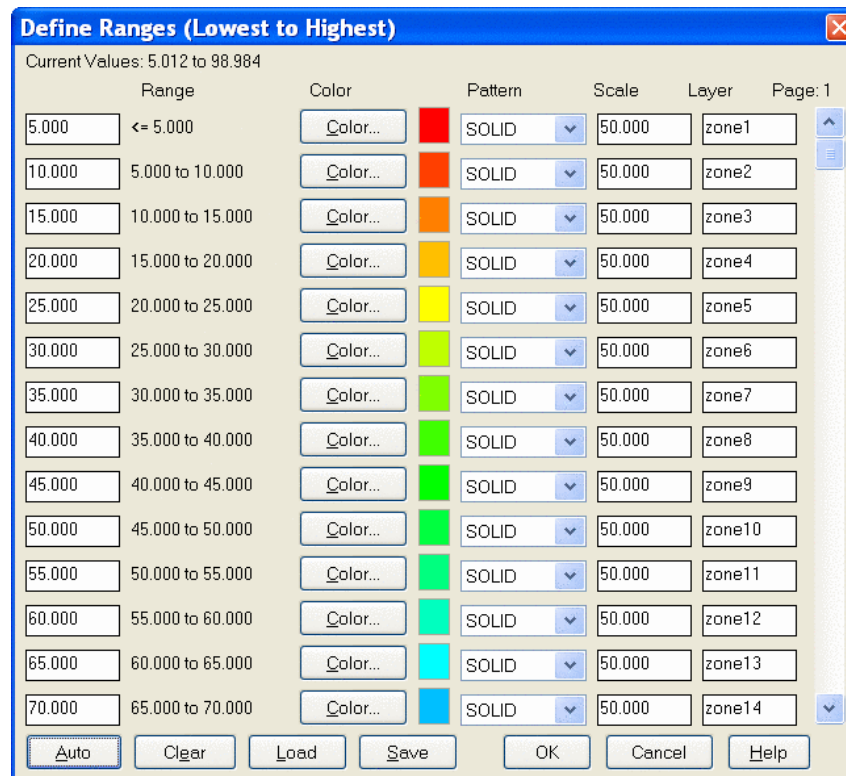
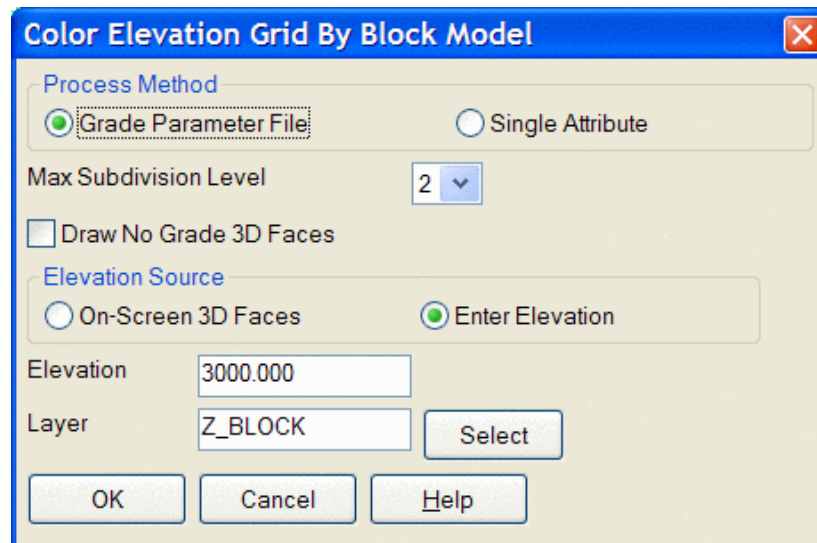
Keyboard Command: pitcolor

Prerequisite: Grade Parameter File (gpf) and Surface Pits with directions and quantities

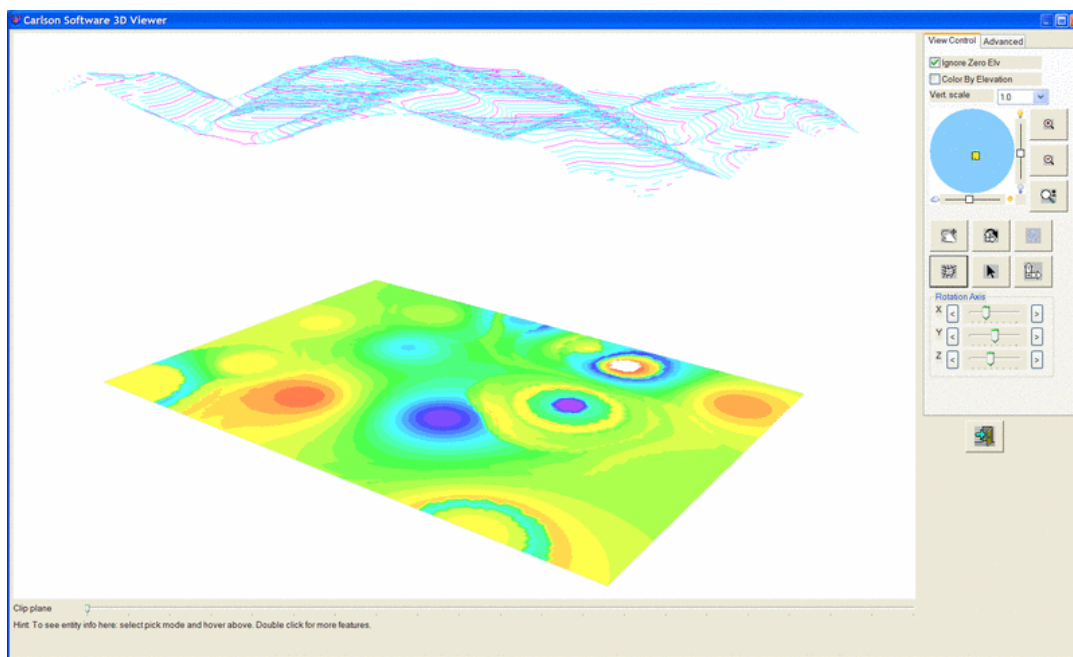
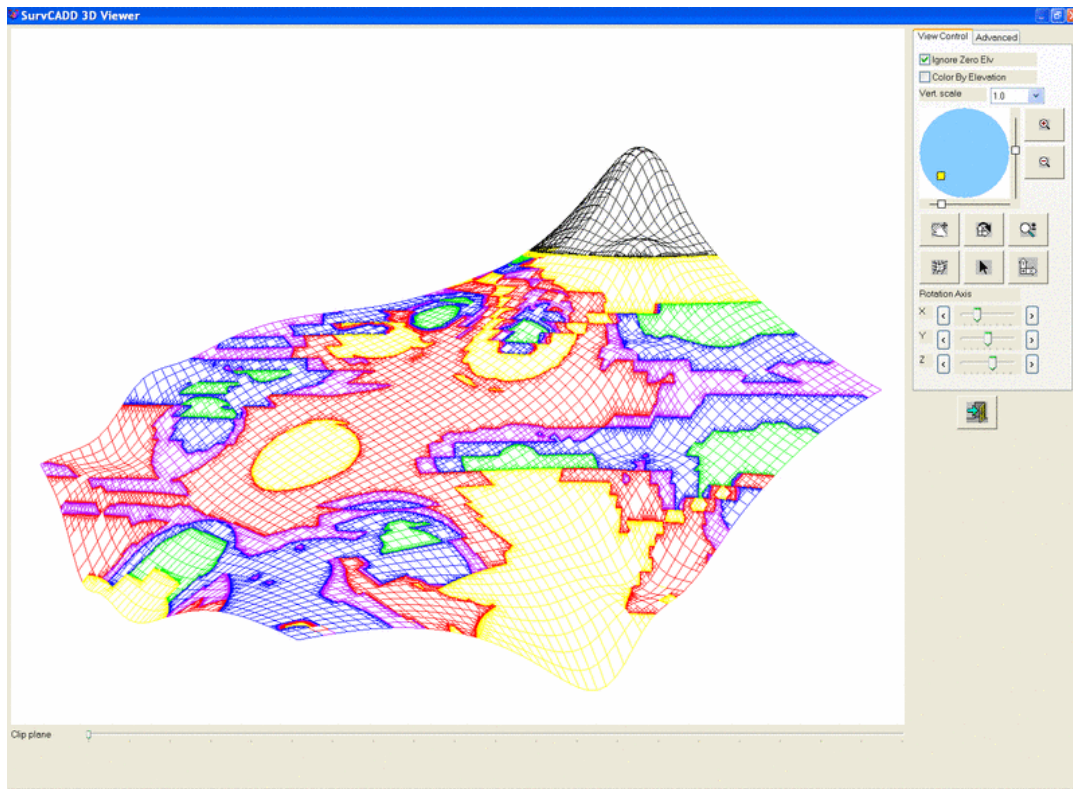
Color Elev Grid by Block Model

This command will color the 3D faces of an elevation grid plotted on screen or of a flat user defined elevation based on the Grade Parameter File or by a single attribute, where the color palette is selected separately.

After selecting the Block model file (*.BLK), the following dialog box prompts for Single or File for the color scheme. Selecting File would use the predefined *.GPF file. Choosing Single leads to the next colorful dialog box to specify the ranges. The Max Subdivision Level allows for a tolerance to subdivide the cells or blocks. Zero (0) will not subdivide them. A maximum of 4 will subdivide them into very small segments. The option to Draw No Grade 3D Faces will leave gaps where the parameters of the block model don't meet any of the grades. There is the option to use On-Screen 3D Faces or to simply enter an elevation to represent the grid.



The result is a colored grid mesh or elevation that can be rendered and viewed in 3D.



Prompts

Reading cell > 4828

Select 3D Faces to Process... type all or window the faces

Select objects: Specify opposite corner: 15779 found, 11552 were filtered out.

Select objects: Enter to accept

Keyboard Command: BLKCOLOR

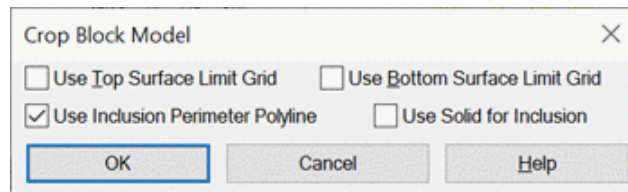
Pull-down Menu Location: Ore

Prerequisite: Need a *.BLK model file and a grid drawn on screen.

Crop Block Model

This command creates a new block model that is cropped from an input block model. When a block model is larger than needed, this command can be used to make a smaller block model that is faster to process and easier to manage.

After selecting the input and output block models, there is a dialog for selecting the methods to crop. The Use Top Surface Limit Grid applies a grid surface to remove all block cells above this grid. Likewise, the Use Bottom Strata Limit Grid removes all block cells below the grid surface. The Use Inclusion Perimeter Polyline removes block cells that are outside the closed polyline perimeter. The Use Solid for Inclusion removes block cells that are outside the solid model.



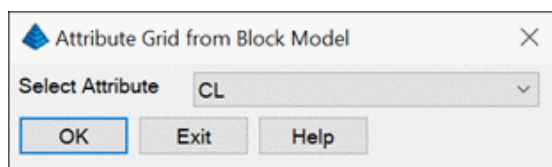
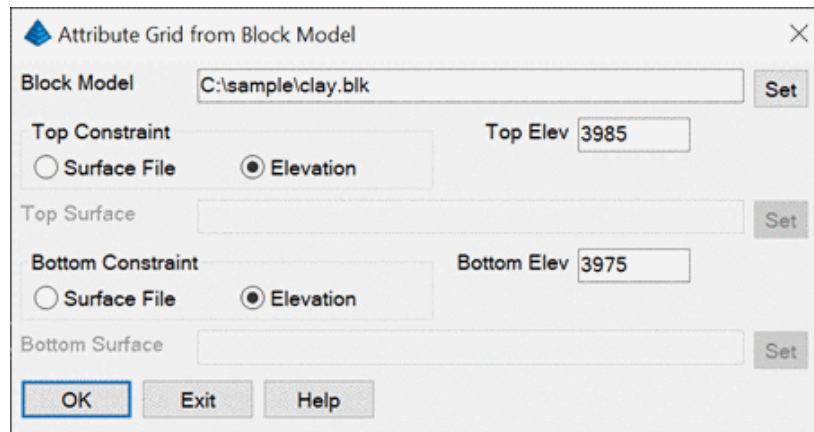
Keyboard Command: crop_blk

Pull-down Menu Location: Block Model

Prerequisite: Block model (blk)

Attribute Grid from Block Model

This command creates strata attribute grids by carving out the attribute data from a block model. The strata attribute grid values are calculated from block model values between a bottom and top constraints. These constraints can be defined either by a grid surface file or a fixed elevation. After selecting the block model to process and the constraints, there is a dialog to choose the attribute to extract and the file name for the attribute grid to create.



Keyboard Command: blk_attr_grd

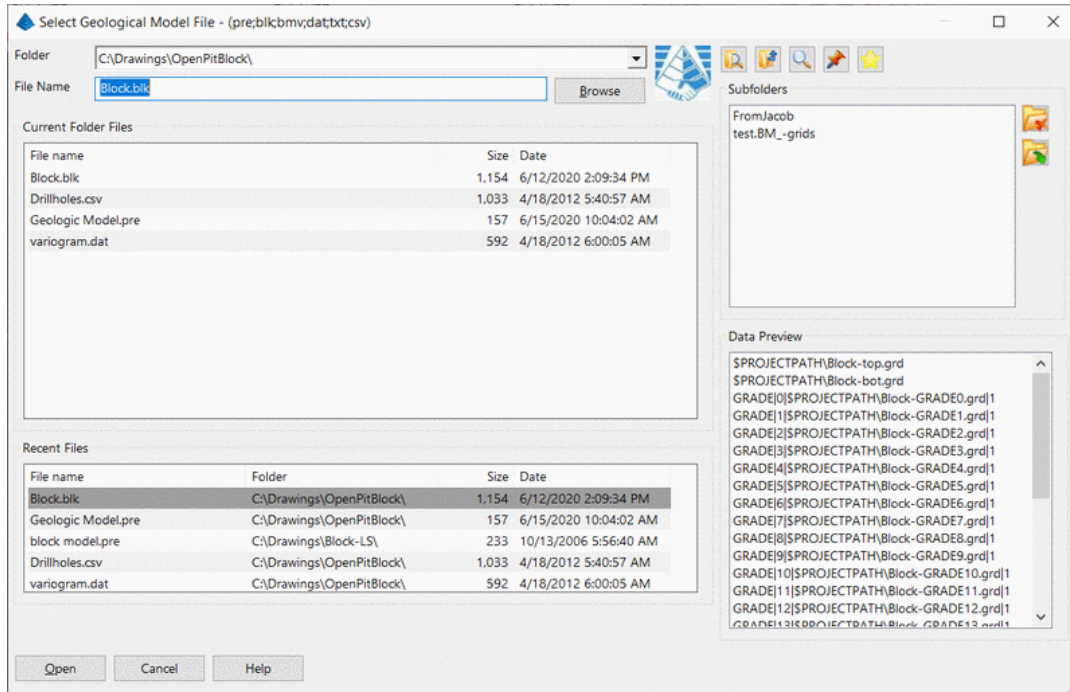
Pull-down Menu Location: Block Model

Prerequisite: Block model (blk)

Prepare Value Block Model

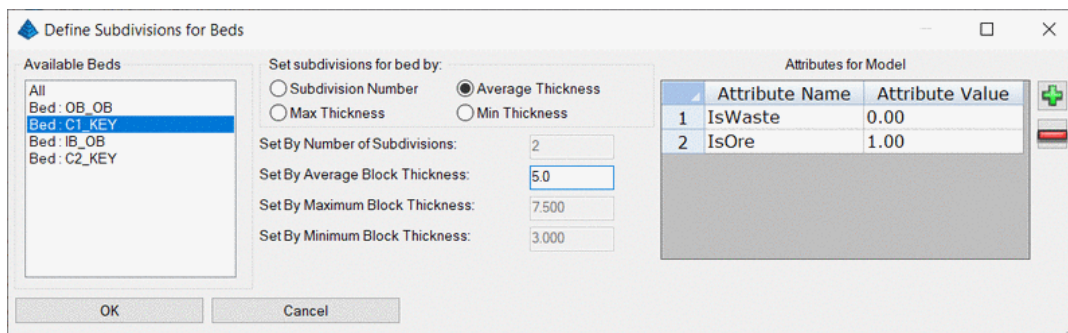
This command creates a block model that represents the economic value of each block. The value block model can be generated from multiple .blk files, a Geologic Model (.pre file), or from an existing .BMV file. The blocks loaded into this command can be modified with user-defined equations to determine the value of each block. The value block model may then be used with the Optimized Pit Design command.

When the command is started, the following dialog will appear.



This command will recognize and use both BLK and PRE files, as well as existing BMV files. To load in .blk files, there should be no gaps between the elevation limits of these block models. The ordering of these files is not important, as they will be sorted automatically according to the elevation limits of each .blk file.

To use a grid based stratified geologic model saved as a .pre file to generate the blocks, select the .PRE file to use. This option will convert a stratified, grid based Geologic Model into a single BMV block model by specifying a number of vertical divisions to apply to each strata/bed. This is basically making a BLK file of each strata/bed in the PRE file. Each seam subdivisions should be set depending on their thickness. This is what determines the block height for each seam. When this option is selected, and the .pre file is chosen, the following dialog will appear.



Available Beds: This column will list all of the strata/beds in the .pre file that are not already defined with a .blk file. Selecting one of these beds allows for defining the subdivisions for that bed. Alternatively, the subdivisions for all beds may be set at once with ALL.

Set Subdivisions for bed by: This option controls how each bed will be vertically subdivided to create a block model. Hit the ENTER button after entering any value, to update the field.

Subdivision Number: This option will subdivide the bed by the number specified. For example, if this value is set to 2, then the resulting block model of the bed will consist of 3 blocks per vertical column. Note that this value must be greater than or equal to 2.

Set By Average Block Thickness: This option will set the number of subdivisions in order to achieve the specified average block thickness. For example, if this value is set to 10, the bed will be subdivided so that the average block thickness of the resulting model will be as close to 10 as possible.

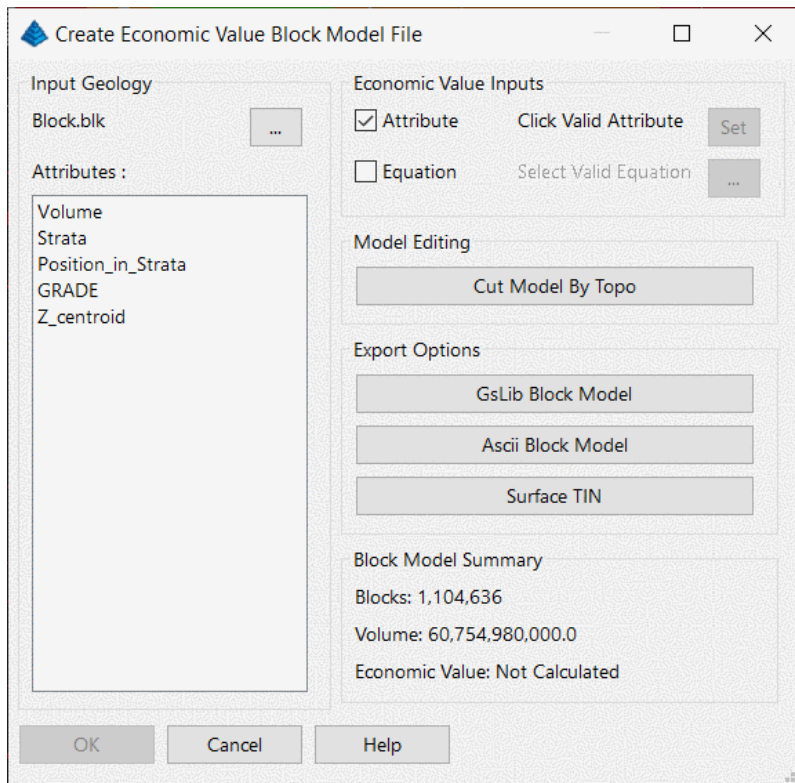
Set By Maximum Block Thickness: This option will set the number of subdivisions in order to achieve the specified maximum block thickness. For example, if this value is set to 10, then the bed will be subdivided so that no blocks in the resulting model have a thickness greater than 10.

Set By Minimum Block Thickness: This option will set the number of subdivisions in order to achieve the specified minimum block thickness. For example, if this value is set to 10, then the bed will be subdivided so that no blocks in the resulting model have a thickness less than 10.

Attributes for Model: This spreadsheet defines new attributes to add to each block in the model, for later use in equations. These attributes can be set to single values, but can be modified when calculating the value of each block later in the command. At least one attribute must be defined for each bed.

Selecting an existing .BMV file. Note that .BMV files are exported from this command, so this option will reload one of these and do additional calculation.

After selecting one of these options, the below dialog will appear.



Any attributes from the input files will be listed on the left side of the dialog. These attributes may be used when writing equations to define the value of each block. Note that when these attributes are referenced, it refers to the value of each block individually - these are not average values for the block model as a whole. In addition to the attributes from input, four additional attributes will always be generated: Z_centroid, Volume, Strata, and Position in Strata.

The Z_centroid attribute is the elevation of each block's centroid. This attribute is useful to calculate the increased mining cost at lower elevations or depths.

The Volume attribute is the volume of each block, as measured in cubic feet or cubic meters.

The Strata attribute is the strata index number. A value of 0 relates to the upper-most strata in the model, a value of 1 relates to the strata below strata 0, etc. If there is just one block model BLK file, then there will be just the one strata index of 0.

The Position in Strata attribute is the block number counted up from the bottom of the model. For example, the lowest block in the bed will be block 1, the next block up will be block 2, etc.

Economic Value Inputs:

Attribute: Click Valid Attribute will set the currently selected/highlighted attribute as the economic value for each block in the value block model. This is an alternative to calculating the block values from a set of equations. Note that this does not modify the input files - you will be prompted to save the updated model as a .BM file after clicking OK.

Define Equations: This option brings up the equation editor (.db3 file). Additional information about this function is provided below.

Model Editing:

Cut Model by Topo: Selecting this option will bring up a dialog to select a grid file. There will be an option to remove blocks found above/below this elevation

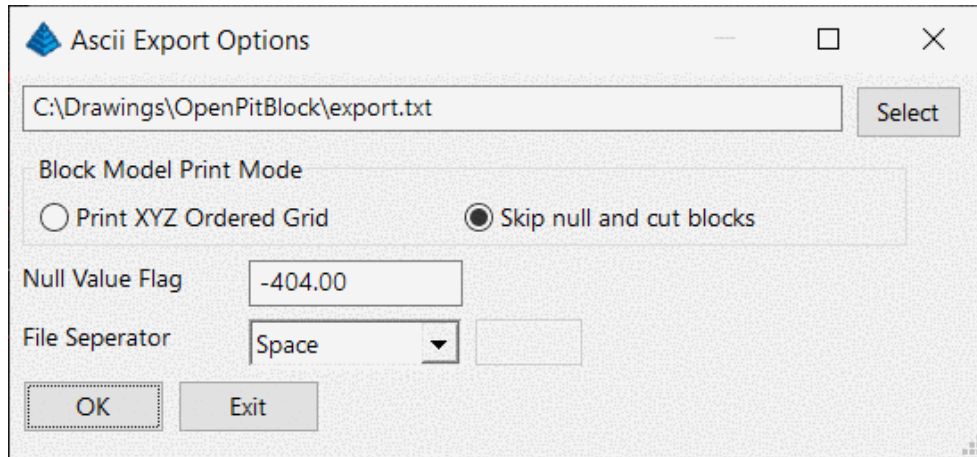


grid.

Export Options:

GsLib File: This button will export the value block model to a .txt file in a GsLib format. Details may be found at www.gslib.com.

ASCII Block Model: This option will output a text file of the block model in xyz format as seen here.



Surface TIN: This button will create a .tin file of the surface of the value block model. Note that this surface will have a block-like appearance and will therefore differ from the original upper limit of the input .blk files.

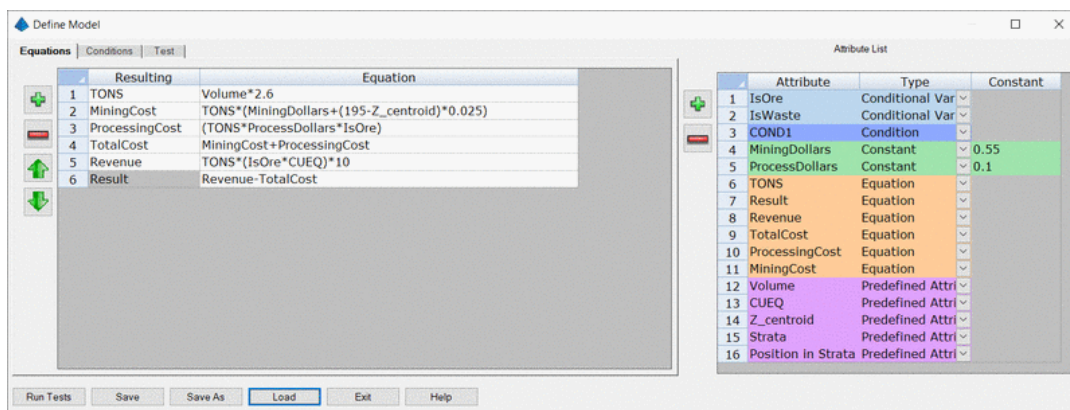
Block Model Summary: These values display basic information about the value block model.

Total Blocks: The total number of blocks in the value block model. Null blocks are not considered in this counter.

Total Volume: The sum of the volume of all blocks in the model.

Total Economic Value: The sum of the economic value of all blocks in the model. This value will not be populated until the equations have been applied to the blocks or until an attribute has been set as the block economic value.

After clicking the Define Equations button, the following dialog will appear. After filling out the tabs with the desired design parameters, save this table to a .db3 file. Choose Exit to return to the previous dialog to calculate the value block model.



The dialog will first appear with only one Equation and Predefined attributes. All other equations, constants, conditions, etc. must be manually added.

Attribute List

The right side of the dialog lists all attributes. Add or remove attributes with the two +/- icons shown to the left of the list. Attributes may be one of five types, with each type showing with a unique color for visual organization.

1. Predefined attributes are attributes that have been found in the input .blk file(s).
2. Equations are attributes that will be calculated when the value block model is created.
3. Constants are attributes that have a non-changing value.
4. Conditions are logical tests that determine which variables to use. For example, a condition may be defined that sets the processing cost for a block based on grade.
5. Conditional Variables are used as a result when Conditions are checked. For example, when setting the processing cost of a block based on grade, the processing cost is the Conditional Variable.

When adding an attribute, the below dialog will appear. This is to assign the attribute a name and set the attribute type, as well as any other relevant options.

Attribute Name	Set Attribute Type	Equation
TONS	Predefined Attribute Equation Constant Condition Conditional Variable	Volume*2.6 Constant Value Condition Expression

Attributes may be renamed by clicking inside the cell of interest and typing a new name. When the name of an attribute is modified, the change will be reflected in the Equations and Conditions tabs on the left side of the dialog. The condition type may be changed by selecting an option from the drop-down menu in the Type column.

Equations Tab

The Equations tab will display all defined equations in a spreadsheet view.

The Resulting Attribute column lists the attribute being calculated by the Equation column. For example, if there is a Resulting Attribute named "Tons", the result of the equation column will be assigned to this "Tons" attribute.

The Equation column is used to write the equation for the attribute being calculated. For example, to calculate the "Tons" for each block, the equation could simply read "Volume*Density". This can be read as "Tons equals Volume multiplied by Density", where Volume and Density are other defined attributes. Equations will be calculated in from top to bottom by row, and therefore equations may reference the results from previously calculated equations. For example, the equation on the row 2 of the equation spreadsheet may reference the result from row 1, but it cannot reference the result from row 3.

The Output to File column controls if the attribute is output to the resulting .bm file. By default, attributes will not be saved to the .bm file. If the .bm file is to be used with the Optimized Pit Design command, only the economic value of each block should be output to the .bm file.

One equation, "Result", will always be defined and output to the .bm file. This equation should always define the value of each block and should always be output to the .bm file.

The below operations and the corresponding symbol are supported in the equation column:

1. Addition "+"
2. Subtraction "-"
3. Multiplication "*"
4. Division "/"
5. Grouping of terms "(" and ")"

Conditions Tab

The conditions tab will display all conditions in a spreadsheet view as shown below.

Condition Name	Condition Expression	True	False	Evaluate Before
1 COND1	Fe>20	COND2 IsWash=,IsOre= None IsWash=0,IsOre=0	None IsWash=0,IsOre=0	Result,Revenue,Process
2 COND2	Al>2.5	None IsOre=,IsWaste= None IsOre=0,IsWaste=0	None IsOre=0,IsWaste=0	Result,Revenue

Condition Name column lists the conditions in the order they were created. These cells are not editable.

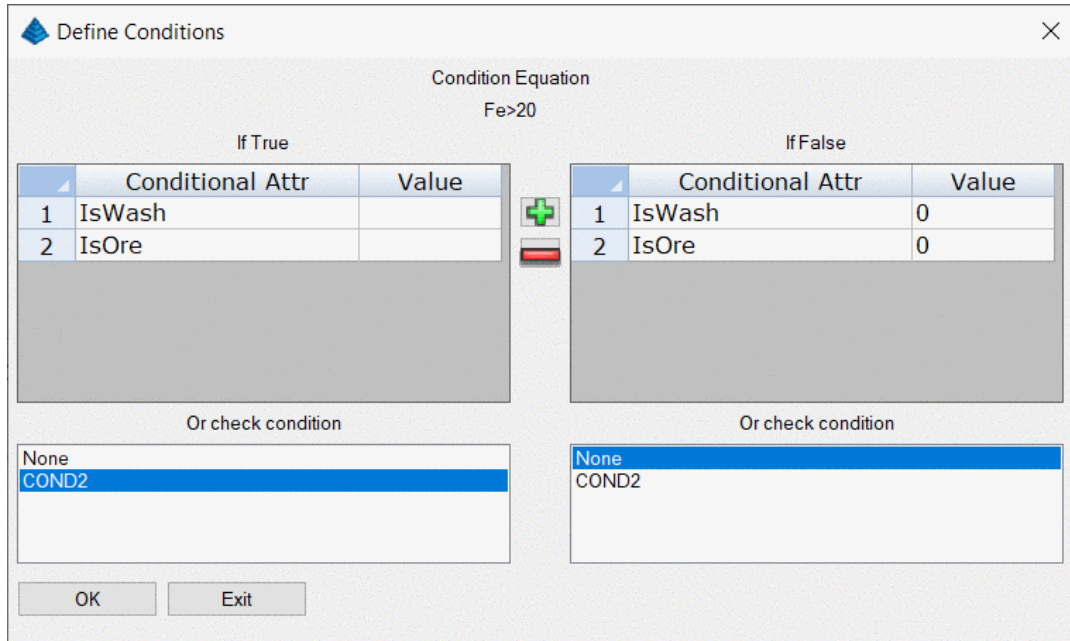
Condition Expression column is the logical test. In the above example, the first logical test checks if the block has an Iron content greater than 20%.

True column lists the actions to take if the logical test is found to be true. At least two actions will be shown, separated by a vertical pipe. The first action tells the program which condition to evaluate next. The second action actually sets values to the Conditional Variables. In the above example, if a block is found to have an Iron content greater than 20%, the program will check Condition #2, but no Conditional Variable values will be assigned based

on this test alone. If Condition #2 is found to be true, the program will not check another condition, but will assign the attributes "IsWash" and "IsOre" values of 1. This is similar to a nested "if" statement.

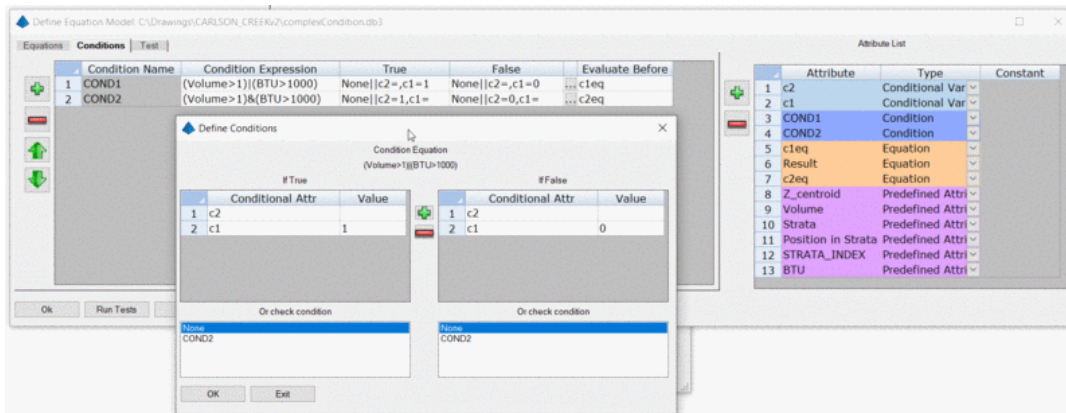
False column lists the action to take if the condition is found to be false. This follows the same format as the True column. In the above example, if a block is found to have an iron content less than or equal to 20%, then no other condition is evaluated and the "IsWash" and "IsOre" conditional variables are assigned a value of 0.

Clicking inside a cell of the True or False columns, or selecting the button next to them will bring up the following dialog.



This dialog sets values for the Conditional Variables based on the logic of the Condition. The two tables list Conditional Variables that will be affected by this Condition (IsWash and IsOre in this case). If the program needs to check another Condition rather than assign values to Conditional Variables as a result of this test, then select the appropriate Condition to check below this Table. Conditional Variables may be added/removed from this list using the appropriate icons between the two tables.

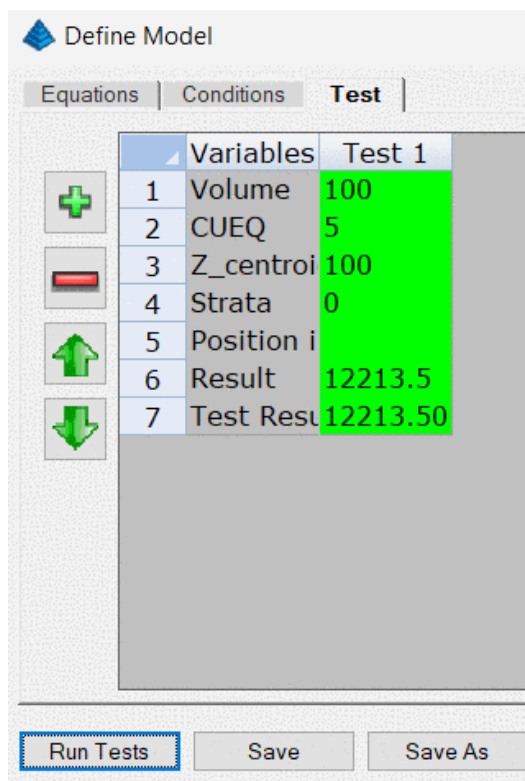
Evaluate Before column lists the equations in which the conditions need to be evaluated before. In the above example, the conditions will be evaluated before the Revenue and Result and Processing Cost equations are calculated. Here is an example of more advanced conditions separated by "and" & and "or" — in the Condition Expression field:

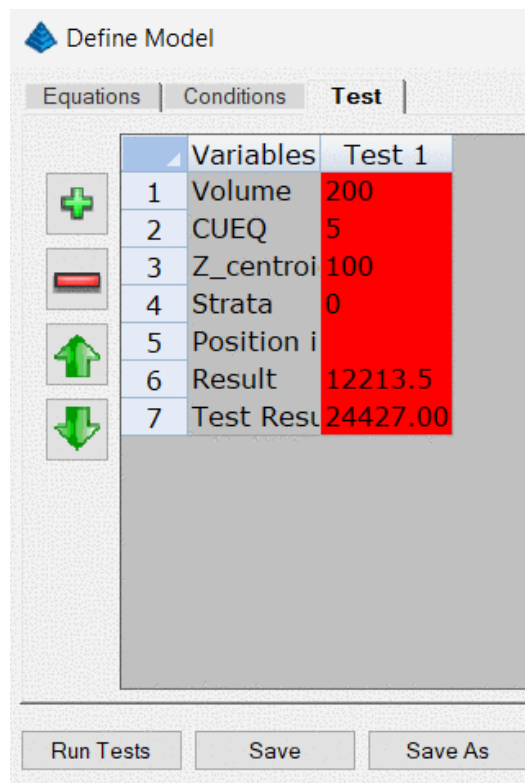


Test Tab

The Test tab displays all defined attributes in a simple spreadsheet. The purpose of this tab is to define example values for each attribute, along with an expected result. Enter in example values and the calculated result, to see if the test of the equations calculates the same result. After defining one or more tests and saving the .db3 file, then click the **Run Tests** button to check the actual RESULT output against the expected, user entered output RESULT. If the Test Result matches the expected result, the cells will be highlighted in green, as shown below. If the values do not match, the cells will be highlighted in red.

Result Passes the Test Result Fails the Test



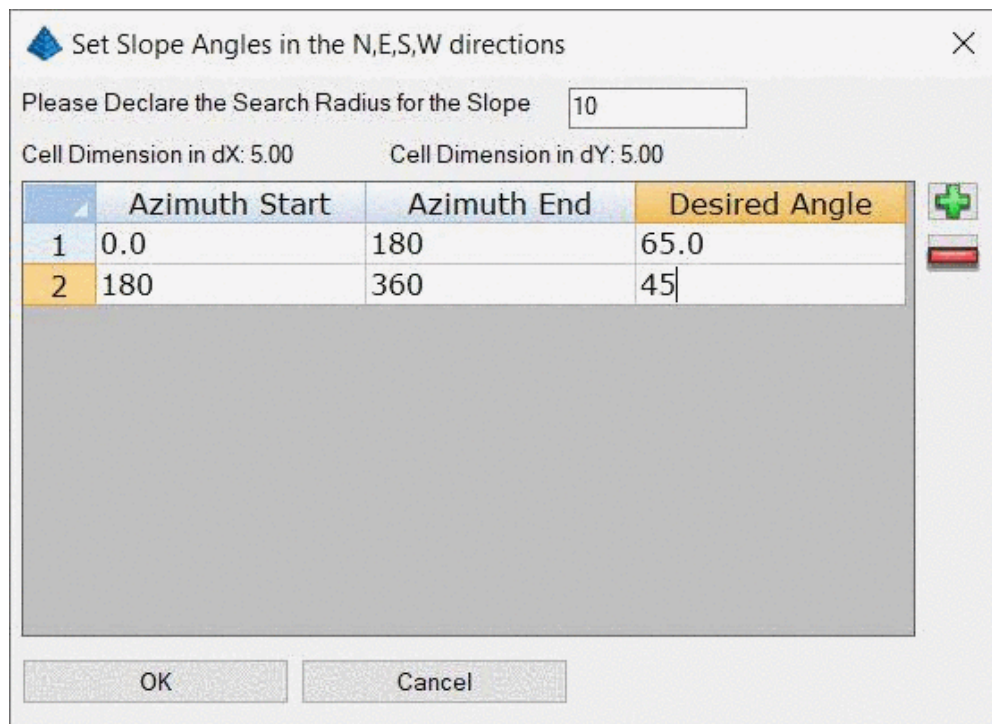


Keyboard Command: mkvalblk **Pull-down Menu Location:** Geology Module > Block Model

Prerequisite: Must have an existing BLK file

Optimized Pit Design

This command uses a max flow algorithm to determine the ultimate mineable economic pit. This is an automated routine with very few prompts. When the command is started, the first prompt is for a value block model (*.BM file). This file is created from the Prepare Value Block Model command which must be run before the Optimized Pit Design to create this file. After selecting the value block model file, the first dialog will then appear to set the slope angles on the various sides of the ultimate pit.



Azimuth and Desired Angle: The Azimuths are defined with 0 to the North, 90 to the East and 180 to the South. The desired slope angles set in the dialog are measured in degrees from the horizontal direction.

After setting these angles, and selecting OK, the program will create a .TIN file of the optimum pit. The surface uses the blocks in full to create the slopes, which can look like benches if the blocks are made at the proper size in the original block model. This TIN file can then be viewed in 3D, contoured, or used to create linework for other pit design routines, such as Design Bench Pit.

The command line displays text after the command stating the monetary value and the volume of the total blocks.

Command: bestpit

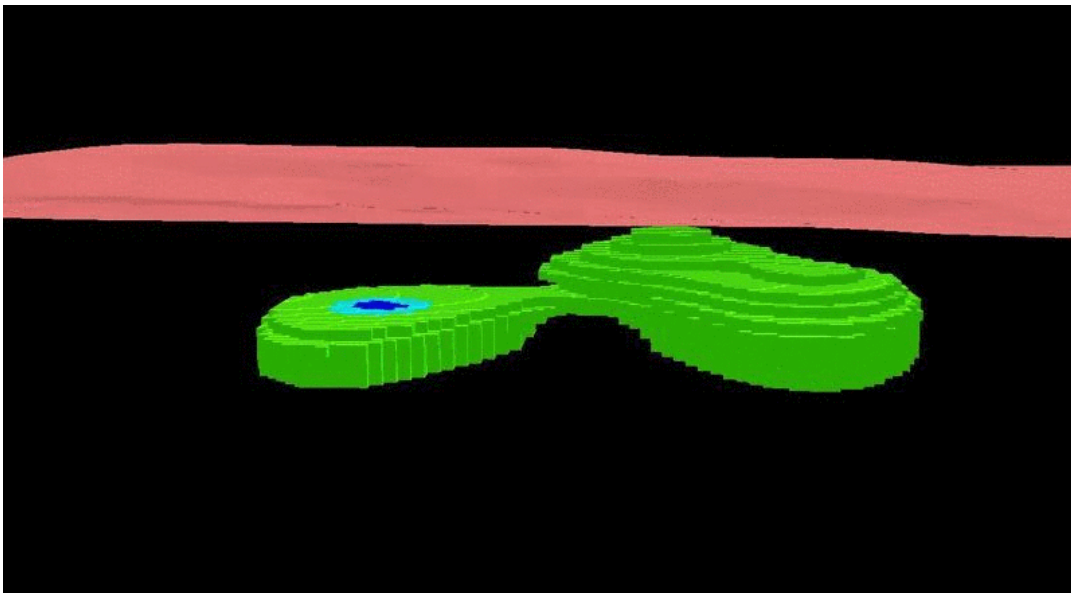
The total size is 135x78x62 blocks.

Inserting points...

Inserted 42,120 points

FINISHED with total monetary value of 640,326.91, and a total volume of 410,250.0 cubic units from 3300 Total Blocks

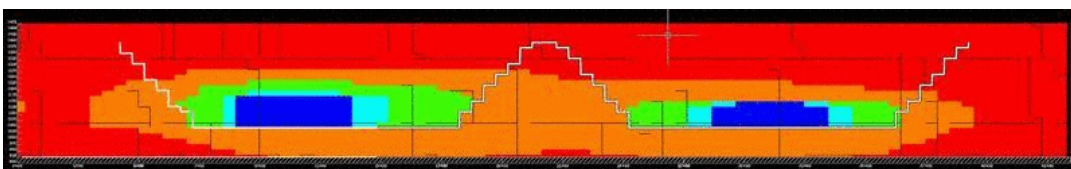
The below image shows a simple block model with the topography file.



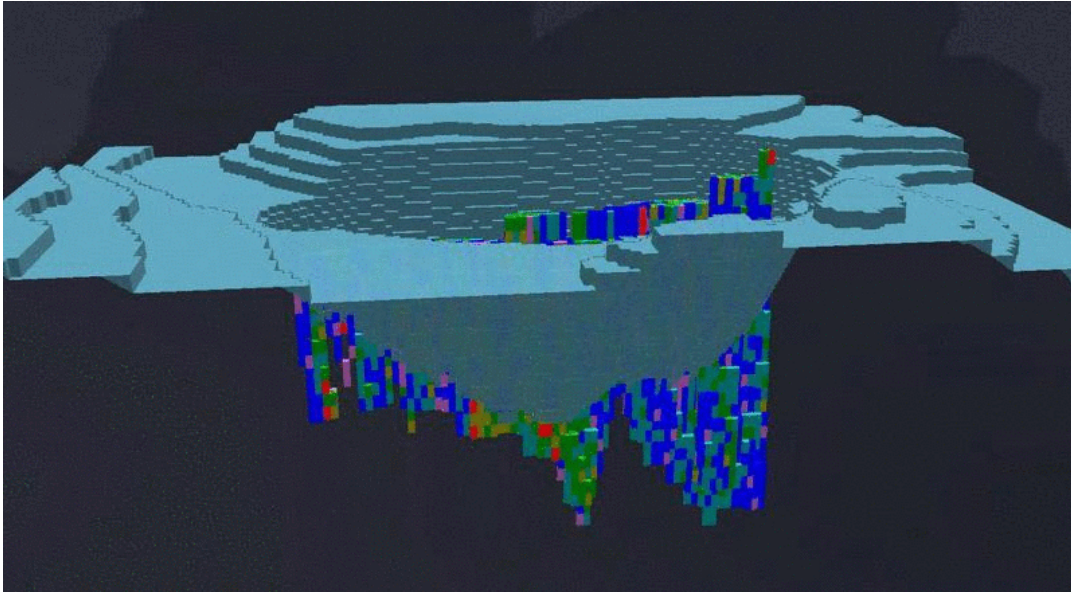
The below image shows the same block model with the optimized pit shell .tin file. The TIN file is clipped in the front, to see into the blocks better.



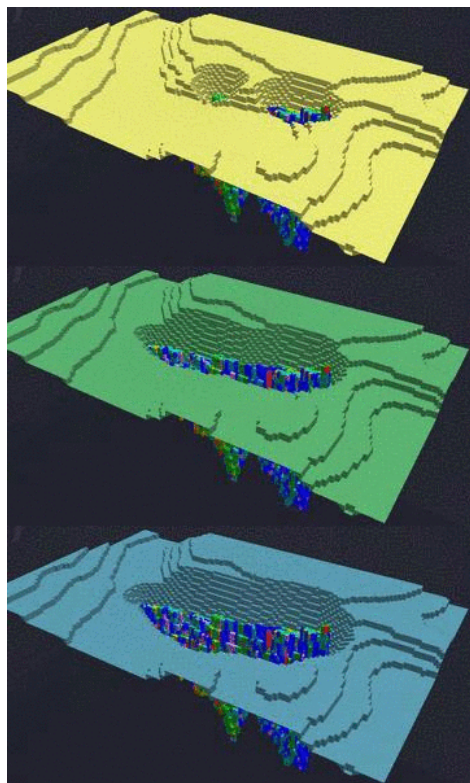
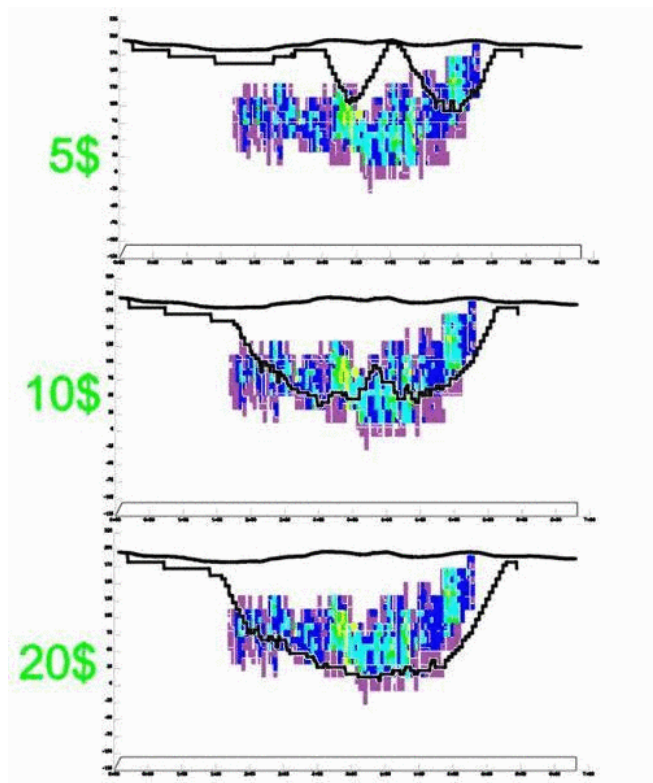
Here is an image of the same block model on a Fence Diagram, with the optimum pit design surfaced drawn as an additional surface. This illustrates the surface combined with the ore, showing the optimum pit in relation to the ore. In this case, the ore is represented by the green and blue blocks, while red and orange is waste.



This image shows a transparent final pit surface in 3D on top of the block model used to calculate it. The irregular appearance of the top of this surface represents the top of the block model itself.

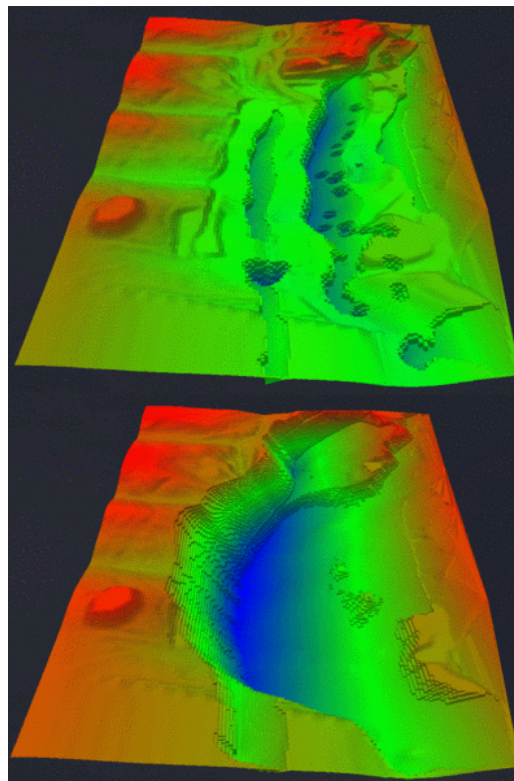
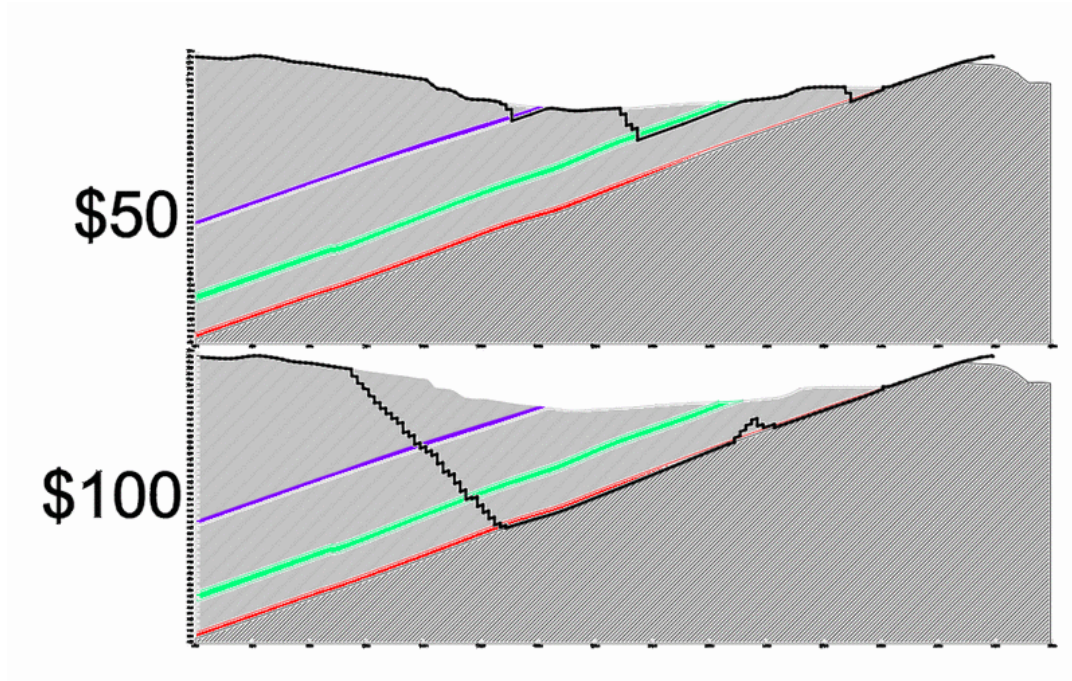


Shown here is another example of multiple optimum pit design TIN files shown with a block model. A fence diagram was also drawn to visualize the pit depth for each design. This was run 3 times, with three different revenue calculations to see the result of the pit at various currency amounts. This command is very easy to run "what if" scenarios with different ore market values to determine the pit design.



The above examples are shown with regular block models. This command can also be used to find the pit design for stratified deposits, such as this dipping coal seam model. The PRE file was used as the input files for Prepare Value Block Model. This will show how deep to follow the dipping seams and still remain profitable. Different revenue amounts may be entered to create different designs showing the mining depth based on varying revenue of the ore. Shown here are geological cross section Fence Diagrams, with the additional surface of the Optimum Pit drawn on top to see the depth to mine to. To the right are the associated pit designs, based on two

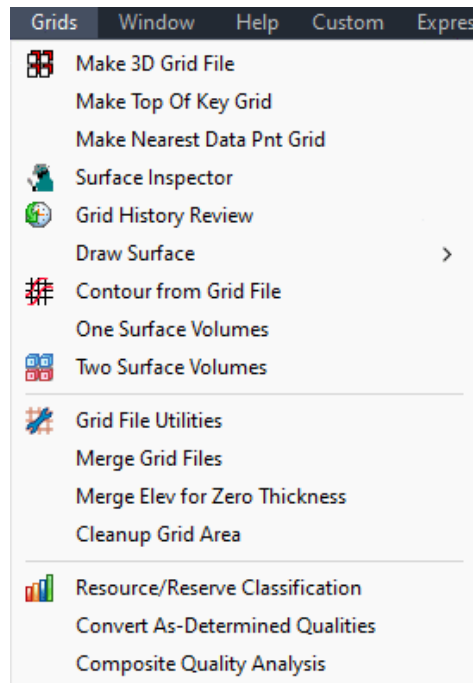
dollar amounts for the coal/ore.



Pulldown Menu Location: Block Model
Keyboard Command: bestpit
Prerequisite: Value Block Model (.BM file)

Grids Menu

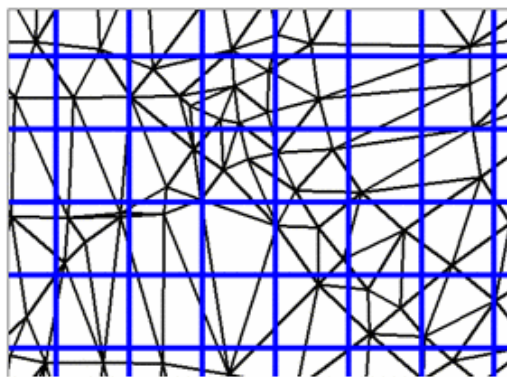
The Grids menu has commands for creating, editing, and visualizing grid files.



Make 3D Grid File

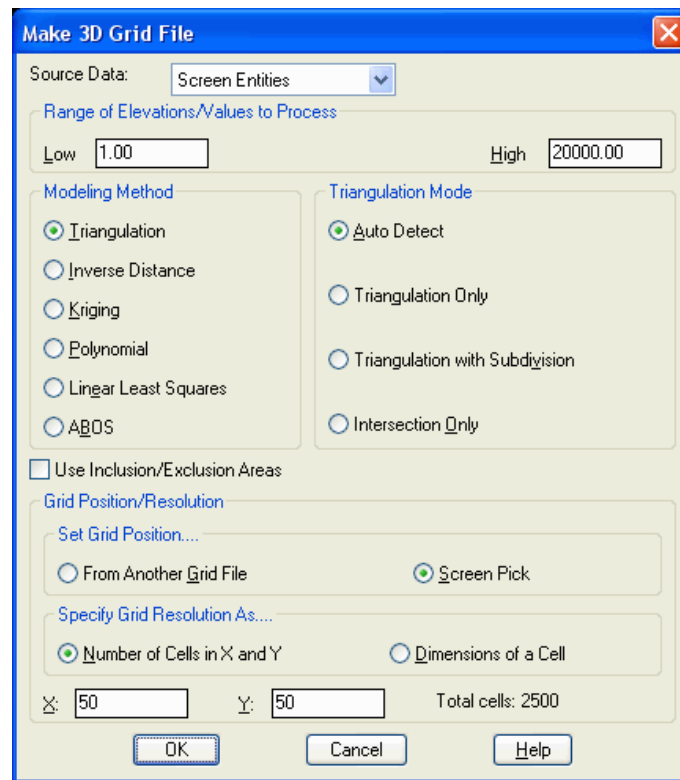
This command creates a grid (.GRD) file which serves as a surface model for use in many of the other Surface routines. The program internally makes a triangular network of the data points (if *Triangulation* is selected as the modeling method) and then interpolates the elevation values of a rectangular grid at the specified grid resolution. Data points can be either points, inserts, lines, or polylines. Lines and polylines are treated as breaklines in the triangulation.

Gridding as a means of modeling surface features is generally less favorable than triangulating as the surface is defined only at the intersection of the grid lines. This can lead to inaccuracies around local features such as ditches or curb lines, since the grid resolution must be small enough to adequately capture the changes in these local regions. Contrast this with Triangulated Networks which carry all this information at every point along the features. Gridding can, however, be useful for modeling large sites in general trends such as watershed analyses and large-scale volume computations.



Grid superimposed over triangulated features

The grid location is specified by first picking a lower left corner and then an upper right corner. The screen cannot be twisted when this is done because grids always run north-south and east-west.



The dialog box sets the range of elevations to process, modeling method and grid resolution. Each of these items is described below.

- **Source Data:** This option selects the type of data to use for gridding. The Screen Entities option processes selected 3D entities from the drawing including points, lines, polylines, 3D faces and inserts. The Coordinate File and Text File options read point data from the selected file. These methods are useful for large datasets that would take extra memory and time to draw as points in the drawing. For the Text File, the program will prompt for the order of the fields and the delimiter. The Triangulation File option will interpolate the grid elevations from the selected triangulation surface.
- **Range of Elevations/Values to Process:** Entities with elevations or values outside the range to process are ignored and will not be used for the gridding.
- **Modeling Method:** The modeling method almost always should be triangulation for surface topographic grid files. Polynomial, inverse distance, kriging and linear least squares apply to random data points for surfaces like underground features, usually sourced by such methods as drillholes, data tables, etc.
- **Triangulation Mode:** When using Triangulation and Polynomial methods, There are four triangulation modes: AutoDetect, Triangulation Only, Intersection with Triangulation and Intersection Only.
 - **Auto Detect** method automatically chooses between the Triangulation Only and Intersection with Triangulation methods. If the selected surface entities are primarily made of polylines, then the Intersection with Triangulation method is used. Otherwise the Triangulation Only method is used.
 - **Triangulation Only** method builds a triangulation surface out of all the selected points, lines and polylines. All lines and polylines are treated as breaklines. Grid node elevations are calculated based on the triangulation.
 - **Triangulation with Subdivision** method uses the subdivisional surfaces modeling method. This option causes each triangle in the triangulation surface model to be subdivided into an average of three smaller triangles per subdivision generation. This gives a much smoother surface model, where instead of one

triangle, there are now three or more.

- **Intersection Only** method goes directly to the Steepest Intersection method using the selected lines and polylines. The Steepest Intersection method is used to assign the grid node elevations from the linework of the triangulation lines and the selected lines and polylines. The triangulation step is skipped and any selected point data is not used. This method can be used for making grids out of polylines such as a contour map as long as the surface is defined just by contour polylines without needing spot elevation points. Skipping the triangulation step makes this method a lot faster especially for large files.
- **Use Inclusion/Exclusion Areas:** This option will prompt for inclusion and/or exclusion perimeter polylines and will only assign grid cell elevations within these areas and leave the rest of the grid cells as Null.
- **Grid Resolution:** The grid resolution is specified by either the number of grid cells or by the size for each grid cell. It is usually best to set the Dimensions of a Cell to a known size, and the program will calculate the "number of cells in X and Y." While the program can handle really large grids with no limit, a general rule of thumb is to keep the total number of grids cells under 500,000 (about 700 by 700 cells) to limit the processing time. The grid location and resolution can also be specified by using the position/resolution from an existing grid file. In this case, the location and resolution of the new grid will match those of the selected grid file which is useful for routines that require two grid files with identical locations and resolutions.

No elevations are calculated on grid cells that extend beyond the extent of the data. The figure shows an example of how the grid is calculated to the limits of the data points. Extrapolation can be used to calculate elevations for the grid cells that are beyond the data limits. When there are grid cells with no elevation in a grid (.GRD) file, many routines will prompt *Extrapolate grid to full grid size?* Extrapolation fills in all the grid cells. The method to extrapolate uses a safe calculation that tends to average out or level the extrapolated values. So extrapolated grid areas are not as accurate as grid areas within the limits of the data. *Grid File Utilities* can be used to apply and save extrapolation to a grid file. The *Plot 3D Grid* command can then draw the grid file so that you can see the extrapolation.

A Carlson grid (.GRD) file has the following format:

Line 1 is the lower left Y coordinate
Line 2 is the lower left X coordinate
Line 3 is the upper right Y coordinate
Line 4 is the upper right X coordinate
Line 5 is the X direction grid resolution
Line 6 is the Y direction grid resolution

The rest of the lines are the Z values of the grid intersects starting from the lower left moving in the left to right direction and ending at the upper right. If the intersect has no value, the letter 'N' is saved instead of the Z value for Null values. An example is shown in the Display-Edit Report dialog.

Cell#	East (x)	North (y)	Elev (
18	4436.21	4639.60	92.58
19	4490.26	4639.60	94.46
20	4544.32	4639.60	94.80
21	4598.38	4639.60	94.62
22	4652.43	4639.60	92.48
69	4436.21	4688.90	92.79
70	4490.26	4688.90	95.29
71	4544.32	4688.90	95.95
72	4598.38	4688.90	96.42
73	4652.43	4688.90	95.46
74	4706.49	4688.90	94.25
119	4382.15	4738.19	89.69
120	4436.21	4738.19	93.14

Gridding from Contour Maps

A grid file can be created from contours represented as polylines with elevation. The program calculates the elevation of each grid corner by looking for contour intersections in eight directions (N, S, E, W, NE, SE, SW, NW) and then interpolating the elevation between the two steepest intersections.

To accurately model the surface, it might be necessary to add entities in addition to the contour polylines. For one, spot elevation points can be added for the high and low points. Otherwise the grid model might plateau at the last contour. Also 3D breaklines need to be added on long narrow ridge and valley contours because in these areas the program will find the same contour when it looks for intersections in the eight directions. When all eight intersections are the same contour, the interpolated grid elevation equals the contour elevation instead of rising up the ridge or dipping in the valley. The 3D breaklines force interpolation along the ridge or valley. To draw these polylines, set the OSNAP to *Nearest* and run the *3D Polyline* command. Then draw the polyline by picking the contour polylines where the breakline crosses them. Another way to quickly create breaklines is to first draw 2D polylines. Then convert these polylines into 3D polylines with the *Screen* option in the *2D to 3D Polyline by Surface Model* command found on the 3Dpoly menu. There is also an automatic way to draw these breaklines. Under 3D Data, use the command: *Create Ridge polylines from Contours*.

Prompts

Grid File to Create File Selection Dialog

Enter a name for the grid file.

Use position from another file or pick grid position [<Pick>/File]?

Pick Lower Left grid corner <8111.88,3985.08>: *pick a point for the lower left limit of the grid*

Pick Upper Right grid corner <8366.88,4195.08>: *pick a point*

Make Grid File dialog box

In this dialog, you specify the grid resolution and whether or not to include data points with zero elevations. You can specify the resolution by entering the number of grid cells in the X and Y directions. By the Dimensions option, you to set the X and Y size for each grid cell.

Reading points ...

Select points, lines, polylines and faces to grid from.

Select objects: Specify opposite corner: 1075 found

Select objects:

Reading points ... 980

Finding points on breaklines ...

Ignored 2729 duplicate points.

Inserting breaklines 3480 ...

Triangulating points ... 980

Assigning grid values > 1800

Writing grid file: C:\Carlson 2008\WORK\example1.grd

Pick the Lower Left grid corner: *pick a point for the lower left limit of the grid*

Pick the Upper Right grid corner: *pick a point*

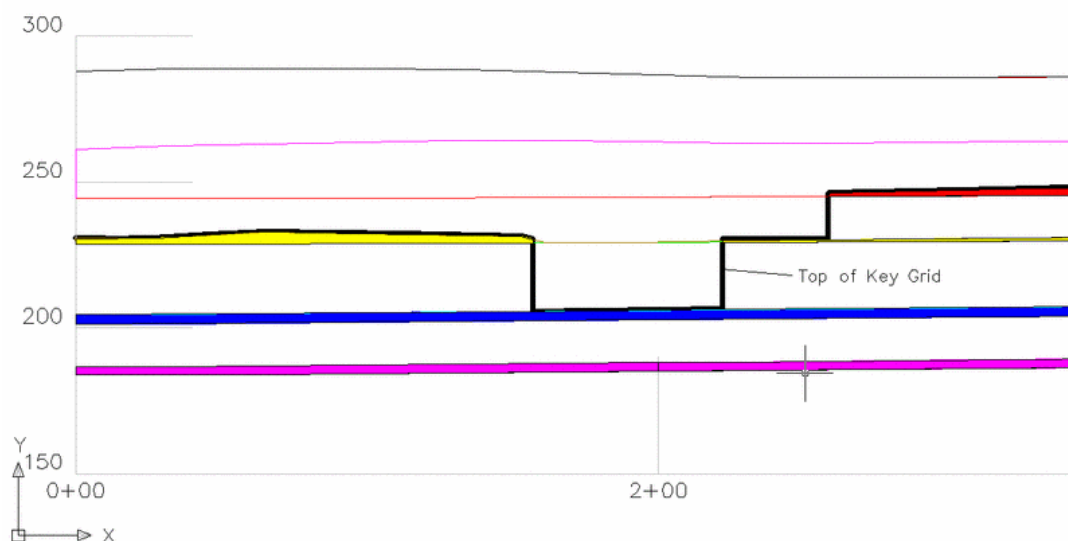
Pulldown Menu Location: Surface

Keyboard Command: mkgrid

Prerequisite: Entities that define the surface

Make Top Of Key Grid

This command analyzes the thicknesses of the Key strata in a Geologic Model file and creates a new base of overburden grid based on a minable key thickness entered. It starts at the top Key strata. Anywhere the thickness is less than the value entered, the grid drops down to the top of the next Key strata. This grid can then be added to a mining model Geologic Model file for calculating overburden volumes and for scheduling. Shown below is a cross section example of the grid dropping down when the key thickness falls below the entered thickness value.



Prompts

Minimum key thickness <0.0>: 1

Reading cell > 194032

Reading cell > 194032

Reading cell > 194032

Writing grid > c:\scad2005\DATA\New Top of Key.grd

Writing cell > 194032

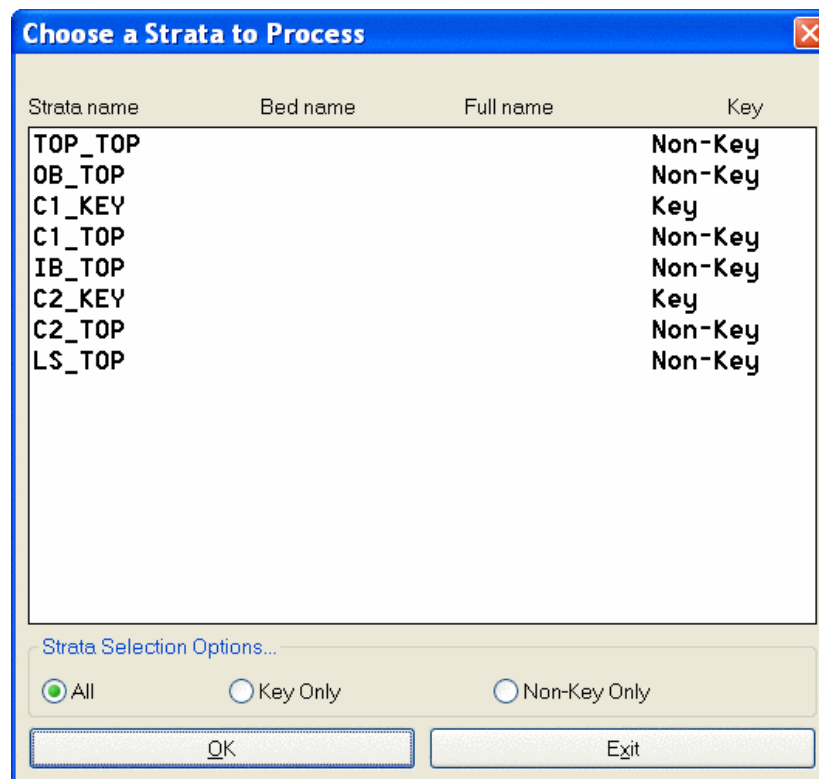
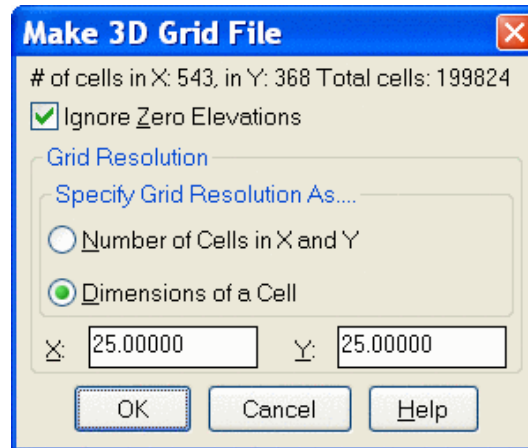
Done.

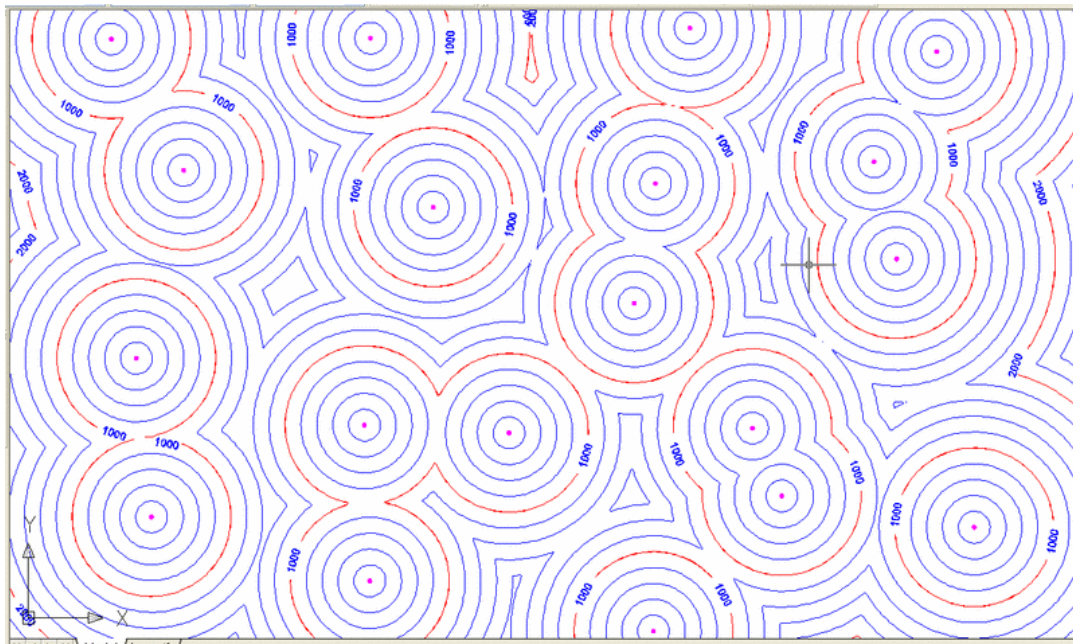
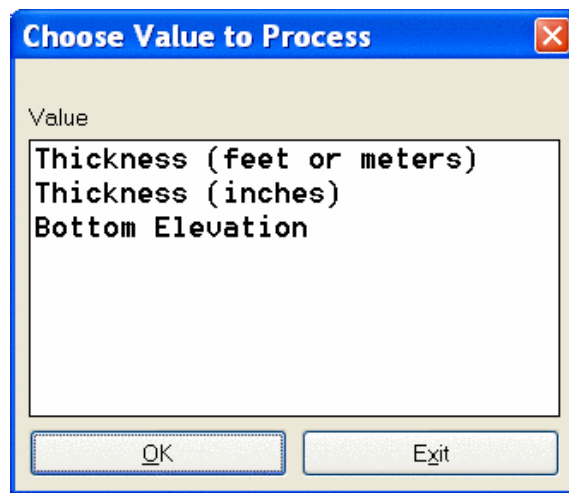
Pulldown Menu Location: StrataCalc

Keyboard Command: top_key

Make Nearest Data Point Grid

This command creates a distance grid file where the value of each grid corner is set to the distance to the nearest data point in a drillhole. This is useful to see the distance between drillholes when planning additional drilling. The grid can be contoured, as shown below, for better analyzing. The grid dimensions are set first, then two dialogs appear to select the strata to grid, and the attribute to grid.





Prompts

Pick Lower Left limit of surface area <1.45355e+006,1.96843e+006>:

Pick Upper Right limit of surface area <1.46712e+006,1.97763e+006>:

Ignore zero values [<Yes>/No]? *N*

Select drillholes, channel samples and strata polylines.

Select objects: *all*

129 found

Select objects:

Reading drillhole 129

Finding splits ...

Finding pinch out ...

Calculating seam stacking ...

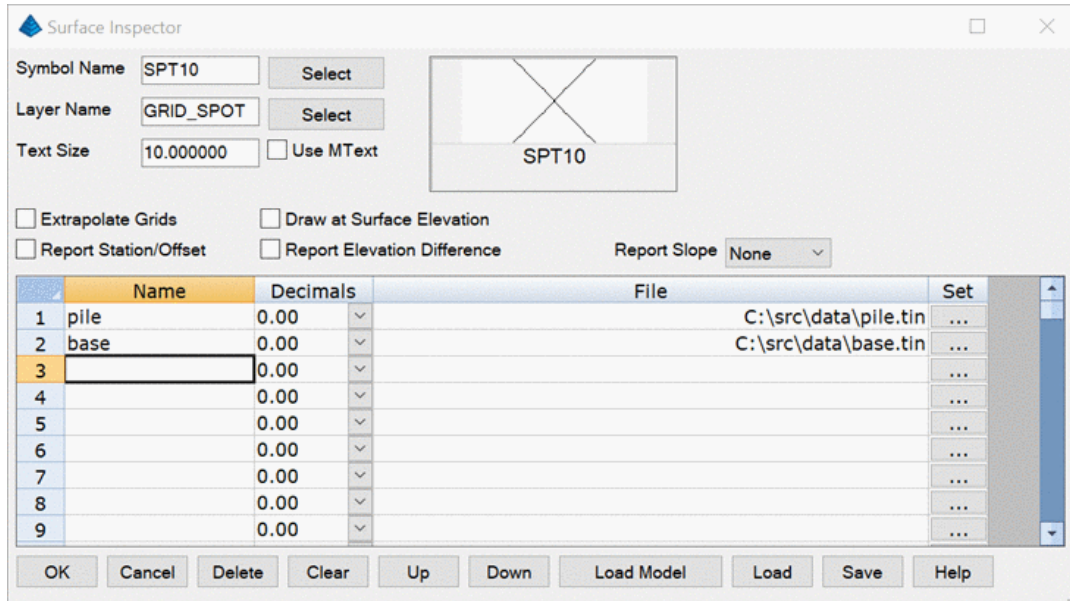
Writing grid file C:\DRAWINGS\GRIDS\Nearest Data Points.grd ...

Pulldown Menu Location: StrataCalc

Keyboard Command: distgrid

Surface Inspector

This command allows you to report and optionally label elevations for selected surface files. You can simultaneously analyze multiple surface files. Surface files can be either triangulation (.flt or .tin) files, grid (.grd) files, or any combination thereof. The following dialog opens when the command is initiated.



Symbol Name sets the symbol name to be plotted. Click **Select** to select the symbol from the symbol library.

Layer Name set the layer to use for the text labels.

Text Size sets the actual size (not scale factor) of the text label placed in the drawing. The **Use MText** controls whether to create the labels as MText or regular Text entities.

Turn the **Draw at Surface Elevation** toggle ON if you want the symbol to be located at the actual elevation of the surface.

The **Extrapolate Grids** option automatically extrapolates any unassigned grid nodes for grid surfaces.

Report Slope displays the slope of the surface along with the elevation.

Report Station/Offset prompts for a centerline and reports the station and offset in addition to the surface elevations.

When there are two surface files, the **Report Elevation Difference** option will add a third report field for the cut/fill between these two surfaces.

Name denotes the name that will be plotted when you label the elevation. The default value is the same as the name of the surface file, but you can change it.

Decimals individually sets the decimal elevation precision for each selected surface.

For **File**, either type in the surface name to use or press the **Select** button to choose the surface file from a browse window.

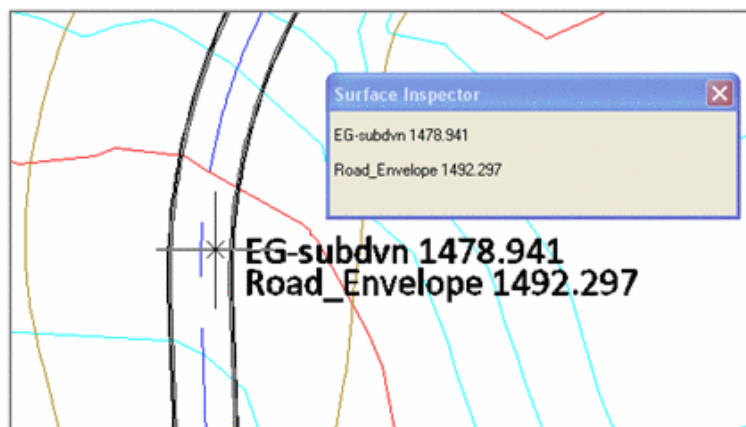
Load Model allows you to select grid files from a list of grids stored in a Geologic or Mining Model file. These files can be created in the Carlson Mining module.

Clear: Clears all values.

Load: Loads a Surface Inspector File (.SIF).

Save: Saves all settings to a Surface Inspector File (.SIF).

After you fill out the dialog box, click OK. Surface Inspector will load the surfaces and begin showing you real-time elevations for each surface as you move the cursor on the screen. If you pick a point or enter coordinates, the elevation will be labeled along with the surface name and selected symbol as shown below.



Pulldown Menu Location: Surface

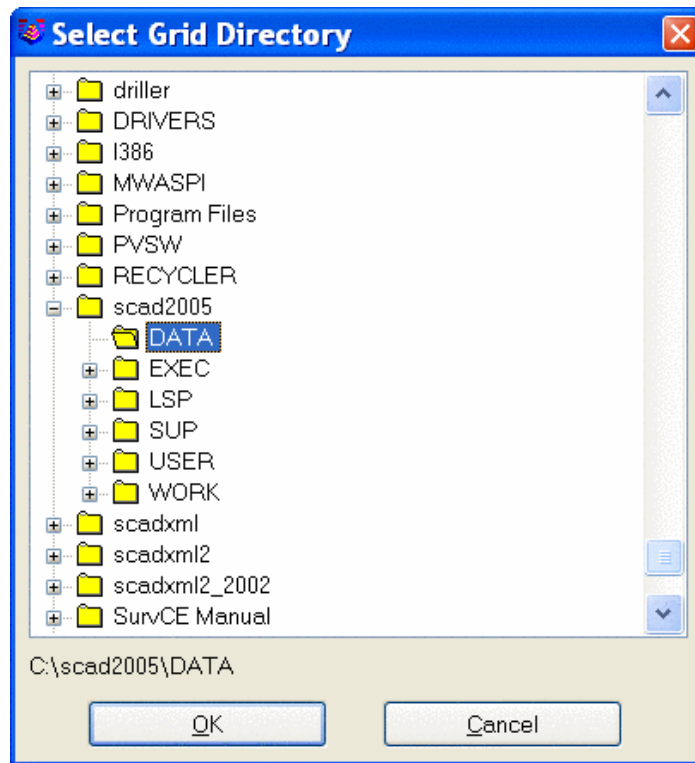
Keyboard Command: surfvals

Prerequisite: Surface Model(s)

Grid History Review

This command will report information on the creation and history of grid files (.GRD) in the selected directory. Such parameters as modeling methods and settings, time and date, grid cell dimensions, file size and even which drillholes used with their values are reported.

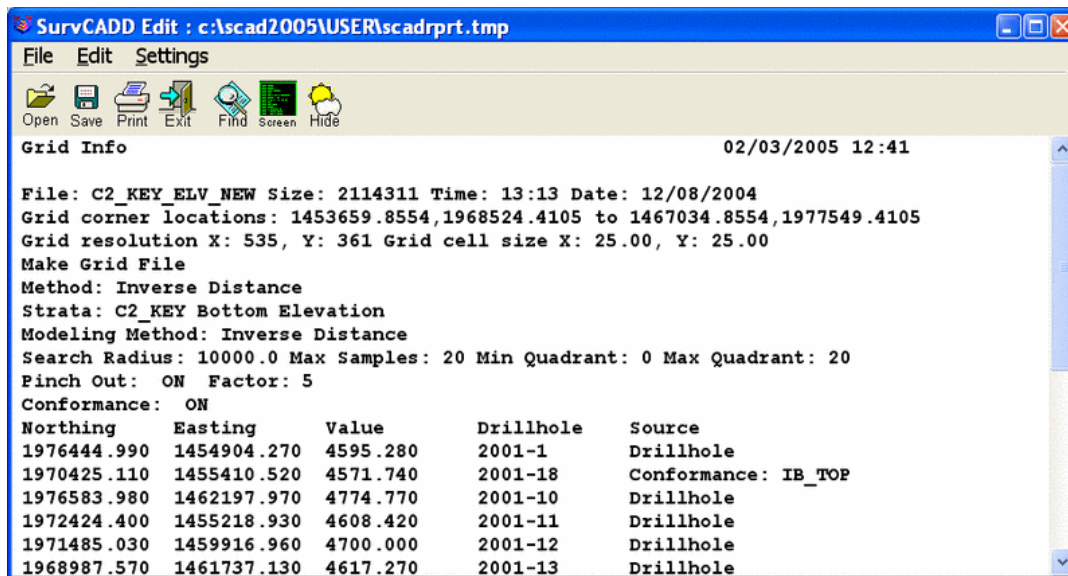
There are no prompts in this routine, only three dialog windows that guide you through this informative command. The first window is where you select the directory containing the grids to analyze. Simply highlight that directory and select OK. The program will read all of the grids in that directory and present them in the Grid History Review window. If there are many grids, this might take a minute or two.



The list of grids shown is sorted alphabetically and displays the name, size, time and date, cell size and modeling method for creation. If the grid was made in a version of SurvCADD from CES (this version is from around 1999) and before, then all the information will show "UNKNOWN".

NAME	SIZE	TIME	DATE	CELL-SIZE	METHOD	
C2_KEY_BTU	2114152	14:05	08/04/2004	25.0	25.0	Inverse Dist
C2_KEY_EL2	2114413	14:05	08/04/2004	25.0	25.0	Inverse Dist
C2_KEY_ELU	2114261	14:05	08/04/2004	25.0	25.0	Inverse Dist
C2_KEY_ELU_NEW	2114311	13:13	12/08/2004	25.0	25.0	GFU
C2_KEY_MOISTUR	1725655	14:05	08/04/2004	25.0	25.0	Inverse Dist
C2_KEY_SUL	1532540	14:05	08/04/2004	25.0	25.0	Inverse Dist
COAL_B1	582220	13:37	02/01/2005	25.0	25.0	Unknown
DL PIT	2054644	20:14	12/08/2004	25.0	25.0	Unknown
FADF	2054054	11:53	12/20/2004	25.0	25.0	Unknown
FADF_1	2054037	11:53	12/20/2004	25.0	25.0	Unknown
FADF_2	2054065	11:53	12/20/2004	25.0	25.0	Unknown
FADF_3	2053984	11:53	12/20/2004	25.0	25.0	Unknown
FADF_4	2053996	11:53	12/20/2004	25.0	25.0	Unknown

Highlight a grid and selecting the info button will bring up the report window showing yet more information of the highlighted grid. This is where the method and settings, strata, and drillholes with their names, locations and values are reported. The report may be edited here, saved to a file, sent to the printer or placed on screen as AutoCAD text.

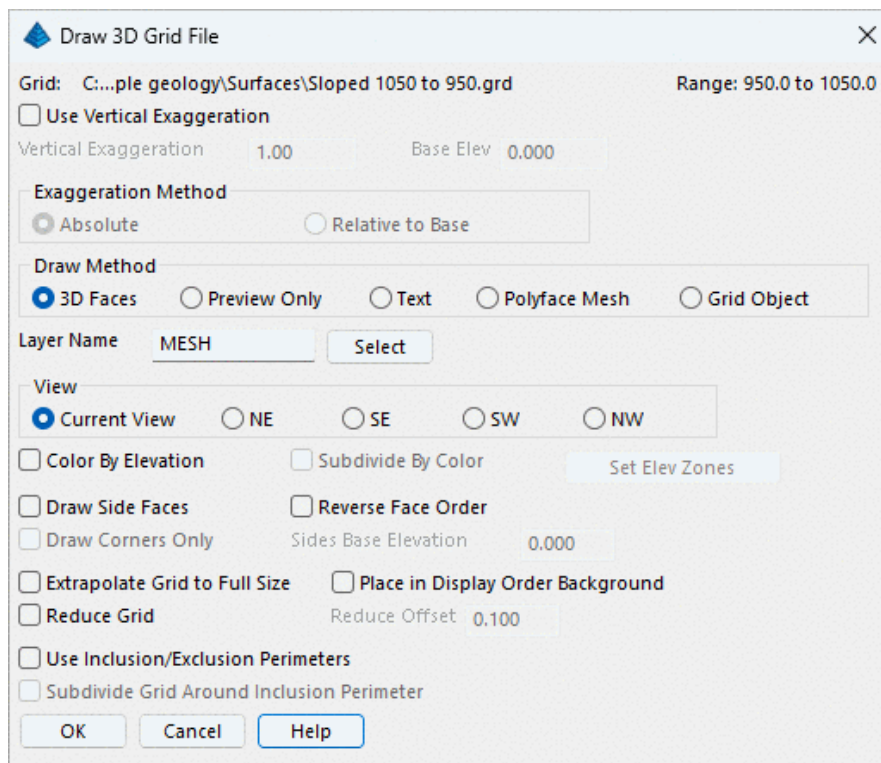


Pulldown Menu Location: StrataCalc
Keyboard Command: grdreview

Draw 3D Grid File

This command draws the 3D grid mesh of the chosen grid (.GRD) file. Each grid cell can be drawn as a Surface Object, 3D Face entities, Polyface mesh, Text or temporary lines. 3D Faces and Polyface Meshes can be viewed/used in the following commands: *3D Viewer Window*, *Viewpoint 3D*, *Hide*, *Shade*, *3D Surface FlyOver*, and *Slope Zone Analysis*.

The Surface Object is a custom object for displaying the grid in the drawing. This custom object also supports the EndPoint snap. The main advantage of the Surface Object is with displaying large grid files because the Surface Object is more efficient than drawing individual 3D Faces. The Surface Object requires Carlson to be loaded to display.



If **Use Vertical Exaggeration** is checked, grid elevations are multiplied by the value specified.

Exaggeration Method specifies whether to use an *Absolute* exaggeration method or *Relative to Base*, which uses the specified base elevation.

Specify the type of entities to draw in **Draw Method**. 3D Faces are described above. The Preview Only option draws the grid using temporary vectors. This method provides a much faster way to view the grid. However these temporary vectors are erased when the viewport is modified. This means as soon as you execute zoom, redraw, regen or plot, this grid will disappear. You can quickly redraw the grid by typing in VG for View Grid at the command prompt. Polyface Mesh is similar to 3D Faces except it is a single entity. The Text option will label the grid elevation at the grid corner. The text is placed center justified over the grid corner. To reduce clutter, there is an option to skip rows and columns.

Specify the layer for the grid entities in **Layer Name**.

Specify the initial viewing direction in **View**.

When **Color by Elevation** is checked, the grid will be colored based on a table of user-defined elevation ranges and the assigned colors. There is also an option to subdivide the grid cells at the color zone transitions, similar to the Elevation Zone Analysis command. Click the Set Elev Zones button to define ranges and colors.

When **Draw Side Faces** is checked, the program will draw vertical faces around the perimeter of the grid. The side faces will be drawn vertically from the grid perimeter to the Sides Base Elevation. You may optionally specify the Sides Base Elevation, it defaults to 0.00.

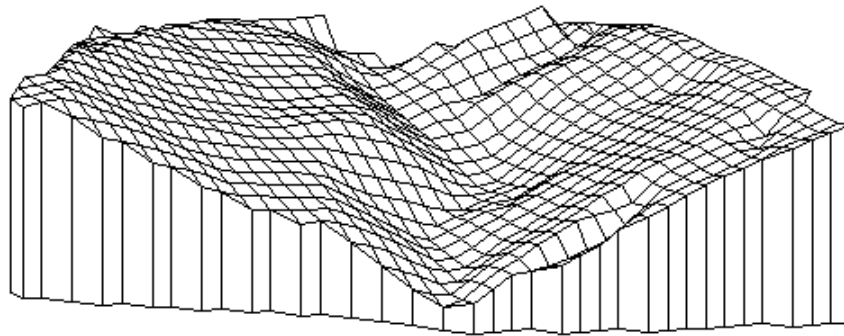
When checked, **Reverse Face Order** changes the direction of the points for a grid cell from clockwise to counter-clockwise. The order applies to shading the grid cell in 3D render viewers such as the *3D Viewer Window* command. The grid cell will only appear shaded when viewing the grid cell from the clockwise side. Viewing from the other side will show a wire frame. The default is to show the shaded side from the top-down view. This option allows you to draw the grid so that the underside of the grid is shaded.

When checked, **Draw Corners Only** will draw the side lines only at the grid corners. Otherwise side lines are drawn down each perimeter grid cell.

When checked, **Extrapolate Grid to Full Size** draws the entire rectangular surface of the grid.

When **Use Inclusion/Exclusion Perimeters** is checked, it allows you to select inclusion and exclusion areas. Only grid cells inside the inclusion polylines will be drawn. Grid cells inside the exclusion polylines will not be drawn.

When checked, **Subdivide Grid Around Inclusion Perimeter** subdivides grid cells that are partially inside and outside the perimeter into smaller resolution grid cells.



Drawn grid file using inclusion perimeter and side faces option
viewed with Viewpoint 3D

Pulldown Menu Location: Surface >> Draw Surface

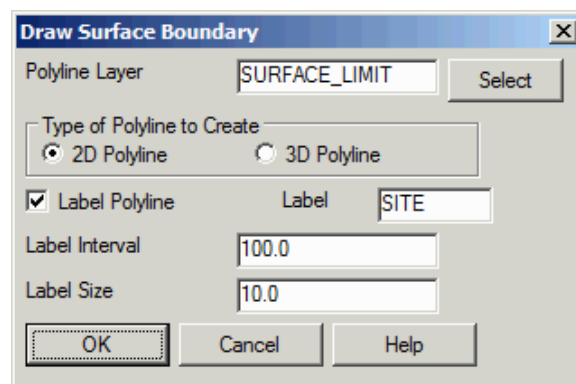
Keyboard Command: plotgrid

Prerequisite: a grid (.GRD) File

Draw Surface Boundary

This command draws the exterior perimeter of a triangulation or grid surface as a polyline. This is a simple way to show the size, shape and location of a surface without adding a lot of drawing entities and file size to the drawing.

The program prompts for the layer of the polyline to create and the type of polyline. A 2D polyline is drawn at zero elevation. A 3D polyline uses the surface elevations. There is an option for Label Polyline which adds text labels along the polyline with the specified name, size and interval. This label can be helpful to identify different surface boundaries.



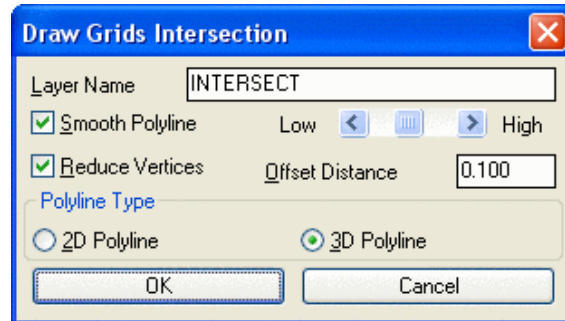
Pulldown Menu Location: Surface >> Draw Surface

Keyboard Command: grdlimit

Prerequisite: a surface file (.GRD, .TIN, .FLT)

Draw Surface Intersection

This command draws a 2D or 3D polyline at the intersection of two surfaces. In addition to this choice, the dialog also allows for the specification of the layer to draw the polylines on, whether or not to smooth them, and whether or not to reduce vertices.



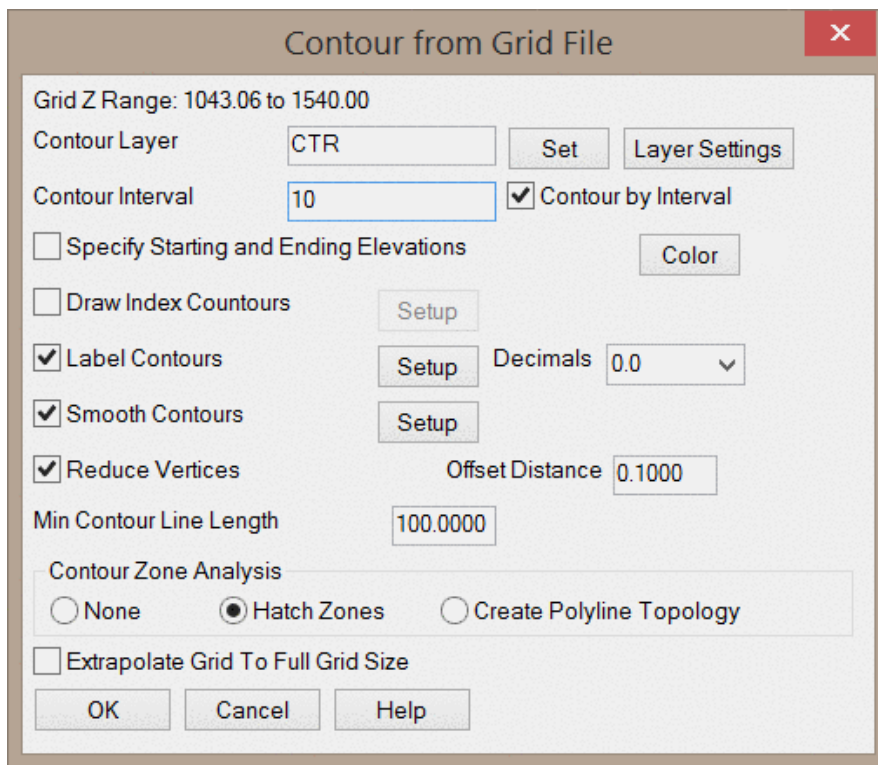
Pulldown Menu Location: Surface >> Draw Surface

Keyboard Command: grdcross

Prerequisite: two surface files (.GRD, .TIN, .FLT)

Contour From Grid File

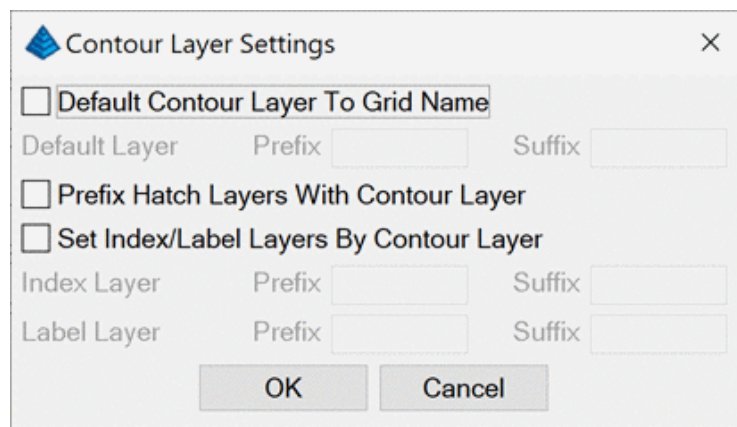
This command creates contours from a grid file. Contouring from a grid employs a different method than from a triangulation network and generally produces contours that loop more. The grid has data points at a regular interval while the triangulation has edges for every point and breakline in the surface. The smoothness of the contours depends a great deal upon the grid resolution. When the command is executed, you will be prompted to select inclusion/exclusion perimeter polylines to limit the contours to certain areas. If no perimeter polylines are selected, the entire grid will be contoured. After selecting a grid file to contour, the below dialog will appear.



Grid Z Range: This text displays the current range of Z values in the grid file to help you determine the optimal Contour Interval/Values.

Contour Layer: This sets the layer for the contours. You may select an existing layer by clicking the **Set** button.

Layer Settings: This button will open the below dialog, which controls default naming for layers of the various linework.



Default Contour Layer To Grid Name: When enabled, this toggle will automatically set the **Contour Layer** to match the grid name. For example, a grid named "Topo.grd" would default to draw contours on the "Topo" layer. The Prefix and Suffix are combined with the grid name. For example, a grid named "Topo.grd" with a prefix of "Contour" would create a layer of "Contour.Topo".

Prefix Hatch Layers With Contour Layer: When enabled, this toggle will use the **Contour Layer** as a prefix for all of the hatch layers. For example, if the first hatch zone is set to be drawn on the "Zone 1" layer and the Contour Layer is set to "CTR", the actual layer name of the hatch will be "CTR Zone 1".

Set Index/Label Layers by Contour Layer: When enabled, this option will automatically set the **Index Layer** and the **Text Layer** to be the same as the **Contour Layer**. However, you may also specify a prefix or suffix for each layer. For example, if the Contour Layer is set to "CTR" and the Index Layer Suffix is set to " Index", then the index contours will be drawn on the "CTR Index" layer. It is important to note that this option only sets these layer names when the command is first executed or when the above dialog is opened, meaning that you may still override these layer names if you wish.

Contour Interval/Values: This value is dependent on the **Contour by Interval** toggle.

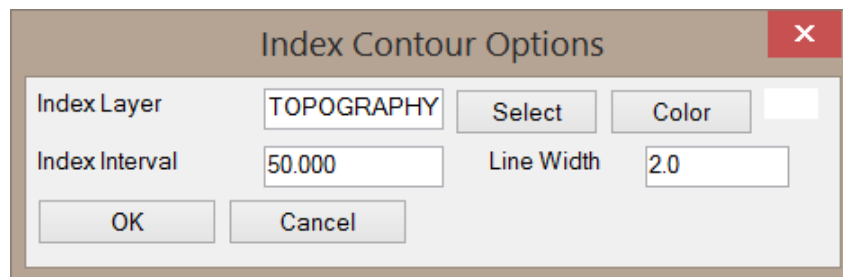
When the toggle is enabled, this value will set the interval for the contours. For example a value of 2 will draw contours along the grid values of 2, 4, 6, 8, etc. The first value contoured will be the first value divisible by that contour value. For example, if an interval of 3 is selected for a grid that ranges from 5 to 20, the first contour will be drawn at a value of 6.

When the toggle is disabled, this value(s) will set the specific value(s) to contour. For example, if a value of 10 is selected, only one contour will be drawn where the grid has a value of 10. Multiple contours may be drawn by entering values separated by spaces. For example, entering "10 12 16" into this text box will draw contours at values of 10, 12, and 16.

Specify Starting and Ending Elevations: When enabled, this toggle will prompt you to enter the starting and ending values to contour. For example, if the grid ranges in values from 0 to 100, you can use this option to only draw contours for values ranging from 30 to 60.

Color: This button will open the color palette, where you can select the color for the contours. Please note that this does not set the color of index contours.

Draw Index Contours: When enabled, this option will draw some contours differently than the main contours. You can control the setup of the index contours by clicking the **Setup** button to the right of this toggle. When clicked, the below dialog will appear.



Index Layer: This sets the layer for the index contours. You may select an existing layer by clicking the **Select** button.

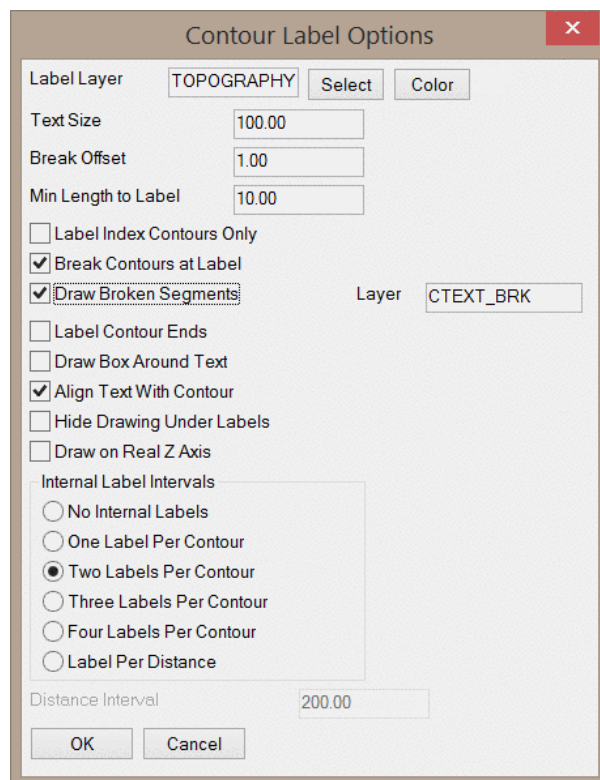
Color: This button will open the color palette, where you can select the color for the index contours. Please note that this does not set the color of non-index contours.

Index Interval: This value sets the interval of the index contours. This value will default to be the **Contour Interval** multiplied by 5.

Line Width: This value sets the width of the index contours. The non-index contours will have a line

width of 0 by default.

Label Contour Lines: When enabled, this toggle will add labels to the contours. The number of decimals in the labels is controlled by the **Decimals** dropdown list. You can control the setup of the labels by clicking the Setup button to the right of this toggle. When clicked, the below dialog will appear.



Label Layer: This sets the layer for the labels. You may select an existing layer by clicking the **Select** button.

Color: This button will open the color palette, where you can select the color for the index contours. Please note that this does not set the color of non-index contours.

Text Size: This value sets the height of the text in drawing units.

Break Offset: This value sets the distance between the end of the trimmed contour and the start/end of the text. This value is only used when the **Break Contours at Label** toggle is enabled.

Min Length to Label: This value sets the minimum length of contours to be labeled. For example, if this value is set to 10, a contour with a length of 5 will not be labeled.

Label Index Contours Only: When enabled, this toggle will only add labels to index contours.

Use MText: Sets whether to create the labels as Text or MText entities.

Break Contours at Label: When enabled, this toggle will break the contour line near the label, making it easier to read the labels.

Draw Broken Segments: When enabled, this toggle will draw the broken segments of the contour (the por-

tion of the contour under the labels). These segments will be drawn on the layer specified to the right of this toggle.

Label Contour Ends: When enabled, this toggle will draw additional labels at the ends of contours. This option will not draw additional labels on contours draw as closed polylines.

Draw Box Around Text: When enabled, this toggle will draw a box around the text labels.

Align Text with Contour: When enabled, this toggle will align the label with the contour itself. When disabled, labels will be aligned horizontally.

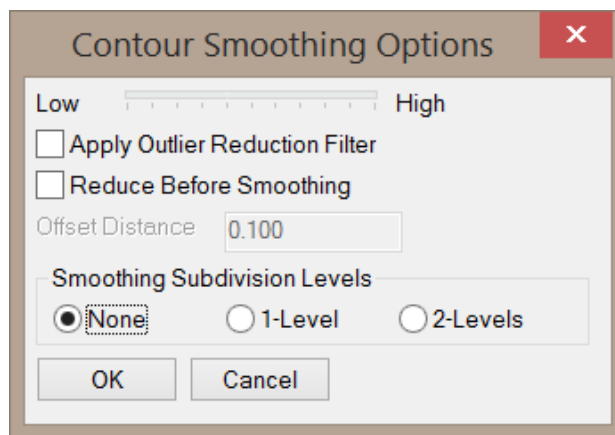
Hide Drawing Under Labels: When enabled, a WIPEOUT entity will be draw around the labels, hiding the contour and any other linework underneath it. This option has the same visual effect as breaking the contour at the label, but allows you to retain a continuous contour.

Draw on Real Z Axis: When enabled, labels will be drawn at the same elevation as the contour. When disabled, labels will be drawn at zero elevation.

--- **Labels Per Contour:** These radio buttons set the number of labels to draw on each contour.

Label Per Distance: This radio button will draw a contour label on a regular interval along the contour, set by the **Distance Interval** value.

Smooth Contours: When enabled, the contours will be smoothed to reduce sharp edges. You can control the smoothing options by clicking the **Setup** button to the right of this toggle. When clicked, the below dialog will appear.



Low-High Slider Bar: This slider bar controls the degree of smoothing. This is based on Bezier smoothing, which holds all the original points calculated from the grid and only smooths between the calculated points.

Apply Outlier Reduction Filter: When enabled, this toggle will remove spikes in the contours that don't follow the general trend direction.

Reduce Before Smoothing: When enabled, this toggle will apply the Reduce Vertices function to the contours before smoothing. The **Offset Distance** is the maximum distance the contour is allowed to move when removing vertices.

Smoothing Subdivision Levels: This option will internally subdivide the grid cells with a quadratic smoothing algorithm to help create smoother contours.

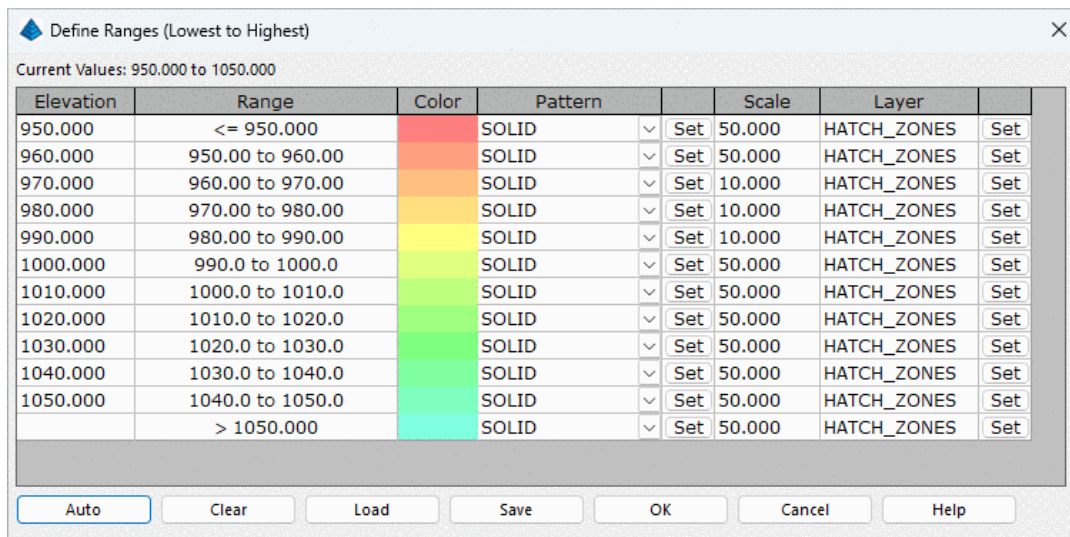
Reduce Vertices: When enabled, this toggle will apply the Reduce Vertices function to the contours. The **Offset Distance** is the maximum distance the contour is allowed to move when removing vertices.

Min Contour Line Length: This values sets the minimum length required for contours to be drawn. For example, if this value is set to 100, a contour with a length of 50 will not be drawn. This filter helps to prevent drawing relatively insignificant contours.

Contour Zone Analysis: This option controls how the contour zones will be drawn. A zone is simply the space between two contour lines.

None: The contour zone boundaries will not be drawn. Only the contours themselves will be drawn.

Hatch Zones: This option will draw the contour zone boundaries and hatch them according to the below dialog. When this option is selected, you will be prompted to pick a location to draw a legend. It is recommended that you select an inclusion boundary when using this option even if you are contouring the entire grid.



The above dialog controls the appearance of the hatch zones. A description of each column is given below. Note that you may need to scroll down to other pages to view all of the contour zones.

Elevation: Sets the maximum contour value for the zone.

Range: Displays the range of values included in the zone.

Color: Sets the hatch color for the zone.

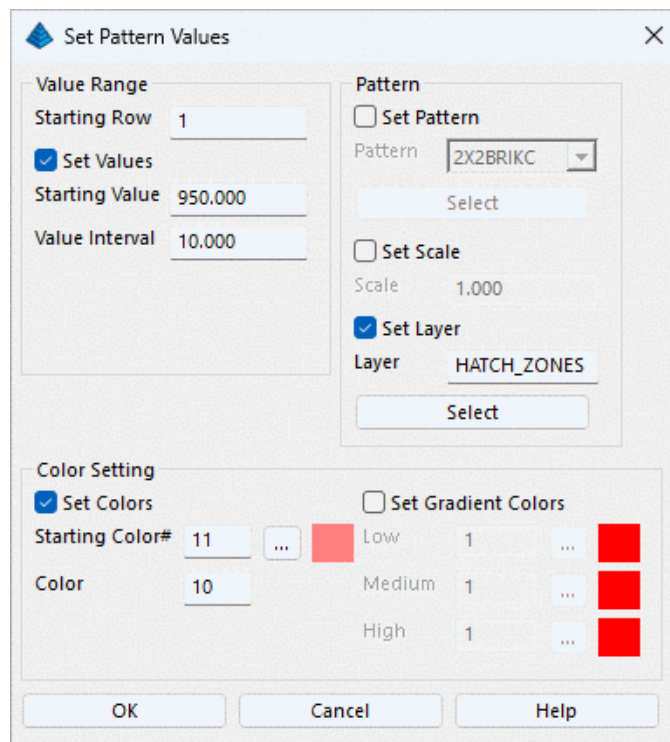
Pattern: Sets the hatch pattern for the zone.

Scale: Sets the hatch pattern scale for the zone.

Layer: Sets the layer for the hatch pattern.

Clear: This button will clear all settings in the dialog.

Auto: This button will display the below dialog, which will let you automatically populate each hatch zone with various properties.



Starting Zone: This value sets the first zone to be modified. If this value is set to 1, all zones in the dialog will be modified. You can toggle the various options on this dialog to only modify certain properties of each zone, however.

Set Values: When enabled, the values of each zone will be modified. The **Starting Value** sets the value of the first zone selected. That value will increment by the **Value Interval** value.

Set Colors: When enabled, the colors of each zone will be modified. There are two ways to set the colors of the zones, controlled by the **Set Gradient Colors** option.

When **Set Gradient Colors** is disabled, the colors for each zone will be defined by a **Starting Color #** and a **Color Increment**. The Starting Color # relates to a color in the CAD color palette. The Color Increment sets the color for the next zone. For example, if the Starting Color # is 10 and the Color Increment is 2, the second contour zone will use color 12. The default values of 10 and 10 will create a rainbow color gradient. Similarly, a Starting Color # of 11 and a Color Increment of 10 will produce a lighter shade of the rainbow gradient. It can be difficult to create other color gradients with this system, but the other method of setting color allows for easier selection of other gradients.

When **Set Gradient Colors** is enabled, you will be able to set color gradients by selecting a **Low**, **Medium**, and **High** color value. All intermediate colors will be selected automatically. It should be noted that this command can only use the colors from the CAD color palette, which includes 255 colors. As a result, attempting to use this option with a large number of hatch zones may result in some hatch zones using the same color.

Set Pattern: When enabled, the pattern for each zone will be set to the selected hatch pattern.

Set Scale: When enabled, the hatch pattern scale for each zone will be set to the **Scale** value.

Set Layer: When enabled, the layer for each zone will be set to the **Layer** value. You may select an existing layer by clicking the **Select** button.

Create Polyline Topology: This option will draw the contour zone boundaries, but will not hatch them. This option will add an extra label in the zone boundary to indicate the range of grid values in that zone. This can

be useful for GIS applications in other modules.

Extrapolate Grid to Full Grid Size: This option will extend the contour lines to the rectangular limits of the grid when values are left as null. Note that this will not modify the grid file itself.

Prompts

Select the Inclusion perimeter polylines or ENTER for none.

Select objects: *pick a closed polyline for the contour boundary if any*

Select the Exclusion perimeter polylines or ENTER for none.

Select objects: *pick a closed polyline for the area to exclude*

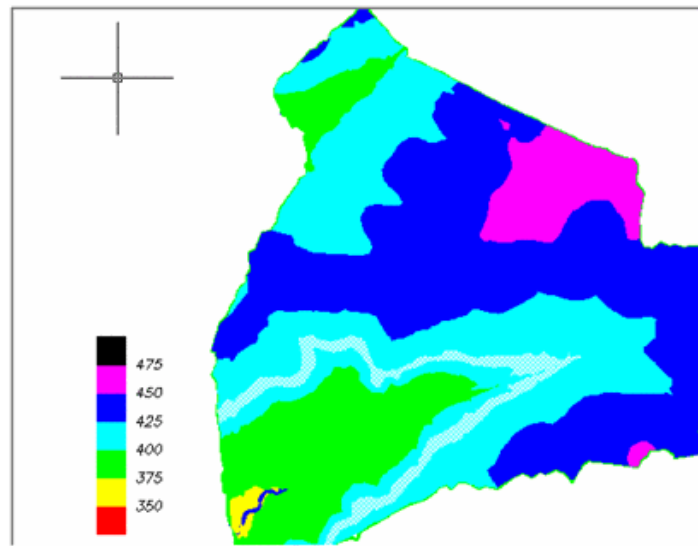
Grid File to Process dialog *select a .grd file*

Contour from Grid File options dialog

Extrapolate grid to full grid size (Yes/<No>)? *press Enter* This prompt appears if your grid extends beyond the limits of your data points in some areas.

Contours interpolated from GRID

Setting color ranges using "Hatch Zones" option



Pulldown Menu Location: Surface >> Contour from...

Keyboard Command: cntgrd

Prerequisite: A grid file

One Surface Volumes

This command calculates the cut and fill volumes between the surface modeled by one grid (.GRD) file and a constant elevation or value. This is the same as *Two Grid Surface Volumes* except that the second surface is a flat plane at a constant elevation instead of a 3D grid surface. Please refer to that section for additional details. If the grid contains grid cells that have no elevations, you have the option to extrapolate elevations from the grid cells with elevations. When you choose not to extrapolate, no volume is calculated for these grid cells. There are also options

to specify inclusion and exclusion areas. When inclusion areas are specified, only the volume within this inclusion area is calculated. Volumes within an exclusion area are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to using this command.

Note: the volume comparison of this routine uses the selected grid file as the *base* surface, and the target elevation plane as the *final* surface, so be aware that if your target elevation is set primarily below the surface defined by the grid file, it will report as cut, when in reality you may be filling above the target elevation plane to reach the defined surface.

Prompts

Select the Inclusion perimeter polylines or ENTER for none:

Select objects: *press Enter*

Select the Exclusion perimeter polylines or ENTER for none:

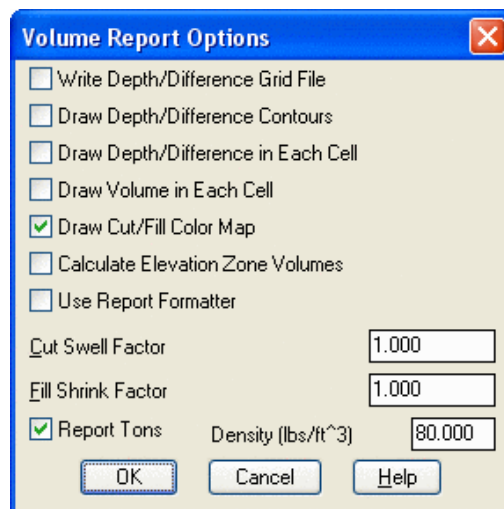
Select objects: *press Enter*

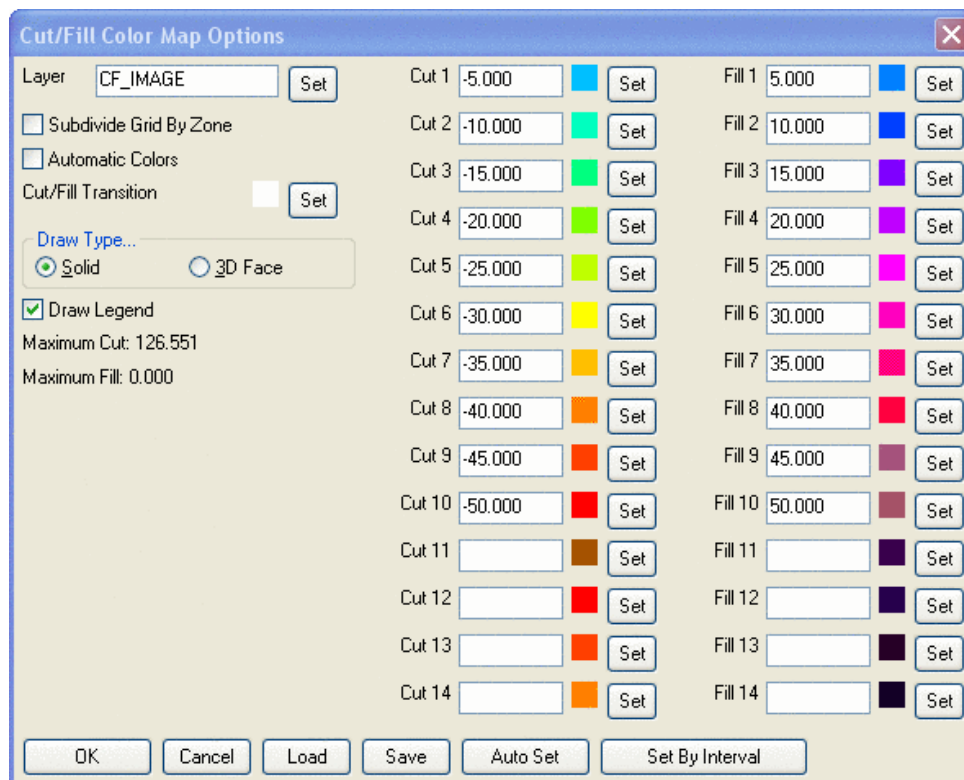
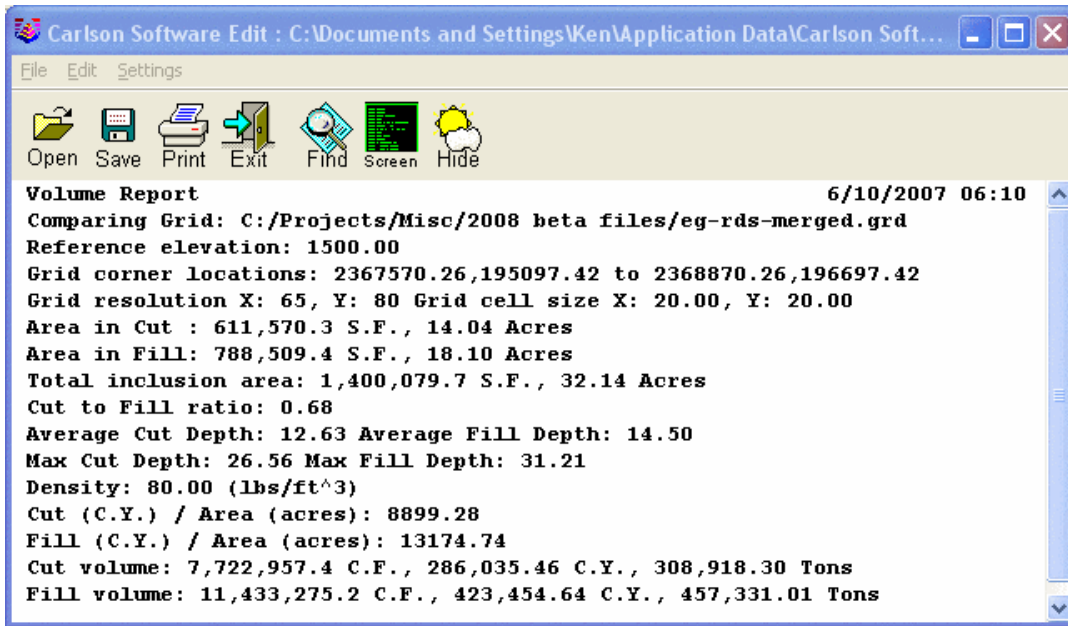
Specify Grid File Selection Dialog Choose a grid (.GRD) file to process.

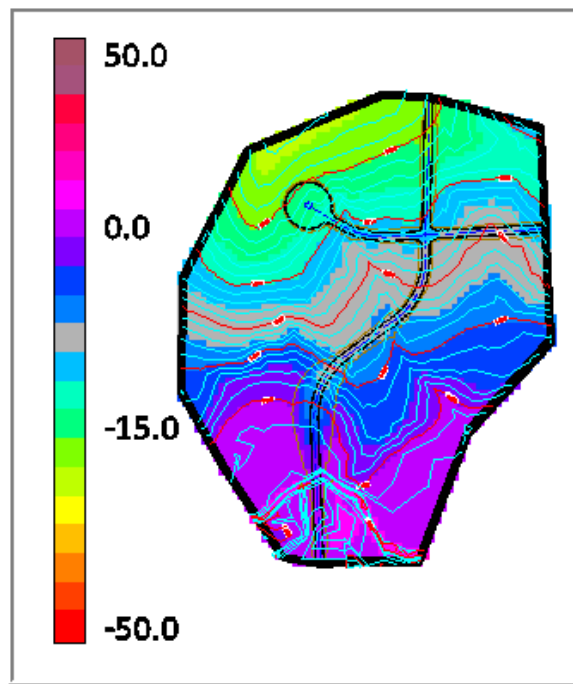
Extrapolate grid to full grid size (Yes/<No>)? *press Enter* If you enter Yes to this prompt, surface elevations will be computed for any grid cells that have null elevations.

Enter the base elevation: *1500* This defines the second surface.

Volume Report Options Dialog







Pulldown Menu Location: Surface >> Volumes By Grid Surfaces

Keyboard Command: volcalc1

Prerequisite: A grid (.GRD) file

Two Surface Volumes

Two Grid Surface Volumes calculates the cut and fill volumes between two surfaces modeled by grid (.GRD) files. These two grid files must have the same location and resolution. To create the grid files, use the *Make 3D Grid File* routine. When creating the second grid file, choose *Use position of another file* and select the first grid file. Using the position of the first grid file sets the location and resolution of second grid to match the first.

There are several other routines that calculate volumes based on grid files. Grid based volumes can be calculated by *One Grid Surface Volumes*, *Volumes by Layer*, *Stockpile Volumes*, and *Pond/Pit Volumes*. These routines have special prompting and calculate the grid surfaces and volume in one step.

Volumes by Two Surface Volumes has three steps:

1. Creating the first grid file with *Make 3D Grid File*
2. Creating the second grid file with *Make 3D Grid File*
3. Running *Two Grid Surface Volumes*

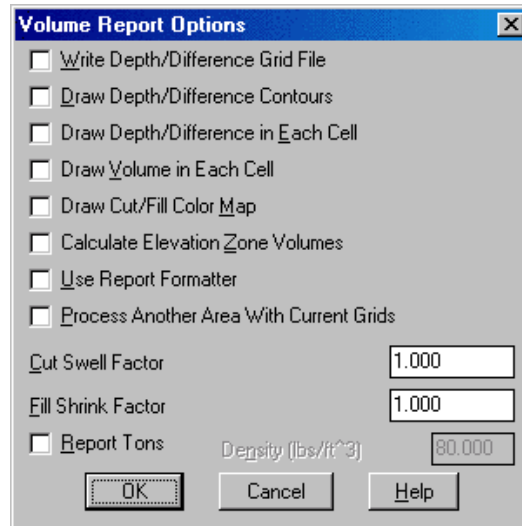
One advantage to this command is that you have more output options to help analyze volumes.

Besides grid based volumes, volumes can also be calculated between triangulation surfaces using the *Volumes by Triangulation* commands. Cross section end area is another volume method that is used by the *Calculate Sections Volume* command in the Civil Design module.

There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the volume within this inclusion area is calculated. **Important:** Whenever possible you should use a polyline that represents the limits of disturbed area as the inclusion perimeter. Volumes within an exclusion area are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine.

If the grid contains grid cells that have no elevations, you have the option to extrapolate elevations from the grid cells with elevations. When you choose not to extrapolate, no volume is calculated for the grid cells left without elevations. In general, extrapolation is not very accurate and should be avoided whenever possible. Sometimes you may get small amounts of cut in stockpiles that should only be fill, or small amounts of fill in pits that should only

be cut. These extraneous quantities are due to extrapolation at the border and should be small enough to be ignored. When inclusion or exclusion polylines are used, the program will automatically extrapolate the grids. In addition to writing a volume report to the file, printer or screen, there are several volume report options.



Write Difference Grid File creates a grid (.GRD) file of the elevation difference of the two grid files.

Draw Difference Contours creates a contour map of the difference or depth between the two grid files.

Draw Elevation Difference in Each Cell plots the elevation difference at the grid corners which is the same as the Elevation Difference routine.

Draw Volume in Each Cell plots the calculated volume for each grid cell and is an excellent way to verify the volume calculation. If a cell contains both cut and fill, both values will be plotted.

Calculate Elevation Zone Volumes calculates the cut and fill between different elevation ranges.

Draw Cut/Fill Color Map fills each grid cell with different shades based on the average cut or fill in the cell. Red shades are used for cut and blue for fill. There is an option to draw a color legend. You can subdivide the grid cells at zone transitions. Also, there is an option to control the zone intervals and range.

Use Report Formatter allows you to customize the report by choosing the fields to report and their order. Also the report formatter can be used to output the report data to Microsoft® Excel or Microsoft® Access.

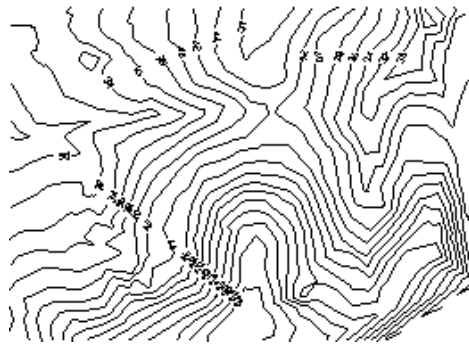
Process Another Area with Current Grids runs Two Surface Volumes again using the same grid files but different inclusion/exclusion polylines. This option saves the step of reloading the grid files to calculate volumes from the same grids for multiple areas.

The **Cut Swell Factor** value is multiplied by the cut volume in the report.

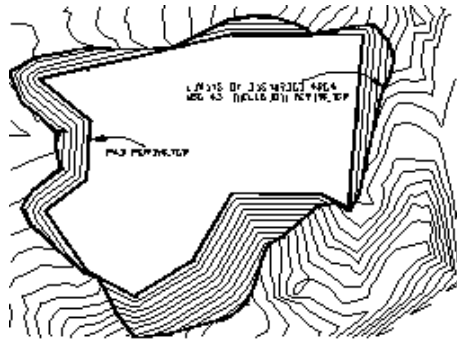
The **Fill Swell Factor** value is multiplied by the fill volume in the report.

Report Tons allows you to enter the material density and the program will report the cut and fill tons in addition to volume.

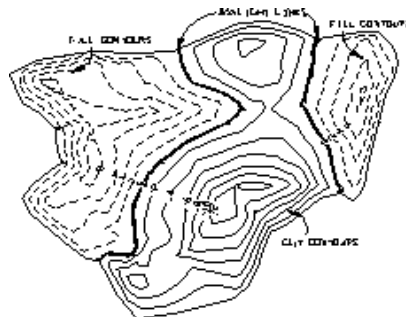
Given two accurate grid (.GRD) files, this routine will calculate accurate volumes. To verify the volume calculation, it is a good idea to check the grid (.GRD) files either by drawing them with *Draw Surface >> Draw 3D Grid File* and viewing them with *the 3D Viewer* or by contouring the grids with the *Contour Grid File* command.



Existing surface



Final surface contours with a closed polyline



Contours from the Draw Depth/Difference Contours option. Cut contours are red, fill contours are blue, daylight contours are green. This is a good way to check that both surfaces are modeled correctly and to verify the volumes.

Sample Two Surface Volumes report:

Volume Report

```

Comparing Grid: C:\scad2006\data\simo.grd
and Grid: C:\scad2006\data\final.grd
Lower left grid corner : 186551.67,57624.98
Upper right grid corner: 186828.81,57897.09
X grid resolution: 75, Y grid resolution: 75
X grid cell size: 3.70, Y grid cell size: 3.63
Total inclusion area: 37016.71 sq ft, 0.850 acres
Cut to Fill ratio: 1.14
Cut (C.Y) / Area (acres): 3642.35
Fill (C.Y) / Area (acres): 3182.70
Cut vol: 83570.89 cubic ft, 3095.22 cubic yards
Fill vol: 73024.56 cubic ft, 2704.61 cubic yards

```

Prompts

- Select the Inclusion perimeter polylines or ENTER for none:
- Select objects: *pick a closed polyline for the limits of disturbed area*
- Select objects: *press Enter*

Select the Exclusion perimeter polylines or ENTER for none:

Select objects: *press Enter*

Specify Base Grid File Selection Dialog

Choose a grid (.GRD) file to process.

Extrapolate grid to full grid size (Yes/<No>)? *press Enter* If you enter *Yes* to this prompt, surface elevations will be computed for any grid cells that have null elevations.

Sample report from the Calculate Elevation Zone Volumes option:

(Calculates the cut and fill in different elevation ranges at a user-specified interval and beginning at a user-specified starting elevation.)

Volumes by elevation zone

Zone 20.00 to 30.00

Cut volume : 0.30 cubic ft, 0.01 cubic yards

Fill volume: 107.90 cubic ft, 4.00 cubic yards

Zone 30.00 to 40.00

Cut volume : 4.88 cubic ft, 0.18 cubic yards

Fill volume: 73021.14 cubic ft, 2704.49 cubic yards

Running total:

Cut volume : 5.18 cubic ft, 0.19 cubic yards

Fill volume: 73129.05 cubic ft, 2708.48 cubic yards

Zone 40.00 to 50.00

Cut volume : 65044.26 cubic ft, 2409.05 cubic yards

Fill volume: 0.25 cubic ft, 0.01 cubic yards

Running total:

Cut volume : 65049.44 cubic ft, 2409.24 cubic yards

Fill volume: 73129.29 cubic ft, 2708.49 cubic yards

Zone 50.00 to 60.00

Cut volume : 17786.85 cubic ft, 658.77 cubic yards

Fill volume: 0.00 cubic ft, 0.00 cubic yards

Running total:

Cut volume : 82836.29 cubic ft, 3068.01 cubic yards

Specify Final Grid File Selection Dialog

Choose a grid (.GRD) file to process.

Extrapolate grid to full grid size (Yes/<No>)? *press Enter*

Volume Report Options dialog

-83.8	-82.1	-63.2	-31.8	-7.5 +0.4	+15.1	+37.2	+59.4
-81.2	-78.8	-61.2	-33.8	-11.6	-0.1 +10.6	+32.7	+54.9
-77.2	-73.9	-60.4	-38.3	-16.1	-0.6 +6.7	+28.2	+50.4
-71.7	-65.6	-53.3	-38.9	-18.7	-2.1 +3.6	+23.7	+45.8
-64.1	-53.8	-40.3	-28.7	-12.7	-1.1 +4.7	+21.8	+41.6
-53.2	-40.3	-26.8	-13.3	-2.1 +2.5	+14.3	+28.5	+44.4

This shows a grid drawn by *Plot 3D Grid File* and volume values drawn by the Draw Volume in Each Cell option of the Two Surface Volumes routine. Cut appears as negative and fill as positive. Notice that cells bordering cut and fill regions contain a little of both.

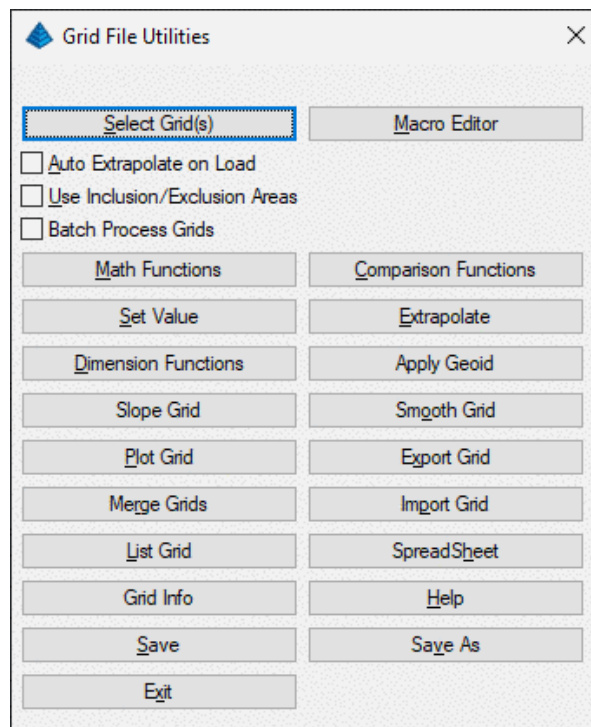
Pulldown Menu Location: Surface >> Volumes By Grid Surfaces

Keyboard Command: volcalc2

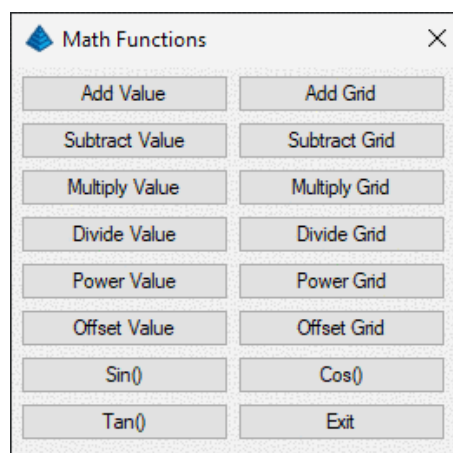
Prerequisite: Two grid files

Grid File Utilities

This command is used to modify and create grid files. The modifications can be done manually on a single grid, on multiple grids in a batch mode, or saved and rerun using the grid macros created with the macro editor. To modify manually, start by picking the Select Grid(s) button. There is an option to use inclusion and exclusion polylines to only modify the grid within/outside these perimeters. With this option active, the program will prompt for inclusion and exclusion polylines when a function is selected. Only grid cells inside the inclusion polylines will be modified. Grid cells inside the exclusion polylines will not be modified. If no inclusion and exclusion polylines are selected, then the entire grid will be modified. Each function is described below.

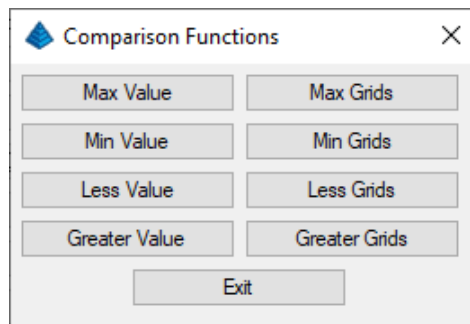


- **Select Grid(s):** This is the first step to load a grid. Usually a grid needs to be loaded before running a function. If Batch Process Grids is turned on, then multiple grids may be selected while holding down the Shift or CTRL buttons.
- **Auto Extrapolate On Load:** This will extrapolate values for any null or empty values in the grid as the grid is loaded.
- **Use Inclusion/Exclusion Areas:** If this is turned on, then the GFU function will only be applied within the selected inclusion polyline and outside the selected exclusion polyline.
- **Batch Process Grids:** When this option is turned on, GFU functions can be executed on many grids at once. It is recommended to move the grids to a backup directory, or create a copy of them, as the grids are over-written with the same name. The functions that cannot be batched are: Plot Grid, Merge Grid, List Grid, Import Grid, and Spreadsheet.
- **Math Functions:**



- **Add Value:** Adds an entered value to the grid values. (GridA + X)
- **Add Grid:** Adds one grid to another grid. (GridA + GridB)
- **Subtract Value:** Subtracts an entered value from the grid values. (GridA - X)
- **Subtract Grid:** Subtracts one grid from another grid. (GridA - GridB)
- **Multiply Value:** Multiplies the grid values by an entered value. (GridA * X)
- **Multiply Grid:** Multiplies the grid values by another grid. (GridA * GridB)
- **Divide Value:** Divides the grid values by an entered value. (GridA / X)
- **Divide Grid:** Divides the grid values by another grid. (GridA / GridB)
- **Power Value:** Raises the grid values to the specified power. (GridA ^X)
- **Power Grid:** Raises the grid values to another grid for the "power". (GridA ^GridB)
- **Offset Value:** Offsets the grid elevations perpendicular to the grid surface. Use a negative value to offset down.
- **Tan(), Sin() and Cos():** Applies these functions to the grid values which need to be in decimal degrees. (tan(GridA), sin(GridA), cos(GridA))

- **Comparison Functions:**



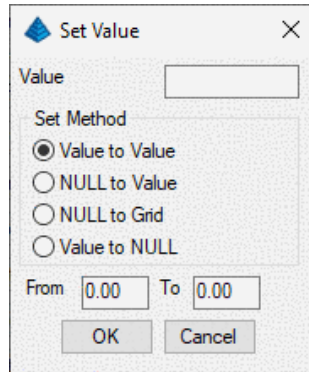
- **Max Value:** Compares a grid and a value and takes the Maximum value of either. This is a way to stop a grid from going negative, below zero.
- **Max Grids:** Compares a grid with another grid and takes the Maximum (higher) value of either.
- **Min Value:** Compares a grid and a value and takes the Minimum value of either. This is a good way to cap a grid off at a certain value so it never goes higher than the specified value.
- **Min Grids:** Compares a grid with another grid and takes the Minimum (lesser) value of either.
- **Less Value:** Asks for a value to compare and a value to assign and uses the following logic:
If GridA < compare_value then GridA = assign_value, otherwise no change
- **Less Grids:** Asks for a grid to compare and a grid to assign and uses the following logic:
If GridA < compare_GridB then GridA = GridC, otherwise no change
- **Greater Value:** Asks for a value to compare and a value to assign and uses the following logic:
If GridA > compare_value then GridA = assign_value, otherwise no change
- **Greater Grids:** Asks for a grid to compare and a grid to assign and uses the following logic:
If GridA > compare_GridB then GridA = GridC, otherwise no change

- **Set Value:** assigns the grid elevations to the user-specified value. Using inclusion and exclusion perimeters are usually required for this command. For example by using Set Value with the inclusion perimeter option,

you could set the grid values to 0.0 within the inclusion polyline for a strata thickness grid. The four options are:

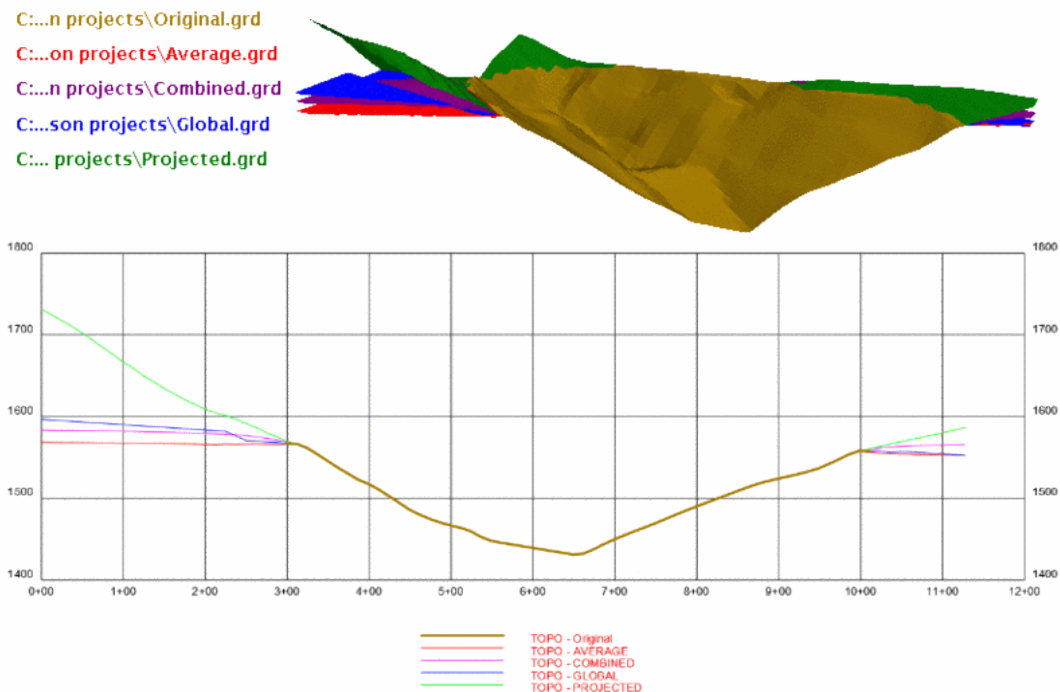
- Value to Value will set all values to one value
- Null to Value will set all Nulls to one value
- Null to Grid will set all Nulls to another specified grid

- Value to Null will set all values to Null.

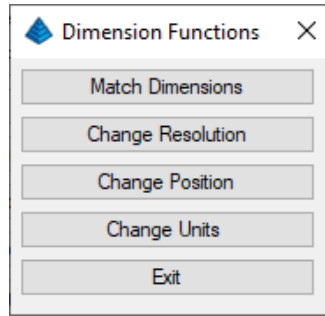


- **Extrapolate:** This function will assign values to null grid nodes by one of four methods. Global Trend finds the average slope and slope direction from the existing grid elevations and applies this slope to calculate the missing elevations. Average method calculates a grid elevation as the average of its nearest neighbors. This will create a very flat extrapolation. Projected method extends the trend at the edge (as opposed to finding an average slope of the entire surface like the Global option). Combined method uses a combination of both the Average and Projected methods.

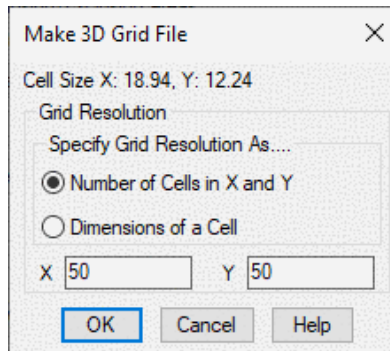
An example extrapolation of a surface using each method is shown below in 3D and in a profile view. The Original surface (brown color) is an example valley that only contains data in the center of the grid.



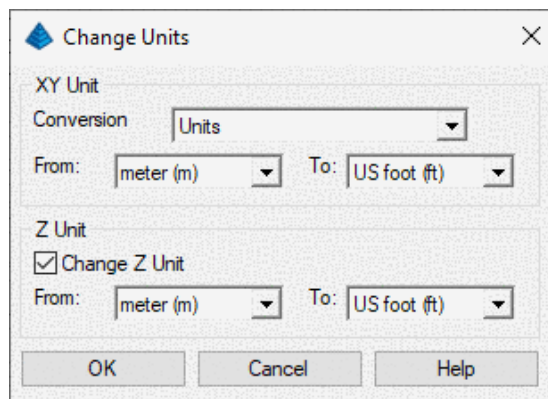
- **Dimension Functions:**



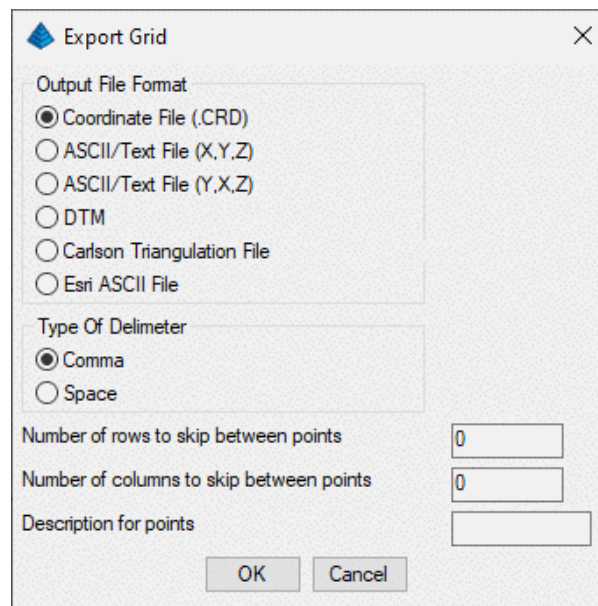
- **Match Dimensions:** Sets the grid position and resolution to match another grid file. The program will prompt for a grid file to get the position from. Certain commands require grids match position and resolution. Running this command will ensure grids will match.
- **Change Position:** This lets you change the lower left and upper right corners of the grid file. For example, you can use this routine to localize a grid file if you have a large grid for the entire site but are currently working on a smaller area. If the new position covers area outside the original position, any grid cells in this area will be assigned a null value. Otherwise the program uses the original grid values for the new grid position.
- **Change Resolution:** This changes the grid resolution (number or dimensions of grid cells). The program uses the original grid values for calculating the grid values at the new resolution. Enter a new value for X and Y number of cells or dimensions of cells.



- **Change Units:** Scales the grid X/Y and/or Z values to switch units such as meters to feet.



- **Apply Geoid:** Adjusts the grid elevations by the selected geoid. To use this function, the grid projection must be defined in the Settings > Drawing Setup command.
- **Slope Grid:** Converts a grid of elevations to a grid of slope values expressed as a percentage.
- **Smooth Grid:** This function has three smoothing method. The **Least-Squares** method applies a moving least-squares algorithm to adjust the grid elevations. The **Eliminate Spikes** method adjusts grid nodes with elevations too different (spikes) from neighbors within the **Search Radius**. The **Search Scaler** is used to identify spikes to adjust by using this scaler multiplied by the standard deviation of the grid values within the search radius. The **Interpolate Radius** is used to select the grid values for recalculating the grid value at the spike. **Quadratic Smoothing** uses neighboring nodes to adjust each grid node. This routine can be used to refine a grid so that the contours from the Contour from Grid routine appear smoother. Typically this adjustment is relatively small. To get more smoothing, run the routine more times.
- **Plot Grid:** This runs the Draw 3D Grid command.
- **Export Grid:** There are several choices for export options: Carlson Coordinate File (CRD), ASCII text as XYZ, ASCII text as YXZ, DTM, Digital Elevation Model (DEM), TIF, Carlson Triangulation (TIN) and Esri (ASC). There are two options for the ASCII delimiter, either a comma or a space. There is an option to skip a number of rows and columns between the exported points. When exporting into the Carlson CRD file, the description for the points is set at the bottom.



Export to DTM writes the current grid file to a DTM format text file. The format of this file is the following:

DTM 1.0 Header Line

test.dtm Name of file

51 Number of cells in X direction

51 Number of cells in Y direction

79442.4697 Lower left grid corner Y coordinate

14899.0326 Lower left grid corner X coordinate

0.0 Lower left grid corner Z coordinate

11.5618 Dimension of cell in X direction

7.0639 Dimension of cell in Y direction

1581.2612 Grid cell values starting from lower left, moving from left to right

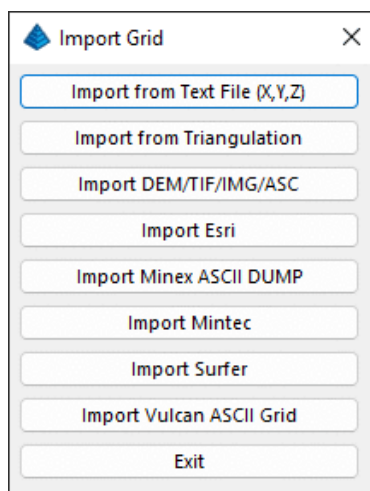
1580.8879

1580.3257

etc...

- **Merge Grids:** creates a grid file by merging together two existing grid files, grid1 and grid2. The current grid is grid1 and the program will prompt for a second grid. These two grids must overlap with the same location and resolution. The inclusion and exclusion perimeters apply to grid2 such that the merged grid will consist of grid2 cells within the inclusion perimeters and outside the exclusion perimeters and grid1 cells everywhere else. The result is stored in the current grid.
- **Import Grid:** There are several formats that may be imported.
 - Text File (ASCII): This function allows for various formats. The data can be comma or space separated or in fixed width columns.
 - Carlson Triangulation (TIN, FLT)
 - Digital Elevation Model (.DEM, .TIF .IMG or .ASC) such as from the USGS (US Geological Survey)
 - Esri (ADF)
 - Minex ASCII DUMP (ASC, CSV)

 - Mintec
 - Surfer (GRD both ASCII and Binary)
 - Vulcan ASCII Grid (ASC, CSV)



Import from Text File (X, Y, Z) creates a grid file from X Y Z data in any text file. There does not need to be a current grid file loaded since this routine will create a grid file. The text file should consist of one X Y Z coordinate per row with the first coordinate being the lower left grid corner and the last coordinate as the upper right grid corner. There are options for space or comma separated coordinates and for the order of the coordinates as either row (left to right) or column (bottom to top). The prompting will be as follows:

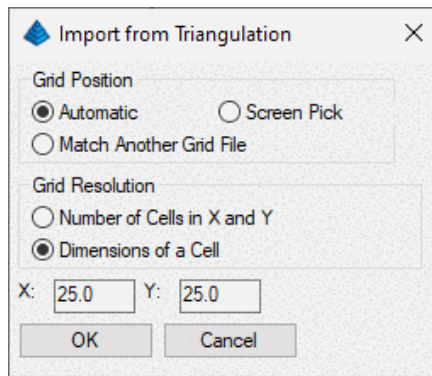
Separation type [<Space>/Comma/FixedWidth]?

Column number for X coordinate <1>:

Column number for Y coordinate <2>:

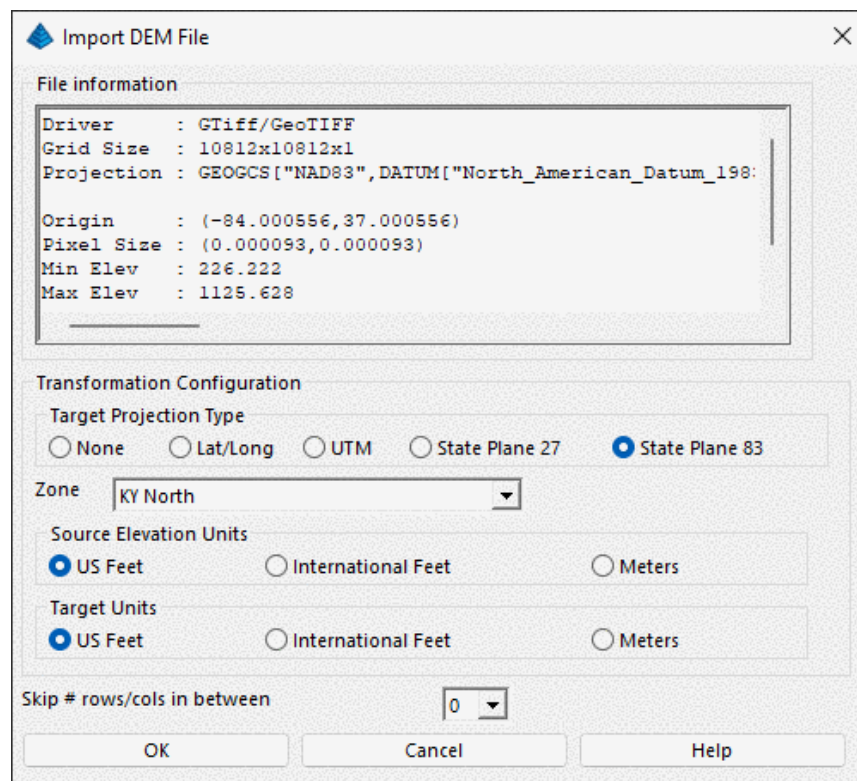
Column number for Z value <3>:

Import from Triangulation prompts user to select a tin or flt file and allows user to adjust grid position and resolution. The grid file is created with the same name in the same directory as the selected tin/flt file.



Import DEM/ESRI prompts the user to select a DEM, ADF, or ASCII ESRI grid file to be imported. The ESRI grid files can be created from ArcMap using the Raster To ASCII tool. If the file format is recognized, the program reads, and displays information about the source projection of the DEM/ADF/ESRI file, and allows the user to define a target projection for transforming the grid to a local coordinate system. the Source Elevation Units define the units of the Z values if they cannot be automatically detected. The Target Units will set the units for the X, Y, and Z values of the new grid. Note that if the Source Elevation Units and the Target Units use different units, the Z values will be scaled accordingly.

The "Skip every # rows/cols" option allows the user to reduce the size of the imported grid file. In the case of a DEM/ESRI files each pixel of elevation information of the DEM/ADF image data is interpreted as representing an elevated point located in the center of that pixel (not multiple points located on the edges of that pixel). In contrast, the "Import DEM/ADF File Dialog" will output the "Origin" of the DEM/ADF file in terms of the raster pixel extents. The output grid file is created with the same name in the same directory as selected DEM/ADF/ESRI file.

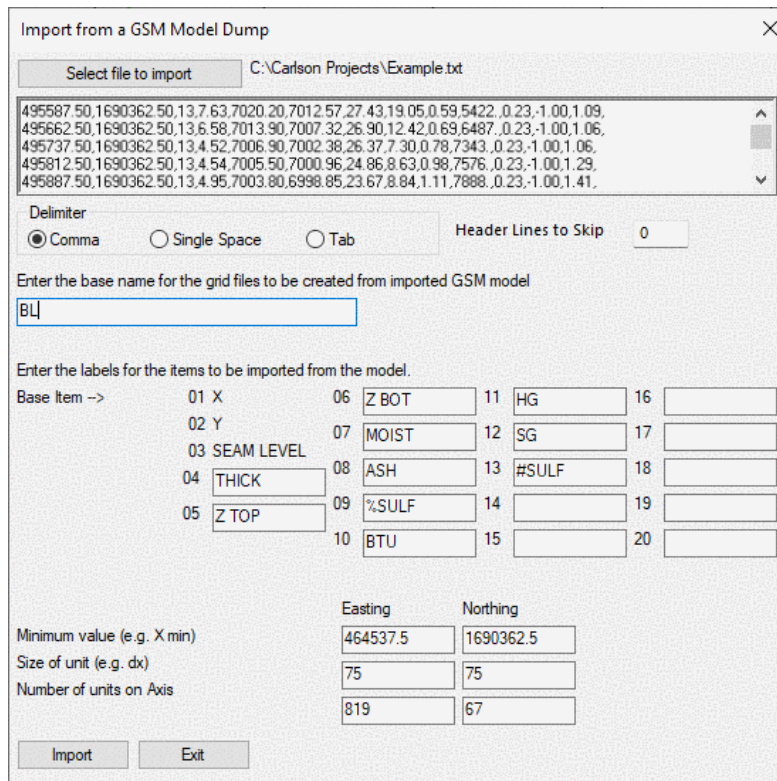


Import Minex ASCII DUMP allows user to import multiple Minex ASCII grids from a single .asc or .csv file. The imported grids will be automatically named according to the header information in the .asc/.csv file. An example

File Format is provided below.

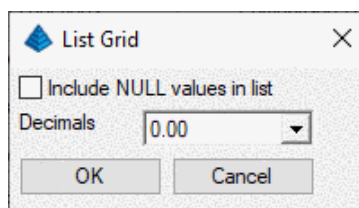
```
; origin      3051045.00    952377.25
; extent      5375.50      4042.63
; mesh        15.00       15.00
; rotation    0 00 00
; x   1   12
; y   13  12
; SeamRoof   25 10
; SeamFloor  35 10
; SeamThick  45 10
3051052.50    953764.75                920.76
3051052.50    953779.75                922.01
3051052.50    953794.75                923.29
3051052.50    953809.75                924.60
3051052.50    953824.75    928.38    925.93
3051052.50    953839.75    940.24    927.30
3051052.50    953854.75    941.63    928.69
3051052.50    953869.75    943.06    930.11
3051052.50    953884.75    944.52    931.56
3051052.50    953899.75    946.01    933.04
3051052.50    953914.75    947.54    934.55
3051052.50    953929.75    949.10    936.10
```

Import Mintec allows user to import Mintec GSM Model Dump as grd files. User is prompted to select GSM Model Dump (txt) file, which is then processed to determine minimum northing, easting (lower left corner), resolution and size of the grid. First three columns of the GSM model dump must represent the X, Y and SEAM LEVEL respectively, a base name for the grid files is specified along with name of the quality that each column represents. User can define up to 17 qualities. When the import button is pressed all the imported grids are created with the name "BASE NAME-SEAM LEVEL-BASE ITEM.grd" in the same directory as source dump file.



Import Surfer prompts to select a grd file from Surfer program and creates a grd file.

- **List Grid:** displays a list of the northing, easting and elevation of each grid corner. There is an option to Include NULL values in the list. A grid node will have no value, or a Null value (listed as None) if the grid node was outside the limits of the data during Make 3D Grid File.



Carlson Software Edit : C:\Users\Tfaulkner.CARLSON\...

File Edit Settings

Cell#	East (x)	North (y)	Elev (z)
55	540100.00	157050.00	1380.25
56	540200.00	157050.00	1404.41
57	540300.00	157050.00	1431.12
58	540400.00	157050.00	1440.00
59	540500.00	157050.00	1440.00
60	540600.00	157050.00	1440.00
61	540700.00	157050.00	1440.00
62	540800.00	157050.00	1440.00
63	540900.00	157050.00	1440.00
64	541000.00	157050.00	1436.77
65	541100.00	157050.00	1432.11
66	541200.00	157050.00	1428.90
67	541300.00	157050.00	1426.21
68	541400.00	157050.00	1424.52
69	541500.00	157050.00	1423.21
70	541600.00	157050.00	1421.83
71	541700.00	157050.00	1418.36
72	541800.00	157050.00	1411.76
73	541900.00	157050.00	1406.39
74	542000.00	157050.00	1398.28
75	542100.00	157050.00	1372.76
76	542200.00	157050.00	1355.95
77	542300.00	157050.00	1352.00
78	542400.00	157050.00	1351.04
79	542500.00	157050.00	1354.00

- **Spreadsheet:** displays the grid elevations in a row and column spreadsheet that is in the same layout as the grid file. Grid elevations can be edited in this spreadsheet and saved upon exiting the spreadsheet.

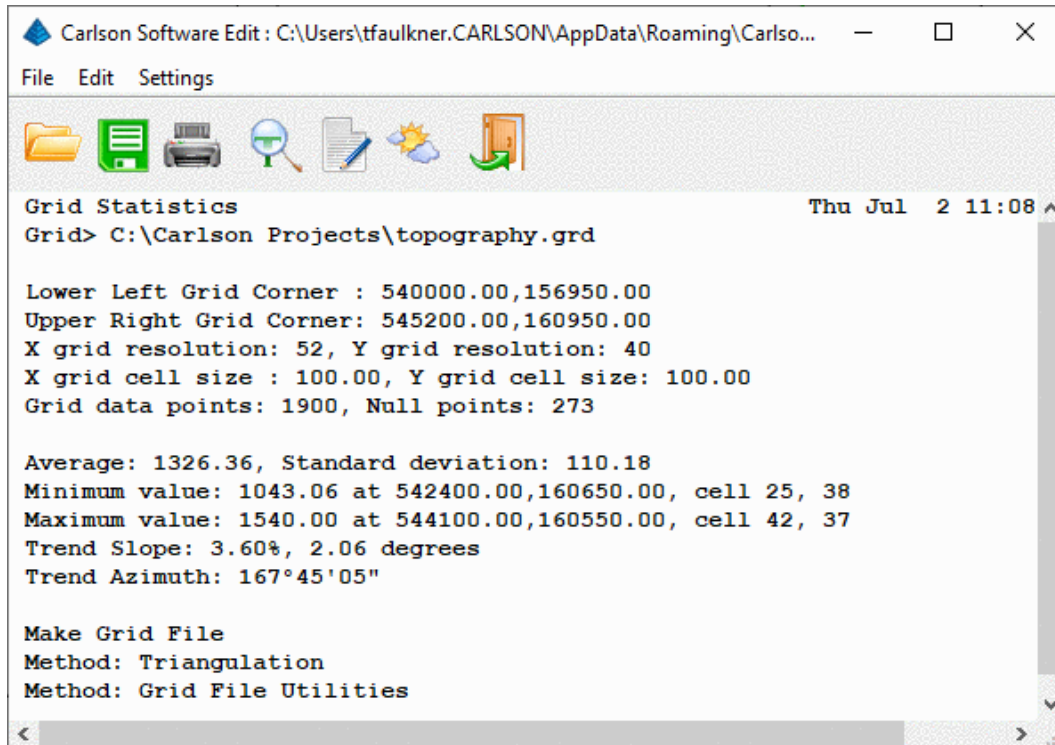
Grid Data Sheet

File

	19	20	21	22	23	24	25	
37	1172.9800	1187.6800	1196.2600	1181.6900	1153.4900	1114.9000	1065.3600	108
36	1222.6500	1240.8700	1252.8000	1216.0700	1173.0200	1130.7200	1078.8100	108
35	1282.9700	1298.5100	1281.1100	1238.4400	1186.5300	1141.9800	1093.7400	106
34	1340.6600	1340.0000	1306.0100	1253.2400	1196.3400	1154.9000	1107.0500	105
33	1343.7100	1322.3400	1285.5400	1249.6900	1191.9600	1163.1700	1115.7900	106
32	1295.3000	1255.6000	1222.1300	1194.7100	1170.8000	1142.8600	1106.1500	106
31	1237.7000	1210.0100	1184.3700	1159.1000	1133.6700	1100.0600	1070.4200	107
30	1190.9800	1172.8700	1148.5300	1119.7800	1088.9600	1101.7200	1115.7900	112
29	1154.5600	1137.0200	1110.5800	1102.7500	1119.2100	1135.5100	1151.2100	116
28	1119.9500	1110.2200	1128.2200	1141.1400	1153.6600	1169.9300	1194.0500	122
27	1140.7700	1150.7100	1158.5900	1172.5700	1188.4700	1212.5000	1243.2100	127
26	1170.0100	1180.8900	1199.6400	1225.0700	1236.0300	1264.4300	1301.2900	133
25	1229.3600	1246.8800	1258.4100	1281.1900	1292.6900	1311.1400	1351.5100	136

- **Grid Info:** This function displays information about the grid file. It is a form of Grid Statistics. The items it

displays are shown in the report below:



Macro Editor: The Macro Command Recorder allows you to create a script for grid modification and save these steps to a .gfu file. These macros can be rerun to repeat grid modifications when the source grids are modified, thus saving time on tedious manipulation. Macros can also be reviewed for accuracy, whereas manual manipulation of grid files cannot be checked with as much detail. When writing a macro, you essentially perform the normal grid manipulations using the functions on the left of the dialog. Although the prompts will be slightly different than manual manipulation, the steps will be automatically added to the text window. You can then copy/paste the text as needed to speed up the macro-writing process.

Quick notes on the macro editor:

- You may enter the script manually (typing each variable, function, etc), or you may use the function buttons on the left. Some functions are more easily written manually compared to using the function dialogs on the left.
- Function templates may be inserted using the "Template to Insert" droplist on the left. Using these templates, you can see an example of the syntax for each function.
- You may add comments to the script to allow for easier review. Comments are simply preceded by a semi-colon (;). With the exception of the PERIM, INCLU, and EXCLU functions, any text following a semi-colon will be seen as a comment (the program will not attempt to read it upon execution of the script)
- Most all function follow a syntax of RESULT=FUNCTION. For example, to evaluate the result of adding Grids A and B together to create Grid C, the function would be written as C=A+B.
- When a variable is first defined, the variable will take on the grid dimensions (cell size and lateral extent) of the right side of the equal sign. For Example, consider an example in which variable A is defined as a grid and B is not yet defined. If you write the equation B = A + 5, then variable B will cover the same lateral extent and will use the same grid cell size as variable A.

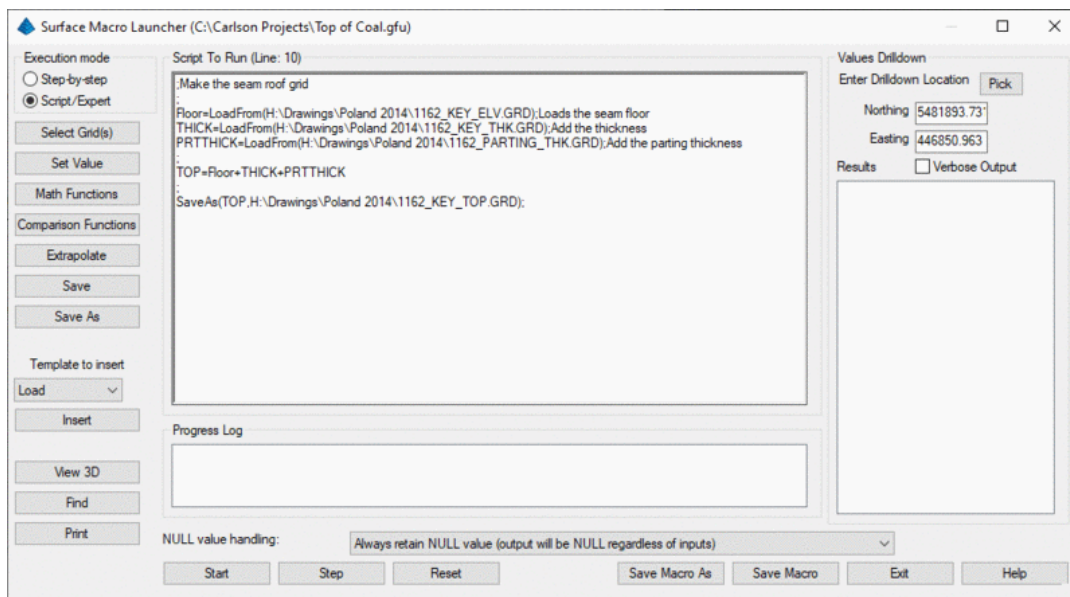
If two grid variables do not use the same grid dimensions, they can still be compared (note that this is not possible with manual manipulation of the grids). For example, if Grid A and Grid B cover only partially overlap, the equation B = A + 5 will allow Grid B to retain it's original dimensions, but it will only be updated where it overlaps A. In this way, you can avoid the intermediate step of settings grids to the same location/cell

size in order to compare them.

- When using the functions on the left side of the dialog (Math Functions, Comparison Functions, etc), you will sometimes be able to select existing grid variables from a dropdown list. In order for this list to be populated with the existing grid variables, you must first Start the macro, otherwise the program will not display the existing grid variables.
- When adding a step that involves another grid file, there are three options for storing this grid into the script. Use Grid Variable will use the grid assigned to the Grid Variable Name. Prompt for Grid File Name will bring up a grid file selection dialog each time the macro is run. Store the Grid File Name will save the specific grid file name into the macro.

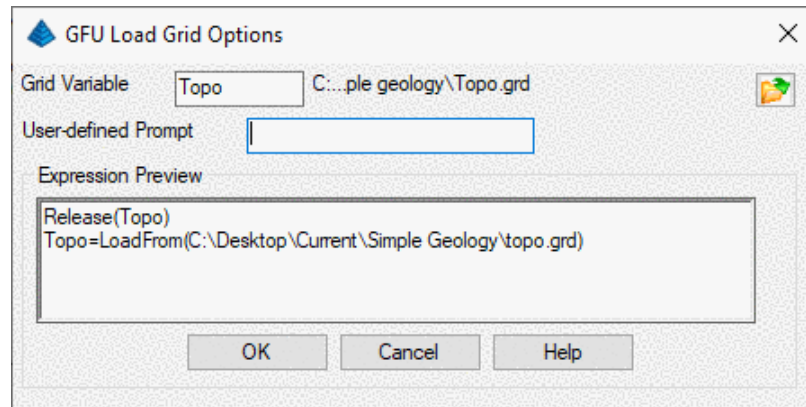
Surface Macro Launcher Details

The Surface Macro Launcher displays the GFU file name in the upper menu bar. This is an editor, and you can use basic functions like CTRL-X, CTRL-C & CTRL-V for cut, copy, & paste. GFU files also can be edited easily in any text editor, such as Notepad, WordPad or K-Edit. If there are any errors in the GFU during its execution, they will be displayed in the error log section when the macro runs. The Values Drilldown on the right side of the dialog allows you to inspect the calculations/grid variables at an actual Northing-Easting location, which is especially useful for checking that the results are correct. You may screen-pick a location by clicking the Pick button, or you may manually enter the Northing and Easting coordinates. As the macro runs, each variable will be display in the text readout below these options. If the Verbose Output option is disabled, only a summary of the grid variables will be displayed, with each variable updating as the macro runs. If the Verbose Output option is enabled, the results from each line of the macro will be displayed in the text readout.



- **Step-by-Step:** This execution mode is a simpler method and is intended to be executed immediately. It is similar to the previous windows simple "button pushing" to execute a function. These two modes have completely different dialogs when the functions are selected.
- **Script/Expert:** This mode stores the history and is a replayable history of the operations. It contains all of the higher level features and options, and is for the users who want to program it manually.

- **Select Grids:** In the Step-by-Step mode, the File Open dialog appears to select the file. In Script/Expert mode, the following dialog initially appears to add the expression to the macro for replaying. This function releases a variable if it is previously defined, and loads a grid file and assigns a variable to it. Another option is to enter an User-defined Prompt instructing the user on which grid to select at the time the macro is run. The dialog appears as shown. The preview shows how it will look when inserted into the macro.



- **Set Value:** This function sets the grid to new values based on the selected options. The Step-by-Step method has the following Set Functions shown in the first image below. The Expert method functions are shown in the second image. The expression #1 can either be set to a Value/Expression or to a New Grid file. The operation type contains four options of Set Value.

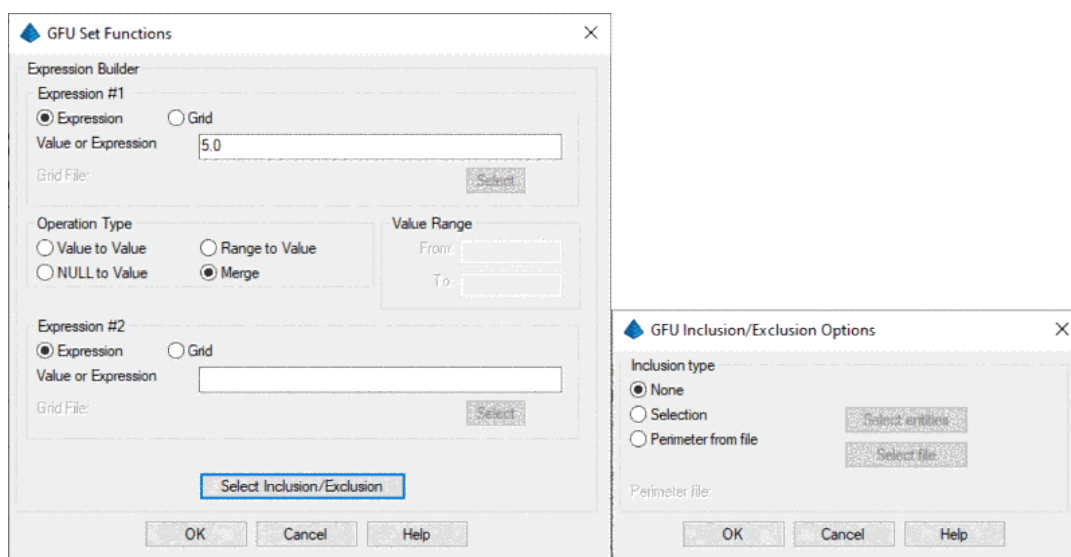
Value to Value (A=B for each point of A, sets the value to the value of B, Expr#1))

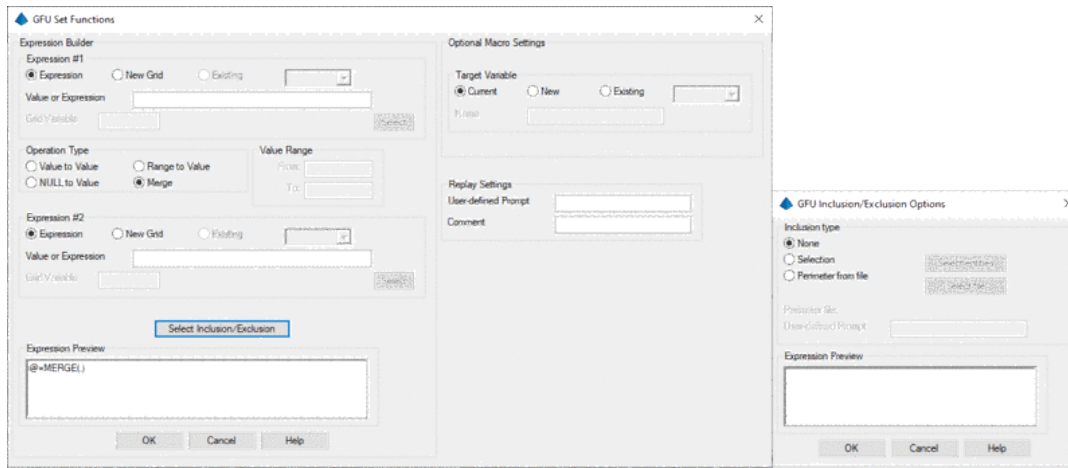
Null to Value (C=SET_NULL (A,B), For each point of C, sets the value to the value of A (Expr#1) if defined and the value of B (Expr#2) otherwise)

Range to Value (A=CHANGE_RANGE(B,C,D,E), for each point of A, if value of B (Expr#1) is between C and D, sets the value to value of E (Expr#2) or Null if not specified)

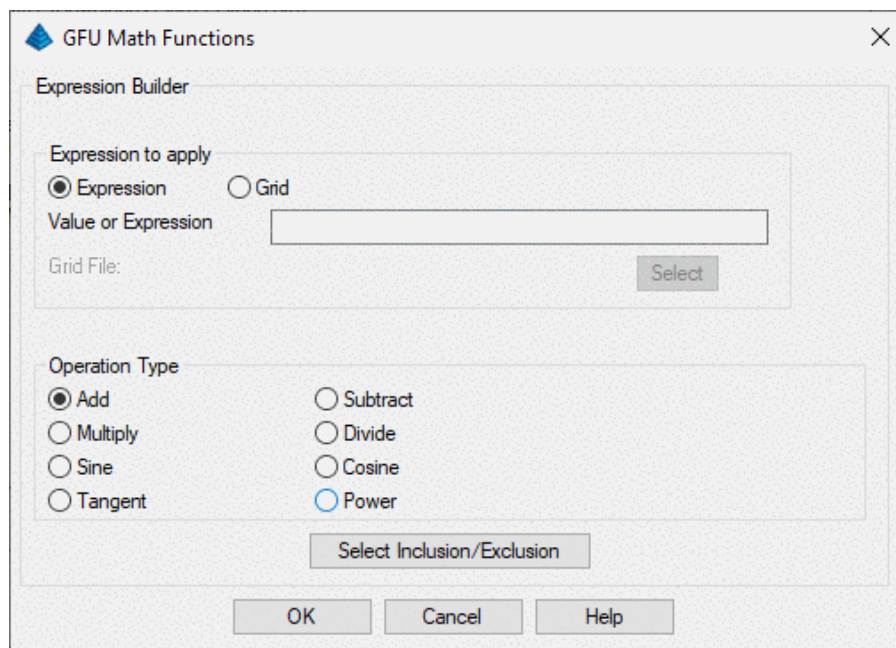
Merge (C=MERGE(A,B), for each point of C, sets the value to the value of B (Expr#2) if defined and the value of A (Expr#1) otherwise). Expression#2 is what the value will be set to, or to a new grid.

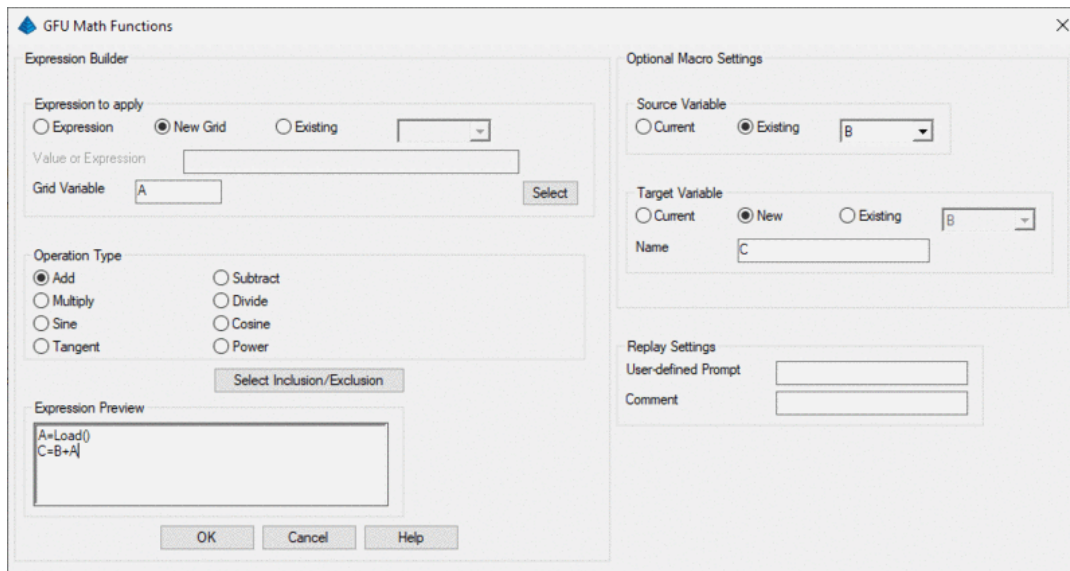
The Select Inclusion/Exclusion options can be set to None, Selection (manually select polyline), or a PLN polyline file. Note that PLN files may be created with the Export Polyline File command. The preview of the expression appears in the expert mode. These two modes appear the same in all the various functions. To set a grid to NULL, where the values fall within a range, use the following format: A=CHANGE_RANGE_VALUE(A,lowrange,highrange,NULL)



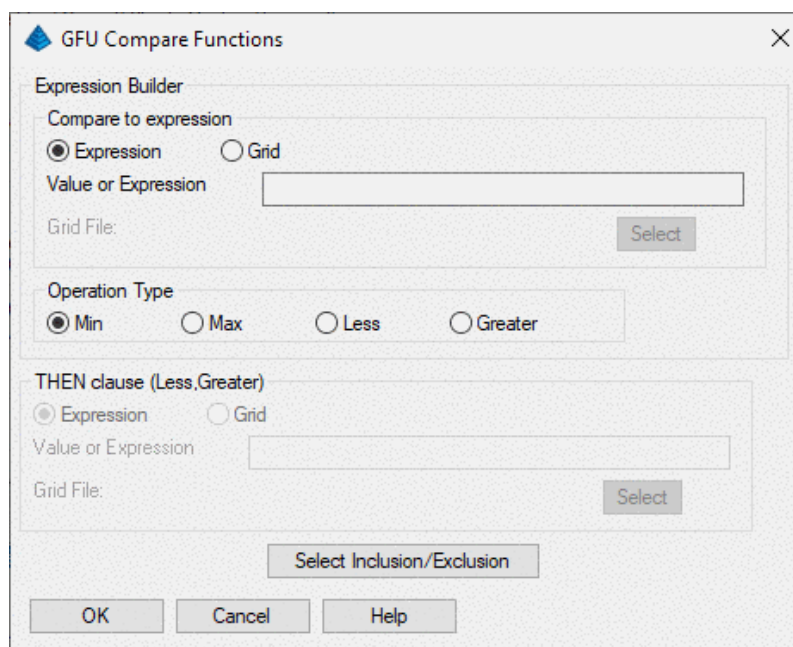


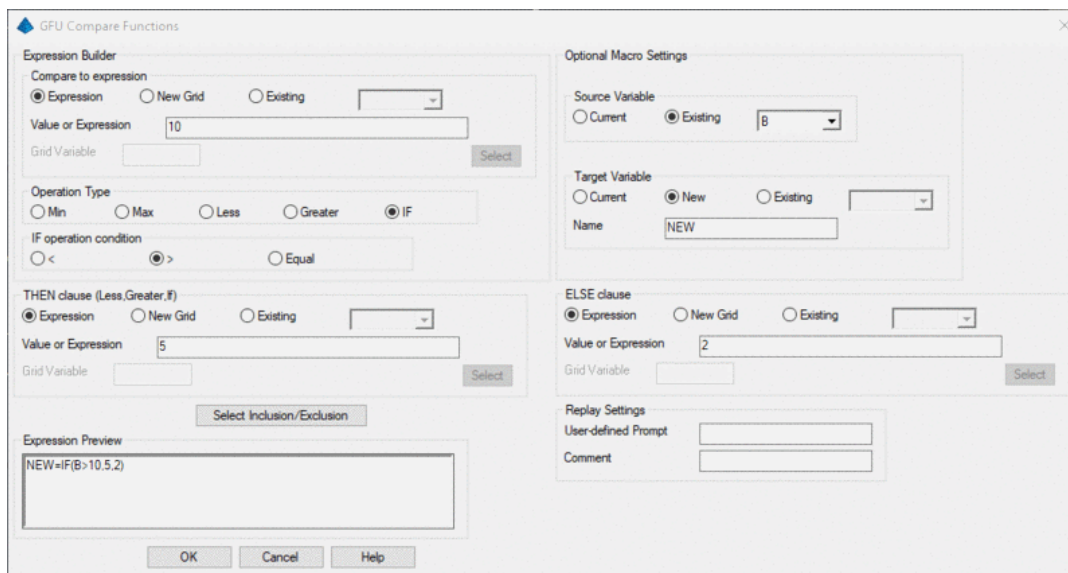
- Math Functions:** These functions apply the mathematical operations selected to the loaded grid. The Step-by-Step mode dialog is shown first, followed by the Script/Expert mode. The operations are Add, Subtract, Multiply, Divide, Sine, Cosine, and Power function.



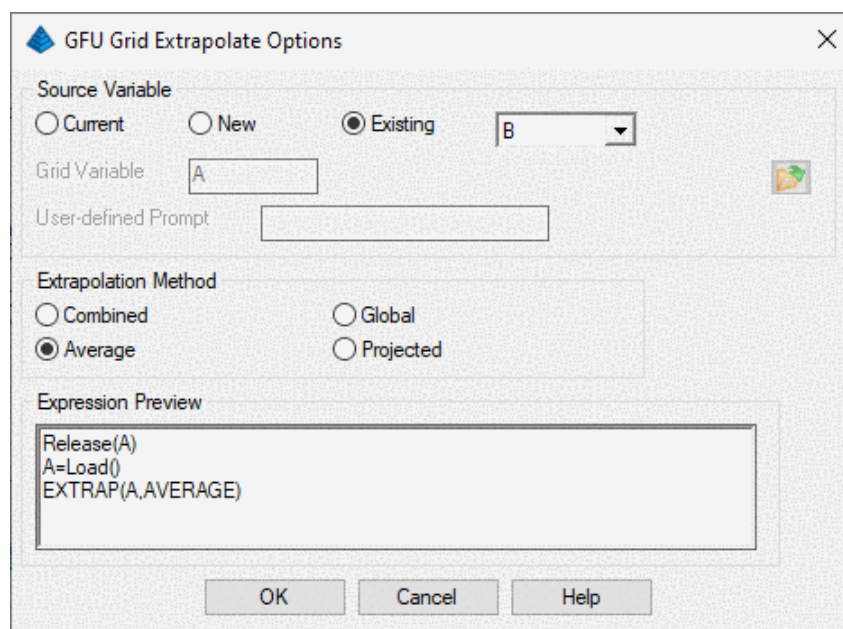
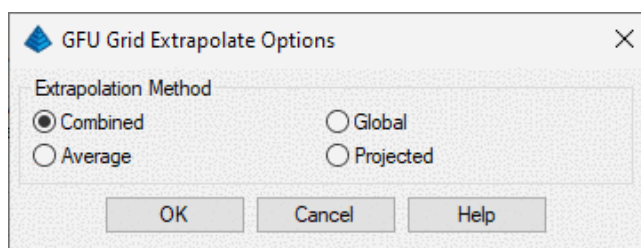


- Comparison Functions:** The Step-by-Step mode uses the Min, Max, Less, and Greater commands for grid editing. This is shown below in the first dialog. The second one shows the Script/Expert dialog that also contains the If Statement functions. The IF statement is the standard format of IF This, Then This, Else This. The Preview shows how it will appear in the GFU syntax.

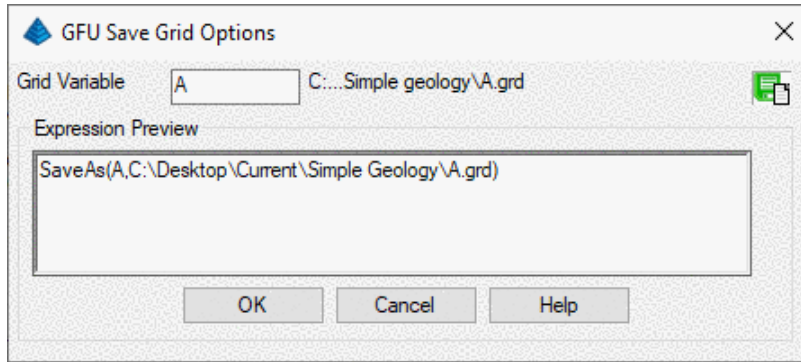




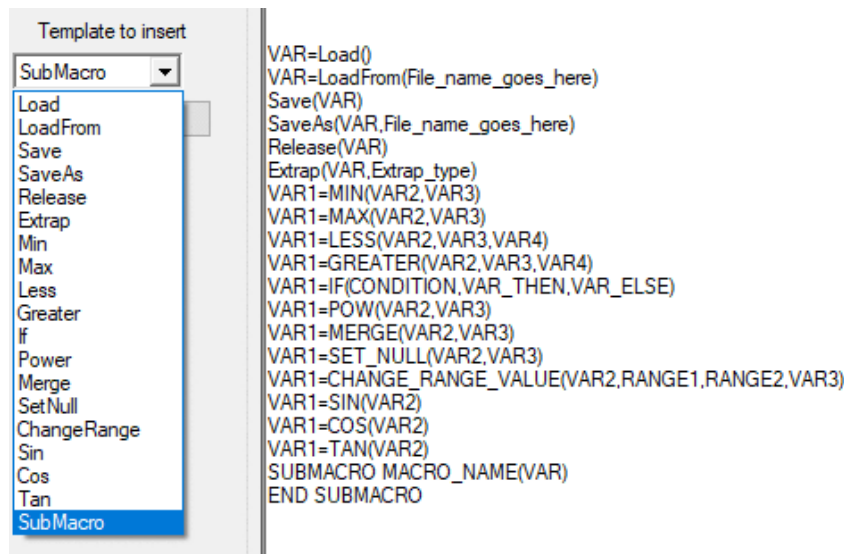
- **Extrapolate:** This function uses the four methods described above to extrapolate the grid to fill in any Null values. Details on each extrapolation method are provided above.



- **Save/Save As:** These commands add the syntax of saving to the macro. The variable can be the existing, or set to a new one here.



- **Template to insert:** This option allows for easy insertion of the syntax for the following functions.



- **Nested Submacros:** It is possible to define a macro for a certain function, and then call that macro and apply it to other variables in either the same macro, or in a different macro. Here are 3 examples of Submacros.

Submacro in the same macro that calls it. There are two submacros defined at the top, and then they are both called at the bottom, THICK2 and THICK.

```

SUBMACRO THICK()
A1=LOADFROM($PROJECTPATH\one.grd)
B1=LOADFROM($PROJECTPATH\two.grd)
C1=A1+B1;
SAVEAS(C1,$PROJECTPATH\three.grd)
END SUBMACRO
;
SUBMACRO THICK2(A,B,D)
C=A+B;
SAVEAS(C,D);
END SUBMACRO
;
A1=LOADFROM($PROJECTPATH\one.grd);
B1=LOADFROM($PROJECTPATH\two.grd);

```

```
THICK2(a1,B1,$PROJECTPATH\three_b.grd);
THICK()
```

Submacro in a different macro that calls it. The first step is to define the submacros in their own gfu, this one contains THICK and THICK2 and is saved as mymacros.gfu.

```
SUBMACRO THICK()
A1=LOADFROM(C:\downloads\one.grd);
B1=LOADFROM(C:\downloads\two.grd);
C1=A1+B1;
SAVEAS(C1,C:\downloads\three.grd);
END SUBMACRO;
;_____
SUBMACRO THICK2(A,B,D)
C=A+B;
SAVEAS(C,D);
END SUBMACRO;
```

INCLUDE: To include a previously defined macro in another macro, the INCLUDE function is used. Once mymacros.gfu is created, then it can be loaded (with INCLUDE) and referenced in another, separate GFU macro.

```
INCLUDE($PROJECTPATH\mymacros.gfu);

A1=LOADFROM($PROJECTPATH\one.grd);
B1=LOADFROM($PROJECTPATH\two.grd);
THICK2(a1,B1,$PROJECTPATH\three_b.grd);
THICK();
```

Here is an example, with comments, where the bottom elevation and the thickness are added together to get the roof, for 6 seams. The Submacro is defined by the equation just once. Then it is called and used 6 times at the bottom of the macro.

```
;Add the Floor and Thickness to get the seam Roof.
;Coal seams to apply this to are: C1,C2,C3,C4,C5,C6
;_____
;LOAD THE GRIDS INTO MEMORY
;
C1_KEY_ELV=LOADFROM(C:\Carlson Projects\C1_KEY_ELV.GRD)
C1_KEY_THK=LOADFROM(C:\Carlson Projects\C1_KEY_THK.GRD)
C2_KEY_ELV=LOADFROM(C:\Carlson Projects\C2_KEY_ELV.GRD)
C2_KEY_THK=LOADFROM(C:\Carlson Projects\C2_KEY_THK.GRD)
C3_KEY_ELV=LOADFROM(C:\Carlson Projects\C3_KEY_ELV.GRD)
C3_KEY_THK=LOADFROM(C:\Carlson Projects\C3_KEY_THK.GRD)
C4_KEY_ELV=LOADFROM(C:\Carlson Projects\C4_KEY_ELV.GRD)
C4_KEY_THK=LOADFROM(C:\Carlson Projects\C4_KEY_THK.GRD)
C5_KEY_ELV=LOADFROM(C:\Carlson Projects\C5_KEY_ELV.GRD)
C5_KEY_THK=LOADFROM(C:\Carlson Projects\C5_KEY_THK.GRD)
C6_KEY_ELV=LOADFROM(C:\Carlson Projects\C6_KEY_ELV.GRD)
C6_KEY_THK=LOADFROM(C:\Carlson Projects\C6_KEY_THK.GRD)
;_____
;DEFINE THE SUBMACRO EQUATION
;
```


SUBMACRO ROOF(X,Y,Z)

TOP=X+Y

SAVEAS(TOP,Z)

END SUBMACRO

;
; _____

ROOF(C1_KEY_ELV,C1_KEY_THK,C:\Carlson Projects\C1_KEY_TOP.grd)

ROOF(C2_KEY_ELV,C2_KEY_THK,C:\Carlson Projects\C2_KEY_TOP.grd)

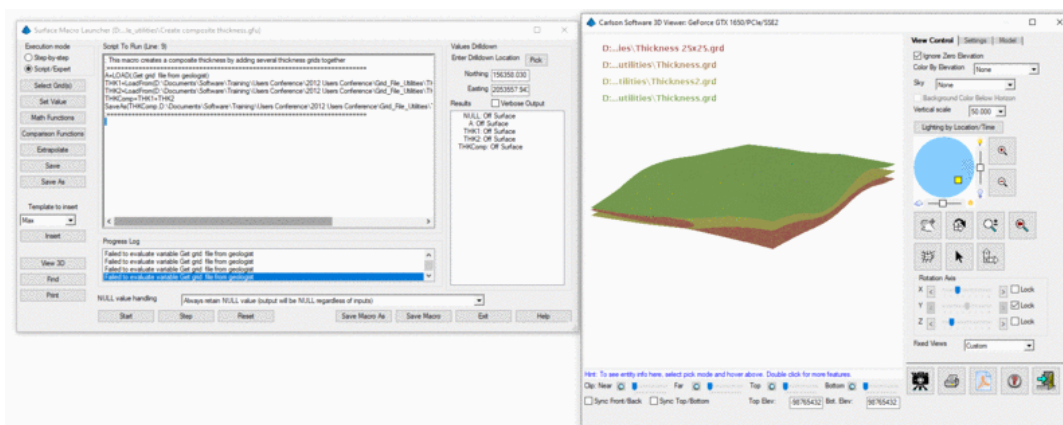
ROOF(C3_KEY_ELV,C3_KEY_THK,C:\Carlson Projects\C3_KEY_TOP.grd)

ROOF(C4_KEY_ELV,C4_KEY_THK,C:\Carlson Projects\C4_KEY_TOP.grd)

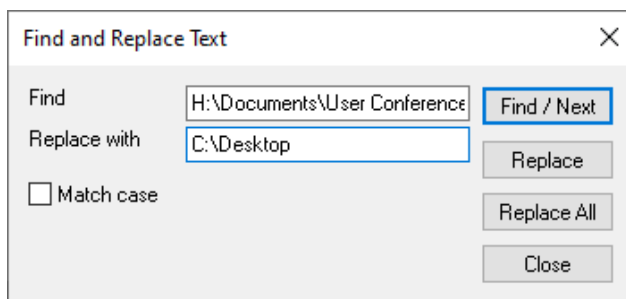
ROOF(C5_KEY_ELV,C5_KEY_THK,C:\Carlson Projects\C5_KEY_TOP.grd)

ROOF(C6_KEY_ELV,C6_KEY_THK,C:\Carlson Projects\C6_KEY_TOP.grd)

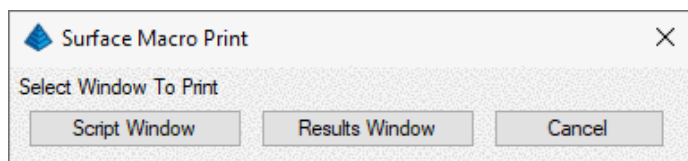
- **View 3D:** Using this option brings up the 3D Surface Viewer that steps through live all of the grids and displays them in 3D. This can be left open and moved to the side, or to another monitor to see all the surfaces and how they are modified, in a 3D view.



- **Find:** This is a Find, or Find and Replace to search the GFU text and make edits. It is useful if a file path has changed and all need to be reset to somewhere else.



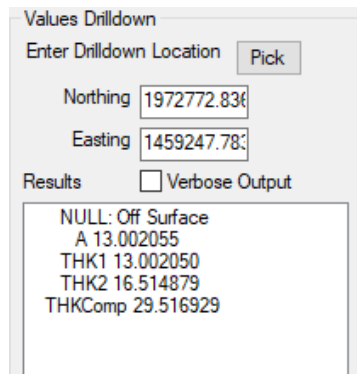
- **Print:** There are two options for printing. The Script Window prints out the entire GFU syntax. The Results Window prints out



- **NULL Value Handling:** This option determines how null values will be handled in the grid calculations. The "Always retain NULL Value (output will be NULL Regardless of inputs)" option will force any calculations with NULL inputs to create a NULL output. The "Treat NULL as 0.0 if one or more inputs are non-NULL";

Retain NULL if all inputs are NULL)" option will replace null values with zeros for calculations, but will retain null values in the output if all inputs are null.

- **Start/Step/Reset:** These buttons allow for individual stepping through the macro. It will advance only one line at a time with the Step. Start will run the entire macro. Reset will begin back at the beginning.
- **Save Macro As/Save Macro:** This saves the GFU macro either as its original name, or as a new name with Save As.
- **Values Drilldown:** The Values Drilldown is a good method for error checking on the macro. Use the Pick button to select a spot in plan view to fill in the Northing and Easting boxes. Then when the GFU is executed, the results of each line will be displayed. The Verbose Output will show the value of each line L1, L2, etc., instead of overwriting the variable each time it is encountered.



Variables

Variables in the GFU can be any keywords providing meaningful identification of the data loaded. A variable can be either just a value or constant; or most commonly a surface (Grid or TIN). Individual macro lines typically have one of the following forms:

Variable1=Variable2

Variable1=Expression

Variable1=Function(Expression1,Expression2)

Whenever new variable name is encountered on the left side of the equation, the new variable will be created. The program will use its knowledge of the right side of equation to define a type of the new variable. For example:

A=1.0 Variable A will be just a value

A=LoadFrom(abc.grd) Variable A is a grid loaded from file

B=A Variable B is same variable type as A

B=(1+C+A)/D Variable B will be 1+C+A, all divided by D. C and D will need to be defined somewhere before this line in the GFU.

Important!: Once a variable is defined, its type (like grid location and resolution) does not change. Therefore, for the existing variable A, the following expression:

A=Min(B,C)

is interpreted in the following way: for every point of the existing surface A calculate values of surface B and C and use the smaller of the two values to set new value of point elevation on surface A.

The following operators may be used in the expressions:

+, -, *, / - regular arithmetic operators

<, >, =, ! (not) - logic operators

— (or), & (and) - binary operators

Changing the scope of the equation

The scope of any line of the script can be modified by adding one of the following inclusion/exclusion operators:

A=Min(B,C);INCLU(handle_here) - will use AutoCAD entity with specified handle for the inclusion

A=Min(B,C);EXCLU(handle_here) - will use AutoCAD entity with specified handle for the exclusion

A=Min(B,C);PERIM() - will prompt for inclusion/exclusion at run-time

A=Min(B,C);PERIM(file_name) - will use specified file with inclusions/exclusions

Multiple inclusions or exclusions can be appended in this manner. Only points of target surface (A) passing inclusion/exclusion filter will be evaluated.

For custom, user define prompting, the following text should be used:

PERIM(,Prompt goes here) for user defined interactive inclusion and exclusion selection in CAD

The following script functions are currently defined:

Macro functions (performing operations on the entire surface at once)

- LOAD() Prompt user for the file to load. Returns a variable.
- LOAD(,Prompt goes here) for user defined prompting
- LOADFROM(string) Load surface from file. Grids (GRD) and TINs (FLT, TIN) are supported. Returns a variable.
- SAVE(Variable) Saves surface back to original file.
- SAVEAS(Variable,FileName) Saves surface into a file with given name.
- RELEASE(Variable) Releases memory used by a surface and undefines it for further use.
- EXTRAP(Variable[,Type])

Micro functions (taking effect on point by point basis as controlled by left side of the equation) Expressions can be complex ones with variables, value and functions

- MAX(Expression1,Expression2) Sets value to larger of two expressions evaluated.
- MIN(Expression1,Expression2) Sets value to smaller of two expressions evaluated.
- LESS(Expression1,Expression2,Expression3) If result of Expression1 is less than Expression2 then result is Expression3. Otherwise the source point is not changed. If Expression3 is not specified value is set to NULL.
- GREATER(Expression1,Expression2,Expression3) If result of Expression1 is greater than Expression2 then result is Expression3. Otherwise the source point is not changed. If Expression3 is not specified value is set to NULL.
- IF(Expression1,Expression2,Expression3) If Expression1 (can be logic expression like $(A+B)>C$ or $A=B$ or $A!B$ 'not equal') not 0 then result is Expression2, otherwise it is Expression3.
- POW(Expression1,Expression2) Result is value of Expression1 in power of Expression2
- MERGE(Expression1,Expression2) If Expression2 is valid at a point, then result is that value, otherwise it is value of Expression1
- SET_NULL(Expression1,Expression2) If Expression1 is valid at a point, then result is that value, otherwise it is value of Expression2
- CHANGE_RANGE_VALUE(Expression1, Range1, Range2, Expression2) If Expression1 is a valid point and its value is greater than equal to Range1 and less than equal to Range2, then result is Expression2. If Expression2 is not specified value is set to NULL.
- TIN_INTERSECT(Expression1,Expression2) Expects both Expression 1 and Expression 2 to be TIN's. This will add vertices to Expression 1 where Expression 2 has vertices. In this way, one TIN can be made to have vertices at the same XY locations as another TIN. Note that this function does not need to be preceded by "Variable=".

Here is an example of a complex IF statement used for coal recovery based on thickness of the seam:

```
COALTHK=LoadFrom(C:\Carlson Projects\Grids\C40_THK.GRD);
ROM_COAL=COALTHK
ROM_COAL=if((COALTHK<2)—(COALTHK=2),COALTHK - (COALTHK * 0.50),ROM_COAL)
ROM_COAL=if(((COALTHK<4)—(COALTHK=4))&(COALTHK>2),COALTHK      -      (COALTHK      *
0.10),ROM_COAL)
ROM_COAL=if((COALTHK<7)&(COALTHK>4),COALTHK - (COALTHK * 0.075),COALTHK - (COALTHK
* 0.05))
SaveAs(ROM_COAL, C:\Carlson Projects\Grids\C40_ROM_THK.GRD)
```

Pulldown Menu Location: Surface and Grids

Keyboard Command: GFU

Prerequisite: Make a grid (.GRD) file with the *Make 3D Grid File* command.

Merge Grid Files

This command creates a grid file by merging together two existing grid files, grid1 and grid2. The current grid is grid1 and the program will prompt for a second grid. These two grids must overlap with the same location and resolution. The inclusion and exclusion perimeters apply to grid2 such that the merged grid will consist of grid2 cells within the inclusion perimeters and outside the exclusion perimeters and grid1 cells everywhere else. The result is stored in the current grid.

No elevations are calculated on grid cells that extend beyond the extent of the data. Extrapolation can be used to calculate elevations for the grid cells that are beyond the data limits. The prompt *Extrapolate grid to full grid size?* shows when there are grid cells with no elevation in a grid (.GRD) file. Extrapolation fills in all the grid cells. The method to extrapolate uses a safe calculation that tends to average out or level the extrapolated values. So extrapolated grid areas are not as accurate as grid areas within the limits of the data.

Prompts

Select Source Grid 1 File Dialog (*file select dialog*)

Reading cell > 93058

Extrapolate grid to full grid size [Yes/<No>]? *press Enter*

Select Source Grid 2 File Dialog (*file select dialog*)

Reading cell > 62137

Extrapolate grid to full grid size [Yes/<No>]? *press Enter*

Overlap method: Hold grid 1, replace with grid 2 or average [Hold/Replace/<Average>]? *press Enter*

Specify inclusion and exclusion areas for grid 2. Grid 1 used everywhere else.

Select the Inclusion perimeter polylines or ENTER for none. *select perimeter(s) or press Enter*

Select objects: *press Enter* to conclude selection

Select the Exclusion perimeter polylines or ENTER for none. *select perimeter(s) or press Enter*

Select objects: *press Enter* to conclude selection

Merged Grid File to Write Dialog (*new file select dialog*)

Pulldown Menu Location: Surface >> Modify Grid File

Keyboard Command: mergegrid

Prerequisite: Two grid (.GRD) files

Merge Elevation for Zero Thickness

This command merges two elevation grid files and outputs the result to a new grid file. The purpose of this command is to "pancake" together the top and bottom elevation grids for a strata so that the thickness is zero in specified areas. The two input elevation grid files should have the same grid position and resolution. One represents the top of strata elevation and the other represents the bottom elevation. The output grid will have the top grid elevations except in areas of zero thickness where the output grid will have the bottom grid elevations. This output grid can then be used as the new modified top elevation grid. StrataCalc routines will have the same elevation for the top and bottom of the strata in the areas of zero thickness.

After prompting for the grid file names, the program prompts for the inclusion and exclusion polylines. These should be closed polylines where everything inside the inclusion polylines and outside the exclusion polylines has zero thickness. For example, you can draw a closed polyline for an inclusion area around an area where the strata pinched out, where you want it to be zero, or has already been mined.

Prompts

Select TOP Elevation Grid File dialog

Select BOTTOM Elevation Grid File dialog

Specify New Elevation Grid File dialog

Specify inclusion and exclusion areas for zero thickness.

Select the Inclusion perimeter polylines or ENTER for none:

Select objects: *select a closed polyline*

Select the Exclusion perimeter polylines or ENTER for none.

Select objects: *press Enter*

Pulldown Menu Location: StrataCalc

Keyboard Command: mergegrid2

Cleanup Grid Area

This routine takes a grid file and a closed 3D polyline, and combines the two. The elevations of the closed polyline are assigned to the grid elevations. Then the polyline is modeled and the resulting surface is applied to the original grid inside the polyline area. The resulting grid is then saved to a new grid file.

Prompts

Select a perimeter polyline:

Reading cell> 12604

Adding intersections with grid to polyline...

Assigning elevations...

Triangulating...

Setting new grid elevations...

Reading cell> 12604

Pulldown Menu Location: StrataCalc

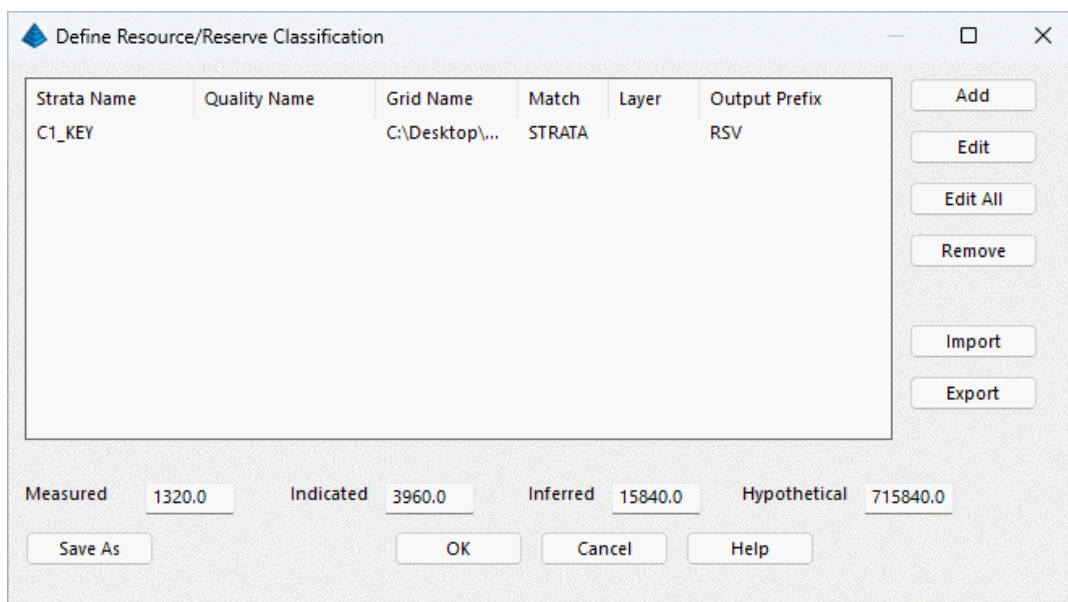
Keyboard Command: cleanupgrid

Resource-Reserve Classification

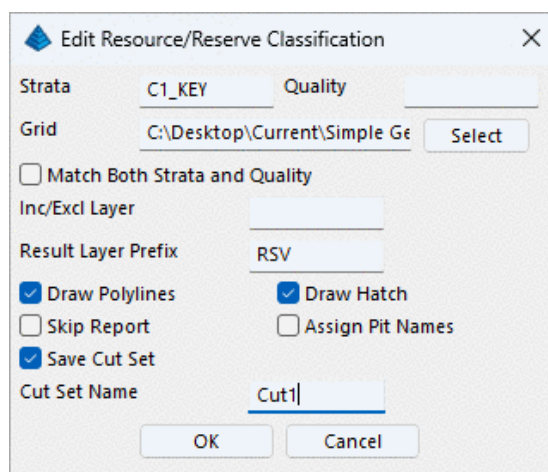
The Resource/Reserve Classification command provides a report based on the area of each zone. The tonnage associated with each zone (Measured, Indicated, Inferred, and Hypothetical) can then be calculated using the Surface Mine Reserves command (found in either the Geology or Surface Mining module). The four classification zones were previously been based off of the following distances from a drillhole, but please note that it is **not** appropriate to blindly assign classification based on these distances under standard reporting requirements such as

- Measured: $x < 1320$ ft (1/4 mile)
- Indicated: $1320 \text{ ft} < x < 3960$ ft (1/4 mile to 3/4 mile)
- Inferred: $3960 \text{ ft} < x < 15840$ ft (3/4 mile to 3 miles)
- Hypothetical: $x > 15840$ (3 miles)

After creating a .rsv file, the below dialog will appear. Here you can add new strata layer to the list.



When adding a new strata to the list, the below dialog will appear.

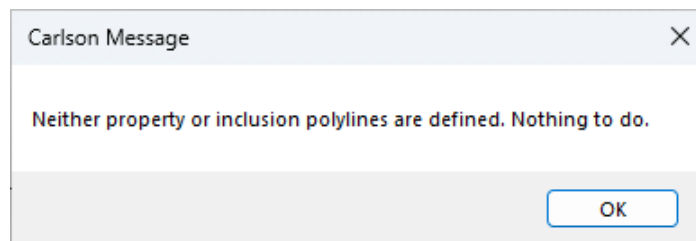


- **Strata:** This sets the name of the strata to analyze. Note that this must perfectly match the strata/bed name in the drillhole. When using bed names, it is important to include the Bed Name and the grouping suffix. For

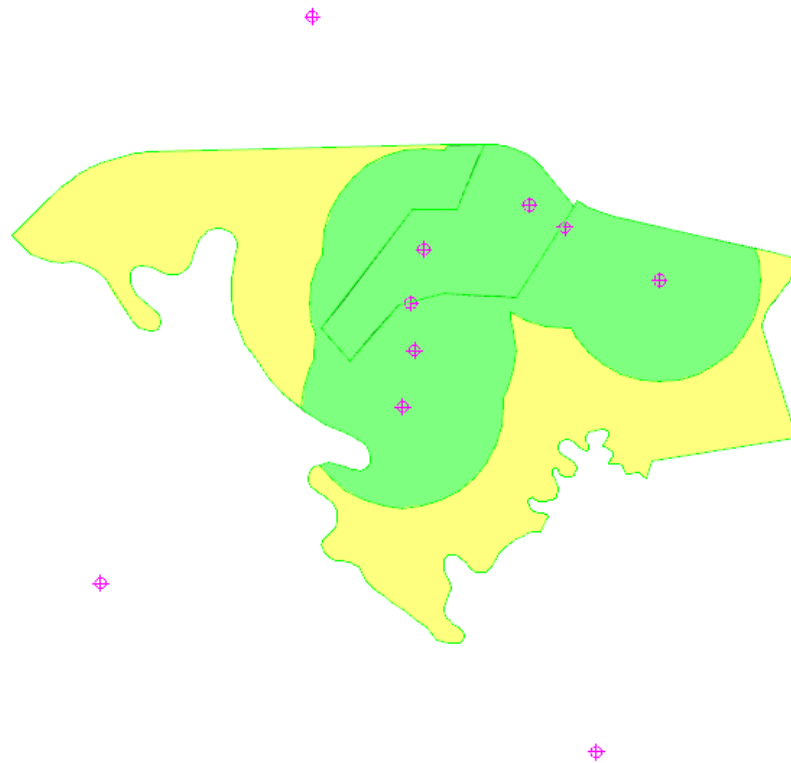
example, the above example shows the name that must be entered for the Key portion of the C1 bed (C1_KEY).

- **Quality:** This is the quality attribute to analyze, such as thickness, BTU, etc. It is not required to enter a value for this field.
- **Grid:** This is the grid name that has been created from the drillholes. It will compare these modeled values with the occurrences and spacing in the drilling.
- **Match Both Strata and Quality:** This will make sure that the quality and strata name both are defined in a drillhole. If just the strata name is there, but no quality sample, then it will not be used as a sample point that is measured.
- **Inc/Excl Layer:** These are the layers that the perimeters are drawn in.
- **Result Layer Prefix:** This is a prefix to add in front of the output layer names. The program will create layer names such as "_A_MEASURED" and "_B_INDICATED". If the prefix is set to "RSV", then the resulting layers would be "RSV_A_MEASURED" and "RSV_B_INDICATED".
- **Draw Polyline:** This will draw the perimeter polylines that contain the hatching for the four classifications.
- **Draw Hatch:** This will draw the hatching inside the perimeters and boundaries of classifications.
- **Skip Report:** This option just will only generate CAD graphics and mapping only - the report will not be generated for this strata when this option is enabled.
- **Assign Pit Names:** This option will give a Pit Name to each perimeter, so it may be used for other volume calculations with commands such as Surface Mine Reserves.
- **Save Cut Set:** This option will create a named cut set for the boundaries, which may be used with Surface Mine Reserves or the Timing commands to further divide volumes/tonnage into relevant categories. Note that when using Surface Mine Reserves or the Timing commands, you must enable the relevant options to report by Property Boundaries in order to use the cut set.

After adding strata to the list and clicking OK, you will be prompted to select drillholes and inclusion boundaries. If no boundaries are selected, the program will automatically use polylines tagged as property boundaries. If no inclusion boundaries are selected, and no property boundaries exist, the command will exit and the below message will appear.



The results are shown as colored maps layered by the chosen quality, with color changing based on the corresponding reserve class.



An example report is show below:

Reserve Classification Wed Jun 04 15:40:18 2014

Owner Harvey
 Property BR549
 Strata name WK6_KEY
 Quality Thickness
 Quality Grid C:\2012 UG\Reserve Classification\WK6_KEY_THK.GRD
 Average Value 2.9
 Total Area (sq.ft) 28953106.4
 Measured Area (sq.ft) 12898291.2
 Measured Acres 296.1
 Measured Value 2.731
 Indicated Area (sq.ft) 16054819.9
 Indicated Acres 368.6
 Inferred Area (sq.ft) 0.0
 Inferred Acres 0.0
 Hypoth. Area (sq.ft) 0.0
 Hypoth. Acres 0.0
 Indicated Value 2.974

Owner Lloyd
 Property ABC
 Strata name WK6_KEY
 Quality Thickness
 Quality Grid C:\2012 UG\Reserve Classification\WK6_KEY_THK.GRD
 Average Value 2.6
 Total Area (sq.ft) 3850616.2

Measured Area (sq.ft) 3850616.2
Measured Acres 88.4
Measured Value 2.628
Indicated Area (sq.ft) 0.0
Indicated Acres 0.0
Inferred Area (sq.ft) 0.0
Inferred Acres 0.0
Hypoth. Area (sq.ft) 0.0
Hypoth. Acres 0.0
Indicated Value

Prompts

Select Reserve File dialog

Reserve Classification dialog

Select Drillholes to process: *select drillholes to use in calculations*

Select Inclusion polyline:

Report Formatter Dialog

Pulldown Menu Location: StrataCalc

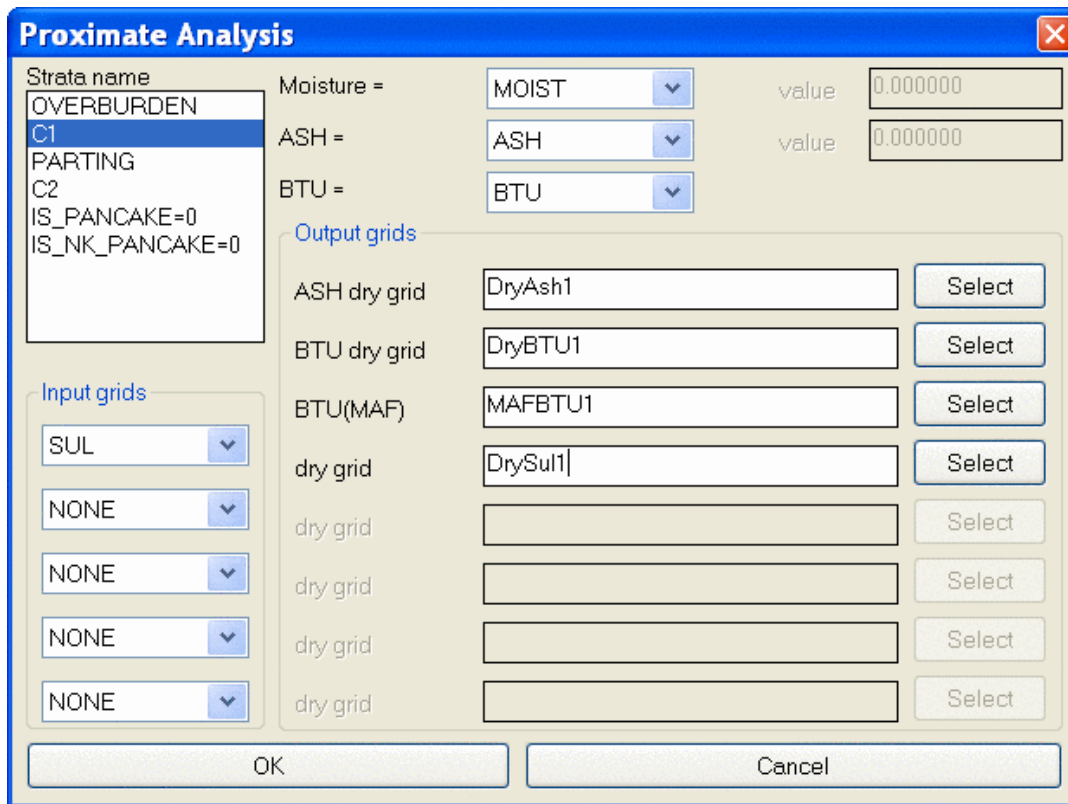
Keyboard Command: reserve_class

Convert As Determined Qualities

This function calculates strata qualities "as delivered" based on in place qualities. A Geologic Model file with all strata and quality grids specified is required to run the command.

The dialog below shows a typical use of the routine. The strata list displays all the strata defined in the Geologic Model file. To calculate "as delivered" qualities of the particular strata pick that strata from the list. Then fill out at least the Moisture field by picking the corresponding quality name from the list or by specifying a value if grid is not present. This allows the program to calculate dry qualities by picking a quality in pull-downs in the lower left corner and specifying an output grid file name in the corresponding dry grid line.

When Ash and BTU fields are filled out, dry Ash, dry BTU and BTU(MAF) grids may also be calculated.



Repeat this procedure for every strata in the list for which dry qualities are to be calculated by the routine. When finished click on the OK button and all the required grids will be generated.

Prompts

Select Geologic Model File Dialog

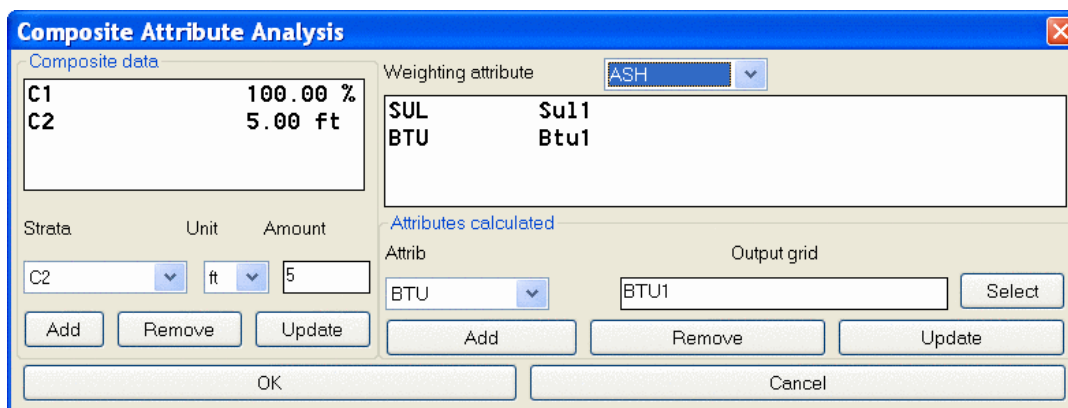
Select a file for Proximate Analysis (the .DIL File Dialog)

Pulldown Menu Location: StrataCalc

Keyboard Command: dilute1

Composite Qualities Analysis

This function calculates quality grids for user-defined composite strata. A Geologic Model file with all strata and quality grids specified is required to run the command.



First you should define a composite strata by specifying which strata it is made of and amounts of strata included in the composite. The amounts are either specified as % of or thickness of the seam. Then specify a quality to weight

the other qualities by, picking from the list on the top. Below it define all the qualities to be calculated along with the corresponding output file names. Click on the OK button when done filling in all the information and the grids specified will be calculated.

Prompts

Select Geologic Model File Dialog

Select .AVG File Dialog

Pulldown Menu Location: StrataCalc

Keyboard Command: dilute2

Case Studies

Case Study #1: Techniques of Geological Compositing

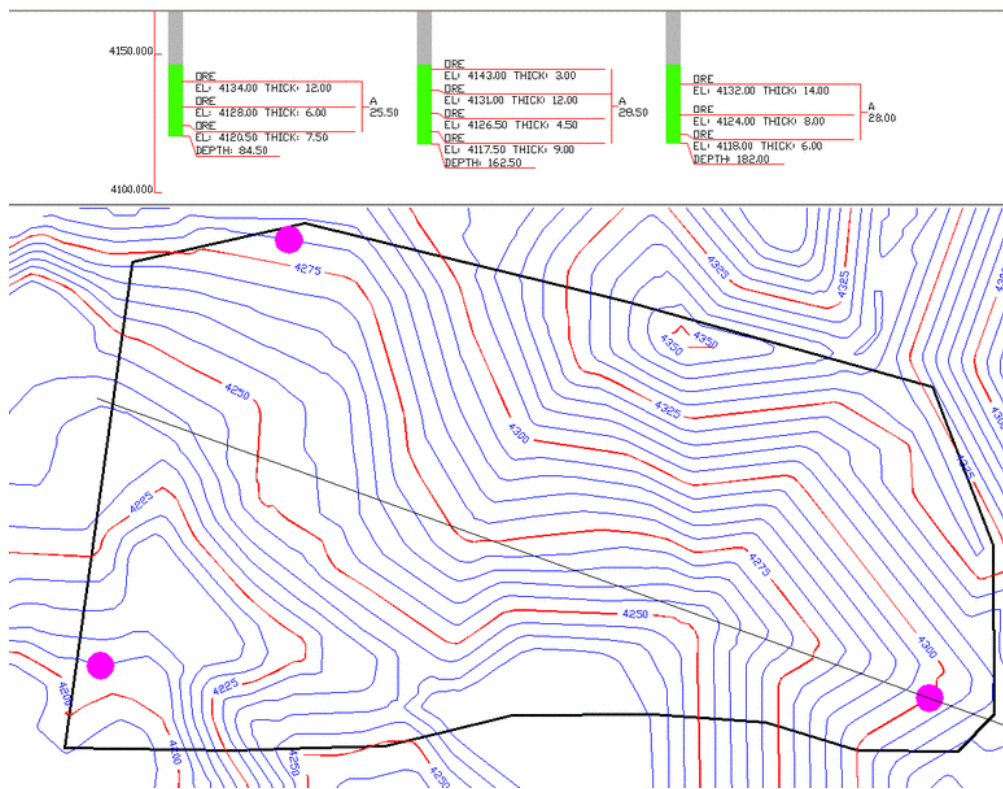
Two Types of Compositing are described here. (1) Single Seam or Ore Body compositing where the seam has been sampled at various intervals, and (2) Combinations of Several Seams Separated by Interburden.

(1) Single Seam or Ore Body

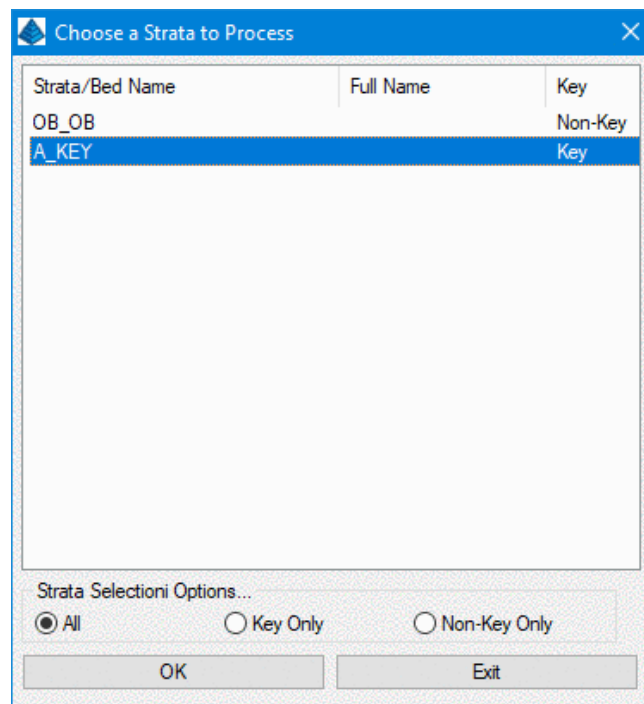
Large ore deposits such as limestone and thick beds of coal are often sampled at multiple points within the single ore body or bed. For long-range planning and reserve estimates, it is often desired to obtain the composite quality for the entire mineral deposit within an inclusion perimeter. For short-range planning, the mine engineer needs to know what quality will be obtained in a particular vertical segment of the ore body or seam, also within a defined perimeter.

Entire Seam Composites In the first example, a body of ore has been sampled at 3 to 4 points vertically, as seen in the geologic columns. Note in the geologic columns (generated with the Draw Geologic Column command) that all the ORE has been assigned bed "A". This will "lump" the various ore samples into one "bed". If the bed name is not assigned when the holes are drawn, this can be accomplished by the commands "Assign Bed Names" or "Fill in Bed Names". The Selected Strata Quantities command will then produce the below report (showing composite A tons) when the A.Key strata is chosen singly and the inclusion perimeter is selected. Compositing of the single A bed is automatic.

Note: The program will not composite qualities for a strata name that is repeated, unless there is a bed name grouping them together.



The command Selected Strata Quantities brings up this screen to select the strata or bed to analyze.



Follow these prompts to get the report screen.

Command: CHQUAN2

Demo version function call#: 8885

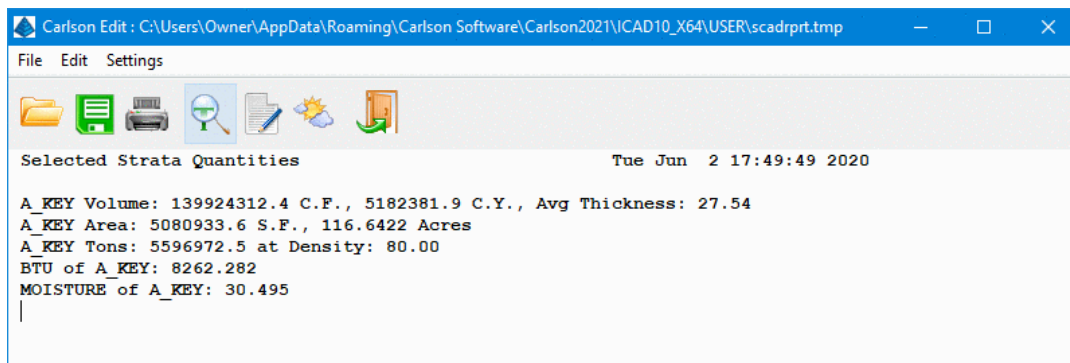
Select objects: Specify opposite corner: 3 found

Select objects: ENTER

Reading drillhole 3

Finding splits ...

Finding pinch out ...
 Calculating seam stacking ...
 Select the Inclusion perimeter polylines or ENTER for none.
 Select objects: 1 found
 Select objects:ENTER
 Select the Exclusion perimeter polylines or ENTER for none.
 Select objects: ENTER
 Choose modeling method [<Triangulation>/Inverse/Kriging/Polynomial/Leastsq/ABOS/Voronoi]? T
 Apply global trend to strata extrapolation [Yes/<No>]? N
 Use Triangulation Subdivision [Yes/<No>]? N
 Ignore zero attributes [<Yes>/No]? Y
 Create composite quantities [<Yes>/No]? N
 Assigning grid values> 2600
 Extrapolate grid for 1685 null values
 Processing cells ...
 Enter strata A_KEY density in lbs/ft³ <0.000>: 80
 A_KEY Volume: 139924312.4 C.F., 5182381.9 C.Y., Avg Thickness: 27.54
 A_KEY Area: 5080933.6 S.F., 116.6422 Acres
 A_KEY Tons: 5596972.5 at Density: 80.00
 Calculate qualities from strata A_KEY values [<Yes>/No]? Y
 Assigning grid values> 2600
 Extrapolate grid for 1685 null values
 Processing cells ...
 BTU of A_KEY: 8262.282
 Assigning grid values> 2600
 Extrapolate grid for 1685 null values
 Processing cells ...
 MOISTURE of A_KEY: 30.495
 Press ENTER to continue.



Vertical Composite Zones

In the next example, a vertical zone of ore has been defined as the upper part of the ore body or "A" bed from 4130 elevation and up. The "floor" of the vertical cut in this case is flat, following the 4130 elevation. This might correspond to a mine plan where the base of the first of two cuts is designed to hit elevation 4130 within the entire pit. The technique required is to divide bed A into two beds. This is done by the command "Split Bed" within Strata/Bed Utilities under Drillhole. The prompting is as follows:

Command: splitbed

Split strata method [<Elevation>/Grid/Thickness]? E

Select the Drillholes for bed split. *Select the holes*

Select objects: Specify opposite corner: 3 found

Enter name of the bed to split: A

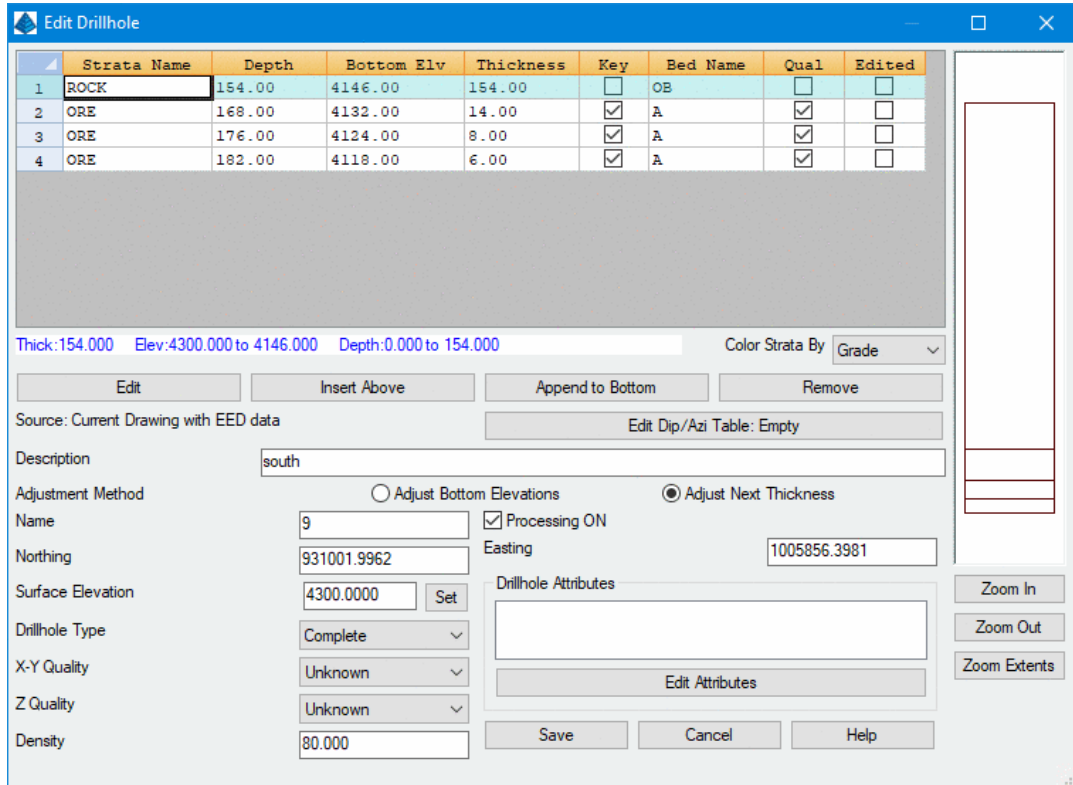
Rename bed or assign key/non-key status [<Name>/Status]? Name

Enter new name for the upper part of the bed: A_Upper

Enter new name for the lower part of the bed: A_Lower

Enter a split elevation: 4130

This shows a drillhole before splitting the A seam into an A Upper and A Lower.



Here is the same drillhole, viewed within "Edit Drillhole" after the command "Split Bed". Notice the new Bed Names, A Upper and A Lower.

Strata Name	Depth	Bottom Elv	Thickness	Key	Bed Name	Qual	Edited
1 ROCK	154.00	4146.00	154.00	<input type="checkbox"/>	OB	<input type="checkbox"/>	<input type="checkbox"/>
2 ORE	168.00	4132.00	14.00	<input checked="" type="checkbox"/>	A UPPER	<input checked="" type="checkbox"/>	<input type="checkbox"/>
3 ORE	170.00	4130.00	2.00	<input checked="" type="checkbox"/>	A UPPER	<input checked="" type="checkbox"/>	<input type="checkbox"/>
4 ORE	176.00	4124.00	6.00	<input checked="" type="checkbox"/>	A LOWER	<input checked="" type="checkbox"/>	<input type="checkbox"/>
5 ORE	182.00	4118.00	6.00	<input checked="" type="checkbox"/>	A LOWER	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Thick:14.000 Elev:4146.000 to 4132.000 Depth:154.000 to 168.000 Color Strata By Grade

Source: Current Drawing with EED data

Description: south

Adjustment Method: Adjust Bottom Elevations Adjust Next Thickness

Name: 9 Processing ON

Northing: 931001.9962 Easting: 1005856.3981

Surface Elevation: 4300.0000

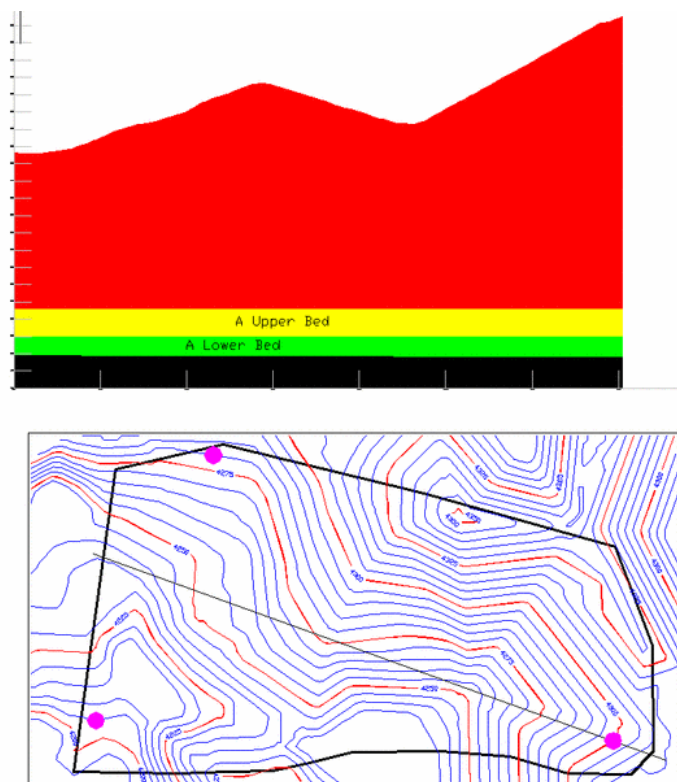
Drillhole Type: Complete

X-Y Quality: Unknown

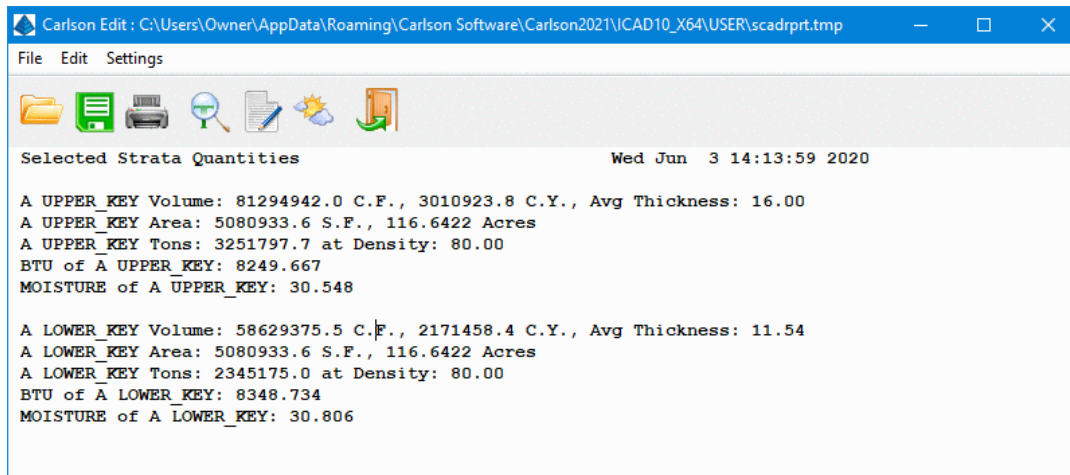
Z Quality: Unknown

Density: 80.000

The result is a new base of A Upper seam. It is important to note that the quality characteristics of the original ore zone have been applied identically to the new ore zones that have been renamed to A Upper and A Lower. The rest of the ore zone, the lower 12.5 feet, has become part of A Lower. If a drillhole does not reach as low as 4130 elevation, then all the ore remains in the A Upper bed and is unaffected. Conversely, if the ore never exceeds elevation 4130 in a particular drillhole, then it is assigned bed A Lower for all the ore sample points. Here is a Fence Diagram showing the two divisions of the bed, at elevation 4130.



Now the command "Selected Strata Quantities" can be run. There is now no need to select the "composite" option because you want to calculate the A Upper and the A Lower beds distinctly. You select them one at a time from the dialog box. This leads to the result shown here in the report. Adding the tons calculated here: 3251797.7+ 2345175.0 = 5596972.7 calculated above, when the bed was just A. This is a good check to make sure the quantities match.



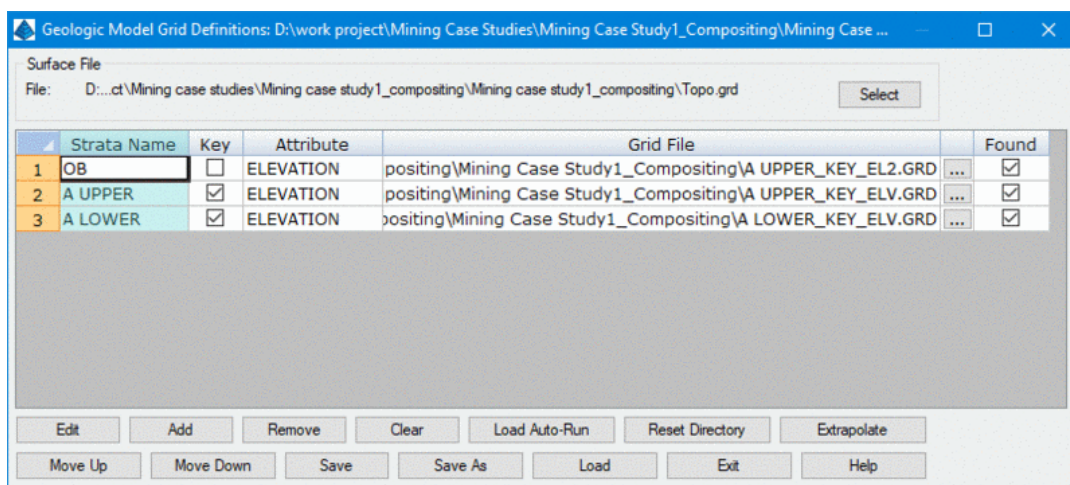
Split Bed by Grid File

In the previous example, we chose to split the bed by elevation. If, by contrast, the goal was to leave 5-feet of lower ore in place to be removed by different equipment, then the vertical demarcation is not a fixed elevation but a grid file defined as base of ore plus 5 feet. This grid file is made by using "Make Strata Grid Files" to make the base of ore, then using Grid File Utilities (in the Grids Pulldown Menu) you would add a value of 5-feet and re-save the grid file as Oreplus5.grd or some such name. This new file would be used to delineate the split elevation between the A Upper and A Lower beds.

Alternative Techniques

In the above example, the command "Selected Strata Quantities" was used. This is an "on-the-fly" selection approach, where the drillholes are selected each time the command is run. It does not take advantage of stored grid files in the Geologic Model file (.PRE), which is generally preferred. To apply the above procedures, follow this sequence:

1. Establish Bed A Upper and Bed A Lower using Split Bed.
2. Make grid files for the surface, the top of Bed A Upper, the base of Bed A Upper and the base of Bed A Lower.
3. Enter these files in appropriate order (A Upper first, then A Lower) as a stored Geologic Model grid set, using the command "Define Geologic Model" under the StrataCalc pulldown.



Then run Surface Mine Reserves, selecting Geologic Model as the modeling method. You will obtain the same quantities for each bed as reported from the command "Selected Strata Quantities".

Surface Mine Reserves

Process | Adjustment | Output

Modeling Method: Geologic Model

Top Constraint: Screen Model Surface File Elevation. Top Elev: 0.000

Bottom Constraint: Model Surface File Elevation Elevation+Surface. Bottom Elev: 0.000

Use Auto-Run Use Surface History. Merge Bench Quantities Percent: 1.000

Strata To Include: All Selected. Grid Subdivision Precision: High

Calculate Strata Qualities. Breakout Quantities By Attributes: Off. Fixed Non-Key Qualities

Use Named Pit Areas. Store Results In Pits. Bench#: 1

Adjust Pits Manager Use Property Boundaries Use Resource/Reserve Classification

OK Cancel Load Save Help

Surface Mine Reserves

Process | **Adjustment** | Output

Use Drillhole Elevations In Surface Ignore Zero Elevations

Use Triangulation Subdivision Use Global Trend Extrapolation

Use Highwall Slopes. Set

Recovery: Recovery Percent: 100.000. Use Strata Definitions. Use Recovery Attribute. Attribute Name: REC

Density: Density: 100.000. Use Strata Definitions. Use Density Attribute. Attribute Name: DENSITY

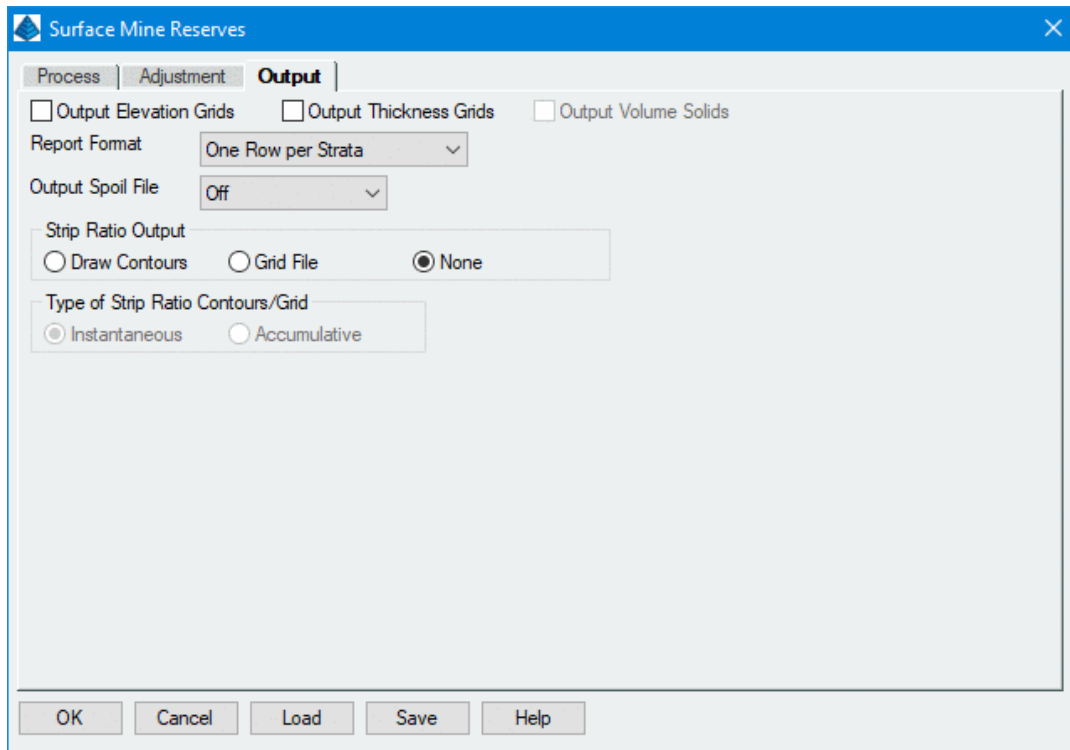
Min Key Thickness To Use (ft): 0.000. Min Depth To Use (ft): 0.000

Min Separable Parting Thickness (ft): 0.000

Non-Key Thickness To Add To Key: Dilution (ft): Above Key: 0.000. Below Key: 0.000. Use Strata Definitions

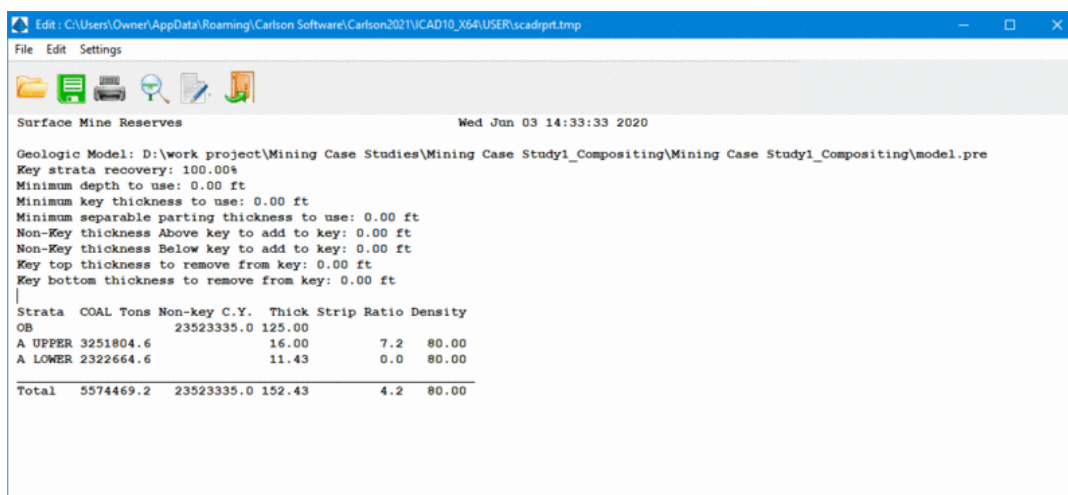
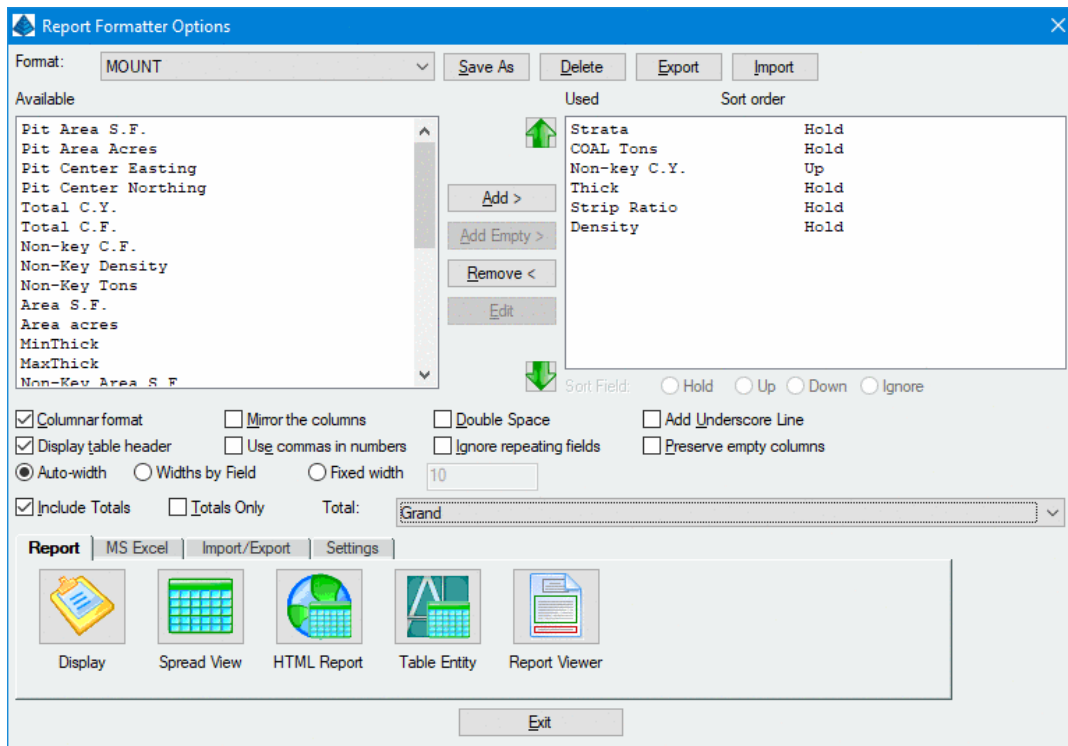
Key Thickness To Remove From Key: Loss (ft): Key Top: 0.000. Key Bottom: 0.000. Use Strata Definitions

OK Cancel Load Save Help



The advantage of setting up grid files is that multiple, pre-named pits can be run within Surface Mine Reserves, and the reporting can be formatted and expanded upon at the user's discretion, and even exported to Excel or Access. The various options within Surface Mine Reserves can be fine-tuned, such as recovery percentage, density, and dilution. Furthermore, "grand totals" are obtained where the qualities and volumes/tonnages of beds A Upper and Lower are composited back into the total reserve values. (Indeed, this is the exact procedure used for multiple seams with interburden.)

Shown here is a formatted report for the single inclusion polygon representing the pit in our example. Slight differences in values versus "Selected Strata Quantities" is a function of the difference in gridding locations and cell sizes. With stored Geologic Model grids, quantities and qualities will be fully repeatable.



(2) Combinations of Several Seams Separated by Interburden

In the mining of stratified deposits, it is very common to have several seams separated by interburden. Surface mines must consider the composite tonnage, composite strip ratio, and composite quality in any reserve study or short-term mining plan. Two main issues come to mind; how deep to mine, and when to mine while removing interburden as if it were Key. Let's look at a coal deposit example.

How Deep to Surface Mine

Strip ratios change as each lower seam is taken, and quantities of coal increase. The goal is to get as many lower seams as possible, but not so many that interburden thicknesses and strip ratios increase excessively, or quality degrades. Currently, about a 15:1 strip ratio approaches the maximum feasible ratio for cost-effective mining. Higher ratios do occur and will certainly occur if the market price of coal increases. Coals are said to "outcrop" at the surface, with "crop loss" referring to unmineable, "weathered" coal at the hillside edge. Usually the crop loss is around 12 to 15 feet measured vertically from the surface. It is often deeper in valleys or even small hillside ravines

and "drains", due to accumulation of debris and erosion. On "points" or ridges, crop loss may be only 10 to 12 feet, particularly in hardrock conditions. The Surface Mine Reserves routine is designed for estimating reserves and includes a built-in crop loss parameter (measured vertically from the surface), appearing as "Min Depth to Use". Here is a graphic representing the crop loss on the side of a hill.



A conservative engineer or geologist would enter 15 feet to obtain a "low-ball" estimate. Someone looking aggressively for all the coal they could possibly obtain might enter 10 or 12 feet for the vertical crop loss. Be aware that with steep 1.5:1 hillside slopes, a 10' vertical crop loss translates to 15' measured horizontally from the hillside. In 3:1, gently sloping terrain, a 10' vertical crop loss translates to 30' measured horizontally from the hillside. If the natural terrain slopes on the order to 2:1 to 3:1 or more, it is reasonable to use a lower vertical crop loss value for "Min Depth to Use". The value used is strictly a judgment call and is ideally based on observations at the mine. There is no option to have a variable crop loss. That is best handled by defining a "Strata Limit Polyline". See the "OutCrops and SubCrops" case study for more information. In many regions, shallow coals will "subcrop" as they hit an alluvial deposit or as it nears the surface and is decomposed due to oxidation and weathering.

Another Example: Getting Composite Qualities

The Surface Mine Reserves command automatically computes composite qualities and strip ratios on all Key strata, provided "Calculate Strata Qualities" is selected in the dialog box (dialog shown earlier above). The low SULFUR and BTU values for OB and IB are not included in the composite quality for SULFUR and BTU. This is because OB and IB are not defined as KEY. When C1 and C2 are imported or placed in the drawing, they are defined as KEY strata. Alternately, the KEY designation can be assigned and changed using the command "Define Strata". Shown below is a drillhole example of this dataset, and the composite KEY quality report.

Edit Drillhole

	Strata Name	Depth	Bottom Elv	Thickness	Key	Bed Name	Qual	Edited
1	ROCK	118.00	4719.98	118.00	<input type="checkbox"/>	OB	<input checked="" type="checkbox"/>	<input type="checkbox"/>
2	COAL	130.00	4707.98	12.00	<input checked="" type="checkbox"/>	C1	<input checked="" type="checkbox"/>	<input type="checkbox"/>
3	ROCK	132.50	4705.48	2.50	<input type="checkbox"/>	IB	<input checked="" type="checkbox"/>	<input type="checkbox"/>
4	COAL	167.00	4670.98	34.50	<input checked="" type="checkbox"/>	C2	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Summary Color Strata By: Grade

Source: Current Drawing with EED data Edit Dip/Azi Table: Empty

Description:

Adjustment Method: Adjust Bottom Elevations Adjust Next Thickness

Name: Processing ON

Northing: Easting:

Surface Elevation:

Drillhole Type:

X-Y Quality:

Z Quality:

Density:

Drillhole Attributes:

Edit : C:\Users\Owner\AppData\Roaming\Carlson Software\Carlson2021\ICAD10_X64\USER\scadrprt.tmp

File Edit Settings

Surface Mine Reserves Mon Jun 08 04:43:06 2020

Geologic Model: D:\work project\Mining Case Studies\Mining Case Study1_Compositing\TEST.pre

Key strata recovery: 100.00%

Minimum depth to use: 0.00 ft

Minimum key thickness to use: 0.00 ft

Minimum separable parting thickness to use: 0.00 ft

Non-Key thickness Above key to add to key: 0.00 ft

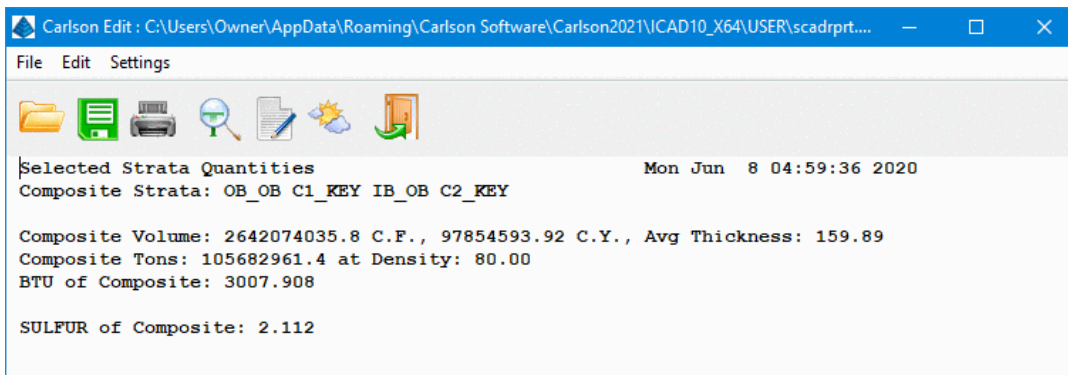
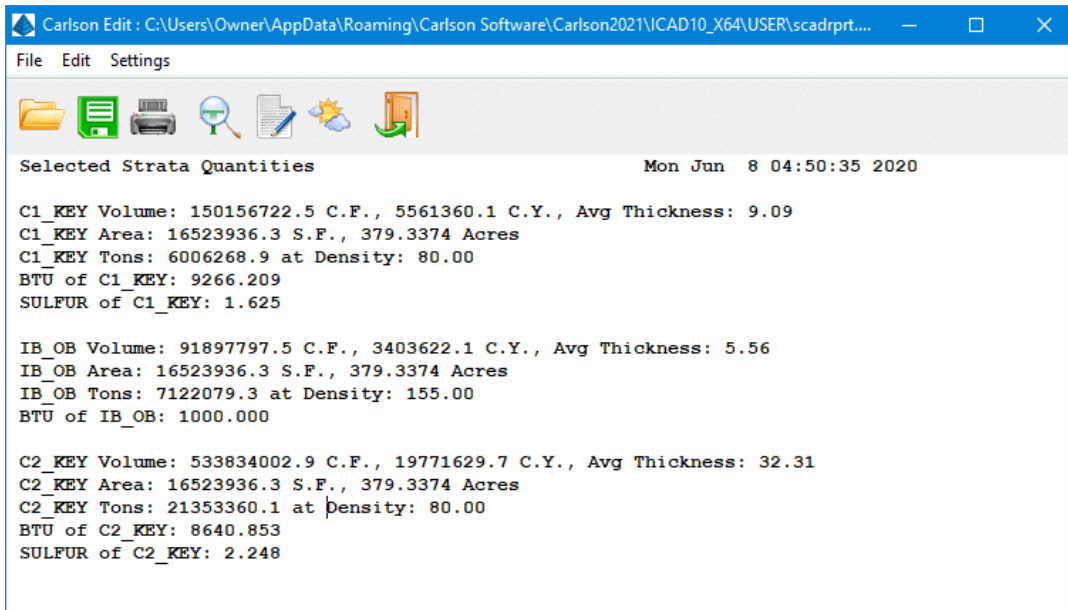
Non-Key thickness Below key to add to key: 0.00 ft

Key top thickness to remove from key: 0.00 ft

Key bottom thickness to remove from key: 0.00 ft

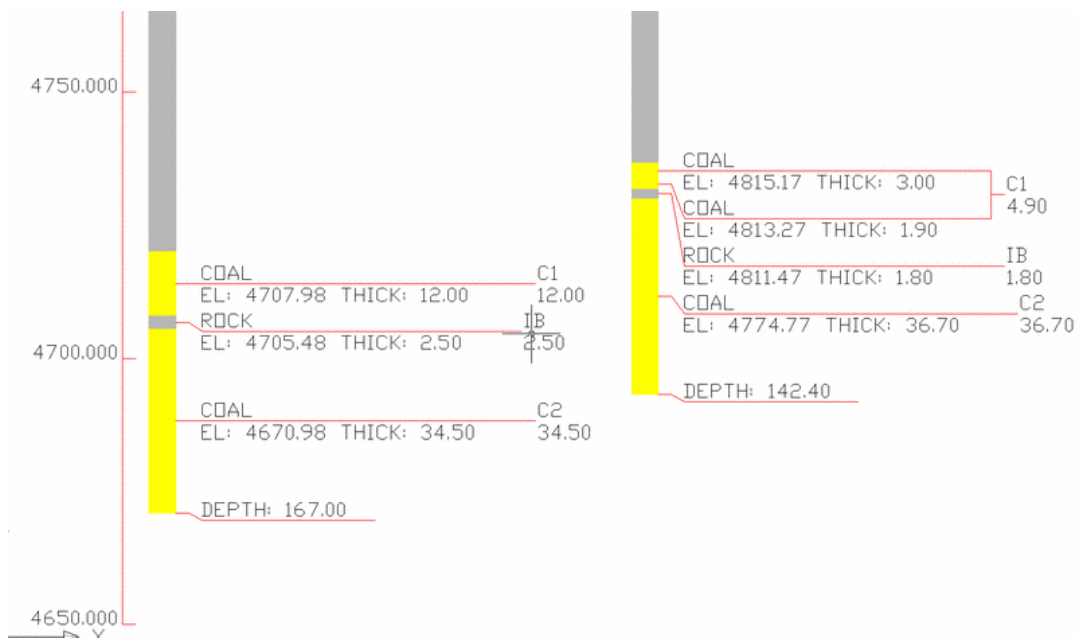
Strata	COAL	Tons	Non-key	C.Y.	Thick	Strip	Ratio	Density	SULFUR	BTU
OB_TOP		69,163,753.9	112.98							
C1_KEY	6,010,514.5		9.09		11.5	80.00	1.625	9,266.074		
IB_TOP		3,406,305.1	5.56							
C2_KEY	21,366,338.9		32.32		0.2	80.00	2.248	8,640.475		
Total	27,376,853.4	72,570,059.0	159.95		2.7	80.00	2.111	8,777.823		

By contrast, composite qualities can also be computed using the command "Selected Strata Quantities", but this routine will composite any selected strata, key or non-key. In our example, if all 4 beds are selected, the composite BTU is only 3008. All of the above calculations were based on screen-selection of the drillholes (not grid files), and use of Triangulation modeling. (Sulfur was not entered for Nonkey beds).



Partings: When to Mine and When to Waste

Drillholes 8 and 10 below have 2.5 and 1.8 feet of interburden, respectively, between coals C1 and C2. It makes sense to take the thin interburden with the coals, even though this will dilute some qualities (and maybe improve others). Surface Mine Reserves has an option called "Min Movable Parting Thickness" designed specifically for this purpose. The effect on qualities is also shown below. This result is obtained automatically by designating 2.0 feet as the "minimum movable parting thickness", meaning that any lesser thickness will not be separated as waste but will be included as coal.



To obtain interburden qualities and to factor them in to the composite quality, it was critical that qualities be associated with non-key strata. This is accomplished within the command "Define Drillhole" or a setting on the main Surface Mine Reserves screen for Fixed NonKey Qualities.

File Edit Settings

Surface Mine Reserves Mon Jun 08 05:06:41 2020

Geologic Model: D:\work project\Mining Case Studies\Mining Case Study1_Compositing\TEST.pre
 Key strata recovery: 100.00%
 Minimum depth to use: 0.00 ft
 Minimum key thickness to use: 0.00 ft
 Minimum separable parting thickness to use: 6.00 ft
 Non-Key thickness Above key to add to key: 0.00 ft
 Non-Key thickness Below key to add to key: 0.00 ft
 Key top thickness to remove from key: 0.00 ft
 Key bottom thickness to remove from key: 0.00 ft

Strata	Non-Key Tons	Non-Key BTU	Non-Key SULFUR	COAL Tons	Thick Strip	Ratio	Density	SULFUR	BTU Non-Key	Density
OB_TOP	140,056,601.6	1,000.000	0.500		112.98					150.00
C1_KEY				6,010,514.5	9.09	11.5	80.00	1.625	9,266.074	
IB_TOP	3,441,521.5	1,000.000	0.500		1.80					150.00
IB_TOP (Min Parting)				3,456,246.3			150.00	0.500	1,000.000	
C2_KEY				21,366,338.9	32.32	0.1	80.00	2.248	8,640.475	
Total	143,498,123.2	1,000.000	0.500	30,833,099.7	156.19	2.3	87.85	1.930	7,905.966	150.00

In our example, the interburden had a density set at 150 within Define Strata, and had BTU values in the 1000 range, and sulfur in the 0.5 range. The net effect of including the thin interburden was to improve sulfur and degrade BTU. If sulfur under 1.0% is the more critical value in meeting quality requirements, and less BTU is satisfactory, then the user could accept thicker interburden. Compare these quality values with the ones calculated above and see how the SULFUR is lower and the BTU is also lower, but still within spec. Just try several reserve runs to see what the parting thickness cut off can be.

Case Study #2: Outcrop and Subcrop Modeling

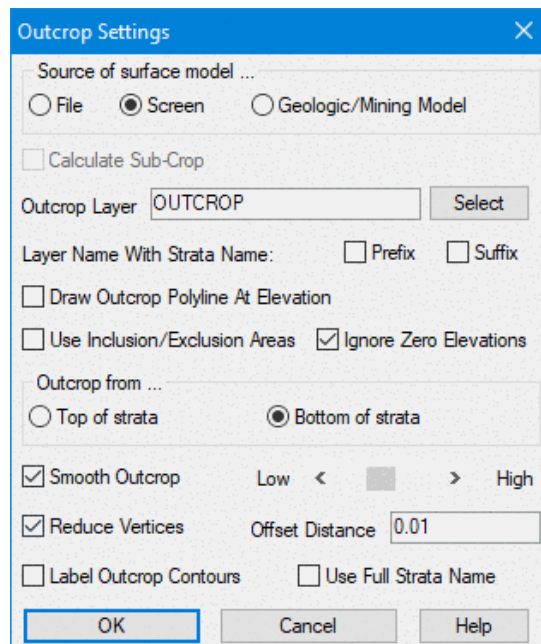
This tutorial steps through examples of how to handle outcrop and subcrop conditions on several examples involving ridge top mines as well as sub-cropping seams. Accurately locating the crop is one of the first things the planner needs for laying out a mine. Carlson automatically detects the location of the crop within the Surface Mine Reserves command. Volumes will never be calculated from above the ground surface. The advantage of the Draw Outcrops routine is that it allows the user to witness where the program is interpreting outcrops and subcrops. It is also useful in establishing a starting perimeter for pit layout. Outcrops are automatically calculated and drawn using Draw Outcrops command under StrataCalc on the fly directly from the drillhole data, or from Geologic Model grids.

For Generating Outcrops Directly from Drillholes ("On the Fly")

- Surface Topography Grid, Geologic Model file or Contours
- Drillholes on the screen

For Generating Subcrops

- Surface Topography Grid or Contours
- Drillholes on the screen
- Thickness Grid for *Unconsolidated* or *Weathered Strata* just below the surface



Outcrop Procedure

Select Draw Outcrops in the StrataCalc menu. Fill in the Outcrop Settings dialog box. The user can specify the source for surface topography (grid file or screen selection) and what layer to store the outcrop in when it is drawn. The user normally will use the settings as shown in the dialog box below with one exception. The offset distance could be set from 0.01 to 1.0 ft to reduce the vertices in the outcrop when drawn. Otherwise, the polyline for the outcrop will contain an unnecessarily large number of points.

Command: outcrop

Select surface entities & at least 3 drillholes

Select objects: Specify opposite corner: 111 found

Use drillhole surface elevations in surface model [Yes/<No>]? Y if they match the contours, otherwise N

Reading points ... 79695

Ignored 562 points with zero elevation.

Ignored 36 duplicate points.

Intersections found 80377

Pass> 10 Null Z values left> 0

Finding splits ...

Finding pinch out ...

Calculating seam stacking ...

Output grids for strata and surface [Yes/<No>]? N The files may be saved out for other uses. No is the typical response.

Choose modeling method [<Triangulation>/Inverse dist/Kriging/Polynomial/LeastSq]? T

Apply global trend to strata extrapolation [Yes/<No>]? Y

Use Triangulation Subdivision [Yes/<No>]? N

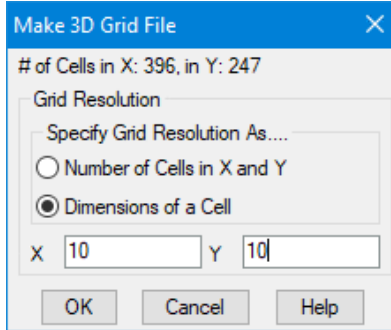
Triangulating points ... 5

Assigning grid values> 98400

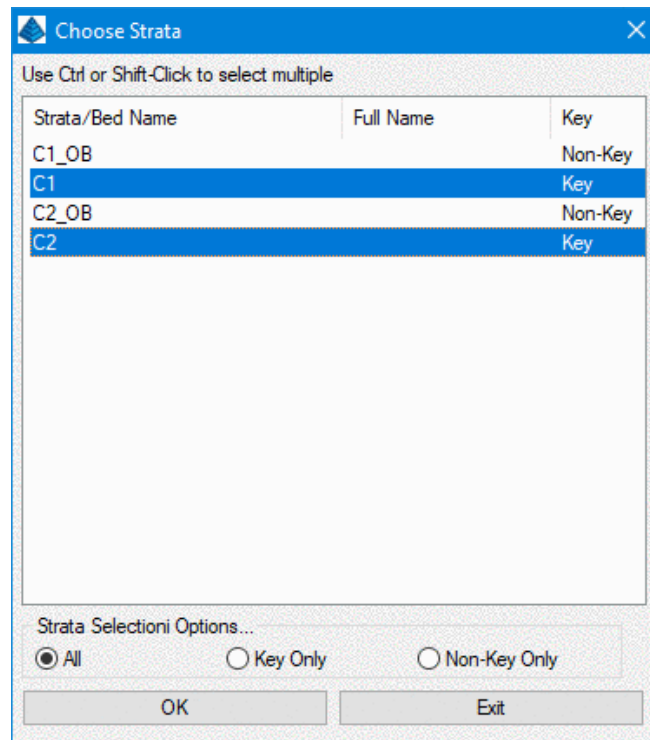
Pass> 282 Null Z values left> 0

Contouring elevation 0.0

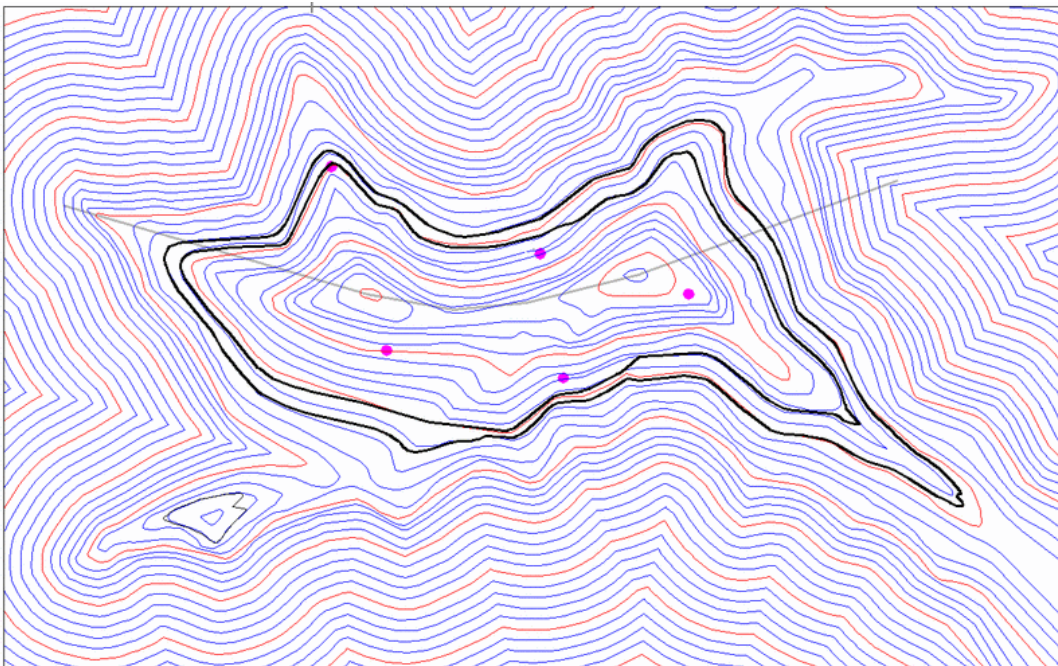
Inserted 348 contour vertices. Complete the **Make Grid File** dialog box. The user may select the total number of rows and columns or the grid spacing pattern option. The program creates the grids "on the fly" from the drillhole data.



Select the preferred method. Once the gridding method is selected the **Choose Strata** dialog box appears. Choose the strata to process. Each strata can be gridded using independent gridding algorithms. If Inverse Distance or Least Squares was selected as the gridding method, when the strata is selected, the prompt is for the power to be used in the algorithm. Once the power is entered, assuming Inverse Distance or Least Squares is the selected method, then select the strata to be processed. Any number of strata may be selected to be processed by holding down the CTRL or SHIFT buttons.

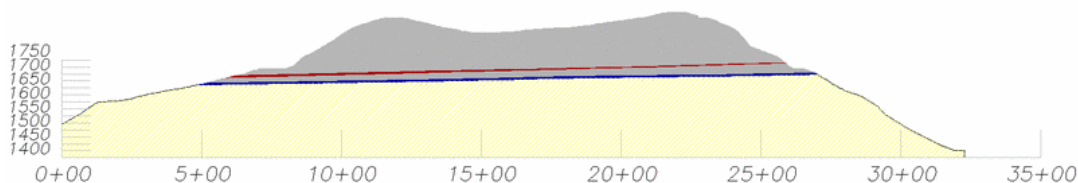


Outcrop lines are produced on elevation zero by default. The outcrop polyline can be drawn on the surface file with the option to Draw Outcrop Polyline at Elevation.



Making Fence Diagrams

As a way to verify the location of the outcrop or make a graphic presentation of a geologic cross section, a fence diagram can be created using the Fence Diagram command under StrataCalc. Fence diagrams are useful for presenting geologic data as well as verifying the depths of seams and thickness of burden. Shown below is a fence diagram taken from the line drawn from west to east through the outcrops on the hill.



Fence Diagram Procedure and Prompting:

Command: fence

Select polyline to pull fence diagram from: *Pick the polyline to get the fence from*

Use drillhole surface elevations in surface model [Yes/<No>]? *N*

Select surface entities and at least 3 drillholes. *Select the drillholes and surface contours*

Select objects: Specify opposite corner: 112 found

Reading drillhole 5

Finding splits ...

Finding pinch out ...

Calculating seam stacking ...

Ignore zero elevations [<Yes>/No]? *Y*

Reading points ... 79695

Choose modeling method [<Triangulation>/Inverse dist/Kriging/Polynomial/LeastSq]? *I* Choose a modeling method you prefer.

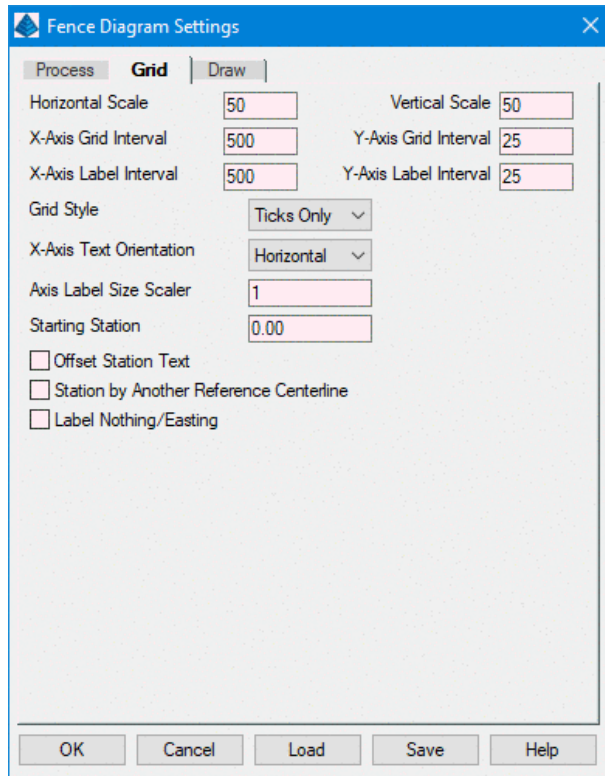
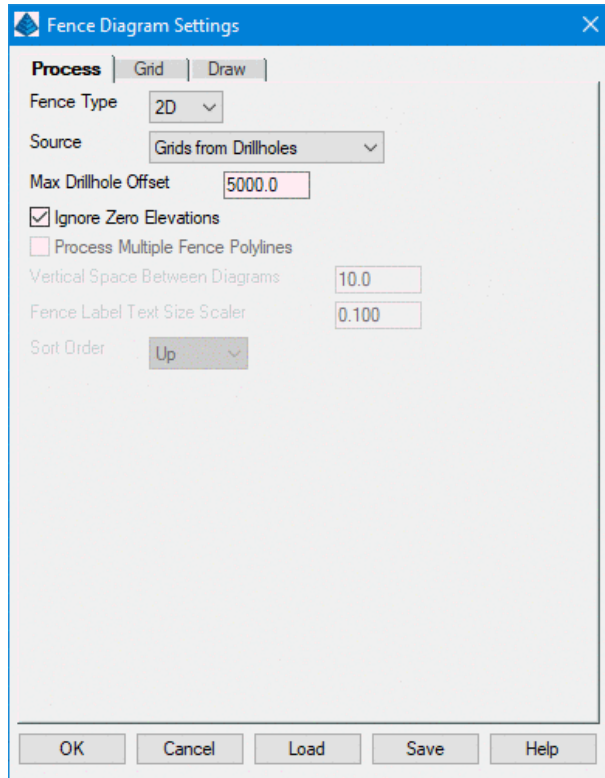
Use inverse distance to which power [First/<Second>/Third/Other]? *S*

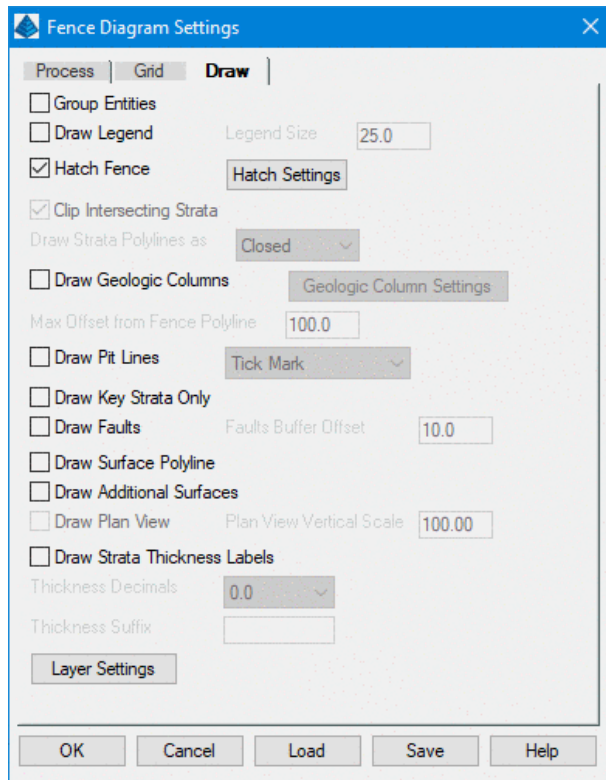
Use elliptical inverse distance [Yes/<No>]?

Calculating grid by inverse distances 98456...

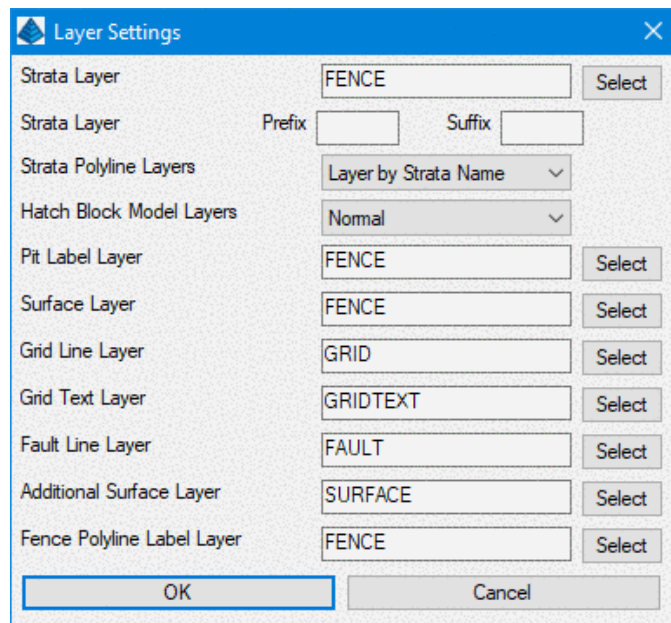
Bottom elevation of grid <1650.00>: 1400

Pick the lower left corner for the diagram: Pick an open area in the drawing for the lower left corner of the diagram.



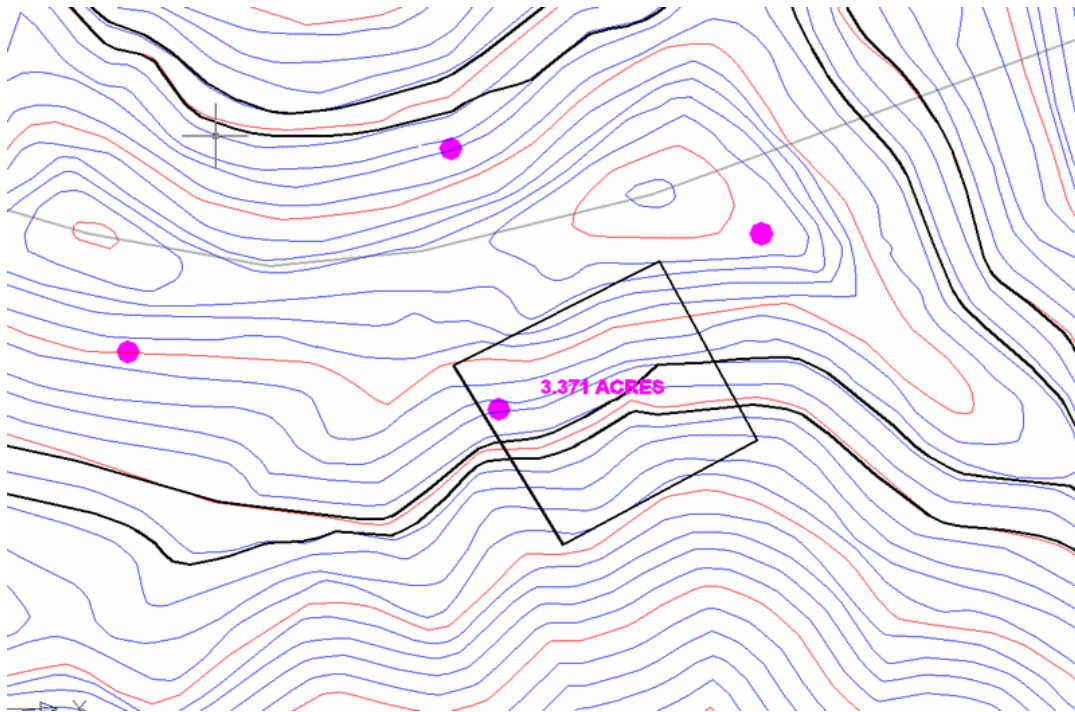


Layer Settings button can be used to set layers for the different components of the fence diagram.



Using Surface Reserves to Check the Outcrop Volumes

To prove that Carlson honors the outcrop in its calculations, layout a rectangular polygon that overlaps the outcrop area. If Surface Mine Reserves recognizes the seam crop, then the area of the seam will be less than the area of the perimeter, and the upper seam crop area will be less than the lower seam area. It calculates reserve areas and handles outcrops automatically. Higher surfaces (the topography in this case) are the limiting factors. Volumes will never be calculated above the next surface up.



The area of the perimeter is shown as 3.371 Acres, some of it within the area of coal, and some outside. Run the Surface Mine Reserves command using the perimeter to get the tons and acres of coal. A typical report from the Surface Mine Reserves command under StrataCalc is shown below. Reports are user-defined in the Report Formatting dialog box.

Surface Mine Reserves Tue Jun 09 09:22:37 2020

Geologic Model: D:\work project\Mining Case Studies\Mining Case Study 2_Outcrops\NEW.pre
 Key strata recovery: 100.00%
 Minimum depth to use: 0.00 ft
 Minimum key thickness to use: 0.00 ft
 Minimum separable parting thickness to use: 2.00 ft
 Non-Key thickness Above key to add to key: 0.00 ft
 Non-Key thickness Below key to add to key: 0.00 ft
 Key top thickness to remove from key: 0.00 ft
 Key bottom thickness to remove from key: 0.00 ft

Strata	Non-key	C.Y.	Thick	COAL	Tons	Strip	Ratio	Area	acres	Pit	Area	Acres
C1_OB	166,287.6	63.98							1.611			3.371
C1		4.87	14,641.9			11.4			1.724			3.371
C2_OB	61,092.0	18.07							2.096			3.371
C2		6.18	23,904.6			2.6			2.221			3.371
Total	227,379.6	93.10	38,546.5			5.9						

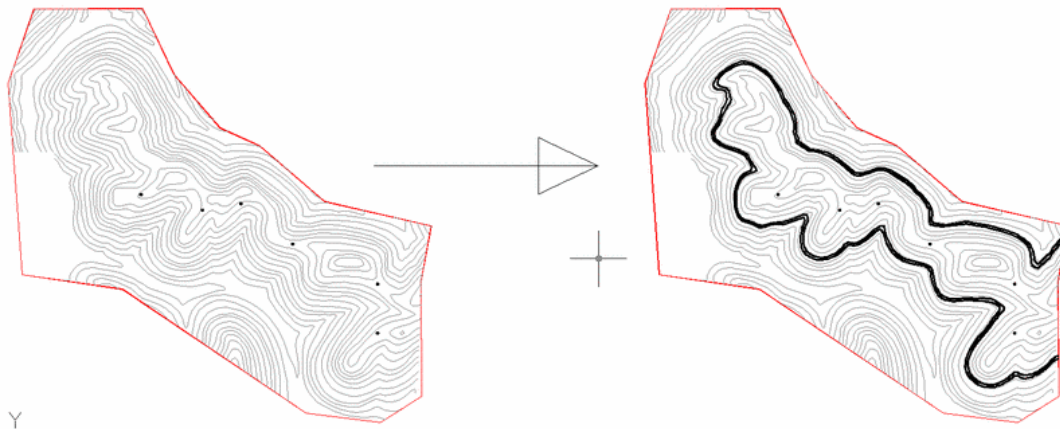
As the report indicates, Carlson recognizes the limits imposed at the outcrop. It properly truncates the seam at the outcrop and reports the included area between the crop and rectangle. Notice the C1 has less acres than the C2 because it crops out higher on the slope. the C2 shows 2.221 Acres compared to the full 3.371 Acres of the polygon. The Pit Acres are reported as an available option in the Report Formatter.

In topographic situations similar to the one shown in this example, for coal, the user will have to deduct from the mineable reserves for crop loss. Crop loss sometimes runs 10' to 15' vertically. The coal in this zone is usually oxidized to such an extent that the BTU is so low and the ash is so high, the coal cannot be sold for steam product,

and chemistry is so poor that it cannot go for metallurgical product either. Oxidized coal is treated as overburden.

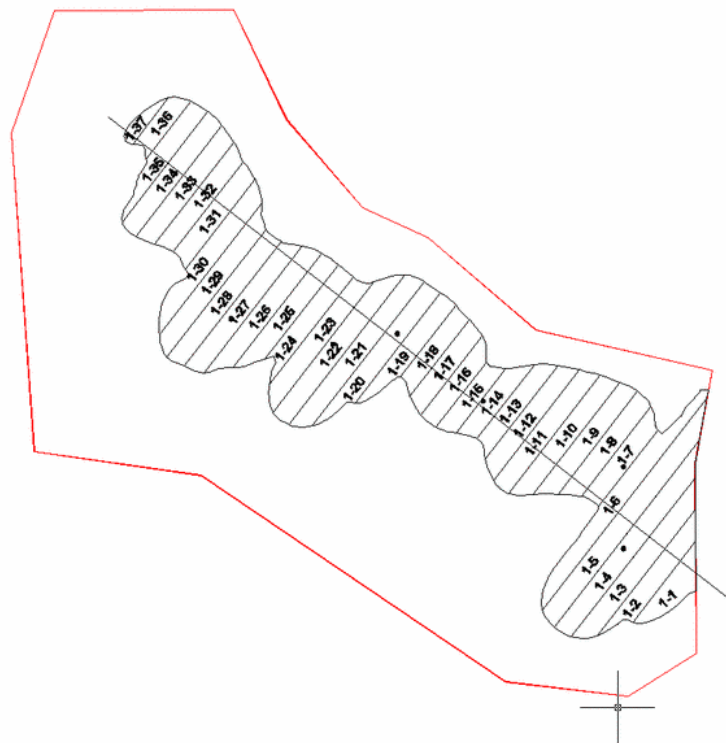
Using Outcrops for Laying Out Pits

In the example below, the outcrop was used to define the limits of the surface mining pits. The outcrop defined the limits of mining on each side of the pit. The direction of mining was input as part of the pit layout routine. The example below shows the Pit Layout By Advance command executed on the closed polylines. First, create a close polyline of the outcrop/mine boundary. Use the command Draw-Boundary Polyline and pick inside the ridge/outcrop lines to get a new interior polyline. You can also use AutoCAD's BPOLY command. Then, draw a direction line down the ridge to assign direction. Pits will be cut perpendicular to this line.



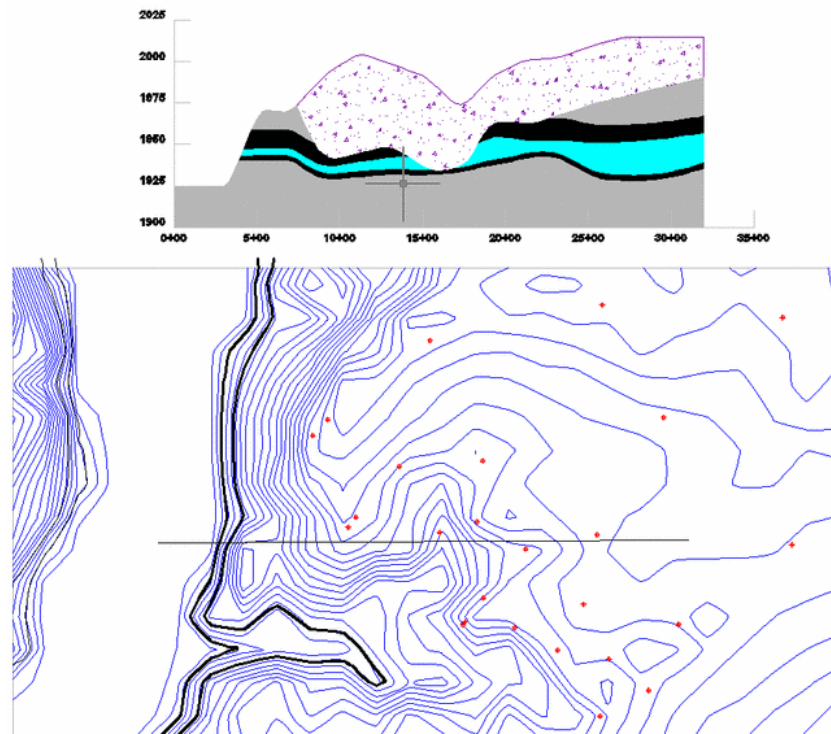
Pit Names, Labels, and Identify Pit Names

Since the process of laying out pits using the Pit Layout By Advance option gives the pits a name, a discussion of naming and labeling procedure is helpful at this point. After the pits are drawn, there are two methods to draw the pit name labels. Go to Label Pit Polylines or Pit Label Formatter. Identify Pit Polylines allows the user to see the Pit Names when the labeling option was not selected.



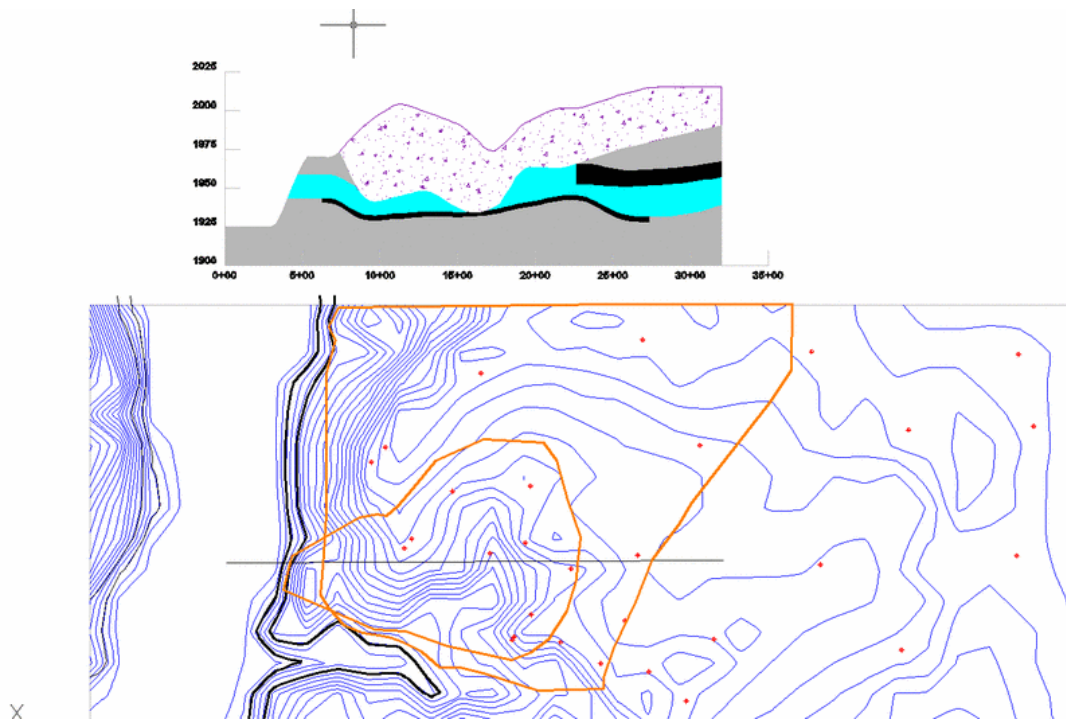
Subcrop Procedures

Subcrops occur where the unconsolidated material (could be glacial till or a channel deposit) has eroded out the key seam below the surface. Subcrops differ from pinchouts, in that unlike pinchouts, which can occur anywhere below the surface, lower strata being eroded by the unconsolidated material cause subcrops. Unlike outcrops which can be created "on the fly" from drillhole data, subcrops are calculated from grids. To calculate subcrops, the user must have the surface grid, thickness grid of the unconsolidated material, and bottom elevations of the strata to test for subcropping. In addition to the gridding algorithms, the user can specify strata limit polylines to override the natural mathematical interpretation of the data. When Carlson calculates the subcrop it starts from the surface and works down through the grids. If an upper grid has a lower elevation lower than that of a lower grid, the elevation of the upper grid is set to the elevation of the lower grid. This leads to a zero thickness in the lower grid, or a subcrop. Select the File option to begin calculating the subcrop. Carlson allows the user to calculate the subcrop from a Geologic Model grid or directly from the drillhole data.



This example was created by making strata grids from the drillholes. Put them in the Geologic Model grids file and then generate a Fence Diagram. The outcrop coincides with the outcrop lines in plan view on the hill side. Notice how the glacial till is subcropping the upper coal seam and the parting.

Strata Limit Polylines Using the same example for the Subcrops above, we will now create Strata Limit Polylines for both the coal seams and the parting. They must be drawn in plan view, then named with Name Strata Limit Polylines. Strata Limit Polylines must be turned on under Setting, Carlson Configure, Mining Modules. There are options to use them, and to also automatically select them. The orange lines are limit lines. The interior line is an Exclusion for the upper coal seam. The outer line is an Inclusion for the lower coal seam. Volumes will be calculated accordingly with Surface Mine Reserves.

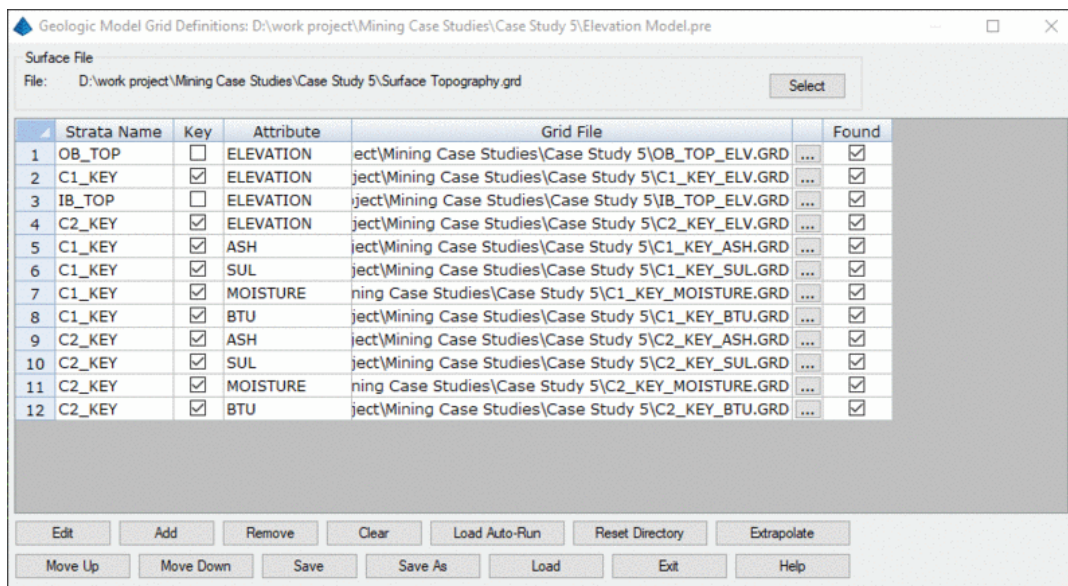


Case Study #3: Techniques Of Gridding

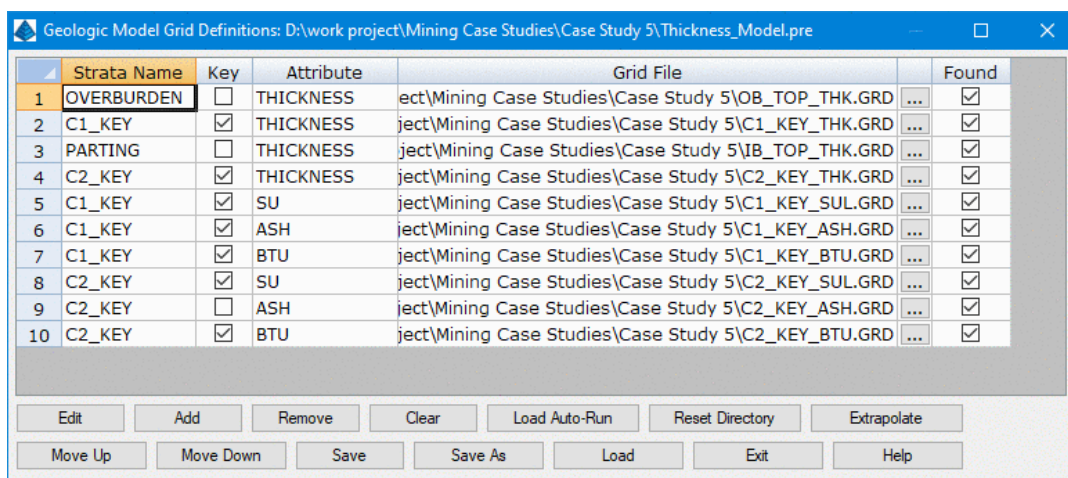
Surface and Underground Reserves are the key routines for developing reserve estimates and qualities and for setting up equipment-based mine scheduling. Along with the Fence Diagram routine, Surface or Underground Reserves will calculate strata elevations and qualities using Geologic Model grids. These grids represent the "mine model". Here we will study how to make the grids strategically to build in well-defined subcrops, outcrops, splits, correct strata thicknesses, and qualities.

What is the "Geologic Model" File?

The Geologic Model File can either be an Elevation model or a Thickness model. The elevation model consists of a grid model of the surface topography as the first and primary grid. Below the surface grid is the bottom elevation grid for each strata to be modeled, ordered top to bottom in the Geologic Model grids dialog box. Any number of quality attribute grids (covering such items as sulfur, ash, moisture, etc.) can be associated with the bottom elevation grid of a particular strata, though they are not required. If a strata is called C1, for example, the C1 name is associated with its bottom elevation grid file, and any attribute grid files should refer to the same Strata name. A Thickness Geologic Model just contains thickness grids (no elevation grids). There is no surface topo grid file defined in a thickness model. Below are typical examples of both types of Geologic Model files.



Geologic Model Elevation Model

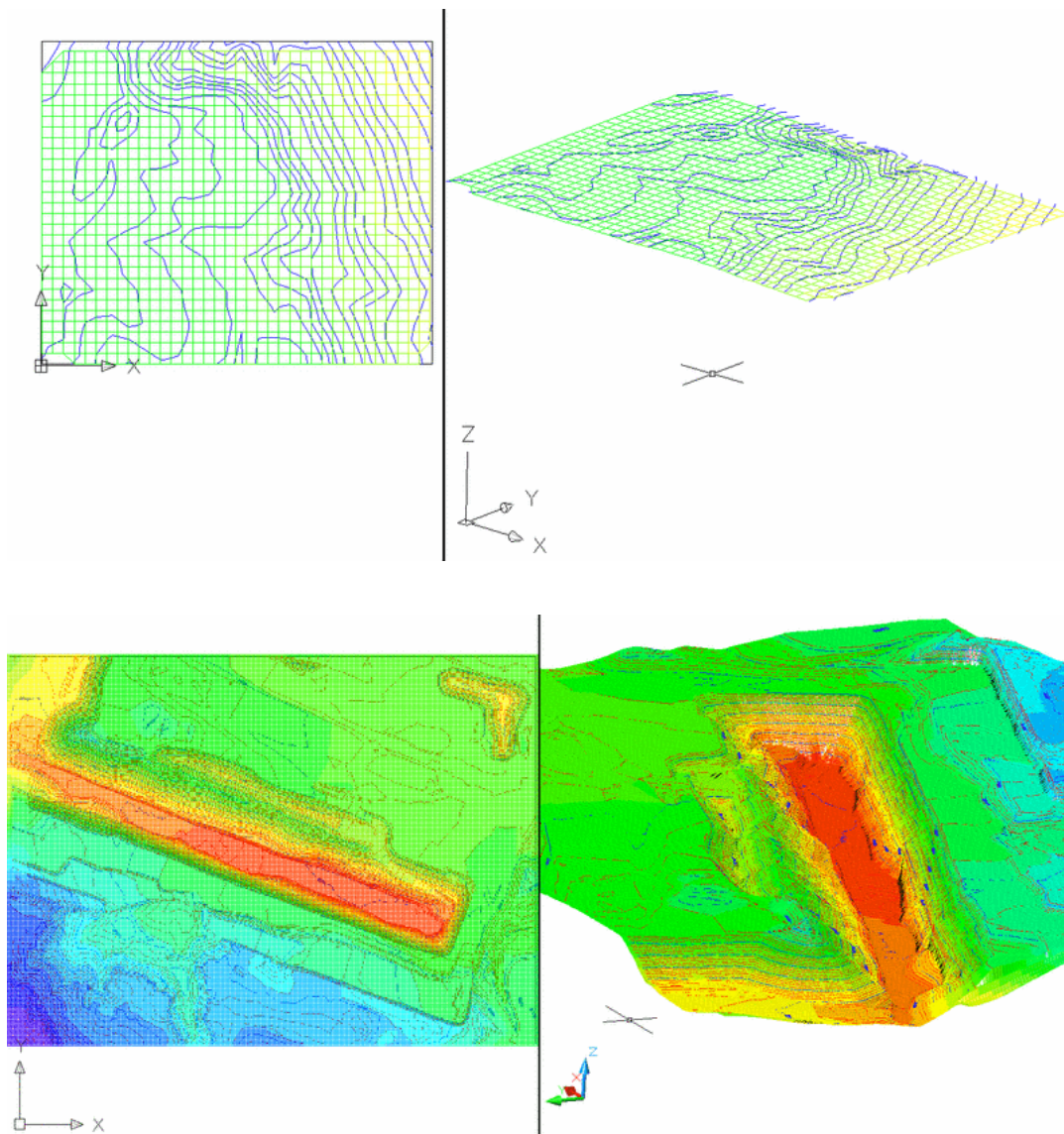


Geologic Model Thickness Model

When naming attributes such as Ash, BTU and Moisture, be sure to attach the attributes to a strata (in this case C1 and C2) which must match the exact spelling of the strata name containing the associated base elevation or thickness grid. These names are not case sensitive, and are converted to upper case automatically.

Grid Cell Dimensions

Drillholes at many mines are often drilled at a spacing of 500 feet or more, making small cell size unnecessary when modeling geologic aspects, especially quality attributes. As a rule of thumb, cell size should be 1/4 the average drillhole spacing between drillholes for most accurate modeling. With some minerals and ores such as quality controlled limestone and clay, drillholes are drilled as close as 50 feet apart. This would suggest the need for a 12.5-foot cell size or less. Surface topography, however, often demand the tightest cell size, because the topography can include steep cliffs, high stream banks and other abruptly changing features, that can be lost or smoothed if cell size is on the order of 100' to 200' spacing. Below are two examples of surfaces. The first surface has gently sloping terrain and widely-spaced contours. This has been gridded at 100'x100', as there are no sharp features that need to be captured in the grid file. The second set of images shows an open pit with benches, spoil and roads. To accurately capture all of this detail, a grid size of 10'x10' is more appropriate. It makes a much larger file, but does not over-smooth the surface, as a larger grid cell size would do. There is no limit, but try to keep the number of total cells in a grid file less than one million total cells. It will run much faster if each grid file is less than one half of a million grid cells.



Cell Positions and Dimensions Should Match

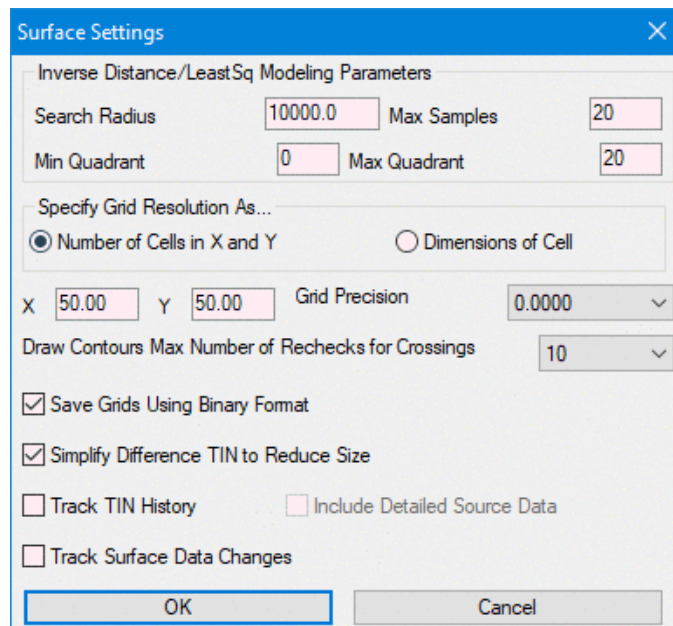
Some routines in Carlson require that the grid position and cell size should be identical for all grid files in a Geologic Model grid model set (Design Bench Pit is an example). To ensure that grids match position and dimension, make the first grid file, then make all additional grid files based on the position of the first grid file. To do this, you select option "F" when prompted: Use position from another file or pick grid position (<Pick>/File)? There are Grid File Utilities to modify grids, such as Change Position, Change Resolution and Match Dimensions.

In Surface and Underground Reserves calculated from a Geologic Model, the grids do not need to match. It is common to have a surface topo grid file with a small cell size, such as 10x10. Then the structure grids for elevation or thickness could have a medium cell size, such as 50x50. Finally, quality attribute grids can have an even larger cell size, such as 200x200. This could be due to the fact that not every drillhole has quality sampled, so the spacing of quality holes is much greater than structure data holes, not needing a tight resolution. Here it is important to note that when volumes are calculated, the elevation and quality grids will be temporarily resized to match the topography (note that the grids themselves will not be permanently modified).

Cell Dimensions Versus Number of Cells

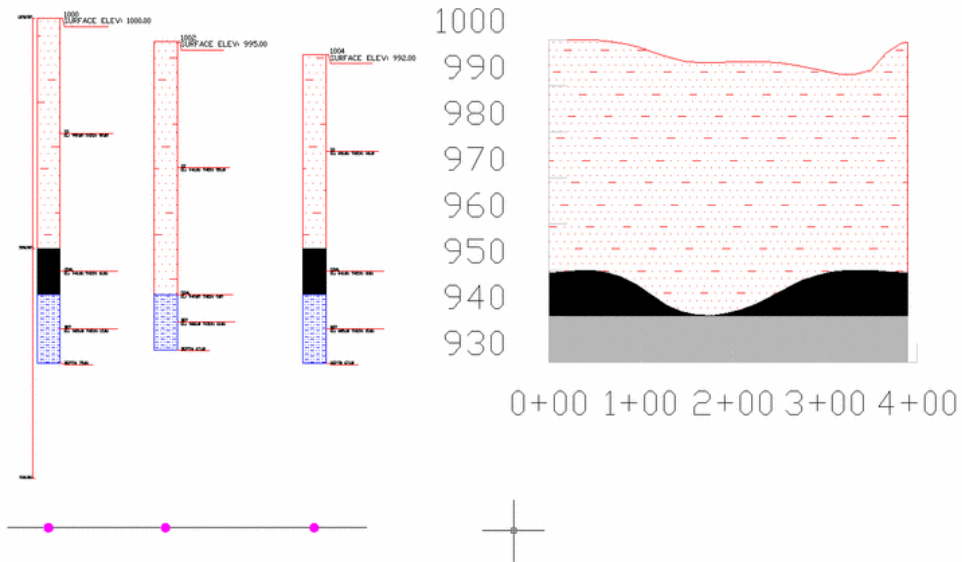
Carlson defaults to 50x50 "number of cells" when first installed, meaning that in any grid window position, there will be 50 cells in the X-direction and 50 cells in the Y-direction, unless altered by the user. If the window is longer in the x-direction, then the cells will be longer in their X-dimension than in their Y-dimension, creating rectangular shaped grid cells. By contrast, the user can specify the cell dimension when making grids, leading to a variable number of cell, depending on the size of the grid window. It should be noted that the default itself can be set by

the user. This is done by selecting Carlson Configure under the Settings Pulldown Menu > Surface Settings, which provides the following dialog. Note that at the bottom of the dialog there is the option to set the number of cells or the dimension of the cells to any desired value.

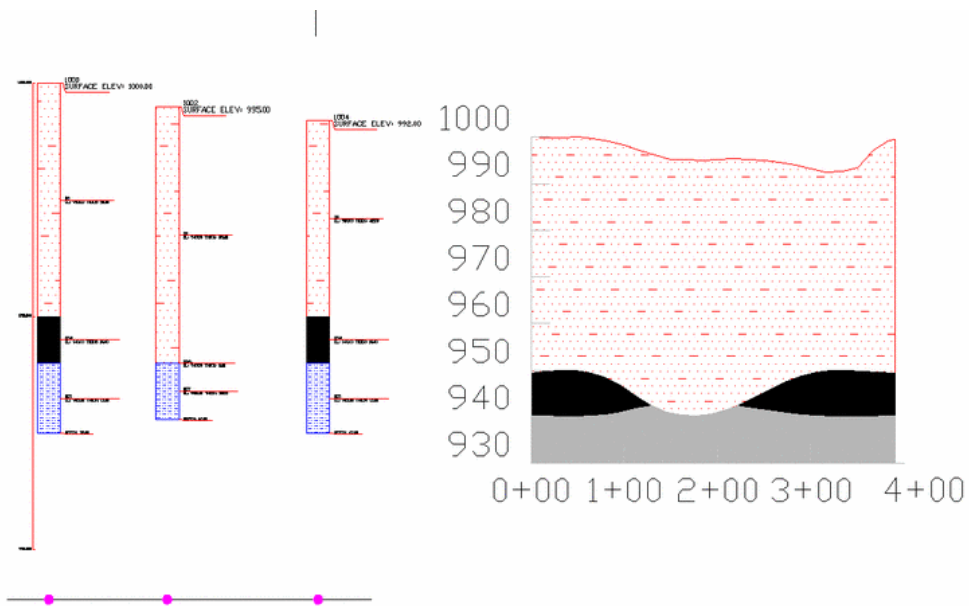


Make Top of Strata Grids by Adding Thickness Grids to Bottom of Strata Grids

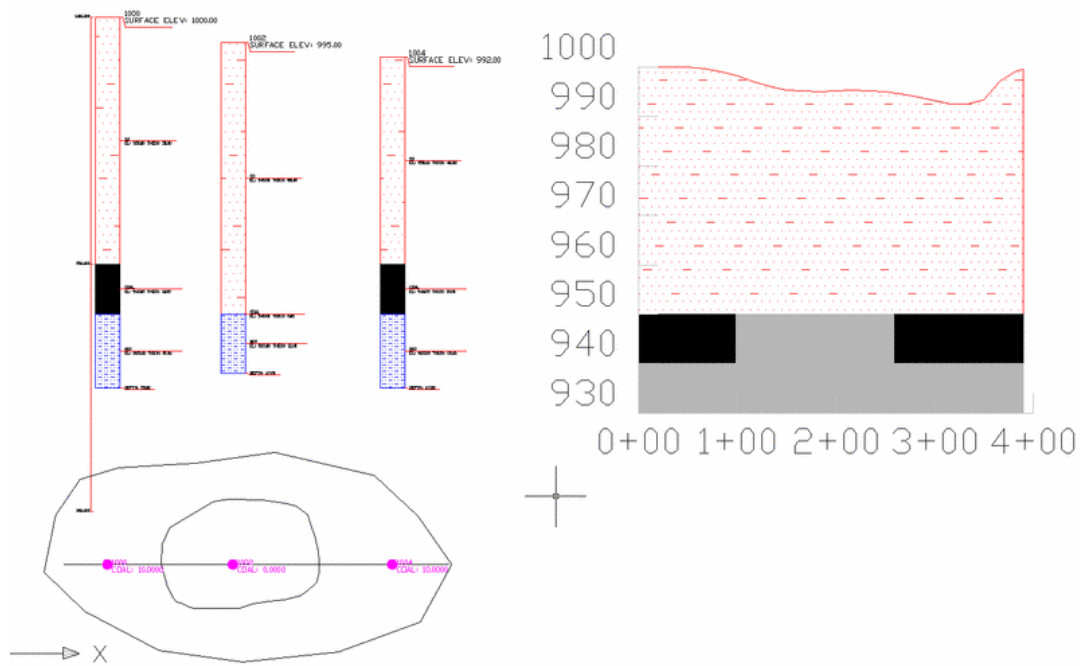
The elevation Geologic Model File is based on having grid files for the bottom and top of all key strata. If you have only the surface grid, and the bottom and top elevation of each Key strata, you are set up for most reserve and scheduling work. Of course, quality grids can be added, and NonKey elevation grids such as base of unconsolidated overburden are valuable. But the point here is that you want bottom and top of Key grids for each seam under consideration. These grids should in most cases be made by making a thickness grid for the strata and adding thickness to base elevation to obtain top of strata grids. There are two settings that should be monitored when doing this type of modeling. Under Settings > Carlson Configure > Mining Settings, be sure that pinch-out is on for modeling thickness. If not, then the seam will never pinch out in cases of zero thickness holes. When modeling elevation grids, it is often helpful that pinchout turned off, because when it pinches out a seam, it sometimes brings that elevation up to the next seam above, to pinch it out. If pinch out is off, it will keep the elevation grid down where it should be, had the seam been there. Add the two grids together to get the roof. Where the seam had zero thickness, the roof will be the same as the floor, and down and the correct elevation. The following example shows this concept, where the middle seam is pinching out in the middle. The middle hole has a zero value for coal thickness. This will bring the coal up to that hole, then pinch it at the hole.



This next example is created from the example where the Coal seam does not exist in the middle hole, not even a zero. This method will pinch the coal 1/2 way between the holes, based on the Pinchout Settings under Carlson Configure > Mining Settings. Most of the time, this is your best guess.



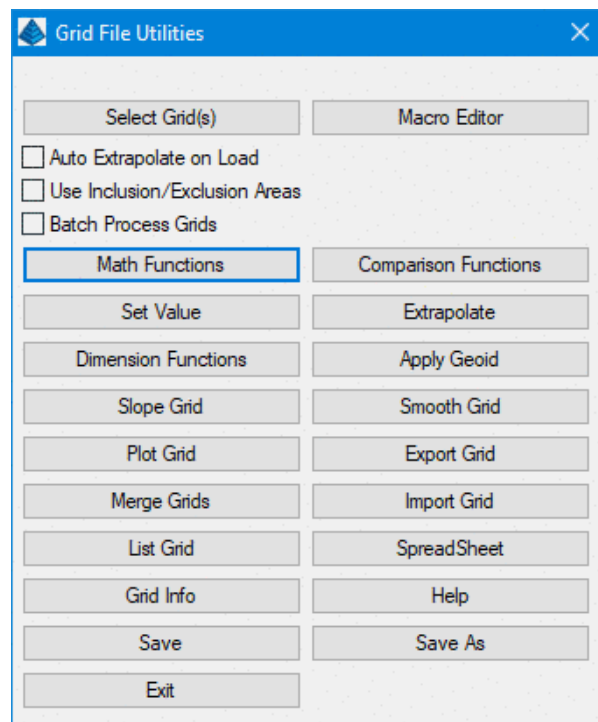
Using slightly different settings, this next Fence Diagram can be obtained.



Two Strata Limit Polylines were drawn around the drillholes, representing crop lines. The interior line is an exclusion limit line. The outer line is an inclusion limit line. The grids were remade, the bottom elevation and the thickness were added together to get the new roof, and here is the result. The seam carries its full thickness to the cropping limit lines, no pinching is taking place.

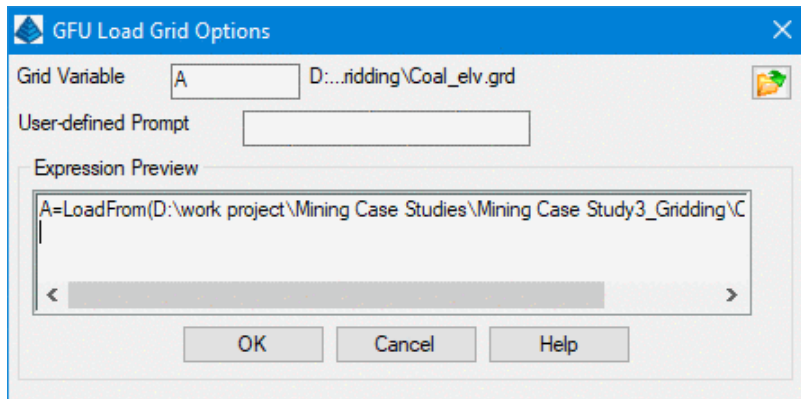
Grid File Utilities

The Grid File Utilities can be accessed from within the Geology Module by entering GFU in the command line, or under the Grids Pulldown Menu > Grid File Utilities. It is also located in the Surface Pulldown Menu of the Civil Module. After starting the command, you must first choose Select Grid(s) to load a grid file. Within GFU, you can click an option to be prompted for Inclusion/Exclusion perimeter polylines. The grid manipulation will only occur inside or outside these perimeters. The commands BPoly and Shrinkwrap under the Draw Pulldown Menu are useful tools to generate these perimeters.

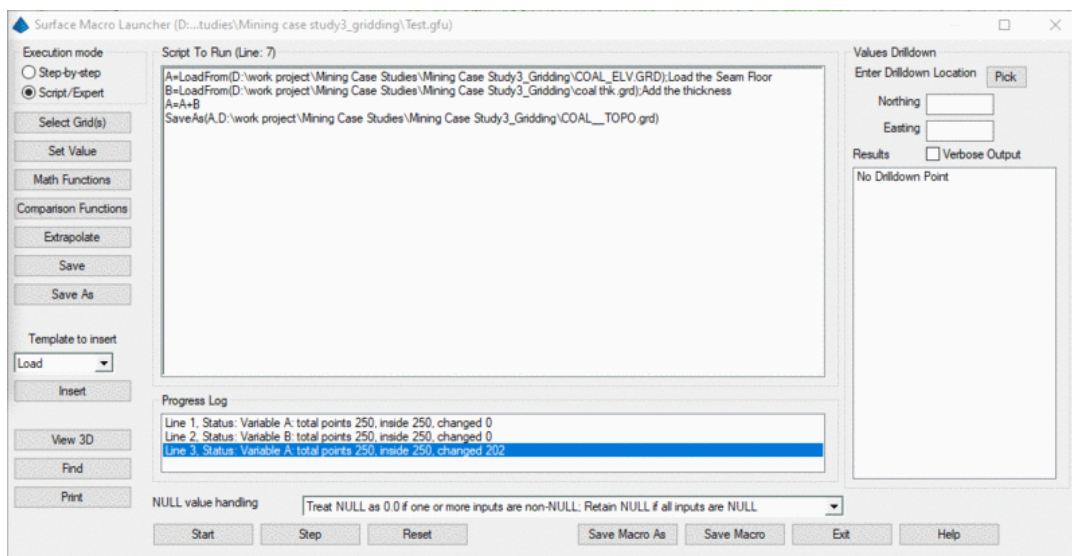


Adding One Grid to Another

In order to add grids, select Grid File Utilities (GFU) to bring up the dialog box shown above. Choose Select Grids and load the base elevation of the coal seam, COAL_ELV.GRD. The next step is to select Math Functions > ADD GRID which asks for the grid file to load (COAL_THK.GRD). At this point you would choose "SaveAs" and save the result as "COAL_TOP.GRD". If this modeling effort is a one-time process, there is no need to save a macro that allows for automatic re-running of the grid addition. However, if the thickness grid or base of coal grid might change due to the addition of more drillholes (or the editing of existing drillholes), then macros can be time-saving devices. To make a macro for our example, you would get to the dialog above by entering GFU as before and clicking the Macro Editor button. First, create a new GFU file for future Auto-Run. Then Choose the Select Grid button. The upper right button in GFU Load Grid Options allow user to choose grid file.



The Grid Variable "A" at the top represents the grid file to be loaded. So A is COAL_ELV.GRD (the base of COAL grid). Repeat loading the second grid and use Grid Variable B for the COAL_THK.GRD thickness grid. Next click Save As, same as above. Now select Start to run the macro. Note that you may manually modify the text in the macro as needed. Comments can be added by typing ";" at the beginning of the comment. For example, the below line will load a grid file, but will recognize the end of the line as a comment rather than part of the actual macro.
A=LoadFrom(D:\work project\Mining Case Studies\Mining Case Study3_Gridding\COAL_ELV.GRD);Load the Seam Floor



The Need to Extrapolate

Some modeling methods (Inverse Distance, Kriging, Linear Least Squares, Nearest Neighbor, and ABOS) will automatically extrapolate the grid, filling in values entirely within the limits of the grid location. If the modeling is

done by Triangulation or Polynomial, the grid values will only exist within the area of the data points - everything beyond the data points will be empty. Thus the area of modeling by Triangulation or Polynomial will always be less than the area of modeling by any other method. This may create a problem in Grid File Utilities where grid addition is involved. Two things can happen. Number one, the program may complete the addition, but in reality grid values have not changed where the base grid file contained null values. Number two, the program may report "Grid files do not match" and refuse to do the grid math operation. To remedy this, choose "Extrapolate" within Grid File Utilities, select the default method, and extrapolate all grids prior to doing grid math. The extrapolate command itself can become part of the macro if record or append is selected. Other routines have the option to extrapolate, such as Grid Inspector, where there is a check-box to Extrapolate Grids. The Reserve commands will always extrapolate the grids upon loading, so it is generally preferred to make sure they are extrapolated to start with. There are two options at the bottom of the Define Geologic Model dialog to extrapolate the grids. One method will extrapolate the elevation out, the other will merge with the next upper seam, pinching the thickness to zero.

Reserves from Geologic Model Grids

It is always preferable to compute volumes from Geologic Model files using the Surface or Underground Mine Reserves. All the care and control that went into making the grid files is then reflected in the improved accuracy and legitimacy of the result. The Geologic Model can contain quality attribute grids, and even a Block Model for detailed quality tracking and breakdown. The alternative is to select drillholes directly from the screen (on-the-fly), which builds in uncertainty as to the nature of the modeling.

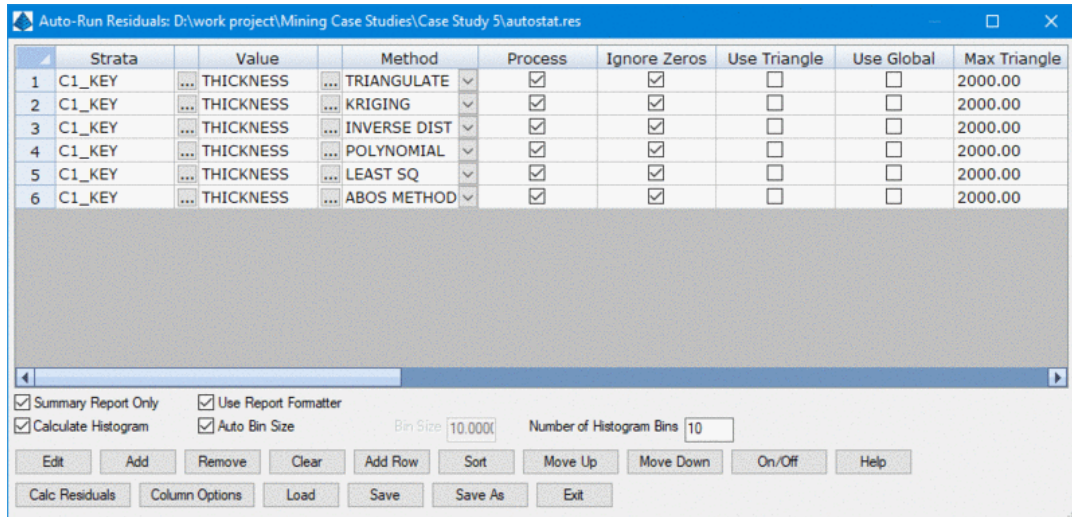
Choosing a Method of Gridding

Every user is confronted with the issue of what gridding algorithm to choose. Here there is no substitute for experience and verification in the field. We at Carlson Software have noticed that many qualities such as sulfur or calcium are modeled most often by Inverse Distance. Base elevation, in general, appears to be amenable to the logic of Triangulation or Polynomial, much like surface topography, though even here Inverse Distance is often used. Strata thickness, however, is again more localized and is often best modeled by Inverse Distance or Least Squares. Polynomial surface modeling utilizes Triangulation, so again lends itself to broad, large area influences. When and how to use Kriging is an art in itself. We have found the "power" form of Kriging to model effectively in evenly distributed drillhole data. The following diagram shows the same drillhole data set modeled with 12 different versions of the algorithms. Notice there are some large differences, yet all have their benefits.



Calculate Residuals

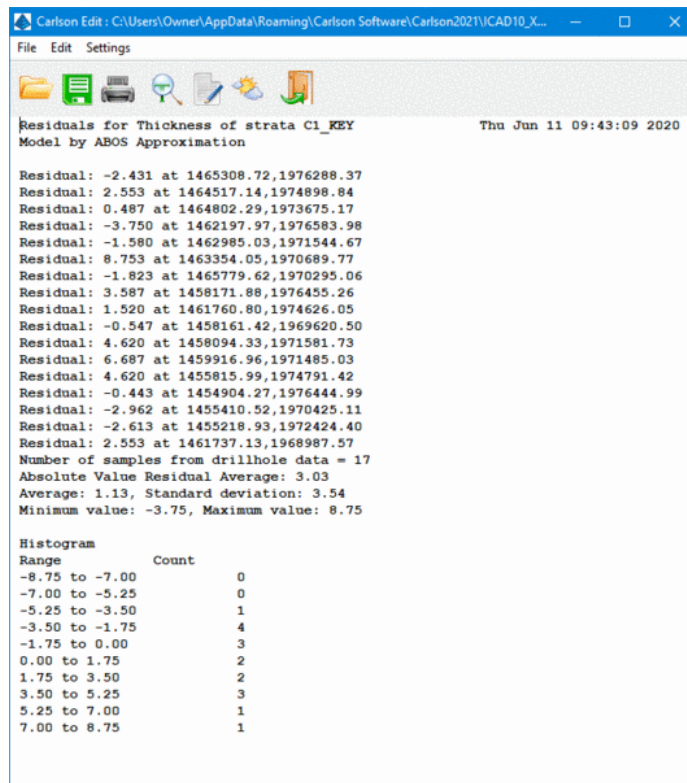
This command guides the user to select a modeling method. The concept is much like field checking. With field verification, you would pick a location for testing, then measure coal thickness or sulfur or base elevation and check the data against the model. If it is close, your model is good. If the testing is repeated, you can add up all the errors (the residuals) and make a determination of the effectiveness of one model against another. But even without field testing, you can take 25 drillholes and model with 24, then check the error residual at the removed drillhole. You can then repeat this "removal" process across all 25 drillholes, and verify the average residual error and the standard deviation of the residual error. This is exactly what the command Calculate Residuals does.



Shown below, for example, is a comparison all the modeling methods exported to Excel. This was done with Auto-Run Residuals command.

	A	B	C	D	E	F	G	H	I	J
1	Strata	Value	Model	Std Deviation	Samples	Absolute Value Residual Average	Avg	Min	Max	
2	C1_KEY	Thickness	Triangulation	4.62	9	4.46	2.37	-8.48	9.11	
3	C1_KEY	Thickness	Kriging	0.00	0	0.00	0.00	0.00	0.00	
4	C1_KEY	Thickness	Inv Dist 2.00 Pwr	4.03	17	3.47	-0.48	-7.00	8.84	
5	C1_KEY	Thickness	Polynomial	4.72	9	3.73	1.13	-8.93	9.58	
6	C1_KEY	Thickness	LeastSq	5.18	17	4.48	-0.94	-11.09	9.70	
7	C1_KEY	Thickness	ABOS	3.54	17	3.03	1.13	-3.75	8.75	
8										
9										

In this example, ABOS has the lowest standard deviation and absolute value residual average, making it a stronger candidate for modeling this drillhole dataset. The command Calculate Residuals will bring up a report that shows every drillhole and what the residual was. It will also create a Histogram, showing the number of residuals within certain ranges. This can be used to look for extreme values and fliers. However, it important to note that even though a gridding method may have the most favorable residual average, the contours and cross sections for that method should still be analyzed for accuracy.



Modeling Options

It is important to set the strata modeling options that are in Settings Carlson Configure command under Mining Settings. These settings are used in modeling commands that process strata from drillholes such as Make Strata Grid, Surface Mine and Underground Mine Reserves, and Strata Isopach Maps, just to name a few.

Mining Settings

StrataCalc Settings

InverseDist/LeastSq Max Samples: 20 Min Quadrant: 0 Max Quadrant: 20
 Search Radius: 10000.0 Use Conformance for Channel Samples
 Fill in Missing Strata Above/Below Existing Strata (Seam Stacking/Conformance):
 None Seam-Specific All
 Calculate Strata Pinchout Pinchout Near Zero < > Non-Zero
 Pinchout Zero Thickness Pinchout Key Only Adjust Non-Key for Estimated Key Strata
 Restrict Pinchout To Drillhole Elevation Range Include Strata Name in Bed Composite
 Composite Bed Qualities by Density Density Attribute Name: DENSITY
 Use Strata Limit Lines Auto Select All Strata Limit Lines
 Process Only Strata with Beds Process Only Strata with Definition
 Store Source Data in Grids Dip Angle Method: Direct
 Hole Dip Angle Direction: 0 = Down | 90 = Horizontal | 180 = Up
 StrataCalc Drillhole Selection Method ...
 On-Screen Drillholes StrataCalc File

Underground Room/Pillar Settings

Use 0 Values for Blank Entries in Coal Sections Draw Coal Sections Z at Thickness
 Prompt for Advancement Plane for Quantities
 Report Format for Quantities by Avg/Grid Method ...
 Standard Column Formatter

General Settings

Key Material Name: KEY SDPS Directory: c:/sdps/
 Bed Name Suffixes: KEY _KEY OB _OB PARTING _PARTING BOTTOM _BOTTOM
 Use Map Object Data as Properties Map Table Name: Key Field:

OK Cancel

The Inverse Distance and Least Squares settings control the data point search radius and the maximum samples which limits calculations to the nearest set of the specified number of data points. Inverse Distance and Least Squares can also be forced to use a minimum and/or maximum number of data points from each quadrant NE, SE,

SW and NW. These inverse distance settings are used whenever modeling by inverse distance.

During strata correlation, the program matches strata with the same name between drillholes. When a strata name is missing in a drillhole, there are three possibilities. Either the program can skip that drillhole for modeling that strata, the strata pinched out, or the drillhole did not reach the strata and the strata position can be modeled by conformance. The method to use is determined by the Pinch Out and Conformance settings in this dialog.

If you turn off Pinch Out, then the program will skip a drillhole with a pinch out case for modeling that strata. Otherwise the missing strata will be given a negative thickness at the drillhole. The thickness is negative so that when modeled with the other positive thickness drillholes, the pinch out or zero thickness position will be somewhere midway between the missing and existing drillholes. The slide bar "Near Zero <-> Non-Zero" controls the amount of the negative thickness. The Near Zero setting will make a smaller negative value which moves the pinch out position closer to the missing strata drillhole. Likewise Non-Zero makes a larger negative value which moves the pinch out position closer to the drillhole with the strata.

For Conformance, turning off conformance will make the program skip a drillhole with a conformance case for modeling that strata. With conformance active, the missing strata position at the partial drillhole will be calculated by modeling the thickness between the missing strata and a marker strata that does exist in the drillhole. This thickness is modeled with inverse distance using drillholes where both strata exist. Then the thickness is added to the marker strata to locate the missing strata in the partial drillhole. Conformance can be set to Seam-Specific which allows only specified strata to be marker strata. Additionally, the specified marker strata will only conform with specified target strata. The marker and target strata names are set in Define Strata.

Case Study #4: Limestone Block Modeling

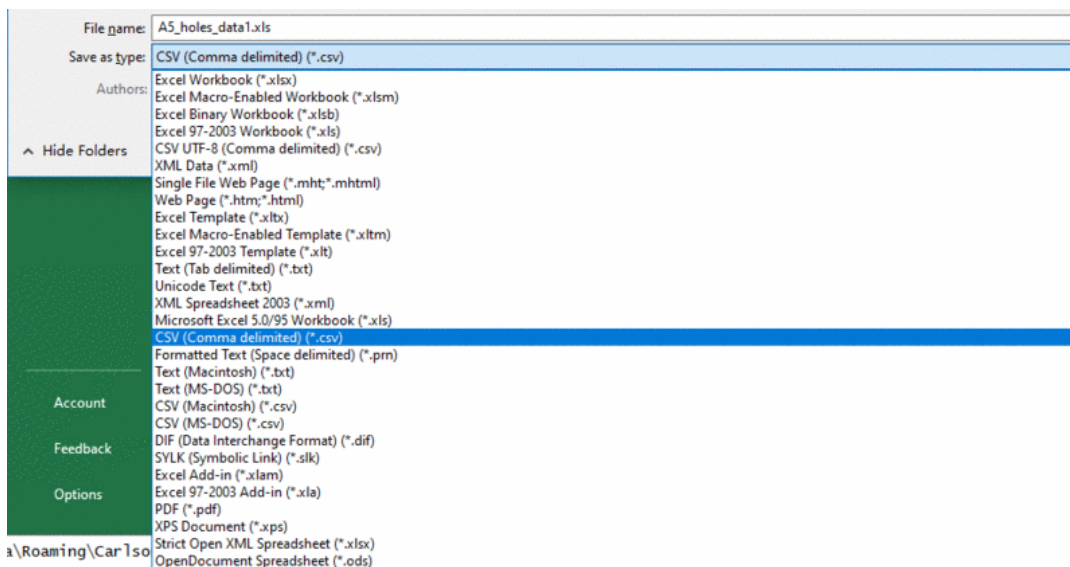
This lesson will step through modeling a limestone quarry using the Carlson block model routines. It goes through the process from start to finish, importing drillholes to final reporting and viewing graphics. Most of these commands are found in the Block Model menu of the Carlson Geology Module.

Step 1-Import the Drillhole and Face Data

To import the drillhole data, it is important to create an ASCII file with a repetitive series of columns for the collar x,y,z position, followed by columns for the downhole information. Shown here is an example of the spreadsheet to import.

	A	B	C	D	E	F	G	H
1	HOLE #	X	Y	Z	Z Top	Z Bottom	DESCRIPTION	Attribute (Color)
2	H-1-04	24463.27	18245.63	676.63	676.63	658.28	Dirt	Dirt
3	H-1-04	24463.27	18245.63	676.63	658.28	651.53	Bastard Stone	Bastard Stone
4	H-1-04	24463.27	18245.63	676.63	651.53	649.24	Void	Void
5	H-1-04	24463.27	18245.63	676.63	649.24	616.82	Bastard Stone	Bastard Stone
6	H-1-04	24463.27	18245.63	676.63	616.82	616.82	Bed	Bed
7	H-1-04	24463.27	18245.63	676.63	616.82	615.15	Bastard Stone	Bastard Stone
8	H-1-04	24463.27	18245.63	676.63	615.15	613.4	Rustic Buff	Rustic Buff
9	H-1-04	24463.27	18245.63	676.63	613.4	611.73	Standard Buff	Standard Buff
10	H-1-04	24463.27	18245.63	676.63	611.73	611.52	Crawfoot	Crawfoot
11	H-1-04	24463.27	18245.63	676.63	611.52	611.1	Rustic Buff	Rustic Buff
12	H-1-04	24463.27	18245.63	676.63	611.1	610.68	Bed	Bed
13	H-1-04	24463.27	18245.63	676.63	610.68	605.76	Standard Buff	Standard Buff
14	H-1-04	24463.27	18245.63	676.63	605.76	605.09	Old Gothic Buff	Old Gothic Buff
15	H-1-04	24463.27	18245.63	676.63	605.09	604.05	Standard Buff	Standard Buff
16	H-1-04	24463.27	18245.63	676.63	604.05	602.38	Old Gothic Buff	Old Gothic Buff
17	H-1-04	24463.27	18245.63	676.63	602.38	599.3	Standard Buff	Standard Buff
18	H-1-04	24463.27	18245.63	676.63	599.3	598.47	Rustic Buff	Rustic Buff
19	H-1-04	24463.27	18245.63	676.63	598.47	598.18	Standard Buff w/ Vertical Seam	Standard Buff w/ Vertical Seam
20	H-1-04	24463.27	18245.63	676.63	598.18	597.89	Standard Buff	Standard Buff
21	H-1-04	24463.27	18245.63	676.63	597.89	597.06	Standard Grey	Standard Grey
22	H-1-04	24463.27	18245.63	676.63	597.06	596.06	Standard Buff	Standard Buff
23	H-1-04	24463.27	18245.63	676.63	596.06	595.6	Old Gothic Buff	Old Gothic Buff
24	H-1-04	24463.27	18245.63	676.63	595.6	593.6	Standard Buff	Standard Buff
25	H-1-04	24463.27	18245.63	676.63	593.6	588.77	Standard Grey	Standard Grey
26	H-1-04	24463.27	18245.63	676.63	588.77	587.85	Variegated	Variegated
27	H-1-04	24463.27	18245.63	676.63	587.85	587.52	Standard Grey	Standard Grey
28	H-1-04	24463.27	18245.63	676.63	587.52	585.77	Standard Buff	Standard Buff
29	H-1-04	24463.27	18245.63	676.63	585.77	580.35	Silver Buff	Silver Buff

To import the drillholes, it will need to be in ASCII form, use Excel to save the file in ASCII form with a ".csv" extension.



Answer Y to the question regarding saving the current worksheet. This results in a comma-separated file ASCII, such as this typical example, viewed with Notepad:

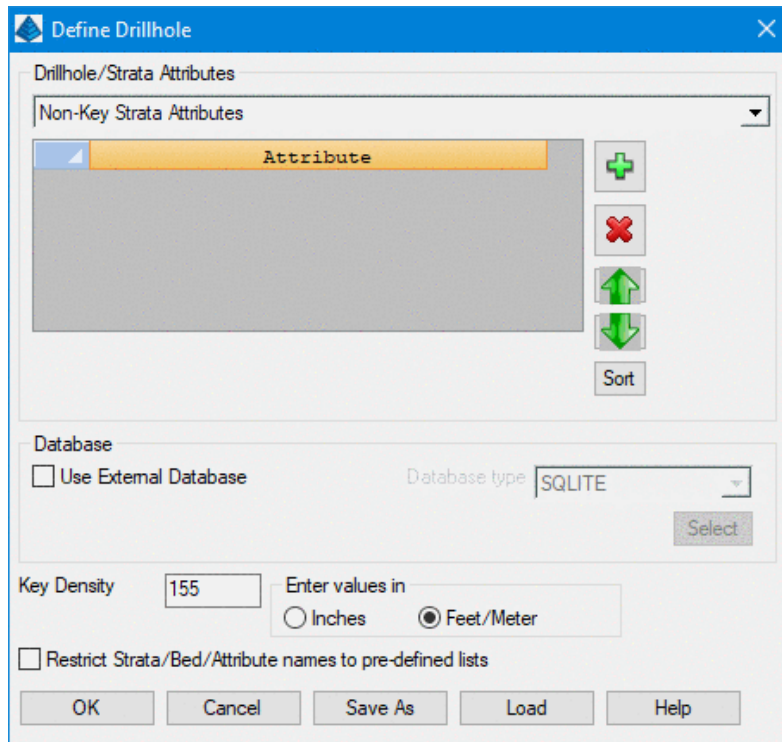
```

HOLE #,X,Y,Z,Z Top,Z Bottom,DESCRIPTION
1-03,25158.6200,18332.4000,652.9800,652.98,634.23,Bastard Stone,Bastard Stone
1-03,25158.6200,18332.4000,652.9800,634.23,634.06,Bed, Bed
1-03,25158.6200,18332.4000,652.9800,634.06,632.98,Standard Buff, Standard Buff

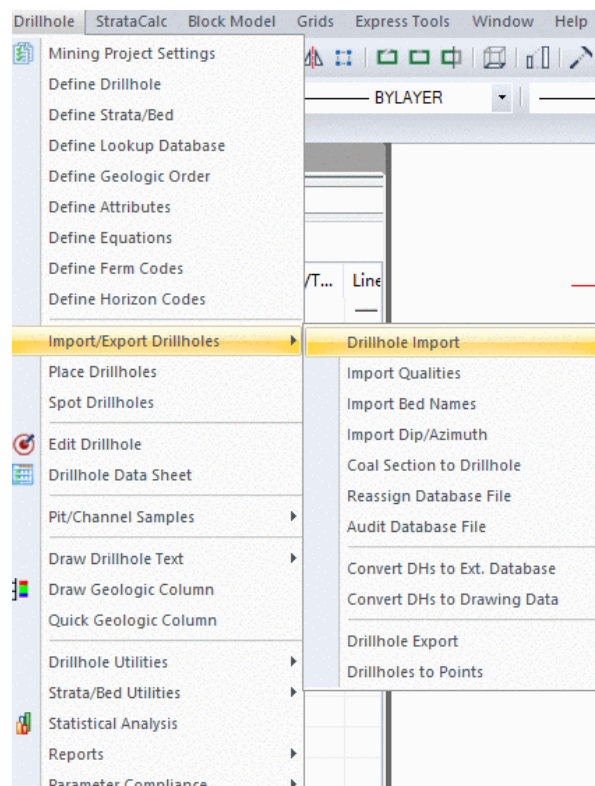
```

Notice that this file contains one "header" line consisting of the title, that we can ignore.

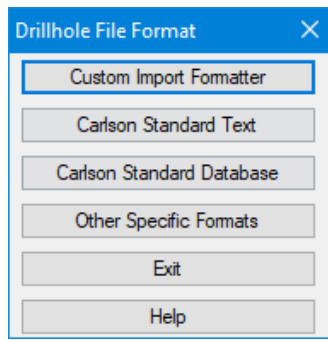
Before importing the drillhole, you will need to create a .CH file via the Drillhole Pulldown Menu > Define Drillhole. Here you only need to specify the default density for key material (note that there are other methods of defining density that will overrule this setting).



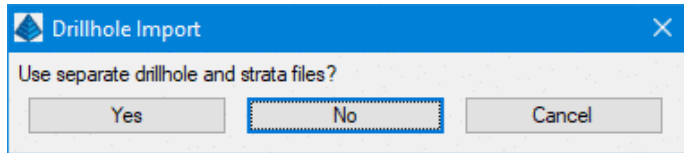
Next we bring this file into Carlson by using the command Drillhole Import found in the Import/Export Drillhole flyout, underneath the Drillhole pulldown menu.



Within this command, choose the Custom Import Formatter.

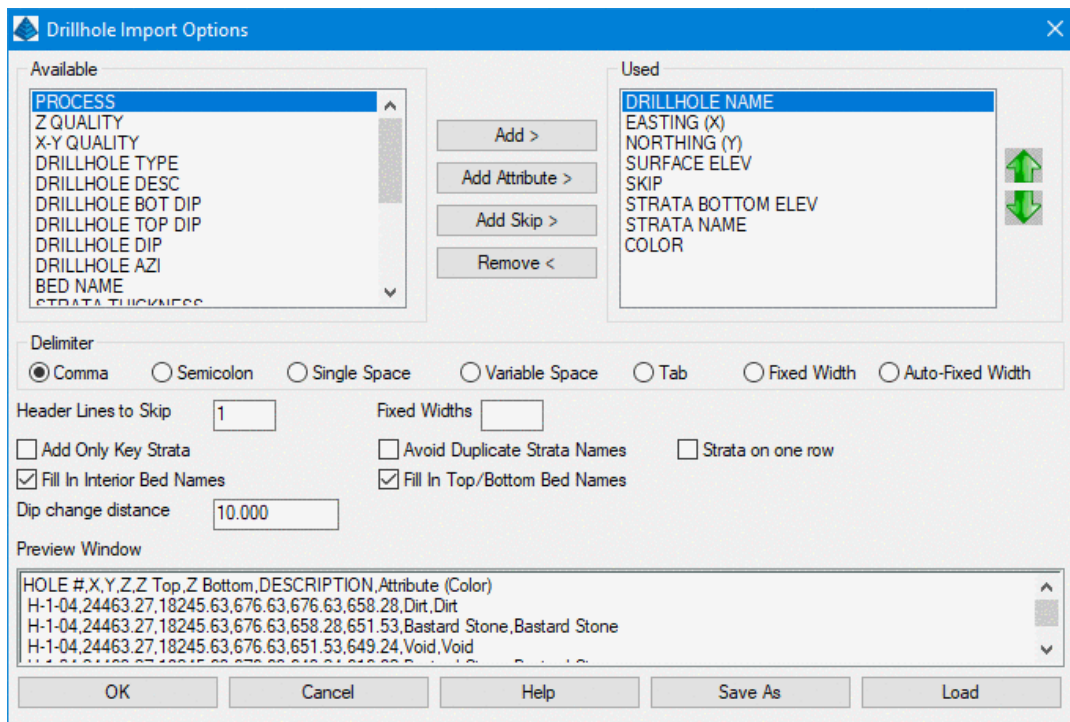


This window will appear:

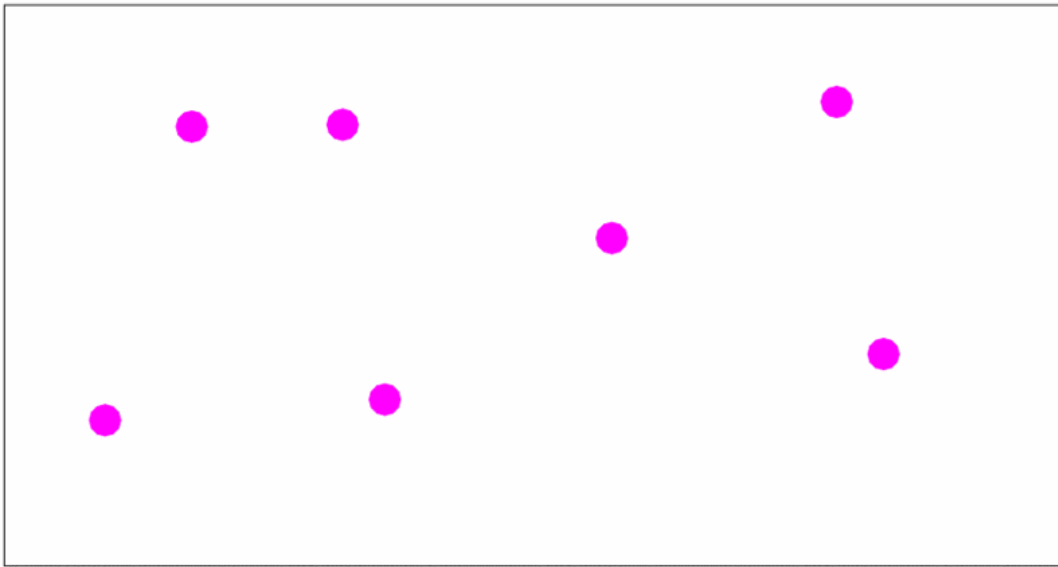


Select "No", as all the data is in one CSV file.

Note that the default answer is "No" and can be selected by pressing Enter. Load the correct ".csv" file containing the ASCII drillhole data. Now organize the right-hand column to correspond to the ASCII file, as shown here:

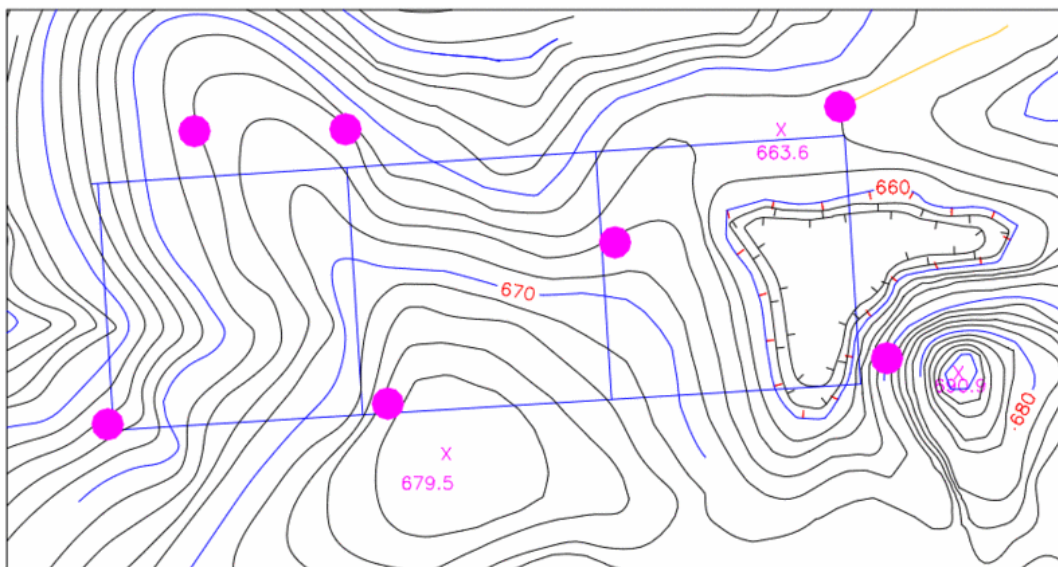


Note how you can study the data file organization in the "Preview Window". This format can be given a name and saved by pressing the Save button. Note that we have specified "1" header line to skip (skip the title line), and also notice that we have chosen to skip the "Z Top" column, since the bottom elevation defines the strata, measuring down from the top, collar elevation. Press OK to continue. As a result of the import, we now have 7 drillholes in our area of study, as shown below:

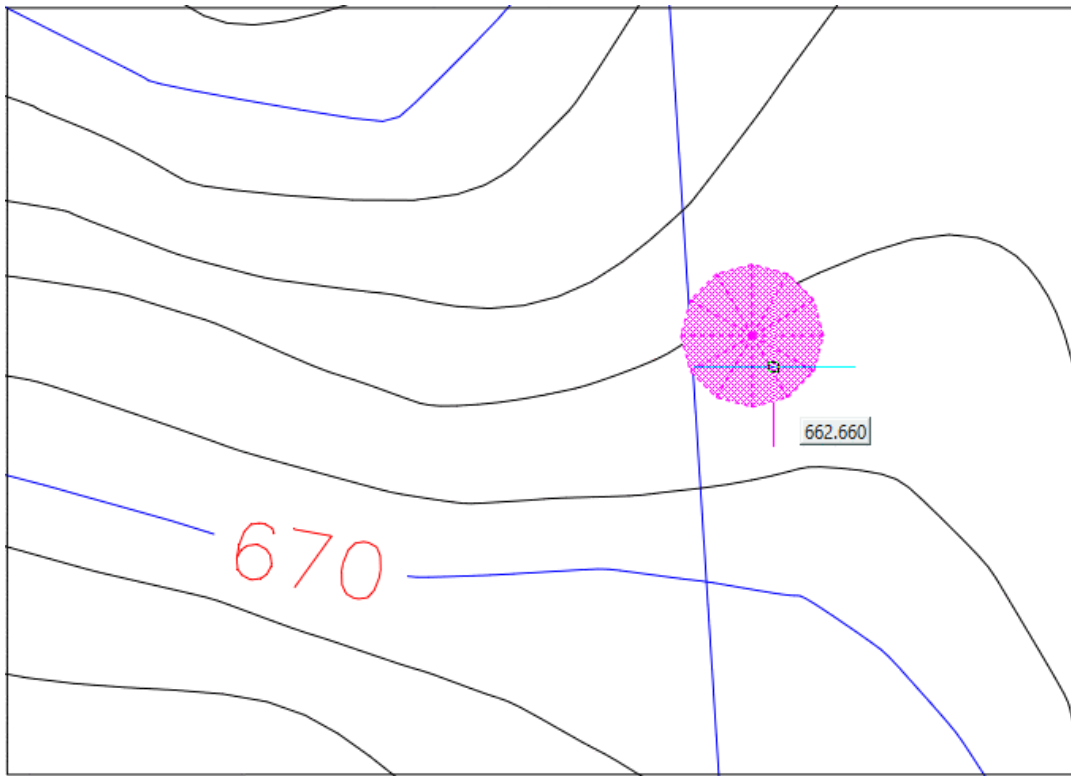


Importing Drillholes is a critical process in geologic modeling, so it is important to be familiar with the precise techniques to accomplish this. Carlson offers extremely flexible importing. The drawing with the 7 new drillholes can now be saved with any file name desired.

You have the option to import these drillholes while already in a drawing (such as a topo map of the site), or you can import the drillholes into a blank drawing then insert additional surface features like contours. The Insert command can be found under the Draw pulldown in Carlson. In this case, we'll insert a contour file called Block Model Contours, to obtain the combined drawing below:



To verify contour elevations, and to check whether the top of the drillholes are close to the contour grades of the contours, use Drawing Inspector under the Inquiry pulldown menu. You will notice that all the holes top out at just about the right elevation you would expect based on the nearby contours, the only slight exception being the slightly lower drillhole shown below, where some of the surface could have been removed where the hole was drilled. It's just a few feet low:



Note also that if you inspect the blue boundary lines, they, too have elevation. Those lines should be set to zero using the command, "3D Entity to 2D" under Edit. Otherwise, they will impact surface modeling. It's a good idea to turn off the Drawing Inspector when you are not using it. Also, by right clicking with the mouse button when Drawing Inspector is on, you can change what you inspect (layers versus elevations, for example).

Step 2-Set up Strata Definitions to Colorize the Strata

This is a one-time process. The Strata Definitions file is used by Draw Geologic Column, Fence Diagram, and by the routines that color and display the block model. It can also impact tonnage calculations, because you can set the strata density within Strata Definitions. The dialog appears as shown below. The asterisk can be used to apply the definition to anything that begins with the letters preceding the asterisk. In this way, Standard Buff with Vertical Seam would be treated as all the other Standard Buff strata in terms of coloring and modeling.

Strata/Bed Name	Full Name	Key	Color	Hatch	Hatch Scale	Density	Layer	Draw Legend
1 STANDARDBUFF*	Standard Buff	<input checked="" type="checkbox"/>	Yellow	SOLID	25.00	BY_DRILLHOLE	Standard Buff	<input checked="" type="checkbox"/>
2 STANDARDGREY*	Standard Grey	<input checked="" type="checkbox"/>	Grey	SOLID	25.00	BY_DRILLHOLE	Standard Grey	<input checked="" type="checkbox"/>
3 SILVERBUFF*	Silver Buff	<input checked="" type="checkbox"/>	Light Green	SOLID	25.00	BY_DRILLHOLE	Silver Buff	<input checked="" type="checkbox"/>
4 RUSTICBUFF*	Rustic Buff	<input checked="" type="checkbox"/>	Orange	SOLID	25.00	BY_DRILLHOLE	Rustic Buff	<input checked="" type="checkbox"/>
5 CRAWFOOT*	Crawfoot	<input checked="" type="checkbox"/>	Blue	SOLID	25.00	BY_DRILLHOLE	Crawfoot	<input checked="" type="checkbox"/>
6 RUSTICGREY*	Rustic Grey	<input checked="" type="checkbox"/>	Dark Blue	SOLID	25.00	BY_DRILLHOLE	Rustic Grey	<input checked="" type="checkbox"/>
7 DIRT	DIRT	<input type="checkbox"/>	Brown	SOLID	25.00	BY_DRILLHOLE	Dirt	<input checked="" type="checkbox"/>
8 VOID	VOID	<input type="checkbox"/>	Red	AR-RROOF	25.00	BY_DRILLHOLE	VOID	<input checked="" type="checkbox"/>
9 BASTARDSTONE*	Bastard Stone	<input type="checkbox"/>	Black	SOLID	25.00	BY_DRILLHOLE	BS	<input checked="" type="checkbox"/>
10 OLDGOTHICBUFF*	Old Gothic Buff	<input type="checkbox"/>	Yellow	BEDCHT	25.00	BY_DRILLHOLE	OGB	<input checked="" type="checkbox"/>
11 OLDGOTHICGREY*	Old Gothic Grey	<input type="checkbox"/>	Grey	BEDCHT	25.00	BY_DRILLHOLE	OGG	<input checked="" type="checkbox"/>
12 BED	BED	<input type="checkbox"/>	Black	SOLID	25.00	BY_DRILLHOLE	BED	<input checked="" type="checkbox"/>
13 RUSTICSILVER*	Rustic Silver Buff	<input checked="" type="checkbox"/>	Blue	SOLID	25.00	BY_DRILLHOLE	RSB	<input checked="" type="checkbox"/>
14 VARIEGATED*	Variagated	<input checked="" type="checkbox"/>	Cyan	SOLID	25.00	155.00	VG	<input checked="" type="checkbox"/>
15 CRAWFOOT		<input type="checkbox"/>	Cyan	ZIGZAG	25.00	BY_DRILLHOLE	crawfoot	<input checked="" type="checkbox"/>

The columns to be shown in the spread sheet can be set using Column Options. The values can be edited with in the spread sheet shown above or by selecting the strata row and using Edit button.

The screenshot shows the 'Strata Definition' dialog box for a strata named 'STANDARDBUFF'. The 'Strata Name' is 'STANDARDBUFF', 'Full Name' is 'Standard Buff', and 'Layer' is 'Standard Buff'. The 'Hatch Name' is 'SOLID', 'Hatch Scale' is '25.00', and 'Hatch Azimuth' is '90°0'0"'. The 'Type of strata' is set to 'Key' (radio button selected). The 'Recovery Percent' is '100.00'. The 'Non-Key Thickness To Add To Key (Dilution)' is '0.00' for both 'Above Key' and 'Below Key'. The 'Key Thickness To Remove From Key (Loss)' is '0.00' for both 'Key Top' and 'Key Bottom'. The 'Use for modeling (pinch out/conformance)' checkbox is checked. The 'Use as seam-specific conformance marker' checkbox is unchecked. The 'Marker Priority Level' is '1'. The 'Strata Split Names' are empty. The 'Additional Attributes' section has 'Attr. Name' empty and 'Update', 'Add', and 'Remove' buttons. The 'OK' and 'Cancel' buttons are at the bottom.

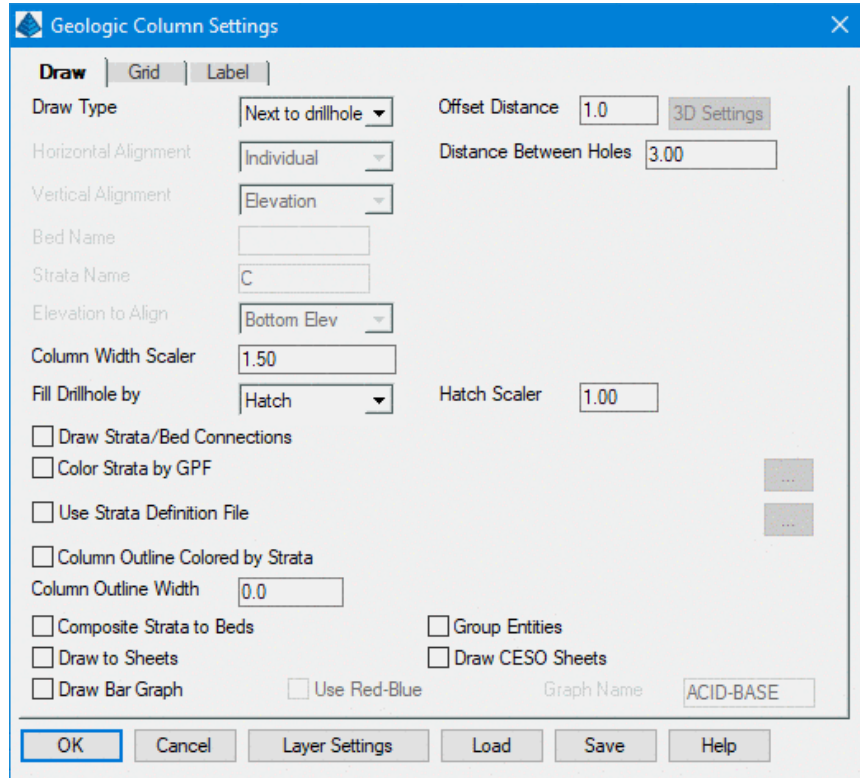
Contrast the StandardBuff settings with those for Dirt, which we have colored black and designated "Non-Key", which means it is waste, not a product you are mining to sell.

The screenshot shows the 'Strata Definition' dialog box for a strata named 'DIRT'. The 'Strata Name' is 'DIRT', 'Full Name' is 'DIRT', and 'Layer' is 'Dirt'. The 'Hatch Name' is 'SOLID', 'Hatch Scale' is '25.00', and 'Hatch Azimuth' is '90°0'0"'. The 'Type of strata' is set to 'Non-key' (radio button selected). The 'Recovery Percent' is '100.00'. The 'Non-Key Thickness To Add To Key (Dilution)' is '0.00' for both 'Above Key' and 'Below Key'. The 'Key Thickness To Remove From Key (Loss)' is '0.00' for both 'Key Top' and 'Key Bottom'. The 'Use for modeling (pinch out/conformance)' checkbox is checked. The 'Use as seam-specific conformance marker' checkbox is unchecked. The 'Marker Priority Level' is '1'. The 'Strata Split Names' are empty. The 'Additional Attributes' section has 'Attr. Name' empty and 'Update', 'Add', and 'Remove' buttons. The 'OK' and 'Cancel' buttons are at the bottom.

Step 3-Draw Geologic Column

Once you have verified your colors for the strata (again, a one-time process using Strata Definitions), it is valuable to confirm the quality of the drillhole import by drawing the geologic column. One method is to draw the columns next to the drillholes. Choose Draw Geologic Column at the bottom of the Drillhole pulldown menu. Possible settings

for the dialog are shown below:

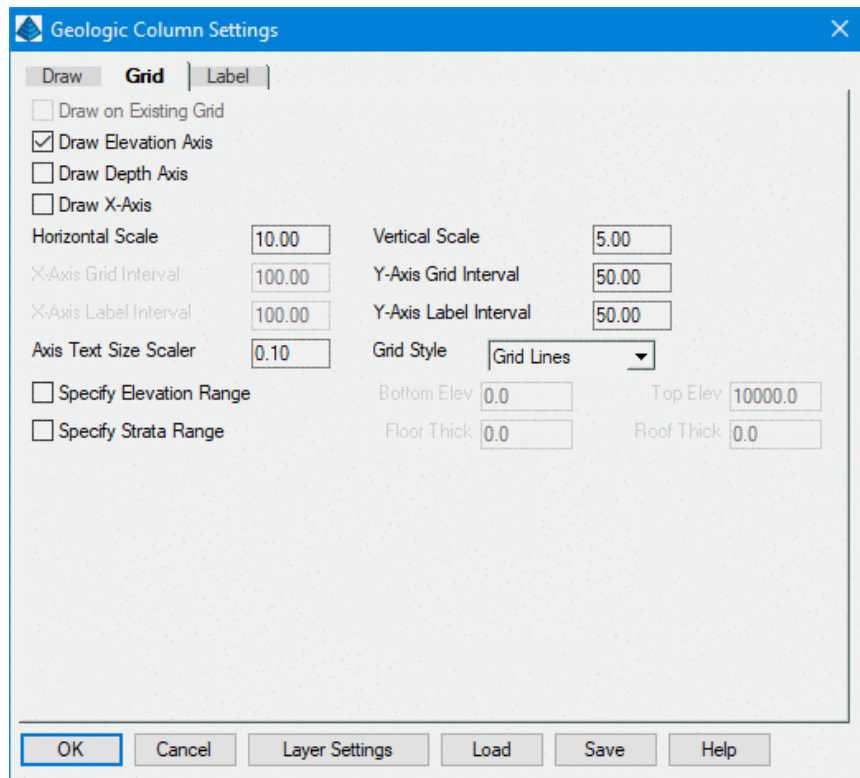


Geologic Column Settings

Draw | Grid | Label

Draw Type: Next to drillhole
Horizontal Alignment: Individual
Vertical Alignment: Elevation
Bed Name:
Strata Name: C
Elevation to Align: Bottom Elev
Column Width Scaler: 1.50
Fill Drillhole by: Hatch
Hatch Scaler: 1.00
Offset Distance: 1.0
Distance Between Holes: 3.00
3D Settings
...
...
 Draw Strata/Bed Connections
 Color Strata by GPF
 Use Strata Definition File
 Column Outline Colored by Strata
Column Outline Width: 0.0
 Composite Strata to Beds
 Group Entities
 Draw to Sheets
 Draw CESO Sheets
 Draw Bar Graph
 Use Red-Blue
Graph Name: ACID-BASE

OK Cancel Layer Settings Load Save Help

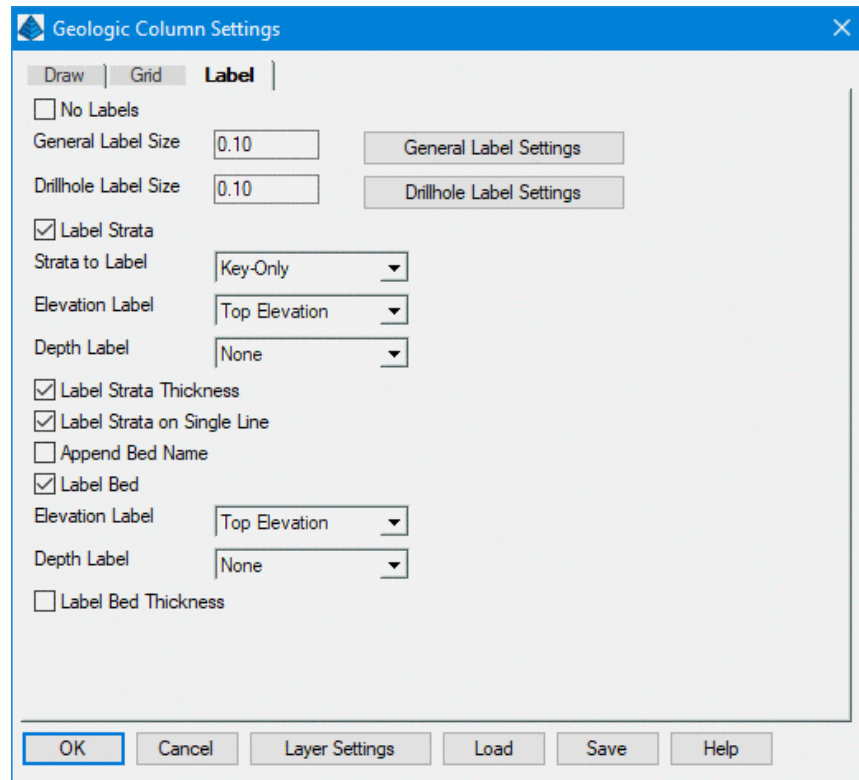


Geologic Column Settings

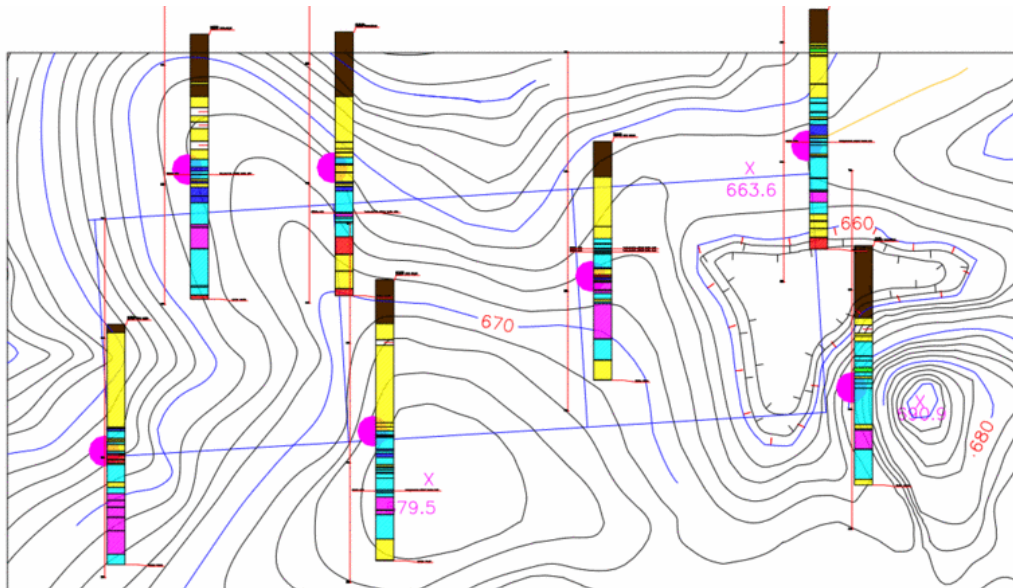
Draw | **Grid** | Label

Draw on Existing Grid
 Draw Elevation Axis
 Draw Depth Axis
 Draw X-Axis
Horizontal Scale: 10.00
Vertical Scale: 5.00
X-Axis Grid Interval: 100.00
Y-Axis Grid Interval: 50.00
X-Axis Label Interval: 100.00
Y-Axis Label Interval: 50.00
Axis Text Size Scaler: 0.10
Grid Style: Grid Lines
 Specify Elevation Range
Bottom Elev: 0.0
Top Elev: 10000.0
 Specify Strata Range
Floor Thick: 0.0
Roof Thick: 0.0

OK Cancel Layer Settings Load Save Help



These settings produce the plot shown here:

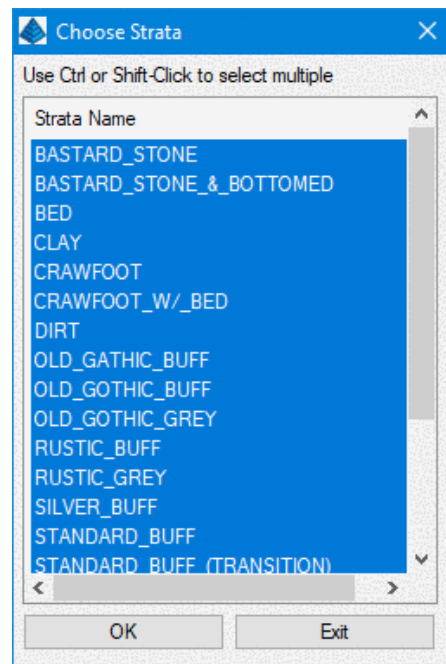


This is the first indication of the location of the yellow standard buff, as well as the location of silver grey and other colored zones. The location of the black overburden material is also clear in the plot.

Step 4-Assign a Bed Name to All Strata

The block modeling routines require that the various strata be organized into beds. You might have a top 80-foot bed and a lower 100-foot bed in large deposits of ore. If you don't want to distinguish beds, just put all the strata into one bed name, like Stone. Note that if a column "Bed Name" was made in the original Excel file, all the

strata could be imported with a bed name without the need for modification. To add Beds "after-the-fact", choose Drillhole Pulldown Menu > Strata/Bed Utilities > Assign Bed Names. Select all drillholes and enter "Stone" for the bed name, for example. Enter N for No when prompted for "Use Parameter Filter?". Then select all strata in the list at right. Highlight and click OK.



Step 5-Surface Mine Reserves

You are now ready to get a volume of any material desired. Before you do the command, you will need a closed polyline perimeter. Note in the above graphic plot with contours, we have 3 blue squares that might be areas of interest for volumes. Each might represent a mining block. But in reality, these are not closed polylines, but are drawn as individual polylines in approximate N-S and E-W directions. To create closed polylines where the zone of interest is "enclosed" by other polylines, a useful command is "Boundary Polyline" under the Draw Pulldown Menu. Choose this command.

Pick all the blue polylines that "bound" the rectangle of interest (or that bound all the rectangles). Do not pick any contours or other polylines. Enter a snap tolerance of 1 (to "bridge" gaps of up to 1 foot). Enter the layer to draw the polylines in (CLAYER would put them in the current layer). Then pick inside, and the closed, rectangular polylines are drawn.

Now run the command Surface Mine Reserves under the Stratacalc Pulldown Menu. Fill out the dialog as shown, being sure to select the "Block Model" method. This selection does an "on-the-fly" block model. The Geologic Model selection would work from a stored Geologic Model file, which can be a block model or a strata-based model. Specify the block by elevation, entering the bottom and top of the zone of interest (we did 580 to 620 here).

Surface Mine Reserves

Process | Adjustment | Output

Modeling Method: Block Model

Top Constraint: Screen Model Surface File Elevation Top Elev: 620

Bottom Constraint: Model Surface File Elevation Elevation+Surface Bottom Elev: 580

Use Auto-Run Use Surface History Merge Bench Quantities Percent: 1.000

Strata To Include: All Selected Grid Subdivision Precision: High

Calculate Strata Qualities Breakout Quantities By Attributes: Interpolate Fixed Non-Key Qualities

Use Named Pit Areas Store Results In Pits Bench#: 1

Adjust Pits Manager Use Property Boundaries Use Resource/Reserve Classification

OK Cancel Load Save Help

Surface Mine Reserves

Process | **Adjustment** | Output

Use Drillhole Elevations In Surface Ignore Zero Elevations

Use Triangulation Subdivision Use Global Trend Extrapolation

Use Highwall Slopes Set

Recovery

Recovery Percent: 100.000 Use Strata Definitions

Use Recovery Attribute Attribute Name: REC

Density

Density: 100.000 Use Strata Definitions

Use Density Attribute Attribute Name: DENSITY

Min Key Thickness To Use (ft): 0.000 Min Depth To Use (ft): 0.000

Min Separable Parting Thickness (ft): 0

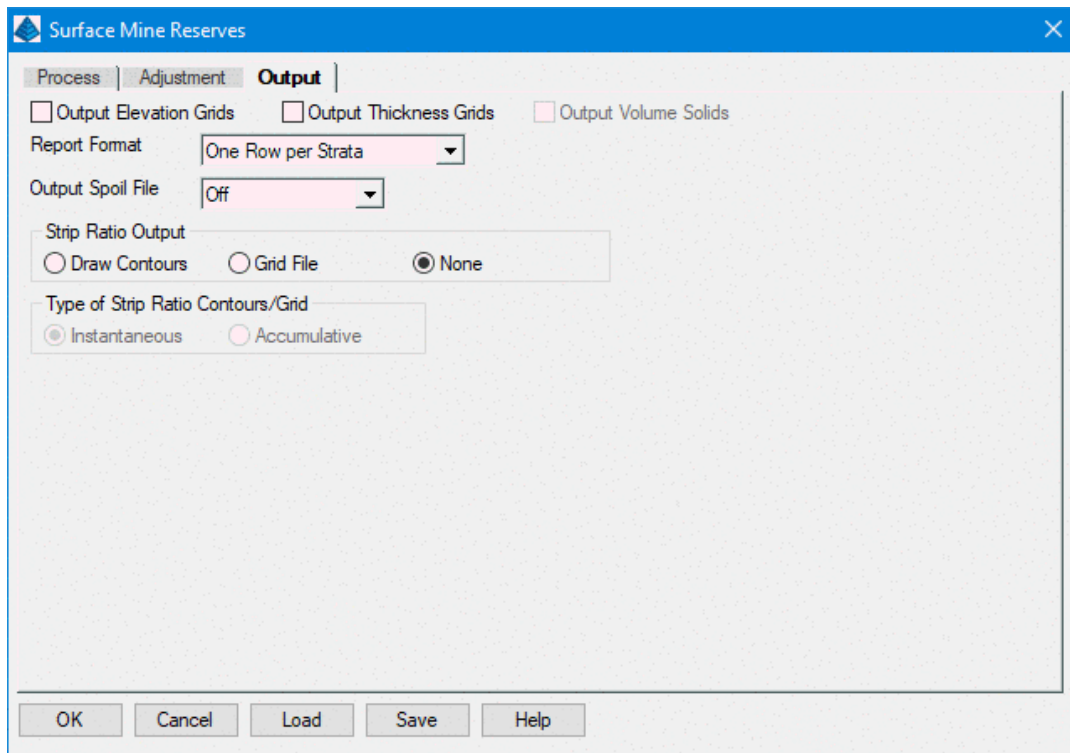
Non-Key Thickness To Add To Key: Dilution (ft)

Above Key: 0.000 Below Key: 0.000 Use Strata Definitions

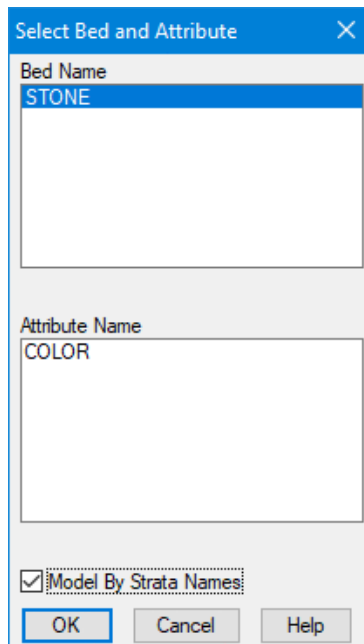
Key Thickness To Remove From Key: Loss (ft)

Key Top: 0.000 Key Bottom: 0.000 Use Strata Definitions

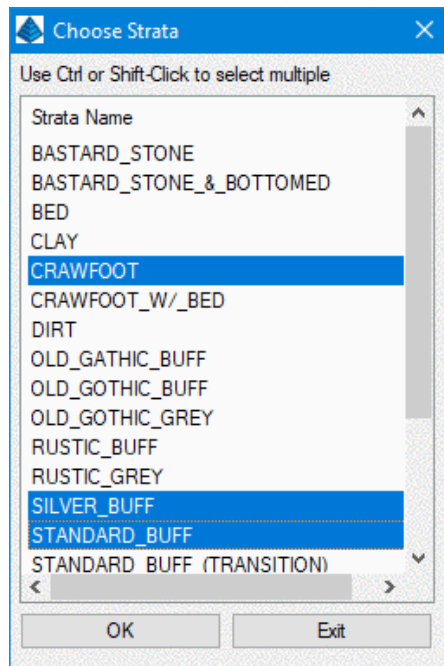
OK Cancel Load Save Help



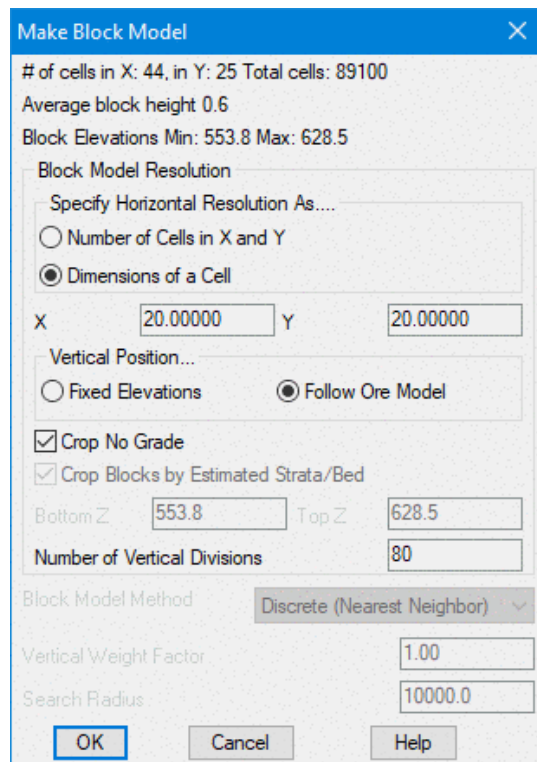
Also be sure to select "Calculate Strata Qualities" and "Breakout Quantities by Attributes", options near the bottom left of the dialog on the Process tab. After you select all 7 drillholes to model, you will get this dialog:



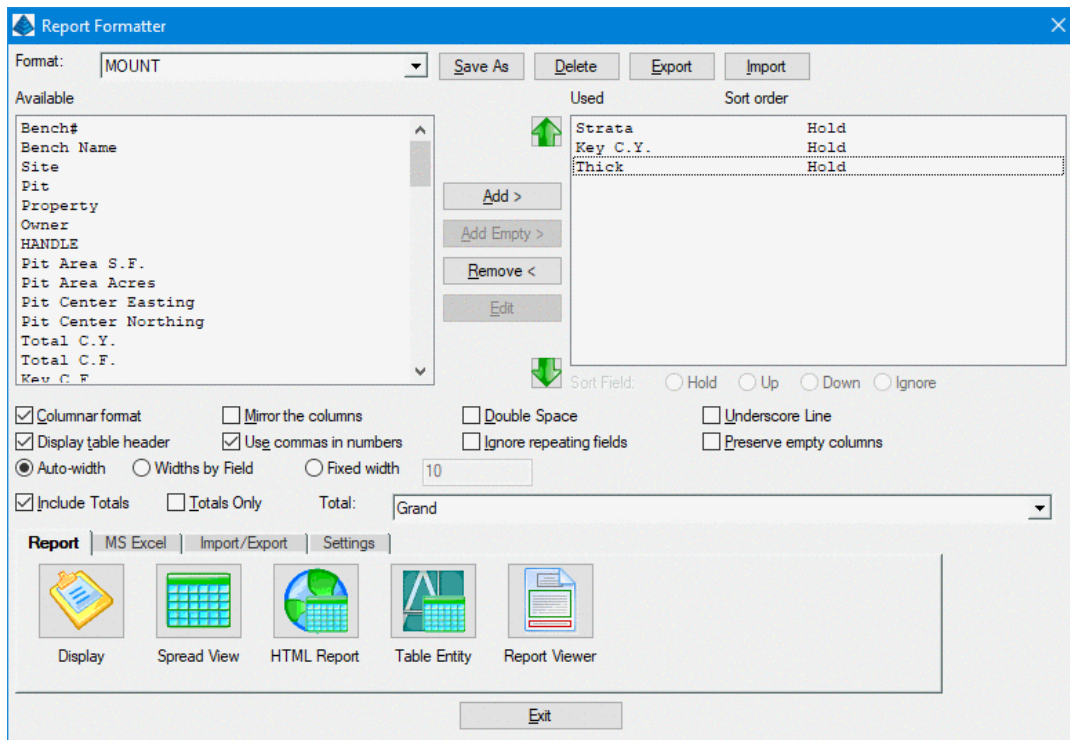
Here we are only interested in modeling the Strata Names, as the color attribute cannot be mathematically modeled (with the exception of the Strata Name, only numeric values may be modeled). Next you can pick any of the strata to report by selecting them:



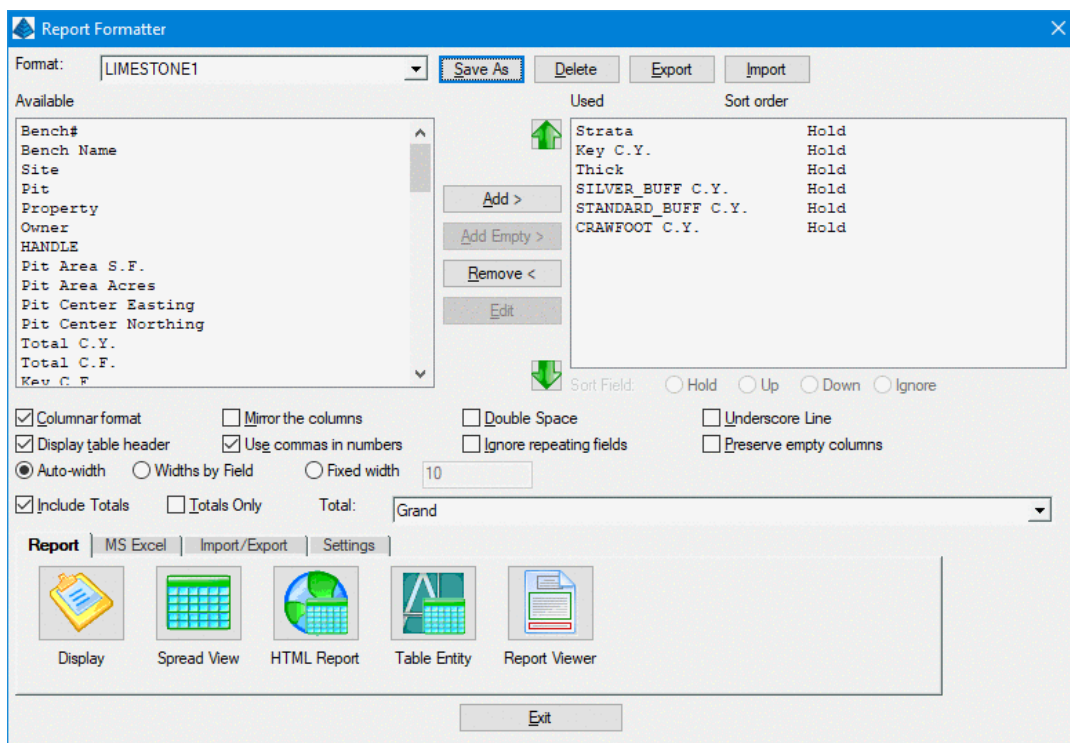
If we choose Crawfoot, SilverBuff and StandardBuff, quantities will be calculated for that block. Next you pick the limits of the study area, from lower left (well below and left of the drillholes) to upper right (well above and to the right of the drillholes) and set your gridding resolution. For example, if you have 40' of elevation range, 80 vertical zones will give you about 0.5' per zone for analysis.



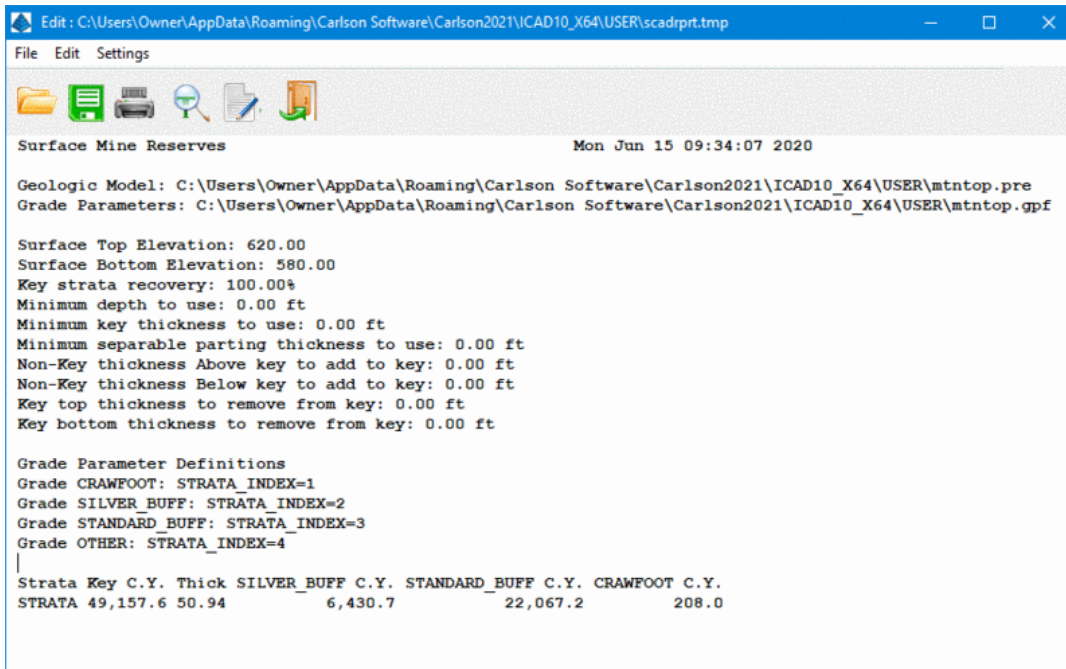
Pick one of the inclusion perimeters, and you are taken to the "Report Formatter" as shown below. Here it is shown as it might appear with a fresh installation of Carlson, completely unformatted.



Move to the right the Silverbuff C.Y., the Standardbuff C.Y. and the Crawfoot C.Y. You can name the format in the upper middle box as Limestone1, so it is a report option available to you in the future (see below):

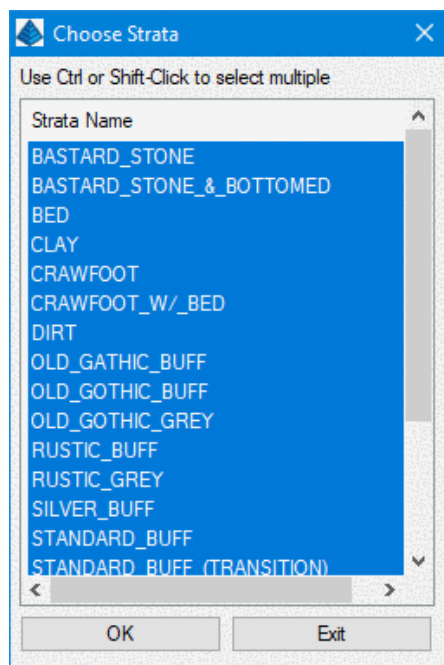


Click Display in the lower left of the screen, and you get the first report:

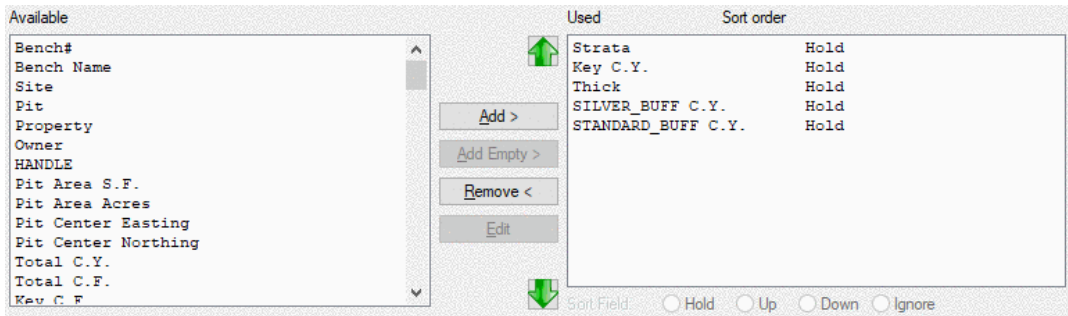


Step 6-Reporting in Cubic Meter: Making User-Defined Attributes for Reporting

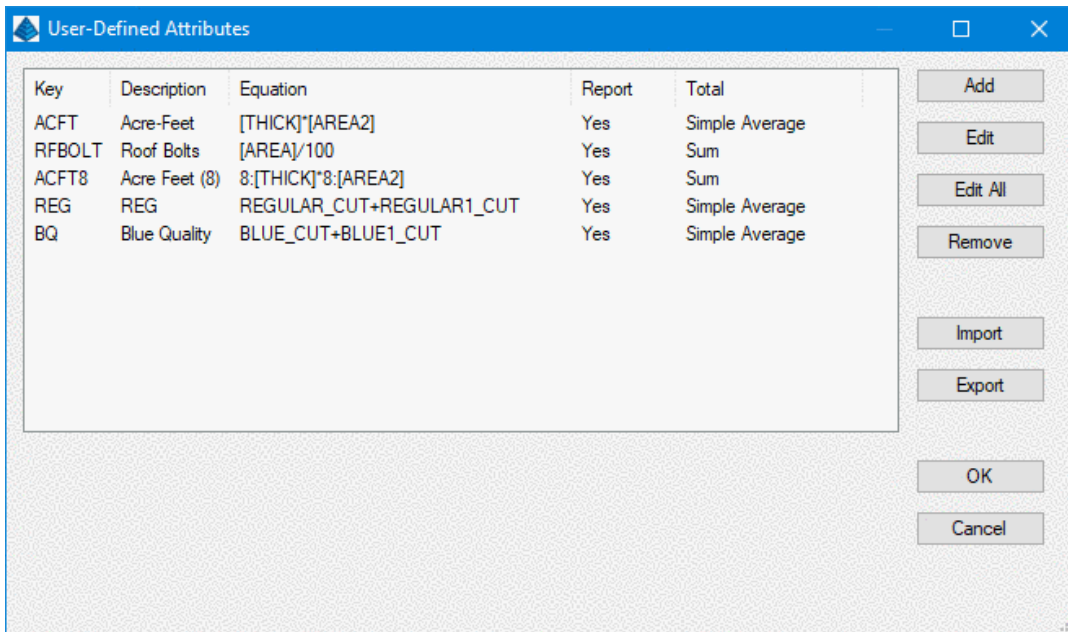
When configured to "English" units, that is, feet, the program defaults volumes in the form of cubic yards. If you wanted to also output cubic meters, then the default output variable for cubic yards needs to be multiplied by 0.765 to produce cubic meter. The variables "known" to the program can be listed within the Report Formatter by clicking the Attribute Options button at the bottom of the screen. A formula must be created for all of the named strata to convert cubic yards to cubic meter. Therefore, to obtain all of the strata in the Report Formatter, it is necessary to re-run Surface Mine Reserves using the same settings in the dialog, but when prompted for Choose Strata, select all of the strata, as shown.



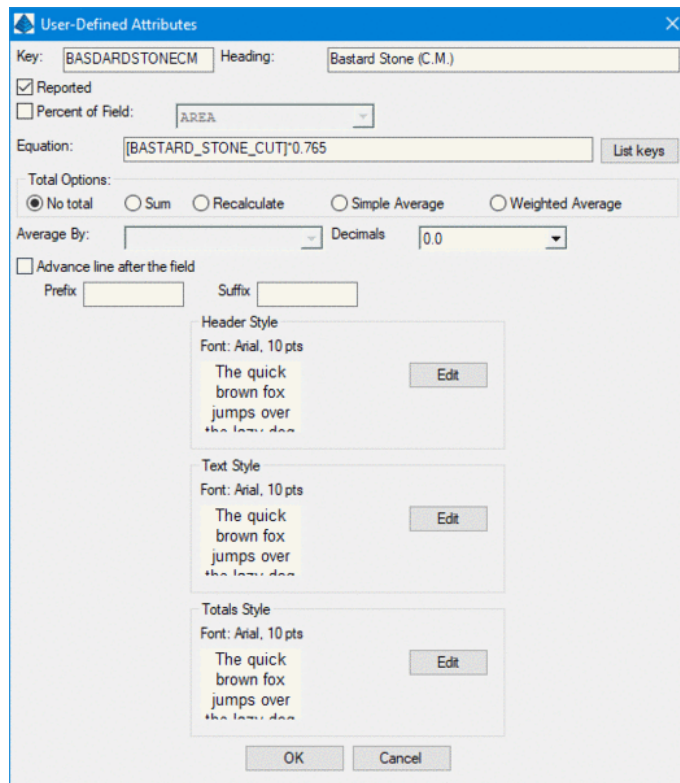
Then when you press OK and complete the gridding process, all of the strata appear in the Report Formatter, either in the left or right column.



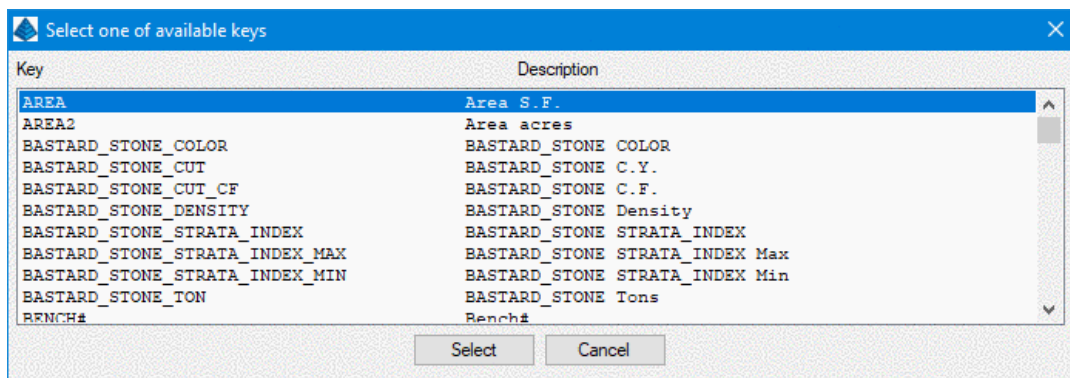
The next step is to click the User Attribute button at the bottom of the screen, which then produces the following dialog:



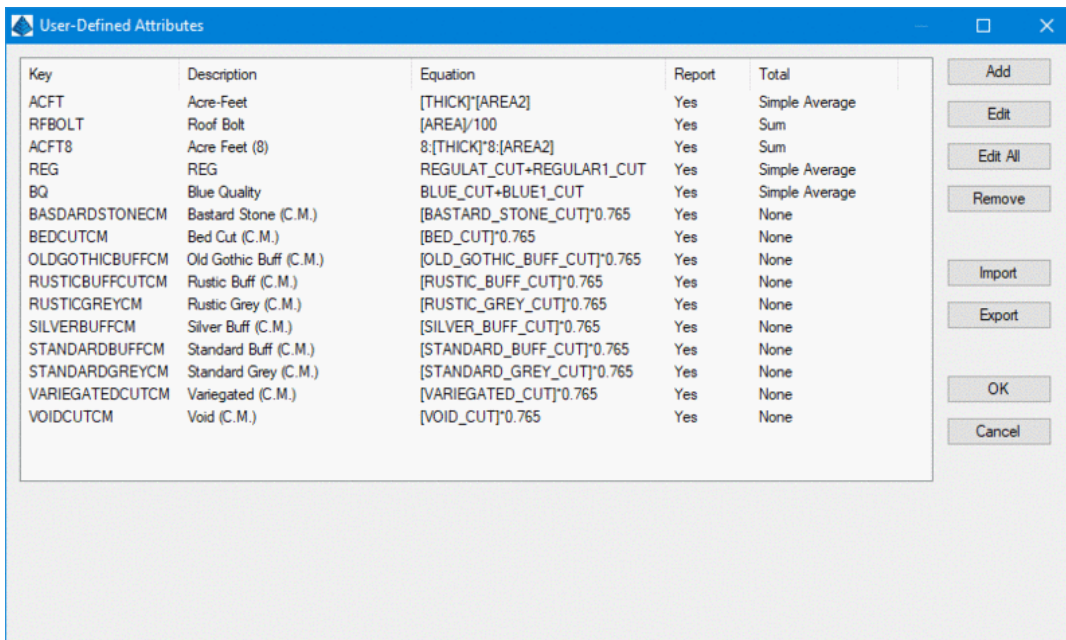
A few conversions are provided by default. We will need to add a conversion to cubic meters for each strata of interest. To begin, click Add.



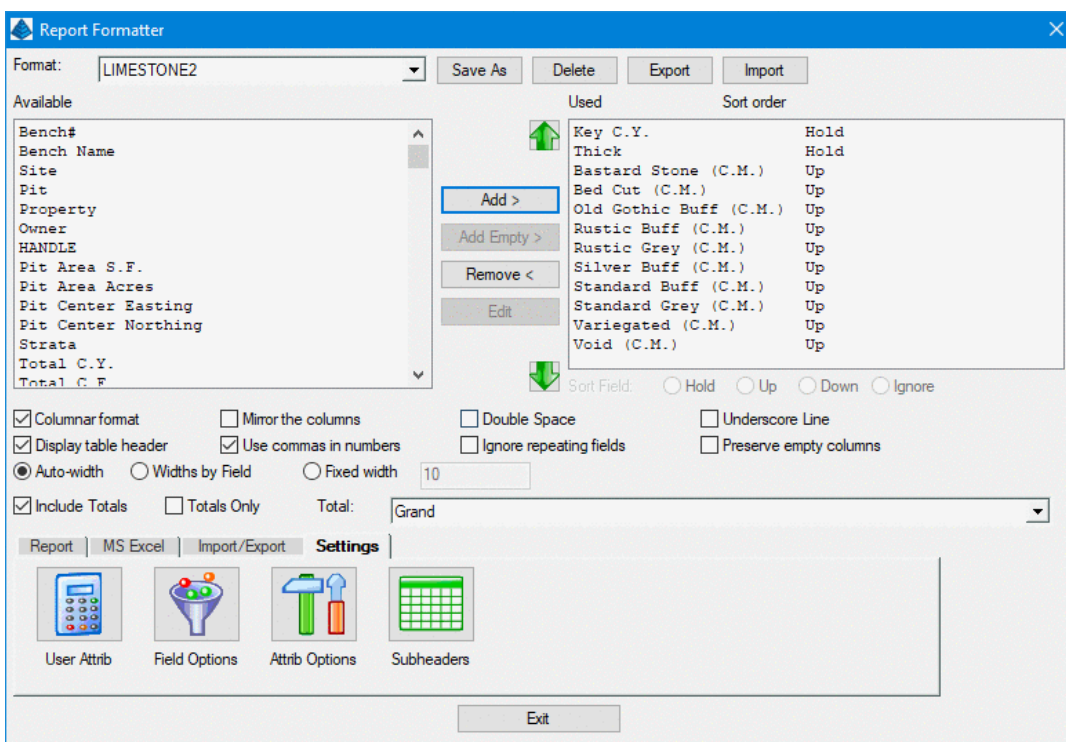
For the example of "BastardStone", you simply multiply the "known" variable for BastardStone by 0.765 to calculate cubic meters. Whatever is filled out for Description is what appears in the report. To see what the program uses as "known" variables, you click "List keys". The variable can be directly selected from this dialog, then the "*0.765" can be appended in the equation. Items are listed alphabetically:



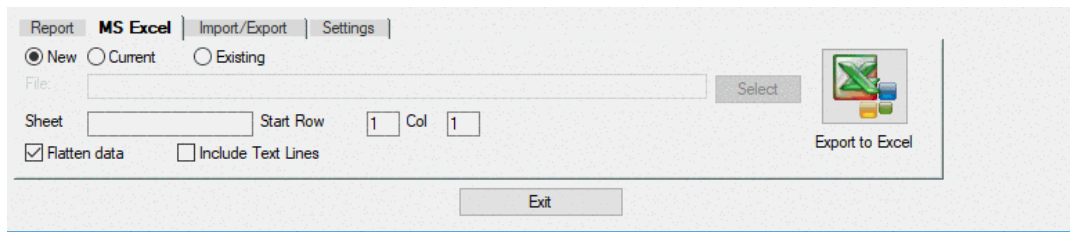
Repeat for each of the stone types involved. The pattern is, every time you click Add, click List Keys, choose the next stone of interest (e.g. BED_CUT), multiply by 0.765 in the equation, fill out the new "key word" for the cubic meter variable, fill out the full description for reporting and the decimal places, consider whether to Total by Sum, use No Total, etc., then click OK. Repeat for each strata. In order to see "Crawfoot" as a strata, you may need to select a larger number of vertical divisions, to "reveal" the very thin Crawfoot seam and get it in the list. Note that this is a one-time process. The new key variables will always be retained on future work. The final table might appear as shown below:



Next, click OK and return to the Report Formatter. Move the generic "Strata" and cubic yard items from the right column back to the left column and move all the desired cubic meter elements over to the right-hand column and save this new format as Limestone2.



Note that using the "User Attrib" button, you can make new elements for reporting, and these elements, like Standard Buff (C.F.), can be used in equations themselves, using their "key word" designation ("STANDARDBUFFCF"). Thus the Report Formatter can be used to produce all sorts of outputs and reports, all deriving from the known elements in the original report. To see results directly in Excel, click the "MS Excel" tab then click Export to Excel at the lower right of the dialog:



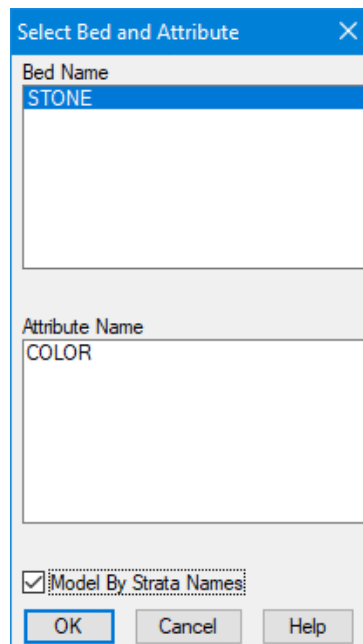
This leads to reporting directly in Excel:

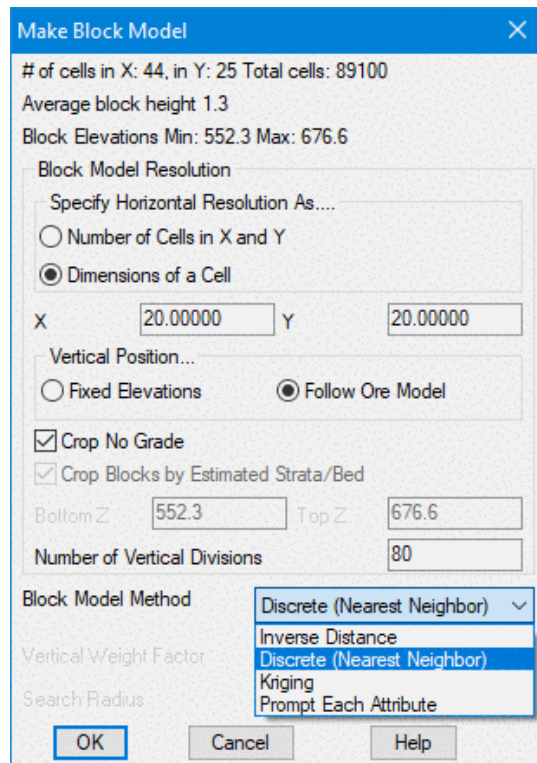
	A	B	C	D	E	F	G	H	I	J	K	L
1	Key C.Y. Thick	Bastard Stone (C.M.)	Bed Cut (C.M.)	Old Gothic Buff (C.M.)	Rustic Buff (C.M.)	Rustic Grey (C.M.)	Silver Buff (C.M.)	Standard Buff (C.M.)	Standard Grey (C.M.)	Variegated (C.M.)	Void (C.M.)	
2	58,907.4	64.34	6,696.2	467.7	3,726.1	6,902.4	1,405.3	5,088.2	15,690.8	3,145.5	402.2	876.3

Step 7-Viewing the Ore Body Model in 3D

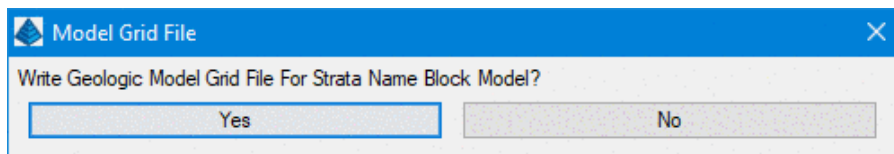
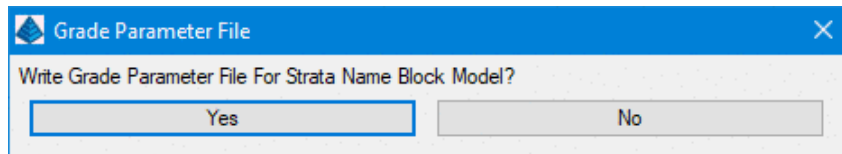
So far, we have produced volumes of material by "on-the-fly" calculations. We have not created a block model of our strata. The block model has only been created internally by the program based on the selection set of drillholes. An official "block model" is made by the command "Make Block Model" at the top of the Block Model pulldown. Select that. Here again, you can make the block model directly from the strata names.

Choose the lower left corner of the grid by the "Pick" method, and using the "end" snap (for endpoint) pick the lower left corner of the site, then the upper right corner of the site, as defined by the rectangular polyline enclosing all contours. Then set the grid resolution as shown below. Since we are dealing with "named" strata that abruptly transition to another form, use the "discrete" method of modeling. Note that for ore bodies, where qualities are typically modeled, "inverse distance" or "Kriging" are more typical modeling methods.



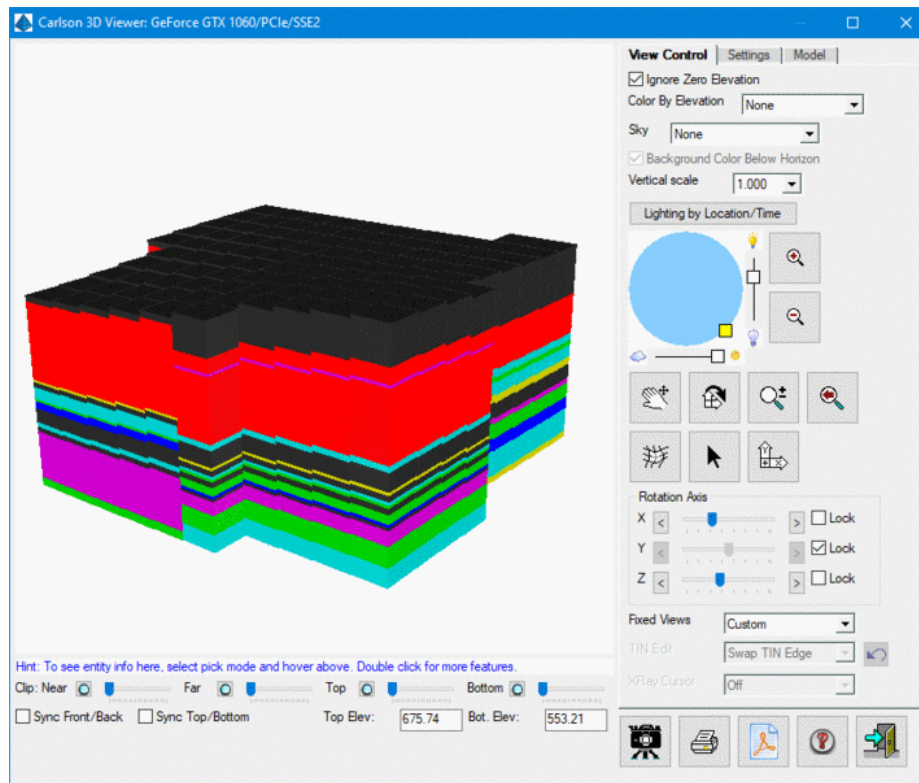


Note also in the dialog above that you can set the vertical position for modeling at fixed elevations or you can follow the ore model. The "follow" method finds the top and bottom of the ore material, then divides the vertical positions as defined, here by 80 divisions. So if the ore material narrowed, then 80 divisions would narrow to a smaller dimension. If you want fixed vertical dimensions, choose "Fixed Elevations". After you click OK, select all strata to process, and name the block model to be created. Also create, for later use, the grade parameter file and Geologic Model grid model. Click "Yes" to both the options below:



These options occur only when choosing the "Model by Strata Name" option above. Otherwise, you must use a pre-made Grade Parameter File and make the Geologic Model file by a separate process. The Grade Parameter File sets the colors (and even pricing) for the named strata or quality zones (in the case of attribute-defined grades). Making the Grade Parameter File this way does not set colors that match the colors set in Strata Definitions. You will need to run the command, "Define Grade Parameters" under the Ore pulldown menu and change the colors to match what were used in Strata Definitions, for consistent viewing, so that the 3D Views match the colors in Draw Geologic Column, for example.

Now run Block Model 3D Viewer under the Block Model pulldown, choose the "blk" Block Model File and the "gpf" Grade Parameter File, press "OK" at the Block Model Viewer dialog, then select a perimeter for viewing. You obtain a plot as shown below:



Note that you can exaggerate the vertical scale, or using the "Settings" tab, you can switch from rendered view (above) to "Leave as Points" view. Note that the yellow "Standard Buff" stone shows clearly in the 3D Block View, as does the red Void zone within the upper portion.

Step 8-Viewing the Ore Body in Profile View (Fence Diagram)

The process of making the Block Model was critical not only for 3D Viewing but also for the Profile View or "Fence Diagram". The 3D View used the "blk" file but the Fence Diagram command makes use of the "Geologic Model" or "pre" file as well as the "blk" Block Model File. Now that we have these, from Step 7 above, we can do the Fence Diagram. Before issuing the Fence Diagram command, draw a polyline through the drillholes or across the site that you will pick for the profile view of the strata (stone). Select Fence Diagram under the StrataCalc pulldown menu. Fill out the dialog as shown below:

Fence Diagram Settings

Process | Grid | Draw

Fence Type: 2D

Source: Geologic/Mining Model

Max Drillhole Offset: 5000.0

Ignore Zero Elevations

Process Multiple Fence Polylines

Vertical Space Between Diagrams: 10.0

Fence Label Text Size Scaler: 0.100

Sort Order: Up

OK Cancel Load Save Help

Fence Diagram Settings

Process | **Grid** | Draw

Horizontal Scale: 50 Vertical Scale: 50

X-Axis Grid Interval: 50 Y-Axis Grid Interval: 50

X-Axis Label Interval: 50 Y-Axis Label Interval: 50

Grid Style: Grid Lines

X-Axis Text Orientation: Horizontal

Axis Label Size Scaler: 0.10

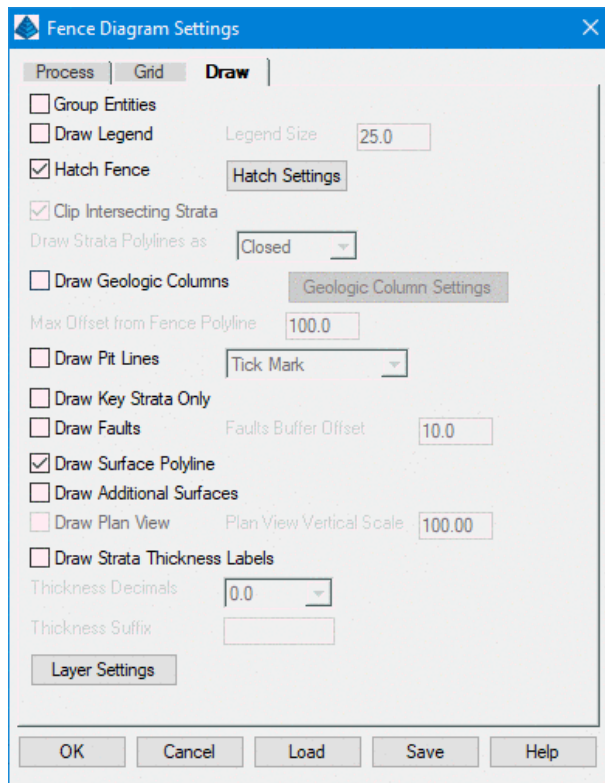
Starting Station: 0.00

Offset Station Text

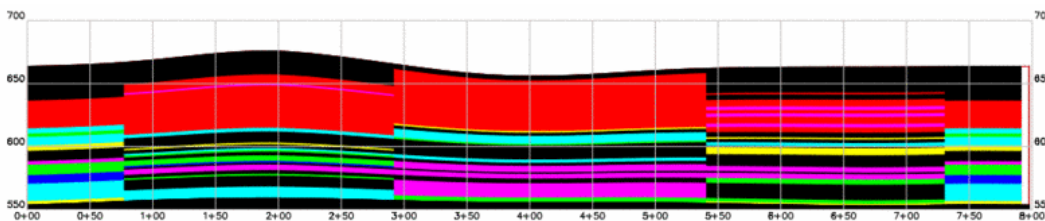
Station by Another Reference Centerline

Label Nothing/Easting

OK Cancel Load Save Help



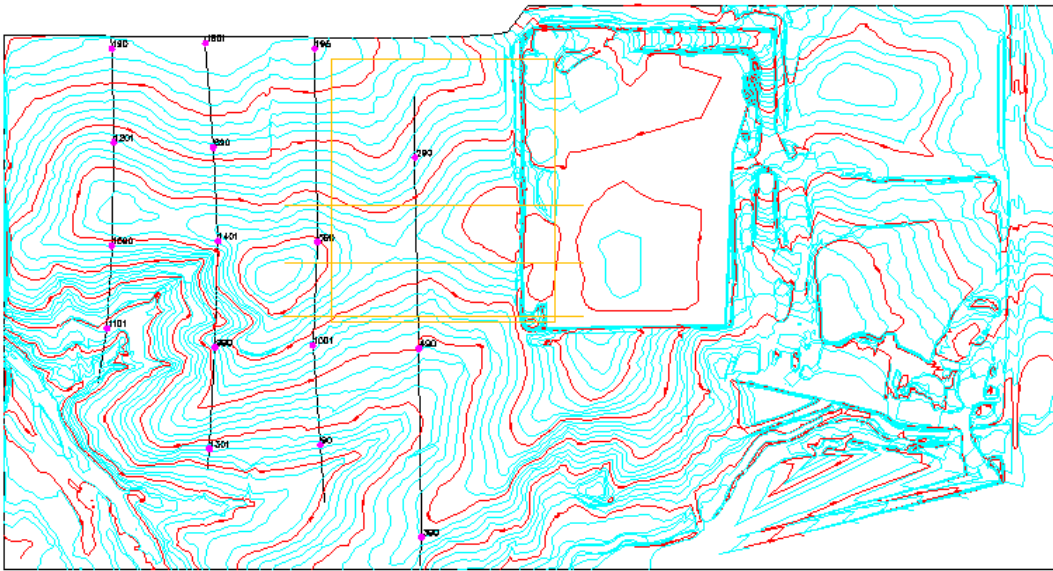
You could choose to exaggerate the vertical scale by changing the vertical entries to 20 rather than 50 (scale, grid interval and text interval). Then pick the polyline to use for the profile, and the "Geologic Model" and the "grade parameter file" as prompted. A typical result is shown below:



Case Study #5: Block Modeling by Quality Attributes

This tutorial takes a set of drillholes and goes through the steps that create the block model. The different grades are defined in the Grade Parameter File. Blocks are drawn and viewed in 3D for analysis. Cross sections are cut through the blocks and volumes by grade are calculated with Surface Mine Reserves. Finally, the Optimized Pit Design is found with the Lerch-Grossman algorithm.

The first step is to import the drillholes and name the beds based on how the seams are to be modeled. This example is a limestone bed with a thin layer of overburden, so there are just two main material types in the drilling, OB and LS are the bed names. The drillholes have already been imported for this example. That process is documented in other documents. There are 16 drillholes in this drawing. The drawing name for this tutorial is *Block Modeling.dwg*. Shown here are the plan view of the topography and the drillholes, and also the drillhole datasheet to display the drilling data.



Drillhole Data Sheet

File Edit Search Display Options

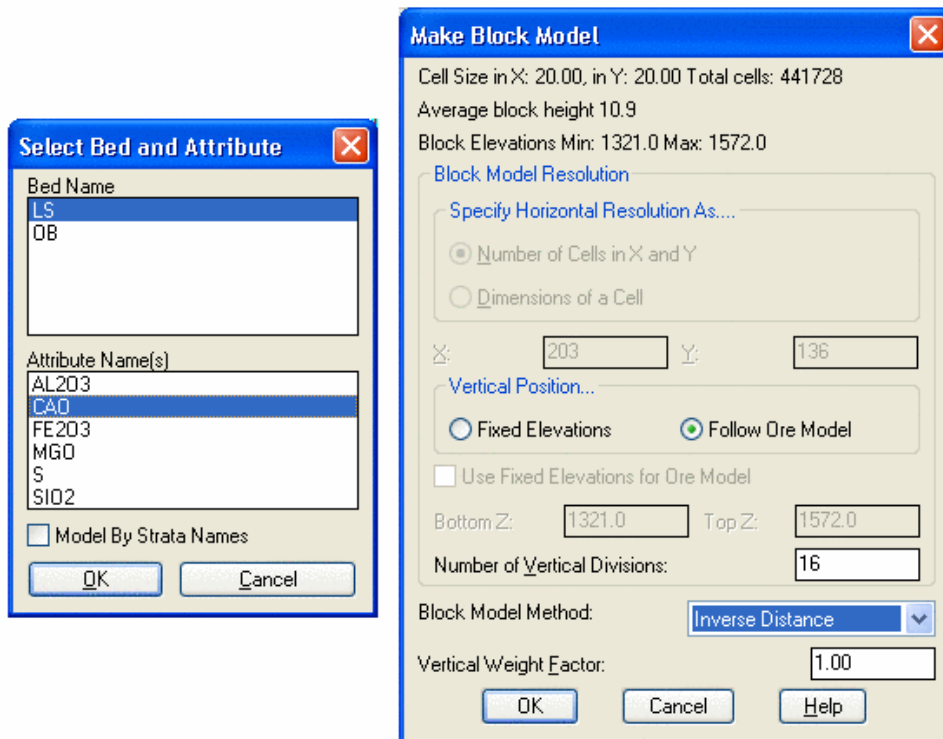
	Name	Description	Northing	Easting	Elevation	Process	Density	Drillhole type
1	1001		10494155.00	3089905.00	1568.000	<input checked="" type="checkbox"/>	155.00	Unknown
2	1090		10494685.00	3088835.00	1559.540	<input checked="" type="checkbox"/>	155.00	Unknown
3	1101		10494245.00	3088810.00	1539.000	<input checked="" type="checkbox"/>	155.00	Unknown
4	1201		10495235.00	3088845.00	1566.000	<input checked="" type="checkbox"/>	155.00	Unknown
5	1301		10493605.00	3089355.00	1548.000	<input checked="" type="checkbox"/>	155.00	Unknown
6	1401		10494710.00	3089400.00	1574.000	<input checked="" type="checkbox"/>	155.00	Unknown
7	1501		10495765.00	3089335.00	1542.000	<input checked="" type="checkbox"/>	155.00	Unknown
8	190		10495735.00	3088835.00	1543.470	<input checked="" type="checkbox"/>	155.00	Unknown
9	195		10495735.00	3089915.00	1544.480	<input checked="" type="checkbox"/>	155.00	Unknown
10	290		10495155.00	3090450.00	1562.260	<input checked="" type="checkbox"/>	155.00	Unknown

	Name	Depth	Elevation	Thick.,ft	Key	Bed Name
1	OVERBURDEN	0.300	1547.700	0.300	<input type="checkbox"/>	OB
2	LIMESTONE	9.000	1539.000	8.700	<input checked="" type="checkbox"/>	LS
3	LIMESTONE	10.000	1538.000	1.000	<input checked="" type="checkbox"/>	LS
4	LIMESTONE	20.000	1528.000	10.000	<input checked="" type="checkbox"/>	LS
5	LIMESTONE	25.000	1523.000	5.000	<input checked="" type="checkbox"/>	LS
6	LIMESTONE	30.000	1518.000	5.000	<input checked="" type="checkbox"/>	LS
7	LIMESTONE	40.000	1508.000	10.000	<input checked="" type="checkbox"/>	LS
8	LIMESTONE	50.000	1498.000	10.000	<input checked="" type="checkbox"/>	LS
9	LIMESTONE	60.000	1488.000	10.000	<input checked="" type="checkbox"/>	LS
10	LIMESTONE	70.000	1478.000	10.000	<input checked="" type="checkbox"/>	LS

	Name	Value
1	CAO	83.000
2	MGO	21.320
3	SiO2	0.136
4	AL2O3	0.059
5	FE2O3	0.061
6	S	0.032

READY

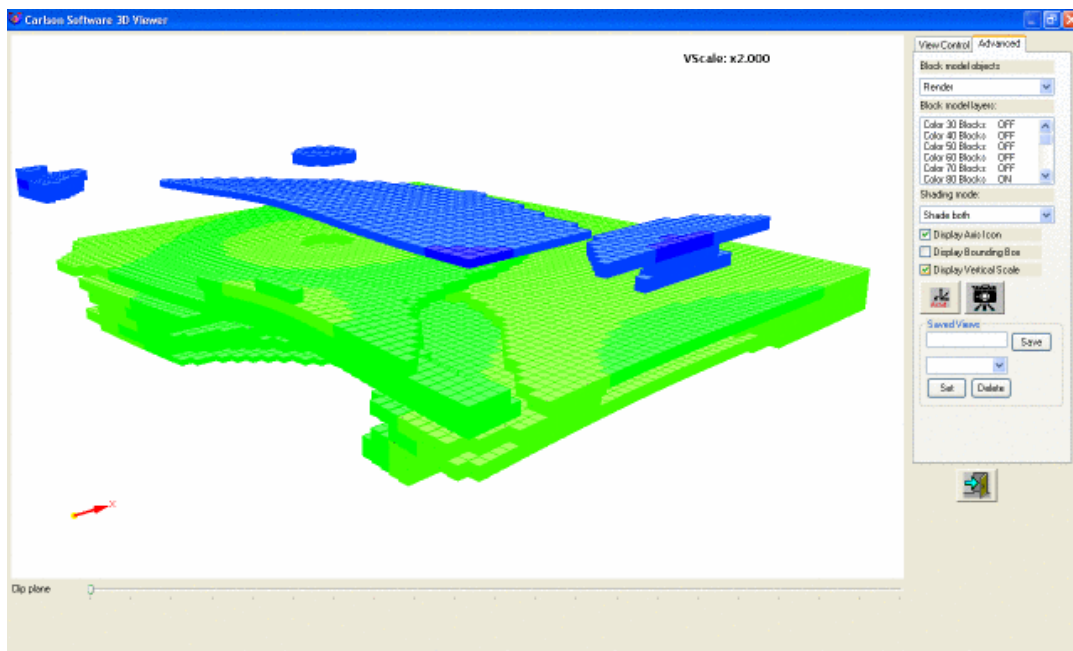
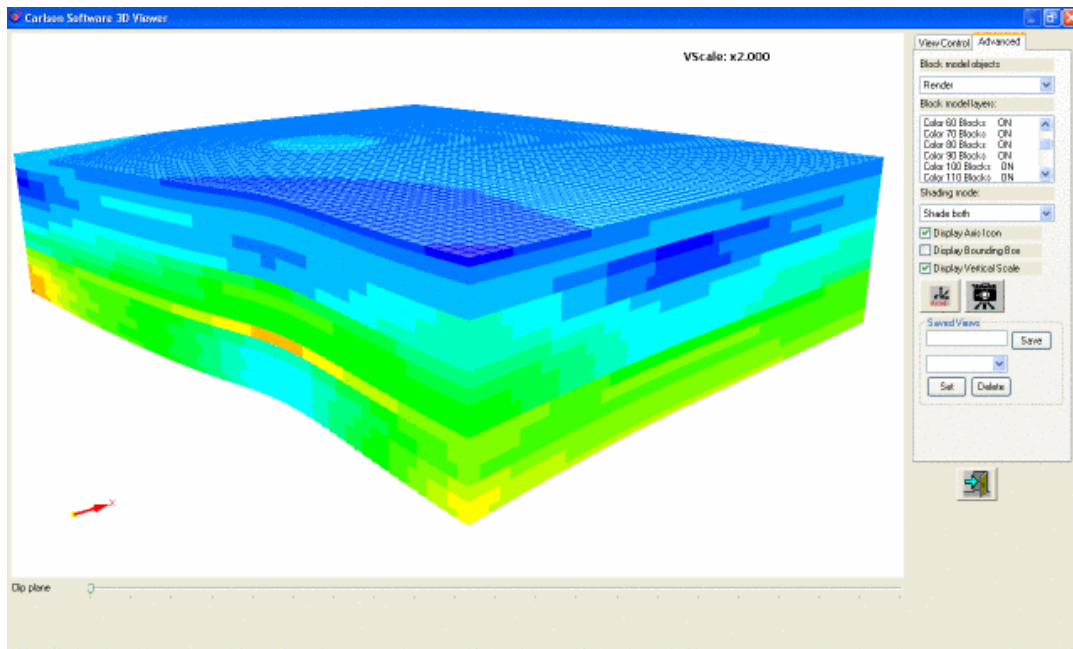
This command is used to create the block model from the drillholes. The first selection is to choose the Bed Name to model, and then the quality or qualities. Just one quality attribute may be used, or several at once. In this example, the LS bed and the CaO attribute will be modeled, as shown in this window. It needs a grid file to set the horizontal block sizes, so either pick the position from screen, and put in a dimension for X,Y, or copy an existing grid for positioning. For this one, the *Surface Topo.grd* can be copied for a position, which is 20x20 in dimension.



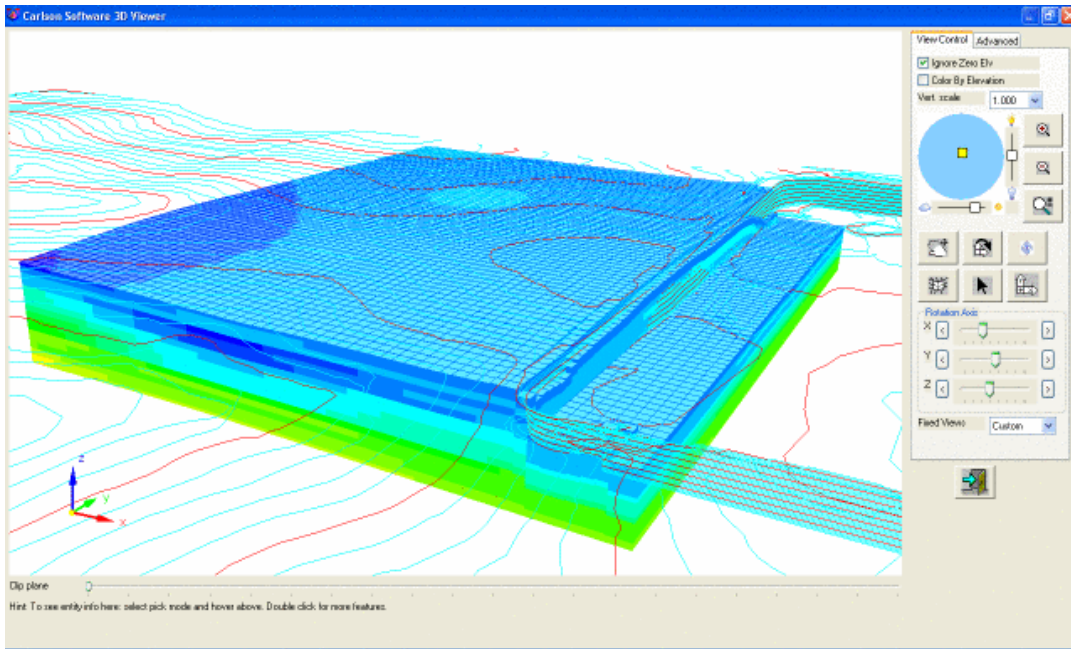
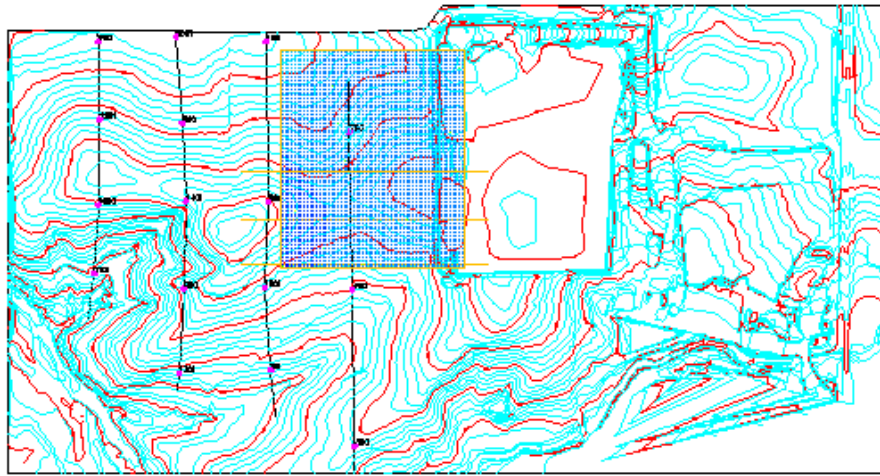
The second window sets the block height and modeling method. There are two distinct methods for setting the block height. They can be set to a fixed elevation and size which is independent of the beds, or can follow the top and bottom of the ore model from the drillholes. This creates almost a stratified block model where the elevations of the blocks follow the top and bottom of the ore elevations. This is like a hybrid of both strata and block modeling. If there are not any strata, such as in a gold or copper deposit, then the Fixed Elevations method is preferred. Both methods work the same. If using the Fixed Elevations method, to set the block height, the top and bottom of the model are entered, with the number of samples chosen to set the average block height, which is calculated and displayed at the top. In this screen, if the Number of Vertical Divisions is set to 16, the average block height listed above is 10.9. This should work well with the horizontal size of 20x20, giving an average block size of 20x20x10. This example will use Inverse Distance as the modeling method with a vertical factor of 1. Selecting OK builds all of the blocks and puts them in the BLK file. Choose a name for the BLK file, such as LS.CaO.BLK.

This command defines the grade ranges of the ore. This is what defines the blocks for colors and divisions for cross sections and volumes. There is a Draw Legend button to put it on the map. The price per pound is also defined here for the cost model, and that will be used for the optimized pit design. Also notice that there are 8 blanks for the various Parameters where the combination of the several attribute ranges can define the grade. For example the CaO > 90 and MgO < 15 defines the "High-Grade". For this example, just the CaO is defined for the different ranges. If another range is already defined, then the program will just use what is available. That is why just the ">" option is used below, starting at the highest grade and working down.

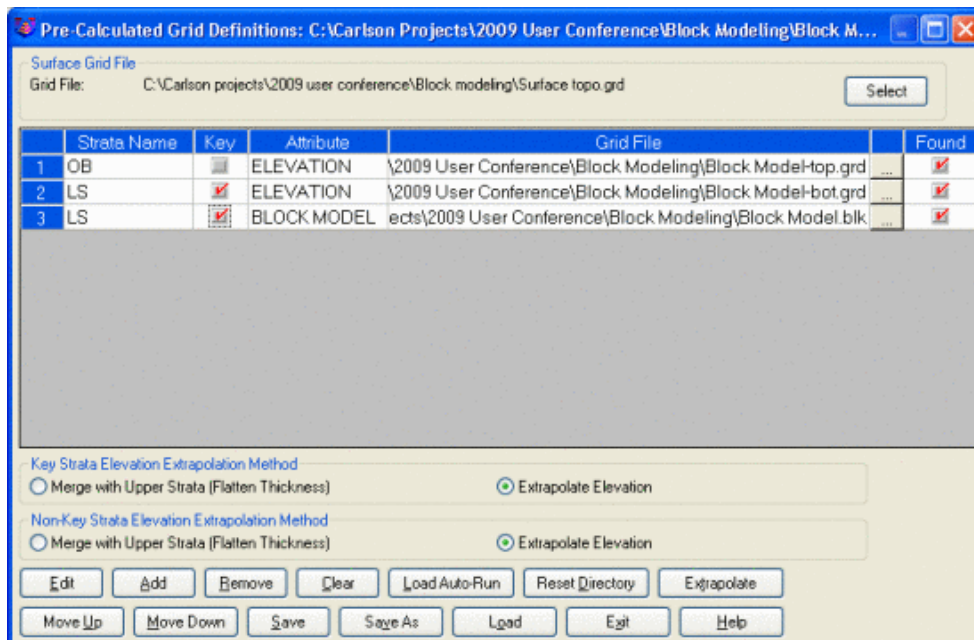
Now that the block model is built and the grade ranges are defined, the model can be inspected and viewed in 3D to check it for any problems. If the model is large, it is best to use an inclusion polyline to view just a subset of the entire model. In the Advanced Tab, there is a way to turn the various blocks on and off like layers. Just click on the line to turn on or off and the blocks are removed or added from the screen. This allows 3D views to see what the quality is inside the middle of the blocks. Notice in this example there are just green and blue blocks remaining.



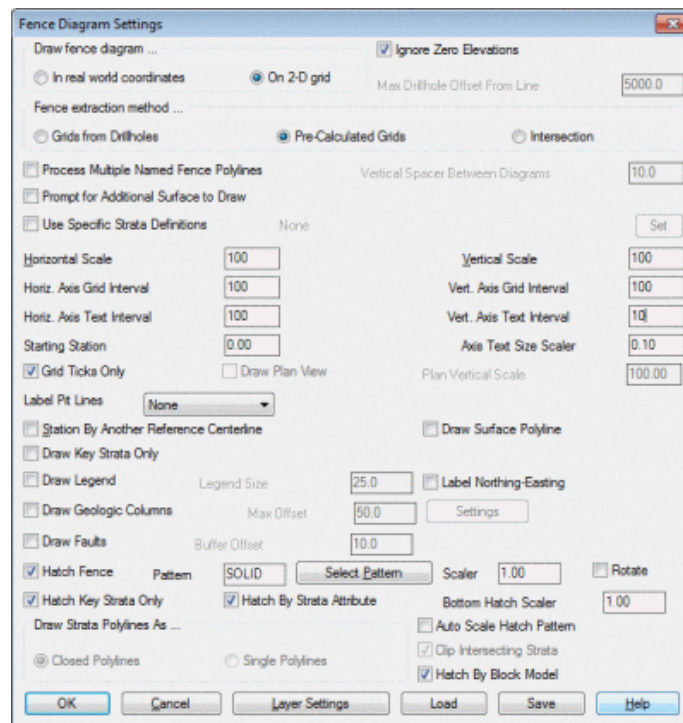
This is not a required step, but is convenient to place the blocks in the drawing permanently. This command will draw the blocks on screen in CAD as nodes or "dots". These nodes can then be brought into the 3D Viewer window and rendered the same as the Block Model 3D Viewer does. The nice option in this command allows to have a top and bottom limiting surface to crop the blocks. That way if just the blocks on a certain bench want to be viewed, use just the top and bottom grids of that bench, or even the topography, and an inclusion perimeter, to contain the blocks to draw, and ultimately view. After selecting the file, just leave all set to "YES" on the Draw Block Model screen. The nodes and contours drawn in CAD can be viewed with the 3D Viewer Window as seen below.

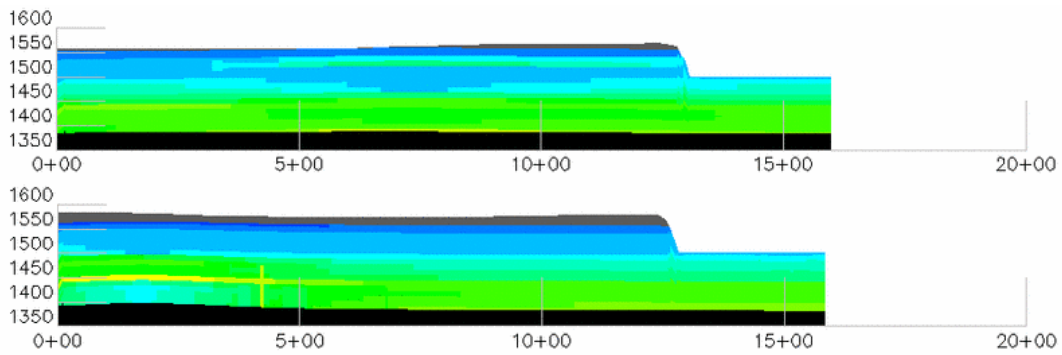


This step is necessary to combine the block model with the surface topo and any top or bottom elevation surfaces that will make up the entire model. The procedure for strata models is to just add the elevation grids as normal, and then add the BLK block model file to the appropriate interval. Flat elevation grids can be used for this, if it isn't a stratified model that has roofs and floors, like many hard rock metal mines and quarries that aren't stratified.

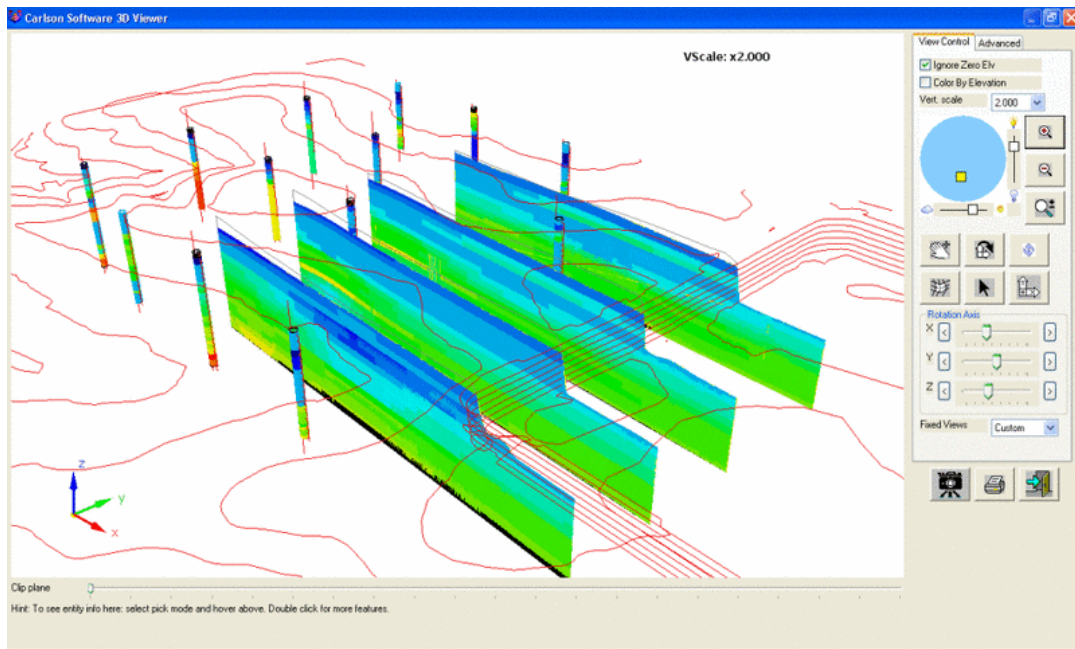


Now that the entire model is built and checked, a Fence Diagram can be drawn to see the geology and blocks in section view. Fence Diagram has an option to Hatch by Block Model. This can be drawn in two ways. The initial section shows it on a 2D Grid, the second one can be seen in 3D where it draws and hatches the fence in 3D below the line, in Real World Coordinates. Shown below are two fence diagrams from the drawing, on a 2D grid. Notice the coloration of the blocks based on grade of limestone.

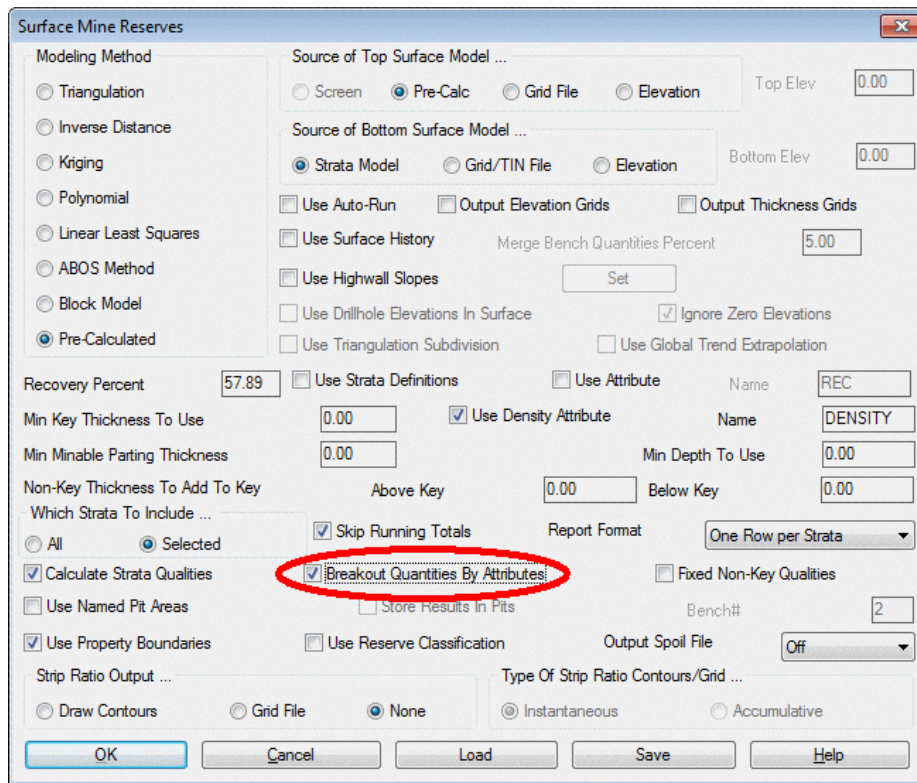




There is also an option to draw the Fence Diagrams in 3D using the Real World Coordinates setting. When this is viewed in the 3D Viewer Window along with 3D Geologic Columns, the result is very useful to visualize the geologic deposit as shown below. The Draw Geologic Column command will draw the drillholes as 3D columns, and they can also be colored by the Grade Parameter File. Notice how the coloration in the drillholes corresponds to the coloration in the fence cross-sections, indicating a good modeling estimation.



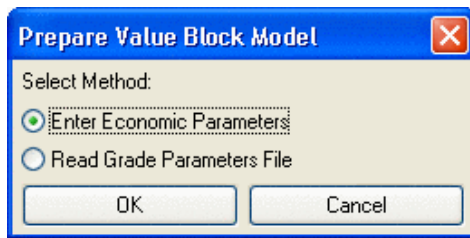
The next step is to get the volume and tons of the different grades of limestone with the Surface Mine Reserves. There is one check box to turn on that will report the tons by grade, it is Breakout Quantities by Attributes. This will not only give total tons of the limestone, but also the tons in the various grades. Here is how the window should appear.



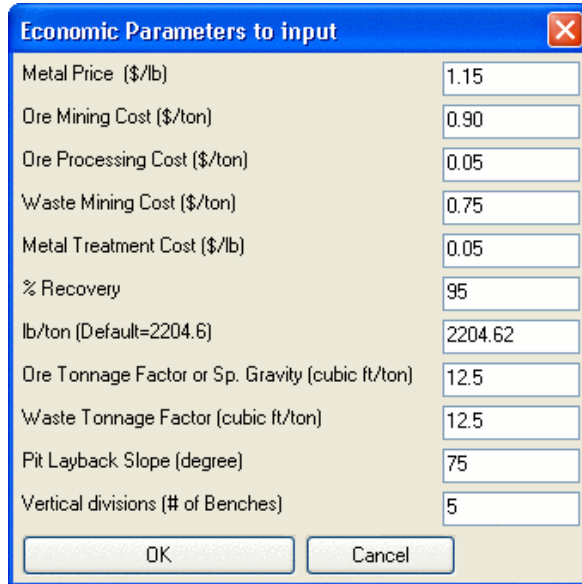
Shown here is the report of the data dumped into Excel using the Report Formatter. Notice how the total Key tons match the individual grade tons added up in the yellow cells. This is a good check to make sure all grades are accounted for in the report. Also confirm that each grade's CaO falls in line with the values defined in the Grade Parameter File.

Pit	Strata	KEY Tons	Waste C.Y.	Thickness	CAO	GRADE3 Tons	GRADE4 Tons	GRADE5 Tons	GRADE6 Tons	GRADE7 Tons	GRADE8 Tons	GRADE9 Tons	GRADE10 Tons
Pit 1 OB			590211.6	11.3									
Pit 1 LS		21197945.2		164.1	46.045	146.2	12406.6	59750.0	253154.6	1060902.4	3945035.3	2172383.6	107092
Pit 1 varies		21197945.2	590211.6		46.045	146.2	12406.6	59750.0	253154.6	1060902.4	3945035.3	2172383.6	107092
		21197945.2	Total from individual grades										
						GRADE3 CAO	GRADE4 CAO	GRADE5 CAO	GRADE6 CAO	GRADE7 CAO	GRADE8 CAO	GRADE9 CAO	GRADE10 CAO
						85.69	81.47	76.86	71.94	67.07	62.41	57.79	52
						85.69	81.47	76.86	71.94	67.07	62.41	57.79	52

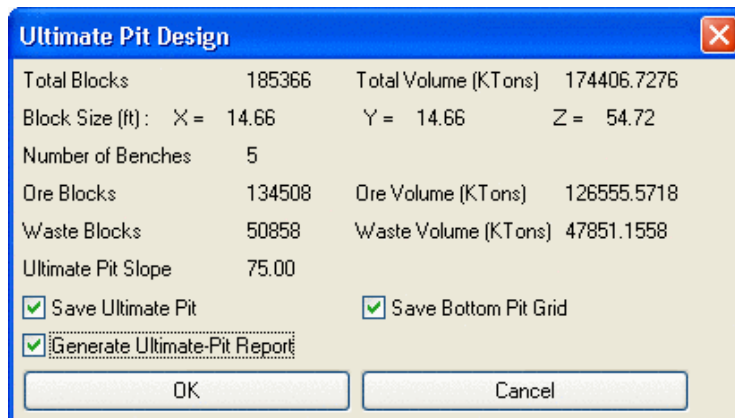
Now to find the optimized final pit of profitable mining, we will run this command to create a value block model, where each block is assigned a cost associated with it. Once this value block model is created, then the Optimized Pit Design can be run. First, select the geologic block model to analyze. This is the file used in the steps above. Then choose either to use the Grade Parameter file, or to Enter the parameters on screen here. For this run, the Economic Parameters will be entered. Select the Surface Topography grid when prompted to do so. It will use this to calculate the overburden on top of the blocks.



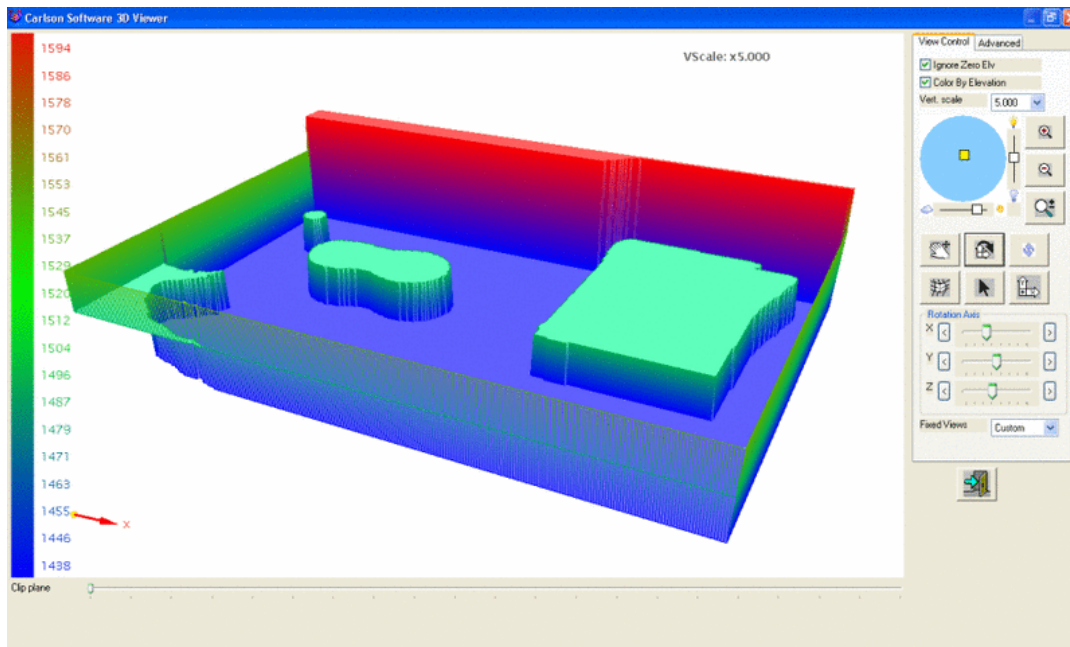
Enter in the Economic Parameters. Shown here is a sample of the costs associated with the various mining stages. This writes the value block model, where each block now has a value assigned to it whether it is profitable or not. This file is named Value Block Model.BLK.



Now that the Value Block Model is written, the Optimized Pit Design is run to create the ultimate pit and create the ultimate pit block model. The Value Block Model is now the file to process, and this one is selected first. All options are turned on to create an ultimate pit grid, block model and a report. The block model is just for calculation purposes and contains cost values.



The final report shows that most of the blocks are mineable. Level 1 doesn't have anything in it that is mineable. The lowest level, 5 is not mineable, though there are blocks in it. The blocks that are not profitable are what is left in the image below. The grid is displayed here, with the Surface 3D Viewer, and colored by elevation. It is easy to make changes in the input parameters and run it again. The cost to mine or process the ore can be modified and the new cost model created to see how it affects the output.



Carlson Software Edit : C:\Documents and Settings\gwenker\Application Data\Carlson Software\Carlson2009R17...

File Edit Settings

Open Save Print Exit Find Screen Hide

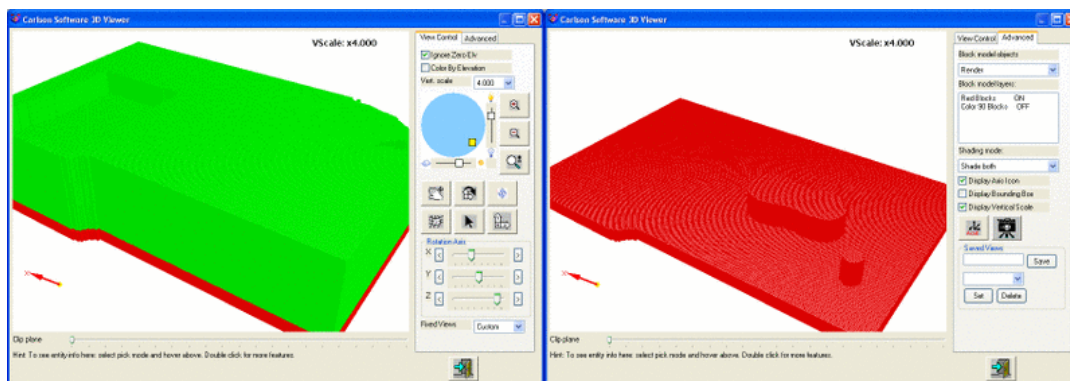
Ultimate Pit Report

=====
 Total Number of Blocks : 185366
 Total Tons in the block model (KTons) : 174406.7276
 Total Number of Ore Blocks : 134508
 Total Number of Waste Blocks : 50858
 Number of Blocks in optimum pit : 132866
 Total mineable reserve (KTons) : 125010.6507

NOTE: All Volume in KTons
 All costs & profit are in (1000*\$)

Level	Total Blocks	Ore Blocks	Waste Blocks	Mineable Blocks	Profit
1	0	0	0	0	0.0000
2	39035	39035	0	39035	29342813.5889
3	48777	48777	0	48270	46463974.9888
4	48777	46696	2081	45561	59158782.2902
5	48777	0	48777	0	0.0000

There is an automatic Grade Parameter File written called the Profit and Loss.GPF. This will colorize the blocks in the Value Block Model green if they are profitable, and red if they are not. This final model can also be viewed in 3D and it will resemble the ultimate pit grid file. Shown below is the full model, and then the profitable blocks are removed or "frozen" and just the red, non-profitable blocks remain.

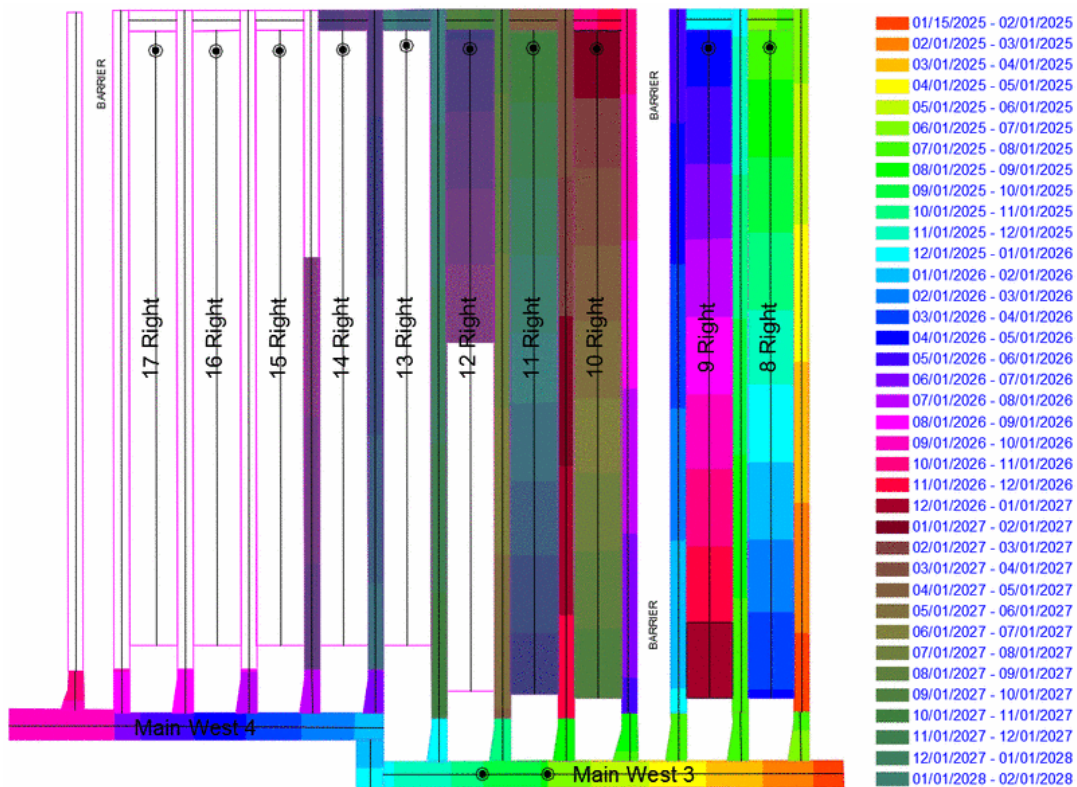




Underground Mining Module

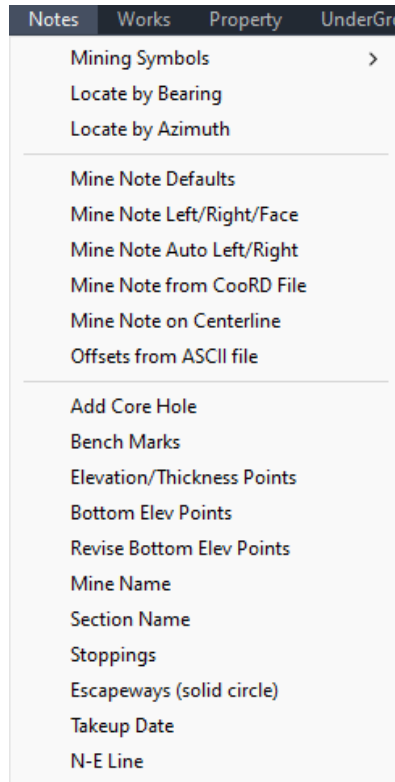
16

The Underground Mining Module consists of various commands for planning and scheduling an underground mining operation. Major features include panel layout, mine mapping, solid modeling/analysis, and mine scheduling. Commands are accessible from the Notes, Works, Property, Underground, Solid, and Subsidence pulldown menus.



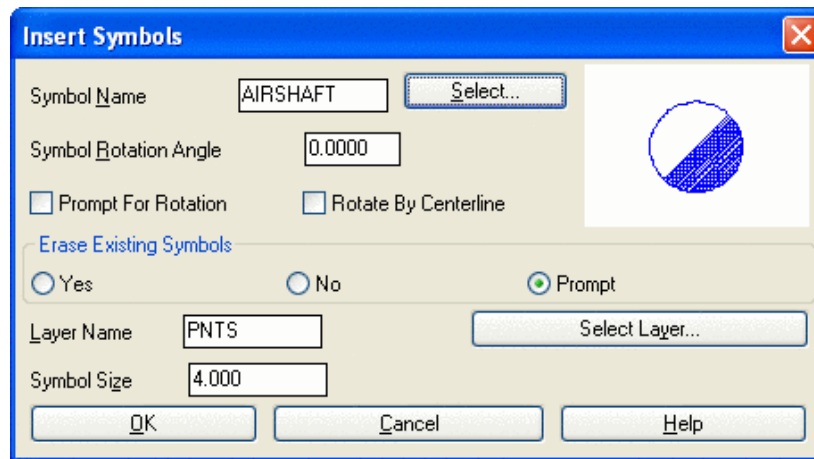
Notes Menu

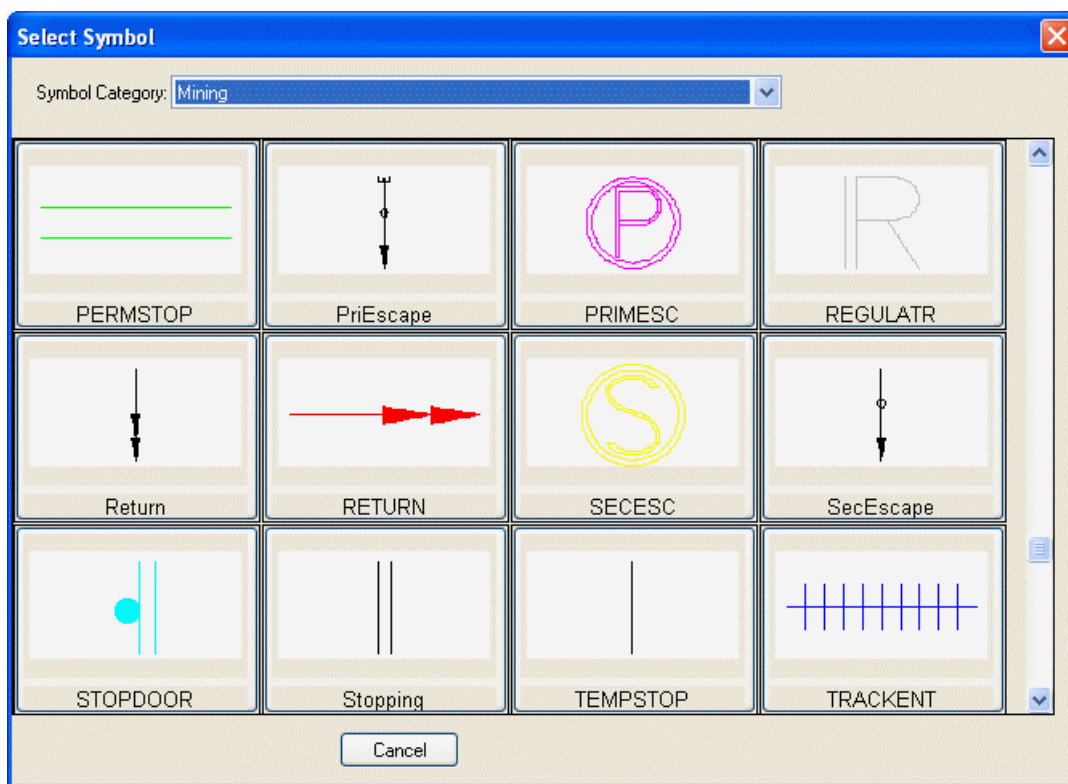
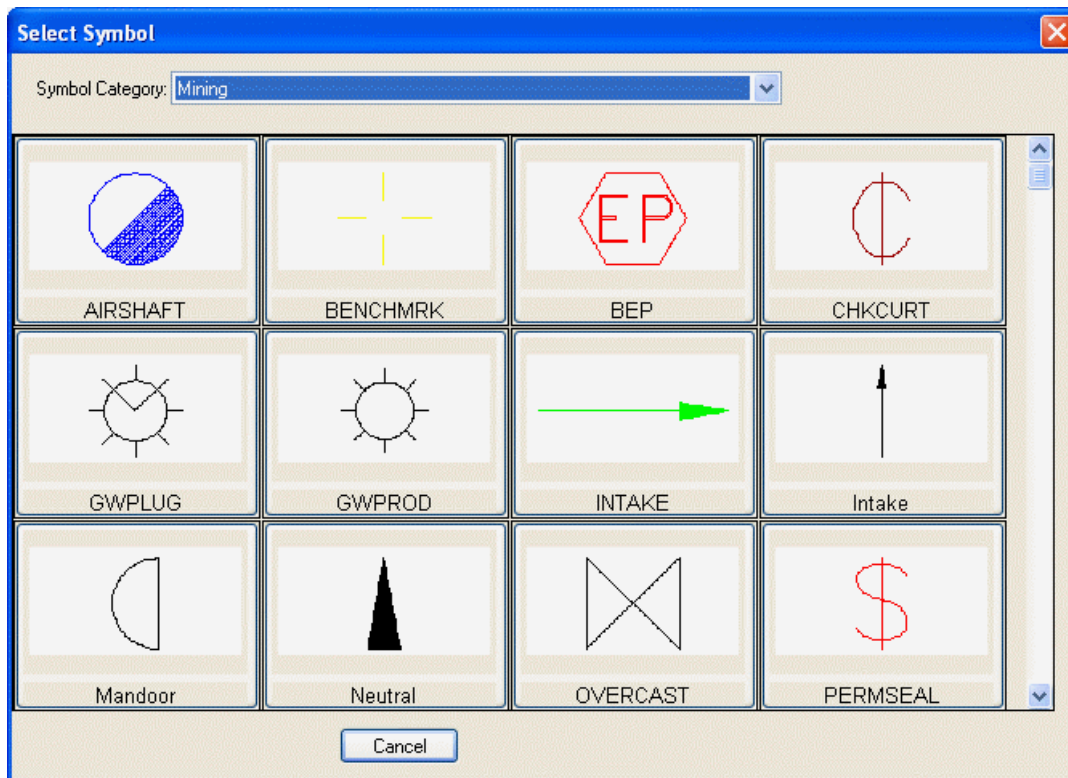
The commands in the top section of the Notes pulldown menu are for underground mine note entry and surveying. The bottom section contains commands for inserted mine drawing embellishments.



Insert Mining Symbols

This routine places icon mining symbols on the drawing. There are 25 symbols to choose from. After selecting a symbol from the pages in the Select option, simply pick a point to place it. A row of that symbol can be placed by picking a destination point, and an interval point. In this case, multiple symbols are placed the specified interval apart along a line from the start to the destination point. There are options for rotating the symbols, to erase existing symbols, layer name and symbol size.





Custom made symbols can be made with the command Edit Mining Symbols Library.

Prompts

Mine Mapping Symbols Dialog

Choose a symbol.

Pick Point for Initial Placement: *pick a point*

Pick Destination Point, or <Enter> For Single Placement: *press Enter*

Rotation angle <E>: *press Enter*

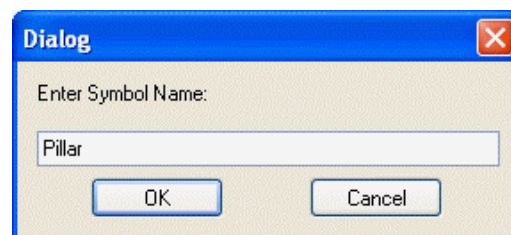
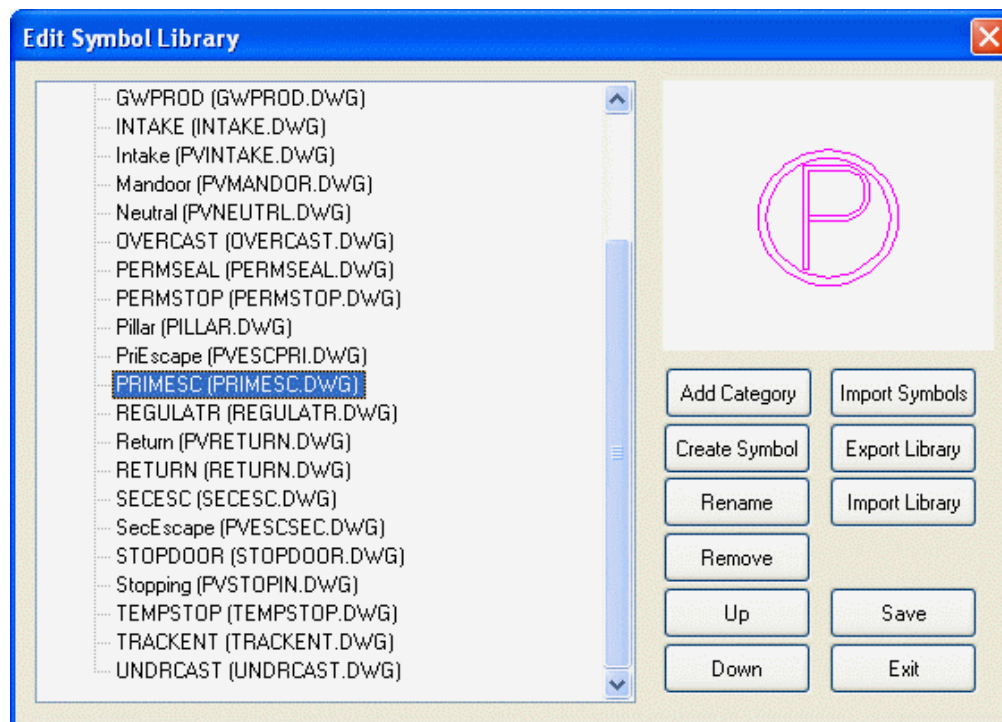
Pulldown Menu Location: Notes

Keyboard Command: ptsym2

Prerequisite: None

Edit Mining Symbols Library

This routine edits the mining symbols library. New symbols can be added and existing symbols can be edited. This is very similar to editing the points libraries in the Survey module. To add a new symbol, it must be drawn as entities in a dwg. Choose Create Symbol and give it a name. Choose a DWG to save it as, select the item, and pick the insertion point (usually the center). The new symbol should now be on the list in the library.



Prompts

Select objects: *select the symbol*

Pick Insertion Point For This Symbol: Symbol saved.

Pulldown Menu Location: Notes
Keyboard Command: editminesym
Prerequisite: None

Locate by Bearing

This command locates points by bearing and distance. Additionally, the AutoCAD text screen provides the horizontal distance and coordinates.

Prompts

[Enter] to use preview point or Select occupied point ?

Pick point/<point Number>: 24

PointNo.	Northing (Y)	Easting (X)	Elev (Z)	Description
24	4922.37	4544.81	0.00	

Enter Bearing (Qdd.mmss) <277.1259>: 435.2317

Enter or pick Distance <104.39>: 200

Enter Vertical Angle (dd.mmss) <0.0000>: Enter

Pulldown Menu Location: COGO > Locate by Bearing-Ang

Keyboard Command: locbrg2, lg

Prerequisite: None

Locate by Azimuth

This command locates points by azimuth and distance. The AutoCAD text screen provides the horizontal distance and coordinates.

Prompts

[Enter] to use preview point/ or Select occupied point ?

Pick point/<point Number>: *_endp of (pick a point)*

Enter Azimuth (ddd.mmss) <22.5632>: 277.1259

Enter or pick Distance <40.32>: 104.39

Enter Vertical Angle (dd.mmss) <0.0000>: Enter

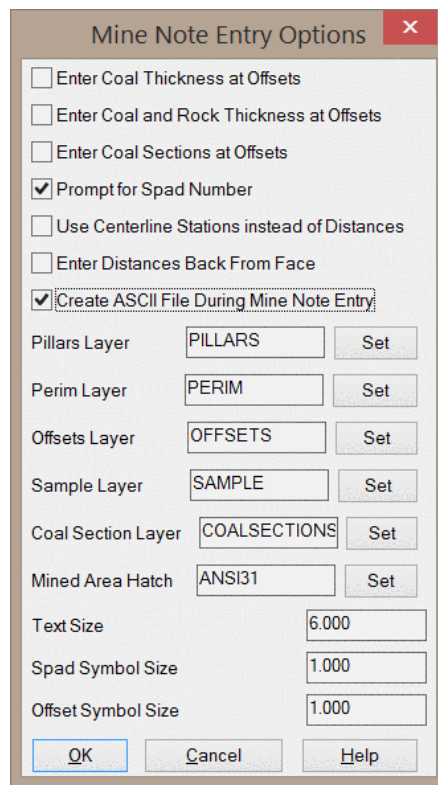
Pulldown Menu Location: COGO > Locate by Bearing-Ang

Keyboard Command: locazi2, az

Prerequisite: None

Mine Note Defaults

These options will toggle various prompts when entering Mine Notes. You can also set the layers for various linework needed for geologic modeling and mine planning.



Enter Coal Thickness at Offsets: This option activates coal section prompts in the mine note entry routines which creates sections of coal thickness at the offsets.

Enter Coal and Rock Thickness at Offsets: This option prompts for both coal and rock thickness.

Enter Coal Sections at Offsets: This option creates a complete coal section at the offsets.

Use Centerline Stations instead of Distances: This option will prompt for a starting station for the entries. Otherwise the starting station is automatically set to zero.

Prompt for Spad Number: This option determines whether the program prompts for spad numbers for the entry points. Turning this option off saves a prompt if you are not using spad numbers.

Enter Distances Back From Face: This option allows all of the mine note entry routines to post back from the face. Once this option is set, you never need to set it again except to change it. When this option is on, you will be asked for the "Distance to Face" when starting a note entry routine. Also, each routine will display a message stating "Mine Note From Face Option Active".

Create ASCII File During Mine Note Entry: This option determines whether or not a record of the offset notes is kept in an ASCII file whose default name is OFFSET.DAT. This file is created by mine note entry routines and can be used by AutoMine Connections and Offsets by ASCII File. If this file already exists when a mine note entry routine is started, there is an option to either append the new notes to the existing file or replace the existing file. All these data files are stored in the Carlson data directory such as C:\SC14\DATA\. Since these files are in ASCII format, they can be easily edited with any text editor. Once ASCII File From Notes is set, you never need to set it again except to change it.

Pillars Layer: This option sets the layer for pillar linework. If pillars are not drawn on this layer, they will not be recognized in many commands such as Place Panel or Recalculate Extraction.

Perim Layer: This option sets the layer for panel perimeters.

Offsets Layer: This option sets the layer for mine notes at offsets.

Sample Layer: This option sets the layer for the location of coal sections. This does not set the layer for text of the coal section.

Coal Section Layer: This option sets the layer for coal sections, including the text and the optional circle around the text. This does not set the layer for the actual location of the coal section.

Mined Area Hatch: This option sets the hatch pattern for mined out areas. This hatch pattern will be drawn with the Quantities by Avg/Grid/Centerlines commands.

Text Size: This option sets the default text size for coal section and mine note labels.

Spad/Offset Symbol Size: This option sets the default size for spad and offset symbols.

Pulldown Menu Location: Notes

Keyboard Command: mconfig

Prerequisite: None

Mine Note Left/Right/Face

Mine Note Left/Right/Face is a simple routine for entering offsets. The flow of this routine is very similar to Mine Note from CoorD File. To get started, the routine only requires a starting point and a direction. Then enter a distance up followed by the offsets. The offsets consist of a code immediately before the distance. The codes are L for left, R for right, CL or *L for corner left, and CR or *R for corner right. For example, the offset CL9 means a corner point offset 9 feet to the left.

Corner points identify a corner of a pillar that is across a cross cut. Corner points are drawn as green dots and other points are drawn as red circles. This distinction is made to assist in pillar connection and is important to AutoMine Connections.

If the ASCII File From Notes option is on, this routine also produces a .DAT file which contains a record of the entries. This file is necessary for AutoMine Connections.

Prompts

Enter Offset File Name <offset.dat>: *press Enter*

Enter/Pick From Station Point: *pick or Enter a point*

Entry Number: *1* This is an identification number for the entries that follow. AutoMine Connections groups spad offsets by this number.

Enter/Pick To Station Point (a for azimuth): *pick or Enter a point or enter A*

Spad Number or <Enter> for none: *1043* This number is drawn at the spad.

Flip Spad Text (Y/<N>)? *N* This determines the orientation of the spad text.

Enter distance From Station/Spad on Centerline (Enter to end): *25*

Offset, U for Undo, or [Enter]for new centerline distance: *L10* Offsets left 10.

Offset, U for Undo, or [Enter]for new centerline distance: *CR9* Offsets a corner point right 9.

Offset, U for Undo, or [Enter]for new centerline distance: *press Enter*

Enter distance From Station/Spad on Centerline (Enter to end): *47*

Offset, U for Undo, or [Enter]for new centerline distance: *L8 Offsets a corner point left 8.

Offset, U for Undo, or [Enter]for new centerline distance: R11 Offsets right 11.

Offset, U for Undo, or [Enter]for new centerline distance: *press Enter*

Enter distance From Station/Spad on Centerline (Enter to end): *press Enter*

Pulldown Menu Location: Notes

Keyboard Command: note1

Prerequisite: None

Mine Note Auto Left/Right

Mine Note Auto Left/Right is a simple routine for entering offsets. The routine only requires a starting and ending point, and then simply distance up, distance left, and distance right over and over until the end of that entry. Corner points may be specified for left or right offsets by preceding the number with a C or *. Corner points identify a corner of a pillar across a cross cut. Corner points are drawn as green dots, while other points are red circles. This distinction is made to assist in pillar connections and is important for AutoMine Connections.

If the ASCII File From Notes option is on, this routine also produces a .DAT file in the data directory which contains a record of the entries. This file is necessary for AutoMine Connections.

Prompts

Enter Offset File Name <offset.dat>: *press Enter*

Enter/Pick From Station Point: *pick or enter a point*

Entry Number: 1 This is an identification number for the entries that follow. AutoMine Connections groups spad offsets by this number.

Enter/Pick To Station Point (a for azimuth): *pick or enter a point or enter A*

Spad Number or <Enter> for none: 1043 This number is drawn at the spad.

Flip Spad Text (Y/<N>)? N This determines the orientation of the spad text.

Enter Distance From Station on Centerline (U to Undo, Enter to end): 25

Enter Left Offset Distance: C10 The C specifies a left corner point.

Enter Right Offset Distance: 8

Pulldown Menu Location: Notes

Keyboard Command: note2

Prerequisite: None

Mine Note From CoORD File

This mine note entry routine is for when the spads are known coordinates or point numbers from a .CRD file. The flow of entering offsets for this routine is very similar to Mine Note Left/Right/Face. First enter the spads point number or coordinates, and specify the azimuth by either entering the number directly or by identifying another point. Then enter a distance up followed by the offsets. Pressing Enter at the offset prompt will back up the routine so that it asks for another distance up. Then, pressing Enter at the distance up prompt will make it ask for the next spad. Finally, pressing Enter to the spad question will exit the routine.

There are two types of offsets points, corner points and regular points. Corner points identify a corner of a pillar that is across a cross cut. Corner points are drawn as green dots and regular points are drawn as red circles. This distinction is made to assist in pillar connection and is important to AutoMine Connections. The offsets have a code immediately before the distance. The codes are L for left, R for right, CL or *L for corner left, and CR or *R for corner right. For example, the offset *R10 means a corner point offset 10 feet to the right.

Prompts

CoORD File Selection Dialog Choose the file that contains the spad point data.

Starting Spad or north & east coord: *1*
Azimuth (DDD.MMSS) or P for Pt.-Defined Direction <0.0>: *P*
Directional Spad or north & east coord: *3*
Distance or 'enter' for new Spad No.: *15*
Offset or 'enter' for new Distance: *L10*
Offset or 'enter' for new Distance: *CR10*
Offset or 'enter' for new Distance: *press Enter*
Distance or 'enter' for new Spad No.: *17*
Offset or 'enter' for new Distance: *CL10*
Offset or 'enter' for new Distance: *press Enter*
Distance or 'enter' for new Distance: *press Enter*
Next Spad No. or north & east coords or 'enter' to exit: *press Enter*
Print file containing offset input data (<y>/n): *n*

Pulldown Menu Location: Notes
Keyboard Command: note3
Prerequisite: None

Mine Note on Centerline

This command is similar to the other Mine Note routines like Mine Note Auto Left/Right. The difference with this routine is that the program does not prompt for offsets left and right. Instead the program only prompts for the distance along the centerline and places the offset points on the centerline (0 offset).

Prompts

Enter Offset File Name <offset.dat>: *press Enter*
Append File (<Yes>/No)? *press Enter*
Enter/Pick From Station/Spad point[node on]: *pick a point*
Entry Number: *1*
Enter/Pick To Station point (a for azimuth): *A*
Azimuth of Heading (DD.MMSS) or p to pick <0.0>: *11*
Enter Spad Number or <ENTER> For None: *101*
Flip Spad Text (Yes/<No>)? *press Enter*
Enter distance From Station/Spad on centerline (Enter to end): *40*
Enter distance From Station/Spad on centerline (Enter to end): *60*
Enter distance From Station/Spad on centerline (Enter to end): *press Enter*
Another Spad (<Yes>/No)? *N*
Print file containing offset input data (<Yes>/No)? *N*

Pulldown Menu Location: Notes
Keyboard Command: note4
Prerequisite: None

Offsets from ASCII File

This command can be used to update a drawing if the offset file is edited outside a Mine Note Entry command or a new drawing is started. Otherwise, the offset dots are drawn only once, while entering them with a Mine Note Entry command.

Offsets from ASCII File draws points that are defined in the specified offset file. The offset file is created by Mine Note Entry commands. In order to draw the points, a SPT99.DWG and a OFFSET.DWG must be in the Carlson Support directory that was set up when the software was installed.

Prompts

Enter the offset file <offset.dat>: Enter a filename.

The default directory for this file is the data directory your Carlson Data directory that was created when the software was installed.

Pulldown Menu Location: Notes

Keyboard Command: moffsets

Prerequisite: An offset file created by one of the Mine Note routines

Elevation-Thickness Points

This command is very similar to the Bottom Elev Points command (BOTELEV), but in addition to drawing symbols and text labels for the bottom elevation of the strata, the thickness and top elevation of the strata will also be labeled. When the command is executed, the below dialog will appear.

Calculate	Value	Label Prefix	Label Suffix	Text Layer	Symbol Layer
<input checked="" type="radio"/> Top	100.0	Top	'	MinePoint - Top	MinePoint - Top X
<input type="radio"/> Thickness	10.0	Thk	'	MinePoint - Thk	MinePoint - Thk X
<input type="radio"/> Bottom	90.0	Bot	'	MinePoint - Bot	MinePoint - Bot X

Text Size: 10.0 Rotation Angle (deg): 45 Pick Decimals: 0.00

OK Cancel

Calculate: This option controls which values will be entered and which value will be calculated. If this option is set to "Top", then the value for the Top Elevation will be automatically calculated based on the values for the Thickness and the Bottom Elevation.

Value: These are the values that will be labeled.

Label Prefix/Suffix: These fields control the label prefixes and suffixes, as shown in the below image.

Text/Symbol Layer: These fields set the layers for the symbols and the labels.

Text Size: This value sets the text size for the labels. All three labels will be the same size.

Rotation Angle (deg): This value sets the rotation angle (in degrees) of the labels and the symbols. Clicking the **Pick** button will prompt you to pick two points on the screen to set the rotation angle.

Decimals: This dropdown list controls the number of decimals to label.

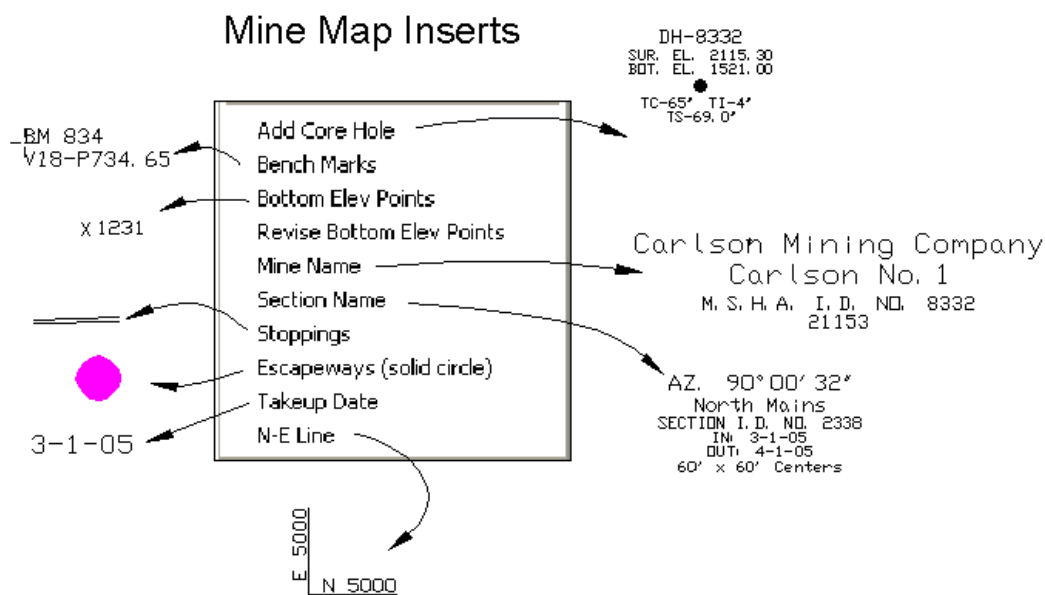
The below image shows the symbols and text that will be drawn using the settings shown in the above dialog.

Top 100.00'
 Thk 10.00'
 Bot 90.00'

Pulldown Menu Location: Notes
Keyboard Command: minepoints

Mine Map Insert Commands

Here are ten very useful drawing embellishment commands. Add Core Hole, Bench Marks, Bottom Elev Points, Mine Name, Section Name, Stoppings, Escapeways (solid circle), Takeup Date, and N-E Line are straight forward routines that place their corresponding information in the drawing using the standard layer name, color, font and height. The prompts for all nine inserts, as well as the Revise Bottom Elev Points routine, are listed below.



Prompts

For Add Core Hole:
Enter or pick 'x' and 'y' co-or of corehole location: *pick a point*
Corehole name: *DH-8332*

Surface elevation: 2115.30
Bottom of coal seam elevation: 1521
Total coal in inches: 65
Total impurities in inches: 4

For Bench Marks:

[node on] Pick Location Of Benchmark Elevation: *pick a point*
Enter BM Number: 834
Enter Elevation of BM: 734.65
Additional Text (Enter if none): V18-P
Pick Location For The BM Text: *pick a point*
Pick Alignment Point For Text: *pick a point*

For Bottom Elev Points:

Elevation : 1231
Pick pt. for elevation mark 'x' : *pick a point*
Pick point at beginning of label: *pick a point*
Pick point for label alignment: *pick a point*

For Revise Bottom Elev Points:

Select Bottom Elevation Marks & Text to Revise: *pick object*

For Mine Name:

Company name : *Carlson Mining Company*
Mine name : *Carlson No,1*
MSHA I.D. N.O.: 8332
State I.D. N.O. : 21153
Pick point at beginning of label: *pick a point*
Pick point for label alignment: *pick a point*

For Section Name:

Degrees of panel azimuth: 90
Minutes of panel azimuth: 00
Seconds of panel azimuth: 32
Panel name : *North Mains*
Panel I.D. n.o. : 2338
Panel in date : 3/1/05
Panel out date : 4/1/05
Additional notes : *60'x60' Centers*
Pick point at beginning of label: *pick a point*
Pick point for label alignment: *pick a point*

For Stoppings:

Pick point for beginning of stopping line orientation: *pick a point*
Pick point for end of stopping line orientation: *pick a point*
Pick point to beginning stopping: *pick a point*
Pick point to end stopping: *pick a point*

For Escapeways:

Pick pt. for escapeway symbol: *pick a point*

For Takeup Date:

Takeup date [mm-dd-yy format]: 3-1-05
Pick point at beginning of label: *pick a point*
Pick point for label alignment: *pick a point*

For N-E Line:

Enter Northing: 5000

Enter Easting : 5000

This will be placed in the drawing at the corresponding northing and easting.

Pulldown Menu Location: Notes

Keyboard Command: chole, benchmark, botelev, botelev2, mname, sname, stop, xways, tdate, neline.

Prerequisite: None

Dynamic Blocks and Tool Palettes

The included dynamic blocks were created for use as Underground Mining symbols; they follow in most part, the requirements from the State of West Virginia guidelines. Other symbols may be found at the West Virginia website <http://www.wvminesafety.org/minemaps.htm>, in the downloadable PDF file and also in the book titled *Mine Mapping & Layout*, by W. Randolph Williams. Only underground mining map symbols and some SCSR Emergency Plan symbols will be released as part of the Carlson Underground Mining Module, version 2015. Dynamic Blocks and Tool Palettes only exist in AutoCAD. They are not found in the IntelliCAD engine of Carlson. Future releases of blocks may consist of CO systems, AC and DC Power, and the Tracking system.

All of the blocks may not work for everyone, based on regional requirements. Blocks and Tool Pallets are readily editable so they can adapt to any desired format. Any existing block can be converted into a dynamic block. All blocks should be placed into one drawing, in the AutoCAD layer 0 to establish mapping standards for blocks and layers for all users in a company.

Dynamic Blocks were first introduced in AutoCAD 2006. Their benefit is they can be edited without exploding them. You can even manipulate the block appearance during or after inserting it into a drawing.

Traditional blocks:

- Multiple steps to place and align
- Design changes require erasing and reinserting or exploding, edit and redefining

Dynamic blocks:

- Automatically align to nearby geometry
- Cycle between multiple insertion points
- Change visibility of geometry to streamline design changes
- Geometry can be edited within a block without exploding it

Dynamic blocks are extremely useful in that they can adapt to the current needs of your design. Blocks representing stoppings or any mining symbols are easily made as dynamic blocks. Dynamic blocks can fill the need for multiple blocks. All blocks will be consistent throughout a company.

Tool palettes are used to manage a block library. Tool palettes provide a consistent interface to insert consistent blocks into all drawings. Tool palettes allow the ability to maintain the blocks in one drawing on a network, so every user reading that drawing is updated instantly. All changes to the blocks should be made in this one drawing. Carlson provides a standard AutoCAD drawing with several mining dynamic blocks. The blocks in this drawing are general standards, and do not imply these symbols will comply with your regional and government agencies. But the blocks are easy to edit so they do comply. The user will need to create a new Tool Palette, and then use the Design Center to drag and drop the blocks from this drawing into the new Tool Palette. Details on this procedure are below.

This document covers the following topics regarding Dynamic Blocks and Tool Palettes

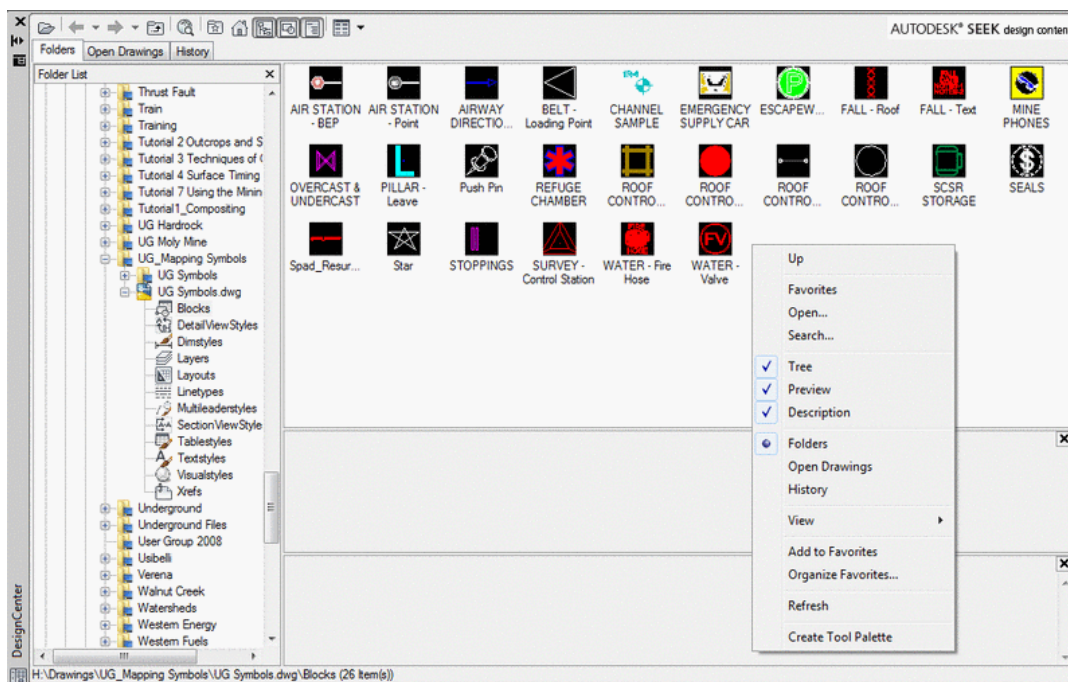
- Tool Palettes

- Creating Dynamic Blocks and Editing
- Standards of blocks and layers
- Design Center
- Revising the blocks
- Updating the drawing
- Reporting information about the Dynamic Blocks in the drawing: Command `_eatttext`

Creating a Tool Palette

To use the included library of underground mining symbols, the steps to create a Tool Palette containing them are as follows. There are two main methods to create a Tool Palette.

1. Use the Tool Palette shortcut menu to create a new, empty palette. This is found by right-clicking anywhere on the tool palette itself, and choosing New Palette. Enter a name for it, such as *UG Symbols* and then add the blocks to it as defined below. The Design Center is the preferred method to add the provided blocks into the new Tool Palette.
2. Use a DesignCenter shortcut menu to create a Tool Palette tab with predetermined content. The DesignCenter button is found in the AutoCAD Standard toolbar or by typing `_adcenter`. Right Click in a blank area of the DesignCenter and Select "Create Tool Palette of Blocks" or "Create Tool Palette". Then hold down "Shift" key and Select the blocks to add to the Tool Palette.



Methods for Adding Tools to the New or Existing Tool Palette. Tools can be added to a new or existing Tool Palette using several methods.

- Drag tools from the drawing space to the palette. Tools include geometric objects, dimensions, fills, blocks, xrefs, raster images, and tables. Others could be lights, cameras, visual styles, and materials (not available in AutoCAD LT).
- **Drag drawings, blocks, and hatches from DesignCenter to the Tool Palette. Drawings that are added to a tool palette are added to the palette.**
- Drag palettes from the Customize. Type `Customize` at the command line to bring it up.

- Drag commands from the Command List pane on the Customize User Interface (CUI) Editor and drop them into a Tool Palette.
- Cut/Copy and Paste tools from one Tool Palette to another.

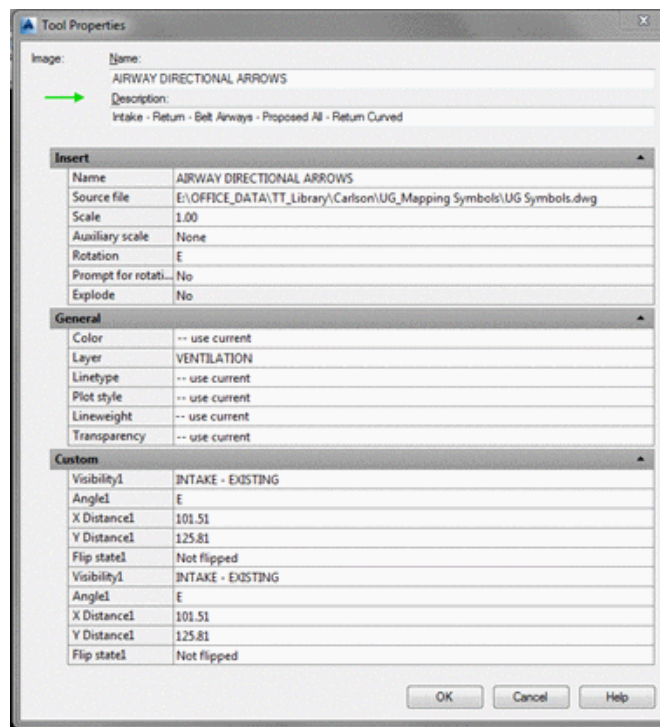
Here is what the new Tool Palette should look like after the Dynamic Blocks are imported.



Customizing the Tool Palette

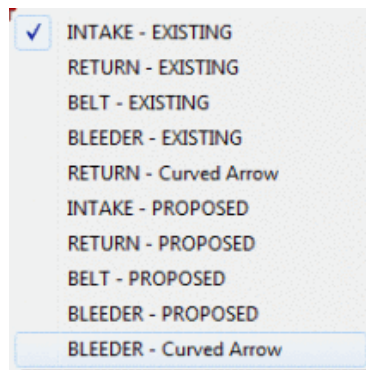
Once a Tool Palette is created, it can be modified and customized by the following methods.

- Rearrange the tools by dragging or sorting them.
- Add text and separator lines.
- Move a Tool Palette tab up and down the list of tabs by using the shortcut menu or Customize dialog box.
- Delete Tool Palettes that you no longer need. (Tool Palettes that are deleted are lost unless they are first saved by exporting them to a file.)
- Set the path to the Tool Palette files location on the Files tab in the AutoCAD Options dialog box. This path can be to a shared network location.
- Associate a customizable Tool Palette group with each panel on the ribbon.
- Change a Tool Palette's read-only status in the palettes folder where they are stored. If a Tool Palette is set to be read-only, a lock icon is displayed in a lower corner of the Tool Palette. This indicates that it cannot be modified beyond changing its display settings and rearranging the icons.
- Tool Properties: How to modify the properties and images of single tools.
- On the Tool Palette, right-click a Symbol/Tool and Select Properties.
- In the Tool Properties dialog box, update or edit the settings.
- Different settings can be used for different tools.



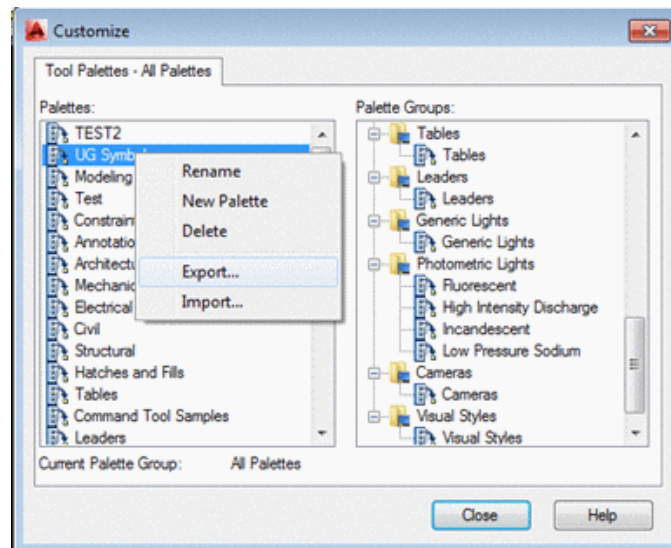
Details of the Tool Properties dialog are listed below.

- **Image Name:** This is the name of the block.
- **Description:** This is a description describing what the block may do.
- **Insert Name:** Block name.
- **Source file:** This is the drawing where all of the Tool Palettes symbols are stored, preferably on the server for others to use for standards. It should be a protected drawing so that users are not able to modify this drawing.
- **Scale:** Default as to what size the Block was drawn at. This can be changed as needed.
- **Layer:** Assigned Layer
- **Custom:** In this case it is only showing the Visibility in the Block.



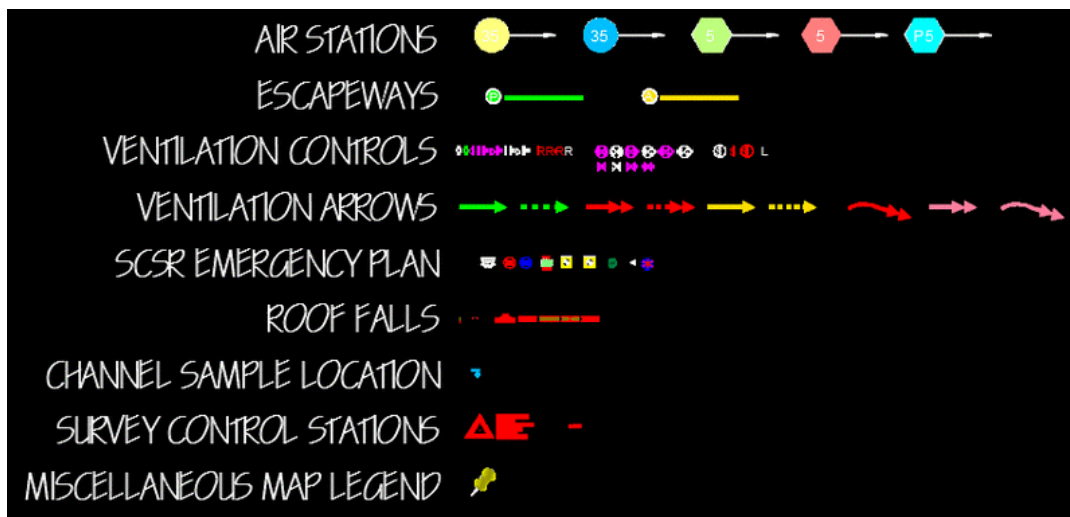
If the Dynamic Blocks are modified in the protected source drawing, such as scale, color or linetype, this revision will change every other drawing that has used it. It may be necessary to do the following to force the change. First, On the Tool Palette Select the tool or Dynamic Block and right-click and Select "Redefine". If these changes are going to be pushed out to all the users then it is necessary to export the Tool Palette and the users will need to re-import the Tool Palette with the changes. That steps to do this are shown below.

- **Exporting a Custom Edited Tool Palette:** To first export a Tool Palette to be shared, first right-click on the Left Side of the Tool Palette or a blank spot inside the Tool Palette and select "Customize Palettes". In the Customize dialog box, to export a Tool Palette, select under Palettes with a right-click on a Tool Palette name and select Export. In the Export Palette dialog box, specify a file name and location and then pick Save. This creates a New Folder with the name of the Tool Palette. A folder for the images is created under it and contains images for the tools. If there is an existing XTP already there, then it can be overwritten.



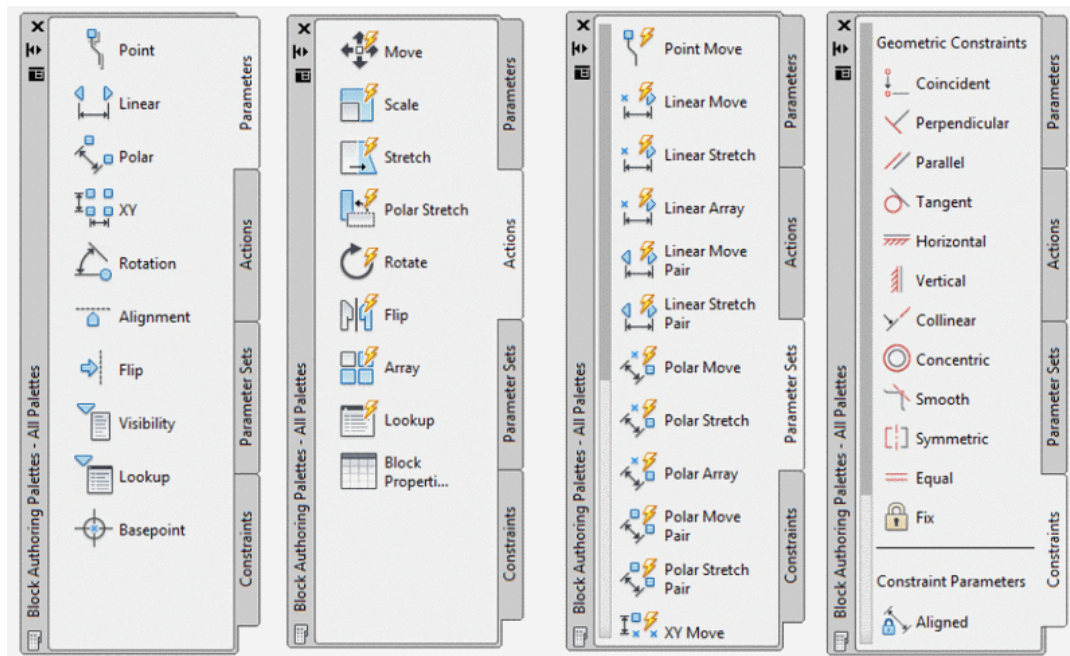
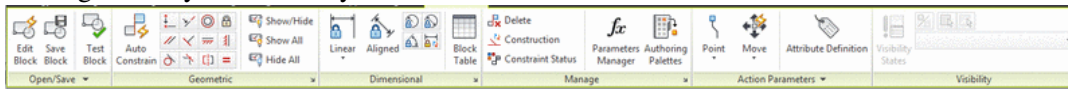
- **Importing a Tool Palette:** This is very similar to Export. Right-click on the Left Side of the Tool Palette or a blank spot inside the Tool Palette. Choose "Customize Palettes". In the Customize dialog box, select Import a tool palette under palettes, right-click on a tool palette name and select Import. In the Import Palette dialog box, specify a file name and location. Click Open.

Here is an image of the source drawing provided by Carlson showing some of the available Dynamic Blocks. User's are allowed to add to it with additional custom Dynamic Blocks, and then include them on the Tool Palette.



Dynamic Block Editing

The AutoCAD ribbon for Blocks is shown here. It appears after picking or selecting a Dynamic Block that is inserted into the dwg, or in the source drawing. To insert a Dynamic Block, turn on the Tool Palettes and just drag and drop a symbol in the drawing. The Authoring Palettes, Parameters and Actions tabs are the functions being used in the Underground Symbols library of blocks.



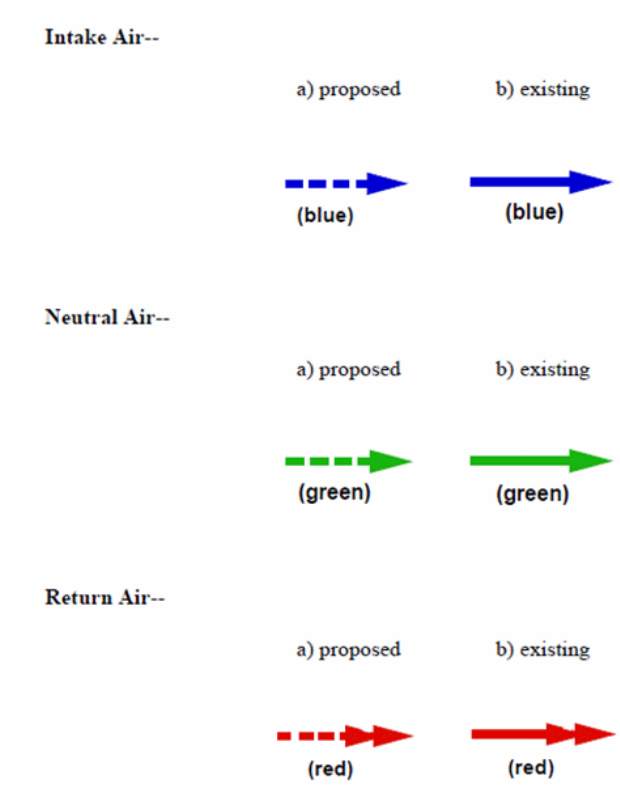
The Parameter Sets and the Constraints, are not covered in this document. For example, the Constraints tab could be used for the Aligned method to place a door or window in an architectural drawing with rules of placing a window in the center of the wall, and the door placed 6 inches from a perpendicular wall.

P	Alignment								
A	Visibility								
R	Basepoint								
A	Point								
M	Linear								
E	Polar								
T	XY								
E	Rotation								
R	Flip								
S	Lookup								
		Move	Stretch	Scale	Array	Polar Stretch	Rotate	Flip	Lookup
		ACTIONS							

Parameters and Action work together to provide the editing capability of Dynamic Blocks. Regular blocks could only have a "Basepoint" parameter. The matrix is a quick reference to determine the appropriate action for each parameter.

Once the Dynamic Block is inserted, pick on it to activate the blue grips. They allow for the editing and orientation

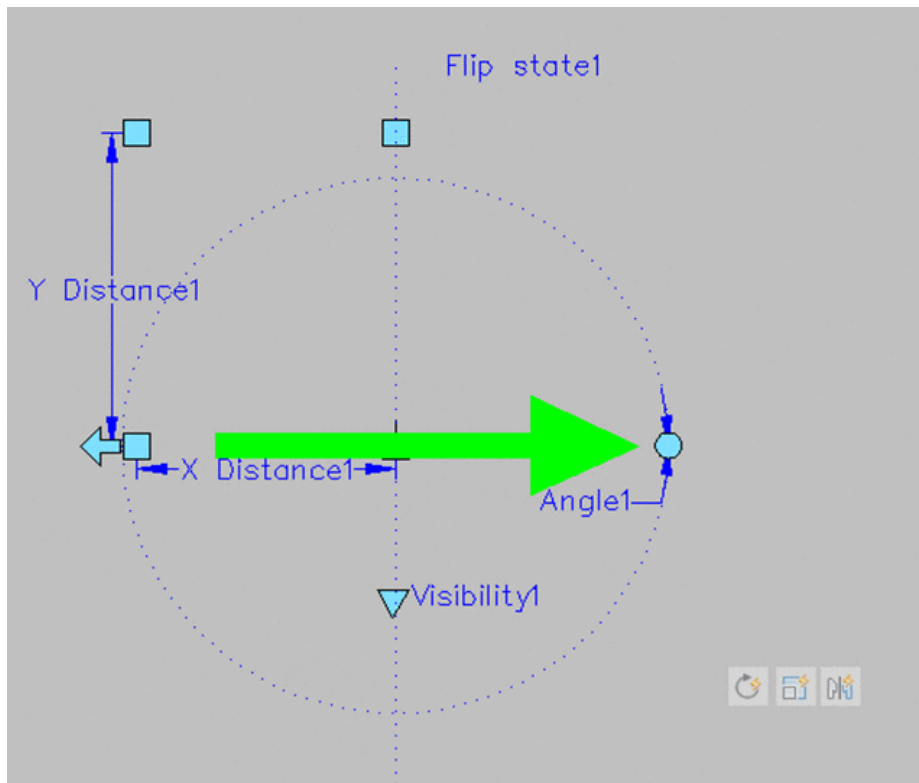
of the block. Picking the down arrow for Visibility1 brings up a list of other blocks and appearances. Here is an example showing ventilation arrows as Dynamic Blocks:



When drawing ventilation arrows, some MSHA districts will not use Neutral Air. They require it to be Belt Air. Some would recommend that the color of the arrows match the reflector underground, or some may require that the double arrow represents the Intake Air instead of the Return Air. Dynamic Blocks allow for editing of colors and arrow heads so they comply.

Dynamic Block Creation and Editing

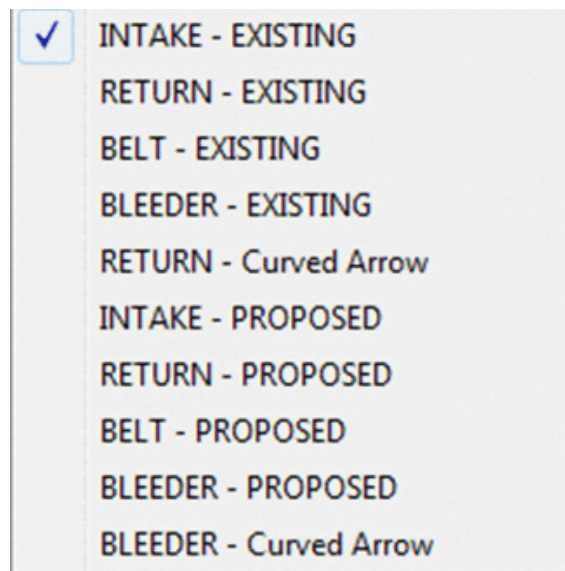
To see the editing features, select the inserted Dynamic Block to show the blue grips and icons. To see the image shown here, right click on a Dynamic Block in the tool palette, and select Block Editor.



1. Visibility1:

- Intake - Existing
- Return - Existing
- Belt - Existing
- Bleeder - Existing
- Return - Curved Arrow
- Intake - Proposed
- Return - Proposed
- Belt - Proposed
- Bleeder - Curved Arrow

Picking on the Visibility1 downward ∇ will bring up the list of visibilities, or types of arrows in this case. This one block can be used for any of the above visibility states in the drawing.



2. Flip state1: Flips the direction of the symbol 180 degrees.

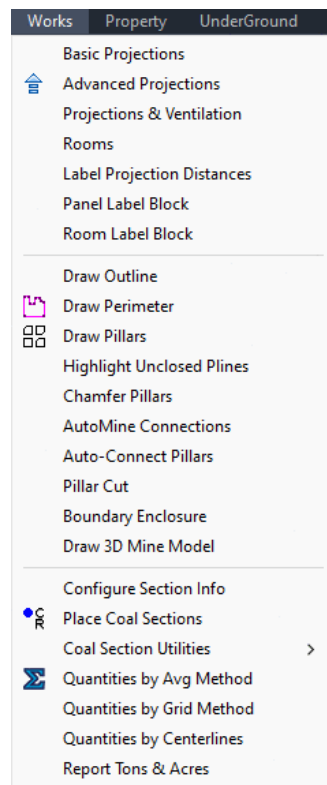
3.X and Y Distance1: Scales and sizes the symbol.

4.Angle1: Rotates the symbol.

All of the above actions are done with the mouse control. As more "Actions" icons are added to the block, they can be moved to any position around the symbol. Other editing icons are for Rotate, Scale and Flip.

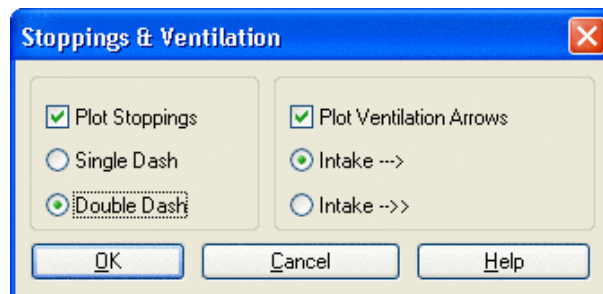
Works Menu

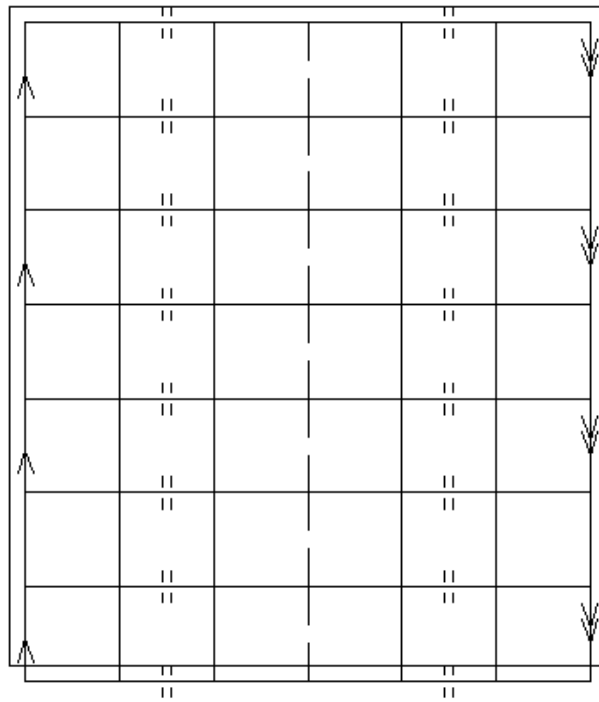
The Works pulldown menu contains commands for underground mines and is divided into three sections with projections commands at the top, placing pillars and perimeters commands in the middle, and quantity commands at the end.



Basic Projections

Basic Projections will only produce a rectangular grid of projections (not angular). The sequencing begins by choosing a starting point, direction, and distance. Then enter the number of entries and their sizing. After the projection is drawn, stoppings and ventilation arrows may be added. This routine has fewer prompts and is more automatic than Advanced Projections. All projection lines are on distinct layers (PROJECTIONS, PROJSTOPPING, PROJVENTARROWS).





Prompts

Start pt. of belt entry: *pick a point* You are automatically placed in intersect and node snap mode.

<P>ick End Point For Belt or [A] For Azi/Dist: *Enter*

Pick end pt. of belt entry: *pick a point*

Number of entries on left side: *3*

Number on right side: *3*

Entry spacing: *60*

Crosscut spacing: *60*

Plot Outer Rib Line y/<n>: *Y*

Cut Width <20>: *press Enter*

Offset to starting rib line (e.g. -10,0,<10>): *press Enter*

Offset to ending rib line (e.g. -10,0,<10>): *press Enter*

Offset xcuts y/<n>: *press Enter*

Pick pt. on xcut for beginning of stopping line: *pick a point*

Draw another stopping line <y>/n: *press Enter*

Pick pt. on xcut for beginning of stopping line: *pick a point*

Draw another stopping line <y>/n: *N*

Pick pt. on entry to begin drawing ventilation arrows [nea on]: *pick a point*

Distance between ventilation arrows: *120*

<I>nake or [R]eturn: *I*

Draw ventilation arrows on another entry <y>/n: *Y*

Pick pt. on entry to begin drawing ventilation arrows [nea on]: *pick a point*

Distance between ventilation arrows: *120*

<I>nake or [R]eturn: *R*

Draw ventilation arrows on another entry <y>/n: *N*

Project another panel <y>/n: *N*

Pulldown Menu Location: *Works*

Keyboard Command: *panell*

Prerequisite: *None*

Advanced Projections

Advanced Projections is a flexible routine that is an extension of Basic Projections. Besides creating projections this routine can also draw pillars and perimeter by using the Draw Pillars option. Other options include angling the crosscuts, offsetting entries, variable length entries, and different labeling schemes. All projection lines are on distinct layers with defaults of PROJECTIONS, PROJSTOPPINGS, PROJVENTARROWS. The pillars are drawn by default in the PILLARS layer and the perimeter is drawn by default in the PERIM layer.

Prompts

Pick Start Point For Belt: *pick a point*

Pick End Point For Belt, or <A> For Azi/Dist: *A*

Enter Azimuth ddd.mmss <>: *90*

Enter Distance: *600*

How Many Entries Left Of The Belt <0>: *3*

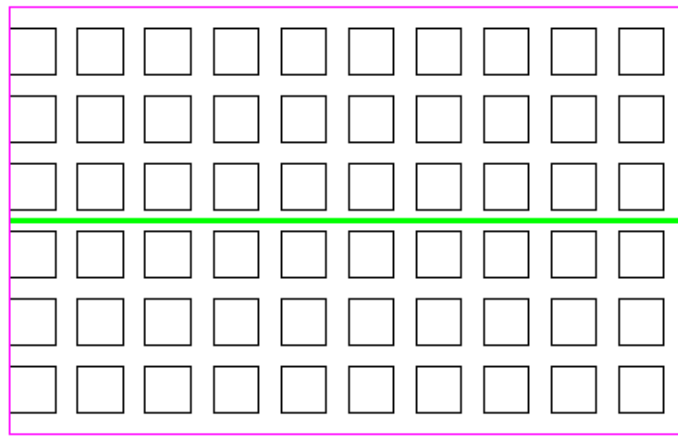
How Many Entries Right Of The Belt <0>: *3*

The screenshot shows the 'Panel Settings' dialog box with the following configurations:

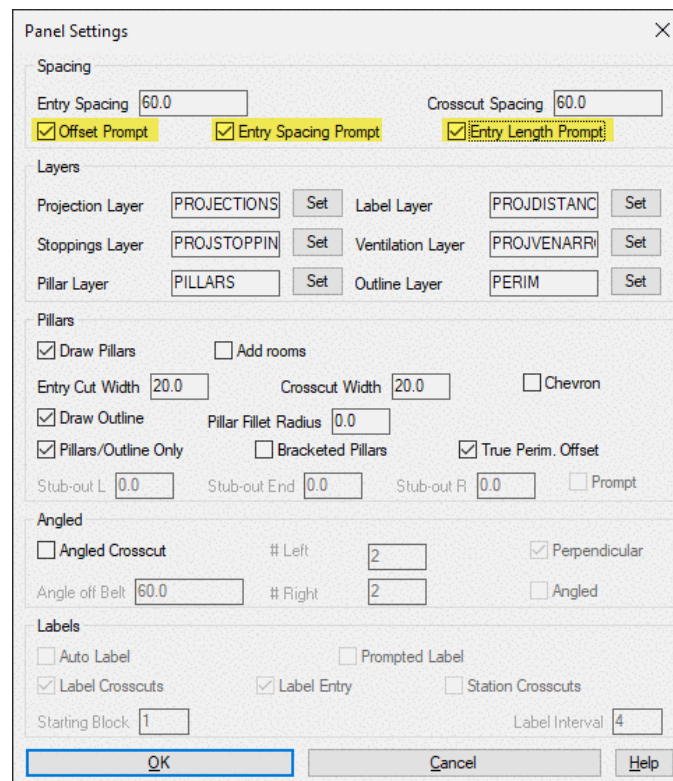
- Spacing:** Entry Spacing: 60.0, Crosscut Spacing: 60.0. Offset Prompt, Entry Spacing Prompt, Entry Length Prompt.
- Layers:** Projection Layer: PROJECTIONS, Label Layer: PROJDISTANC, Stoppings Layer: PROJSTOPPIN, Ventilation Layer: PROJVENARR, Pillar Layer: PILLARS, Outline Layer: PERIM.
- Pillars:** Draw Pillars, Add rooms. Entry Cut Width: 20.0, Crosscut Width: 20.0, Chevron. Draw Outline, Pillar Fillet Radius: 0.0. Pillars/Outline Only, Bracketed Pillars, True Perim. Offset. Stub-out L: 0.0, Stub-out End: 0.0, Stub-out R: 0.0, Prompt.
- Angled:** Angled Crosscut, # Left: 2, Perpendicular. Angle off Belt: 60.0, # Right: 2, Angled.
- Labels:** Auto Label, Prompted Label. Label Crosscuts, Label Entry, Station Crosscuts. Starting Block: 1, Label Interval: 4.

Buttons: OK, Cancel, Help.

Based on the preceding dialog box responses, the following mine layout is created:



To create various entry spacings and lengths choose the Offset Prompt, Entry Spacing Prompt, and/or Entry Length Prompt check boxes.



Panel Settings Dialog

Spacing for Left Entry No. 1 <60.0>: 90

Spacing for Left Entry No. 2 <60.0>: *press Enter*

Spacing for Left Entry No. 3 <60.0>: *press Enter*

Spacing for Right Entry No. 1 <60.0>: 90

Spacing for Right Entry No. 2 <60.0>: *press Enter*

Spacing for Right Entry No. 3 <60.0>: *press Enter*

Enter Offset For This Heading <0>: *press Enter*

First Length along Entry: 60

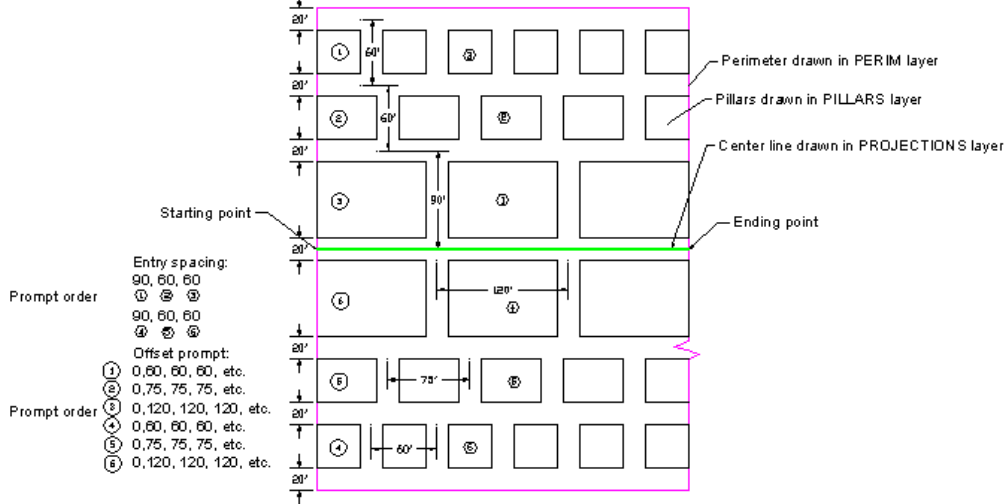
Enter Length along Entry <540.0>: 60

Enter Length along Entry <480.0>: 60

Enter Length along Entry <420.0>: 60

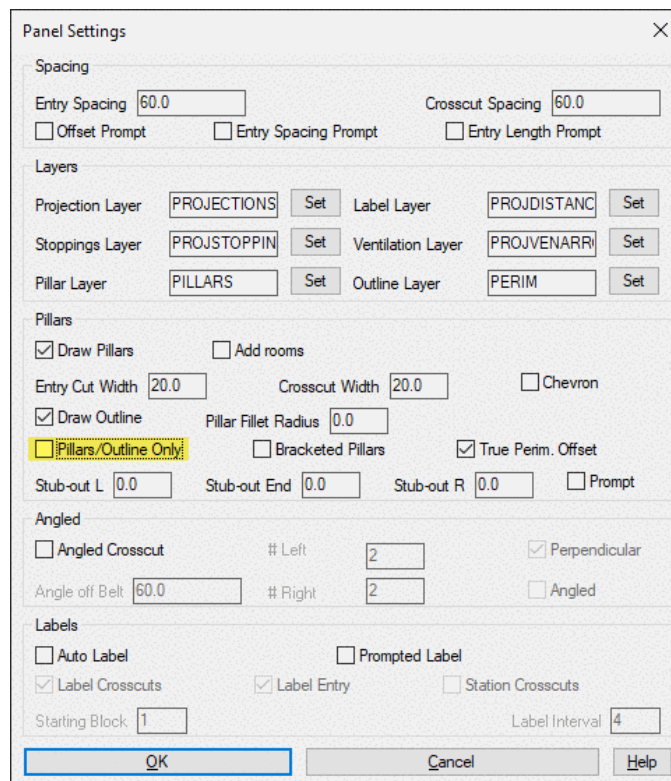
Enter Length along Entry <360.0>:60
Enter Length along Entry <300.0>: 60
Enter Length along Entry <240.0>: 60
Enter Length along Entry <180.0>: 60
Enter Length along Entry <120.0>: 60
Enter Length along Entry <60.0>: 60
Enter Offset For This Heading <0>: 0
First Length along Entry: 75
Enter Length along Entry <525.0>: 75
Enter Length along Entry <450.0>: 75
Enter Length along Entry <375.0>: 75
Enter Length along Entry <300.0>: 75
Enter Length along Entry <225.0>: 75
Enter Length along Entry <150.0>: 75
Enter Length along Entry <75.0>: 75
Enter Offset For This Heading <0>: *press Enter*
First Length along Entry: 120
Enter Length along Entry <480.0>: 120
Enter Length along Entry <360.0>: 120
Enter Length along Entry <240.0>: 120
Enter Length along Entry <120.0>: 120
Enter Offset For This Heading <0>:
First length along Entry: 60
Enter Length along Entry <540.0>: 60
Enter Length along Entry <480.0>: 60
Enter Length along Entry <420.0>: 60
Enter Length along Entry <360.0>: 60
Enter Length along Entry <300.0>: 60
Enter Length along Entry <240.0>: 60
Enter Length along Entry <180.0>: 60
Enter Length along Entry <120.0>: 60
Enter Length along Entry <60.0>: 60
Enter Offset For This Heading <0>:
First length along Entry: 75
Enter Length along Entry <525.0>: 75
Enter Length along Entry <450.0>: 75
Enter Length along Entry <375.0>: 75
Enter Length along Entry <300.0>: 75
Enter Length along Entry <225.0>: 75
Enter Length along Entry <150.0>: 75
Enter Length along Entry <75.0>: 75
Enter Offset For This Heading <0>:
First length along Entry: 120
Enter Length along Entry <480.0>: 120
Enter Length along Entry <360.0>: 120
Enter Length along Entry <240.0>: 120
Enter Length along Entry <120.0>: 120

Advanced Projections showing variable entry spacing and offset prompts

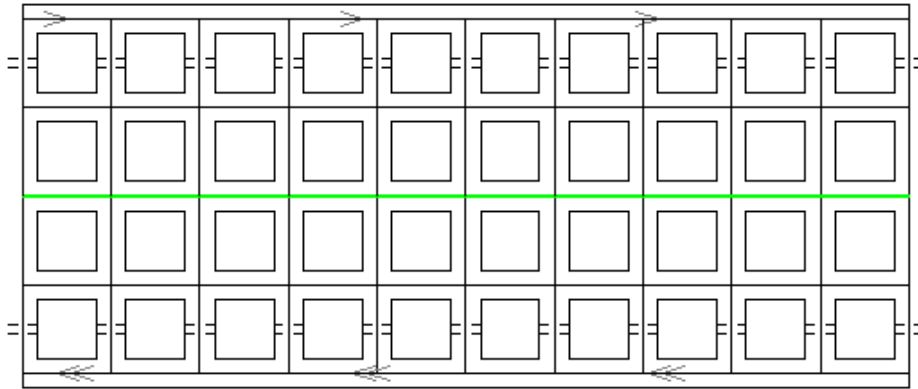


Note: The length of the panel was shortened for display purposes. The PILLARS and PERIM layers are reserved layer names used in other routines in Carlson. The layer names can be changed here if you have other reasons to do so.

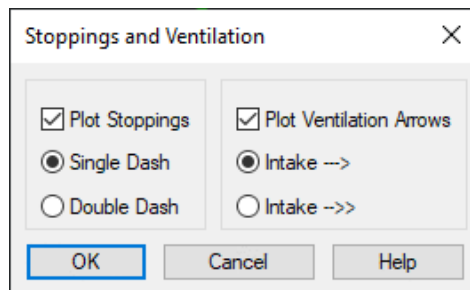
If you want to display the centerlines with the projections and be prompted for ventilation symbols (stoppings and ventilation arrows), then at the Panel Settings dialog box uncheck the Draw Outline and Pillars only checkbox.



The following panel layout is drawn without the ventilation stoppings or arrows.



Then you will see the Stoppings & Ventilation dialog box and upon filling it out you will be prompted as follows:



Stoppings & Ventilation Dialog

Pick pt. on xcut for beginning of stopping line: *screen pick*

Draw another stopping line <y>/n: *Y*

Pick pt. on xcut for beginning of stopping line: *screen pick*

Draw another stopping line <y>/n: *N*

Pick pt. on entry to begin drawing ventilation arrows [nea on]: *screen pick*

Distance between ventilation arrows: *200*

<I>ntake or [R]eturn: *I*

Draw ventilation arrows on another entry <y>/n: *Y*

Pick pt. on entry to begin drawing ventilation arrows [nea on]: *screen pick*

Distance between ventilation arrows: *200*

<I>ntake or [R]eturn: *R*

Draw ventilation arrows on another entry <y>/n: *N*

Upon filling in the prompts, the map will be drawn as shown above.

Angled Projections

It is possible to plot projections as centerlines, as pillars with a perimeter, or both. Within the Pillars section of the dialog, if the three dialog items on the left (Draw Pillars, Draw Outline, Pillars/Outline only) are selected, then only pillars and perimeters are drawn. This type of projection is particularly useful to represent the actual mine for purposes of scheduling equipment, using the Underground Mining Module. Here is an example of angled projections, with a "winged" look created by angling only the first 2 of 4 left and right entries:

If all 4 entries left and right are angled (rather than the 2 above), then the entire projection would be in angled form.

Panel Settings

Spacing

Entry Spacing Crosscut Spacing

Offset Prompt Entry Spacing Prompt Entry Length Prompt

Layers

Projection Layer Label Layer

Stoppings Layer Ventilation Layer

Pillar Layer Outline Layer

Pillars

Draw Pillars Add rooms

Entry Cut Width Crosscut Width Chevron

Draw Outline Pillar Fillet Radius

Pillars/Outline Only Bracketed Pillars True Perim. Offset

Stub-out L Stub-out End Stub-out R Prompt

Angled

Angled Crosscut # Left Perpendicular

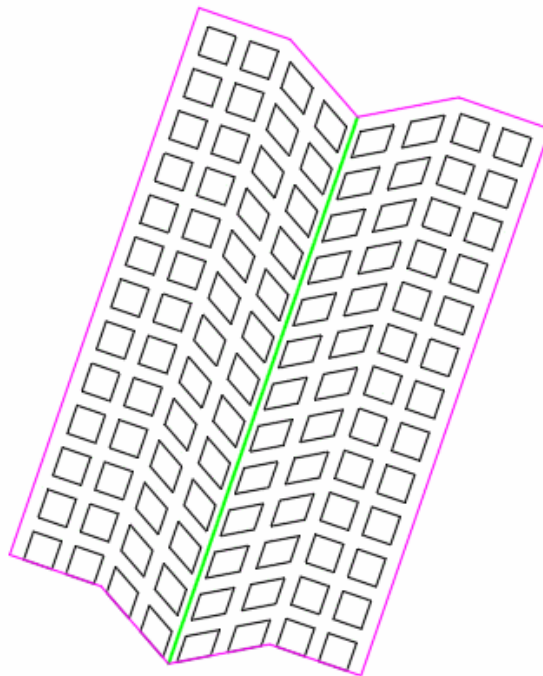
Angle off Belt # Right Angled

Labels

Auto Label Prompted Label

Label Crosscuts Label Entry Station Crosscuts

Starting Block Label Interval



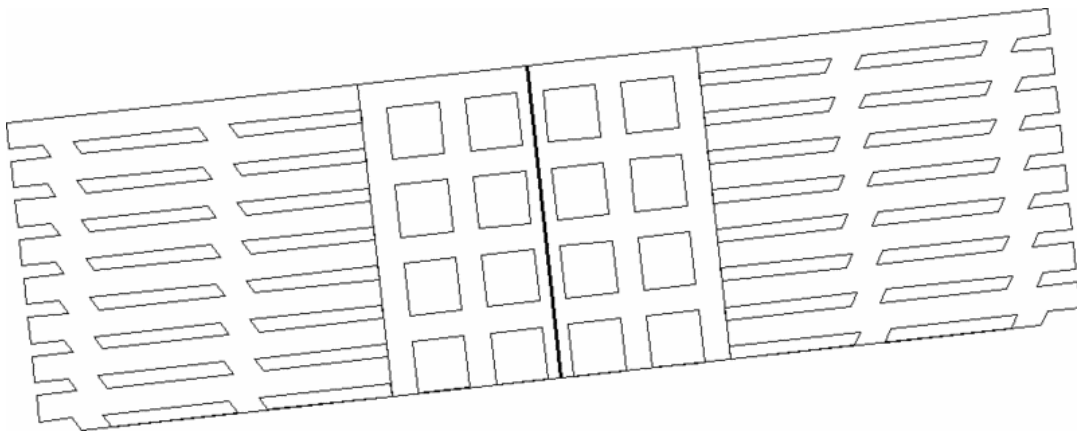
Add Rooms

Typical, Default Room Plotting requires that you click on the left-side Draw Pillars, Draw Outline and Pillars/Outline only, as well as click 'Add Rooms'. A typical 2 left, 2 right standard projection with 120' longrooms (current default) has the following dialog appearance and plot.

The default angles lean the left rooms to the left and the right rooms to the right at 60 degrees. The crosscut spacing on the longrooms default to 1/2 the crosscut spacing on the main projections (60 in our case). I have noticed some irregular rooms if you use different crosscut spacing left and right. In that case, it might be better to plot the right side or left side separately. If the Join Perimeter with Center option is not selected, the projections will be in 3 parts: left, center and right. The outer left and right "extents" of the perimeter default to 30' plus the distance to the center of the left- and right-most entries. In our case, that distance to the far entry is $2 \times 120 = 240$, meaning that the outer punchout is defaulting to 270' left and right. I can change that addition of 30' to any other desired number.

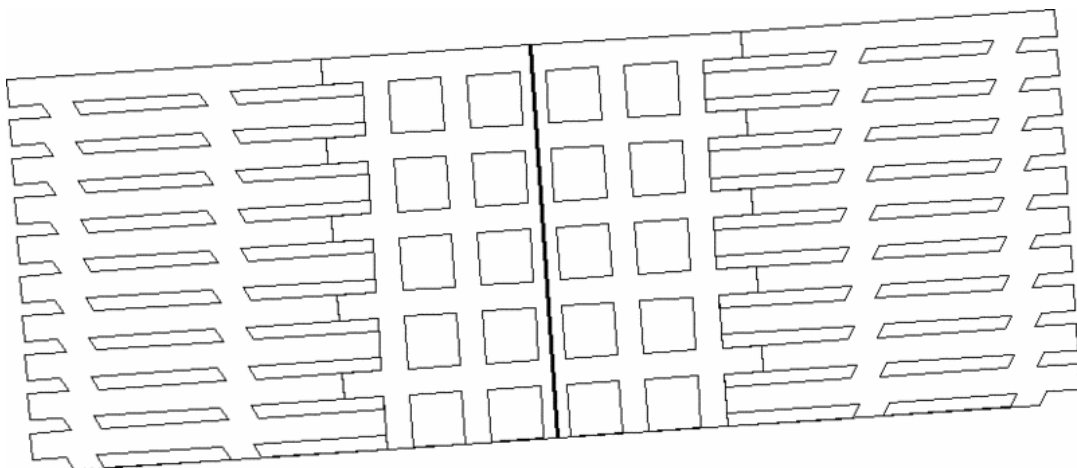
Rooms with Stubouts: You can add stub-outs within the main dialog, and the rooms will attach appropriately. In this example, Stub-out L and Stub-out R were set to 30.

Joining with Center Section: The next variation is to join a left or right section to the center (we don't allow joining all 3 at present). This can be done with or without stub-outs. To activate this option, turn on the Join Perimeter with Center option in the Room Dimensions dialog. Shown below is a left section joined to the center with no stubs involved. Below that is a right section joined with the center with stub-outs. The perimeters have been made bold for emphasis.

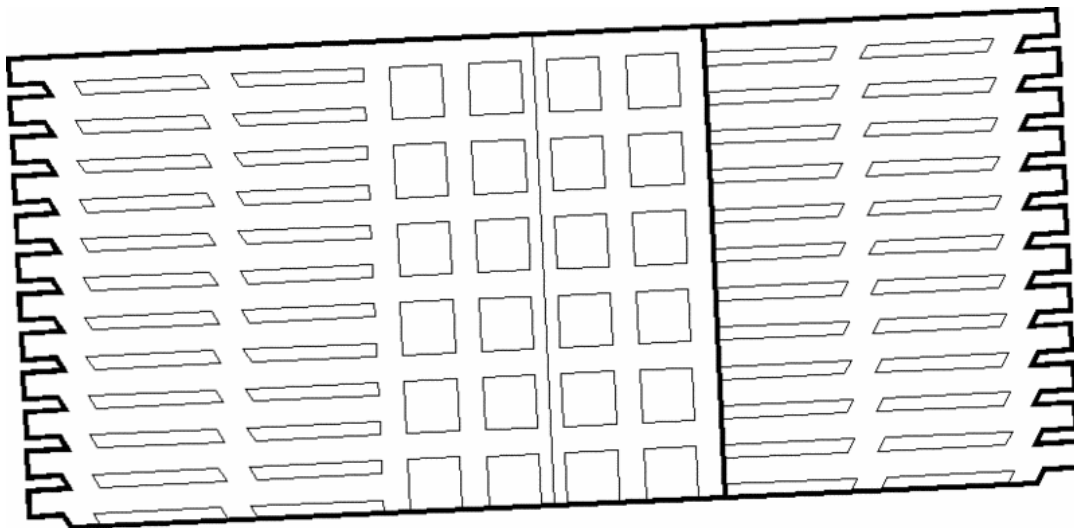


Rooms with stubouts connected to a standard panel

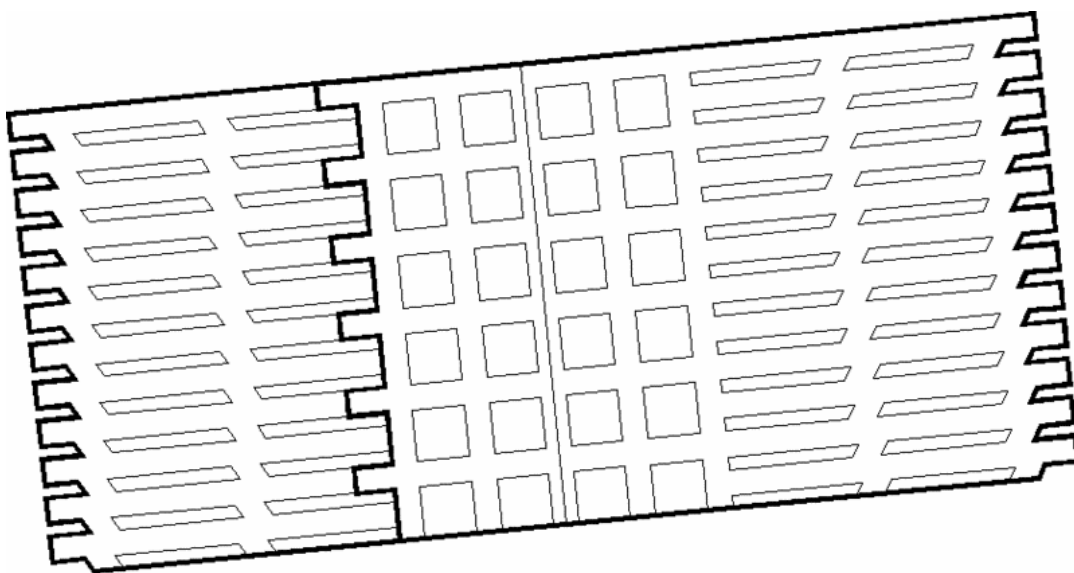
Changing Angles in the Rooms: Rooms can have a variety of angles for cutting through. To allow variation in angles, turn on the Angle Left Prompt in the Room Dimensions dialog. In this example, we did .60 (the default), then 60 and 60 in 3 entry left room. We did not vary the right condition. The prompting for the angle will remember the last entry, so the second "60" becomes just an "Enter" default on the left side.



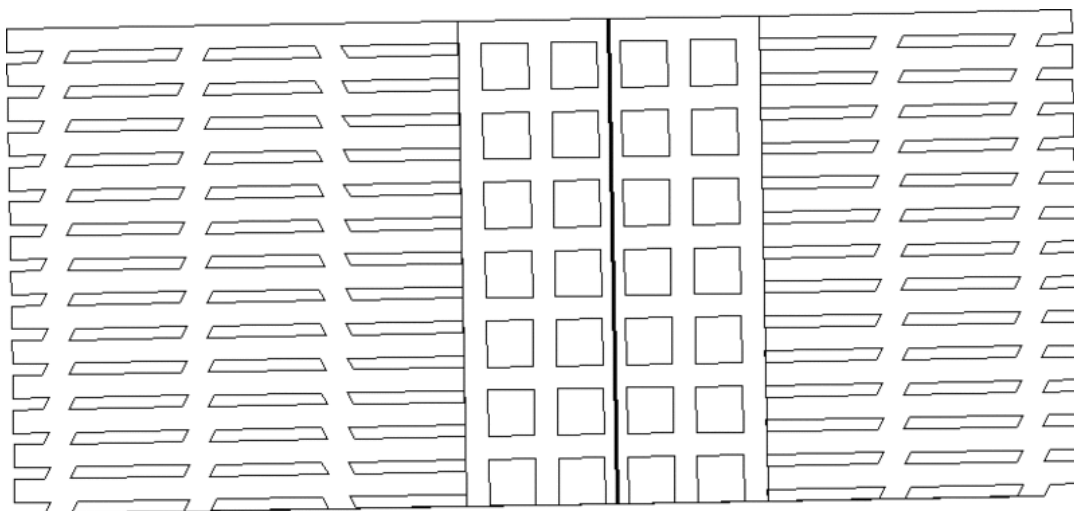
Room with stubouts connected to a panel with stubouts



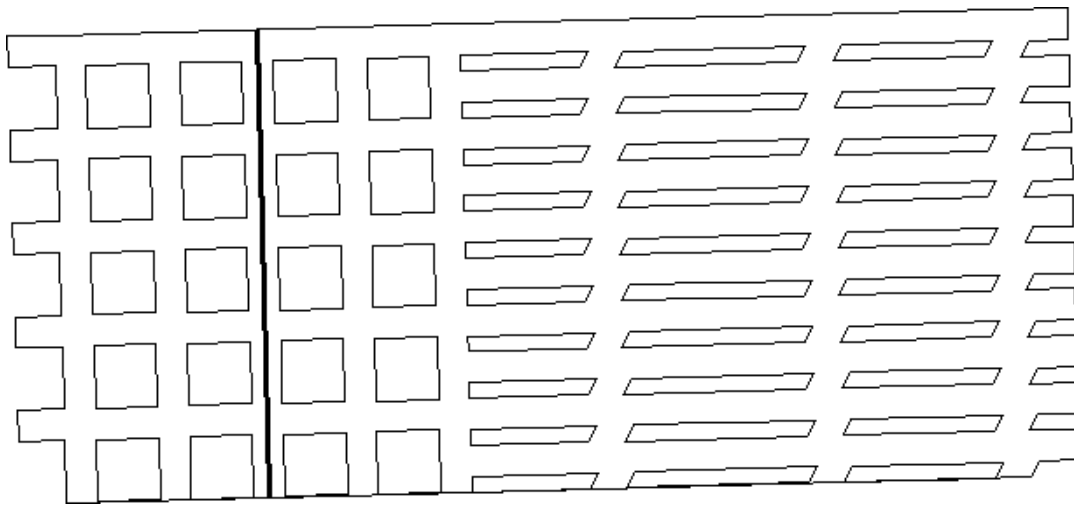
Left room connected to advancing panel



Right room connected to advancing panel



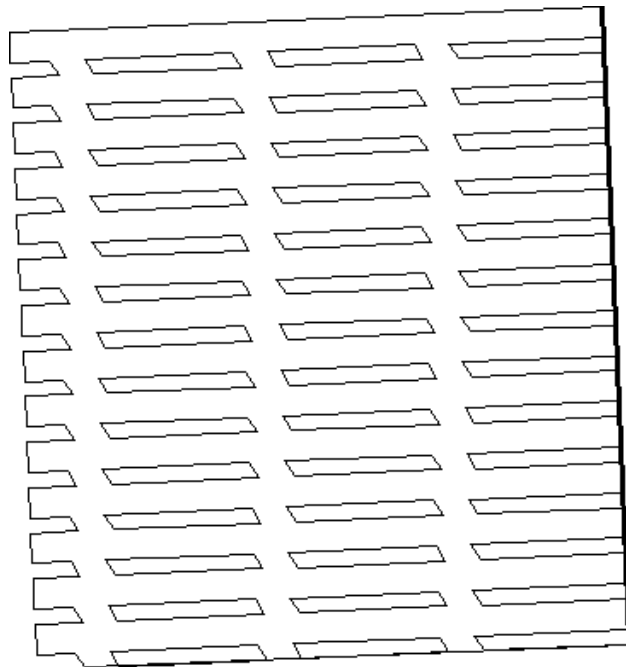
Angled rooms left (3+ X-cuts deep) and angled room right (2+ X-cuts deep)
 Notice neither set of rooms were joined to the panel on the advance



Rooms with entry spacing variations

Entry Spacing Prompt: We can also choose to vary the spacing. In the example below, we did 0 left entries and 3 right entries, and we varied the entry spacing prompt on the right (and included stub-outs and Join Perimeter with Center). We entered 100, 140 and 120 for the three entry spacings and obtained the plot shown below:

Plotting Rooms with No Central Projection: This is useful for attaching left and right rooms to an existing mine plot. You pick your "centerline" as normal to get starting, but you answer 0 to both the number of left and right entries. Normally, this would simply exit the routine, but a new prompt asks "Do Longroom (y/n)". Enter "y" and you can place rooms to the left or right. (The program will only do one side or the other in a "no central panel" projection scheme. Repeat the process to add a right side to the left side rooms.) Here is the "look" of a left-side, 3 room section with no central projection:



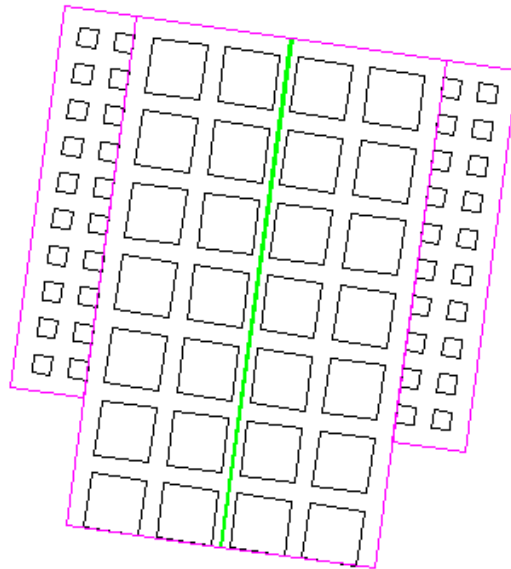
Perpendicular Rooms and Offset Rooms

If the Room Dimension dialog is completed as shown below, rooms can be drawn at 90 degrees to the main entry, and can be offset as many crosscuts as desired. The offset up is measured by counting the number of room-size crosscuts. So if you have 40x40 rooms coming off 80x80 mains, and want to offset two main-size crosscuts (160 feet to the first rooms), then enter 4 (4*40 is 160) for the number of crosscuts to offset. Here is the result:

Room Dimensions

No. of Entries Left <input type="text" value="2"/>	No. of Entries Right <input type="text" value="2"/>
Start at Crosscut # <input type="text" value="4"/>	Start at Crosscut # <input type="text" value="4"/>
Entry Spacing Left <input type="text" value="40"/>	Entry Spacing Right <input type="text" value="40"/>
<input type="checkbox"/> Entry Spacing Prompt	<input type="checkbox"/> Entry Spacing Prompt
Angle Left <input type="text" value="-90.0"/>	Angle Right <input type="text" value="90.0"/>
<input type="checkbox"/> Angle Left Prompt	<input type="checkbox"/> Angle Right Prompt
Crosscut Spacing Left <input type="text" value="40"/>	Crosscut Spacing Right <input type="text" value="40"/>
<input type="checkbox"/> Join Perimeter with Center	<input type="checkbox"/> Join Perimeter with Center

OK Cancel



Pulldown Menu Location: Works

Keyboard Command: panel2

Prerequisite: None

Projections & Ventilation

Projections/Ventilation makes heavy use of specification files. Once parameters have been established for a set of projections within a drawing file, you can save those parameters to a specifications file. When changes or additions are necessary, simply erase any undesired projections, load the file, make changes, and process. The specification file contains all the information contained within the dialog, including the projection and ventilation parameters, mandoor and stopping information, even the layer names to use.

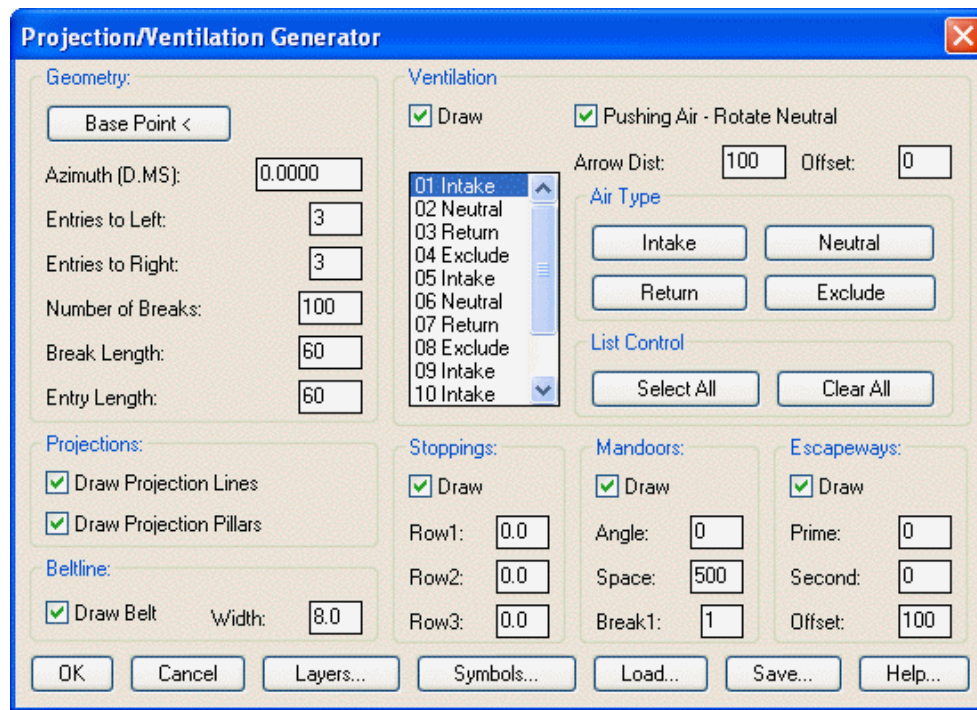
These files can reside in any directory (usually the same directory as the drawing), and have the extension (*.PVS). When the program is activated for the first time within a drawing session, it will look for a PVS file with the same name as the drawing, in the drawing's directory, and if found will load it. If a PVS file with the drawing's name is not found, it will look for a file named DEFAULT.PVS in \PROJECTIONS/VENTILATION subdirectory, and load it.

You can utilize this feature to your advantage by saving the general specifications about a particular drawing in a PVS file with the same name as the drawing. This way, if only the azimuth changed in a set of projections, you could open

the drawing, activate Projections/Ventilation, change the azimuth, and process. You can also access the program's dialog box, enter and then store the specifications you would use most often across drawings as DEFAULT.PVS in \SCAD\LSP subdirectory.

Dialog Control:

When Projections/Ventilation starts, it will display the dialog shown below. All specifications can be entered inside this dialog, and execution of the entity generation process can be initiated as well. Each section of the dialog will be explained in detail in appropriate subsections that follow.



All of Projections/Ventilation's options are accessed easily through this dialog with edit fields for parameters and buttons for each routine available in the program.

Geometry Specifications:

Before generating any entities using Projections/Ventilation, you must specify the geometric specifications of base point, azimuth, entries left/right, number of breaks, break and entry length.

Base Point: This button is used to specify the base point of the geometric entities. After picking this button, the dialog will temporarily disappear, allowing you full screen access to your drawing, and a prompt will appear at the command line requesting you to select a base point.

Choose the base point by picking or typing in coordinates. You MUST use an object snap (NODE, CENTER, or INSERTION) if picking the point graphically, to guarantee that the point selected is exactly on the point, spad circle, or insertion point of a point block.

Azimuth: This edit field is used to enter the projected azimuth. Enter the value in the format of DDD.MMSS, where the whole number of degrees are followed by a period, then minutes and seconds. If the minutes or seconds are less than ten (10), put a leading zero in front such as 01, 05, etc. If the minutes or seconds are zero, enter two zeros 00 as a place holder, especially for minutes.

Entries Left: This edit field is used to enter the number of entries to the left of the base point. A whole number is expected with no decimal points allowed.

Entries Right: This edit field is used to enter the number of entries to the right of the base point. A whole number is expected with no decimal points allowed.

Break Count: This edit field is used to enter the number of breaks to travel. The number of breaks times the break length will determine the total length of the set of projections. A whole number is expected with no decimal points

allowed.

Break Length: This edit field is used to enter the length of each individual break. The break length and entry length do not have to be the same.

Entry Length: This edit field is used to enter the length (width) of each entry from the base point. The break length and entry length do not have to be the same.

Projections:

If the **[x] Draw Projection** Lines toggle is turned on, the program will create projection lines with cross break lines. These projection lines are drawn as lines, having a bylayer color and linetype, and placed on the layer designated in the layer dialog. See the section on layers for more information about changing layers of Projections/Ventilation entities.

Checking the **[x] Draw Projection Pillars** toggle can be used to simulate mine pillars. The simulated mine pillars will be polylines, having a bylayer color and linetype, and placed on the layer designated in the layer dialog.

Beltline:

Checking the **[x] Draw Belt** toggle will generate a set of parallel lines, centered about the base point, and running the entire length of the projections on the specified azimuth. The beltline will be lines, having a bylayer color and linetype, and placed on the layer designated in the layer dialog

Ventilation:

Ventilation can be generated along with projections by checking the **[x] Draw toggle** in the ventilation section and supplying a few additional parameters.

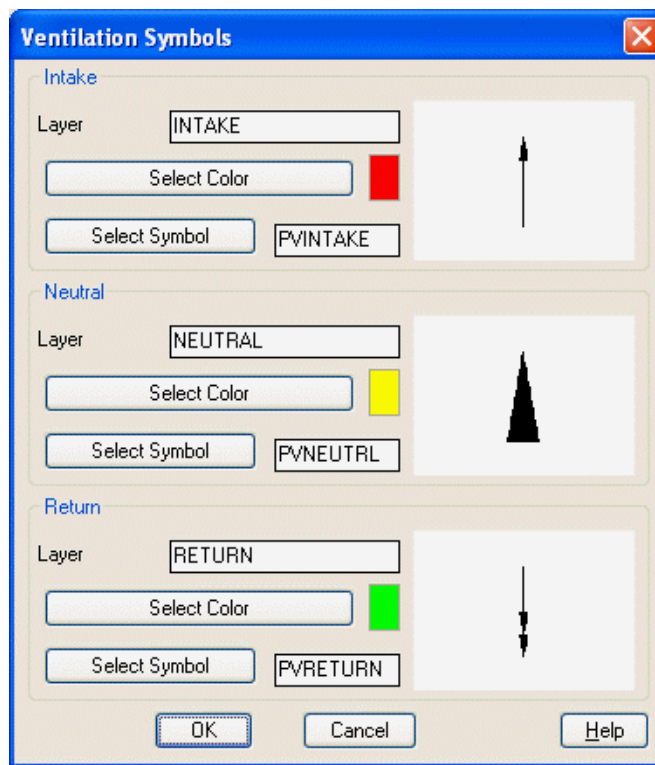
Distance Between Arrows []: This edit field is used to enter the distance between the sets (rows) of ventilation arrows. The sets will be placed along the entire run of projections.

The entry type list box shown, is used to setup the type of air flow in each of the entries. The program is setup to handle up to thirteen entries, but you can use as few as needed.

The process works by clicking your pointer button on the entries (in the list box) that you wish to change. When clicked they will become highlighted, or if they are already selected they will be un-highlighted. To change the air flow once entries are selected, press the button on the right corresponding to the type of air flow in each entry as Intake, Neutral, Return, or Exclude. If no air flow is to be designated for the entry, choose the Exclude button, which will draw no arrows. There are more entries available and will fit in the list box, therefore you may need to use the down arrow button on the scroll bar to view and change these.

If all items in the list need to be selected, press the Select All button, which highlights all items in the list. Additionally you can press the Clear All button, which un-highlights all items in the list.

Selecting the Symbols button allows customization of the arrows. Any of the mining symbols can be selected here.



Stoppings:

To generate stoppings while generating projections and ventilation, turn on the [x] **Draw toggle** in the stoppings section and enter the headings where they should appear. The program can handle up to three rows of stoppings for split ventilation systems.

Row 1-3: These edit fields are used to indicate the heading in which the stopping will be placed. Since stoppings are normally placed halfway between headings, the valid input will usually have a decimal value of (.5). For example, as shown above, if a row of stoppings is to be placed between the 4th and 5th entries, the valid input is 4.5. To exclude a row, set the value to 0.0.

The block inserted can be modified, and is stored as PVSTOPPIN.dwg in the SCAD\SUP subdirectory.

Mandoors:

Mandoors can be automatically added to stoppings. Turn on the [x] **Draw toggle** in the mandoor section and specify the additional parameters below.

Angle: Indicates the rotation angle of the door symbol when placed on the stopping. The default value of 0 should provide the desired results in most cases, but can be changed if needed.

Space: Indicates the distance between doors along the row of stoppings. Usually set to 300 or 500 feet, however any numerical value is acceptable. The program will track the running distance and automatically choose the correct break to place the mandoor in.

Break 1: Indicates the break, relative to the Base Point, that the first mandoor should fall into. The program will begin tracking the space (described above), and place additional doors at the designated distance from this first break.

The block inserted can be modified, and is stored as PVMANDOR.dwg in the SCAD\SUP subdirectory.

Escapeways:

You can optionally place sets of escapeway symbols. Turn on the [x] **Draw toggle** in the escapeway section and specify the additional parameters below.

Prime: Indicates the entry number for the primary escapeway. The block is stored as PVESCPRI.dwg. Valid input would be a whole number with no decimal places.

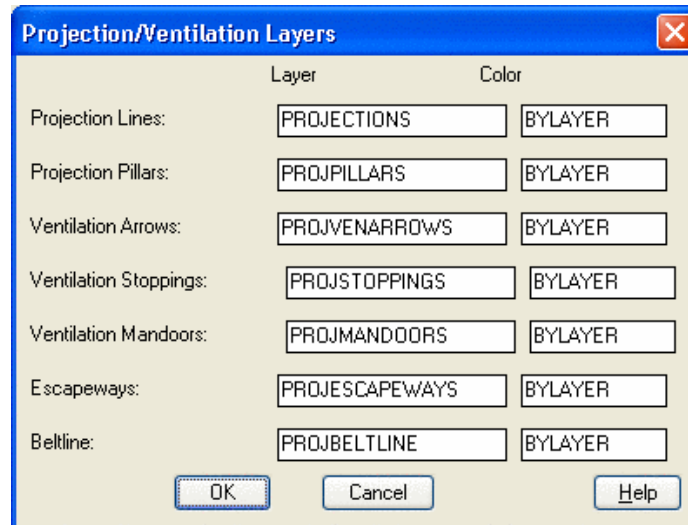
Second: Indicates the entry number for the secondary escapeway. The block is stored as PVESCSEC.dwg. Valid input would be a whole number with no decimal places.

Offset: Indicates the distance to measure back from the row of ventilation symbols for the placement location of the escapeway symbol. Valid input would be a whole number with no decimal places.

The blocks inserted can be modified, and are stored in the SCAD\SUP subdirectory.

Layer Control:

Projections/Ventilation always generates the different sections of its geometry (projection lines, pillars, ventilation arrows, stoppings, manddoors, escapeway, and beltline) on separate layers. These layers can be easily changed within the layer dialog and are automatically saved when the specifications are saved. The layers used are self-explanatory and are shown below.



LAYER CONTROL SUB-DIALOG

To access the layer control dialog shown, press the layer button on the main dialog. After making changes to the layers, choose the OK button. If you wish to reject the changes made to the layer names, simply press the Cancel button.

Specification Save:

Projections/Ventilation can store all specifications previously discussed to a file for later recall. To save a set of specifications, choose the Save button from the main dialog. When pressed, a standard file dialog will appear (defaulting to drawing name) allowing you to save the *.PVS file. PVS stands for Projections/Ventilation specification file. When chosen, the default drawing filename will usually be the filename used for the PVS file. Accept this name or enter a new one, changing directories if appropriate. The normal scheme is to have one PVS file per drawing, having the same name as the drawing and stored in the same directory.

Specification Load:

Projections/Ventilation can re-load all specifications previously discussed to a file for later recall. To load a set of specifications, choose the Load button from the main dialog. When pressed, a standard file dialog will appear, allowing you to enter the *.PVS file. The default drawing filename will usually be the filename used for the PVS file. Accept this name or enter a new one, changing directories if appropriate.

Pulldown Menu Location: Works

Keyboard Command: panel3

Prerequisite: None

Rooms

This routine simply draws centerlines for the rooms. First pick two points to define the beginning side of the rooms. Then specify the room dimensions. Finally, enter the distance that the rooms extend from the edge and on which side of the edge to draw the rooms.

Prompts

Start pt. of rooms: *pick a point*

End pt. of rooms: *pick a point*

Distance between room entries: 40

Distance between room crosscuts: 40

Room Depth: 200

Rooms on <R>ight or [L]eft side: R

Pulldown Menu Location: Works

Keyboard Command: rooms

Prerequisite: None

Label Projection Distances

As its name suggests, this routine labels the distances of a segment in a projection. It applies to projections created by Basic Projections or Rooms.

Prompts

Belt entry [point to point mode] y/<n>: *press Enter*

Select line to label distance on: *pick a projection line*

Pulldown Menu Location: Works

Keyboard Command: labelpan

Prerequisite: None

Panel & Room Label Block

Panel Label Block and Room Label Block both draw labels with their corresponding dimensions.

60' x 60' PANEL

40' x 40' ROOMS ON ADVANCE

Prompts

For Panel Label Block:

Insertion point: *pick a point*

Rotation angle: *choose a rotation*

SECOND DIMENSION <60'>: *press Enter*

FIRST DIMENSION <60'>: *press Enter*

For Room Label Block:

Insertion point: *pick a point*

Rotation angle: *choose a rotation*

ON ADVANCE OR RETREAT <ADVANCE>: *press Enter*

SECOND DIMENSION <40'>: *press Enter*

FIRST DIMENSION <40'>: *press Enter*

Pulldown Menu Location: Works

Keyboard Command: labelrms

Prerequisite: None

Draw Outline

Draw Outline creates a polyline in the OUTLINE layer out of the points you pick. The only intent of this routine is to draw old mine works.

Prompts

[node on] First Point: *pick a point*

Next Point, U to Undo, C to Close: *pick a point*

Next Point, U to Undo, C to Close: *pick a point*

Next Point, U to Undo, C to Close: *C for close*

Pulldown Menu Location: Works

Keyboard Command: oline

Prerequisite: None

Draw Perimeter

Draw Perimeter creates a polyline in the PERIM layer out of the points you pick. These points should compose the perimeter of a mine section. It is necessary to end picking points, and by entering C in order to make a closed polyline. The polyline must be closed, or inaccurate tonnages may result. Before using this routine, the points to connect can be placed using one of the Mine Note Entry commands.

Prompts

[qui end,node on] First Point: *pick a point*

Next Point, U to Undo, C to Close: *pick a point*

Next Point, U to Undo, C to Close: *pick a point*

Next Point, U to Undo, C to Close: *C for close*

Pulldown Menu Location: Works

Keyboard Command: perim

Prerequisite: None

Draw Pillars

Draw Pillars creates a polyline in the PILLARS layer out of the points you pick. These points should compose the perimeter of a pillar. It is necessary to end picking points by entering C in order to make a closed polyline. Otherwise, inaccurate tonnages may result. Before using this routine, the points to connect can be placed using one of the Mine Note Entry commands. If, after drawing a series of pillars, you want to check to see that all of the pillars have been properly closed, use the Highlight Unclosed Polylines under the Works dropdown menu.

Prompts

[node on] First Point: *pick a point*

Next Point, U to Undo, C to Close: *pick a point*

Next Point, U to Undo, C to Close: *pick a point*

Next Point, U to Undo, C to Close: *pick a point*

Next Point, U to Undo, C to Close: *C to close*

Pulldown Menu Location: Works

Keyboard Command: pillars

Prerequisite: None

Highlight Unclosed Plines

As its name suggests, this function will temporarily highlight polylines that are not closed. The intention of this function is for checking that the pillars and perimeter polylines are closed, since their closure is necessary for calculating quantities.

The procedure is to simply select the polylines to check. Then, after highlighting the selected polylines that are unclosed, the routine will pause for user input and then unhighlight the polylines. Thus, no changes are made to the drawing.

A polyline is considered closed if the endpoints are the same or if the closed flag is set. The closed flag can be set when creating a polyline, with the pline command, by using 'C' to close at the end.

Prompts

Select the polylines to check.

Select objects: *select polylines*

The program then reports either:

Unclosed polylines are highlighted.

Close all or selected polylines (All/<Selected>)? *press Enter* Look for the highlighted polylines and select the ones to close.

or

All the selected polylines are closed.

Pulldown Menu Location: Works

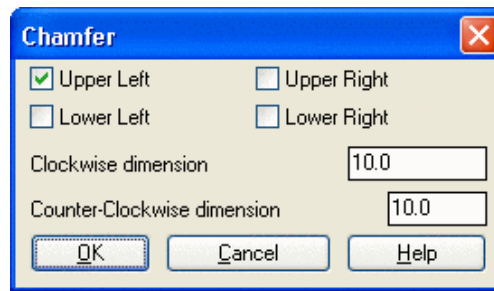
Keyboard Command: unclosed

Prerequisite: Polylines

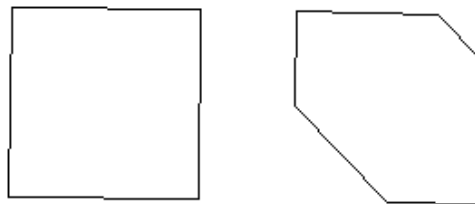
Chamfer Pillars

Chamfer Pillars is meant for redrawing existing pillars that are notched at a corner by mine equipment.

Chamfer Pillars is similar to AutoCAD's chamfer except that AutoCAD's works only on lines and Chamfer Pillars works only on polylines in the PILLARS layer. Chamfering basically creates a new edge that replaces a corner point and extends down each side a specified distance. In order to determine which corner to chamfer, this routine displays the dialog box shown below.



Chamfer Pillars dialog box



A 40' x 40' pillar before and after chamfering the upper right 10' and the lower left 20'

Prompts

Chamfer Dialog Box

Select the pillars the chamfer (polylines in the PILLARS layer).

Select objects: *Select the pillars*

Pulldown Menu Location: Works

Keyboard Command: pillar_chamfer

Prerequisite: Pillar polylines

AutoMine Connections

This command automates the connecting of dots for pillars and perimeters.

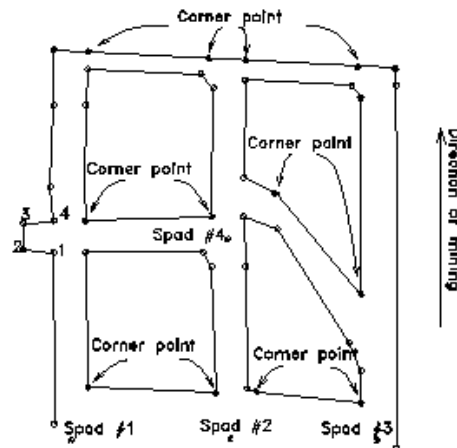
AutoMine Connections draws the pillars and perimeter of a mine based on point data in an offset file. This file can be created by one of the Mine Note Entry commands. There are two types of points in this file, corner points and regular points. Corner points, which appear as green dots, identify the first point across a cross cut for both the left and right side of a spad. Regular points are all other points, and are drawn as red circles. Corner points are critical to AutoMine. If a corner point is missing, AutoMine will become offset and the pillars will appear shifted. Each pillar must have exactly two corner points, one on the left and the other on the right. There can be no regular points between the two corner points on the pillar since AutoMine always directly connects the corner points together. For the perimeter, corner points are not important for the left and right faces. At the end face, however, corner points are required in order to identify the cross cut between the perimeter and the farthest pillar.

The order of entry of points in the offset file is significant in several ways. For one, the spads need to be in left to right order in the direction of mining. To do this, the spads can be entered in order, or the spads can be assigned sequential entry numbers. Another reason to assign entry numbers to the spads is to allow appending of the offset file. When reading appended files, AutoMine will group spads according to their entry number. The spads with the same entry number may be entered in any order since AutoMine sorts these spads by the distance along the direction of mining. For example, provided that spad 2 and spad 4 have the same entry number, then spad 2 and spad 4 offsets

may be entered either spad 2's offsets then 4's or spad 4's offsets then 2's. The offsets within each spad, however, are not sorted, but are drawn in order. Consider, for instance, the cut in the left face of the perimeter in the figure below. These points must be entered in the order 1 then 2 then 3 then 4.

AutoMine removes any pillar or perimeter that is already in the drawing at the same location.

After drawing the pillars and perimeter, AutoMine leaves a pick box hanging off the perimeter so that you may connect it with a previous perimeter.



Starting Point: N 5000.00 - E 5000.00

Entry no. 1

Azimuth: 0.4100

Distance on Centerline: 3.00

Left: 9.00

Distance on Centerline: 23.00

CRight: 9.00

Distance on Centerline: 61.00

Left: 9.00

Right: 9.00

Distance on Centerline: 62.00

Left: 19.00

Distance on Centerline: 77.00

Left: 17.00

Distance on Centerline: 78.00

Left: 9.00

CRight: 9.00

Distance on Centerline: 84.00

Left: 11.00

Distance on Centerline: 111.00

Left: 11.00

Right: 8.50

Distance on Centerline: 117.00

Right: 9.00

Distance on Centerline: 131.00

Left: 9.50

CRight: 9.50

Starting Point: N 5000.00 - E 5065.00

Entry no. 2

Azimuth: 0.4100

Distance on Centerline: 22.00

CLeft: 8.00
CRight: 10.00
Distance on Centerline: 24.00
Right: 8.00
Distance on Centerline: 56.00
Left: 10.00
Right: 9.00
Distance on Centerline: 61.00
Left: 12.00
Starting Point: N 5070.00 - E 5065.00
Entry no. 2
Azimuth: 00.4100
Distance on Centerline: 4.00
Right: 9.00
Distance on Centerline: -3.00
Right: 21.00
Distance on Centerline: 8.00
CLeft: 11.00
Distance on Centerline: 20.00
CRight: 16.00
Distance on Centerline: 24.00
Right: 9.00
Distance on Centerline: 43.00
Left: 9.00
Distance on Centerline: 49.00
Left: 12.00
Right: 9.00
Distance on Centerline: 62.00
CLeft: 12.00
CRight: 9.00
Starting Point: N 5000.00 - E 5130.00
Entry no. 3
Azimuth: 00.4100
Distance on Centerline: 4.00
Right: 10.50
Distance on Centerline: 19.00
CLeft: 10.50
Distance on Centerline: 24.00
Left: 8.50
Distance on Centerline: 36.00
Left: 15.00
Distance on Centerline: 54.00
CLeft: 12.00
Distance on Centerline: 108.00
Left: 9.00
Distance on Centerline: 114.00
Left: 13.00
Right: 9.00
Distance on Centerline: 129.00
CLeft: 9.00
Right: 9.00

Prompts

File Selection Dialog Box

Select a mine note file. This file must contain points in the format described above.

Next Point, U to Undo, P for Polyline, C to Close: C

Pulldown Menu Location: Works

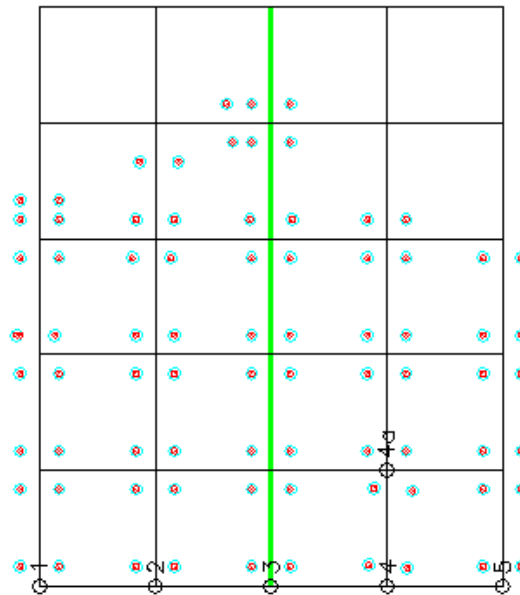
Keyboard Command: automine

Prerequisite: An offset file created by one of the mine note routines

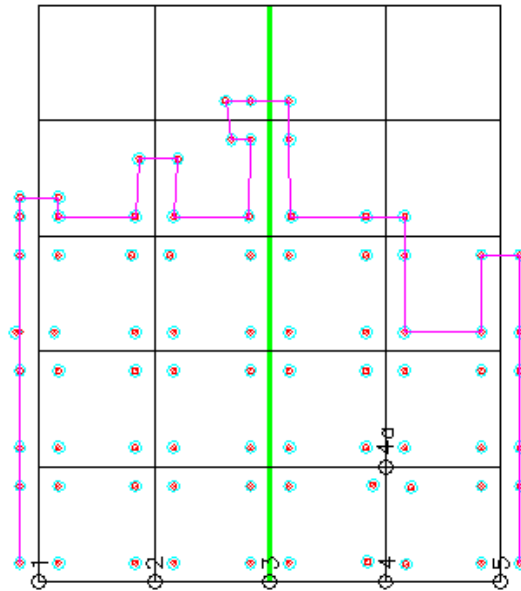
Auto-Connect Pillars

This routine automatically draws pillar polylines by connecting the selected offset points. The points must be offset points created by one of the Mine Note routines such as Mine Note Auto Left/Right. In addition to the points, polylines must be drawn through each crosscut. The routine works by connecting the points such that no connection crosses a crosscut polyline. The crosscut polylines can be drawn individually with the Draw Polyline command or created as projection polylines with the Advanced Projection routine.

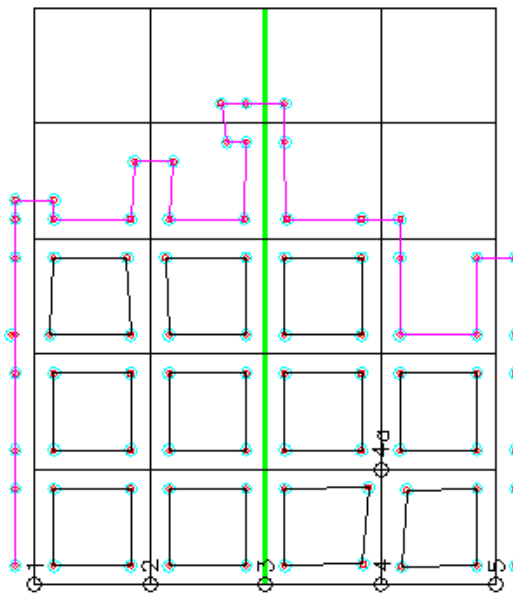
To use the routine see the layouts below. The first layout is created using Mine Note Auto Left/Right to locate the offset points. The centerlines were laid out using Advanced projections.



It is a good idea to manually connect the outer offset points manually and let this routine connect the internal points. Using the Polyline command and with the Osnap setting on Nodes, connect the perimeter points and create them in the PERIM layer as shown:



Next, connect the internal points using the Auto-Connect Pillars command. Select the points to be connected and the center lines that separate them. See the results here:



Prompts

Select pillar points and projection lines.

Select objects: *select the points and lines*

Pulldown Menu Location: Works

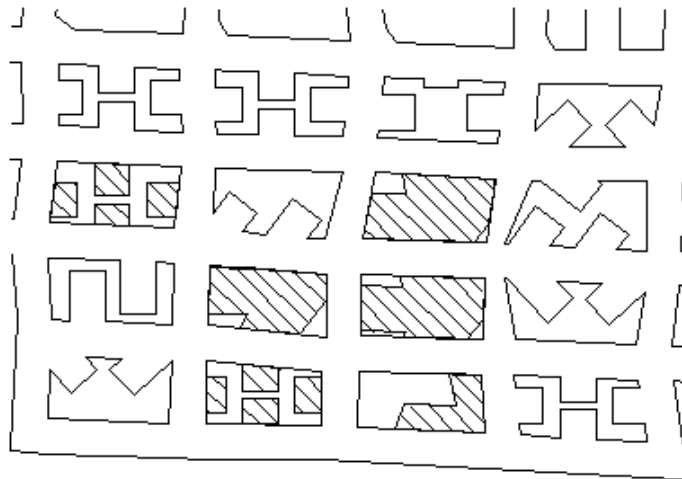
Keyboard Command: autopillar

Prerequisite: Offset points and crosscut polylines

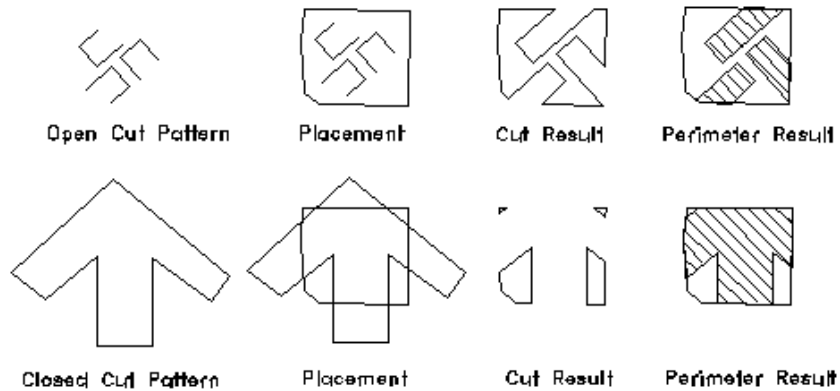
Pillar Cut

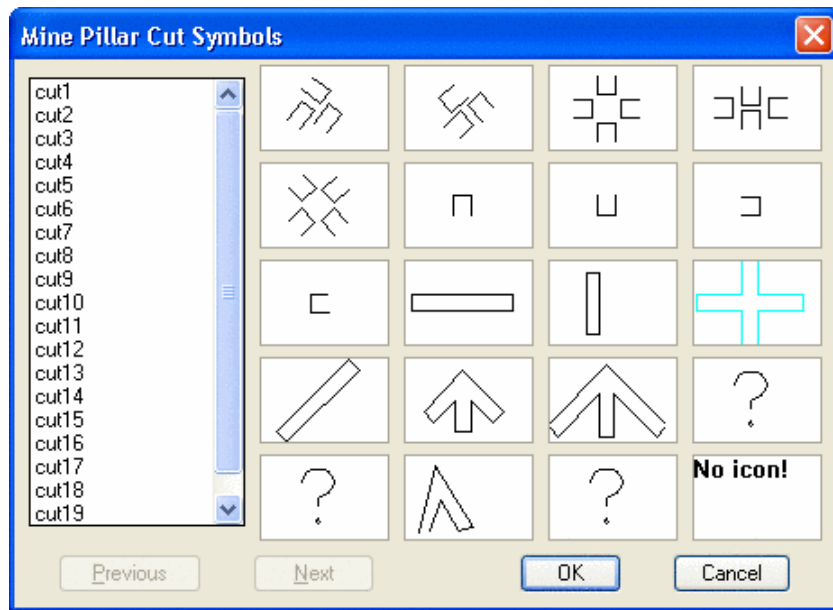
Pillar Cut serves two purposes relating to final cuts that are made into pillars. It automates the process of drawing cut pillars, and it creates new pillars and perimeters for use in calculating the tonnage removed from the cuts.

The command is run by first choosing a cut pattern, then selecting the pillars to cut, and finally placing the pattern inside a pillar. The pillars must be closed polylines in the PILLARS layer. The cut pattern can contain either open or closed polylines. When an open pattern symbol is placed in a pillar, Mine Pillar Cut calculates the intersections between the pillar and the end segments of the symbol, and fits the pattern into the pillar. With a closed symbol, the cut pattern is clipped against the pillar. There are two effects this placement can have. One is to recreate the pillar with the cuts taken out. The other creates perimeter polylines inside the pillar in the spaces that are cut out.



Example of Mine Pillar Cuts





Open cut patterns will not work if they divide a pillar into two or more separate pillars. Use a closed cut pattern if the pillar will be cut through.

To create custom cut patterns, create polylines in the desired pattern either as closed polylines or open polylines whose end segments will intersect the pillar when extended. Make sure that the polylines are the correct size to fit with a pillar. Then start Pillar Cut as usual and select the User-Defined symbol. It will ask you to select an insertion point, and pick a place in the table to store the new symbol. The next time Mine Pillar Cut is called, the new pattern should be in the table.

Prompts

Mine Pillar Cut Symbols Dialog

Select a symbol to cut with or select User-Defined to begin the procedure to define your own symbol.

Enter the azimuth for the cuts <0.0>: *press Enter* This allows the cuts to be rotated when placed.

Cut the pillar or create new perimeter? (<Cut>/Perim) *press Enter a string* If Cut is selected, the pillar is re-drawn with the cut hacked out. If Perim is selected, perimeter polylines are created inside the pillar in the spaces that are cut out. The removed portion can be hatched with PERIM.

Select mine pillars (polylines in the PILLARS layer).

Select objects: *select polylines* This creates the set of pillars that may be cut. These pillars must be closed polylines in the PILLARS layer.

Pick a point for the symbol: *pick a point* This point must be inside one of the selected mine pillars.

Pulldown Menu Location: Works

Keyboard Command: cutsym

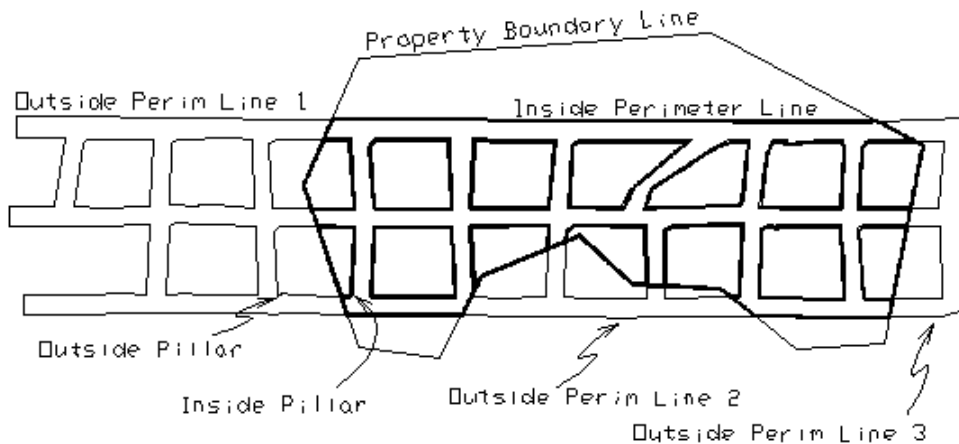
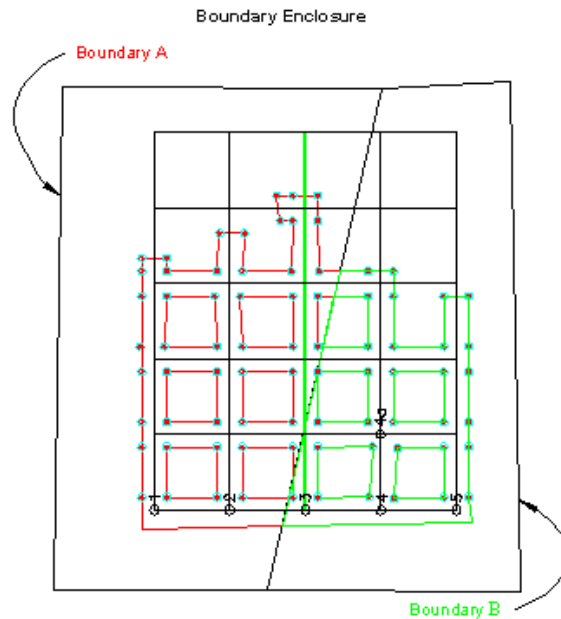
Prerequisite: Pillars to cut and drawn as polylines in the PILLARS layer

Boundary Enclosure

This routine is intended to be used in mines that include different leases or property ownerships. If the property is bounded by a polyline, Boundary Enclosure can be called to divide the mine into separate sets of pillars and perimeters. Then other routines such as Quantities by Average Method can use these sets to determine the exact tonnage quantities for that property.

Boundary Enclosure separates the pillars and perimeter of a mine into two sets based on a boundary line. One set consists of pillars and a perimeter from inside the boundary line and the other set consists of pillars and perimeter from outside the boundary line. If a pillar or perimeter crosses the boundary line, it is divided into two or more new polylines. In some cases, part of the new polyline may follow the boundary line. The boundary line must be a

closed polyline. The one restriction on the boundary line is that it cannot create a disjoint inside set of pillars and perimeter. Disjoint outside sets may be created.



Prompts

Enter a color (1-7) <No change>: 2 The pillars and perimeter inside the boundary will change to this color.

Select a boundary polyline: *pick a polyline* If the point where the polyline is picked crosses additional polylines, all polylines will be highlighted. Pick another point on the desired polyline that distinguishes it from the others.

Select mine pillars and perimeter.

Select objects: *pick the selection set of pillars and perimeter* These pillars and perimeter must be closed polylines in the PILLARS or PERIM layer.

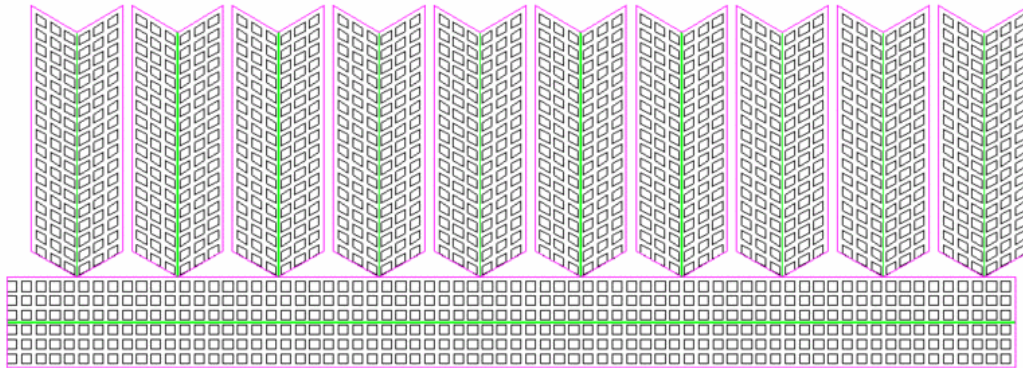
Pulldown Menu Location: Works

Keyboard Command: bound

Prerequisite: Pillars drawn as polylines in the PILLARS layer and perimeter drawn as polylines in the PERIM layer

Draw 3D Mine Model

This command takes the 2D series of linework and converts it to a 3D underground representation of the room and pillar mine. It uses the pillars and perimeter polylines and extrudes them to the thickness of the seam that is being mined.



The above starting linework for 3D conversion

This command will extrude the linework to either the thickness from a flat base, or to a roof and floor defined by either fixed elevations, coal sections, or elevation grid files as set by the **Mine Height Method** in the dialog. The input layers are defined so that the program knows what layers to look for. For the Draw 3D Faces option, the Output Layers are defined for the new 3D faces that are to be drawn in the assigned layers. The Output Solid Model option creates an MDL file.

Draw 3D Underground Mine

Mine Height Method

Thickness Coal Sections Elevations

Thickness

Thickness: 20 Select

Elevation

Roof Elevation: 1020 Select

Floor Elevation: 1000 Select

Input Layers

Perimeter Layers: PERIM ...

Pillar Layers: PILLARS ...

Output Layers

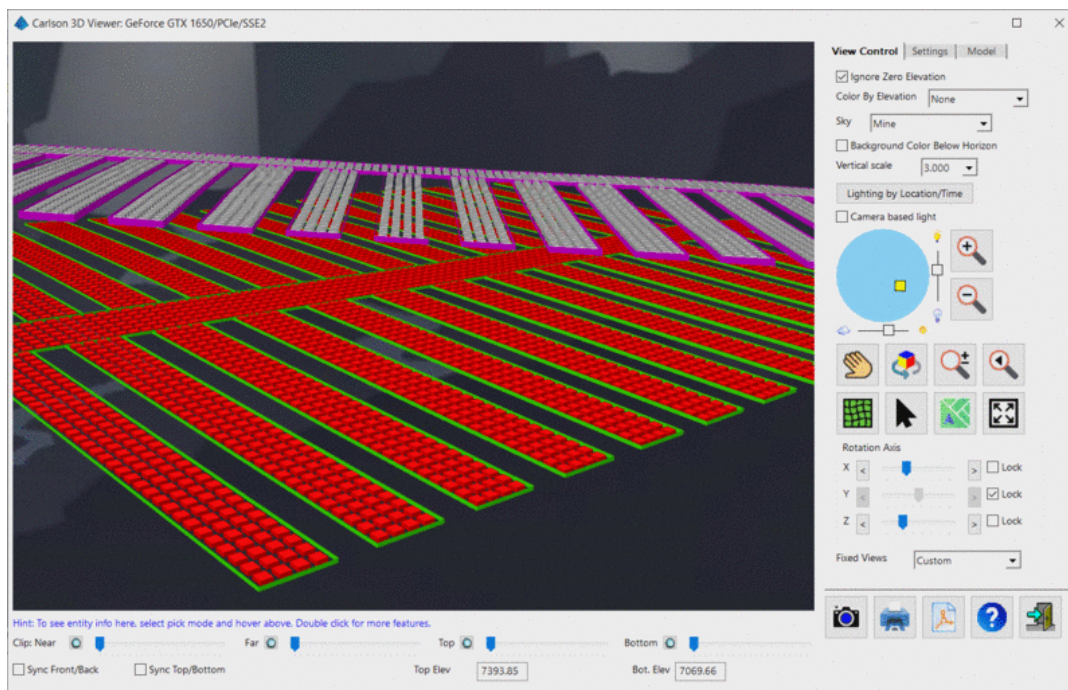
Perimeter Layer: 3D_PERIM ...

Pillar Layer: 3D_PILLARS ...

Draw 3D Faces Output Solid Model

OK Cancel Help

Once the 3D faces have been drawn, they can be viewed in the 3D Viewer Window under the View menu. In the Settings tab, the Shading Mode can be set to Both so that both the top and bottom of the faces will be rendered.



3D view of the underground mineplan

Prompts

Select Pillar and Perimeter entities to be drawn in 3D: *select the mineplan polylines*

Select objects: *Specify opposite corner: 2 found*

Inserting points...

Processed: 4 Perimeters and 1217 Pillars

Pulldown Menu Location: Underground Module, Works

Keyboard Command: ug3d

Prerequisite: Pillars and Perimeter polylines, elevation or thickness grids.

Configure Section Info

This command configures the names, abbreviations, and density of the various strata that appear in your underground mine. Also, the appearance of the section is specified by this routine. This information is needed for placing sections on the drawing which in turn are needed for Quantities by Average and Grid Method. If you reconfigure section info of an existing coal section, the sample points placed using the previous section info will become invalid and must be replaced.

The strata of a section can be configured as either individual or composite. With the individual configuration, each strata has its own name and density. With composites, each strata is still named separately but they also are divided into groups that have a group name and density. The principle advantage to composites is that it allows you to enter and list out each strata height and then combine the strata into their corresponding composite category when generating tonnage quantities.

This command creates a block drawing for the coal section with block attributes for each strata. The default folder for this .dwg file is your current SUP support folder which works fine for an individual user of the section info. To share the coal section with others, both the section .sc file and block .dwg file should be stored on a shared network folder. The location of the block .dwg file is stored inside the .sc file.

If you have a drawing with existing coal sections but don't still have the associated .SC file, then you can use this

command to recreate the .SC file. Near the beginning of the command prompting, answer Yes to the *Select existing coal section for strata names* prompt. Then the program will prompt to pick an existing coal section and will read the strata names from this section. The program still needs to prompt for the full strata names, densities and some format options.

Warning: After Configuring Section Info be sure to save your drawing. This will get AutoCAD to update the section Block information. Otherwise there can be a conflict between the version of the Block settings and the *.SC file which Carlson creates during Configure Section Info.

Prompts

For an Individual Section:

Enter Coal Section Configuration file name <Drawing_name.SC>: *SECTION1* (Or the name of your choice.)

Mine Name: Enter a name.

Select existing coal section for strata names [Yes/<No>]? *press Enter*

Enter Abbreviated Strata Name/<ENTER> to End: *C* This name is drawn next to the corresponding height when a section is located in the drawing.

Enter The Full Strata Name <C>: *Coal* This name is used in quantity reports.

Enter Abbreviated Strata Name/<ENTER> to End: *B*

Enter The Full Strata Name <C>: *Bone*

Enter Abbreviated Strata Name/<ENTER> to End: *R*

Enter The Full Strata Name <C>: *Rock*

Enter Abbreviated Strata Name/<ENTER> to End: *press Enter*

Enter Individual densities or Composite densities (I/C) <I>: *press Enter*

Average wt. of C (Coal) [lbs/ft³]: *80*

Average wt. of B (Bone) [lbs/ft³]: *150*

Average wt. of R (Rock) [lbs/ft³]: *150*

Circle the Coal Section (y/<n>)? *Y* This specifies whether a circle is drawn around the section when it is placed in the drawing.

Plot the Numeric Value Only (y/<n>)? *press Enter*

Text Size <6.0>: *press Enter*

Prompt for entry width (Yes/<No>)? *press Enter* The entry width value is used in Quantities by Centerlines.

For a composite section:

Enter Coal Section Configuration file name <Drawing_name.SC>: *SECTION2* (Or the name of your choice.)

Mine Name: Enter a name.

Select existing coal section for strata names [Yes/<No>]? *press Enter*

Enter Abbreviated Strata Name/<ENTER> to End: *TC*

Enter The Full Strata Name <C>: *Top Coal*

Enter Abbreviated Strata Name/<ENTER> to End: *TR*

Enter The Full Strata Name <C>: *Top Rock*

Enter Abbreviated Strata Name/<ENTER> to End: *BC*

Enter The Full Strata Name <C>: *Bottom Coal*

Enter Abbreviated Strata Name/<ENTER> to End: *BR*

Enter The Full Strata Name <C>: *Bottom Rock*

Enter Abbreviated Strata Name/<ENTER> to End: *press Enter*

Enter Individual densities or Composite densities (I/C) <I>: *C*

Coal Section Block To Write Specify dwg file name

Define the Composite Categories:

Enter Composite Category/<Enter> to END: *Coal*

Average wt. of COAL [lbs/ft³]: *80*

Enter Composite Category/<Enter> to END: *Rock*

Average wt. of ROCK [lbs/ft³]: 150

Enter Composite Category/<Enter> to END: *press Enter*

Assign the strata to a composite category:

Enter Composite Category for Top Coal TC (COAL ROCK): *coal*

Enter Composite Category for Top Rock TR (COAL ROCK): *rock*

Enter Composite Category for Bottom Coal BC (COAL ROCK): *coal*

Enter Composite Category for Bottom Rock TC (COAL ROCK): *rock*

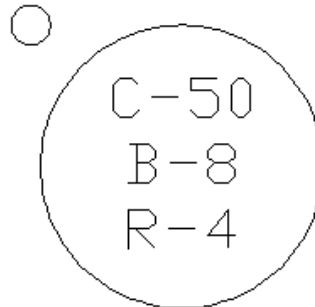
Circle the Coal Section (y/<n>)? *press Enter*

Plot the Numeric Value Only (y/<n>)? *press Enter*

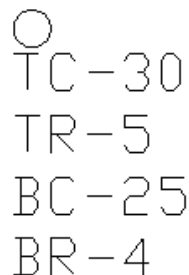
Text Size <6.0>: *press Enter*

Prompt for entry width (Yes/<No>)? *press Enter* The entry width value is used in Quantities by Centerlines.

Coal Section Block To Write Specify dwg file name



Sample Coal Sections



Sample Coal Sections

Pulldown Menu Location: Works

Keyboard Command: csconfig

Prerequisite: None

Place Coal Sections

This routine places a block circle and text of measured section information on the drawing. It serves two purposes: to display the sections for plotting, and to calculate tonnage. Carlson reads the location and values of the section from the block circle and not from the text.

For blank thickness entries, the program can either assume a zero thickness or ignore the section point for that strata. Which method to use is defined in Configure .

Prompts

Coal Section Configuration File Dialog

Pick sample point for coal section: *pick a point* This places a block circle.

Pick Start Point: *pick a point* This is where the text begins.

Pick Alignment Point: *pick a point*

Now enter your measurements.

How many Inches of Coal C: *51* Which quantities it asks for here depends of the configuration file.

How many Inches of Rock R: *9*

Pulldown Menu Location: Works

Keyboard Command: pcs

Prerequisite: A coal section configuration file

Edit Coal Sections

This routine edits the measured section information already placed on the drawing. To edit this information, pick the section block circle and the text. Then a dialog box will appear which allows you to change the current amounts of that section.

Pulldown Menu Location: Works

Keyboard Command: ecs

Prerequisite: A coal section placed in the drawing

Import Coal Sections From Text File

This command creates coal sections by reading data from a text file. The format of the text file must be northing, easting, thickness and strata name separated by commas. This is the format that the Export Coal Sections To Text File command creates.

Pulldown Menu Location: Works > Coal Section Utilities

Keyboard Command: importcs

Prerequisite: Text file to import and Coal Section definition file (.sc)

Export Coal Sections To Text File

This command creates a text file with the coal section data from the selected coal sections. The program first prompts for the text file name to write. Then the Coal Section Configuration file (.sc) is set. The last step is to select the coal section blocks from the drawing. Then the text file is written.

The format of the text file is northing, easting, thickness, and strata name. The values are comma separated and there is one record per line.

Prompts

Coal Sections Text File To Export

Coal Sections Configuration File

Select coal sections.

Select objects: *pick coal sections*

Pulldown Menu Location: Works > Coal Section Utilities

Keyboard Command: exportcs

Prerequisite: Coal sections

Quantities by Avg Method

Quantities by Average Method is one of three commands that calculate tonnages based on coal cross section data. The other routines are Quantities by Grid Method and Quantities by Centerlines. The grid method is more accurate but the average method is faster and requires fewer coal section sample points. The centerline method uses entry row centerlines instead of pillar and perimeter polylines.

Before executing this command, there must be sample coal section points and pillar and perimeter polylines in the PILLARS and PERIM layers. The pillars and perimeter polylines can be made with the Draw Pillars, Draw Perimeter, or AutoMine Connection commands, and sample points are placed by the Place Coal Sections command.

The quantities can also be reported by property owner. The property areas are defined by closed polylines with an attached property name. There are routines in the Boundary menu of the Underground Mining module for assigning the property names to the polylines. Quantities by Average Method will prompt you to select property polylines. If you do not select any, then just the total quantities will be reported. Otherwise the quantities are divided between the properties and reported separately.

Quantities by Average Method generates detailed a report when it is done calculating. The format of the report method can be Standard text, Column text, or reported with the Report Formatter. Besides this detailed report, the command Report Tons and Acres gives coal tonnage reports based on data files that Quantities by Average Method updates. In order to have these data files updated, you must enter beginning and ending dates and an ownership name when prompted. If this information is entered, Quantities by Average Method will add the current tonnage to the data file for the specified ownership name. The data files for the mine and panel names will also be updated if they are specified. The first time a mine or panel data file is accessed, it will ask for the estimated coal reserves.

Quantities by Average Method

SC File: C:\Drawings\1\Miscellaneous\JULY99.sc

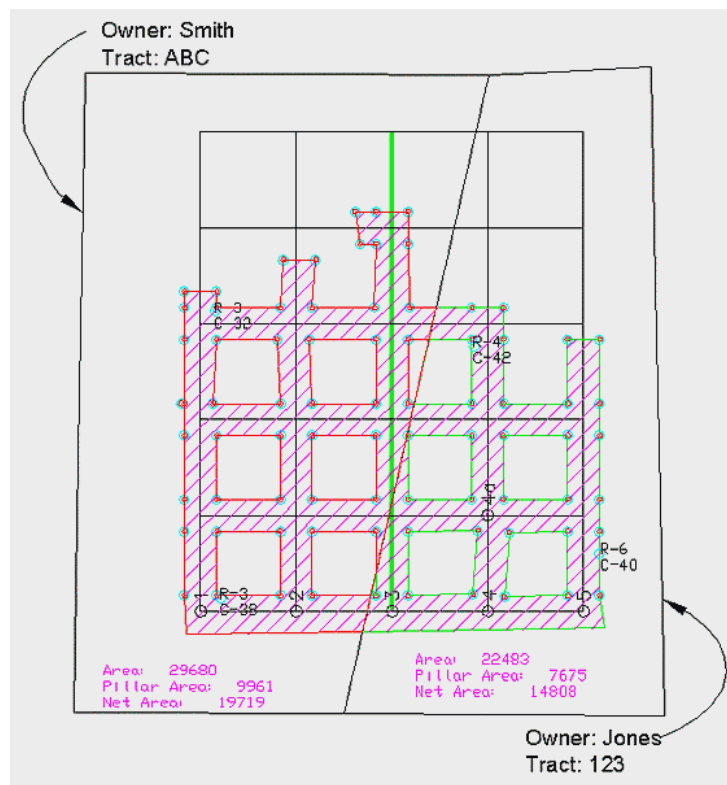
Beginning date of takeup (mm/dd/yyyy) 8/28/2007 Select

End date of takeup (mm/dd/yyyy) 12/28/2007 Select

Report Method

Standard Column Formatter

OK Cancel



Hatched drawing with the area summaries for each property boundary

Carlson Software Edit : C:\Documents and Settings\Grant Wenker\Application Data\Carlson Software\Carlson2008VR17.0\USER\scadrpr...

File Edit Settings

Open Save Print Exit Find Screen Hide

NUMERICAL AVERAGE COAL SECTION METHOD 8/28/2007 13:50

Individual Stratas Configuration

MINE: BIGRIVER PORTAL 2 MINED FROM 8/28/2007 TO 12/28/2007

AREA NO. 1 DESCRIPTION: ownership

SECTION/UNIT ID: CM MINE TYPE: Advance MINING METHOD: Shuttle Cars

GROSS AREA MINED (S.F.):	494925.00	DEPLETED ACRES:	11.362
AREA OF PILLARS (S.F.):	233185.00	ACRES OF PILLARS:	5.353
NET AREA MINED (S.F.):	261740.00	NET ACRES MINED:	6.009

AVERAGE ROCK THICKNESS	(INCHES): 8.73	(FEET): 0.73
AVERAGE COAL THICKNESS	(INCHES): 63.71	(FEET): 5.31
AVERAGE FIRECLAY THICKNESS	(INCHES): 6.55	(FEET): 0.55
TOTAL MINING HEIGHT	(INCHES): 78.98	(FEET): 6.58

AVERAGE ROCK WT. (LBS/CU. FT.):	145.00
AVERAGE COAL WT. (LBS/CU. FT.):	81.00
AVERAGE FIRECLAY WT. (LBS/CU. FT.):	125.00

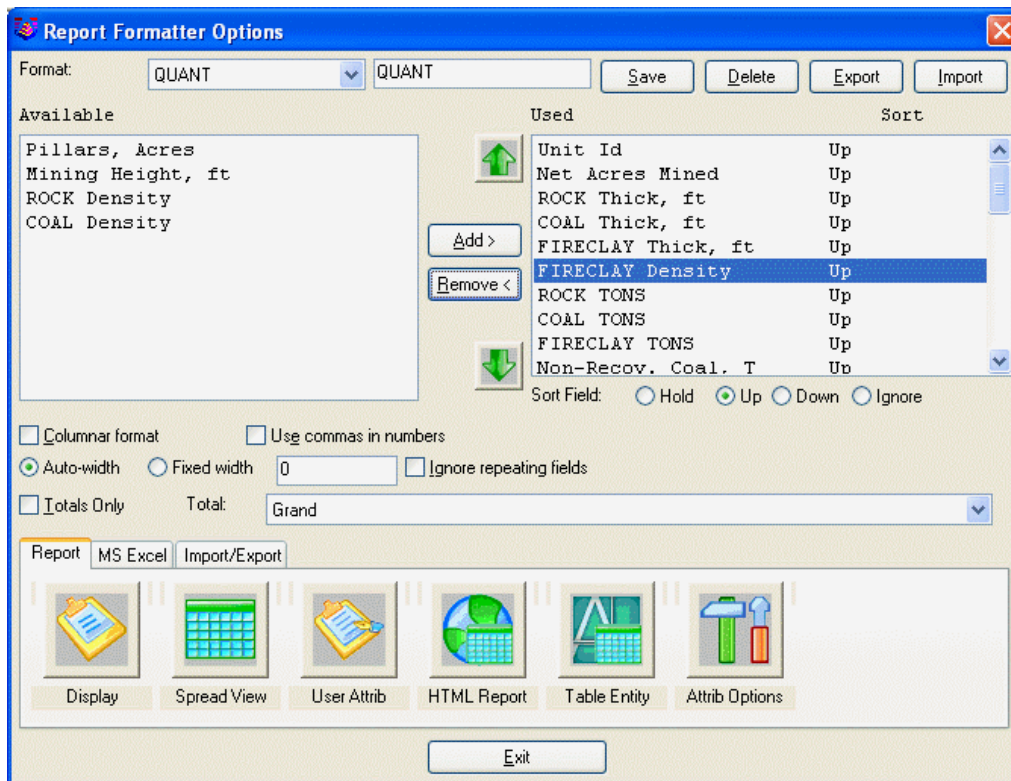
ROCK (TONS):	13800.83	COAL RECOVERY PERCENT:	100.00%
COAL (TONS):	56278.87	PERCENT COAL BY WGT.:	71.24%
FIRECLAY (TONS):	8922.95		
NON-RECOVERABLE Coal (TONS):	0.00		
TOTAL TONS :	79002.65		

COAL ACRE-FEET:	31.901
CAVITY ACRE-FEET:	39.548

Qualities:

CROSS WIDTH	20.0
CROSS SPACING	60.0
ENTRY WIDTH	20.0
ENTRY SPACING	60.0
FULL EXTRACT	0.0
ANGLE RIGHT	90.0
NUM_RIGHT	2.0
ANGLE LEFT	90.0
NUM_LEFT	2.0

Text Report Example



Report Formatter option

Prompts

Coal Section Configuration File Dialog

Select property polylines or press Enter for none: *select the property boundaries*

Coal recovery percent <100.00>: *press Enter*

Which type of selection? [<Standard>/Cuts]: *press Enter*

Select pillars, perimeters, and section sample points. Select these carefully so you only choose the pillars inside the perimeters. Section sample points can be selected outside the perimeter, if they contribute to a more accurate answer.

Pick location to draw results or Enter for none: This is where the Area/Pillar Area/Net Area will be posted for the first property boundary.

Pick Alignment Point: This sets the text direction.

Pick location to draw results or Enter for none: This is where the Area/Pillar Area/Net Area will be posted for the second property boundary.

Pick Alignment Point: *This sets the text direction.*

Another area [Yes/<No>]? *N*

Start point or Center/Middle/Right/?: This refers to the insertion point for the text report to be posted.

Height <6.00>: *10*

Rotation angle <90d0'0">: *press Enter for East to West (Horizontal) text*

Layer for text <REPORT_TXT>: *You can set any layer for the text report.*

Insert as MText or Text [<MText>/Text]?: *press Enter for MText*

Update coal tonnage files [<Yes>/No]? *N*

Pulldown Menu Location: Works

Keyboard Command: qavg

Prerequisite: Pillars and perimeter polylines and coal sections

Quantities by Grid Method

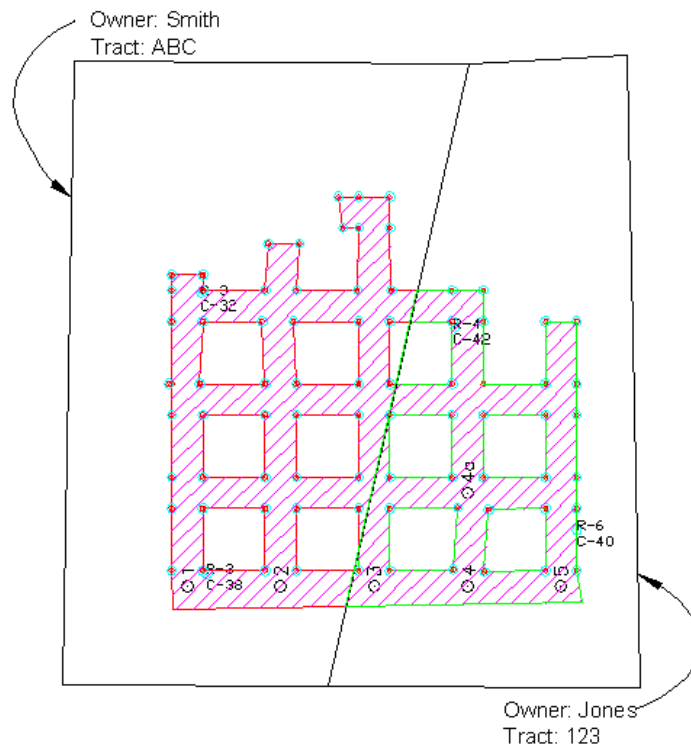
Quantities by Grid Method is one of two commands that calculate tonnages based on pillars, perimeter and coal cross section data. The other routine is Quantities by Average Method. The average method is faster and requires fewer coal section sample points, but the grid method is more accurate because it creates a 3D model of the strata which finds deviations the averaging misses. Either Triangulation or Inverse Distance methods can be used for the grid model.

Before executing this command, there must be at least three sample coal section points and pillar and perimeter polylines in the PILLARS and PERIM layers. Perimeters may only be used one at a time. The pillars and perimeter polylines can be made with the Draw Pillars, Draw Perimeter, or AutoMine Connection commands, and sample points are placed by the Place Coal Sections command. Also property owner polylines can be used as described in the Quantities by Average Method command.

Quantities by Grid Method generates a detailed report when it is done calculating. Besides this detailed report, Report Tons & Acres gives coal tonnage reports based on data files that Quantities by Grid Method updates. In order to have these data files updated, you must enter beginning and ending dates and an ownership name when prompted. If this information is entered, Quantities by Grid Method will add the current coal tonnage to the data file for the specified ownership name. The data files for the mine and panel names will also be updated if they are specified. The first time a mine or panel data file is accessed, it will ask for the estimated coal reserves. There are three options for reporting, Standard, Column and Formatter.

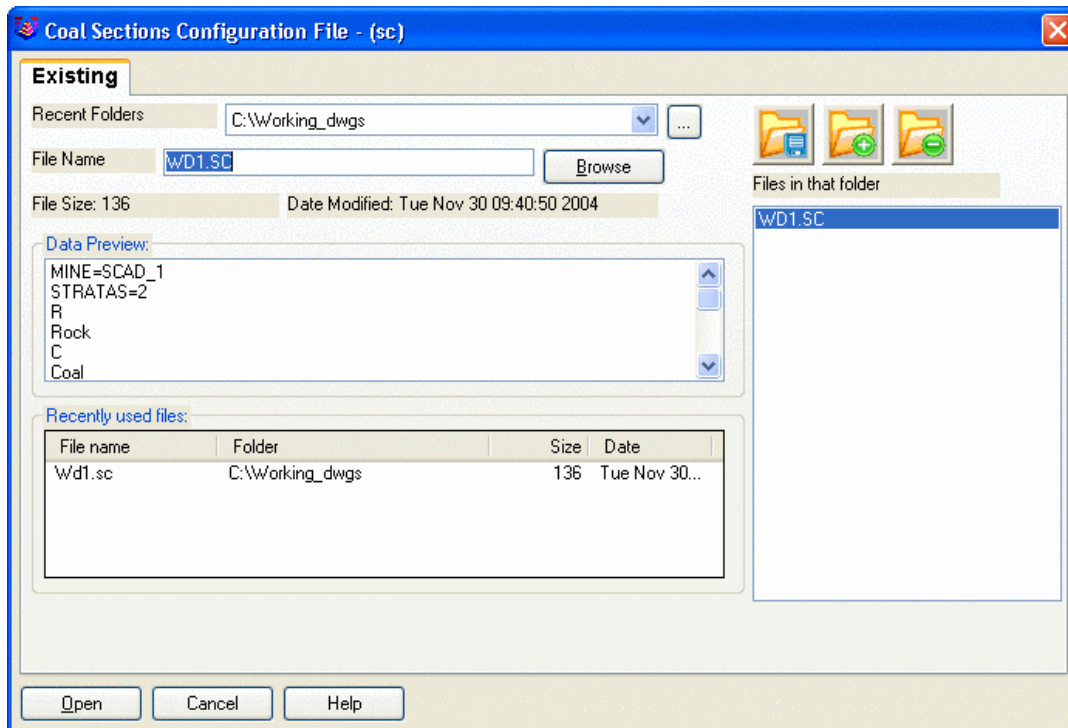
The screenshot shows a dialog box titled "Quantities by Grid Method". It contains the following fields and options:

- SC File: C:\sample\leaf.sc
- Grid Modeling Method: Triangulation, Inverse Distance
- Automatic Grid Size, Grid Cell Size: 10.0
- Beginning date of takeup (mm/dd/yyyy): 4/1/2012 (with a "Select" button)
- End date of takeup (mm/dd/yyyy): 4/30/2012 (with a "Select" button)
- Report Method: Standard, Column, Formatter
- Select Property Polyines
- Buttons: OK, Cancel



Prompts

Select the file that defines the coal section sample points.



Coal recovery percent <100.00>: *press Enter*

Select property polylines or press Enter for none: *select property boundaries only*

Select pillars, perimeters, & at least 3 coal section points Select these carefully to only include the ones for the calculation. If you select more than you need and the hatching does not look correct, undo and repeat.

Processing cells ...

Another area [Yes/<No>]? If you want to process more than one area and calculate them together select another area. When you have selected all of the areas the program prompts you for the location if the report.

Start point or Center/Middle/Right/?:

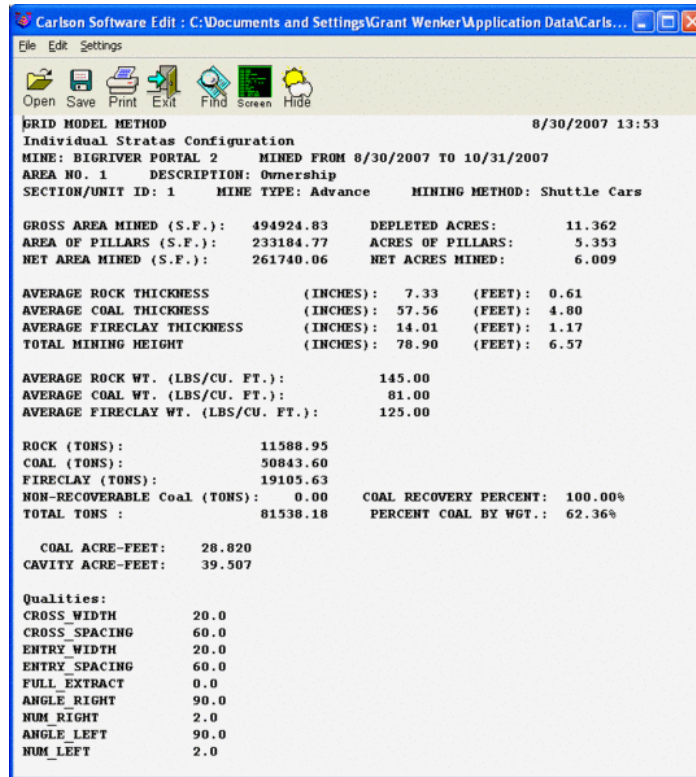
Height <6.00>: 10

Rotation angle <90d0'0">: *press Enter*

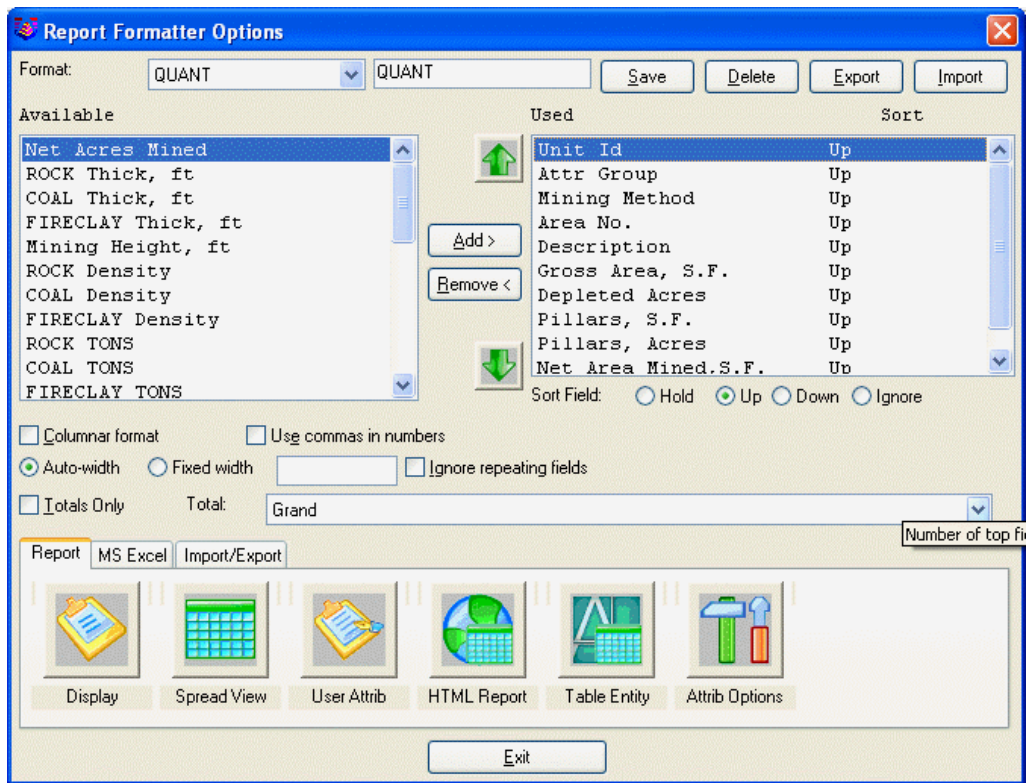
Layer for text <REPORT_TXT>: *press Enter* You can assign a new layer name if you choose.

Insert as MText or Text [<MText>/Text]?: *press Enter*

Update coal tonnage files [<Yes>/No]?: *N*



Report window



Report Formatter

Pulldown Menu Location: Works

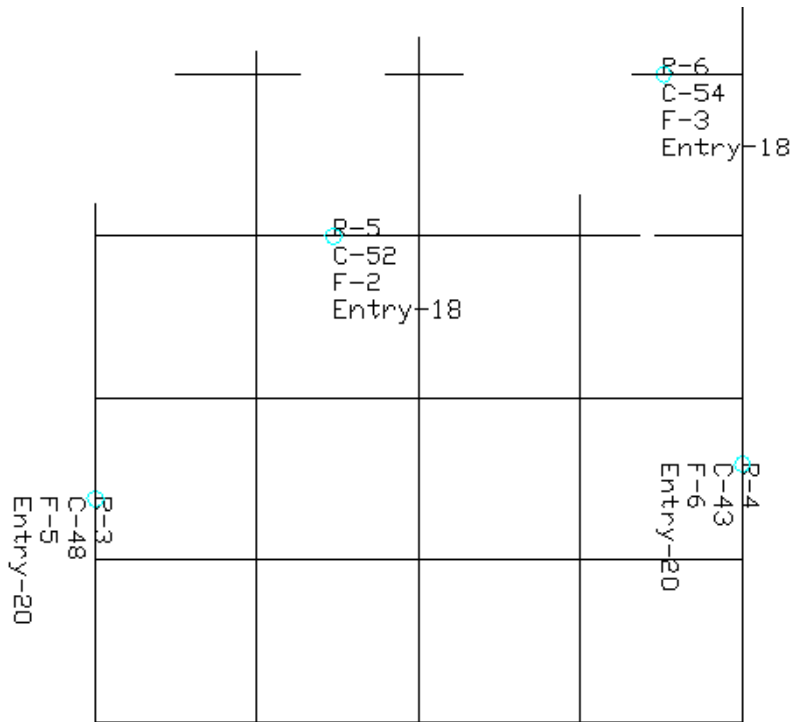
Keyboard Command: qgrid

Prerequisite: At least three coal sections and pillar and perimeter polylines

Quantities by Centerlines

This command calculates strata quantities based on coal sections and polylines along each cut. The coal sections contain the thickness for each strata and the entry width. When defining the coal section format in Coal Section Info, the Entry Width option must be activated.

Before running Quantities by Centerlines, polylines should be drawn along each cut. This routine sums the polyline lengths to find the linear feet of advance. Crossing cut polylines are handled by subtracting one entry width from the length for each crossing. Multiplying the linear feet of advance by the average entry width gives the mined area which is multiplied by the average strata thicknesses to obtain the strata volumes.



Prompts

Coal Section Configuration File

Select the file that defines the coal section sample points.

Beginning date of takeup [format mm-dd-yy] <3-1-05>: *press Enter*

Ending date of takeup [format mm-dd-yy] <4-1-05>: *press Enter*

Ownership/description <Smith(ABC)>: *press Enter*

Coal recovery percent <100.00>: *press Enter*

Select centerlines and section sample points. *select objects*

Another area [Yes/<No>]? *N*

Start point or Center/Middle/Right/?: *Left pick start point or input option*

Height <6.00>: *10*

Rotation angle <90d0'0''>: *press Enter*

Layer for text <REPORT_TXT>: *press Enter to accept default, or input another layer name for the report*

Insert as MText or Text [<MText>/Text]?: *press Enter to accept default, or input std for standard text*

Update coal tonnage files [<Yes>/No]? *Y to update or N to end function*

The report is shown here:

1/29/2005 13:34

NUMERICAL AVERAGE COAL SECTION METHOD
 Individual Stratas Configuration
 MINE: CARLSON NO. 1 MINED FROM 3-1-05 TO 4-1-05
 AREA NO. 1 DESCRIPTION: Smith(ABC)
 LINEAR FEET OF ADVANCE: 1785.44
 AVERAGE ENTRY WIDTH: 19.00
 NET AREA MINED (S.F.): 33923.40 NET ACRES MINED: 0.779

AVERAGE Rock THICKNESS (INCHES): 4.50 (FEET): 0.38
 AVERAGE Coal THICKNESS (INCHES): 49.25 (FEET): 4.10
 AVERAGE Floor THICKNESS (INCHES): 4.00 (FEET): 0.33
 TOTAL MINING HEIGHT (INCHES): 57.75 (FEET): 4.81

AVERAGE Rock WT. (LBS/CU. FT.): 160.00
 AVERAGE Coal WT. (LBS/CU. FT.): 80.00
 AVERAGE Floor WT. (LBS/CU. FT.): 165.00

Rock (TONS): 1017.70
 Coal (TONS): 5569.09
 Floor (TONS): 932.89
 NON-RECOVERABLE COAL (TONS): 0.00 COAL RECOVERY PERCENT: 100.00%
 TOTAL TONS : 7519.69 PERCENT COAL BY WGT.: 74.06%

COAL ACRE-FEET: 3.196
 CAVITY ACRE-FEET: 3.748

Pulldown Menu Location: Works
Keyboard Command: qentry
Prerequisite: Coal sections and polylines

Report Tons & Acres

When you run the take-up procedures Quantities by Average, Quantities by Grid, or Quantities by Centerline, at the end of the procedure after the routine has posted the take-up report you are prompted to update the Tonnage Report. The Tonnage Report gives you the option of keeping track of tons for an Owner, Mine, and each Panel for specific periods of time.

Mine Report produces coal tonnage reports for a specified mine, panel, or ownership.

NUMERICAL AVERAGE COAL SECTION METHOD

Individual Stratas Configuration

MINE: LEAF MINED FROM - 3-1-05 TO - 4-1-05

AREA NO. 1 DESCRIPTION:

LINEAR FEET OF ADVANCE: 2005.95

AVERAGE ENTRY WIDTH: 18.20

NET AREA MINED (S.F.): 36508.30 NET ACRES MINED: 0.838

AVERAGE Coal THICKNESS (INCHES): 59.00 (FEET): 4.92

AVERAGE Hard Rock THICKNESS (INCHES): 4.00 (FEET): 0.33

TOTAL MINING HEIGHT (INCHES): 63.00 (FEET): 5.25

AVERAGE Coal WT. (LBS/CU. FT.): 77.00

AVERAGE Hard Rock WT. (LBS/CU. FT.): 104.00

Coal (TONS): 6910.72

Hard Rock (TONS): 632.81

NON-RECOVERABLE COAL (TONS): 0.00 COAL RECOVERY PERCENT: 100.00%

TOTAL TONS : 7543.53 PERCENT COAL BY WGT.: 91.61%

COAL ACRE-FEET: 4.121

CAVITY ACRE-FEET: 4.40

Reports are created for a certain time period. If no dates are entered, a report is generated that includes all the known data. Besides reporting the tons mined for the period, Mine and Panel reports also include the total reserves and remaining reserves.

Coal tonnage data is made with the Quantities by Average Method or Quantities by Grid Method commands, and the data is stored in *.MIN, *.PAN, or *.OWN files in the data directory. These files are in a straightforward ASCII format and may be edited by the Inquiry Dropdown > Display Edit File command.

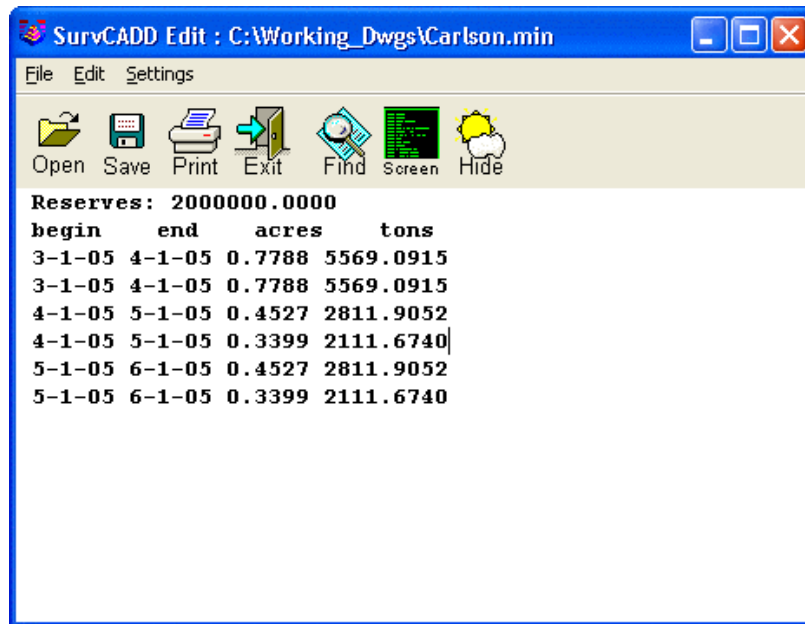
Prompts

Type of report (Mine, Panel, <Owner>): *Enter a name*

Report Source File selection dialog The available data files are displayed for selection.

Enter a beginning date (MM-DD-YY): *enter a date* If no date is entered, the earliest date in the data file is used.

Enter an ending date (MM-DD-YY): *enter a date* If no date is entered, the latest date in the data file is used. You can also select the files from the dialog box.



Sample source mine data file

Mine Report 1/29/2005 16:39

```

Mine: C:\Working_Dwgs\Carlson.min
Time period: 3-1-05 to 4-1-05
Tons mined: 11138.18
Acres mined: 1.56
Total reserves: 2000000.00
Remaining reserves: 1979014.66
    
```

Mine Report 1/29/2005 16:41

```

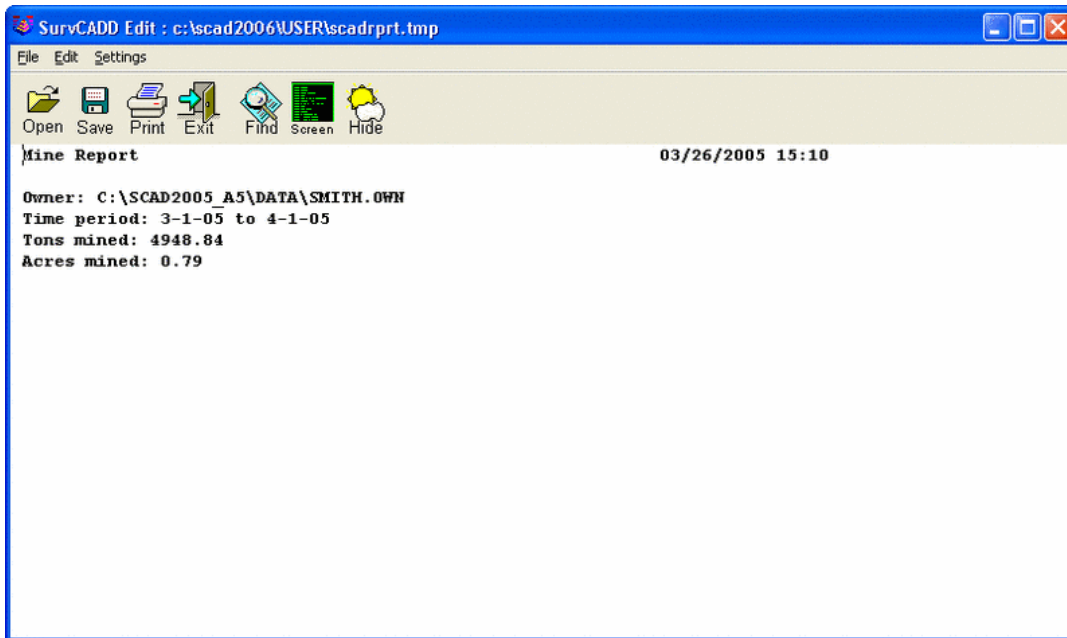
Mine: C:\Working_Dwgs\Carlson.min
Time period: 4-1-05 to 5-1-05
Tons mined: 4923.58
Acres mined: 0.79
Total reserves: 2000000.00
Remaining reserves: 1979014.66
    
```

Mine Report 1/29/2005 16:39

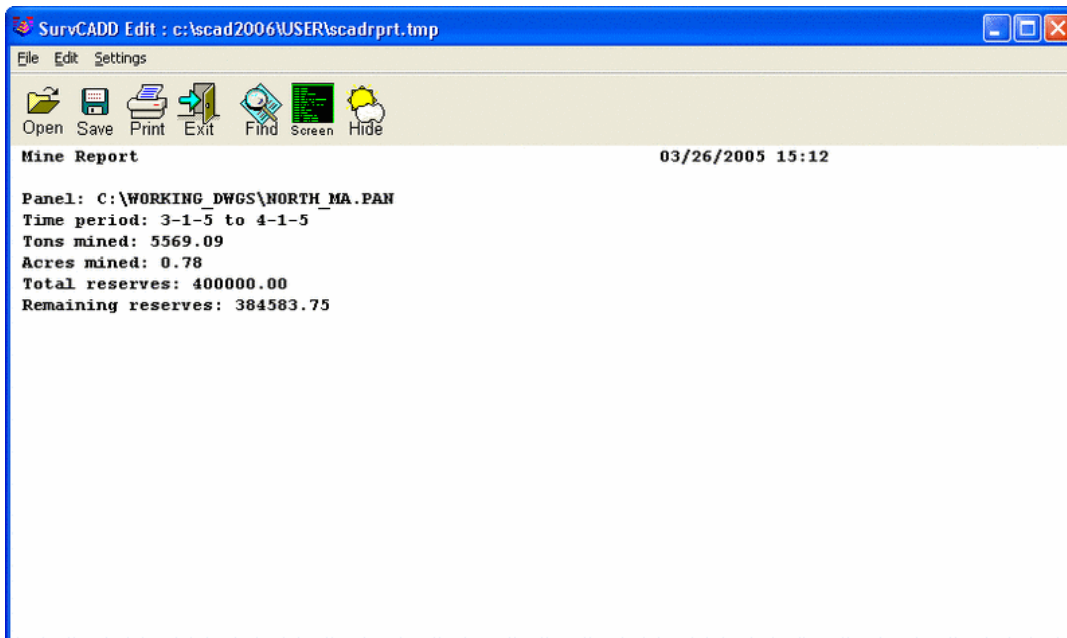
```

Mine: C:\Working_Dwgs\Carlson.min
Time period: 3-1-05 to 5-1-05
Tons mined: 16061.76
Acres mined: 2.35
Total reserves: 2000000.00
Remaining reserves: 1979014.66
    
```

Sample Mine Report



Sample Owner Report



Sample Panel Report

Output report to printer (Y/(<N>): enter *Y* or *N*

Besides outputting to the screen, the output can be sent to a printer in the PRN slot.

Do another report (Y/<N>): enter *Y* or *N*

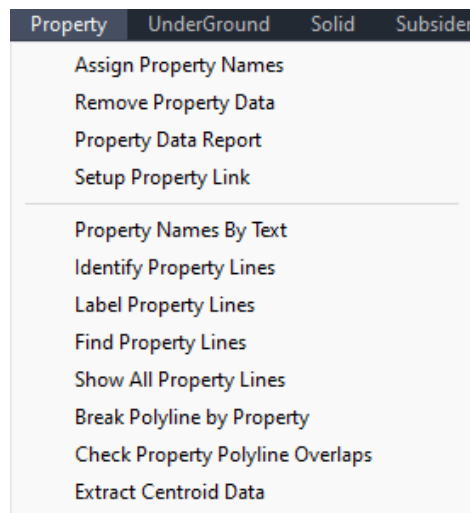
Pulldown Menu Location: Works

Keyboard Command: mreport

Prerequisite: Quantity report files

Property Menu

The Property menu has commands for managing property boundary polylines.



Assign Property Names

In order to evaluate the quantity and/or quality of ore mined on the particular property, Carlson employs the use of closed polylines representing the property boundary. This function is used for assignment and modification of the owner and property ID names to these polylines. The following commands support property lines:

Quantities by Average
Quantities by Grid
Surface Equipment Timing
Surface Production Timing
Underground Timing
Surface Mine Reserves
Reserve Classification

This command simply prompts to select the polyline and type in the owner and ID. Property polylines supersede the Boundary Enclosure command for the purposes of calculation, since the use of property polylines does not require actual subdivision of the mine plan into properties. Instead the subdivision is performed internally "on-the-fly" by the cutting of a given polyline (such as whole mine property or monthly production pit polyline) against an underlying set of property lines.

In its current implementation the use of property lines has the following limitations: property lines should be closed polylines and make sure that they do not overlap other property lines. If a portion of a pit or panel is outside of all property lines, then in the report, the owner will be reported as UNKNOWN.

Prompts

Select property polyline:

Property Owner Name<>: *Federal*

Property ID<>: *#J1267*

Pulldown Menu Location: Boundary

Keyboard Command: property_line

Remove Property Data

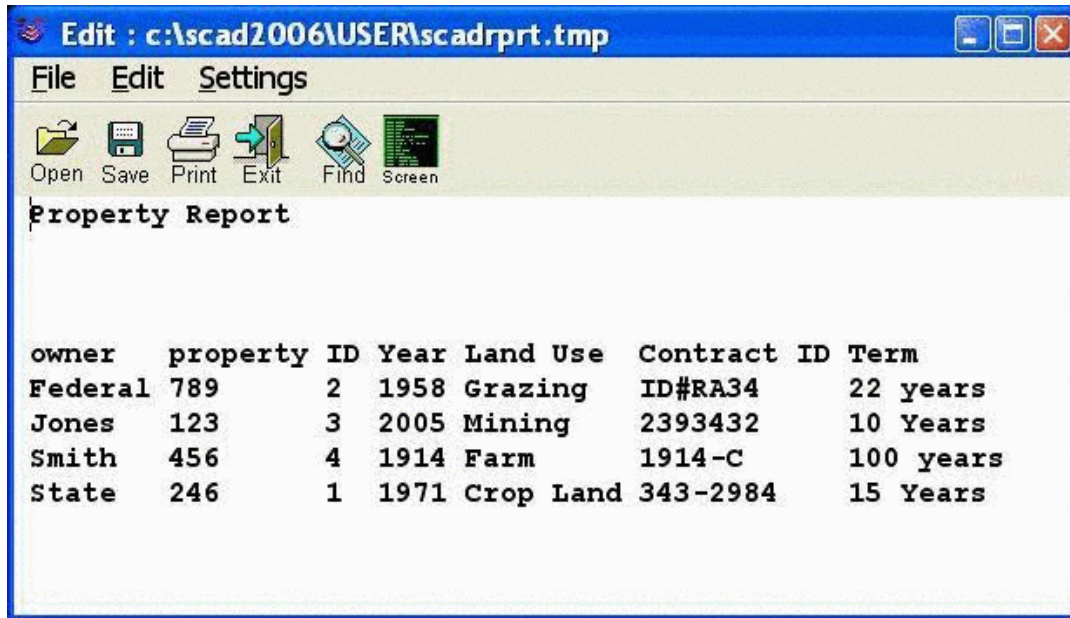
When property information attached to the polyline is no longer needed, it may be removed by using this command. Multiple polylines may be processed at once.

Pulldown Menu Location: Boundary

Keyboard Command: clearprop

Property Data Report

This command uses the report formatter to create a report showing all property data associated with the selected property lines. They must be linked to the MPD file where all of the property data is stored.

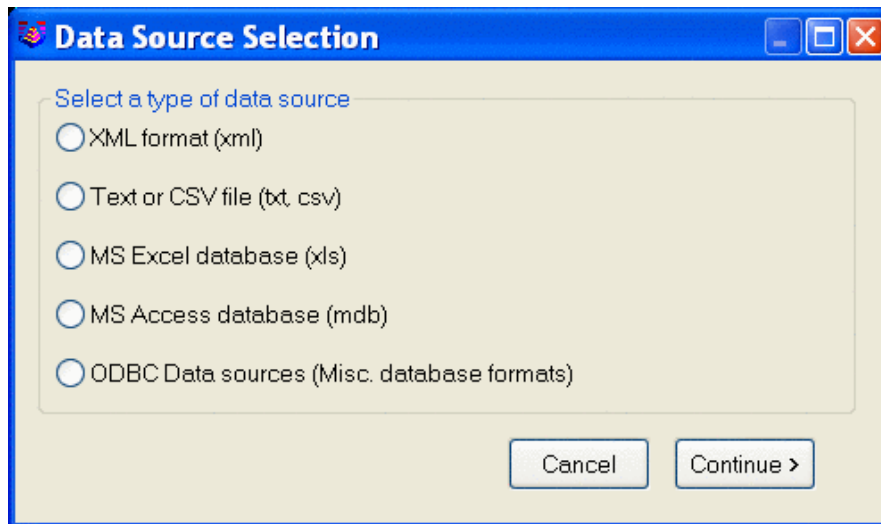


Pulldown Menu Location: Boundary, Property Line Tools

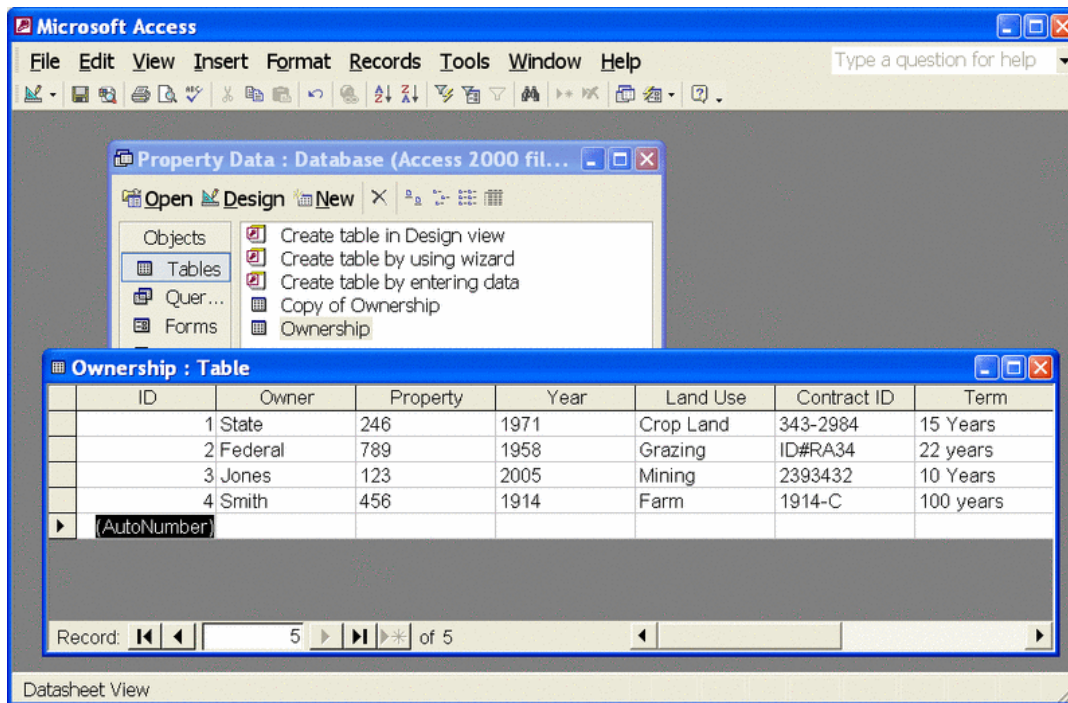
Keyboard Command: reportprops

Setup Property Link

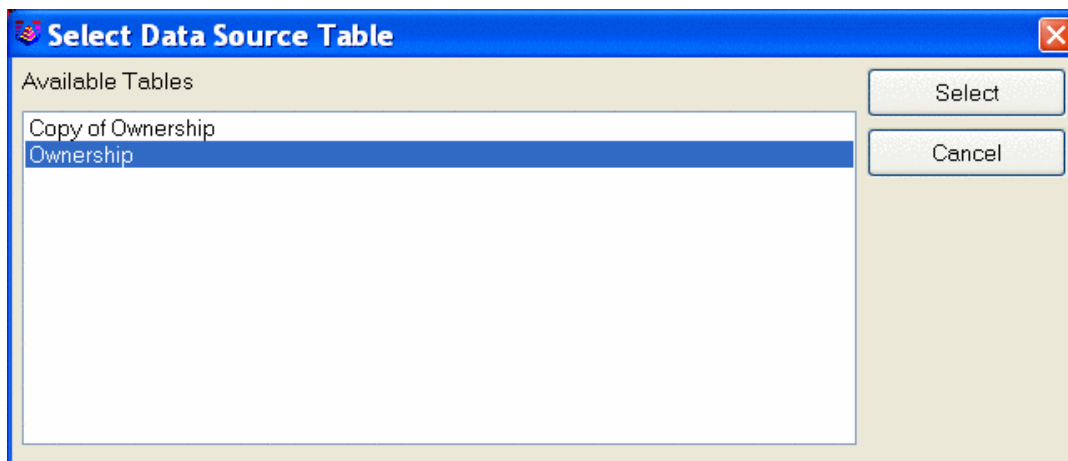
This command creates the link between the property lines in the drawing and the database. The data can come from an XML file, a TXT or CSV file, an XLS file, an Access MDB file, or miscellaneous ODBC data sources.



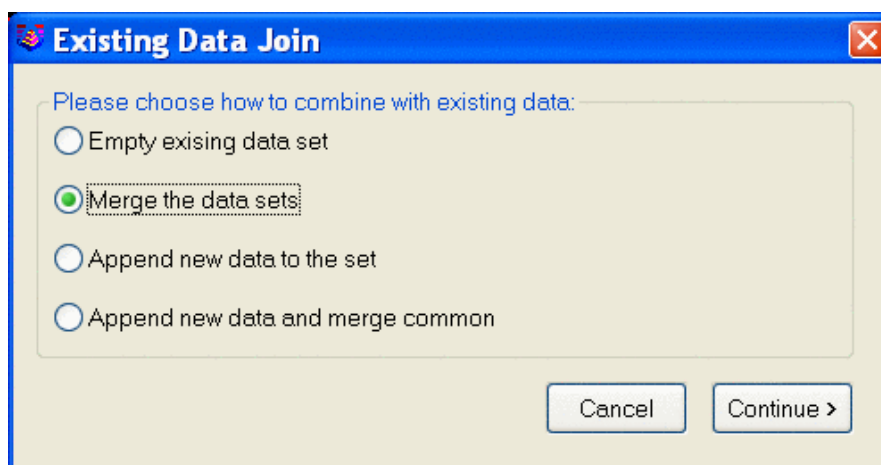
After choosing which format it is, select the Continue button.



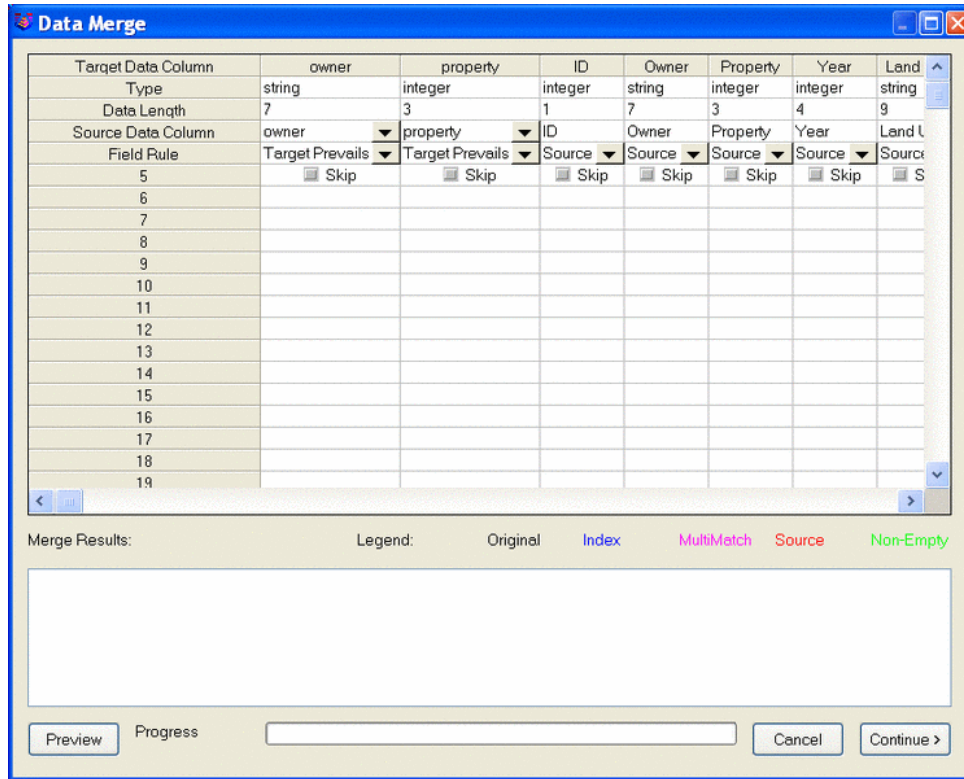
This Access database shows some data entered in for four different properties.



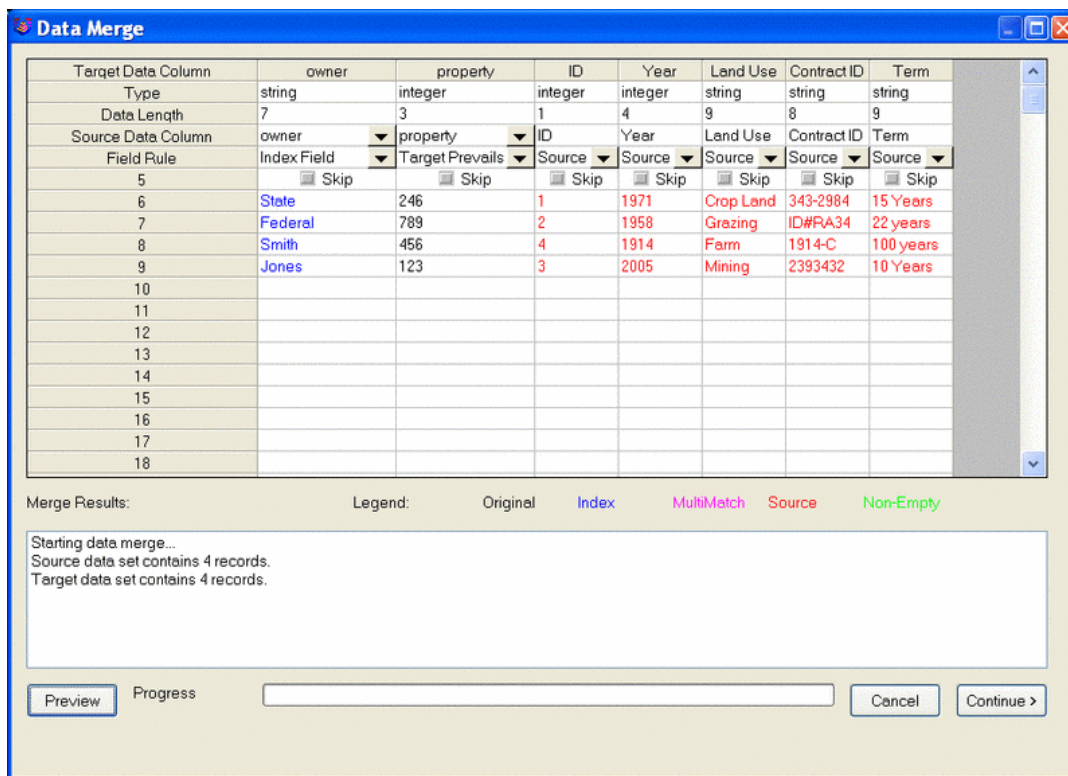
After choosing the source file, this selection window appears if it is from an Access database. Choose the table the data is stored in.



There are then four choices on how to join the data. The first is to empty the existing data set which would delete the property and ownership line data. The second is to merge the data, using both data sets. The third option is to append all to the end of the data, and finally, the last option is to append any new data, and merge the common data. The next Data Merge table displays all of the available fields. Notice in this example, there are two Owners and two Properties. In this case, the Merge option is the way to go.



Choosing Index Field on the first Owner gives the following result. The data is now merged and ready to be used for reporting and reserve runs. Any time these property lines are used in reserves, all of this data is available for reporting. Choose Continue will process the data and return to the command line.



Pulldown Menu Location: Boundary, Property Line Tools

Keyboard Command: setuppropdblclick

Property Names By Text

This command converts a large number of closed polylines into property lines by using text label placed inside of the polyline. You are prompted to select closed polylines to become property lines and then the program tries to locate a text label inside each closed polyline that has the same layer as that closed polyline. The text is used as the owner name and the property ID is set to blank.

When you have text labels to start with but no closed polylines, use the Boundary Polyline command under Draw menu to create closed polylines from appropriate linework by using the text insertion point.

Pulldown Menu Location: Boundary, Property Line Tools

Keyboard Command: grabprop

Identify Property Lines

This command is one of several property line tools available to identify and trouble shoot property lines. To identify a property line, simply click inside the polyline and the name of the owner and property ID will be displayed in the command text window.

Prompts

Pick on or inside of property polyline:

Owner: State

Property: 459a

Pick on or inside of property polyline:

Owner: Jones

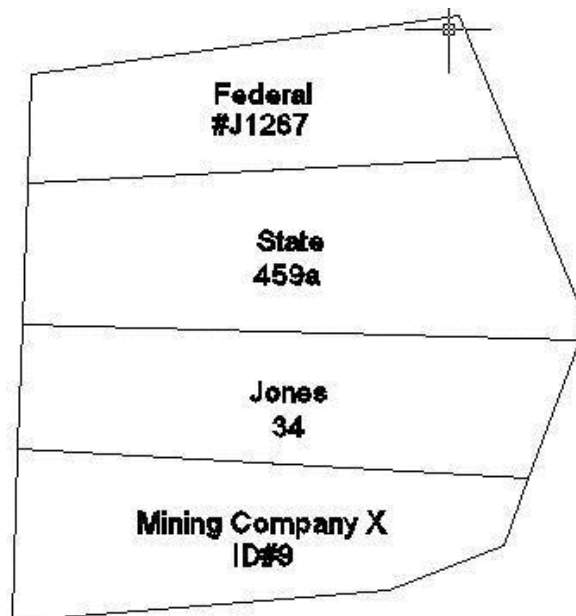
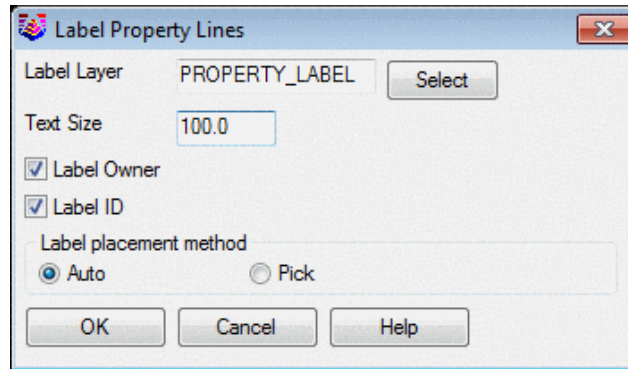
Property: 34

Pulldown Menu Location: Boundary, Property Line Tools

Keyboard Command: idprop

Label Property Lines

This command labels property lines in the drawing. The dialog contains option set the label layer and text size in feet or meters. There are options to label the owner, ID or both. When the Automatic mode is selected, the label is placed at the center of the property line. If the placement method is Pick, then each property line text location will need to be prompted for.



Prompts

Select all property polylines to label:

Select objects:

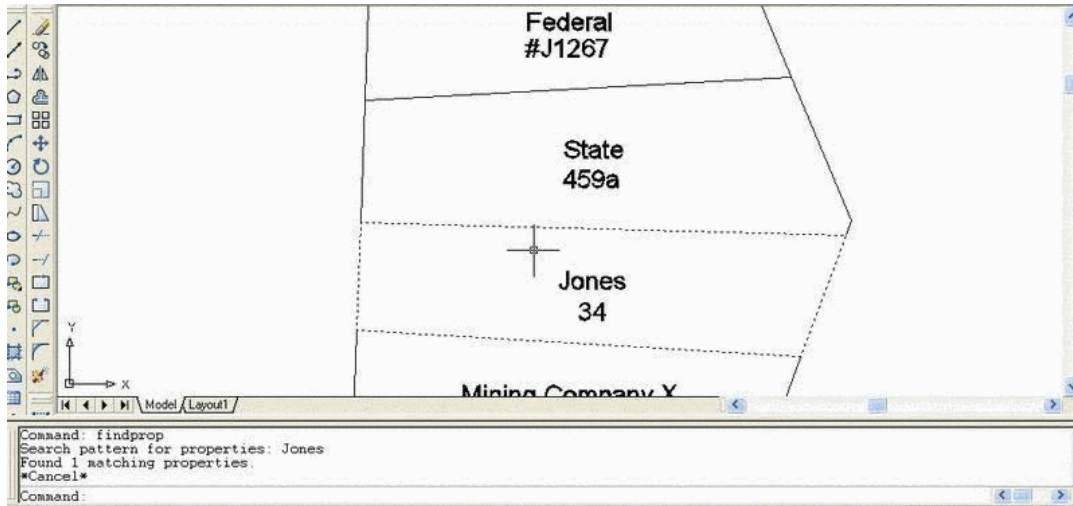
Dialog Box.

Pulldown Menu Location: Boundary in Surface Mining and Underground Mining Modules

Keyboard Command: labelprop

Find Property Lines

This command locates property lines by matching the user provided search pattern against owner name and property ID of all property lines in the drawing. All matching property lines will be highlighted. The wildcard character (*) can be used in the search pattern.

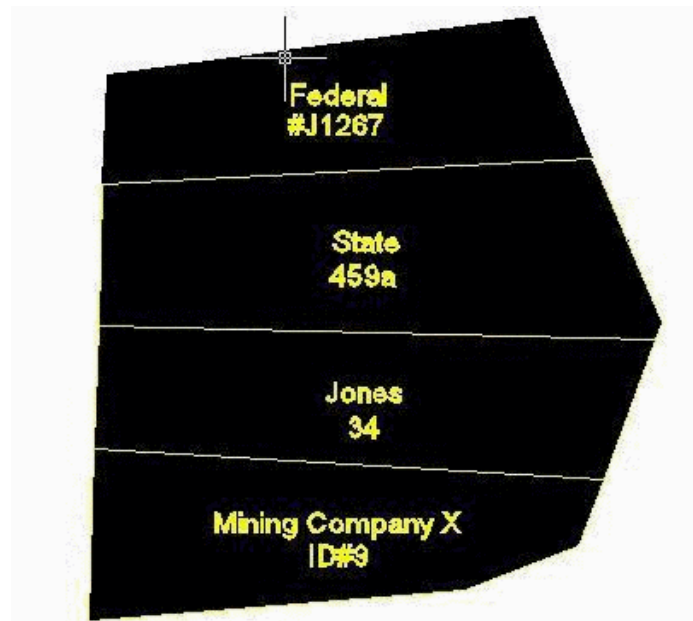


Pulldown Menu Location: Boundary, Property Line tools

Keyboard Command: findprop

Show All Property Lines

This function displays all property lines in the drawing either by highlighting them or by using solid fill. The latter option works for detection of the unassigned properties totally surrounded by the other properties. It is also convenient for location of tiny gaps between properties shown in the timing report as small areas with an UNKNOWN owner.



Prompts

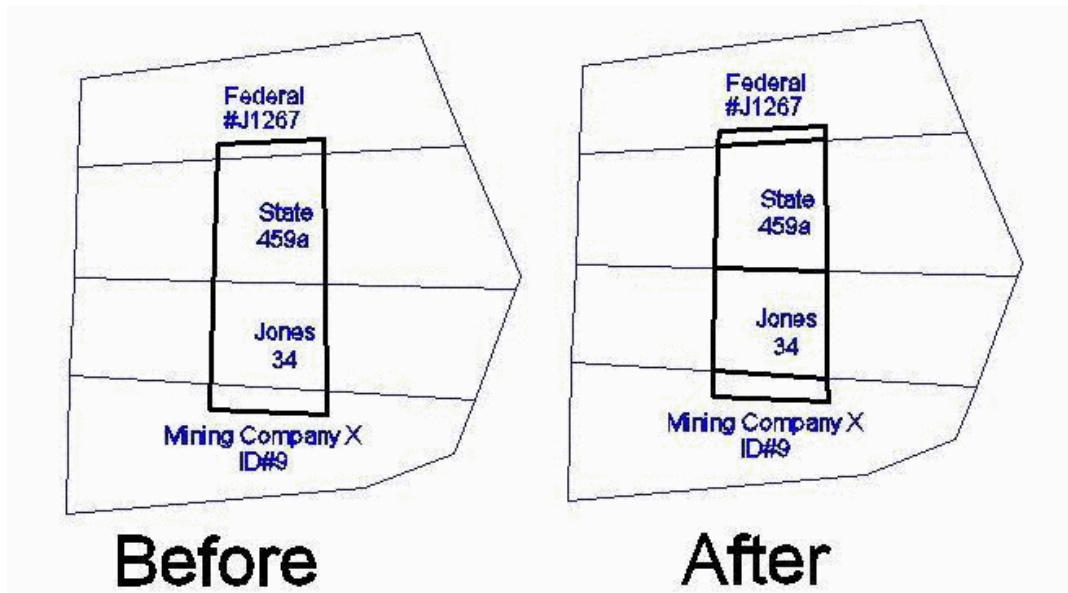
Solid-fill the properties [Yes/<No>]? Y

Pulldown Menu Location: Boundary, Property Line Tools

Keyboard Command: showprop

Break Polyline by Property

This command breaks a closed polyline into separate closed perimeters where they cross property lines. This is useful when smaller, individual perimeters are desired for pits or panels.



Prompts

Select polyline to divide by owners:

Pulldown Menu Location: Boundary

Keyboard Command: ownerdivide

Check Property Polyline Overlaps

This command checks whether property polylines overlap each other. Properties should never overlap since this would lead to extra volumes with the quantities in the overlap area getting reported twice. The program reports any overlaps at the command line.

Prompts

Select pit polylines to check.

Select objects: *pick the polylines*

Found 0 overlaps between 56 properties

Pulldown Menu Location: Boundary > Property Line Tools

Keyboard Command: propcheck

Prerequisite: property polylines

Extract Centroid Data

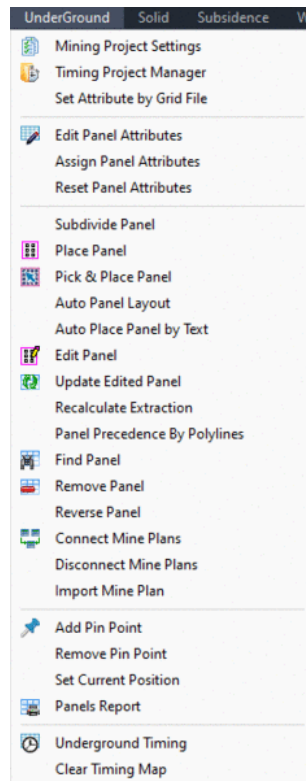
This command is used to convert property information imported from ArcInfo using shape files into property lines. It searches for polyline centroids, extracts the necessary information stored there and attaches it to the property line. This will only work in AutoCAD MAP.

Pulldown Menu Location: Boundary, Property Line Tools

Keyboard Command: import_shape

Underground Menu

The Underground menu has commands for mine layout and timing. The Mining Project Manager command is described in the Geology section of the manual. There are several different methods for placing panels for the mine layout. The differences between these routines is outlined in the chart below.



	Place Panel	Quick Place Panel	Pick & Place Panel	Auto Place Panel By Text
Level of details	Provides the high level of detail, allowing user to specify every setting for every panel during mine layout.	Assumes default settings except for every panel during mine layout.	Provides the high level of detail, allowing user to specify every setting for every panel during mine layout.	Except for extraction in the beginning all panel settings are default.
Layout speed	Moderate, depends on amount of settings edited.	Fast.	Slowest, but generates progress.	Quickest, most streamlined results as layout progresses. Very efficient in placing perim with direction.

Mine plan versatility	Creates a complete mine plan with "tree-like" connectivity, providing maximum information for timing precedence.	Creates a complete mine plan with "tree-like" connectivity, providing maximum information for timing precedence.	Generates mine plan consisting of separate panels with equipment already assigned. May be edited and replayed. Correct timing sequencing is user responsibility.
Main application	Established mine plan with relatively minor modifications in future.	Established mine plan with relatively minor modifications in future.	Best for laying out a complicated mine plan with multiple units working and numerous unconnected panels. Ideal for trial runs on large number of panels, where fast creation and extensive modifications are the must.

Mining Project Settings

This command sets the current file names used for several Carlson definition files. For example, this command assigns the Drillhole Configuration file (.ch file) to be used for drillhole placement in other routines such as Place Drillhole and Drillhole Data Sheet. The Mining Project Manager lets you set up different groups of definitions. For example, if you work with two mines, then you can have two sets of strata definitions (strata1.sdf and strata2.sdf), two sets of attribute definitions (attrib1.atr and attrib2.atr), etc. When switching between the projects, you can load the .mpj file to instantly update all of your relevant settings.

Whenever a command is executed that creates one of the below files, it will be automatically added to the Mining Project Manager. However, you can also specify the file names directly from this command.

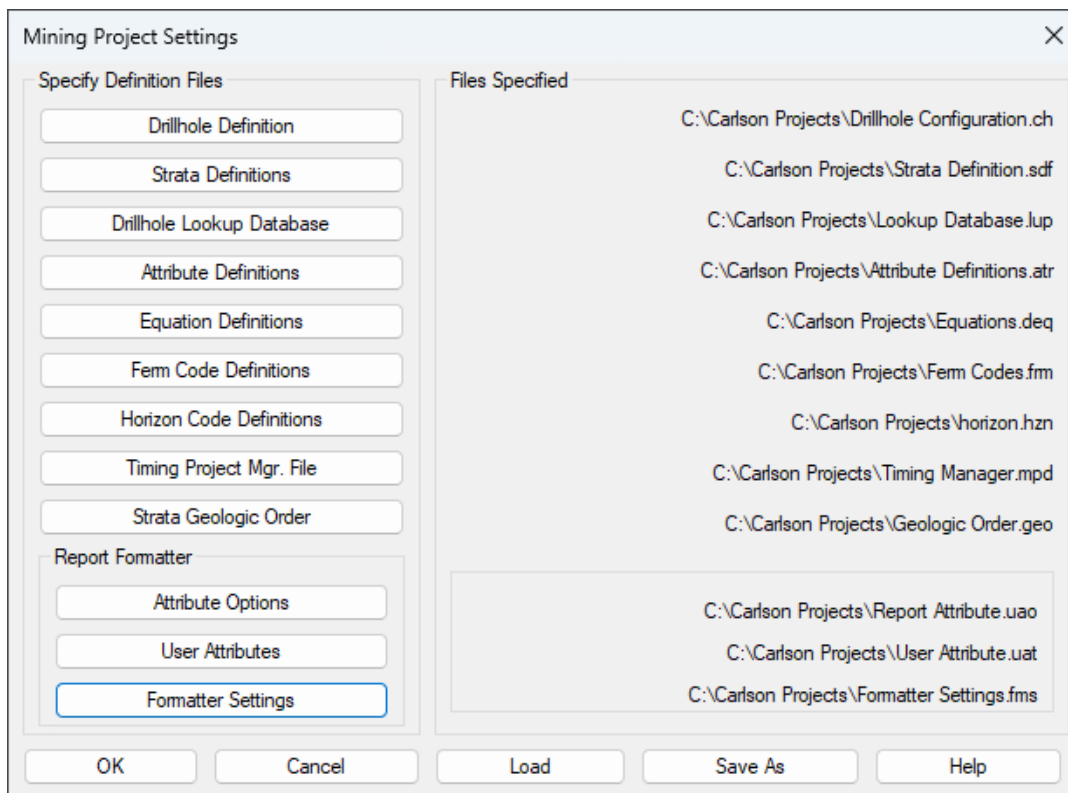
All the file names in Mining Project Manager are saved with the drawing in the drawing .INI file. You can also save this set of file names to a mining project file (.mpj) by picking the Save button. The Load button will then recall the set of file names from the selected .MPJ file.

Following is a list of commands that will create the files shown in the Mining Project Manager. The extensions of the files are shown in the dialog window below.

1. Drillhole Definition: Define Drillhole
2. Strata Definitions: Define Strata/Bed
3. Drillhole Lookup Database: Define Lookup Database

4. Attribute Definitions: Define Attributes
5. Equation Definitions: Define Equations
6. Ferm Code Definitions: Define Ferm Codes
7. Horizon Code Definitions: Define Horizon Codes
8. Timing Project Mgr. File: Underground Timing, Surface Equipment Timing, Solids Timing

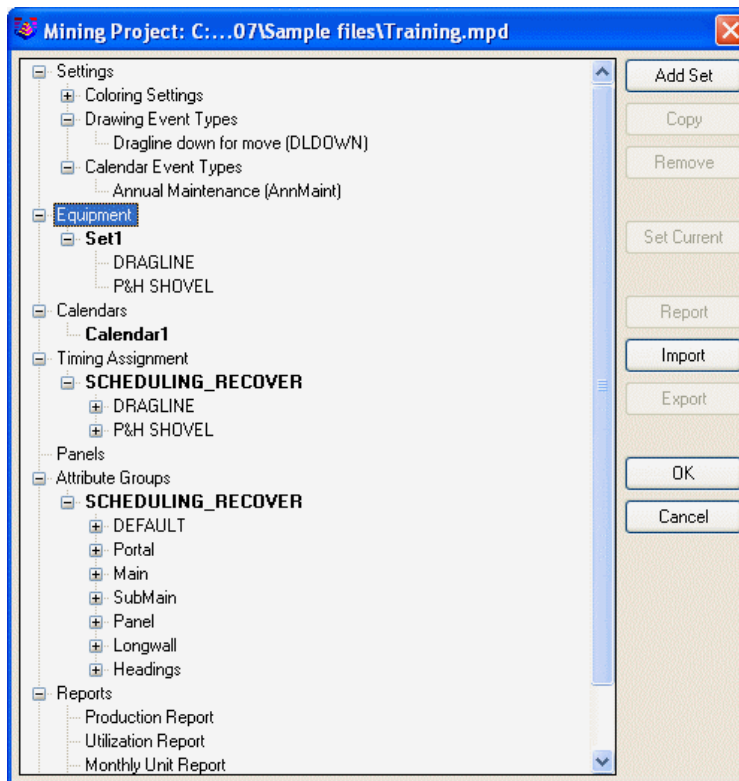
9. Strata Geologic Order: Define Geologic Order
10. Attribute Options: In the Report Formatter, Attribute Options
11. User Attributes: In the Report Formatter, User Attributes.
12. Formatter Settings: In the Report Formatter, Export



Pulldown Menu Location: Drillhole
Keyboard Command: mproject

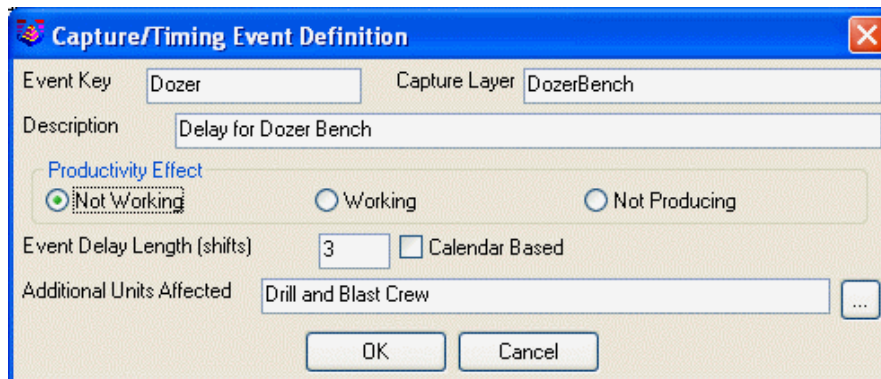
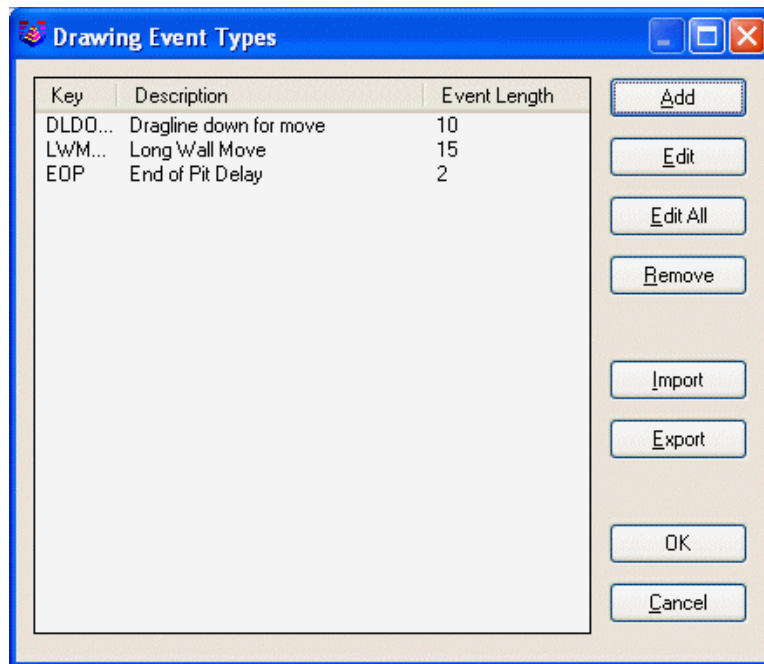
Timing Project Manager

The Timing Project Manager (Surface Project Manager or Underground Project Manager) applies to both surface and underground mining for file organization and selection. All files and most settings and configurations used in the mine scheduling are found in the manager. Each topic is described below in detail. Note that some topics listed below may refer to panels for Underground Timing or Pits for Surface Equipment Timing.

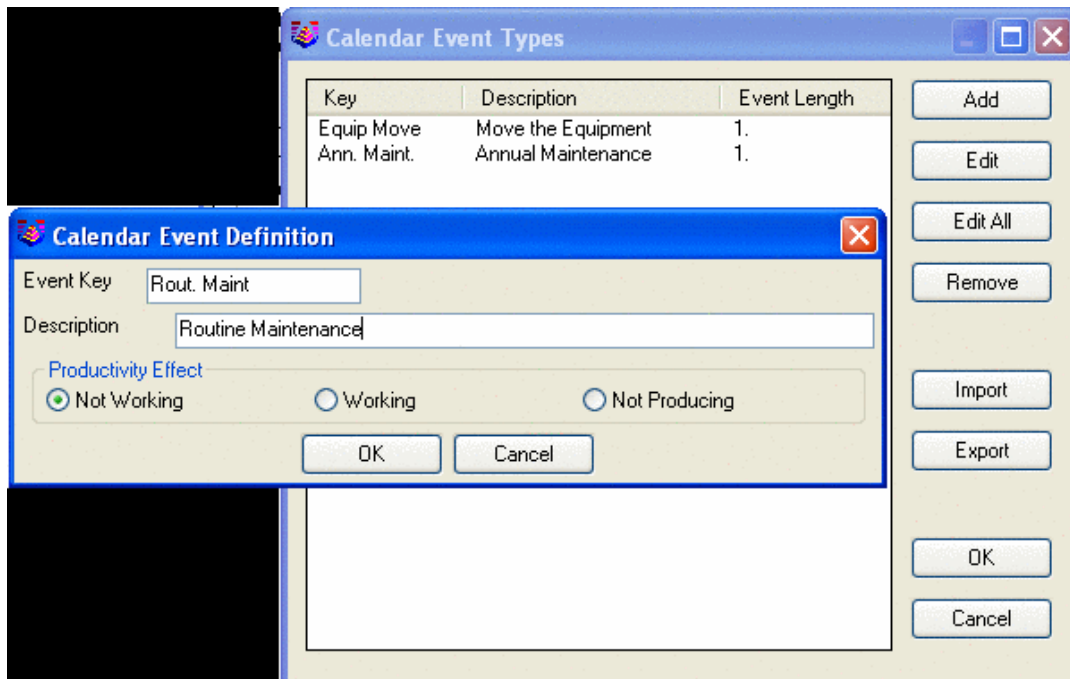


Settings

- Coloring Settings:** These settings are from the Report Options screen from the equipment timing command. Defaults can be set here before running the actual schedule. Explanation of this dialog is found in Underground Timing or Surface Equipment Timing.
- Drawing Event Types:** Drawing events are entities in the drawing at a specific location that will be used for a marker in the schedule as an option for a delay. This event can be used to delay equipment, switch crews, move other equipment, etc. When the schedule crosses over the insertion point of the drawing event (usually text) the event will trigger the delay or effect. The text/drawing entity in the drawing should be inserted to the position where the delay where occur and in the AutoCAD layer defined in the Capture Layer window. The Event Key and description can be used in the reporting of the schedule to create additional reported items.

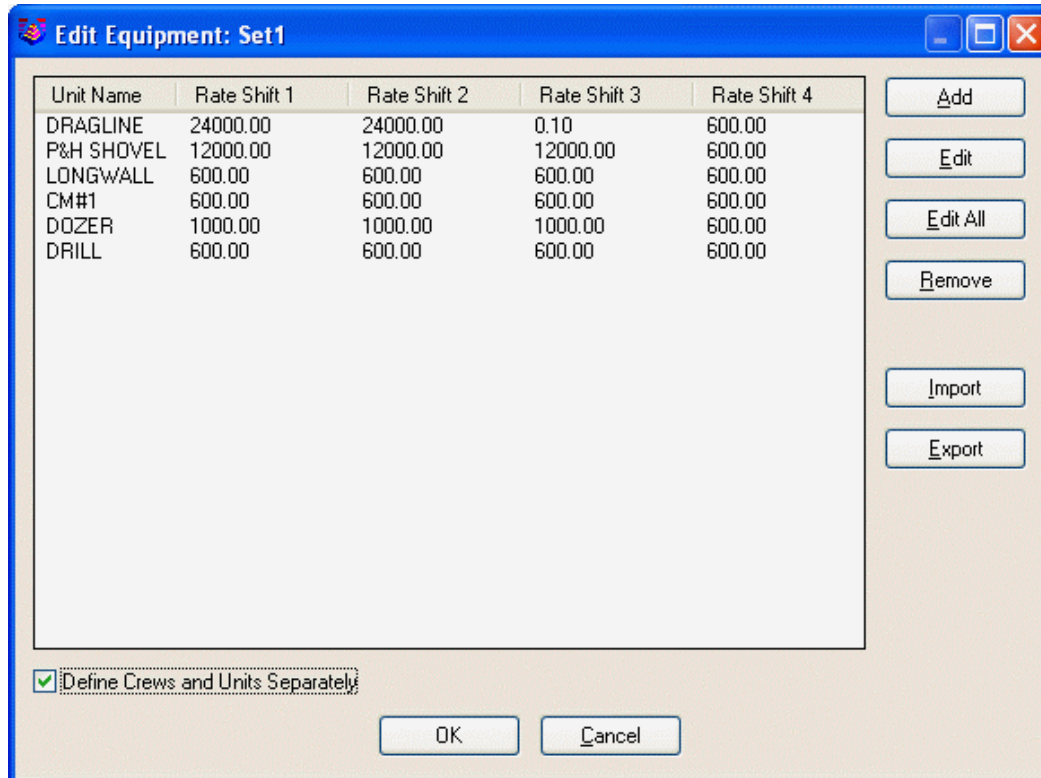


- Calendar Event Types:**Calendar events are named events that will determine equipment down time. The default event in the calendar is Manual. The other events are set under the Calendar Event Types and Calendar Event Definition. Calendar events can be any event that will cause a delay, such as a routine maintenance, a long wall move, or a dragline dead head. The Productivity Effect allows the option of Not Working at all, Working as normal, or Not Producing (yet still working with crews, using hours, but not mining).

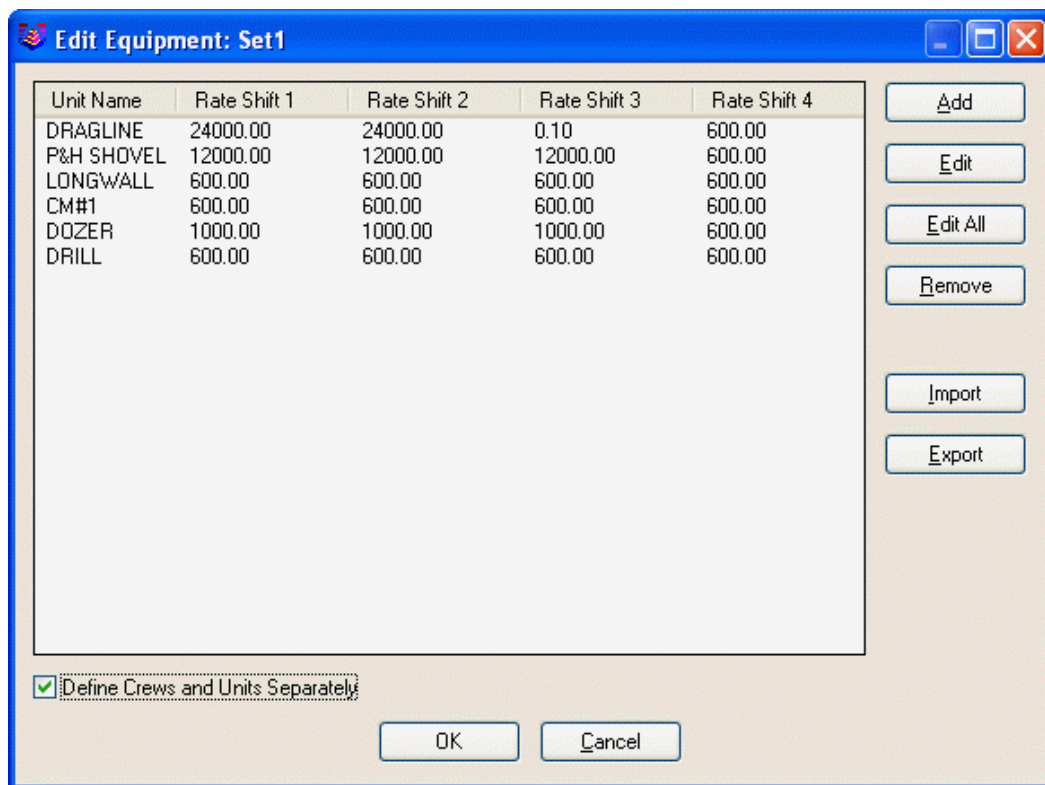


Equipment

- **Equipment Set:** The productivity of the unit (crew) may vary from shift to shift, up to 4 shifts a day. The number of shifts is set in the Timing window where the sequence of pits or panels is set. The average productivity rate is used for scheduling, however the production rate for the particular pit or panel may be adjusted by setting the difficulty factor or adding delays for that pit or panel. The first window is the edit equipment screen with a columnar display of equipment. Crews can be defined to work with a specific unit, or can switch from unit to unit based on the schedule.



Add and Edit will bring up the next screen for detailed entry.



The Unit of the Production value is defined by a rate unit setting. Equipment may mine either tons, tonnes, CY or CM, Distance and Linear Advance. The difference between distance and linear foot of advance settings is that latter one is a combined length of all pathways/entries mined (underground only), and distance is just distance moved, of a longwall for example. Linear Feet of Advance is calculated as the area extracted divided by the entry width assigned to the panels. Note that this calculation can be skewed if the entry width varies, or if the pillar dimensions vary.

$$\text{Linear Ft Advance} = \text{Pillar Area} * \text{Extraction Ratio} / \text{Entry Width}$$

Enter the Advance Rate/shift or the Advance Rate/hour and the other will be automatically calculated based on the Hours/shift. The Retreat Rate/shift is for Distance and Linear ft of Advance in underground equipment only. The availability value of less than 1.0 will reduce effective production rate of the equipment. 0.94 is a 94% productivity of the full shift. Rates may also be specified in the drawing itself by placing a piece of text on the RATE layer. The value in the text should represent the rate per hour. Note that a piece of text on the RATE layer will temporarily override the normal rate specified on this dialog.

Underground units may be assigned an Advance minimum and/or maximum height, so that the extra rock will have to be mined or correspondingly unmined coal will be left in the seam if the maximum miner height is less than combined coal and rock parting thicknesses. These settings affect the underground mine timing only. The maintenance settings provide the ability to schedule a delay for routine or major maintenance/repair of the equipment based on the number of shifts worked. Add in the length of the delay and the number of shifts to determine the frequency.

If operational cost per hour is specified, the total cost will appear in the production report.

The extraction (recovery) factor is used to adjust the amount of material mined from given area to account for certain technical limitations of the equipment such as inability to mine out corners, or not cleaning the top and bottom of ore. This machine will always use that recovery rate.

Advanced Options:

Define Production Change

PH SHOVEL

Date Specific Difficulty		
	End Date	Difficulty
1	11/1/8	1.000
2	4/1/9	1.200
3	11/1/9	1.000
4	4/1/10	1.200
5	11/1/10	1.000
6	4/1/11	1.200
7	11/1/11	1.000
8	4/1/12	1.200
9		
10		
11		
12		
13		
14		
15		
16		

Thickness Specific Difficulty and Rehandle			
	Thickness	Difficulty	Rehandle
1	0	1.000	
2	10	1.050	
3	25	1.100	
4	50	1.150	
5	75	1.170	
6	90	1.200	
7	100	1.500	
8			
9			
10			
11			
12			
13			
14			
15			
16			

Bench Specific Difficulty		
	Bench	Difficulty
1	1	1.000
2	2	1.100
3	3	1.150
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		

Reset OK Cancel Load Save

Under the Advanced options the variation of equipment-related Difficulty Factor with time, depth or bench number may be specified. Resulting from the advanced options, the difficulty factor is a product of coefficients calculated for given date, thickness and bench number. The final difficulty factor used in calculations is a product of location-specific and equipment-specific difficulty factor.

In the Period End Date column, the difficulty for the date is for next, later date, or last one if no later entry exists. For the thickness column the value of difficulty factor between two entries in the table is a linear approximation. The rehandle value is calculated in the same fashion and passed over to the report, not being used in calculations, but can be used in the report in equations to calculate the total amount mined. The difficulty will modify the rate, and the rehandle is reported. These are not necessarily always the same, linear factors. The Bench Specific Difficulty will be used for surface equipment mining on that bench number, and use that difficulty factor to change the rate of the equipment.

Selecting Edit All from the first screen brings up this editor where all equipment may be viewed and edited. Both the first screen and this one have the import and export buttons.

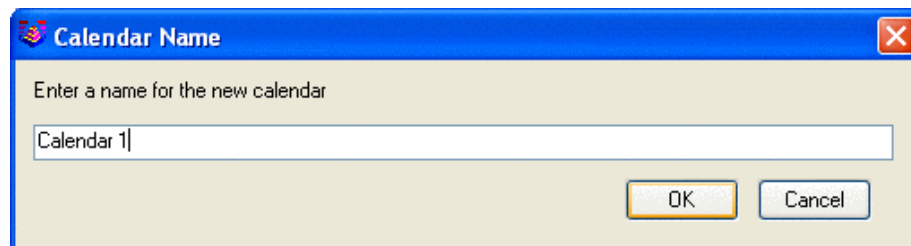
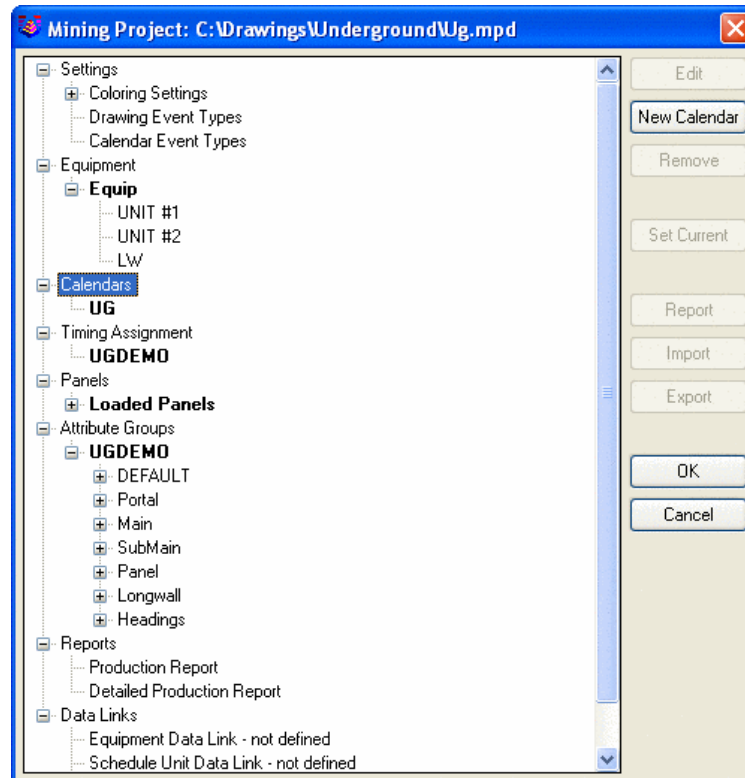
Edit Equipment: Set1

	Unit Name	Rate Shift 1	Rate Shift 2	Rate Shift 3	Rate Shift 4	min height	max height
1	DRAGLINE	24000.00	24000.00	0.10	600.00	0.00	98765432
2	P&H SHOVE	12000.00	12000.00	12000.00	600.00	0.00	98765432
3	LONGWALL	600.00	600.00	600.00	600.00	0.00	98765432
4	CM#1	600.00	600.00	600.00	600.00	0.00	98765432
5	DOZER	1000.00	1000.00	1000.00	600.00	0.00	98765432
6	DRILL	600.00	600.00	600.00	600.00	0.00	98765432

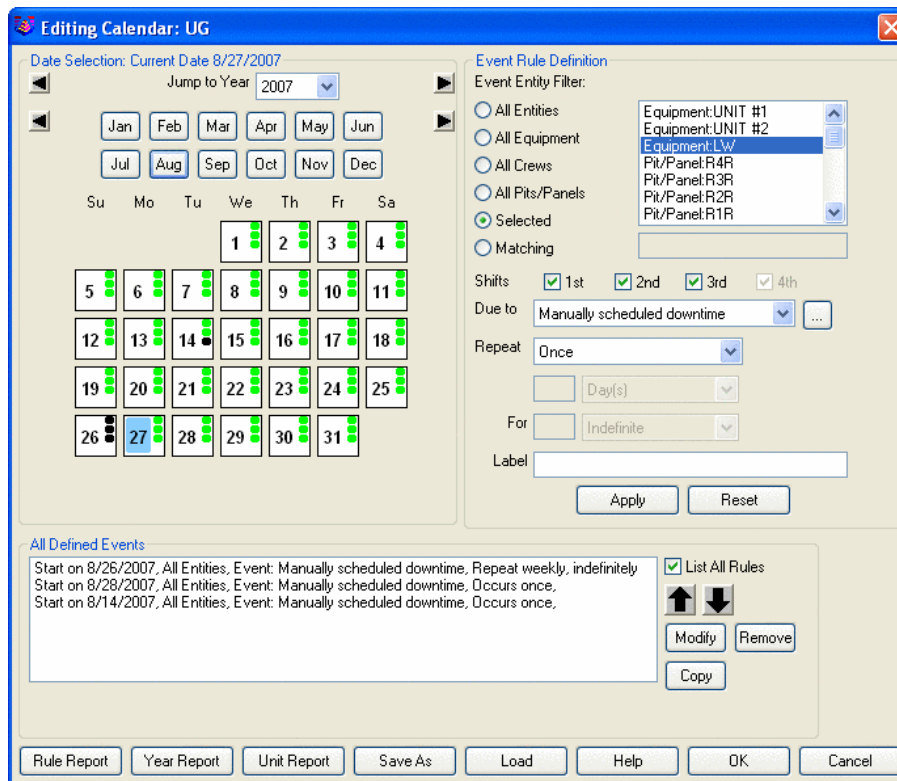
Import Export OK Cancel

Calendars

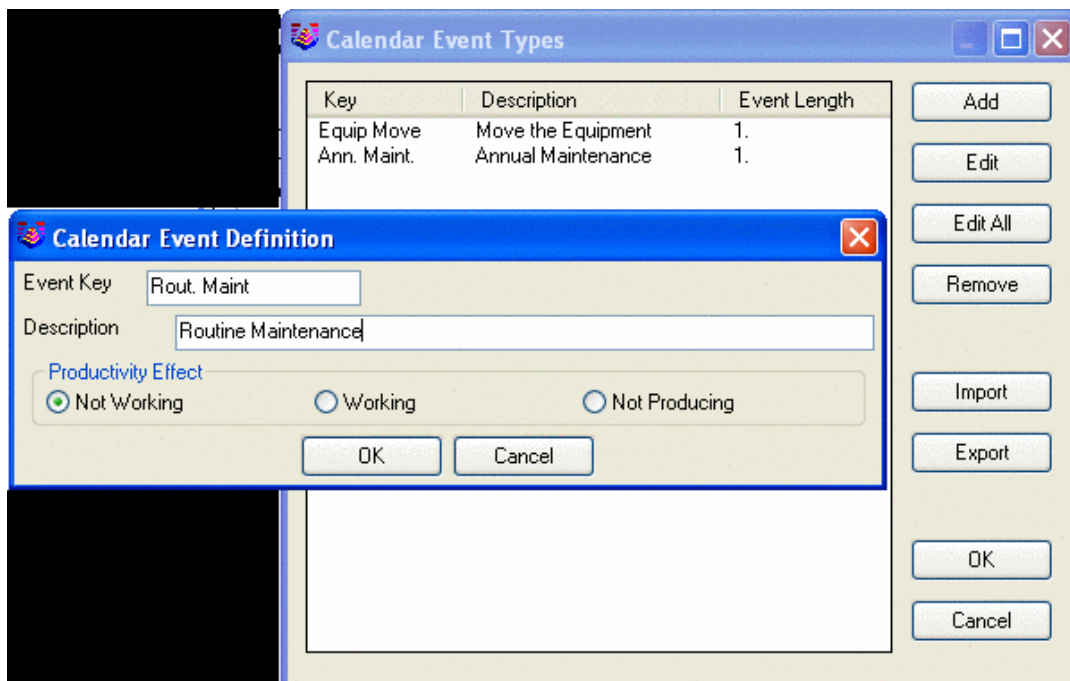
- New or Existing Calendar: Highlight the Calendar tree, and the New Calendar button is activated if no calendars are present. If some are present, the edit button will also be active for editing a calendar. If it is a new calendar, the Calendar Name box appears to enter a name.



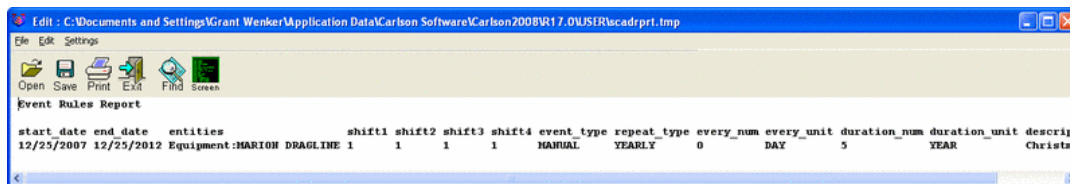
The Equipment Calendar allows for entity production down time. By default, entities are working every day, every shift. Entities include equipment, crews, pits and panels. Days and shifts defined in this calendar as down-time are taken into account during scheduling routines. The assignment pattern is very flexible: it works for a particular crew or all crews, for a particular shift or the whole day and allows replication of the defined behavior over period of time as desired. The calendar will clean out, or purge a schedule for a crew which is no longer present.



- **Date Selection:** The calendar should initiate on the current day. Use the left/right arrows to scroll through the months and years. The current or selected day is shown in blue. The shifts appear on the right of each day. There can be up to 4 shifts. If the shift is green, then all entities are working; if it is yellow, then some are working and some are down. If the shifts are black, then all entities are down on that day and shift.
- **Event Rule Definition:** This section is where the event down times are created and edited. Apply adds the event, Reset clears it for the next assignment.
- **Event Entity Filter:** The event list contains the Equipment, Pits or Panels and Crews. The choices for filtering are All Entities, All Equipment, All Crews, All Pits/Panels, Selected (highlight the desired Entity), and Matching (highlight the Entity to find the match).
- **Shifts:** The number of shifts checked here will display next to each day.
- **Due to/Calendar Events:** The options for down time are selected here. The default event is Manual. The other events are set under the Calendar Event Types, activated by clicking the button next to the "Due to" window. Calendar events can be anything that will cause a delay, such as a routine maintenance, a long wall move, or a dragline dead head. The Productivity Effect allows the option of Not Working at all, Working as normal, or Not Producing (yet still working).



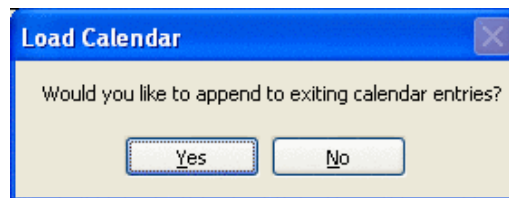
- **Repeat for/Label:** This setting is how often the down time will repeat. This can be from a day, month, year or indefinite. This event can have a name, which is entered in the Label window. The calendar will refer to this label when encountering and reporting this event.
- **Events for Date Selected/All Defined Events:** This window displays the events for the highlighted day. If the List All Rules is on, then all the Events will be displayed here, no matter which day is highlighted.
- **Rule Report:** This report displays all of the rules defined for down time within the calendar.



- **Year Report:** This reports all days for the selected calendar year.
- **Unit Report:** This reports all equipment and dates defined in the calendar. See example below.

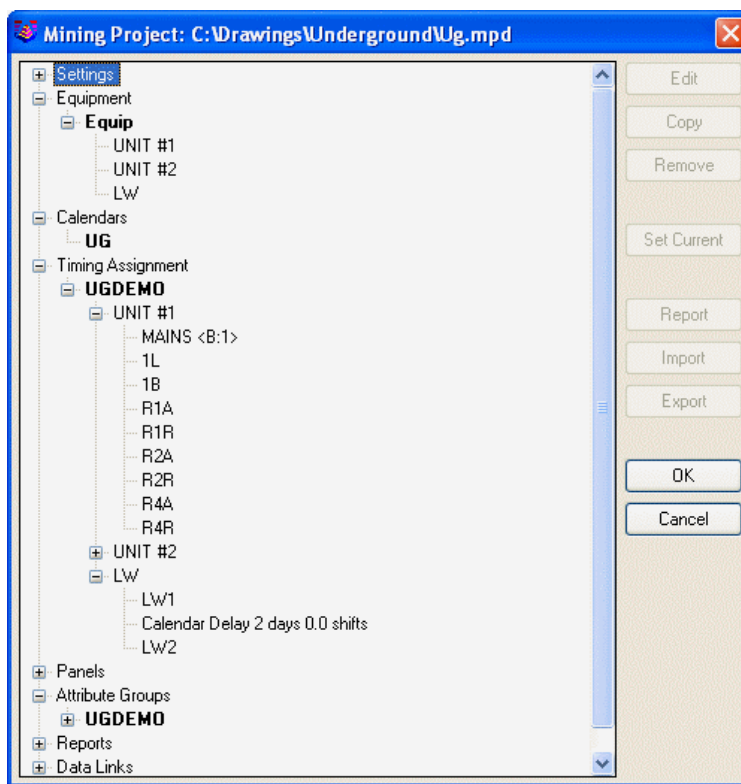
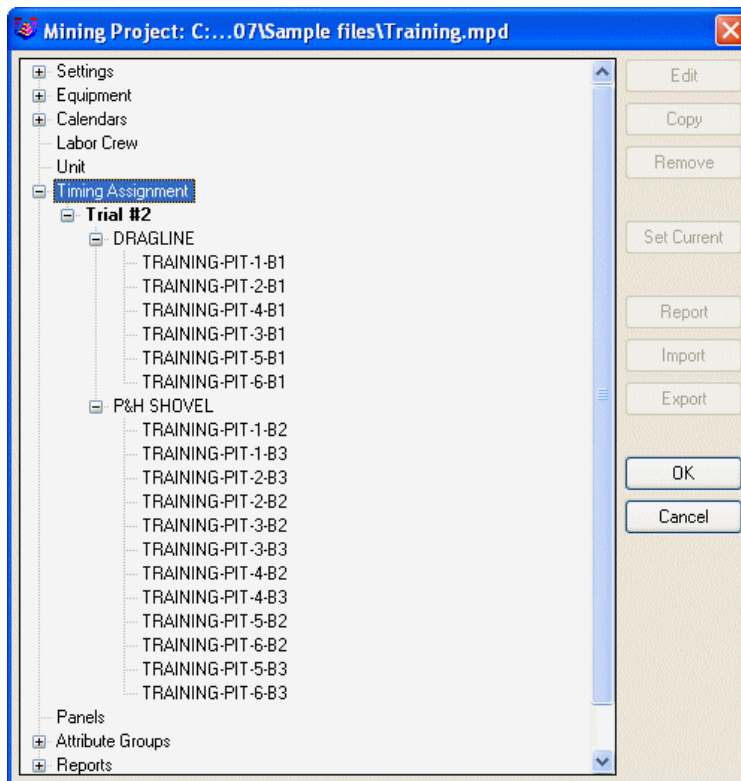
Unit	Month	Year	Operating days	Days	Weekdays	Worked	Saturdays	Worked	Sundays	Worked	Calendar days	Work shifts	Hours
LW	January	2011	28.00	28.00	19.00	4.00	4.00	5.00	31.00	84.00	0.00	0.00	
LW	February	2011	28.00	28.00	20.00	4.00	4.00	0.00	28.00	84.00	0.00	0.00	
LW	March	2011	31.00	31.00	23.00	4.00	4.00	0.00	31.00	93.00	0.00	0.00	
LW	April	2011	19.67	19.67	17.67	2.00	0.00	0.00	30.00	59.00	0.00	0.00	
LW	May	2011	21.67	21.67	19.67	2.00	0.00	0.00	31.00	65.00	0.00	0.00	
LW	June	2011	18.67	18.67	16.67	2.00	0.00	0.00	30.00	56.00	0.00	0.00	
LW	July	2011	13.33	13.33	12.00	1.33	0.00	0.00	31.00	40.00	0.00	0.00	
LW	August	2011	22.67	22.67	21.33	1.33	0.00	0.00	31.00	68.00	0.00	0.00	
LW	September	2011	21.00	21.00	19.67	1.33	0.00	0.00	30.00	63.00	0.00	0.00	
LW	October	2011	19.00	19.00	17.67	1.33	0.00	0.00	31.00	57.00	0.00	0.00	
LW	November	2011	18.67	18.67	17.33	1.33	0.00	0.00	30.00	56.00	0.00	0.00	
LW	December	2011	18.00	18.00	16.00	2.00	0.00	0.00	31.00	54.00	0.00	0.00	
LW	January	2012	13.67	13.67	13.67	0.00	0.00	0.00	31.00	41.00	0.00	0.00	
LW	February	2012	15.33	15.33	15.33	0.00	0.00	0.00	29.00	46.00	0.00	0.00	
LW	March	2012	12.67	12.67	12.67	0.00	0.00	0.00	31.00	38.00	0.00	0.00	
LW	April	2012	12.00	12.00	12.00	0.00	0.00	0.00	30.00	36.00	0.00	0.00	
LW	May	2012	16.33	16.33	16.33	0.00	0.00	0.00	31.00	49.00	0.00	0.00	
LW	June	2012	11.00	11.00	11.00	0.00	0.00	0.00	30.00	33.00	0.00	0.00	
LW	July	2012	11.33	11.33	11.33	0.00	0.00	0.00	31.00	34.00	0.00	0.00	
LW	August	2012	15.33	15.33	15.33	0.00	0.00	0.00	31.00	46.00	0.00	0.00	
LW	September	2012	13.67	13.67	13.67	0.00	0.00	0.00	30.00	41.00	0.00	0.00	
LW	October	2012	15.67	15.67	15.67	0.00	0.00	0.00	31.00	47.00	0.00	0.00	
LW	November	2012	13.67	13.67	13.67	0.00	0.00	0.00	30.00	41.00	0.00	0.00	
LW	December	2012	12.00	12.00	12.00	0.00	0.00	0.00	31.00	36.00	0.00	0.00	

- **SaveAs/Load:** This saves the calendar, optionally as a new name. Load will open a new one, and give the option to append to the existing, open calendar.



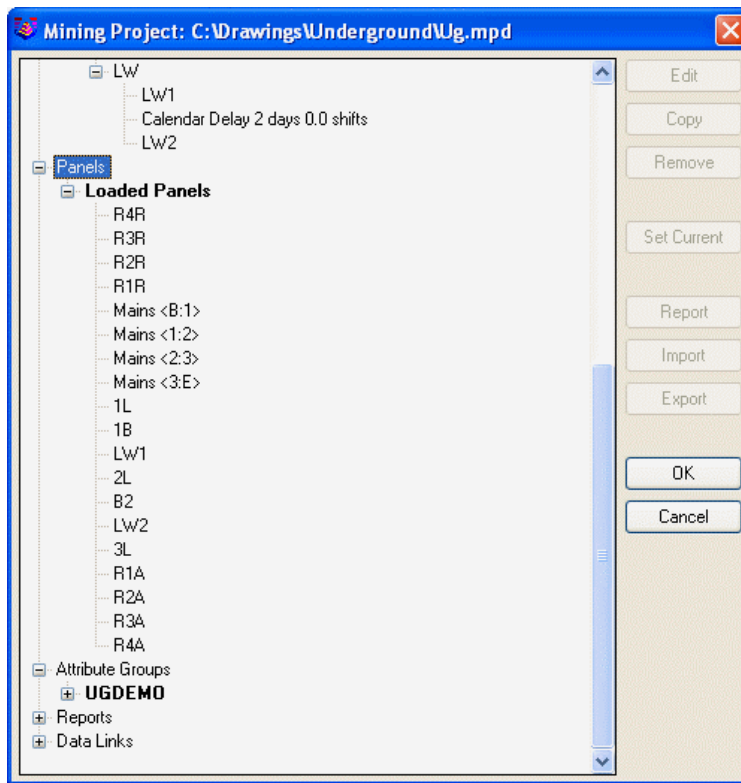
Timing Assignment

The Timing Assignment allows for setting the current sequence. The Assignment is similar to the TIM file in the timing routines. The equipment used and which panels or pits they are mining are shown. The Set Current button controls which one will be loaded and used. Shown here is both a surface mine and an underground mine example.



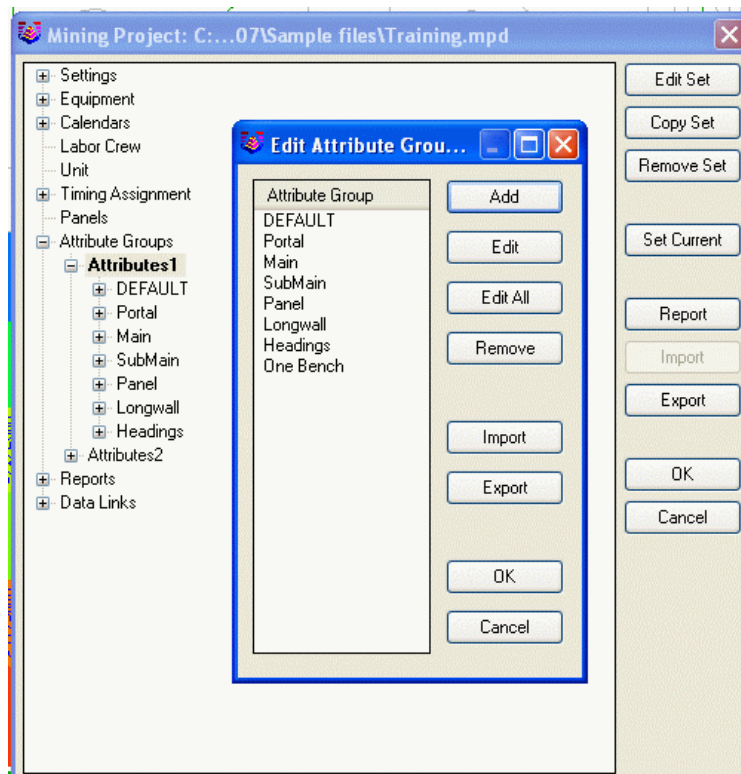
Panels

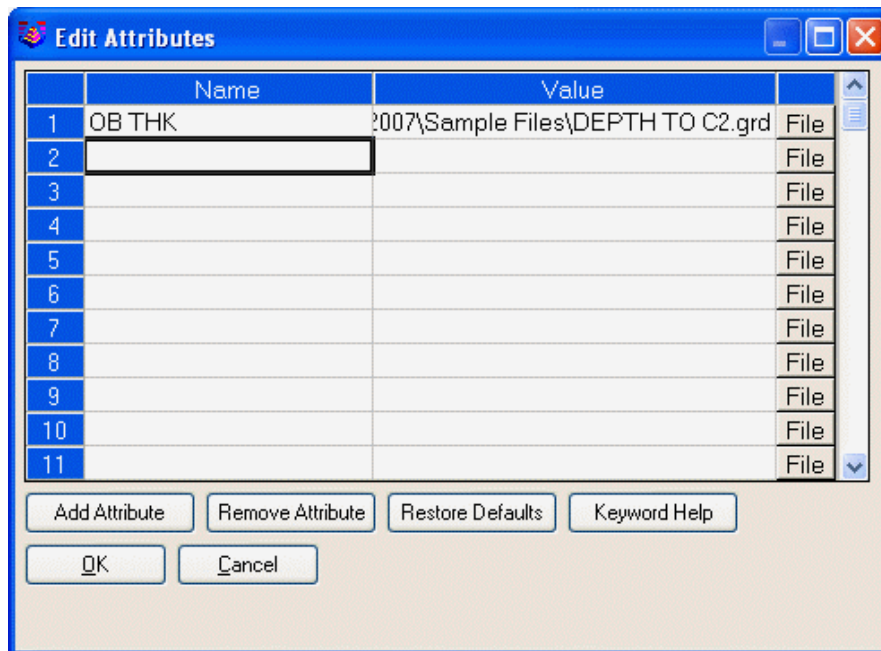
This displays the selected underground panels in the mine plan assignment. An example is shown here.



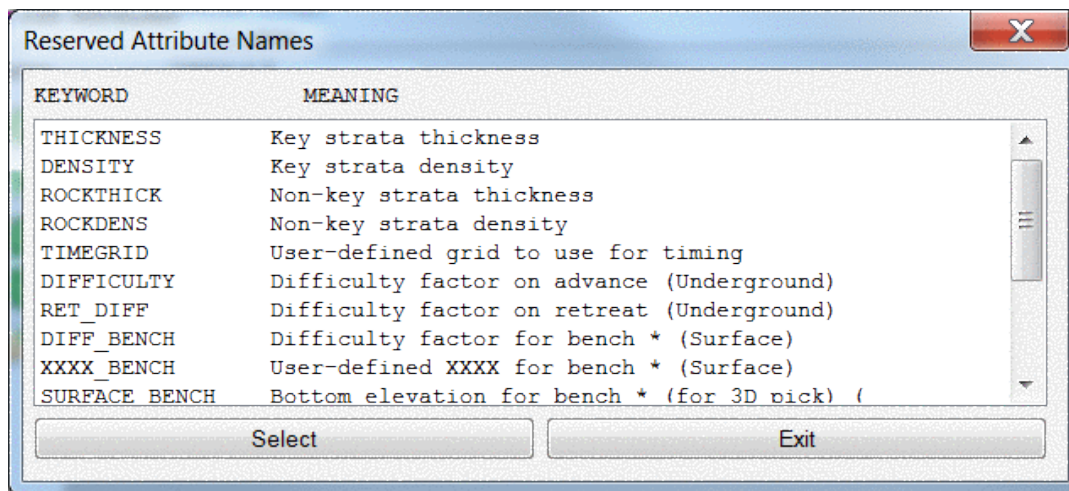
Attribute Groups

Attribute groups are Pit or Panel Attributes assigned to the pits and panels for timing. When timing the pits or panels, there are additional attributes that will be calculated and reported in addition to the Non-Key and Key quantities. If these are defaults that should be applied to all the panels or pits, then they should be entered here. Any quality values, density, or difficulty attributes can be defined here either just as a value, or as a grid file with varying values. The Reserved Attribute Names window displays the reserved words that will be recognized in the timing routines, and how they should be entered in. Selecting Keyword Help brings up the Reserved Attribute Names.





The Reserved Attribute Names are defined in a little more detail here:

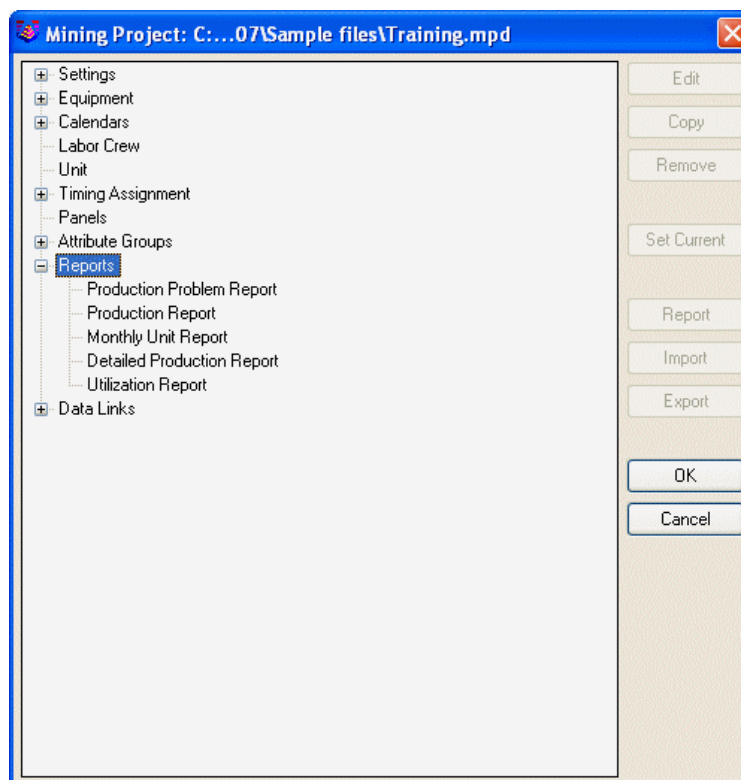


- **THICKNESS:** Key strata thickness: Will report any other thickness values, this is used mostly for underground timing.
- **DENSITY:** Key strata density: Will report Key density, is used mostly in underground timing to calculate the tons. In surface timing, the tons are already in the pits.
- **ROCKTHICK:** Non-key strata thickness: Reports the Non-Key thickness mostly for underground timing.
- **ROCKDENS:** Non-key strata density: Reports the Non-Key density mostly for underground timing.
- **TIMEGRID:** User-defined grid to use for timing: This will be any additional grid the user would like to add for reporting.
- **DIFICULTY:** Difficulty factor on advance just for underground timing: This will alter the underground equipment rate on advance.
- **RET_DIFF:** Difficulty factor on retreat for underground timing: This will alter the underground equipment rate on retreat.
- **DIFF_BENCH:** Difficulty factor for bench * for surface timing: This will speed up or slow down the equipment as it mines the specified bench. If a value is above 1, such as 1.2, then it will mine 20% slower at that

- point in the grid, or everywhere if it is set to value. If it is less than 1, such as 0.84, then it will mine 16% faster at that point.
- XXXX_BENCH: This is any other attribute that will be reported out for a specific bench in the Surface Timing routines. All quality grids will be defined in this fashion, for each bench. Examples would be BTU_BENCH or Rehandle CY. All quality parameters need to be entered as the NAME_BENCH*.
 - SURFACE_BENCH: This is the bottom elevation grid for each bench. They are used in Surface Equipment Timing in the 3D Pick window so that the mine pit blocks are draped onto the actual elevation surfaces.
 - SURFACE: This is the surface topography elevation grid. It is used in Surface Equipment Timing in the 3D Pick window so that the mine pit blocks are draped onto the actual elevation topo.
 - TIMING_SPLIT: This will globally split all blocks that contain both Key and Nonkey material into two blocks that may be scheduled separately, by either the same unit, or different units. The split is into two portions - key and non-key. Entering a non-zero value will cause a split. Additionally three values are supported as pertains to precedence. Enter in one of these values in for the attribute to split it.
 - 1 = Both OB and KEY needs to be completed to satisfy precedence
 - 2 = OB portion needs to be completed to satisfy precedence
 - 3 = KEY portion needs to be completed to satisfy precedence
 - TIMING_SPLIT_BEN: This attribute is the same as TIMING_SPLIT, but is only applied to the specified bench number.
 - RET_THICK: This attribute is applied to Underground Timing. The retreat mining Key thickness grid can be different than the advance Key thickness grid.
 - RET_ROCKTHICK: This attribute is applied to Underground Timing. The retreat mining NonKey Rock thickness grid can be different than the advance Rock thickness grid.

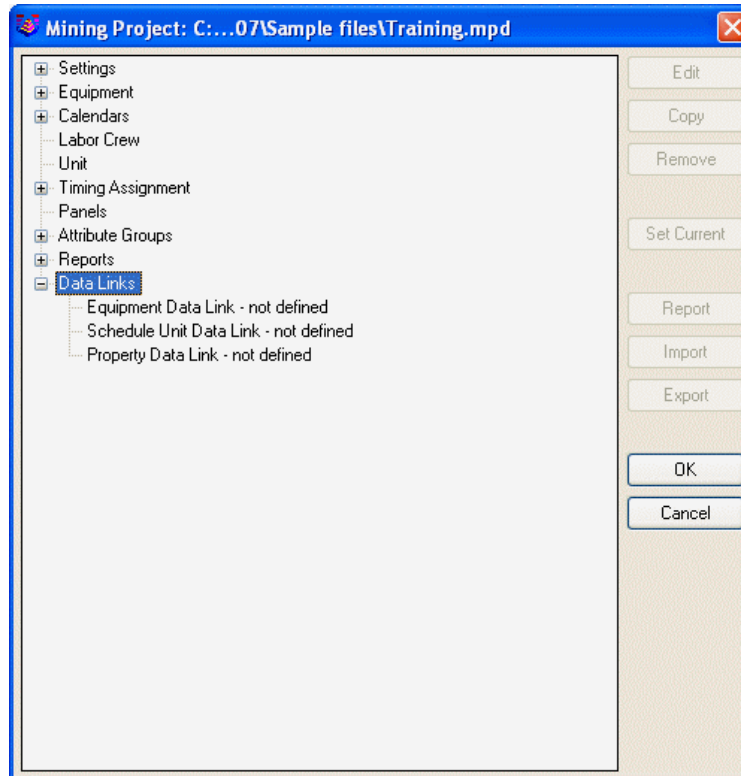
Reports

Some of the reports generated in the scheduling will appear here for other options to report. These are also accessed in the calendar or after a schedule is run.



Data Links

This shows the link of the data when it is linked to an external source for reporting.



Skip Past Report Archiving in Project Database: By default, all reports generated with timing commands will be saved in a backup of the Timing Project Manager as an .mrd file (the file name will be the same as the .mpd file, just with a different extension). If many reports are generated, this archive can become quite large, and may even slow down processing time when running the timing command. When this option is enabled, no reports will be archived, preventing potential slow-down from large .mrd files. It is recommended to enable this option when you have no need for old timing reports.

Pull-Down Menu Location: Reserves/Timing in Surface Mining and Underground in Underground Mining
Keyboard Command: stime_project or utime_project

Set Attribute by Grid File

This command assigns an attribute value such as thickness or BTU to a panel or pit area based on the average value from a grid file. The program prompts for the attribute name and the grid file that represents the attribute. Then multiple panel/pit perimeter polylines can be selected and the program calculates the average grid value in each panel/pit polyline and stores this value.

Another method for using grids is to define the attribute as the actual grid file name instead of the average value. Then the timing routines will calculate attribute values for each timing block from the grid value. The advantage to using Set Attribute by Grid File instead of using the grid file is speed.

The Calculate Seam Split Quantities option is for calculating coal and rock thicknesses by analyzing multiple strata elevation grids using minimum and maximum miner heights. When the coal thickness is less than the minimum height, the minimum height is used. The maximum height is used when the coal thickness is greater than the maximum height. A fixed amount of rock thickness that is always taken can be specified. Given the miner height,

the program will find when the miner takes main coal seam plus the parting and any of the second coal seam. The average coal and rock thickness for each panel polyline is calculated and stored.

Prompts

Type of polylines (<Underground>/Surface)? *press Enter*

Calculate seam split quantities (Yes/<No>)? *press Enter*

Attribute Name: *BTU*

Select Grid File dialog

Select panel polylines.

Select objects: *pick the panel perimeter polylines*

Pulldown Menu Location: AdvMine

Keyboard Command: gridattr

Edit Panel Attributes

The function edits the panel attributes attached to a panel perimeter polyline. The list of attributes is loaded from a polyline or created anew using the Default Panel Attributes defined. The dialog used is the same dialog as in the Define Panel Attributes function. Edit Panel Attributes allows you to specify a distinct set of attributes for the particular perimeter polyline. This set has precedence over the default one.

Edit Section Data

Panel name: LW1 Panel start: []

Easting: 3270.7272 Northing: 6826.7719 Azimuth: 270d0'0''

Entry Width: 20.0000

Extraction ratio: 1.0000

Prompt for coal qualities

Prompt for retreat options

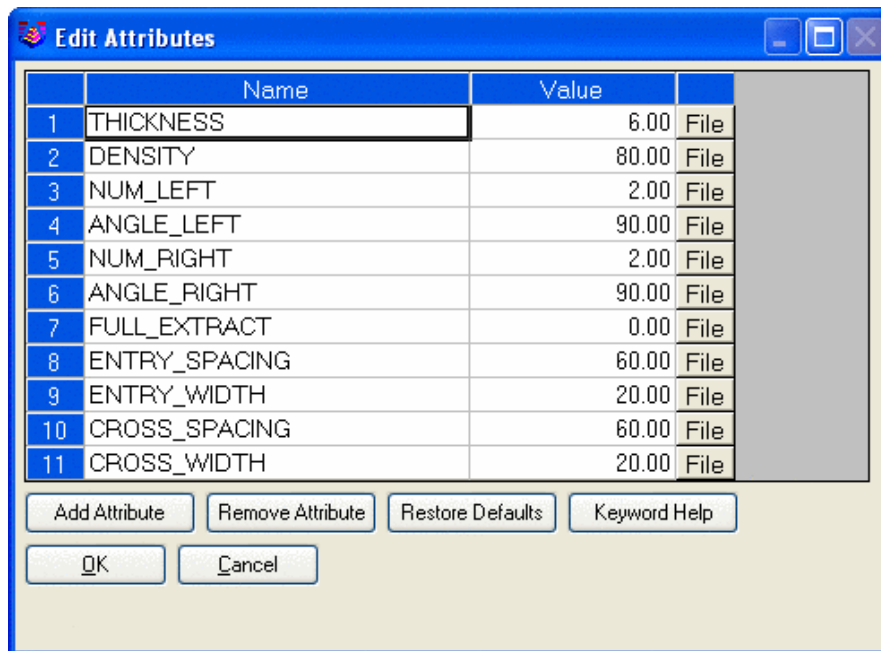
Retreat extraction: 0.0000

Owner: []

Advance Difficulty Factor: 1.0 [Select file]

Retreat Difficulty Factor: 1.0 [Select file]

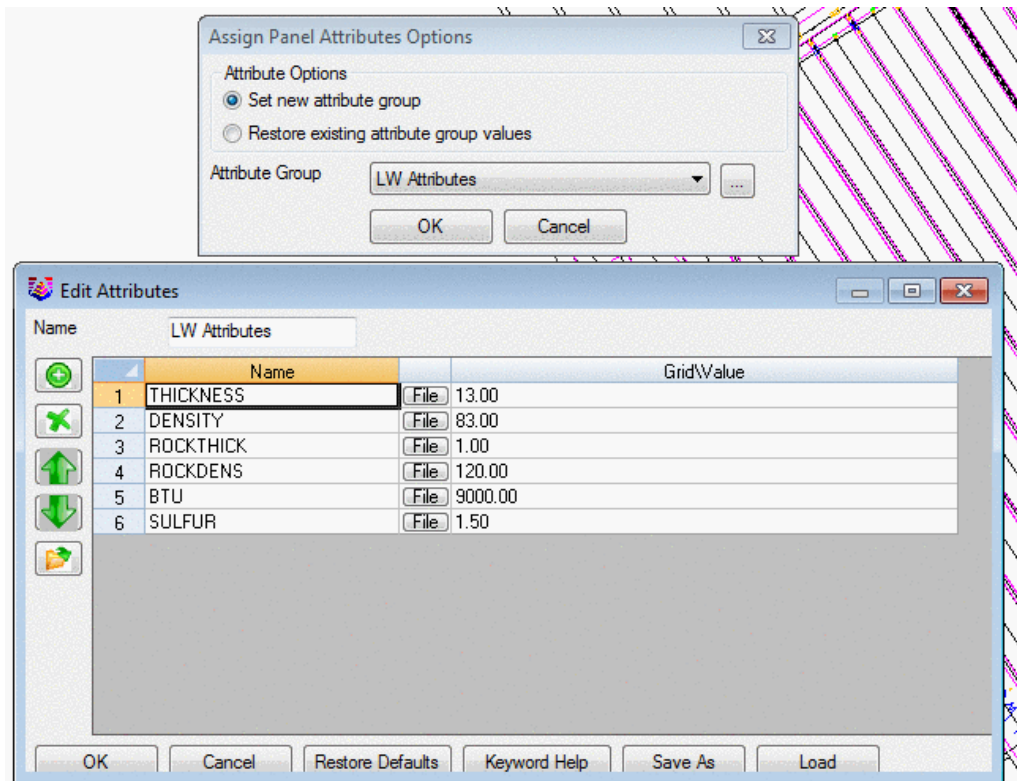
[OK] [Cancel]



Pulldown Menu Location: Underground
Keyboard Command: editpanel

Assign Panel Attributes

The command allows for selecting multiple panels and assigning a Panel Attribute Group to them all at once. A typical use for this is where there are different sets of attributes used in different panels, such as a continuous miner and a longwall. These two different attribute groups must be defined first, in the Timing Project Manager, then they appear on the dropdown list of Attribute Groups to choose from. Choosing the button next to the group dropdown will bring up the Edit Attributes screen, where changes can be made if needed. Once the group is selected in the dropdown, choose OK and select all the panels to assign that group to.



Prompts

Select panel polylines to have attribute group assigned.

Select objects: 1 found, 14 total

Select objects:

Saving changes to project database...

Pulldown Menu Location: Underground

Keyboard Command: setpanelattr

Reset Panel Attributes

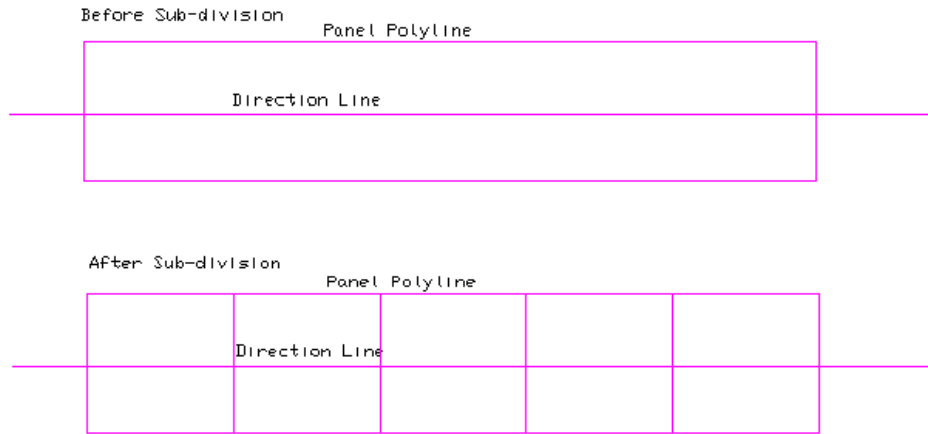
Whenever the Edit Panel Attributes function is used to make changes in attributes for the particular panel, a completely separate set of attributes is created and stored for this panel. From that point on, the changes in default panel attributes will not affect this edited panel. To revert these changes and make the panel use the default set of attributes, use the Reset Panel Attributes function.

Pulldown Menu Location: AdvMine

Keyboard Command: resetpanelattr

Subdivide Panel

The simple way to specify different coal qualities and/or other panel properties is to break a large panel into a number of smaller sections. These sections are later associated with a single panel within the Place Panel routine. This routine divides panels into a given number of smaller ones. The slower alternative to this method, giving better precision, is to use grids instead of average values for panel attributes.



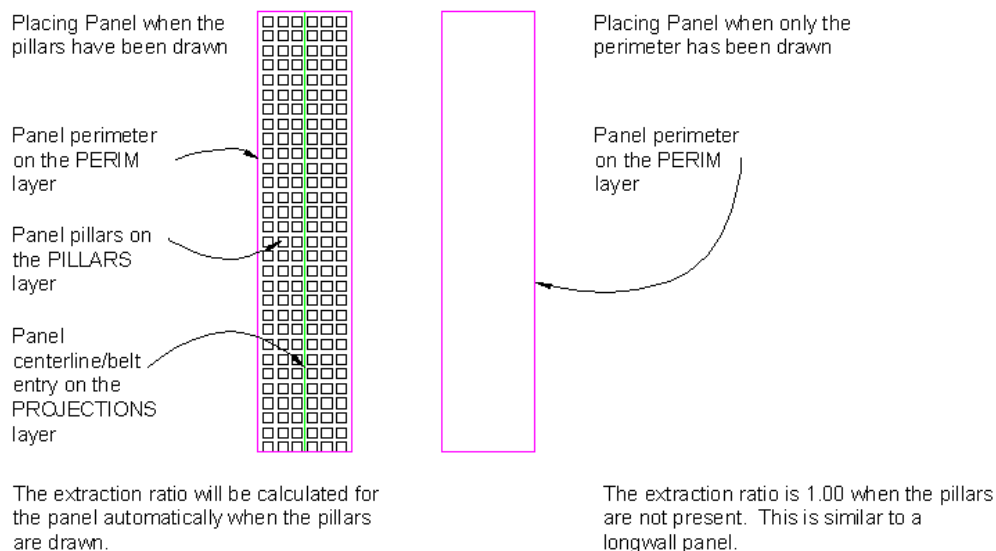
Note: Panel is NOT placed yet, just sub-divided into 5 equal parts along the direction Polyline.

Pulldown Menu Location: AdvMine
Keyboard Command: subdivide

Place Panel

The Place Panel function requires a panel perimeter drawn in the PERIM layer. If you draw the panel with Advanced Projections in the Basic Mining Module and use the option to draw the pillars, the program will automatically calculate the extraction ratio (as shown in the example panel on the left). The extraction ratio can be entered into the Edit Section dialog box, if you start with the perimeter only drawn in the PERIM layer (as shown in the example panel outline on the right).

Place Panel Pre-Requisite Linework Options



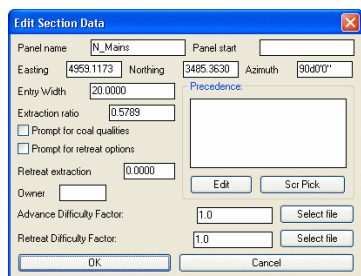
The underground timing routine operates with a tree-like structure of the panels built using a centerlines network. The new panel may be added or an existing panel (i.e. created using Advanced Projections) may be attached to the existing network of centerlines using this function.

Each panel may consist of a number of sections, where each section is characterized by the uniformity in the section

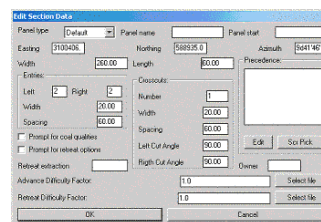
attributes such as owner, coal thickness and qualities, complexity of mining, extraction ratio, azimuth and length. Therefore each panel is represented by a centerline and a set of panel perimeter polylines, one for every section.

Due to the tree-like structure of the Mine Plan, each new panel should branch from the existing panel. This allows for proper timing calculations and data management. The user is prompted for the starting point and direction of each section, with dimensions dependent on the existing perimeter polyline or on user input.

If an existing polyline is used to define the panel, the program evaluates the section length by intersecting the centerline with this polyline. Otherwise the sizes of the section are defined in the Edit Section Dialog. The appearance of the dialog depends on the way the section is defined. Both kinds of dialogs are shown here.



Dialog for pre-drawn panel



Dialog for panel defined on the fly

User has a number of options to define a direction of the panel. The following methods are available:

Perim: The direction of panel is defined by the direction of the first segment of polyline.

Text: Direction is from "branch off" point to insertion point of the text found inside the panel. The text must be in PANELNM layer and it defines panel name.

Pick: Direction of the panel is from "branch off" point to pick point.

Segment: Direction is defined by user-picked segment of the panel polyline.

Panel name: Identifies the panel for Underground Timing.

Difficulty factor: Represents the effectiveness of the mining equipment within the section. Values above 1.0 will cause a proportional slowdown of the production and below 1.0 will speed it up. This value may be specified using difficulty factor grid file by selecting file with *Select File* button or typing it in instead of the actual value. The run-time difficulty factor will then be evaluated for corresponding area.

Precedence: Defines whether development of the section may not start until all listed sections are completed. Precedence is especially important for the longwall sections, since production there is not possible until all headings are completely developed. The listbox shows panels which have to be developed before the section being edited. Since some sections may be defined after the current section, it is advisable to edit precedence later with Edit Panel function. The precedence can be set by either picking from the list, or screen picking.

Panel start: Defines fixed start date of the development. This field is optional and is ignored if the actual starting date is later than date specified.

To define attributes for this panel section (such as THICKNESS, DENSITY and etc. as explained in Define Panel Attributes function) select **Prompt for coal qualities** toggle.

In the case of using an existing perimeter, the extra unknown is **Extraction Ratio**, whereas in the other case the dimensions of the section and entry/crosscut spacings need to be defined. The extraction ratio will default to value calculated using pre-drawn pillars and panel perimeter.

The **Entry Width** for predefined panel is only used if the section is mined by the piece of equipment with the linear feet of advance production rate specification.

Retreat Extraction: Defined as ratio of area of all coal extracted on retreat to the area of whole panel. Therefore if extra side coal is picked up on retreat this value combined with advance extraction ratio, exceeds 1.0. To simplify

estimation **Prompt for retreat options** may be used to calculate the retreat extraction for given the extraction polyline.

Retreat Difficulty Factor: Similar to Difficulty Factor except it applies on the retreat stage.

Users who are creating an extensive mine plan, may prefer to use a streamlined Quick-Place mode to create layout and later on use Edit Panel Dimensions to modify panel properties.

Prompts

Use Quick-Place mode (Yes/<No>)? *press Enter*

Start new Mine Plan (Yes/<No>)? *press Enter* Choose Yes to start Mine Plan.

Select an entry to branch from: *pick the panel to branch from at desired location*

Enter the azimuth for the segment:

Select a panel polyline or press Enter for dialog:

Edit Section Dialog

Enter the azimuth for the segment:

Select a panel polyline or press Enter for dialog:

Edit Section Dialog

Pulldown Menu Location: AdvMine

Keyboard Command: place_panel

Pick & Place Panel

This routine is parallel to the Place Panel and Underground Timing functions. The timing through the panels is performed as soon as the user picks the panel perimeter. The sequence of picking and all the associated parameters necessary for "replay" are stored in the same form as with the Timing routine. That allows you to modify the mine plan, equipment rates and/or calendar and obtain an updated plan without having to redo the sequence.

The routine goes through the following steps:

- Prompt user for next panel to mine in.
- Check if the picked panel is a part of the mine plan already. Then prompts to pick an existing mine plan or to start new mine plan. The existing plan is loaded in and rerun to obtain the information about the availability of the equipment.
- Bring up the dialog and prompt the user for various parameters and the part of the panel to be mined. The proposed mining is previewed along with the dates and tonnages associated with the proposed segment.
- When editing is finished the colored and labeled timing blocks are displayed in the user-defined style and the whole process is repeated for the next panel.

The starting date of mining in the picked polyline depends on three dates: if the optional panel start date is specified and the unit is available this date is used. Otherwise if the unit is tied up within some other panel, the starting date is the date when the unit finished the last panel offset by the specified moving delay. Lastly if the unit has not been used in the mine plan at all, the start date for the panel is the same date as for the whole mine plan.

Define Timing Step

Panel Settings

Panel name: 99RT Panel start: Owner:

Starting Date: 1/1/98 Number of shifts: 2

Entry Width: 20.0 Panel attributes: Prompt for retreat options

Advance Diff. Factor: 1.0 Select file: Extraction: 0.59

Retreat Diff. Factor: 1.0 Select file: Extraction: 0.41

Equip. Name: CM#1 Development Results

Advance Retreat

Portion To Mine

Whole perimeter

Distance: 641.62336

Period: Month

To Date: 7/19/98

Moving time (shifts): 0

Test Report Options OK Cancel

Date Start	03/03/00
Date Finish	04/08/00
Crew	CM#1
Panel Name	99RT
Coal (tons)	32735.560000
Rock (tons)	11159.850000
Area mined	252199.999998
Linear adv.	7439.900000
Difficulty	1.000000
Density	80.000000

The portion of the panel to be mined is determined by the selection made:

Whole perimeter The panel will be mined to the end of the panel or to the next pin point if present (retreat).

Distance Mining will advance a given distance along the centerline past the last mined point. A pin point will be added at the end of new segment. Use of this option on retreat is associated with automatic reassignment of advance development, followed by rerun of timing plan (advance and retreat use the same set of pin points).

Period and **To Date** are similar to Distance, since the program will automatically calculate a distance corresponding to given date.

When the **Test** button is used the portion to be mined is highlighted in the drawing and the timing and quantities results are displayed in the dialog.

The amount of time necessary to mine given portion of the panel depends on the following factors:

Equipment production rate (use Define Equipment to modify)

Coal and Rock Attributes (Define Panel Attributes, Edit Panel Attributes)

Difficulty Factor (may be specified as value or grid on per panel basis or using Define Panel Attributes and by placing a text labels on certain layer within the panels)

Extraction Ratio (pre-calculated based on pillars draw inside the panel, but may be modified) Equipment Calendar assignments.

Report Options are the same as these used in Underground and Surface Timing Routines. The final results for the portion laid out during current session are displayed in the user-defined form using the Report Viewer.

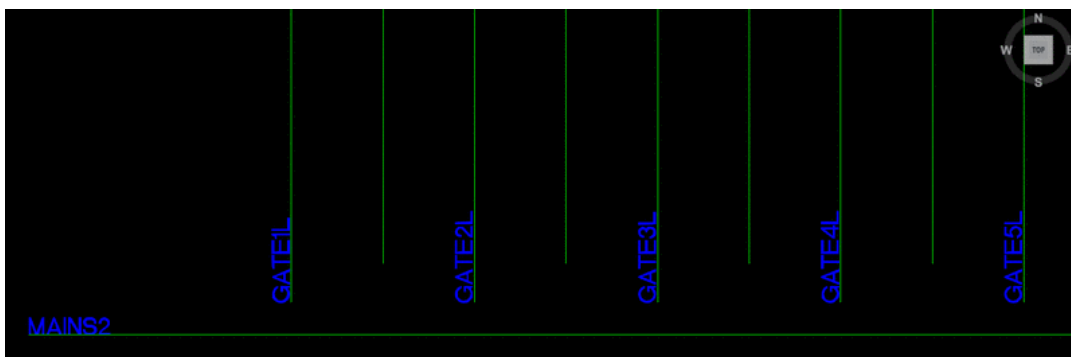
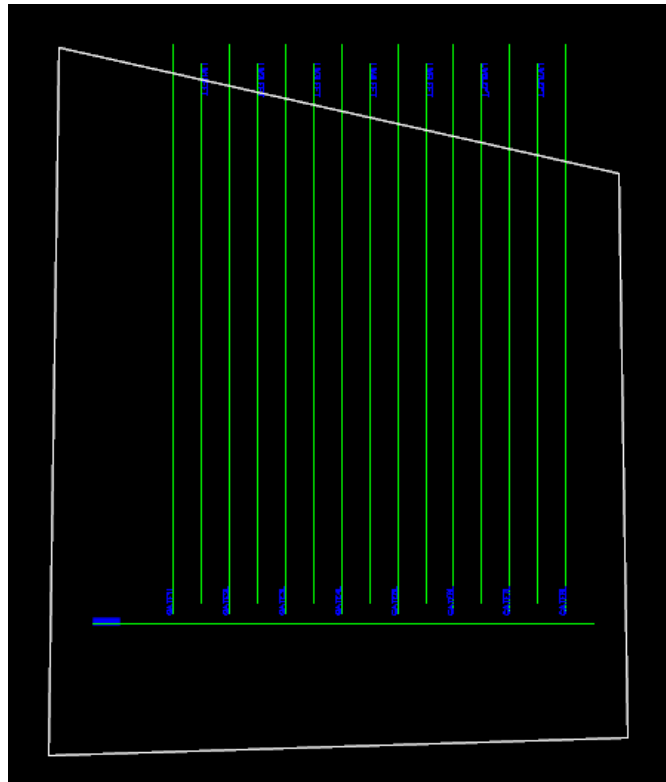
The generated mining plan (the system of green centerlines) may be modified and rescheduled using Underground Timing routine.

Pulldown Menu Location: UnderGnd

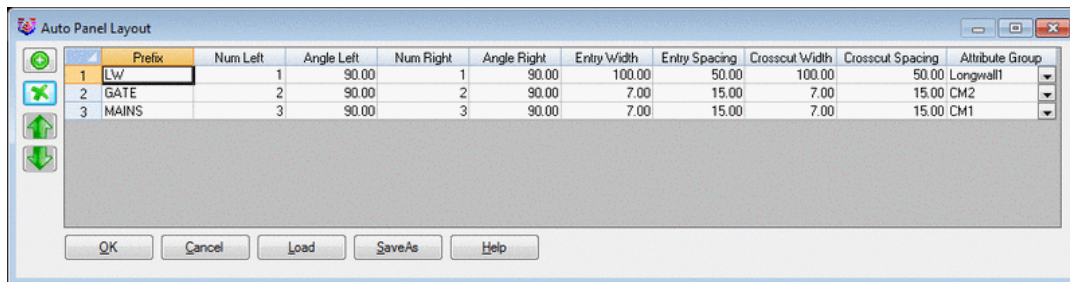
Keyboard Command: pick_panel

Auto Panel Layout

This command takes center lines and text in the drawing, and creates named underground panels based on the names, offset distances, and outer perimeter. The prerequisites are the center lines and text for each panel, and they must be drawn on the proper layers. The center lines need to be on the layer MINEPLAN, and the text needs to be on the layer PANELNM.



Starting the command brings up the following dialog. Shown here are examples on three prefixes that will be used on these lines and text. The prefixes listed must be in the PANELNM text in the drawing, to match up the layout. The width and spacing (shown here in metric) are used to draw the panel perimeters and assign the names. The layout of the center lines in the MINEPLAN layer is the tricky part that takes some time to calculate and draw them correctly offset, to allow for the panels to draw around them. Keep in mind the barriers, and spacing between panels needs to be accounted for too. The Attribute Group will automatically assign these Panel Attributes from the MPD timing file to the specified panels when the panels are drawn.



After choosing OK, the boundary polyline is prompted for. This is necessary to clip or stop the panels as they approach the boundary.

Prompts

Select boundary polylines or press Enter for none:

Select objects: 1 found

Select panel center lines (MINEPLAN layer):

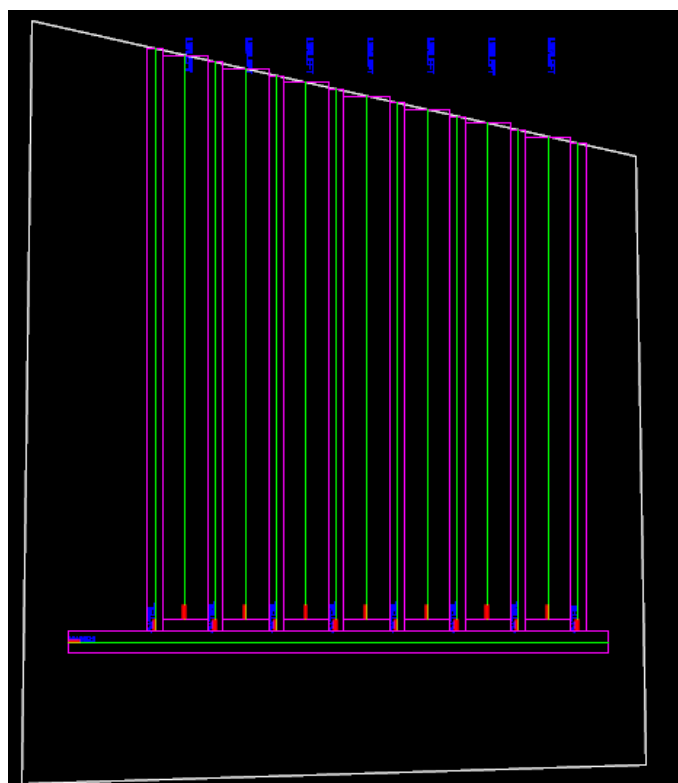
Select objects: Specify opposite corner: 16 found

Select panel names (PANELNM layer):

Select objects: Specify opposite corner: 16 found

16 panels created.

The resulting panels are shown here, and only extend up to the boundary polyline that represents the mining limits for the panels.



Pulldown Menu Location: Underground

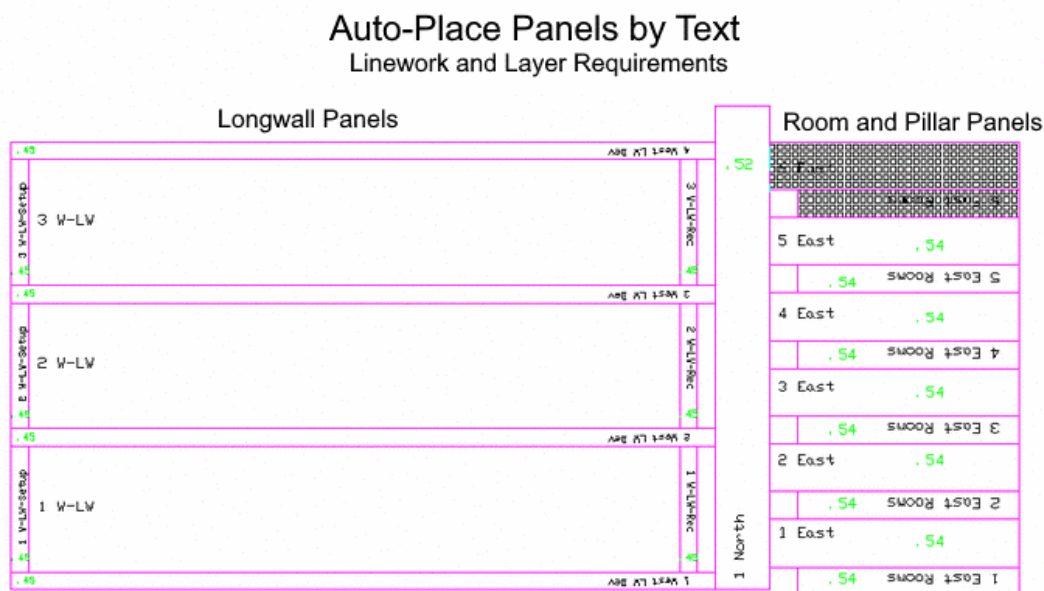
Keyboard Command: auto_panel

Related Commands: Place Panel, Pick and Place, Auto Place Panel by Text

Auto Place Panel by Text

This is the best method to place many panels fast. The panel is directed by text orientation with centerline going through text insertion point. The text itself becomes a panel name and it has to be in layer PANELNM to be found. User can specify retreat options, difficulty factors and entry width to be used for all panels before running command. To place the text in the panels easily use Draw Sequential Numbers, found under the Draw dropdown menu.

All panels created in one run become a single connected mine plan with no equipment assigned.



Required Input

Perimeter on PERIM Layer

Panel name on PANELNM Layer

The linework must be closed polylines. Each panel must have its own closed polyline (including longwall panels). The layer names do not have to be written in all caps. The text insert points must be inside the panel polylines.

Optional Input

*Advance Extraction Ratio on EXTRACTION Layer

*Retreat Extraction Ratio on RET_EXTRACT Layer

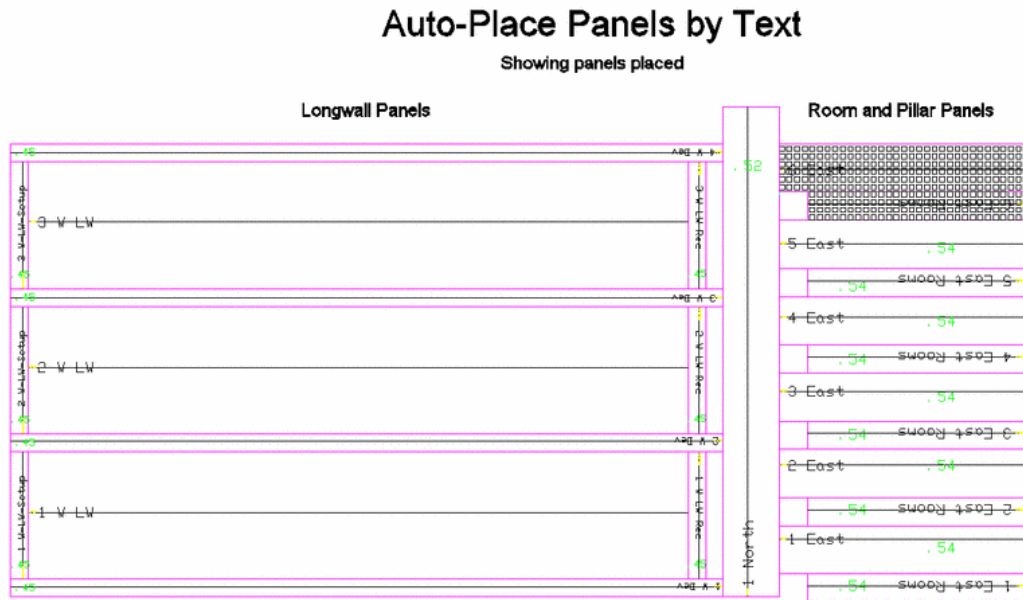
**Advance Difficulty Factor on DIFFICULTY Layer

**Retreat Difficulty Factor on RET_DIFF Layer

*The Extraction Ratio is not updated from the text on the EXTRACTION layer until running the Recalculate Extraction command on the Underground Pulldown Menu. There can only be one Extraction Ratio in a panel. Note that the Extraction Ratio will be automatically calculated if the pillars are drawn.

**There can be more than one Difficulty and Retreat Difficulty Factor per panel if the text is separated by Pin Points.

When you note the prompts below for entry width you will want to place the "longwall panels" and "room and pillar" works in two steps in order to get the correct advance distances reported. For this example the room and pillar works are on 20ft width, and the longwall width is 1,000 feet. Placing them in two steps will create two mine plans that can be connected using the Connect Mine Plans option.



Prompts

Assign zero retreat extraction [Yes/<No>]? Y

Advance difficulty <1.0> :

Entry width <20.0> : Use the panel width for longwalls, entry width for room and pillar panels.

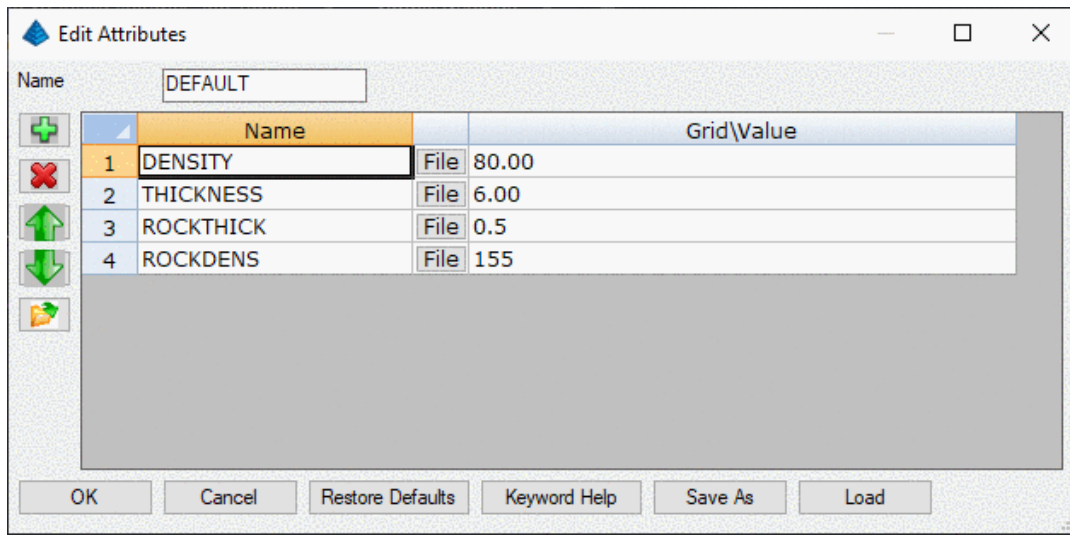
Pulldown Menu Location: UnderGround

Keyboard Command: auto_place

Edit Panel

The function is identical to the section definition portion of Place Panel function. It is used to edit properties of the section of the panel after the panel has been completed. The precedence can be set by either picking from the list, or screen picking.

- **Attribute Group:** Specifies the attribute group for the panel. Attribute groups may be configured in the Timing Project Manager.
- **Panel name:** This is a required field.
- **Panel start:** This is an optional field. The Underground Timing routine have an option for start date that usually makes this field unnecessary. This field can delay the start of a panel independently of the plan start date.
- **Easting:** The easting coordinate value of the first picked point when the panel was placed.
- **Northing:** The northing coordinate value of the first picked point when the panel was placed.
- **Azimuth:** The azimuth from the first to the second picked points when the panel was placed.
- **Entry Width:** This is used to calculate face feet/meters of advance. This should be set to the width of the panel when working with longwall panels to get the correct advance down the panel. With regards to the scheduling/timing command, this value has a direct impact on the linear feet of advance for the panel. Note that the linear feet of advance accounts for the length of all cross cuts and entries in the panel. The linear feet of advance for a panel is calculated as the Mined Area divided by the Entry Width. For a panel covering 60,000 sq ft, an extraction ration of 0.6, and an entry width of 20, the linear feet of advance in the panel would be calculated as $(60,000 * 0.6) / 20 = 1800$. Please note that this calculation assumes that all pillars in panel have similar dimensions).
- **Prompt for coal qualities:** A check in this box brings up the Define panel Attributes dialog box after clicking OK in the main dialog. This is the same dialog that is displayed when defining attribute groups in the Timing Project Manager.



- **Prompt for retreat options:** This option will allow you to select an inclusion polyline for the retreat area of the panel. When this option is enabled, you will receive three command line prompts after clicking OK in the main dialog. An example panel with a separate retreat polyline is shown below.

Pick a polyline representing a shape of the retreat area: *During this prompt, you may select an inclusion polyline for the retreat area of the panel.*

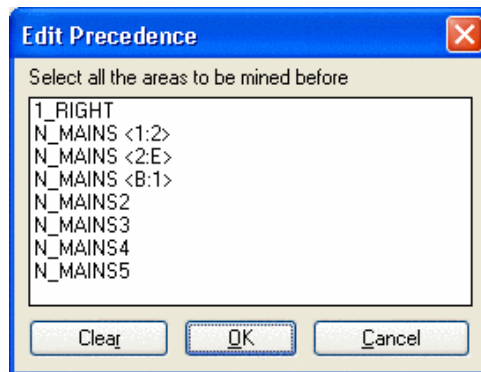
Enter the extraction ratio for remaining pillars: *The value entered here defines the portion of the pillars that will be mined. For example, if the pillar will be entirely mined out, enter a value of 1. If only half of the pillar will be mined, enter a value of 0.5.*

Enter the extraction ratio for KEY outside the panel: *The value entered here defines the extraction ratio for the portion of the retreat panel outside of the main panel area. If all of the key material in this area will be removed, enter a value of 1. If half of the material in this area will be mined, enter a value of 0.5.*

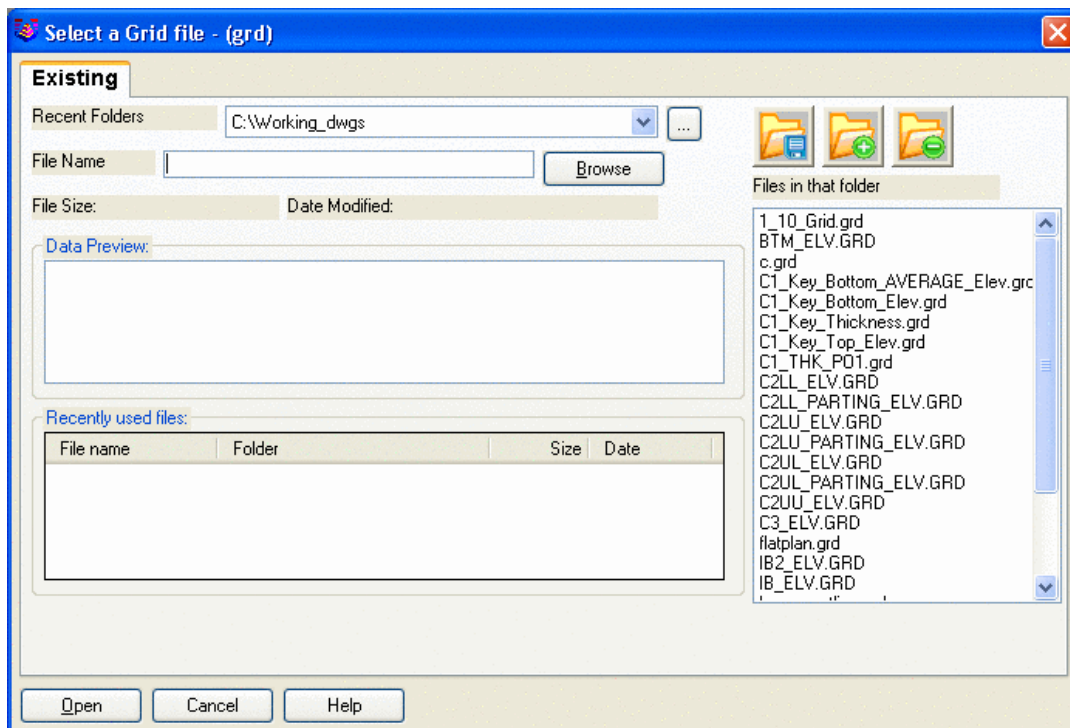
In the below example, we have a panel with an area 60,000 ft². The orange polyline represents the retreat area (10 of the 15 pillars will be retreat mined, and a portion of the rib will also be retreat mined). When prompted for the retreat options, the orange polyline is selected. The extraction ratio for the remaining pillars was set to 0.8 (meaning 80% of the pillars will be mined), and the extraction ratio for Key outside the panel area is was set to 0.6. The table below the dialog shows the effective pillar area to be mined and the effective outside area to be mined. These areas combined give the total retreat area to be mined, which is then divided by the Panel Area to give the Retreat Extraction Ratio.

Area of ten pillars inside retreat polyline, ft ²	16,000	Panel Area, ft ²	60,000
Extraction Ratio for Remaining Pillars	0.80	Retreat Area Mined, ft ²	19,520
Pillar Area mined, ft ²	12,800	Retreat Extraction Ratio	0.3253
Area outside Panel, ft ²	11,200		
Extraction Ratio for Key Outside the Panel	0.60		
Area mined outside panel, ft ²	6,720		

- **Retreat Extraction:** This value represents the area of the panel to be mined on retreat. This is the retreat area divided by the panel area.
- **Owner:** A single owner's name can be assigned to the panel.
- **Level Layer Suffix:** This is only needed when multi-level mining is to be used. Entering "_LEVEL2" for this field would tell the program that this panel is part of the second level.
- **Precedence:** This box gives the opportunity to set fixed scheduling relationships. Any panels selected must be completed before this panel can start.



- **Advance Difficulty Factor:** A value or a grid can be used to provide difficulty factor input for advance mining in the panel.
- **Retreat Difficulty Factor:** A second value or grid can be specified for retreat mining independent of the advance mining difficulty rate.



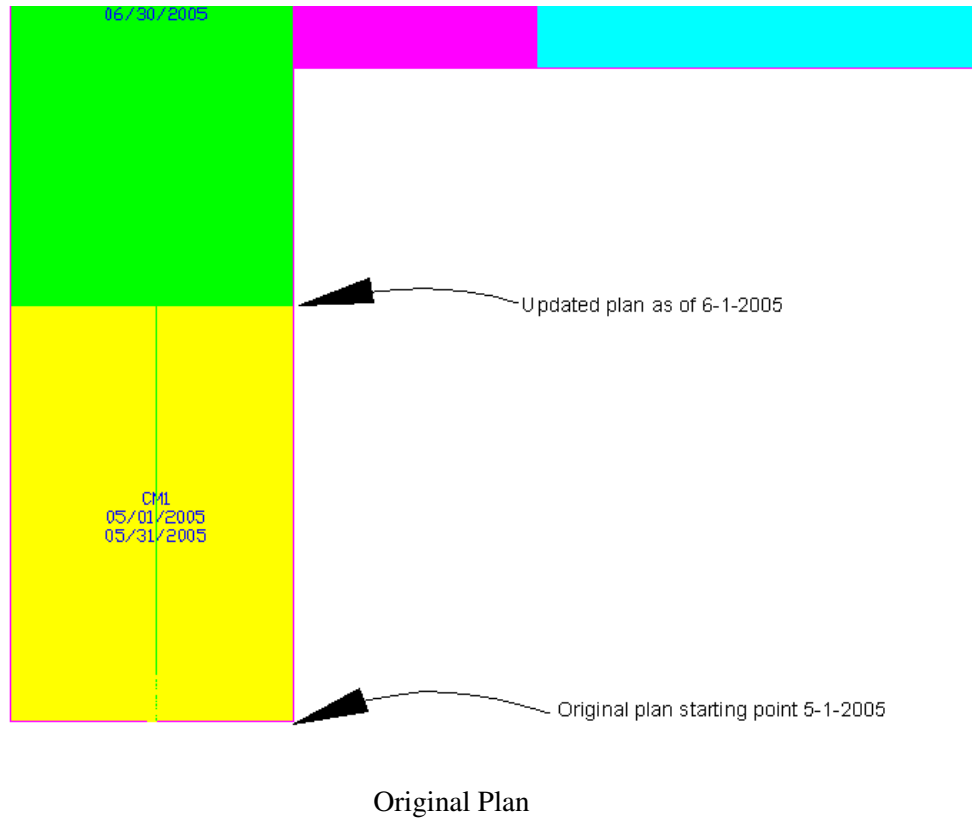
Pulldown Menu Location: UnderGround

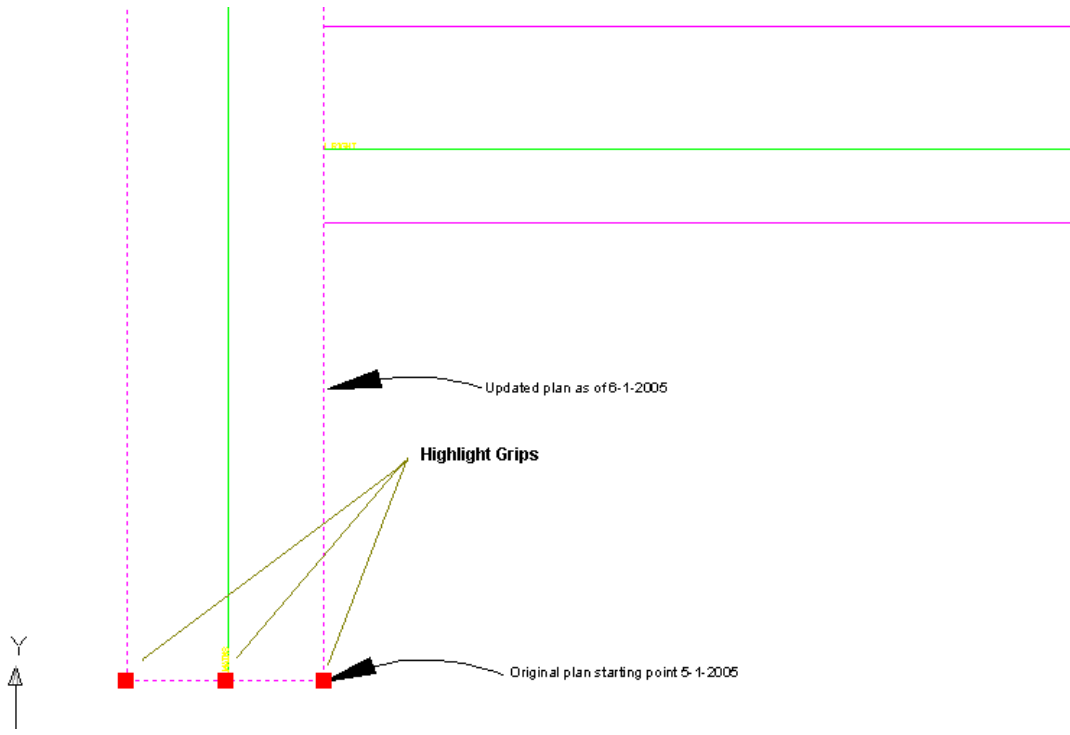
Keyboard Command: edit_panel

Update Edited Panel

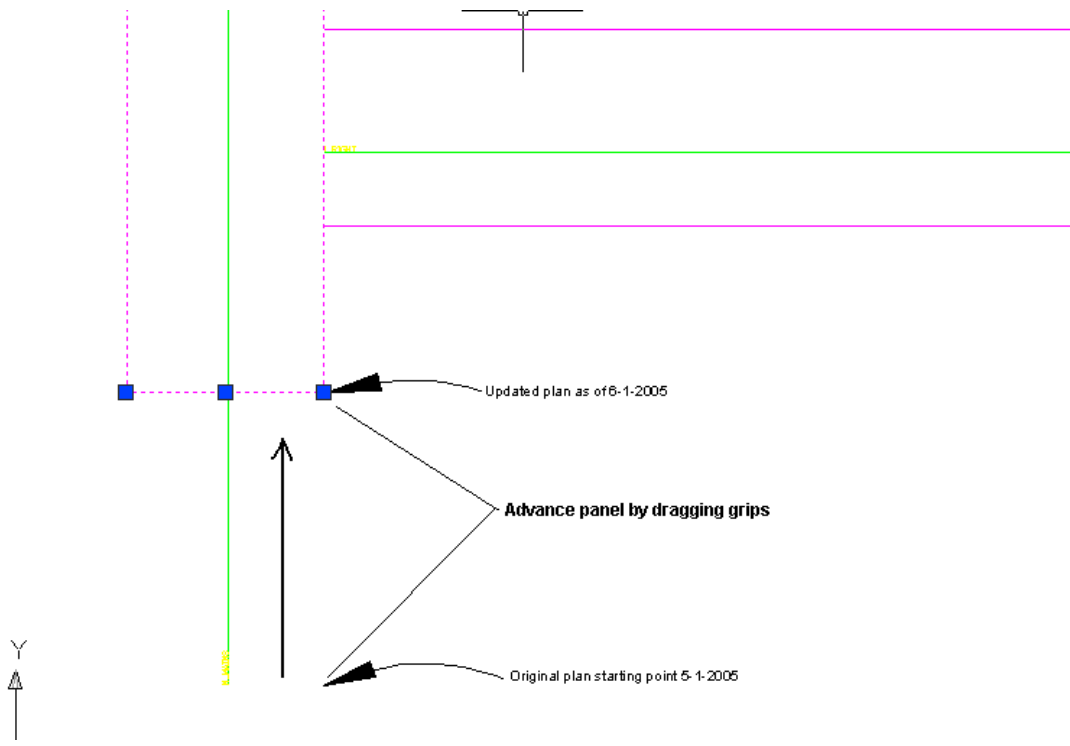
The mining plan created once should undergo changes as mining progresses. The user may modify, replace or erase panel perimeters in correspondence with changes in the existing mine and then use this command to update the mine plan data. Do not modify or erase the remaining linework, since this linework and internal plan data will be updated to match changes in the associated perimeters. This command should also be used every time the panel polyline is moved, replaced or edited for any other reason.

Naturally the mine starting date should be moved forward each time plan is updated, so that timing of the remaining plan remains correct.

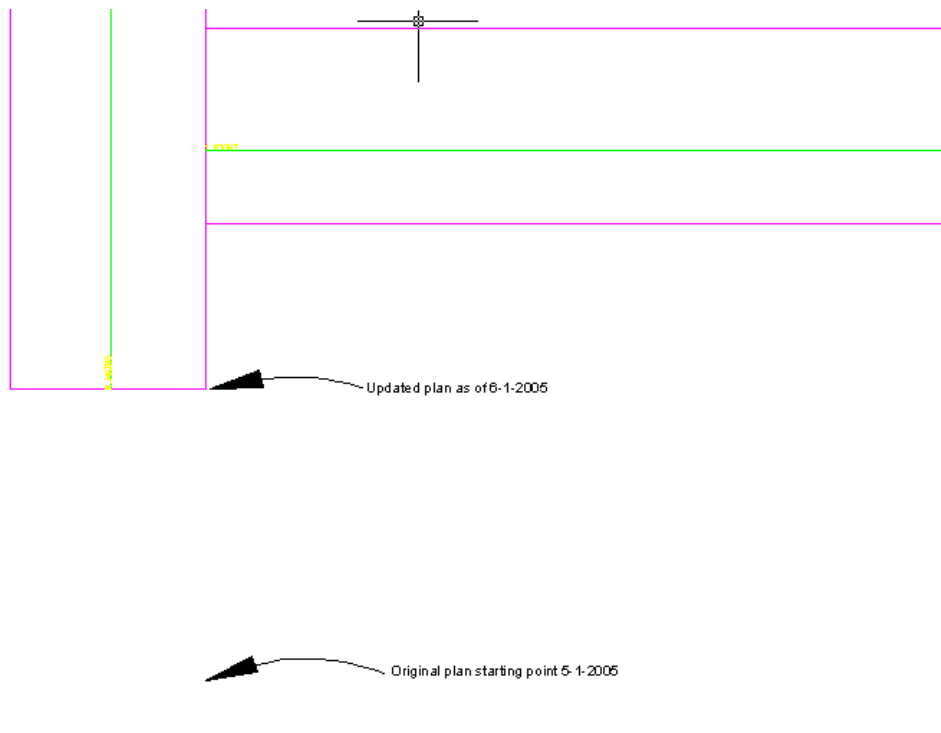




Highlight grips



Drag grips



Updated panel

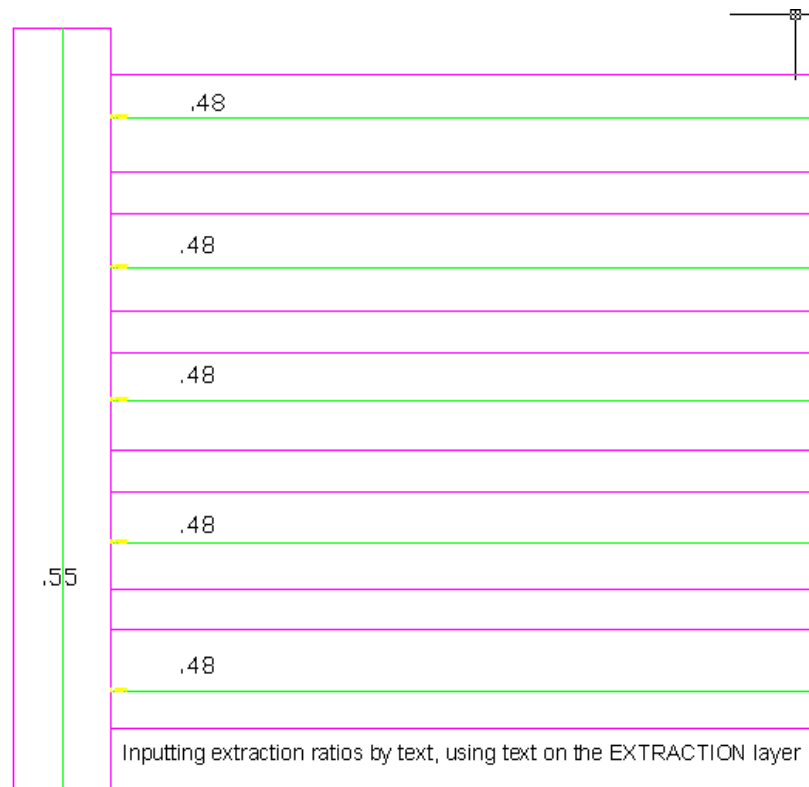
Once the panel linework is updated, re-run the Underground Timing updating the plan start.

Pulldown Menu Location: UnderGnd

Keyboard Command: update_plan

Recalculate Extraction

When changes are made to the panels that result in a change in the extraction ratio this function must be run to update the extraction ratio in the panels. This also must be run after placing panels using Auto-Place Panels by text. If the text values for extraction are used to input the extraction ratio for advance or retreat extraction ratios this function must be run. Text for the extraction ratio must be put on the CAD layer of EXTRACTION. Two other layer names that will be recognized by this command to update the panels are ATTR_GROUP to assign a certain Panel Attribute Group to certain panels (such as longwall vs. continuous miner), and ENTRY_WIDTH to set that to the value of the text in that layer.



Prompts

Select all panel polylines to have extraction ratio updated:

Select objects: Specify opposite corner: 6 found

Select objects:

Polyline offset amount <0.0>: *press Enter*

Retreat recovery <1.0>: 0

Panel 5 RIGHT has extraction of 0.48000

Panel 4 RIGHT has extraction of 0.48000

Panel 3 RIGHT has extraction of 0.48000

Panel 2 RIGHT has extraction of 0.48000

Panel 1 RIGHT has extraction of 0.48000

Panel N_MAINS has extraction of 0.55000

Pulldown Menu Location: AdvMine

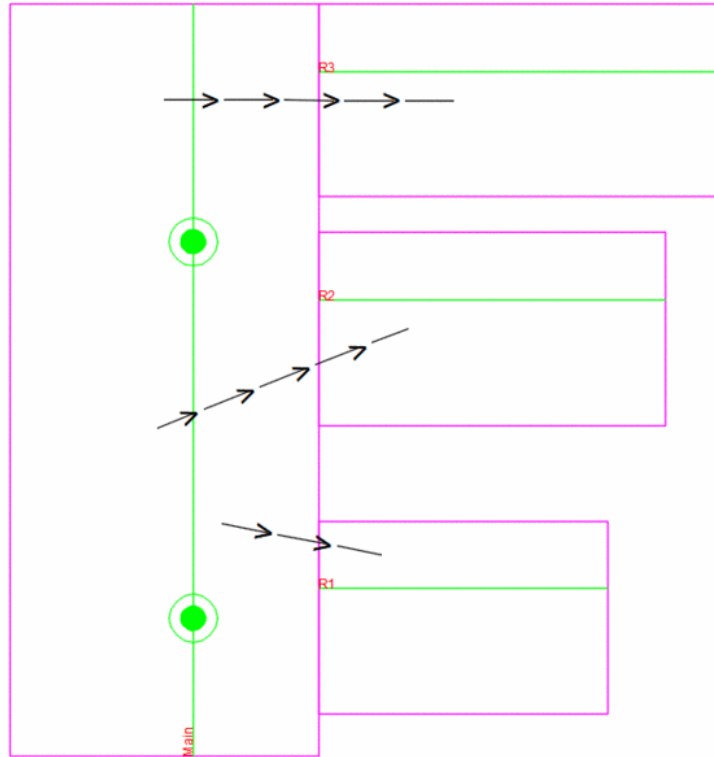
Keyboard Command: redoextract

Panel Precedence By Polylines

This command allows you to quickly assign precedence to mine panels by drafting directional polylines which indicate the direction of precedence. Before the command is executed, a polyline must be drawn on the PRECEDENCE layer. The vertices of the polyline must exist inside the panels, as shown below. It is important to note that a vertex must exist inside each panel perimeter (a polyline with 2 vertices will only modify 2 panels, even if it crosses multiple panels).

When the command is executed, you will be prompted to select the panels to modify, followed by the Precedence polylines. You will then be prompted if the precedence panel is Advance or Retreat, followed by a prompt to assign the precedence rule to the Advance or the Retreat of the target panel.

An example Setup is show below. In all three cases, the Main Panel must be mined before R1, R2, and R3.



The resulting Precedence rule for Panel R3 is shown below. In this example, we assigned the Advance of panel R3 <3:E> as a precedence rule to the Advance of Panel R1.

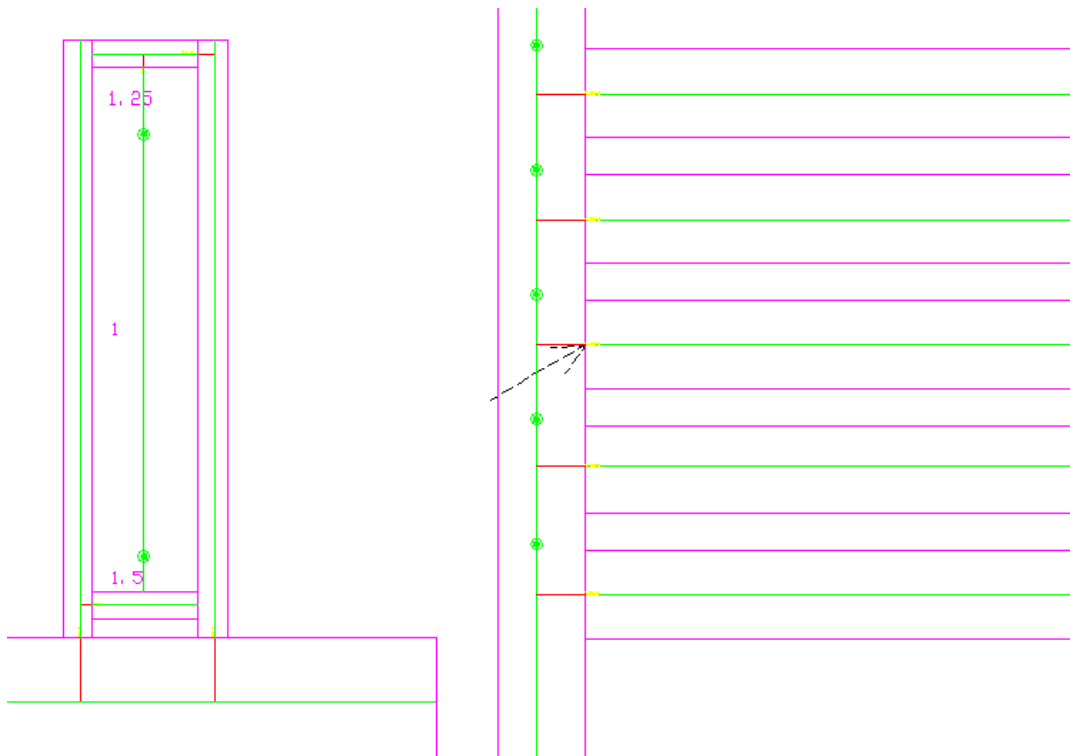
Pulldown Menu Location: Underground

Keyboard Command: pl2panelprec

Prerequisite: Mine Panels and directional polyline(s) on the PRECEDENCE layer

Find Panel

Use this command to find a panel with a given name in a complex mine plan. Type the name of the panel in at the command prompt and the ghosted dashed arrow will appear in the center of the screen pointing at the requested panel.



Pulldown Menu Location: AdvMine

Keyboard Command: find_panel

Remove Panel

The mine plan consists of visible linework and internal data, including cross-references, containing all data necessary for timing. For this reason a simple deletion of the linework of an unwanted panel is not enough for removing all references to it and is likely to corrupt whole mine plan.

Use the Remove Panel command to remove all the linework and clean up internal data related to panel being removed and all its children panels.

In case if user just wants to remove a panel, but keep its children in the mine plan, remove a panel perimeter polyline and then use Update Edited Panel to tell it what that panel is completed.

Pulldown Menu Location: AdvMine

Keyboard Command: delete_panel

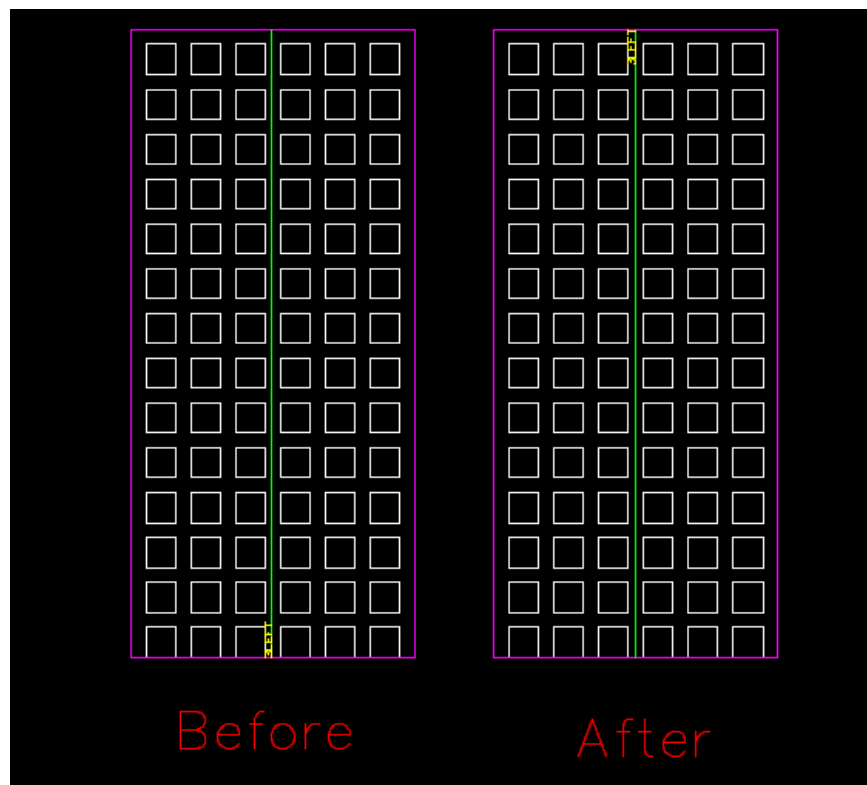
Reverse Panel

This command reverses the direction of mining within a panel. The panel must still be mined from the other end after it is exposed, otherwise that could introduce a problem with the mining sequence.

Prompts

Select a part of the mine plan to be reversed:

Select objects:



Pulldown Menu Location: Underground

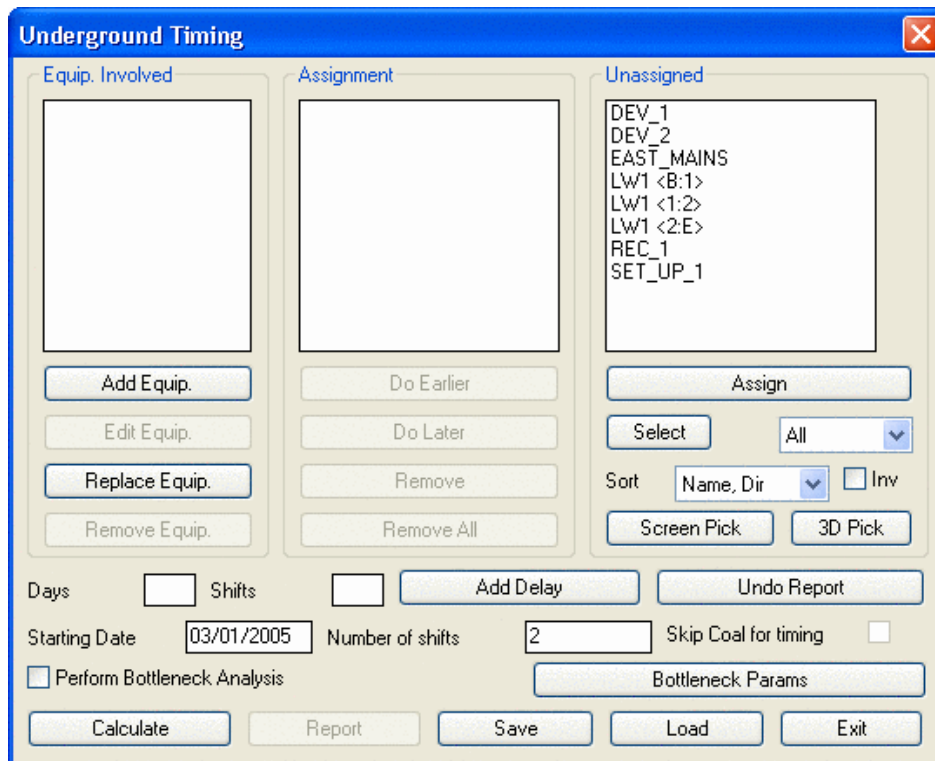
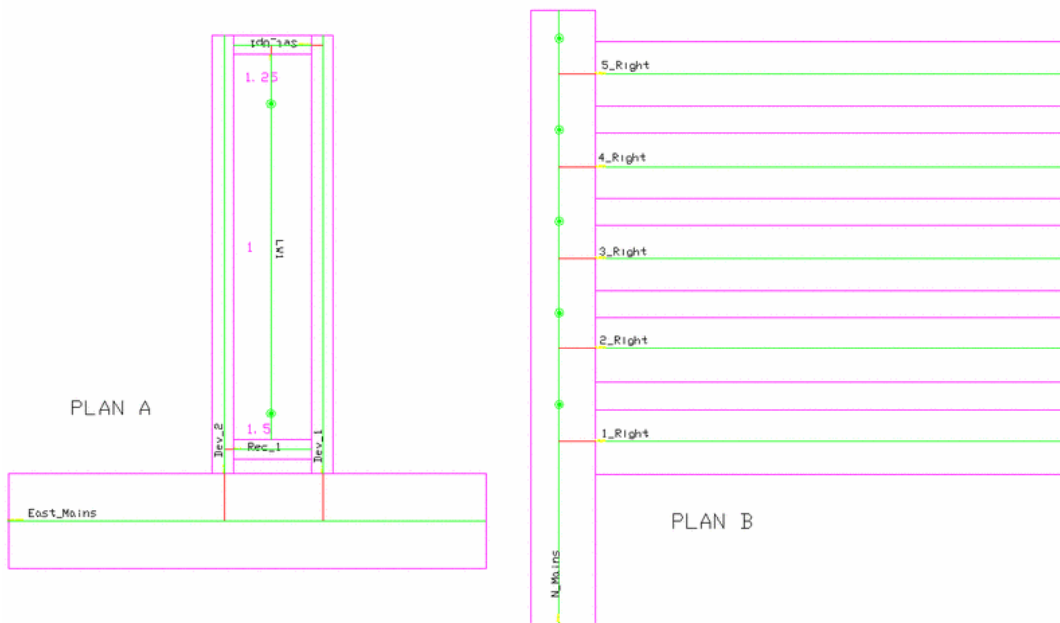
Keyboard Command: reverse_panel

Related Commands: Edit Panel, Remove Panel, Update Edited Panel

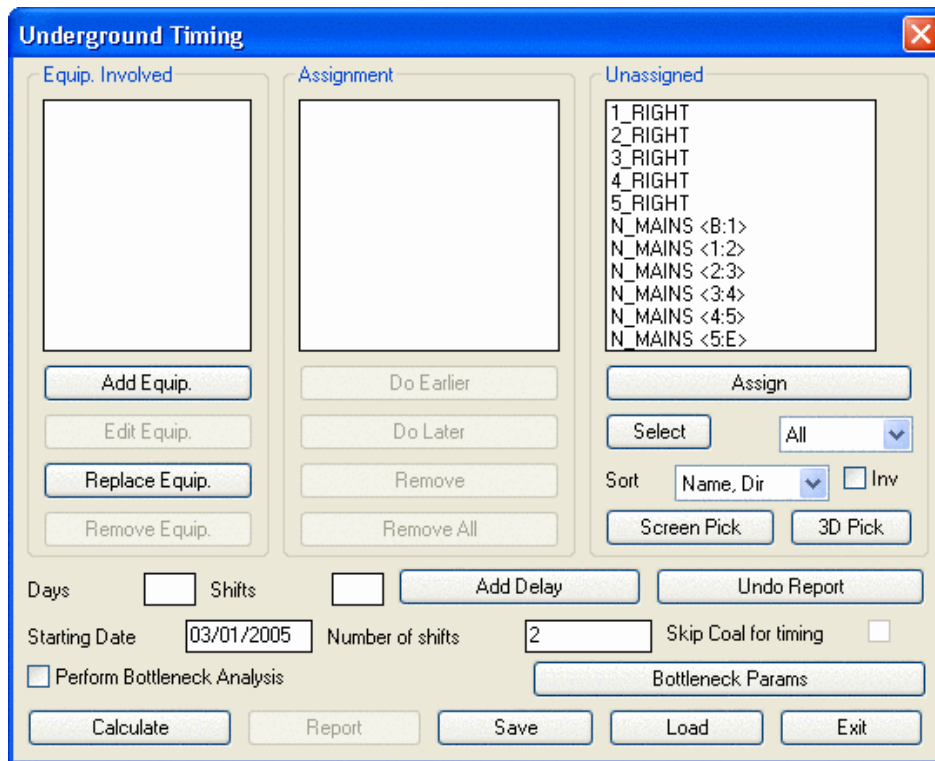
Connect Mine Plans

Two mine plans which are located in two separate parts of the mine but are sharing the same equipment or just wanted to appear on the same time plan, may be connected into a single mine plan using this command. The only difference between joint mine plan versus conventional one is that mining progresses simultaneously in all connected plans, if equipment use permits that.

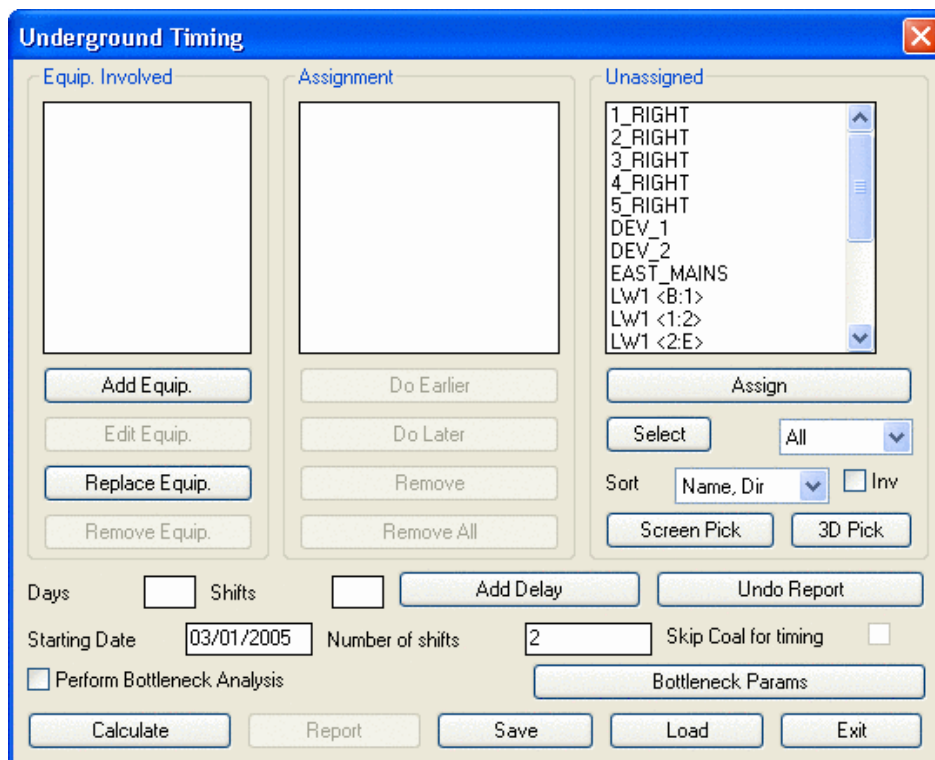
The following drawing contains two separate mine plans: Plan A and Plan B.



Plan A panels are shown when the Underground Timing function is selected. (Before connecting)



Panel B panels are shown when the underground timing option is selected and you pick on the Plan B panels.
(Before connecting)

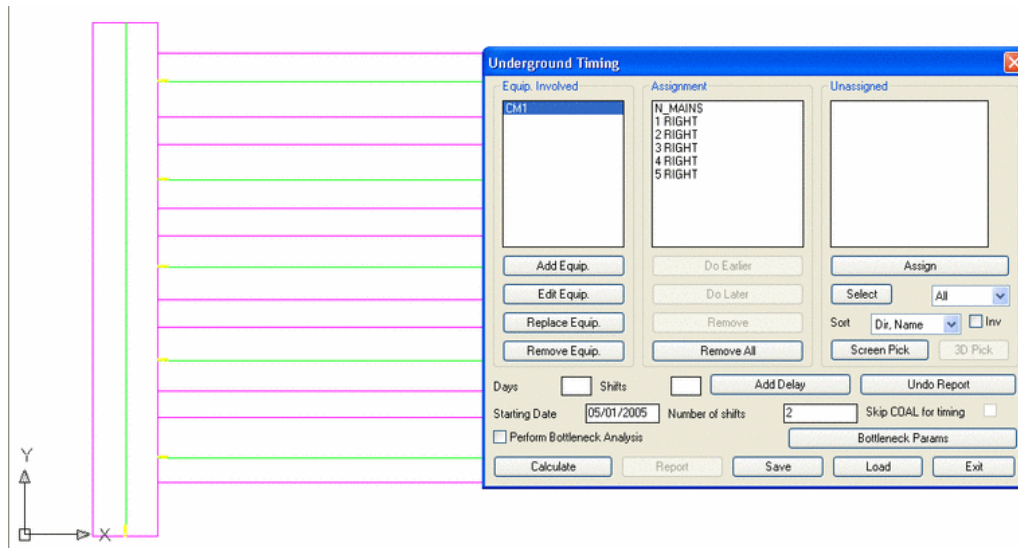


After connecting Plans A & B the list of available panels is composed of both lists.

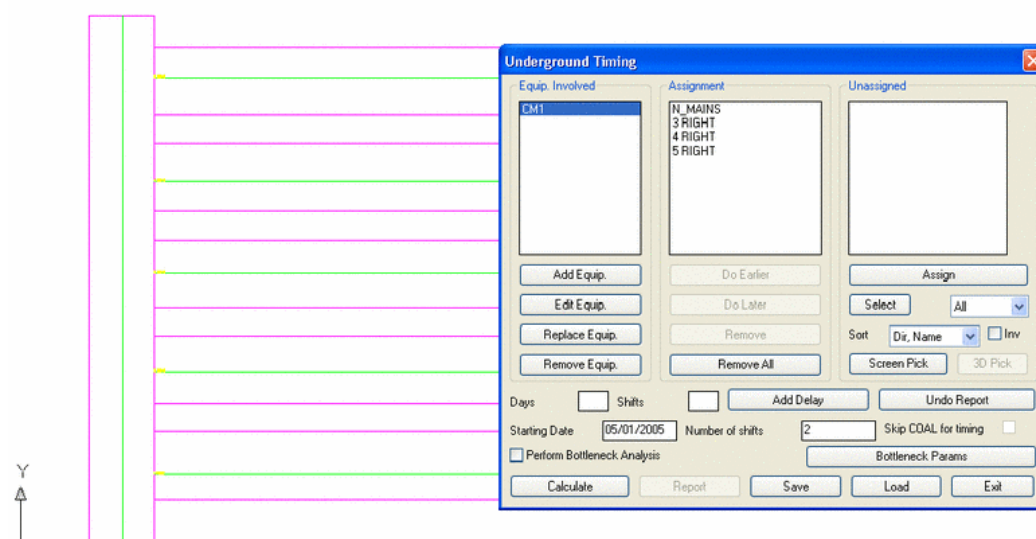
Pulldown Menu Location: AdvMine
Keyboard Command: connect_plans

Disconnect Mineplans

A mine plan is a group of placed panels on a common list grouped together for scheduling. Disconnect Mine Plans is used to break a mine plan into two or more groups of placed panels. To select a mine plan pick on the green backbone lines of the placed panels.



After disconnecting 1 Right and 2 Right panels



List after disconnecting 1 Right and 2 Right

Prompts

Select a part of the mine plan to be disconnected: *select backbone polyline*

Select a part of the mine plan to connect to or press Enter to leave disconnected: *press Enter*

Pulldown Menu Location: AdvMine

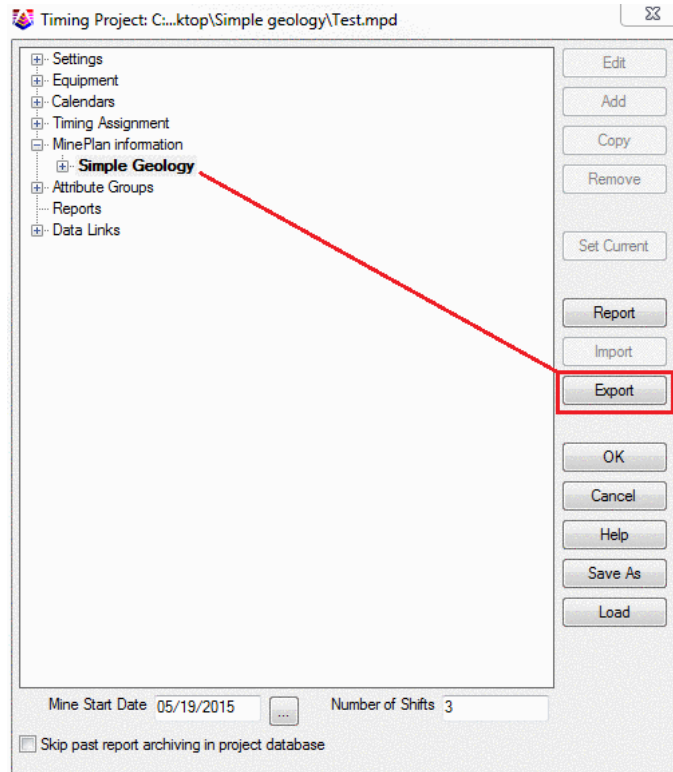
Keyboard Command: disconnect_plans

Import Mine Plan

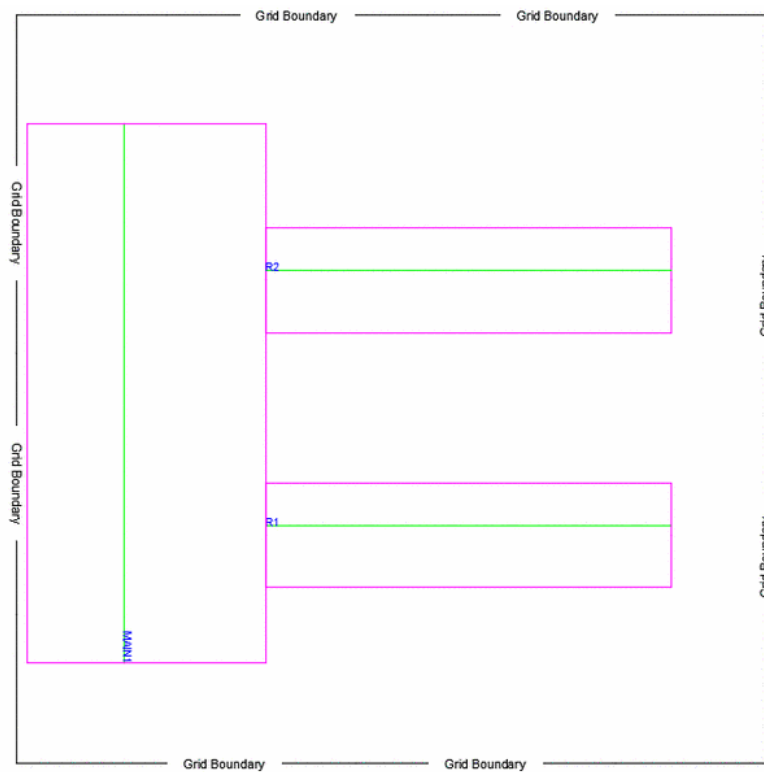
This command allows you to import an Underground Mine Plan into a drawing. The Mine Plan includes panel perimeters, panel names, panel centerlines, and section data. This command can be used to transfer planning

information between drawings to evaluate multiple mining scenarios.

The Mine Plan is created when you create the mining panels and assign them to a piece of equipment in the Underground Mine Timing command. The Mine Plan can be saved as a .ppi file via the Timing Project Manager. To save this file, open the Timing Project Manager (Underground Pulldown Menu > Timing Project Manager). Then select the Mine Plan under the Mine Plan Information Menu. You can then click **Export** to save the Mine Plan as a .ppi file.



To import a Mine Plan, click Underground Pulldown Menu > Import Mine Plan. Here you can select a .ppi file to import into the drawing. Once the file has been imported, the basic linework of the Mine Plan will be drawn. The below figure shows a simple Mine Plan that has been imported into a new drawing. Notice that the panel perimeters, names, and belt lines have all been imported.



Once the Mine Plan has been imported, you can use the Edit Panel command to inspect the Section Data relating to the panels.

Pulldown Menu Location: Underground Mining/Underground

Keyboard Command: import_plan

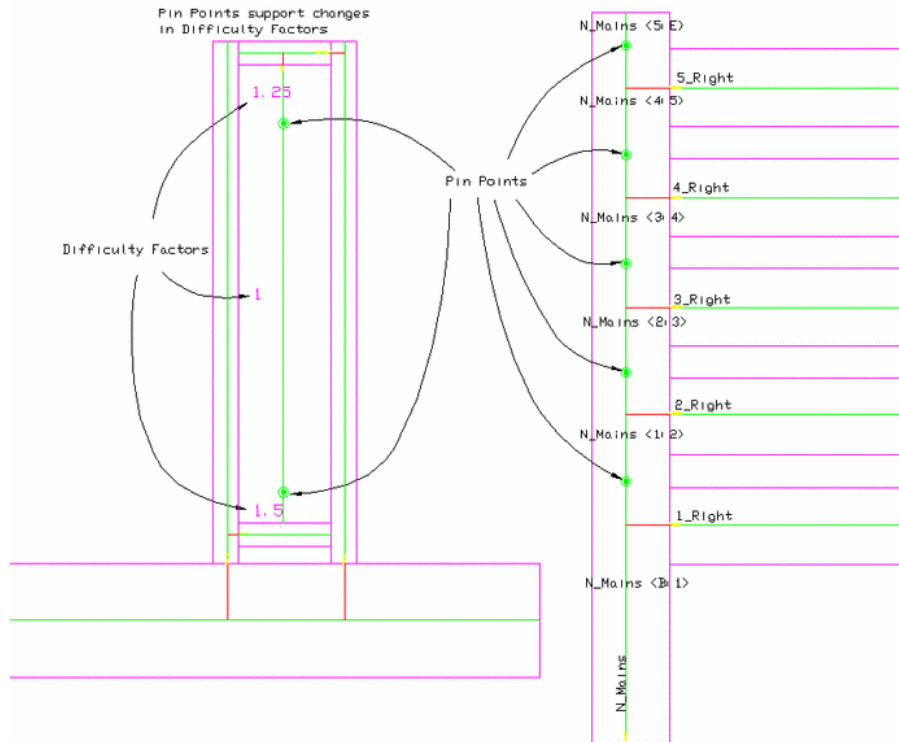
Prerequisite: PPI file previously saved

Add Pin Point

Pin Points are used to provide stopping points within a panel. This is similar to, but not exactly the same thing as sub-dividing as panel. Pin Points can be used to sequence mining up to the stopping point for both advance and

retreat mining. Difficulty factors can be changed between pin points by placing the difficulty factor by text on the DIFFICULTY layer to impact the mining rate as well. Units can mine up to a pin point, mine in another panel, and then return to continue mining later. The N_Mains panel below has been broken into 6 parts that can be scheduled with the 5 pin points as shown.

The function Add Pin includes new pin points into the panel as desired.



For the example shown the sequence of advance mining is as the following:

```
N_MAINS <B:1>
N_MAINS <1:2>
N_MAINS <2:3>
N_MAINS <3:4>
N_MAINS <4:5>
N_MAINS <5:E>
```

The different pieces may also be assigned to the separate units as well.

The same set of pinpoints is used for both advance and retreat mining with the following naming convention:

Advance

```
N_MAINS <B:1>
N_MAINS <2:E>
```

Retreat

```
MAINS<B:1>
RET MAINS<2:E> RET, so that a section name on retreat is the same as on advance except for suffix the RET.
```

Prompts

Pick pinpoint or Enter distance or Define size[<Pick>/Enter/Size]: Whenever this prompt is displayed, you can click directly on a mine panel to place a pin point. This will continue until you select a new command line

option or press ESC to exit the command.

Pick pinpoint or Enter distance or Define size[<Pick>/Enter/Size]:E *This option will let you select a single mine panel and enter distances down the length of the panel to place a pinpoint*

Select mine panel:

Enter distance from starting point:100 *Places a pinpoint at exactly 100 ft (or meters) from the start of the panel*

Enter distance from starting point:200 *Places a pinpoint at exactly 100 ft (or meters) from the start of the panel*

Enter distance from starting point: *Press ESC to return to the main prompt*

Pick pinpoint or Enter distance or Define size[<Pick>/Enter/Size]:S *Allows you to change the size of any new pinpoints created*

Pinpoint size <50.00>: 25 *Sets the size of the pinpoints to 25*

Pulldown Menu Location: Underground

Keyboard Command: addpin

Remove Pin Point

The Remove Pin Point command deletes the unwanted pin point from the panel. See Add Pin Point for detailed information on pin points.

Prompts

Click on centerline close to pinpoint to be removed: *pick on the centerline close to unwanted pin point*

Pulldown Menu Location: AdvMine

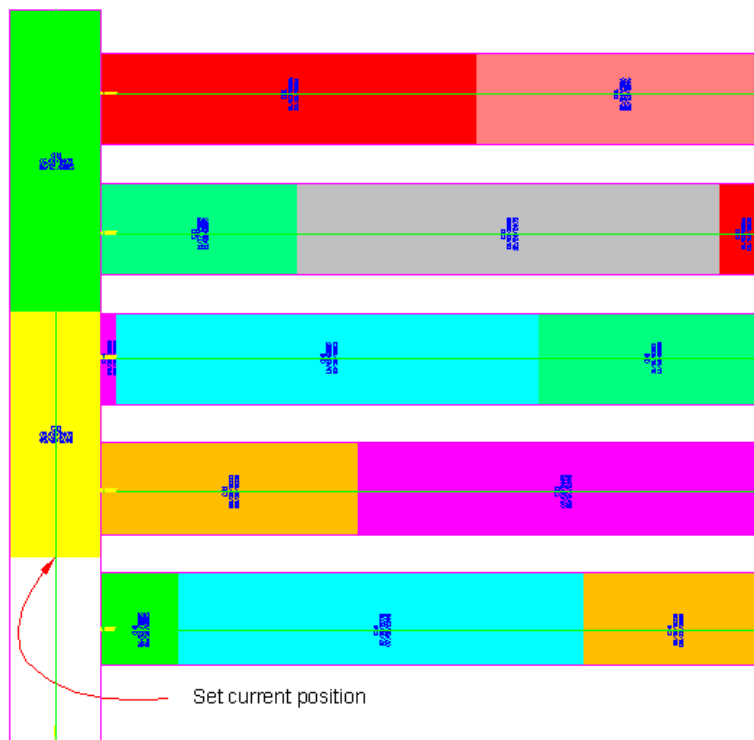
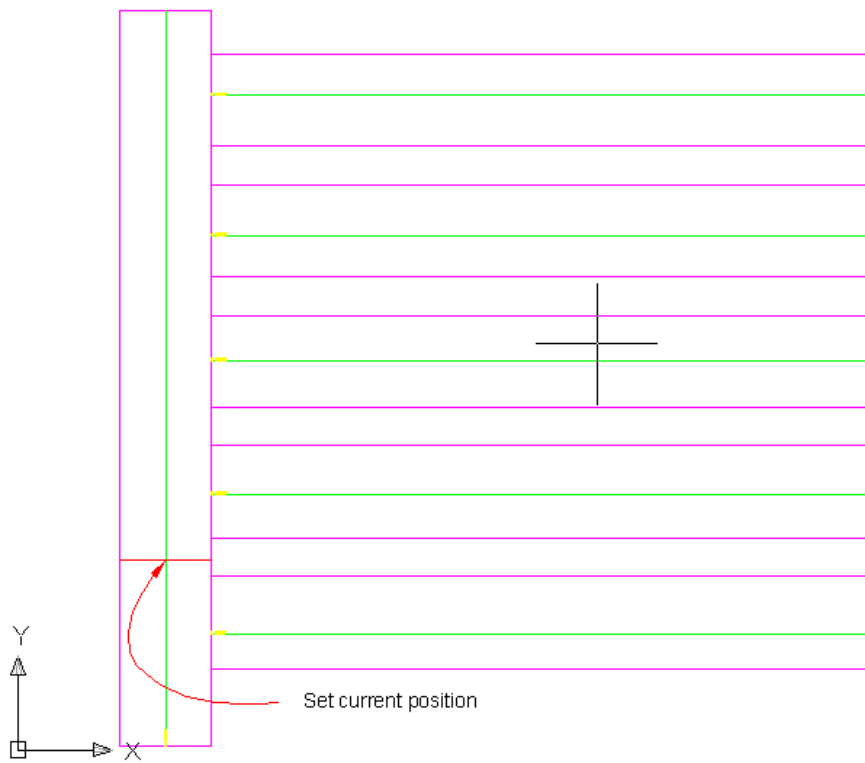
Keyboard Command: delpin

Set Current Position

The purpose of this function is to set the current state of the mine development.

When a mine plan is first created, the mine plan is a snapshot of the layout of the mine. As mining progresses some panels are completed, while mining in the others just starting. Whenever the user wants to advance the Starting Date, the mine plan should be adjusted to the mine state on the new date.

Some of the updates can and should be done using Update Edited Panel after corresponding modification, but whenever a panel is going to be retreated, it is impossible to modify panel perimeter to show changes. In this case a current position of the crew within the panel is marked using Set Current Position function.



Prompts

Select a current stage of panel development (<Advance>/Retreat)?

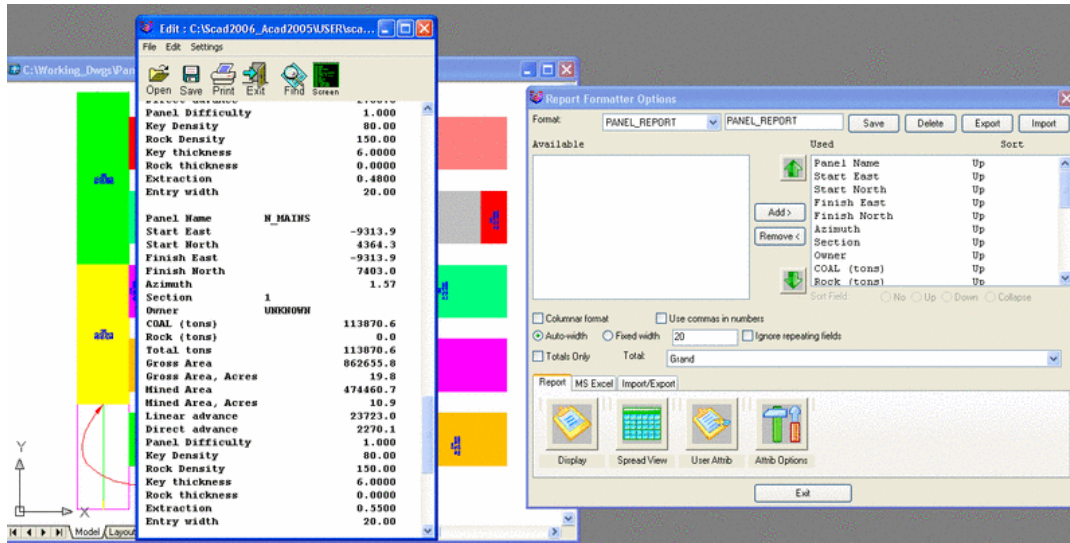
Click on centerline at current position: *pick on the centerline*

Pulldown Menu Location: UnderGnd

Keyboard Command: pstart

Panels Report

Panels Report provides a way to report out all of the information stored in the panels in a common mine plan using the report formatter. Select any backbone polyline in a mineplan and the report formatter will appear ready to print the report.



Pulldown Menu Location: AdvMine
Keyboard Command: panels_report

Underground Timing

The diagram above shows the general steps associated with the creation of a mine plan with the Underground Timing routine. Prerequisites include:

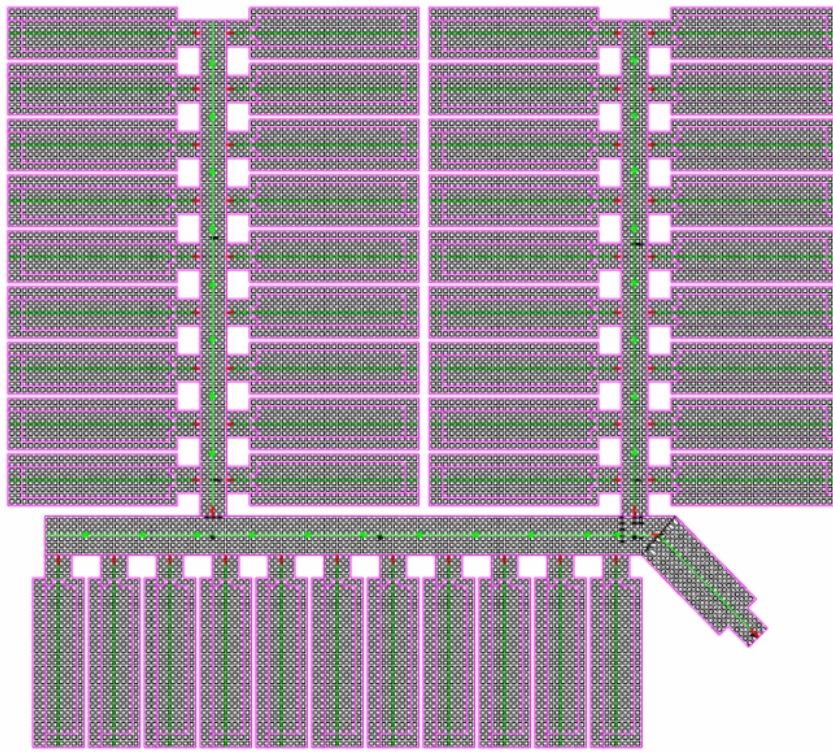
- Creating thickness, density, and quality grids, and/or average values for the total coal and rock to be mined.
- Defining the productivity rates of the equipment and operating schedule for each section of equipment in the Timing Project Manager.
- Preparing the mine linework on specific layers (see below).
- Converting linework into panels with one of four commands: Place Panel, Auto-Place Panel by Text, Auto Panel Layout, and/or Pick and Place Panel.
- Adjusting the mine plan by adding Pin Points, Recalculating Extraction, Connecting Mine plans, and Editing panels.

A complete list of the reserved layer names used in the timing command is below. Linework and/or text on these layers will play a specific role in the timing.

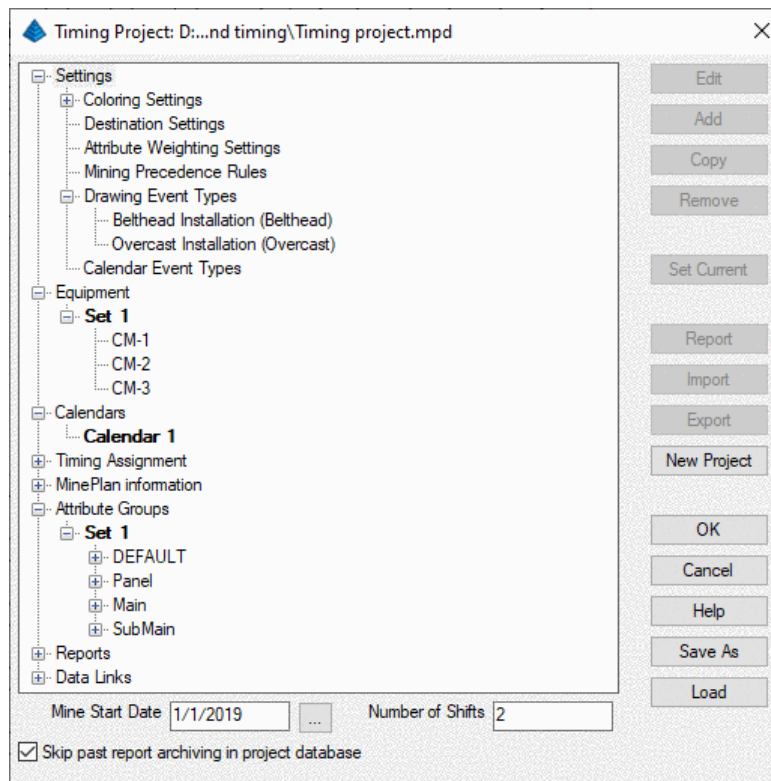
Layer	Data Type	Description
ATTR_GROUP	Text	Assigns Attribute Group for the panel when using Auto-Place Panel by Text and Recalculate Extraction commands
DIFFICULTY	Text	Assigns panel difficulty factor during Underground Timing command (value is read during timing calculation)
ENTRY_WIDTH	Text	Assigns Entry Width for the panel when using Auto-Place Panel by Text and Recalculate Extraction commands

EXTRACTION	Text	Assigns extraction ratio for the panel when using Auto-Place Panel by Text and Recalculate Extraction commands
MINEPLAN	Linework/symbols	Displays Pin Points and panel centerlines when panels have been created - This linework is automatically generated; users do not need to manually create this linework
MINEPLTX	Text	Displays panel names when panels have been created - This text is automatically generated; users do not need to manually create this text
PANELNM	Text	Assigns the panel name when using the Auto-Place Panel by Text command
PERIM	Linework	Defines the panel perimeter - Note that these must be closed polylines
PILLARS	Linework	Defines pillars in the panel - Note that these must be closed polylines
PRECEDENCE	Linework	Assigns precedence rules when using the Panel Precedence by Polylines command
PROJECTIONS	Linework	Displays entry and crosscut projections when using the Basic Projections or Advanced Projections commands
RET_DIFF	Text	Assigns the panel difficulty factor during Underground Timing for retreat mining (value is read during timing calculation)
RET_EXTRACT	Text	Assigns extraction ratio for retreat mining the panel when using Auto-Place Panel by Text and Recalculate Extraction commands

An example mine layout is shown below.

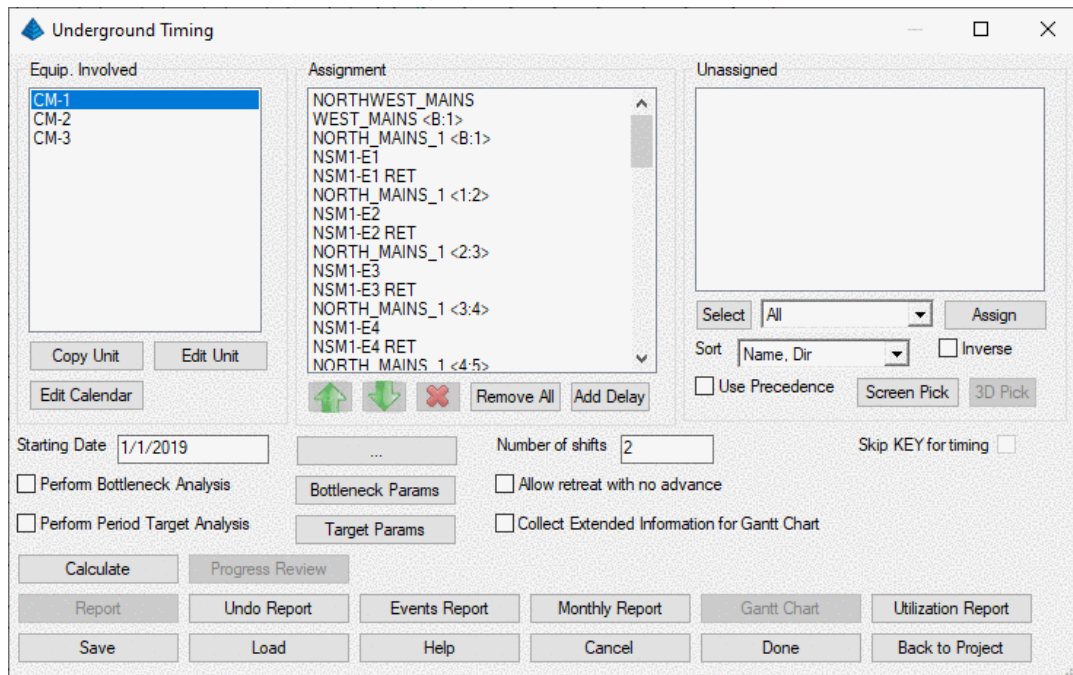


When the command is executed, you will be prompted to select a part of the mine plan. The Timing Project Manager, shown below, will then appear. Here you can define equipment, calendars, attribute groups, and more. For more information on the Timing Project Manager, see the corresponding section of the help manual.

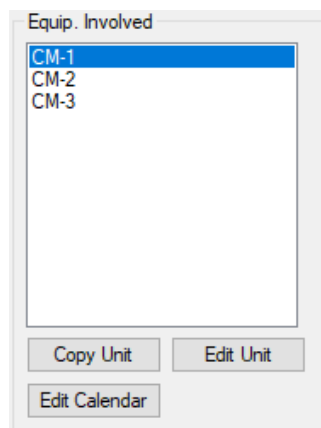


After ensuring all settings are correct in the Timing Project Manager, you can click OK to move on to the assignment dialog, shown below. The primary purpose of this dialog to move panels from the Unassigned Column to the

Assignment column in the order in which they will be extracted. Each piece of equipment has its own assignment list.



The Equip. Involved Column lists all pieces of equipment to be used for the mine scheduling. Panels must be assigned to a specific piece of equipment in order to be mined.

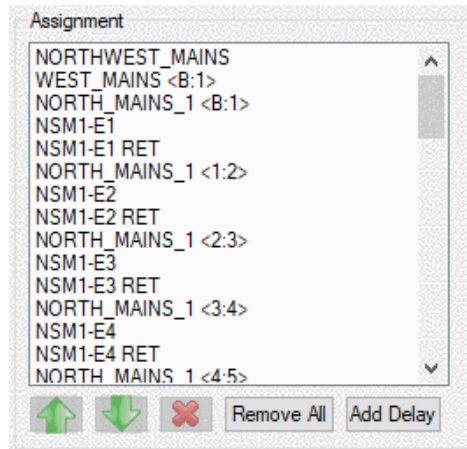


Copy Unit: This button will create a copy of the currently select unit.

Edit Unit: This button will allow you to edit the specifications of the currently selected equipment. More information on defining equipment specifications is available in the Timing Project Manager section of the help manual.

Edit Calendar: This button will allow you to edit the current work calendar. More information on defining the work calendar is available in the Timing Project Manager section of the help manual.

The Assignment Column lists the panels assigned to the currently selected equipment. Selecting a different unit will update this column with the list of panels assigned to that unit. This column lists panels in the order that they will be mined, with panels at the top of the list being mined first. Once panels have been assigned to the various equipment, the assignments can be saved to a .tim or a .csv file. Both can be edited with a simple text editing program such as Notepad, but the .tim format is not as user-friendly as the .csv format. When exported to a .csv, you can very easily editing the timing assignments in Microsoft Excel, then load the timing assignments back into the Underground Timing dialog. In the .csv format, the first column is the equipment name and the second column is the panel name. The assignments will be retained between calculations of the timing, but it is highly recommended to save your timing assignments to an external file.

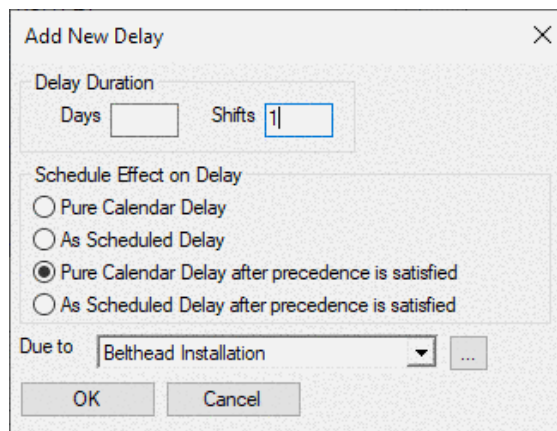


The green **Up/Down Arrow Icons** will move the currently selected panel(s) upward or down in the list, causing it to be mined earlier or later in the sequence.

The red **X icon** will remove the currently selected panel(s) from the list, allowing it to be re-assigned to another piece of equipment.

Remove All: This button will remove all assignments from the currently selected unit.

Add Delay: This button will open the dialog shown below, which controls how delays are handled. A delay will appear in the Assignment Column as if it were another panel to be mined, and may be moved up or down in the sequence.



Delay Duration: These values control how long the delay lasts. In the above example, the equipment will be delayed for a full shift. Note that partial shift delays are allowed.

Pure Calendar Delay: This option will not force a delay. If the equipment already has scheduled downtime when the delay is encountered, additional time will not be taken off.

As Scheduled Delay: This option will force a delay even if the equipment already has scheduled downtime when the delay is encountered. The delay will wait until the scheduled downtime is complete, then the delay will be applied.

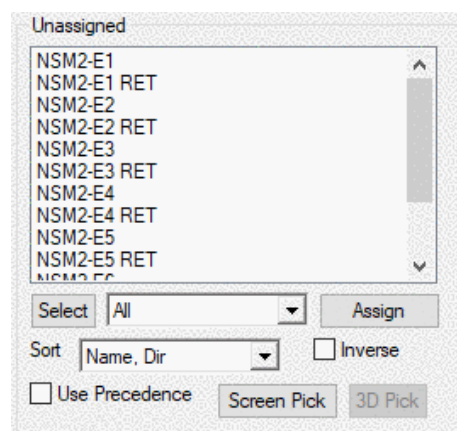
Pure Calendar Delay after precedence is satisfied: This is similar to a Pure Calendar Delay, but will not attempt to apply the delay until precedence rules have been satisfied. Consider the scenario shown in the below table. Equipment E1 is scheduled to mine Panel A and Panel C. Equipment E2 is scheduled to mine Panel B and Panel D. A precedence rule has been created that does not allow Panel D to be mined until Panel A is complete. A delay of this type is placed on Equipment E2 after Panel B. In this scenario, the finish date of Panel A will first be calculated, then the delay will be applied. If the equipment already has scheduled downtime when the delay is encountered, additional time will not be taken off.

As Scheduled Delay after precedence is satisfied: This option is similar to an As Scheduled Delay, but will not attempt to apply the delay until precedence rules have been satisfied. Consider the below scenario. Equipment E1 is scheduled to mine Panel A and Panel C. Equipment E2 is scheduled to mine Panel B and Panel D. A precedence rule has been created that does not allow Panel D to be mined until Panel A is complete. A delay of this type is placed on Equipment E2 after Panel B. In this scenario, the finish date of Panel A will first be calculated, then the delay will be applied. If the equipment already has scheduled downtime, the delay will be applied in addition to this downtime.

Due To: This dropdown menu lists the Drawing Event delays defined in the Timing Project Manager. The ellipsis button will allow you to define a new Drawing Event.

Equipment	E1	E2
Assignment	Panel A	Panel B
Sequence	Panel C	Delay
		Panel D (Precedence Rule: Panel A)

The Unassigned Column lists all panels that have not yet been assigned to a piece of equipment.



Assign: This button will assign the selected panel(s) to the selected piece of equipment.

Select: This button will select all panels according to the drop-down menu to the right of this button. If the drop-down menu is set to NONE, all panels will be unselected. If the drop-down menu is set to ALL, all panels will be selected. If the drop-down menu is set to Advance or Retreat, all panels of that type will be selected.

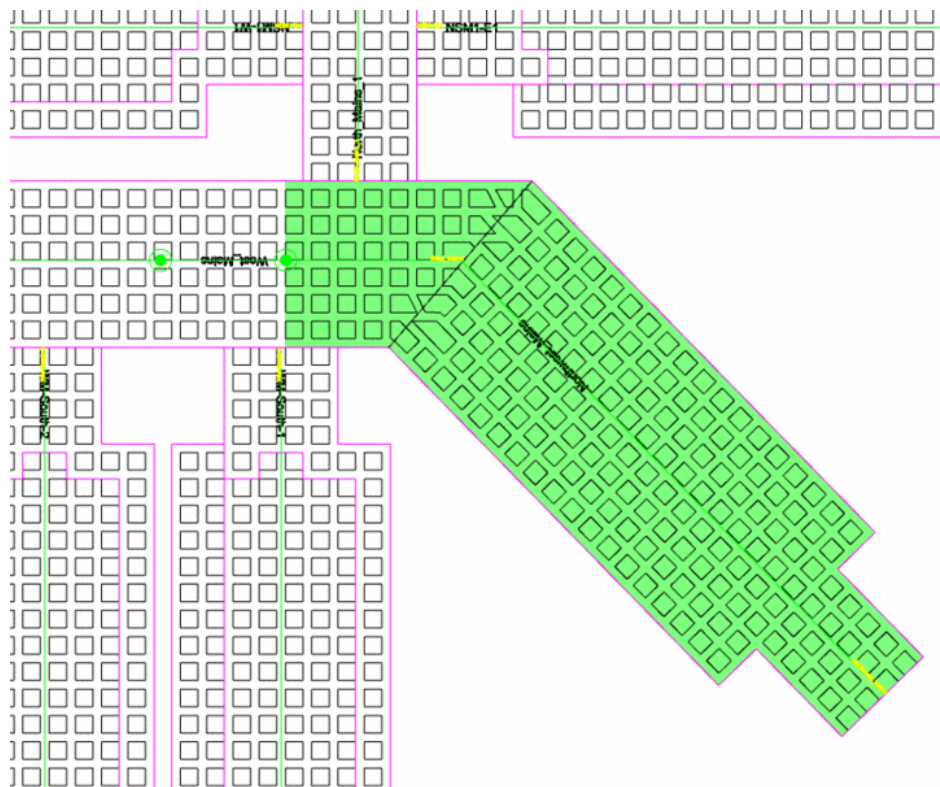
Sort: This drop-down list will sort the panels in the list.

- Name, Dir: This option will sort the list by panel name, then by direction. For example, if you have retreat panels, they will be listed immediately after the advance heading (notice how the RET panels immediately follow the non-RET panels in the image above).
- Dir, Name: This option will sort the list by direction, then by name. This will simply group all of the advance panels together, and all of the RET panels will be grouped as well.

Inverse: This checkbox will invert the sorting order of the Unassigned Column.

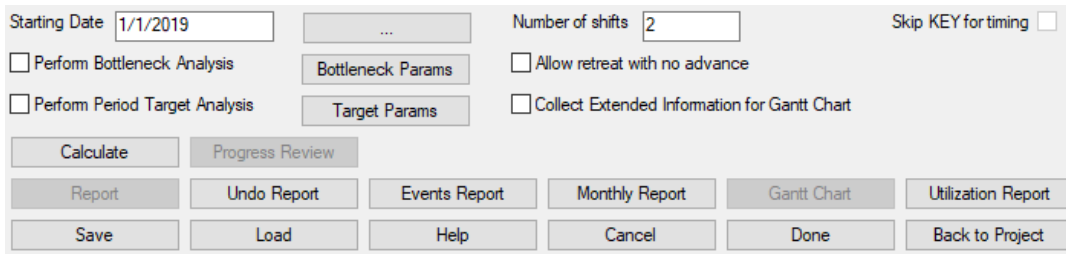
Use Precedence: This checkbox will ensure that precedence is satisfied in the list. For example if Panel1 is assigned to precede Panel2, toggling this option on will reorder the list so that Panel1 is listed before Panel22.

Screen Pick: This button will allow you to sequence the panels by selecting them in the drawing. When activated, the Underground Timing Dialog will disappear and the drawing area will be visible. The command line will prompt you to click inside the panel to assign to the currently selected piece of equipment. Panels that have already been assigned to a piece of equipment will be filled with a transparent color. Pressing ENTER after selecting the panels will return to the Equipment Timing dialog. An example of the Screen Pick method of selection is shown below. The green filled panels have already been assigned to a piece of equipment. If pin points are in the panel, it will only hatch up to the next pin point, allowing the panel to be assigned in order as separate entities.



3D Pick: This option is not available in the Underground Timing command - it is only available in the Surface Equipment Timing command.

Additional scheduling options are shown below.



Starting Date: This value sets the date to begin mining operations. The ellipsis button to the right of this field will open a small calendar to assist in date selection. By default, this value will be set the Mine Start Date set in the Timing Project Manager.

Number of Shifts: This value will set the number of working shifts in a day.

Skip COAL (Key Material) for timing: This checkbox controls if key material is used for the timing or not. When this option is on, the key material (be it coal or some other resource) will not be included in the scheduling. Instead, it will be assumed that all key material will be mined quickly enough so that it never delays the removal of waste material.

Perform Bottleneck Analysis and Bottleneck Params: This option is intended for use in the Surface Equipment Timing command, and is therefore not discussed here.

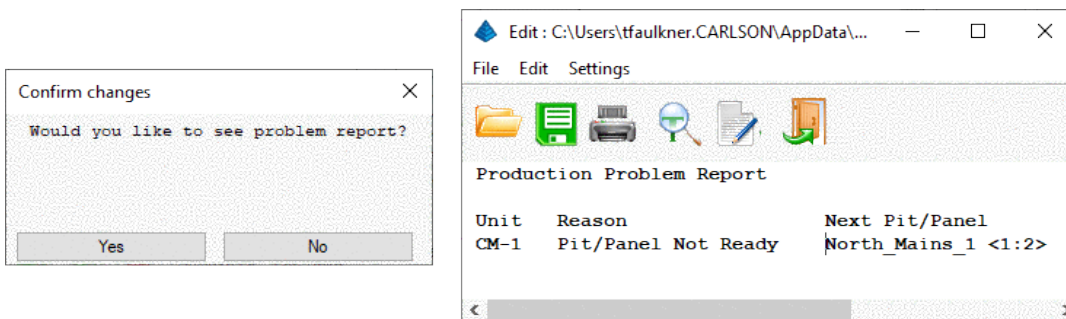
Perform Period Target Analysis and Target Params: This option is intended for use in the Surface Equipment Timing command, and is therefore not discussed here.

Allow Retreat with No Advance: This option allows retreat panels to be mined without first mining the advance panels. This can be useful if it is more advantageous to simply refer to a panel as retreat, even though it is only going to be mined through once.

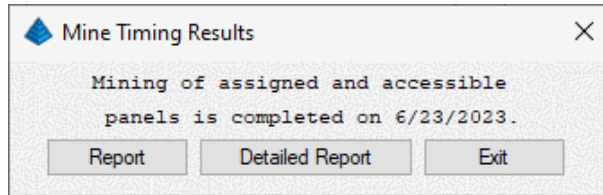
Collect Extended Information for Gantt Chart: When enabled this option will allow the Gantt Chart to include extra information such as the tonnage, quality, etc.

Calculate: This button will calculate the mine schedule. In cases when the mine plan includes retreat (second) mining, the system will make sure that when the parent panel is mined on retreat, the mining (advance or retreat) in all child panels is completed. If not it tries to offset the starting date of retreat in the parent panel until the timing conflict is resolved. Sometimes it is impossible to do so and the program reports the problem to the user for manual resolution.

If any errors are encountered, the below dialog (left) will appear. The problem report generated will appear similar to below (right). In this example, a precedence rule has been violated for the CM1 equipment. Here, the North_Mains_1 <1:2> panel has violated a precedence rule. This will require an edit to the assignment order for proper timing.



If no problems are encountered, the schedule will be calculated and the below dialog will appear.

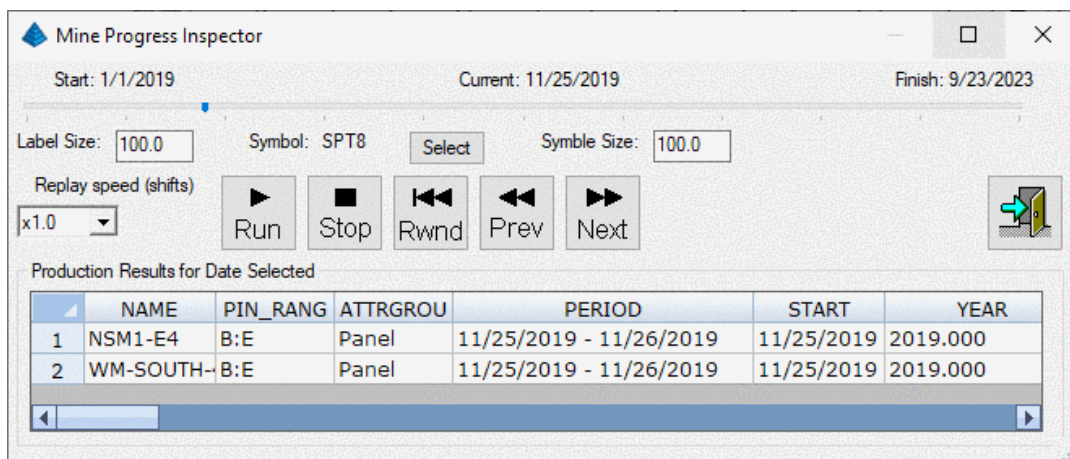


Report: This button will generate a very simple report as shown below. Note that this report does not include information about the amount of material mined. The Days Waiting in this report refer to the number of days in which a panel is waiting to be excavated due to precedence rules, or due to a scheduled delay.

Unit	Days waiting	Panel name	Pinpoints	Sections	Sect. 1 finish	Days Operating	days	Weekdays	Worked	Saturdays
CM-1	0.0	NORTH_MAINS_1	B:1	1	3/26/2019	7.8	9.46	8.46		
CM-1	0.0	NORTH_MAINS_1	1:2	1	5/31/2019	6.9	6.89	5.89		
CM-1	0.0	NORTH_MAINS_1	2:3	1	8/8/2019	7.1	7.06	6.06		
CM-1	0.0	NORTH_MAINS_1	3:4	1	10/15/2019	7.1	7.11	6.11		
CM-1	0.0	NORTH_MAINS_1	4:5	1	12/25/2019	8.8	9.79	8.79		
CM-1	0.0	NORTH_MAINS_1	5:6	1	3/25/2020	8.5	8.46	7.46		
CM-1	0.0	NORTH_MAINS_1	6:7	1	6/4/2020	6.8	6.82	6.82		
CM-1	0.0	NORTH_MAINS_1	7:8	1	8/11/2020	6.8	6.85	6.29		
CM-1	0.0	NORTH_MAINS_1	8:E	1	10/14/2020	5.0	4.99	3.99		
CM-1	0.0	NORTHWEST_MAINS	B:E	1	2/13/2019	29.7	33.22	30.22		
CM-1	0.0	NSM1-E1	B:E	1	5/7/2019	33.0	32.97	29.97		
CM-1	0.0	NSM1-E1 RET	B:E	1	5/23/2019	13.5	13.47	12.47		
CM-1	0.0	NSM1-E2	B:E	1	7/12/2019	33.0	32.97	29.97		

Detailed Report: This button will generate a more detailed report, including information such as the amount of material mined, quality of material, downtime, etc. This report is discussed in more detail later in this help document.

Progress Review: This button will open the Progress Inspector shown below. The Progress Inspector will only be generated with information after the mine schedule has been calculated.



The Progress Inspector allows you to step through the day-by-day schedule of each piece of equipment. The slider bar at the top of the window may be used to cycle through the dates quickly. Various information about each piece of equipment is shown in the spreadsheet view. As you progress through each day with the below controls, a symbol and label will appear on the drawing to indicate the location of the equipment. The size of the labels/symbols may be set with the **Label Size** and **Symbol Size**. A different symbol may be selected by clicking the **Select** button.

Run: This button steps through the overall mine progression day-by-day. You may modify the playback speed by changing the **Replay speed** option.

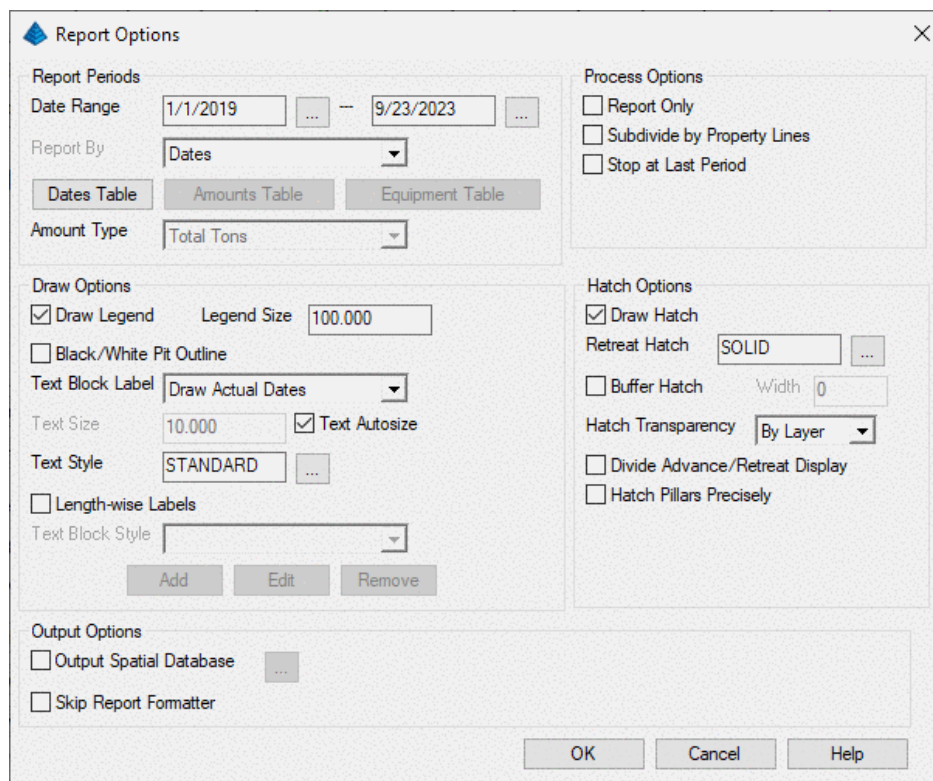
Stop: This button stops the action of the Run button.

Rwnd: This button returns the inspector to the first working day of the schedule.

Prev: This button steps to the previous day in the mine schedule.

Next: This button steps to the next day in the mine schedule.

Report: This button is only available after calculating the mine schedule. This button will open the Detailed Report dialog. This is the same dialog that will appear after selecting the Detailed Report option after clicking the Calculate button.



Date Range: This option will set this range of dates to report.

Report by: This drop list will determine how the timing map is generated.

If the *Dates* option is selected, the timing map will be colored by date ranges.

If the *Amounts* option is selected, the timing map will be colored by amount intervals.
 If the *Equipment* option is selected, the timing map will be colored by equipment.

Dates Table: This button will open the Dates Table, shown below, which is used to define the various time periods used for reporting.

	End Date	Range	Color	Pattern	Scale	Layer	Label
1	2/1/2028	< 2/1/2028	Red	SOLID	Set	250.0 MINETM	Set
2	3/1/2028	2/1/2028 - 3/1/2028	Dark Red	SOLID	Set	250.0 MINETM	Set
3	4/1/2028	3/1/2028 - 4/1/2028	Red-Orange	SOLID	Set	250.0 MINETM	Set
4	5/1/2028	4/1/2028 - 5/1/2028	Orange	SOLID	Set	250.0 MINETM	Set
5	6/1/2028	5/1/2028 - 6/1/2028	Light Orange	SOLID	Set	250.0 MINETM	Set
6	7/1/2028	6/1/2028 - 7/1/2028	Yellow-Orange	SOLID	Set	250.0 MINETM	Set
7	8/1/2028	7/1/2028 - 8/1/2028	Yellow	SOLID	Set	250.0 MINETM	Set
8	9/1/2028	8/1/2028 - 9/1/2028	Light Yellow	SOLID	Set	250.0 MINETM	Set
9	10/1/2028	9/1/2028 - 10/1/2028	Light Green	SOLID	Set	250.0 MINETM	Set
10	11/1/2028	10/1/2028 - 11/1/2028	Green	SOLID	Set	250.0 MINETM	Set
11	12/1/2028	11/1/2028 - 12/1/2028	Dark Green	SOLID	Set	250.0 MINETM	Set
12	1/1/2029	12/1/2028 - 1/1/2029	Light Green	SOLID	Set	250.0 MINETM	Set
13	2/1/2029	1/1/2029 - 2/1/2029	Green	SOLID	Set	250.0 MINETM	Set
14	3/1/2029	2/1/2029 - 3/1/2029	Dark Green	SOLID	Set	250.0 MINETM	Set
15	4/1/2029	3/1/2029 - 4/1/2029	Light Green	SOLID	Set	250.0 MINETM	Set
16	5/1/2029	4/1/2029 - 5/1/2029	Yellow-Green	SOLID	Set	250.0 MINETM	Set

The above dialog can be used to create custom time periods to show in the timing map. The green and red icons at the top of the dialog will add and delete rows of information.

End Date: This column sets the ending date of the period.

Range: This column is automatically generated, and simply identifies the Date Range for the period. Dates may be preceded by > or < to mark indefinite time periods.

Color: This column sets the color of the time period. Double clicking one of these cells will open the CAD color palette.

Pattern: This column sets the hatch pattern for the timing blocks.

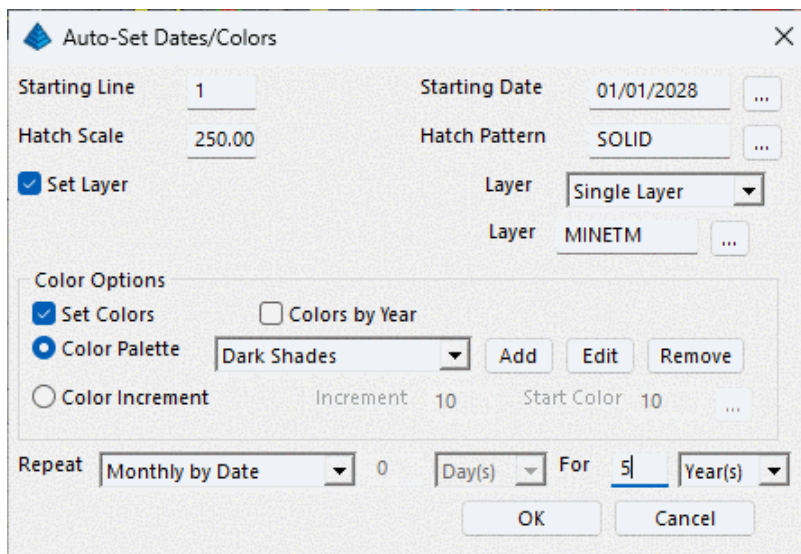
Scale: This column sets the scale of the hatch pattern.

Layer: This column sets the CAD layer on which to draw the timing block.

Label: This column sets a custom label for the timing block.

Clear: This button will clear the table of all information.

Auto Set: This button will open the below dialog, which allows for quick population of the Custom Dates table.



Starting Line: This value will set the row to begin the automatic population of the Custom Date table.

Starting Date: This is the Date From that will be used on the Starting Line. The ellipsis button will open a small calendar to assist in date selection.

Hatch Scale: This value will set the hatch scale for the new time periods.

Hatch Pattern: This field will set the hatch pattern for the new time periods. The ellipsis button will open the list of available hatch patterns.

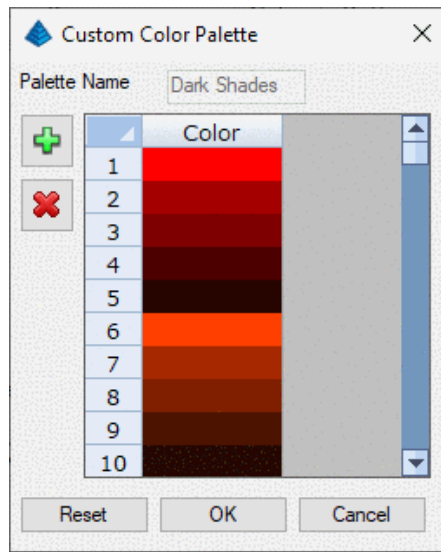
Set Layer: This option will overwrite the Layer names when enabled.

Layer by: This drop list determines how the layers will be populated. If the *Single Layer* option is selected, all layers will be the same, according to the **Layer** text box. If the *Year* option is selected, the layer will be set by the year of the time period.

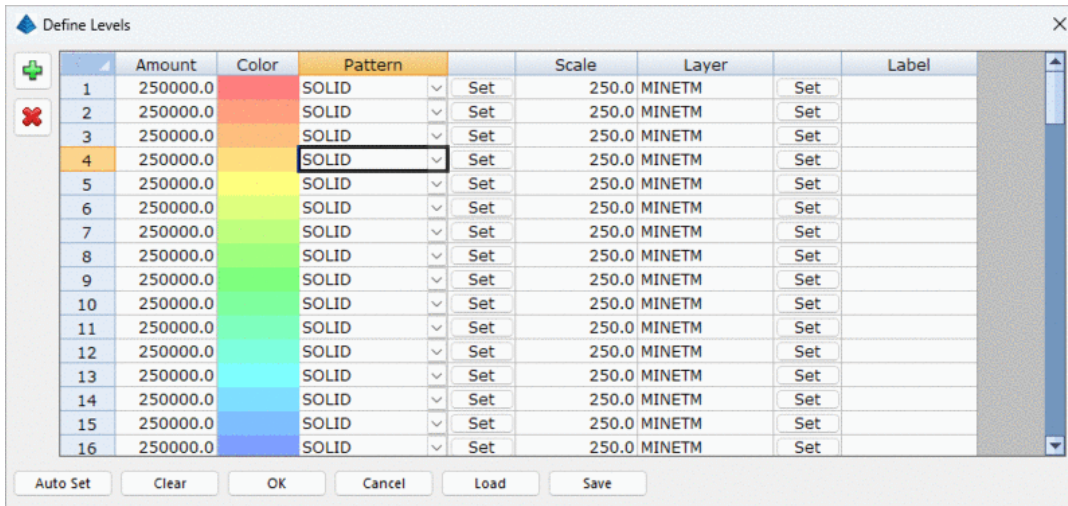
Set Colors: This option will set custom colors for the new time periods. There are two options for setting colors - by using a Color Palette, or by using a Color Increment.

Colors by Year: When this option is enabled, the colors will only increment when the starting date for a row changes. This option is handy for drawing timing maps that have the same color for an entire year, but generate the report on a monthly basis.

Color Palette: This option will use a predefined color palette to color the time periods. When the end of the palette is reached, the colors will start over. Palettes may be added, edited or removed using the corresponding buttons. An example color palette is shown below. Colors may be added or removed from the palette by clicking the Plus or X icons, and colors may be modified by double-clicking them. Any of the predefined palettes may not have their name modified. If you modify the colors in a predefined palette, you may click the **Reset** button to return to the default colors.



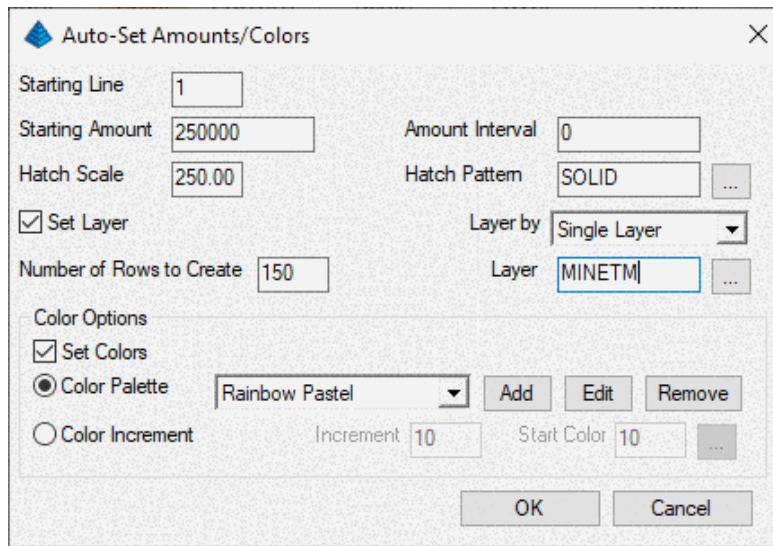
Amounts Table: This button will open the Amounts Table, shown below, which is used to define the various amount intervals used for reporting.



The above dialog is very similar to the Custom Dates table previously discussed. The primary difference is that the dates are replaced with the **Amounts** column. These values may represent one of the following values to be mined, set according to the **Amount Type**:

- Total Tons
- Key Material Tons
- Waste Tons
- Total Area
- Mined Area
- Total Volume
- Key Material Volume
- Waste Volume
- User Grid

Auto Set: This button will open the below dialog, which is very similar to the Auto Set dialog used in the Custom Dates table, as previously discussed. This dialog differs in that amounts are specified rather than dates, and the periods are set to repeat for a set number of times, rather than being based on the dates.



Equipment Table: This button will open the Equipment Table, shown below, which is used to set the colors for each piece of equipment to be reported. This is like a simplified version of the Dates Table and Amounts Table. Here there is no option to set a report period - only the color, hatch, layer, and label for each piece of equipment.

	Equipment	Color	Pattern	Scale	Layer	Label
1	CM-1	Red	SOLID	100	MINETM	CM1
2	CM-2	Yellow	SOLID	100	MINETM	CM2
3	CM-3	Green	SOLID	100	MINETM	CM3

Report Only: This option will not create a timing map; only the mine schedule will be reported. When selected, many other options on the dialog will be ghosted out. This runs much faster than creating a timing map.

Subdivide by Property Lines: This option will sub-divide the report by property owner.

Stop at last period: This option is only available when the Custom Amount Table report option is selected. This option will end the schedule at the last entered user row in the Custom Amount Table.

Draw Legend: This option will draw a color legend of the timing map. The **Legend Size** can be used to make the legend larger or smaller.

Black/White Panel Outline: This option will draw the boundaries of the timing periods with a white/black color as opposed to matching the color of the hatch.

Text Block Label: This drop list will determine how the labels are drawn in the timing map.

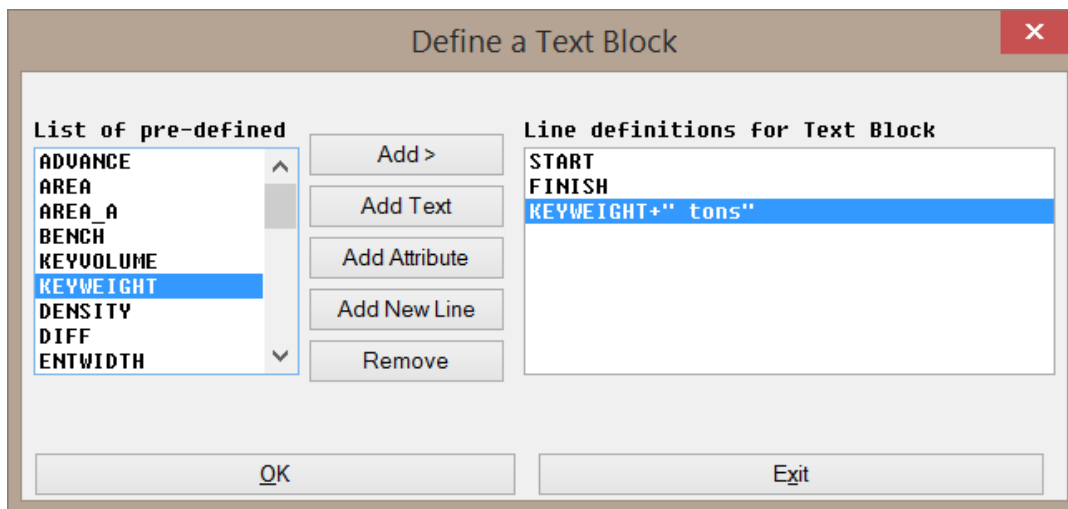
The *No Block Labels* option will not draw labels.

The *Draw Actual Dates* option will label the starting and ending date of each period.

The *Draw Period Names* option will draw labels defined in the Custom Date Table.

The *Use Custom Names* option will draw labels defined in the Custom Amount Table.

The *Use Custom Text Block* option will draw labels according to the below dialog. When selected, the **Text Block Style**, **Add**, **Edit**, and **Remove** buttons will become available. The Text Block Style dropdown list will list all available text block labels. The Add button will create a new text block. The Edit button will edit the currently selected text block. The Remove button will remove the currently selected text block.



The left side of the above dialog contains pre-defined keys that may be used in the labels. The right side of the dialog shows how the text block label will appear.

Add: This button will add the selected attribute to the current line. Multiple attributes may be added to a single line.

Add Text: This button will add custom text to the current line. In the above example, "tons" has been added as a suffix to the KEYWEIGHT attribute.

Add Attribute: This button will add any attribute that is not defined in the pre-defined list. This attribute name must match an attribute available in the final text report.

Add New Line: This button will create a new line in the text block. In the above example, three lines have been created.

Remove: This button will remove the selected line from the text block.

Text Size: This value will set the text size of the block labels.

Text Autosize: This option will automatically size the text of the block labels.

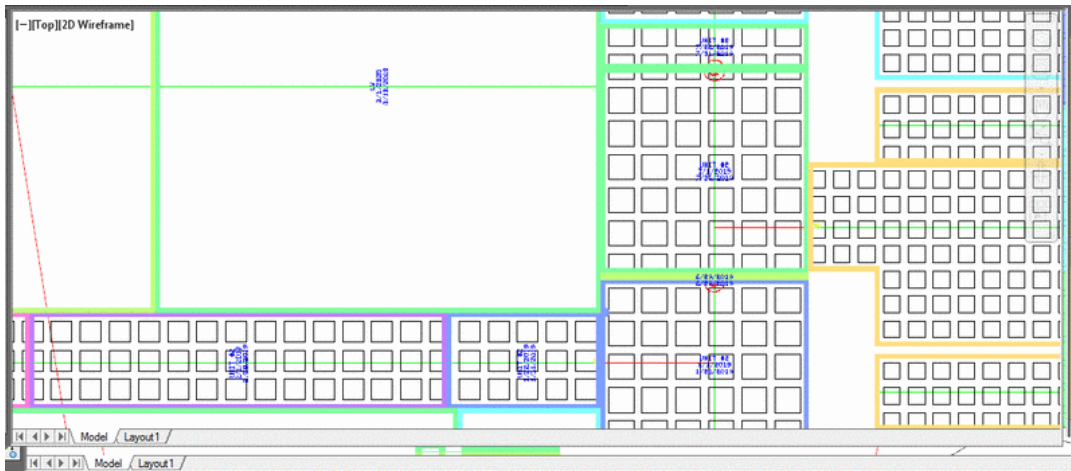
Text Style: This option will set the text style to use for the period labels. The **Select** button will allow you to select one of the existing text styles.

Length-wise labels: This option will rotate the block labels in line with the mining direction of each panel. If not selected, labels will be drawn perpendicular to the mining direction.

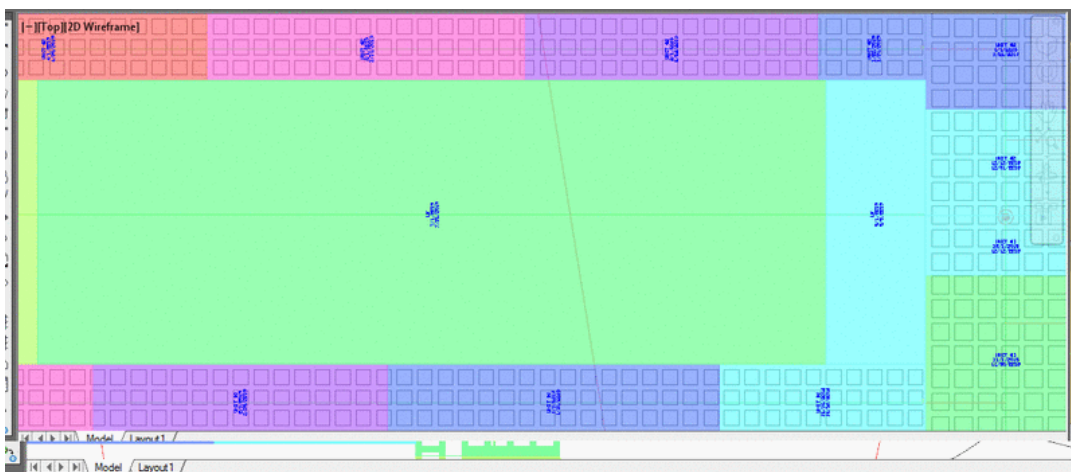
Draw Hatch: This option will create boundaries for the timing periods and fill them in with the specified hatch pattern/color.

Retreat Hatch: This option sets the hatch pattern for the retreat hatches. The ellipsis button to the right of this option will open a list of available hatch patterns.

Buffer Hatch/Width: This option will not hatch in the full period, but will instead create a thick boundary of a specified width. The width determines the offset from the perimeter. An example of a Buffer Hatch is shown below.



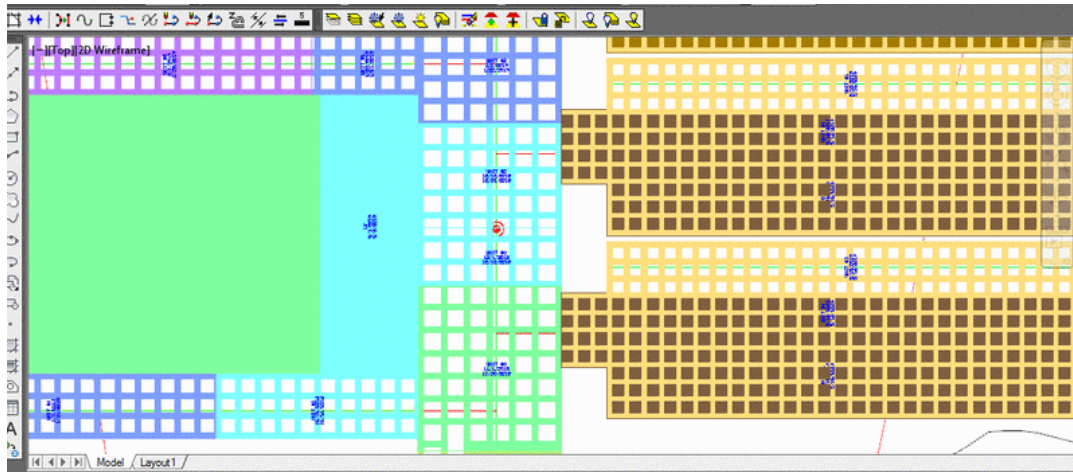
Hatch Transparency: When using the SOLID hatch, this setting controls the transparency of the hatch. The range is from 10%-90%. This allows for viewing the entities behind the hatch. An example of transparent hatches is shown below. Note that if transparency is not used, you would not be able to see the pillars and projection lines behind the hatches.



Divide advance/retreat display: This option will divide the hatches for advance and retreat hatches down the length of the panel. Advance hatches will be drawn on the right of centerline and retreat hatches will be drawn

on the left.

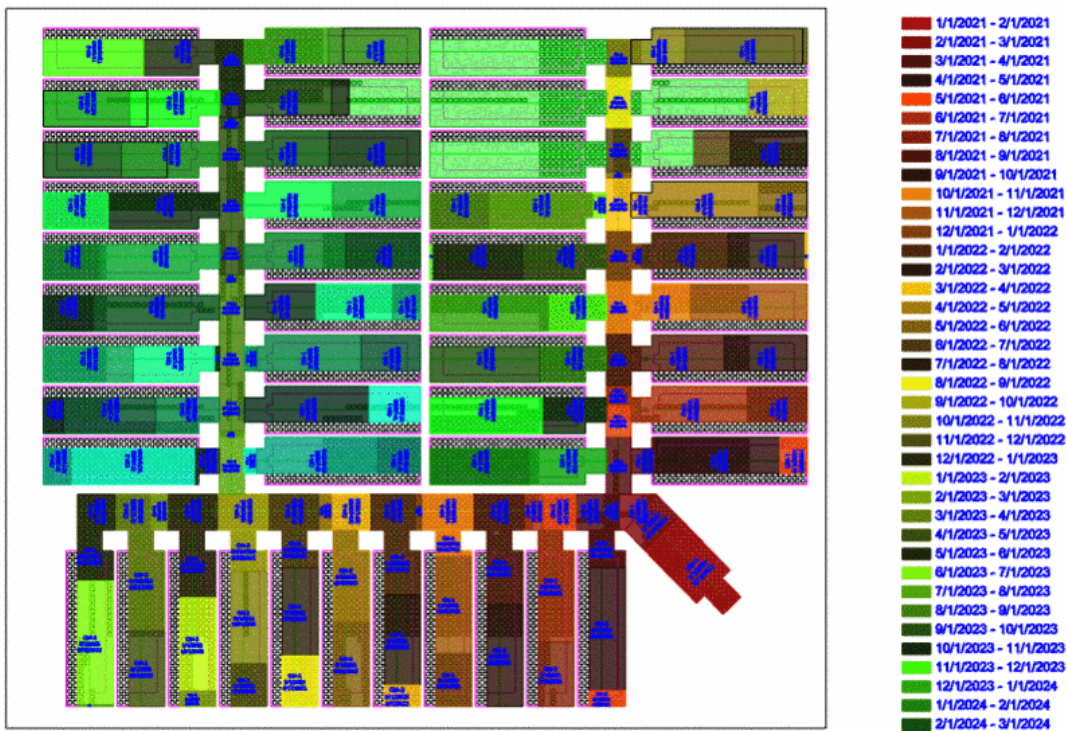
Hatch pillars precisely: Hatches just the entries on the advance, leaving the pillars open. Then if there is retreat mining, the pillars are hatched separately as they are retreat mined.



Output Spatial Database: This option will output the report information to an SQLITE database. The database will retain the coordinate system of the drawing for linkage with GIS software packages. You may specify the file by clicking the ellipsis button to the right of the checkbox.

Skip Report Formatter: This option will skip the report formatter. Rather than customizing the format of the report, the information will be automatically sent to a text file preview.

Examples of a timing map and a generated text report are shown below. Note that there are MANY ways that a timing map and report may be formatted. The Report Formatter is used to generate the text report, but this dialog is not discussed at length here.



Edit : C:\Users\tfaulkner.CARLSON\AppData\Roaming\Carlson Software\Carlson2022\ICAD10_X64\USER\scadrprt.tmp

File Edit Settings

KEY production report Wed Dec 30 11:40:44 2020

Period	Date Start	Date Finish	Equipment	Panel name	KEY (Tons)	Non-Key (Tons)	BTU
1/1/2021 - 2/1/2021	1/1/2021	1/31/2021	CM-1	NORTHWEST MAINS	58223.371	43368.874	14432.959
2/1/2021 - 3/1/2021	2/1/2021	2/12/2021	CM-1	NORTHWEST MAINS	19043.354	15411.990	14299.882
2/1/2021 - 3/1/2021	2/12/2021	2/28/2021	CM-1	WEST MAINS	10747.897	9023.685	14273.549
3/1/2021 - 4/1/2021	3/1/2021	3/15/2021	CM-1	WEST MAINS	7910.411	6903.561	14337.738
3/1/2021 - 4/1/2021	3/16/2021	3/25/2021	CM-2	WEST MAINS	13092.205	11678.153	14414.144
3/1/2021 - 4/1/2021	3/16/2021	3/27/2021	CM-1	NORTH MAINS_1	20132.873	9390.948	14056.301
3/1/2021 - 4/1/2021	3/25/2021	3/31/2021	CM-2	WM-SOUTH-1	14954.896	0.000	14418.362
3/1/2021 - 4/1/2021	3/27/2021	3/31/2021	CM-1	NSM1-E1	9472.397	0.000	14067.392
4/1/2021 - 5/1/2021	4/1/2021	4/30/2021	CM-1	NSM1-E1	62231.197	0.000	14257.628
4/1/2021 - 5/1/2021	4/1/2021	4/30/2021	CM-2	WM-SOUTH-1	62028.413	0.000	14350.145
5/1/2021 - 6/1/2021	5/1/2021	5/6/2021	CM-2	WM-SOUTH-1	7890.053	0.000	14343.465
5/1/2021 - 6/1/2021	5/1/2021	5/8/2021	CM-1	NSM1-E1	13761.483	0.000	14385.956
5/1/2021 - 6/1/2021	5/6/2021	5/22/2021	CM-2	WM-SOUTH-1 RET	69360.292	0.000	14361.410

There are many attributes available in this report. Most of these attributes are self explanatory, but some are not as clear. The below definitions are therefore provided.

- **Difficulty Factor:** This mining rate factor represents the difficulty of mining a specific area. A value of 1.0 means that mining progresses at a normal rate. This value is the product of the Equipment Difficulty and the Location Difficulty.
- **Equipment Difficulty:** This value is the product of the optional difficulty rates set in the equipment definitions. This can include
 - Date Difficulty
 - Thickness Difficulty
 - Bench Difficulty (Surface Mining Module only)
- **Location Difficulty:** This value is a difficulty factor determined by a hierarchy of the following three options. If the high priority item is detected, it will be used instead of the other two options. If the high priority option

is not detected, the medium priority item will be used. If neither of these items are detected, the low priority item will be used. The low priority item defaults to a value of 1.0, so if no location difficulty is defined, the mining rate will not be affected.

- Highest priority: text in the drawing on the DIFFICULTY layer
 - Medium priority: DIFFICULTY attribute value/grid in the Timing Project Manager Attribute Groups
 - Lowest priority: Difficulty value assigned to the specific panel (set with the EDIT_PANEL command)
- Calendar Days: Actual period length between start date and end date (allows for partial days). Note that a day is not necessarily a 24 hour period, but is instead defined by the equipment definition. For example a piece of equipment working two 8-hour shifts means that one calendar day is equivalent to 16 hours.
 - Operating days: Scheduled shifts divided by number of shifts per day (as defined in equipment definition)
 - Days: Scheduled Hours divided by day length in hours (as defined in equipment definition). Note that if all shifts are the same length, this attribute will be the same as the Operating Days.
 - Weekdays Worked: Shifts scheduled on weekdays divided by number of shifts per day
 - Saturdays Worked: Shifts scheduled on Saturdays divided by number of shifts per day
 - Sundays Worked: Shifts scheduled on Sundays divided by number of shifts per day
 - Days Waiting: Number of days equipment cannot work due to a precedence rule
 - Work Shifts: Number of scheduled work shifts. A shift that is not scheduled to work (due to a Calendar Delay) will not contribute to this value.
 - Hours Scheduled: Number of hours equipment is scheduled to work (this includes Hours Utilized, Hours Delayed, and Hours Available)
 - Hours Down: Number of hours equipment is not scheduled to work (due to downtime scheduled on the calendar)
 - Hours Utilized: Number of hours equipment is actually working
 - Hours Delayed: Number of hours equipment is delayed due to a Drawing Event or Scheduled Delay (note, this is not the same as Hours Down caused by a Calendar Delay). Note that a Scheduled Delay will appear on the Assignment List for each piece of equipment
 - Hours Available: Number of hours is scheduled to work, but is neither being utilized nor is it actually delayed. If a piece of equipment encounters a delay during the middle of the shift, the partial shift time remaining will be counted as Hours Available (the delay will be applied to the next shift)

The below image helps visualize the organization of the different definitions of Hours.

Hours in a day	Hours Scheduled	Hours Utilized
		Hours Delayed
	Hours Available	
	Hours Down	

Undo Report: This button will delete a previous timing map and clear the calculated schedule. This allows for quick what-if analysis of the scheduling.

Events Report: This button will display the Events Report, which lists all Drawing Events encountered during the mine schedule. Drawing Events are discussed in more detail in the Timing Project Manager. This report is only available after a Detailed Report has been generated. An example of the Events Report is shown below.

Events Report Wed Dec 30 11:55:49 2020

Event Key	Event Layer	Description	Period	Unit	Pit/Panel Name
Belthead	Belthead	Belthead Installation	1/1/2021 - 2/1/2021	CM-1	NORTHWEST_MAINS
Belthead	Belthead	Belthead Installation	2/1/2021 - 3/1/2021	CM-1	NORTHWEST_MAINS
Belthead	Belthead	Belthead Installation	2/1/2021 - 3/1/2021	CM-1	WEST_MAINS <B:1>
Belthead	Belthead	Belthead Installation	3/1/2021 - 4/1/2021	CM-1	NORTH_MAINS_1 <B:1>
Belthead	Belthead	Belthead Installation	3/1/2021 - 4/1/2021	CM-2	WEST_MAINS <1:2>

Monthly Report: This button will generate a monthly report, similar to the Detailed Report, but with less information available and without the ability to set custom time periods for reporting. An example of the Monthly Report is shown below.

Monthly Unit Report Wed Dec 30 11:57:08 2020

Unit	Month	Year	Operating days	Days Weekdays Worked	Saturdays Worked	Sundays Worked	Calendar days
CM-1	January	2021	23.00	23.00	20.00	3.00	31.00
CM-1	February	2021	22.00	22.00	20.00	2.00	28.00
CM-1	March	2021	25.00	25.00	23.00	2.00	31.00
CM-1	April	2021	24.00	24.00	22.00	2.00	30.00
CM-1	May	2021	23.00	23.00	21.00	2.00	31.00
CM-1	June	2021	24.00	24.00	22.00	2.00	30.00

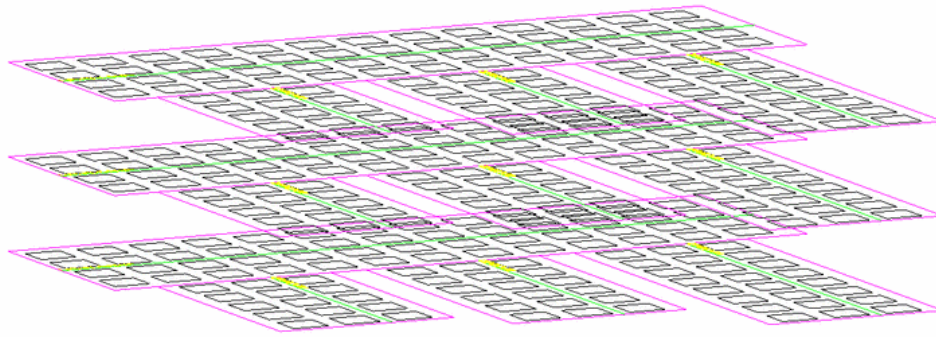
Gantt Chart: This button will generate a Gantt chart of the mine schedule. This button is only available after calculating the mine schedule. For more information, see the Gantt Chart section of the help manual.

Utilization Report: This button will generate a Utilization Report of the mine schedule. This report lists each shift for each piece of equipment. Scheduled delays will be reported as such. An example of the Utilization Report is shown below.

Event Date	Shift	Year (start)	Month/Year	Unit	Equipment	Type	Calendar Rule
1/1/2021	1	2021	01/2021	CM-1	CM-1	Scheduled down	New Years Day
1/1/2021	2	2021	01/2021	CM-1	CM-1	Scheduled down	New Years Day
1/1/2021	1	2021	01/2021	CM-2	CM-2	Scheduled down	New Years Day
1/1/2021	2	2021	01/2021	CM-2	CM-2	Scheduled down	New Years Day
1/1/2021	1	2021	01/2021	CM-3	CM-3	Scheduled down	New Years Day
1/1/2021	2	2021	01/2021	CM-3	CM-3	Scheduled down	New Years Day
1/2/2021	2	2021	01/2021	CM-1	CM-1	DELAY	
1/2/2021	1	2021	01/2021	CM-1	CM-1	UTILIZATION	
1/2/2021	1	2021	01/2021	CM-2	CM-2	AVAILABLE	

Back to Project: This button will return to the Timing Project Manager window.

If a mine has multiple levels with similar panels stacked above one another, a special set of layer names must be used when creating the panels to allow for the distinction between levels. The key to distinguishing mining levels is setting the linework on the correct layer names, which must use a suffix similar to '_LEVELX'. For example, the mine plan in the below figure has three levels. The linework for the first level includes the following layers: MINEPLTX, PILLARS_LEVELX, MINEPLAN, and PERIM_LEVELX. When these panels were made using the Auto Place Panel by Text, the panel name was on a layer named PANELNM_LEVEL1 and the panel perimeter was on a layer named PERIM_LEVEL1. Levels 2 and 3 were made in a similar fashion, but with the respective level name as the suffix for each layer. Although it isn't necessary, the linework for each level in the below image has been drawn at a specific elevation to allow for further visual inspection by the 3DORBIT command.



After the panels have been created, you may also inspect the mining level via the Edit Panel command, which will allow you to enter the Level Layer Suffix, as outlined in red below.

Prompts

Select any part of mine plan: *Pick any centerline in the mine plan*
Underground Timing Dialog

Pull-Down Menu Location: Underground

Keyboard Command: schedule_mine

Clear Timing Map

Removing the coloring after the timing routines are completed may be a tedious task. This routine removes and erases all text labels and hatching from surface pits and underground panels for a specific bench or mining direction in one step. It is a quick and efficient procedure, even for objects on many layers. Items are permanently removed from the drawing screen. It supports both surface and underground mining.

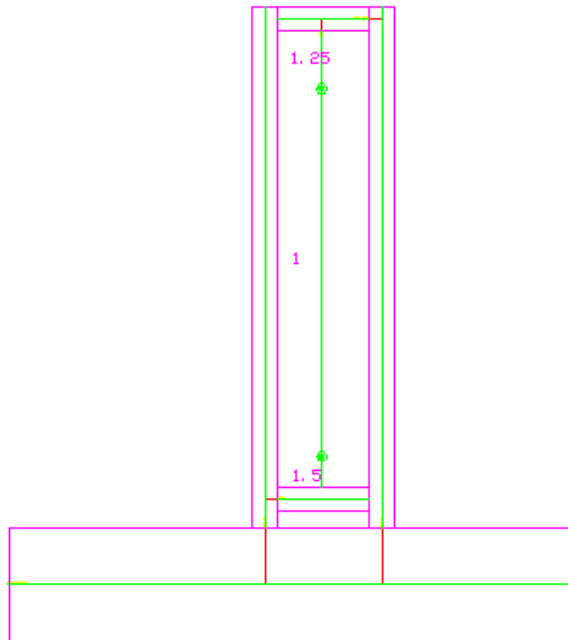
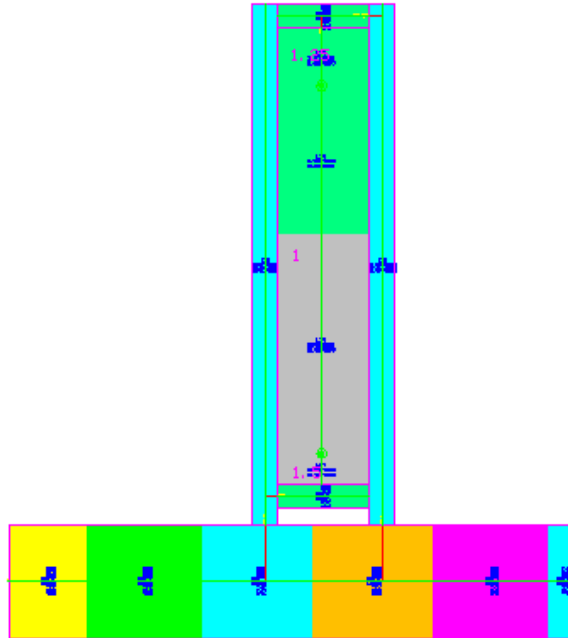
Prompts

For Underground Timing:

Remove coloring for [Advance/Retreat/<Both>]?

Select parts of mineplan to have timing coloring removed:

The following drawings show panels with blocks colored and without blocks colored after using Clear Timing Report.

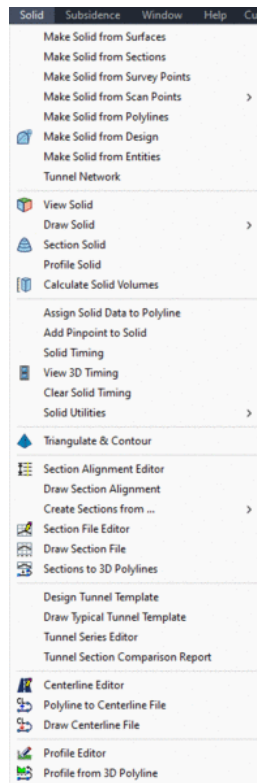


Pulldown Menu Location: Underground

Keyboard Command: utime_clear

Solid Menu

The top portion of the Solid menu has commands specific to the Underground Mining module and are explained in this chapter. The bottom portion of the menu has commands for cross sections and centerlines which are explained in the Civil module of the manual.



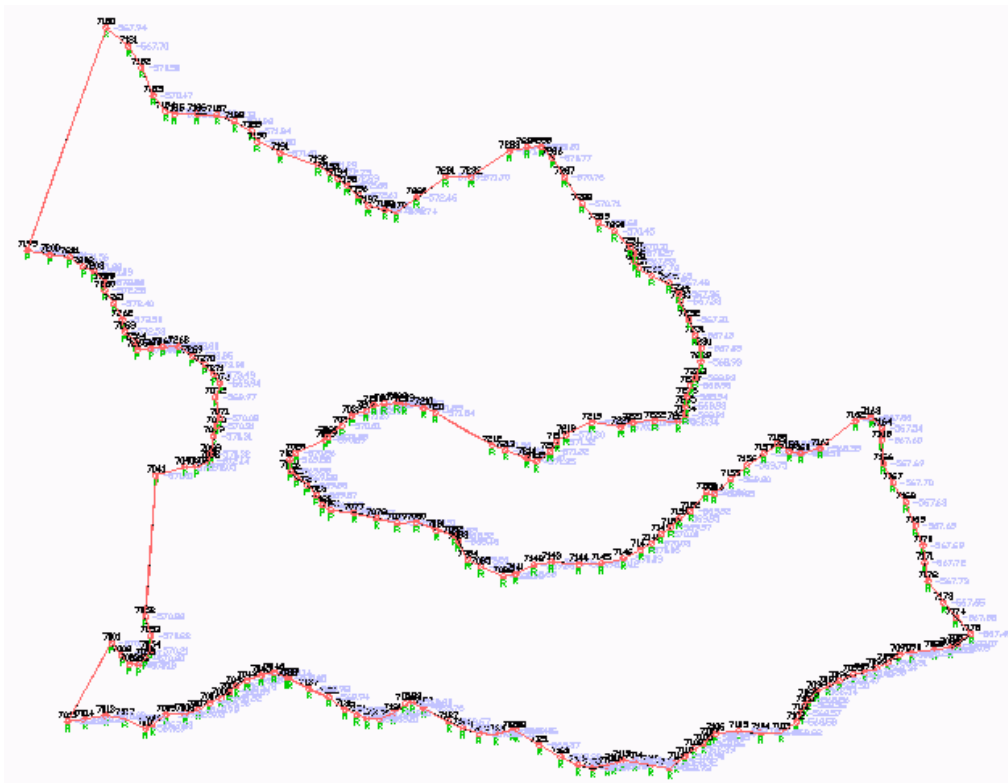
Make Solid From Surfaces

This command creates a solid model using TIN surfaces for the roof and floor plus perimeter polylines. This method for creating a solid model applies when there are survey points on the roof, floor and walls.

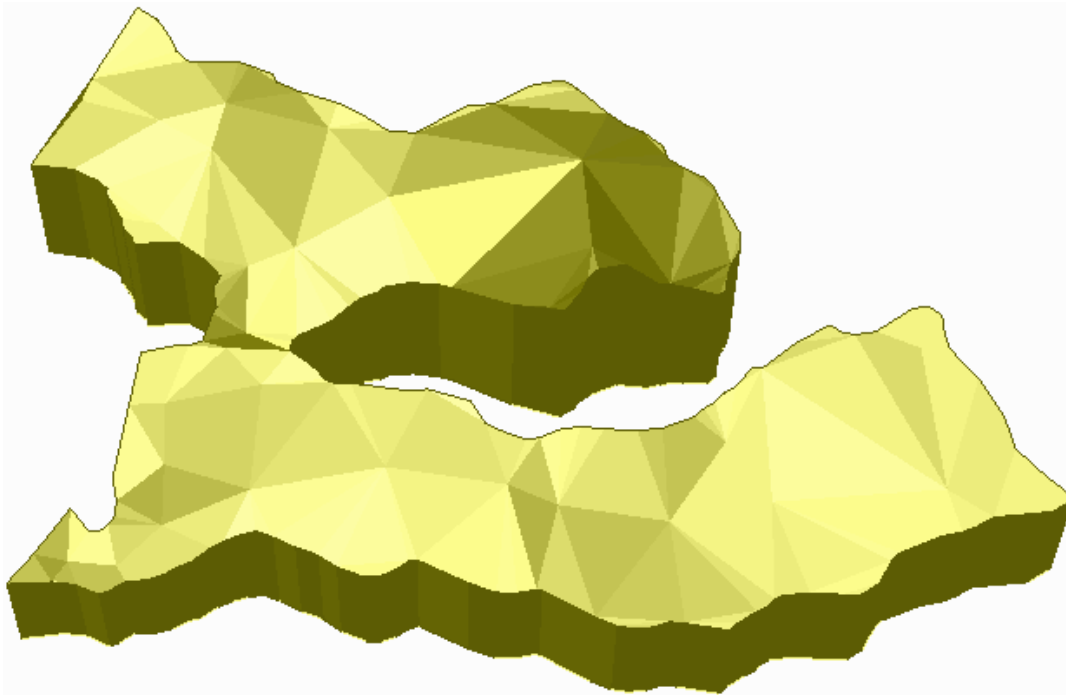
The TIN surfaces can be prepared with routines like Triangulate & Contour in the Civil module. In Triangulate & Contour to build the roof surface, turn on the option to Write Triangulation File and select the roof points. Then repeat Triangulate & Contour to build the floor surface by selecting the floor points.

The perimeter polylines represent the walls of the solid and can be either 2D or 3D polylines. For 3D, the polylines should have elevations between the floor and roof. If there are survey points on the walls, these can be connected into 3D polylines by using commands like Draw 3D Polyline. Multiple 3D polylines can be used to create the solid model. 3D polylines that are inside other 3D polylines are used as exclusion areas for the solid model and create holes in the model. To build the solid model, the program connects the 3D polylines to the roof and floor surfaces.

For 2D perimeters, the polyline is used to crop the top and bottom surfaces. The solid is built by then connecting the top and bottom surfaces.



3D rib polyline and survey points



3D view of solid model

Prompts

Top Surface To Read *Select TIN file*

Bottom Surface To Read *Select TIN file*

Select rib polylines.

Select objects: *pick 3d polyline(s)*

Solid File To Write *Select MDL file*

Example Workflow

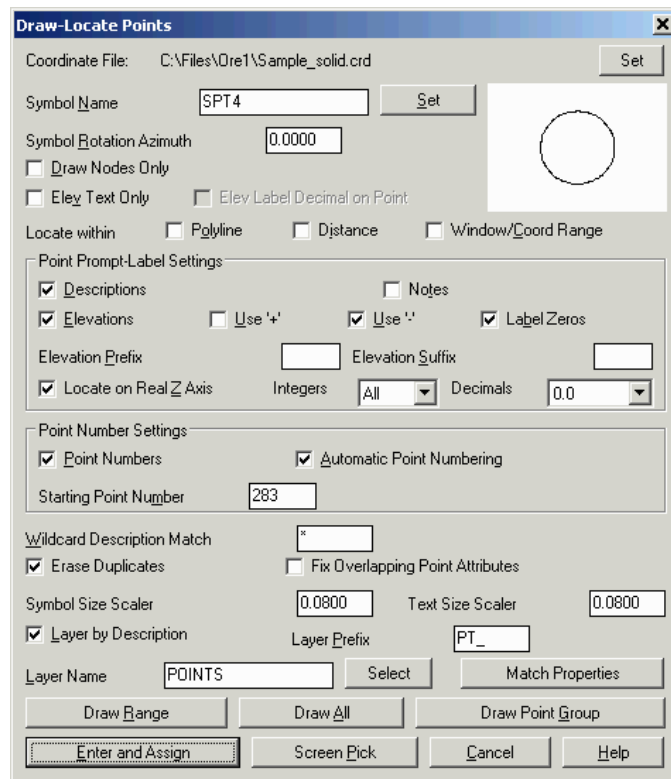
This example uses survey point data to build the solid by creating the rib 3D polyline and top and bottom surfaces.

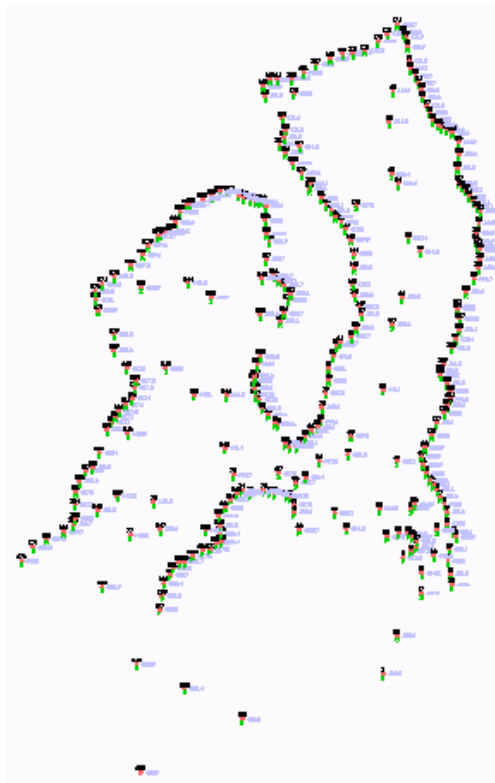
Step 1: Import Points

From a new drawing, run **Points > Set Coordinate File** and select a coordinate file named sample_solid.crd from the Carlson Projects folder.

To have the points drawn at a good size, run **Settings > Drawing Setup** and set the Horizontal Scale to 10 and the units to English.

Next run **Points > Draw/Locate Points** and turn on the option to **Layer By Description** and then pick **Draw All**. In this example, the points have different description codes to identify roof, floor and rib points.

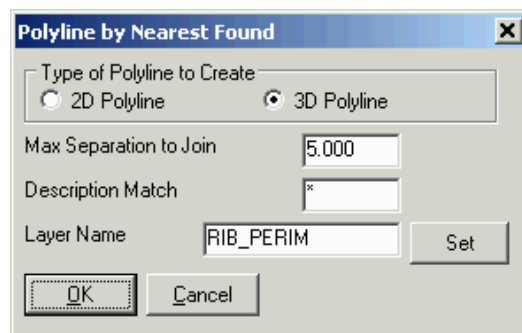




Step 2: Draw Rib 3D Perimeter Polyline

There are two codes of "R" and "P" for points on the perimeter. To view only the perimeter points, run **View > Isolate Layers** and pick on a point with a "R" description and a point with a "P" description.

Next run **Draw > Polyline > Polyline By Nearest Found**. Set the type to 3D Polyline, the max separation to 5 and the layer to RIB_PERIM.



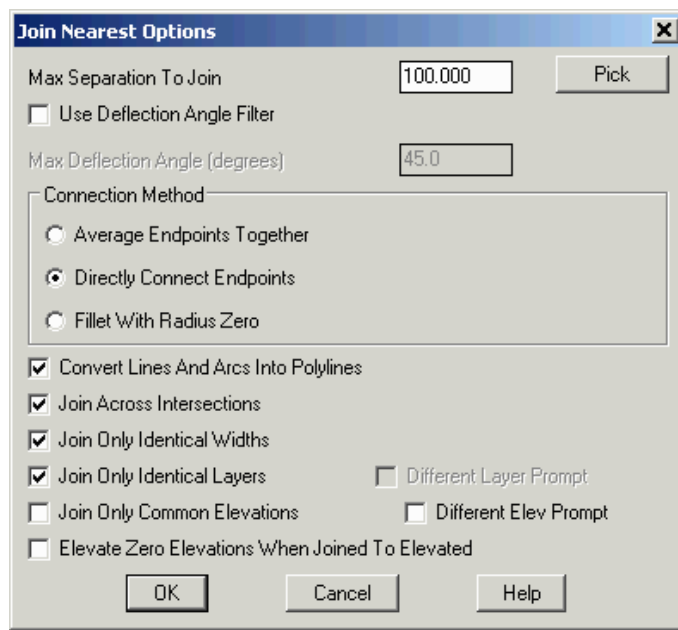
Then choose the Screen selection mode and then select all the points.

Select point from screen or by point number [<Screen>/Number]? *press Enter*

Select points.

Select objects: *all*

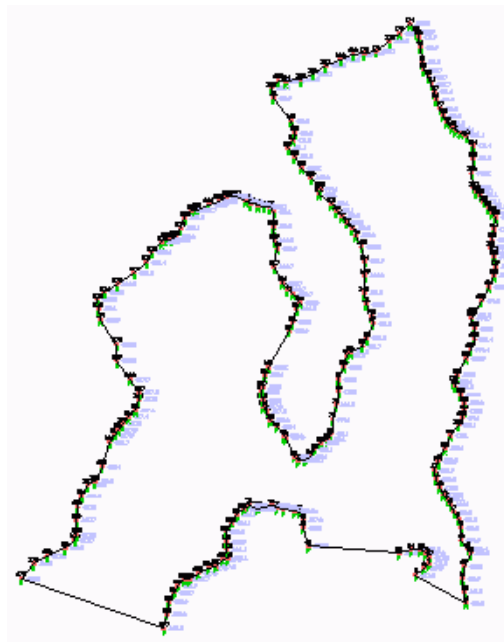
The Polyline By Nearest Found connected the points except for the large gaps. To create a single closed polyline and span these gaps, run **Edit > Join Nearest**. Set the max separation to 100 and the method as Directly Connect Endpoints.



Then select all the polylines to be connected.

Select lines, arcs and unclosed polylines to join.

Select objects: *all*



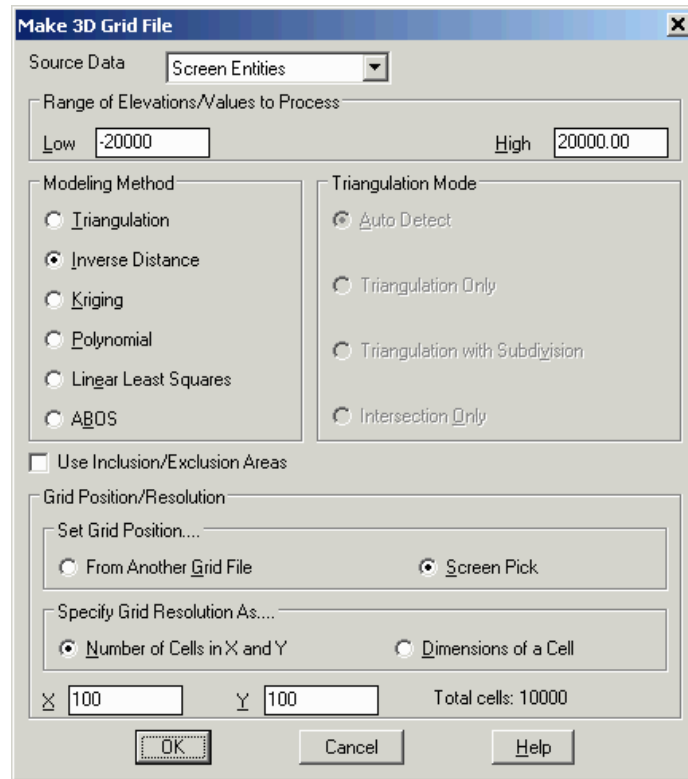
Step 3: Draw Top and Bottom Perimeter Polylines

To use for building the top and bottom surfaces, the next step is to create top and bottom 3D perimeter polylines. First, turn all the points back on by running **View > Restore Layers**.

The bottom points have a description of "S". Run **View > Isolate Layers** and pick on a "S" point.

To elevate the perimeter polyline at the bottom surface, a grid file is a good way to extrapolate the bottom points to

cover the perimeter. Run the **Surface > Make 3D Grid File** command from the Civil module. Set the file name as bottom.grd. In the options dialog, set the method as Inverse Distance and the grid position as Screen Pick.



For the grid corners, pick a large window around the points to generously make an area large enough to cover the perimeter.

Pick first grid corner: *pick the lower left point*

Pick opposite grid corner: *pick the upper right point*

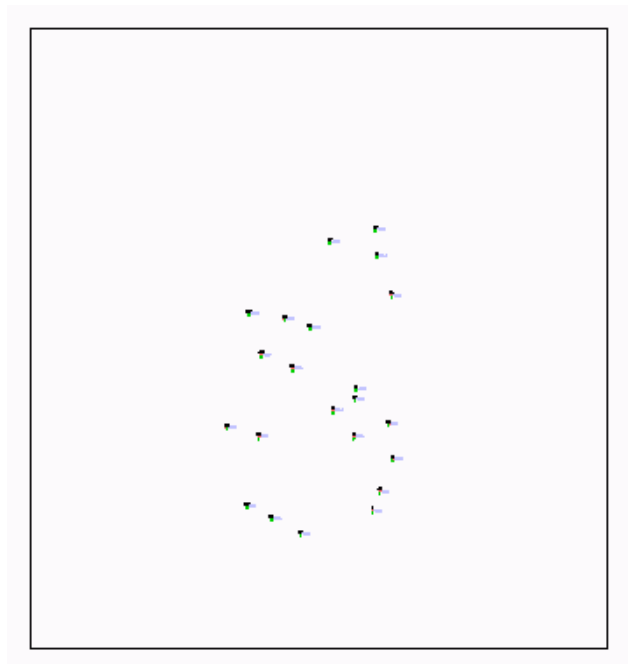
Select points, lines, polylines and faces to grid from.

Select objects: *all*

Reading points ... 29

Use inverse distance to which power [First/<Second>/Third/Other]? *press Enter*

Use elliptical inverse distance [Yes/<No>]? *press Enter*



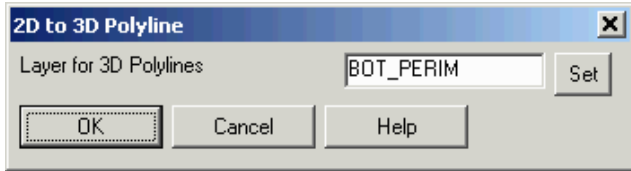
Now that we have this bottom surface, we can create a bottom 3D perimeter polyline. Turn on all the layers again by running **View > Restore Layers**. Then run **3D Data > 2D to 3D Polyline > By Surface Model** from the Civil model. Choose the bottom.grd as the surface to process. Pick the rib polyline as the polylines to convert. Be sure to answer YES to the Keep Existing Polyline prompt. Set the layer to BOT_PERIM.

Select polylines to convert.

Select objects: *pick the rib polyline*

Use current polyline elevations as vertical offset from surface [Yes/<No>]? *press Enter*

Keep existing polylines [Yes/<No>]? *Y*



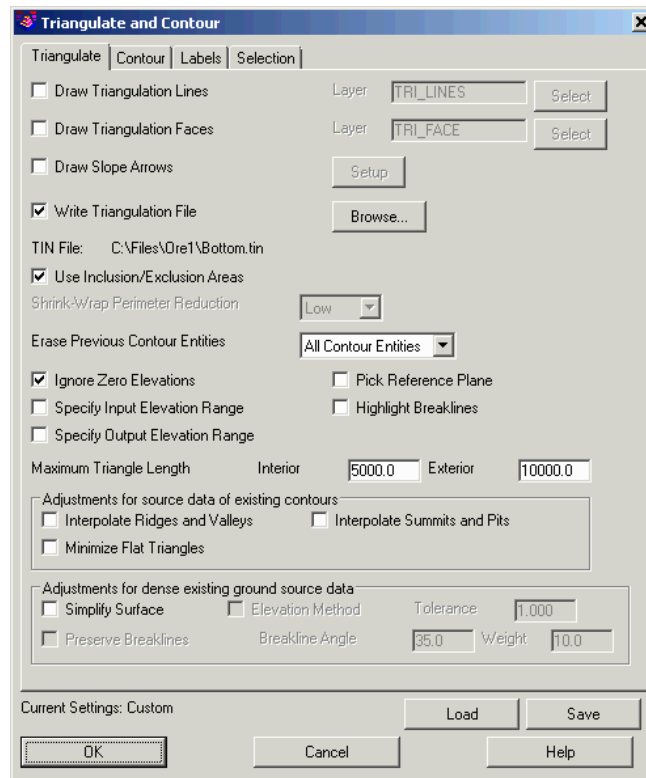
The bottom perimeter polyline is done. Now let's create the top perimeter polyline using the same procedure. Run **View > Isolate Layers** and pick on a point with a "B" description. Run **Surface > Make 3D Grid File** again and use the same inputs except create a grid called top.grd. Then turn the layers back on by running **View > Restore Layers**. Next run **3D Data > 2D to 3D Polyline > By Surface Model** and pick the rib polyline as the polyline to convert, answer YES to the Keep Existing Polyline and set the layer to TOP_PERIM.

Step 4: Create Top and Bottom Surfaces

The data is prepared and now the surfaces can be created. Let's build the bottom surface first. Since the perimeter polyline for rib, top and bottom are on top of each other, it is tricky to select them individually. Run **View > Layer Control** and freeze the layers for RIB_PERIM and TOP_PERIM. Then run **View > Isolate Layers** and pick on a "S" point and the bottom perimeter polyline.



Next run Surface > Triangulate & Contour. On the Triangulate tab, turn on Write Triangulation File and set the file to bottom.tin. Also, turn on Use Inclusion/Exclusion Areas. On the Contour tab, turn off Draw Contours.



Pick OK on the dialog. Then select the perimeter polyline for the Inclusion perimeter. For the Exclusion perimeter prompt, press Enter for none. Then for the select points prompt, type ALL.

Select Inclusion perimeter polylines or ENTER for none.

Select objects: *pick perimeter polyline*

Select Exclusion perimeter polylines or ENTER for none.

Select objects: *press Enter*

Select points and breaklines to Triangulate.

Select objects: *all*

The bottom surface is all done. Now let's create the top surface. Run View > Restore Layers. Then run View >

Layer Control to thaw the TOP_PERIM layer and freeze the BOT_PERIM and RIB_PERIM layers. Then Run View > Isolate Layers and pick on a "B" point as well as the perimeter polyline.

Then run Surface > Triangulate & Contour. Set the TIN file to write as top.tin and then pick OK. At the command line, select the perimeter polyline as the Inclusion and select all for the points to triangulate.

Step 5: Create Solid Model

We need to view the rib perimeter polyline for creating the solid. Run **View > Layer Control** to thaw the RIB_PERIM layer and freeze the TOP_PERIM and BOT_PERIM layers.

Finally, run **Solid > Make Solid From Surfaces**. Select the top and bottom surfaces created in step 4 and the rib polyline created in step 2. Then specify the solid model file name to create.

Top Surface To Read *Select TOP.TIN file*

Bottom Surface To Read *Select BOTTOM.TIN file*

Select rib polylines.

Select objects: *pick rib polyline*

Solid File To Write *Enter a file name*

Pulldown Menu Location: Solid

Keyboard Command: make_solid

Prerequisite: 3d perimeter polyline and two surface files

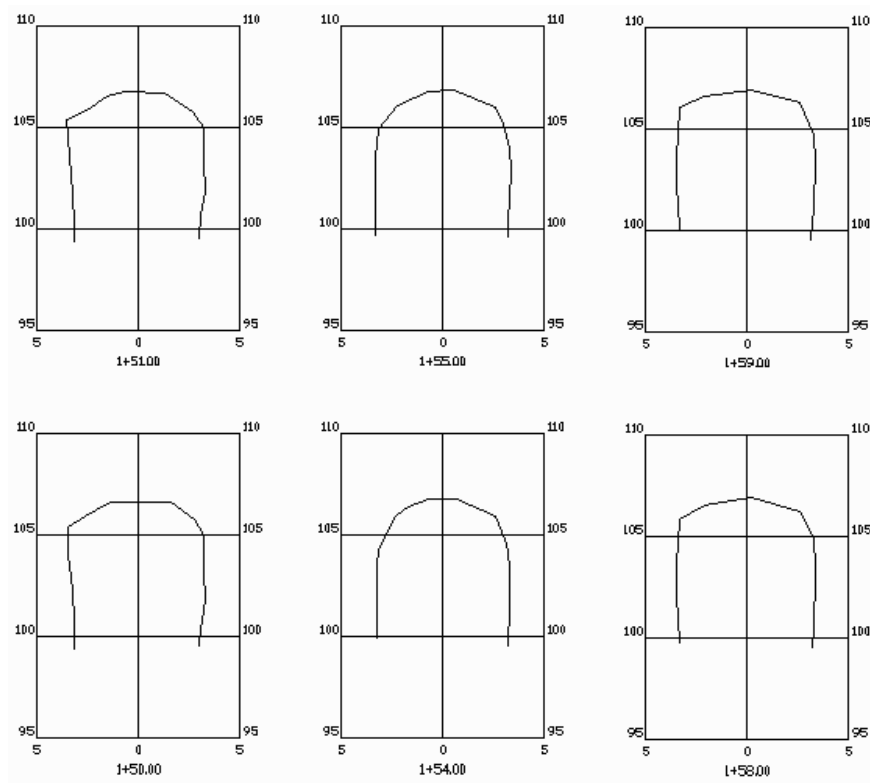
Make Solid From Sections

This command creates a solid model using cross sections and a centerline alignment. For example, tunnel cross sections can be used for this method.

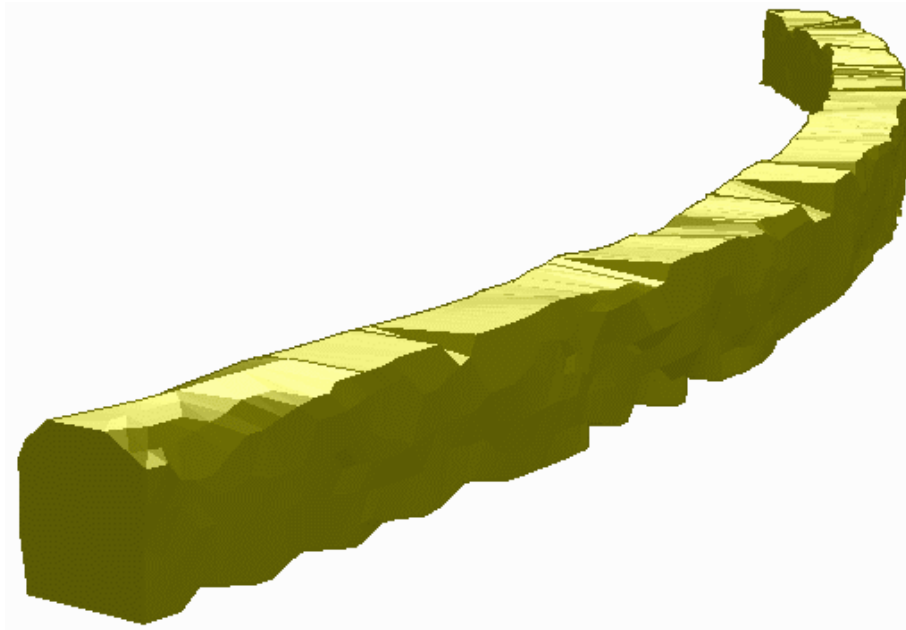
The cross sections can be prepared with the Create Sections routines in the Solid menu. There are also routines to prepare the centerline file in the Solid menu.

The settings dialog has an option to Merge With Existing Solid for merging the solid model for the new cross section data with an existing solid model. This option applies to combining data for intersection tunnels. The Merge Tolerance is the distance for replacing the previous solid model data points with the new data points.

To build the solid model, the program connects the data between the cross section stations and caps the solid model at the ends.

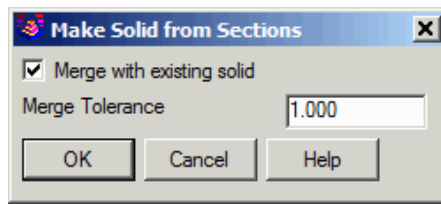


Sample input cross section data



3D view of solid model from cross sections

Prompts



Make Solid From Sections options dialog

Section File To Process *Select SCT file*

Centerline File To Process *Select CL file*

Solid File To Write *Select MDL file*

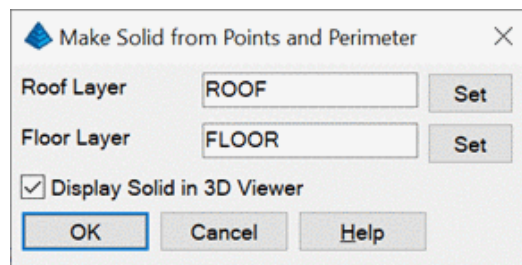
Pulldown Menu Location: Solid

Keyboard Command: sct2solid

Prerequisite: cross sections and centerline files

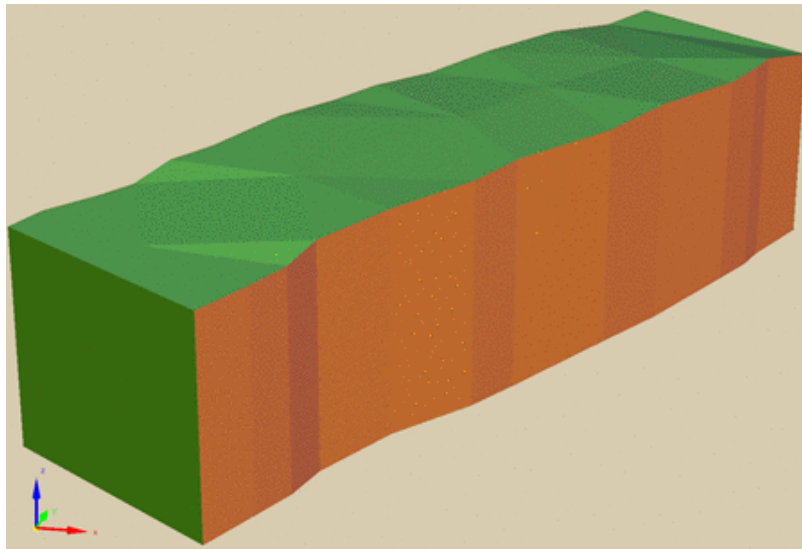
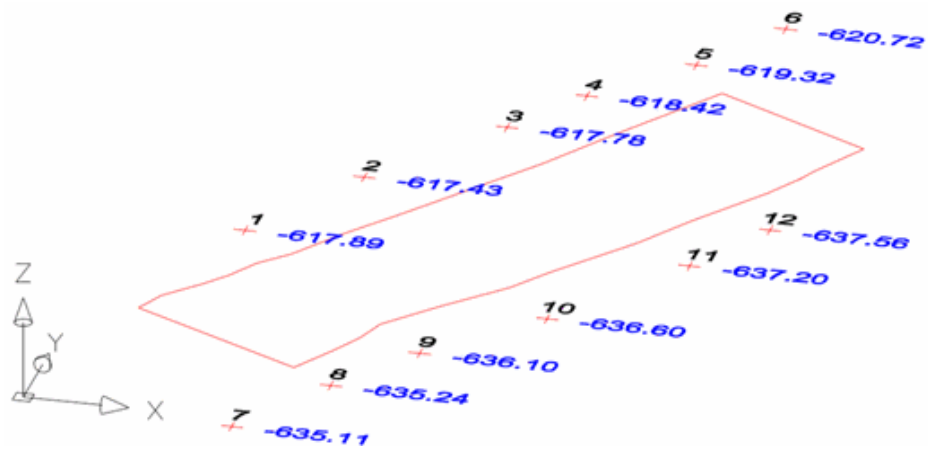
Make Solid From Survey Points

This command creates a solid model from survey points on the roof and floor plus a 3D perimeter polyline. This method applies to input data of relatively few data points compared to a point cloud. All the data points must be on the roof or floor and not the sides. The roof and floor points must be drawn on separate layers that the program prompts for in the options dialog.



The solid is built by modeling a surface from the roof points and a separate solid from the floor points. The surfaces are trimmed by the perimeter polyline and then stitched together with the perimeter polyline to make a watertight solid. The resulting solid is stored to the specified MDL file.

In the example shown here, points 1 to 6 are on the ROOF layer and points 7 to 12 are on the FLOOR layer. The perimeter polyline is shown in 3D between these points.



Prompts

Make Solid dialog

Select points to process.

Select objects: *pick the points*

Select rib polylines.

Select objects: *pick the 3D perimeter polyline*

Solid File To Write *Select MDL file*

Pulldown Menu Location: Solid

Keyboard Command: survey_solid

Prerequisite: points and perimeter polyline

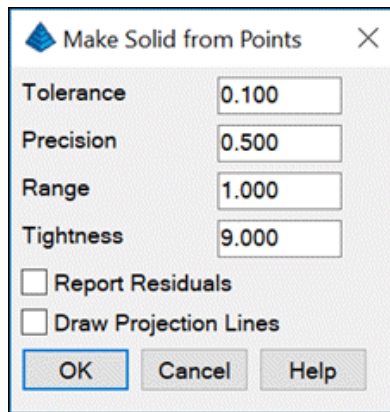
Make Solid From Points By Poisson

This command creates a solid model from a collection of x,y,z points by using the Poisson method. The points need to cover all sides of the surface (top, bottom, left, right, front, back). The Poisson method is a 3D surface

reconstruction algorithm employed to rebuild 3D surfaces from point samples. The screened Poisson formulation considers all the points at once, without resorting to heuristic spatial partitioning or blending, and is therefore highly resilient to data noise. Since the input for this routine uses points in the drawing, the difficulty of CAD to handle a very large number of points limits the number of points this routine can process. This function is also available in the Point Clouds module which stores the data outside of CAD and can process more input points.

The program prompts for CAD point entities in the drawing as the input for the model. Next the program has a dialog for the processing options.

The solid model is designed to follow the source points but does not exactly pass through each point. The options in the dialog control how well the model fits the points and has options for reviewing how well the model fits.



Tolerance: Indicates how much noise is expected in the point data. This setting helps remove outlier points. If the resulting surface is too spikey, adjust this value higher.

Precision: Controls pre-processing calculations. A smaller value results in a tighter solid.

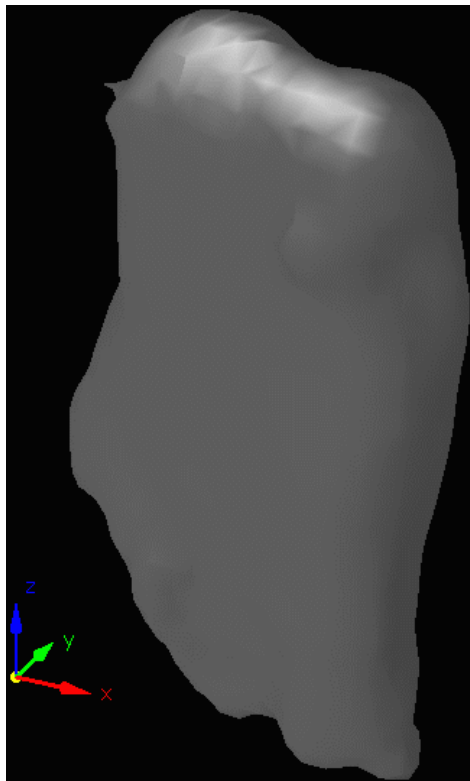
Range: Controls how far the model looks for connecting points. If the solid does not span over expected gap areas, raise this value higher. If the solid spans over unexpected areas, lower the value.

Tightness: Controls the density of the triangles. The higher the value, the smaller the triangles of the resulting solid.

Report Residuals: Reports the residual distances between the source points and the solid model.

Draw Projection Lines: Draws lines from the source points to the solid model.

Smooth points before triangulation: This option applies a moving least-squares smoothing to the points to reduce noise. The Save Sparse Points controls whether to save sparse points which would be thrown out with point smoothing. If points do not have at least 3 neighboring points within the search radius they are discarded. This setting will preserve them.



Prompts

Select points to process.

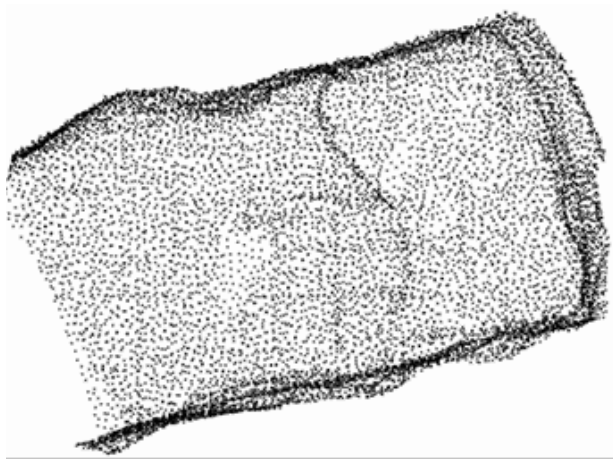
Select objects: *pick points*
Points to Solid options dialog
Solid File To Write *Select MDL file*

Pulldown Menu Location: Solid
Keyboard Command: pt2solid
Prerequisite: points

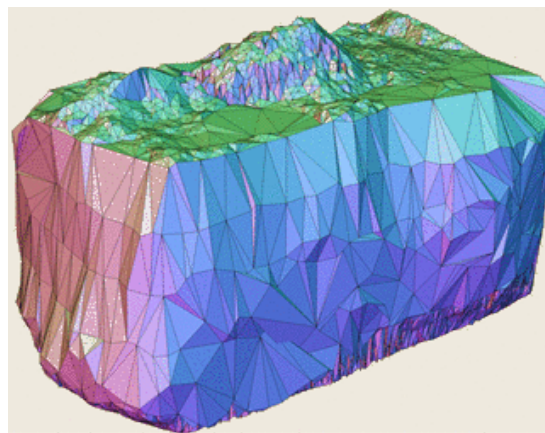
Make Solid From Points By Sections

This command creates a solid model from a collection of x,y,z points by creating cross sections from the points and then stitching the sections together.

Here is an example of an underground cavity with data points all around: top, bottom and sides. One way to model this case is to set the Model Type to Closed, Triangulate Ends to 3D TIN and Section Alignment to Vertical.



Plan view of data points

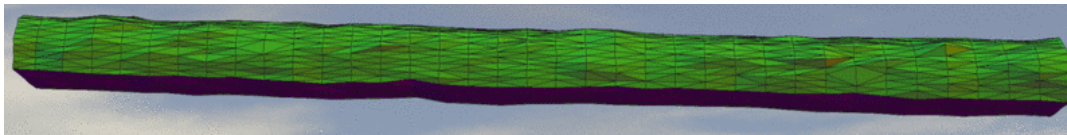


Resulting 3D Solid colored by normal and showing edges

Here is an example of a tunnel with points at a section interval. For this case, use the CAD Orbit command to rotate the CAD view so that you're looking down the tunnel. Then in this command, set the Section Interval to match the data, Model Type as Closed, Section Alignment as Current 3D View and Triangulate Ends as None.

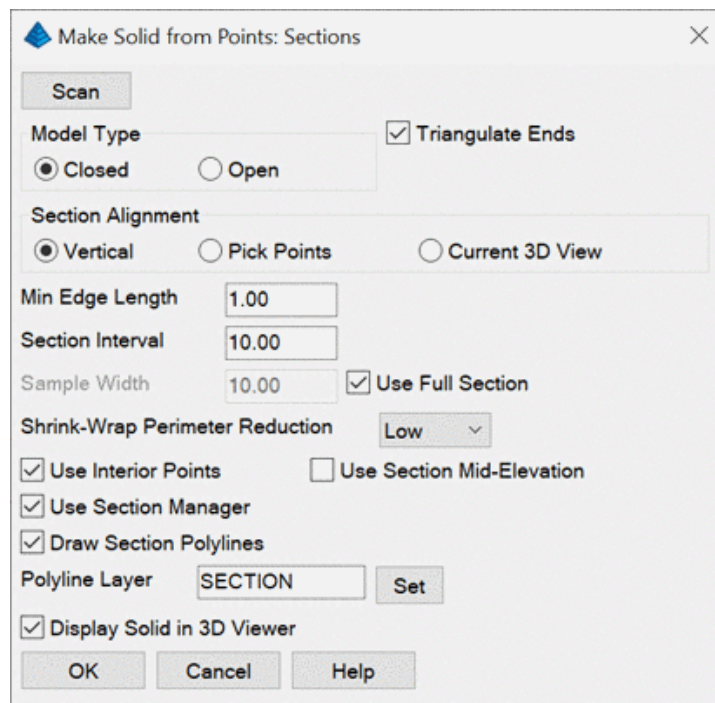


CAD 3D View from Orbit aligned with tunnel



Resulting 3D Solid

colored by normal and showing edges



Scan: Reads the input points to find the min and max elevations which are reported at the top of the dialog.

Model Type: Controls whether to create a watertight closed solid such as an underground cavity, or create an open surface solid such as a cliff face. For a Closed model, the points need to cover all sides of the surface (top, bottom, left, right, front, back).

Triangulate Ends: Controls whether to triangulate and cap off the end sections. The By Section method triangulates the section end polylines. The 3D TIN method triangulates all the end points. The None method leaves the ends open such as for a tunnel.

Section Alignment: Sets how to section the points. It is important to align the sections with the data to get good sections. For example, with a tunnel the alignment should go along the length of the tunnel so that the sections are

like slicing a carrot into discs. The Vertical method slices by elevation range. The Pick Points method prompts for two points and uses the 3D line between these points for the alignment. The Current 3D View method uses the CAD view. This way you can use the CAD 3D view commands to line up a good view for the points.

Min Edge Length: Sets the minimum length of a triangle edge in the solid. When there are several points closer than this distance to each other, some of these points will be left out of the solid model.

Section Interval: Sets the distance between sections along the alignment. This distance needs to be big enough to ensure that plenty of points are used for each section, and the distance needs to be small enough to create enough sections to model the solid.

Sample Width/Use Full Section: Controls whether to use all the data points between sections or limit the data points to the specified distance from the section line.

Shrink-Wrap Perimeter Reduction: The program triangulates the points for each section and uses the boundary of this triangulation for the section. This option controls how much to tighten this boundary polyline by removing long edges.

Use Interior Points: This option shrink-wraps the interior points and then finds the median with the outer shrink-wrap to create the section polyline.

Use Section Mid-Elevation: Controls whether the section lines use the individual elevations of the data points or the same average elevation for the section.

Use Section Manager: This option shows the generated sections in an editor for making changes before building the model. See the Edit Solid topic for more information on this feature.

Draw Section Polylines: This option draws 3D polylines for the sections on the specified layer.

Display Solid in 3D Viewer: This option will automatically run the 3D Viewer for the resulting solid at the end of the command.

Prompts

Select points to process.

Select objects: *pick points*

Points to Solid options dialog

Solid File To Write *Select MDL file*

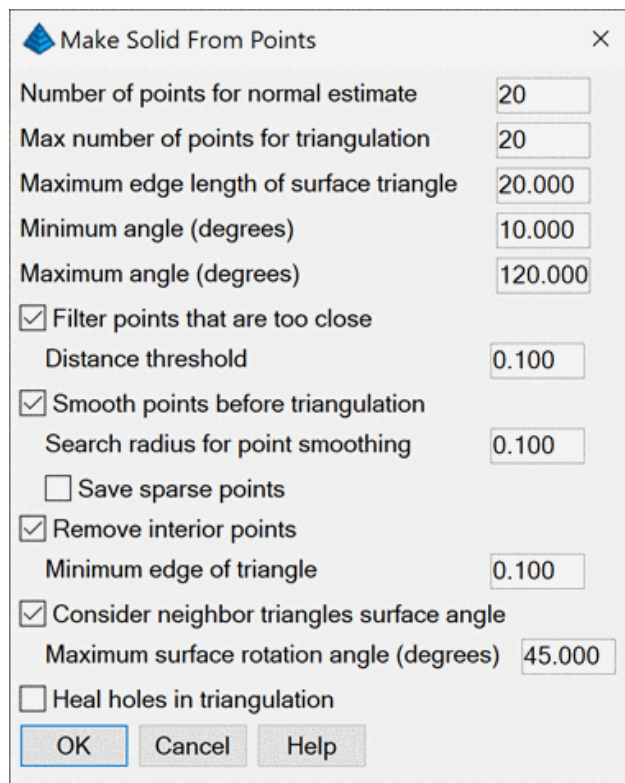
Pulldown Menu Location: Solid

Keyboard Command: pt2mdl

Prerequisite: points

Make Solid From Points By 3D Triangulation (Projection)

This command creates a solid model from a collection of x,y,z points using the projection or "greedy" 3D triangulation method. The points need to cover all sides of the surface (top, bottom, left, right, front, back).



Number of points for normal estimate: For each point, the program calculates the normal using the specified number of neighboring points.

Max number of points for triangulation: The number of points to consider when connecting each point. More points should increase accuracy, but increase run time.

Maximum edge length of surface triangle: Sets the maximum distance for 3D triangulation to connect points.

Minimum angle: The minimum acceptable interior angle of a triangle which is used to make up the triangulated mesh.

Maximum angle: The maximum acceptable interior angle of a triangle which is used to make up the triangulated mesh.

Filter points that are too close: Sets the minimum length of a triangle edge in the solid. When there are several points closer than this distance to each other, some of these points will be left out of the solid model.

Smooth Input Points: This option applies a moving least-squares smoothing to the points to reduce noise which can help a lot with calculating the normals and improving the triangulation. The Save Sparse Points controls whether to save sparse points which would be thrown out with point smoothing. If points do not have at least 3 neighboring points within the search radius they are discarded. This setting will preserve them.

Remove interior points: Runs a routine to remove the interior points of the data set. The routine checks that points fall within a triangulated mesh of varying triangle sizes. Using smaller triangles should capture more detailed exterior surfaces, but are prone to labeling interior points as exterior points. Larger triangles will capture less surface detail but are more likely to drop interior points. The triangle size is set by "Minimum edge of triangle".

Consider neighbor triangles surface angle: Whether to limit the rotation between neighboring triangles on the mesh surface. Considering it will limit the possible rotation angle to user-specified setting in degrees. 0 would require a totally flat surface, 90 degrees would allow for a cube surface.

Heal holes in triangulation: triangulates across small holes in the solid that remain after the 3D triangulation so that the final surface is watertight.

Prompts

Select points to process.

Select objects: *pick points*

Points to Solid options dialog

Pulldown Menu Location: Solid

Keyboard Command: pt2triang

Prerequisite: points

Make Solid From Points By 3D Triangulation (Tunnel)

This command creates a solid model from a collection of x,y,z points and an alignment polyline using the 3D triangulation. The points need to cover all sides of the surface (top, bottom, left, right). The alignment polyline needs to be a 3D polyline that goes through the center of the points for the entire length.

Min Edge Length: Sets the minimum length of a triangle edge in the solid. When there are several points closer than this distance to each other, some of these points will be left out of the solid model.

Max Edge Length: Sets the maximum distance for 3D triangulation to connect points.

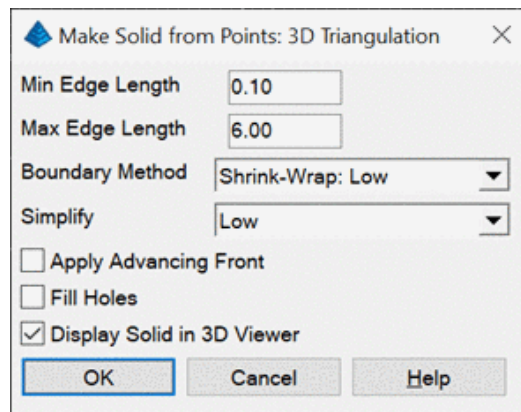
Boundary Method: controls how tightly the solid model wraps around the perimeter points.

Simplify: removes points that cause steep triangles.

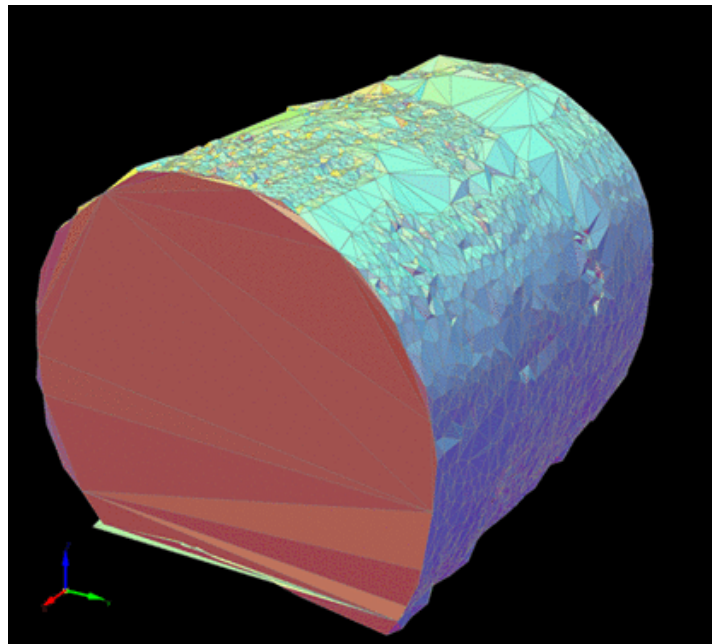
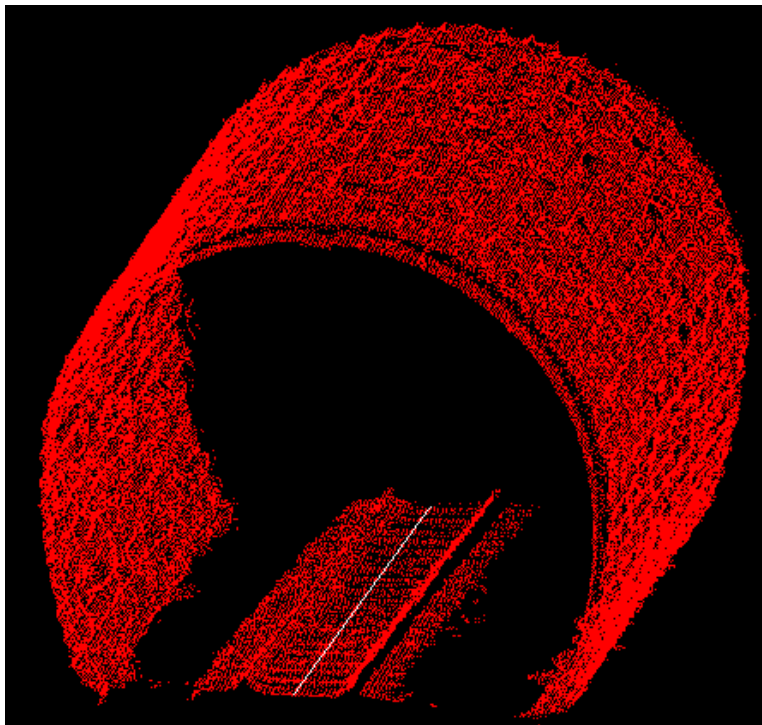
Apply Advancing Front: expands the solid by adding triangles to unconnected edges.

Fill Holes: triangulates across small holes in the solid.

Display Solid in 3D Viewer: automatically runs the View Solid command with the model results.



This example of a tunnel shows the input points in CAD along with the alignment polyline along the center of the train tracks and then the resulting solid model viewed with color by normal.



Prompts

Make Solid dialog

Select alignment polyline: *pick 3D polyline*

Select points to process.

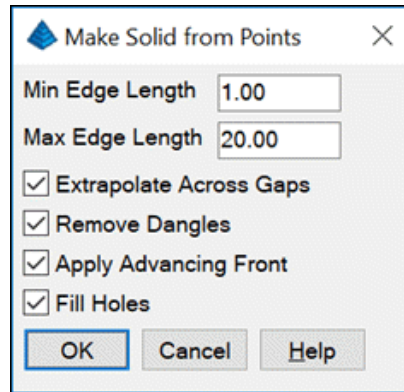
Select objects: *pick points*

Solid File To Write *Select MDL file*

Pulldown Menu Location: Solid
Keyboard Command: pt3dtunnel
Prerequisite: points

Make Solid From Points By 3D Triangulation (Advancing Front)

This command creates a solid model from a collection of x,y,z points using the "advancing front" 3D triangulation method. The points need to cover all sides of the surface (top, bottom, left, right, front, back).



Min Edge Length: Sets the minimum length of a triangle edge in the solid. When there are several points closer than this distance to each other, some of these points will be left out of the solid model.

Max Edge Length: Sets the maximum distance for 3D triangulation to connect points.

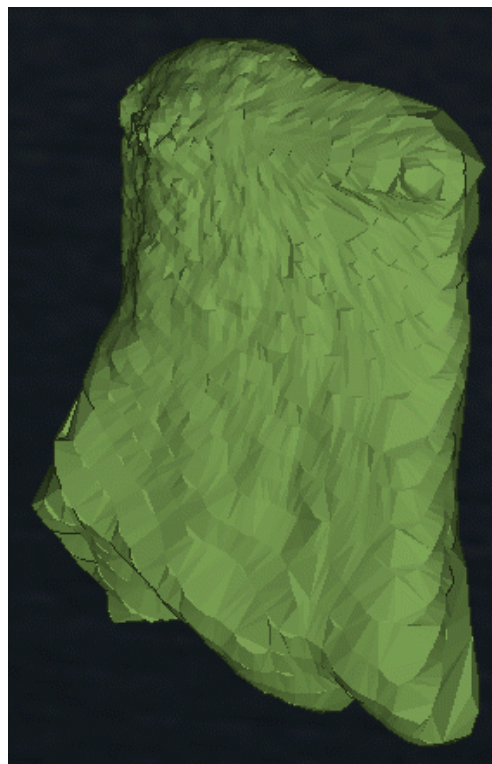
The rest of these options apply to expanding and improving the solid after the 3D triangulation has connected all the points that it can.

Extrapolate Across Gaps: looks for large gaps in the input data and uses a Poisson model points to fill in those gaps.

Remove Dangles: removes triangles that are not connected to the majority of the solid model.

Apply Advancing Front: expands the solid by adding triangles to unconnected edges.

Fill Holes: triangulates across small holes in the solid.



Prompts

Points to Solid options dialog

Select points to process.

Select objects: *pick points*

Solid File To Write *Select MDL file*

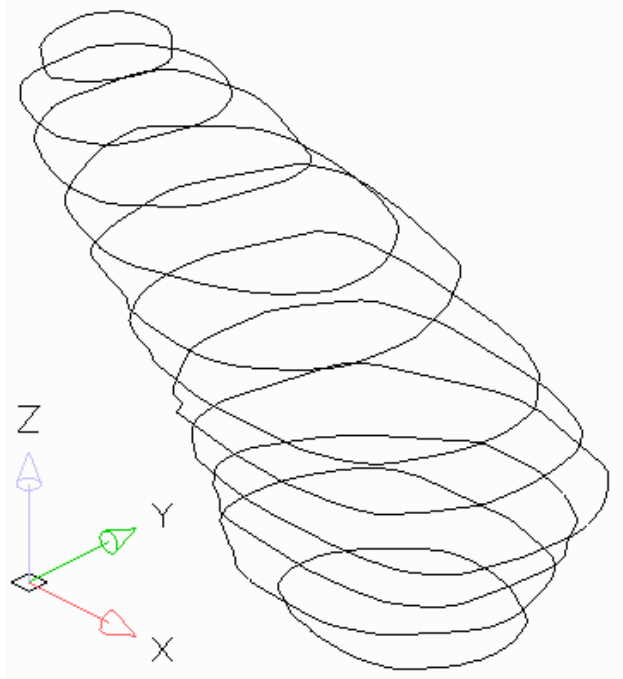
Pulldown Menu Location: Solid

Keyboard Command: pt2mdl

Prerequisite: points

Make Solid From Polylines

This command creates a solid model file (.MDL) by stitching together a series of closed 3D polylines that form the ribs or series of hoops for the solid.





Prompts

Select rib polylines to process.

Select objects: *pick the 3D polylines*

Solid File To Write *Select MDL file*

Pulldown Menu Location: Solid

Keyboard Command: pl2solid

Prerequisite: closed 3D polylines

Make Solid From Design

This command creates a solid model using a tunnel template, centerline and profile. The tunnel template defines the cross section. The centerline defines the horizontal alignment and the profile define the vertical alignment. Using the design files, the program creates a cross section for the solid at the specified station interval along the alignments.

Before running this command, the design files must already be created. The tunnel template is created with the Design Tunnel Template command. The centerline can be created by many commands including Centerline Editor. The profile can be created by many commands including Profile Editor.

Use 3D Polyline For Alignment: prompts for a 3D polyline for the horizontal and vertical alignment instead of using centerline and profile files.

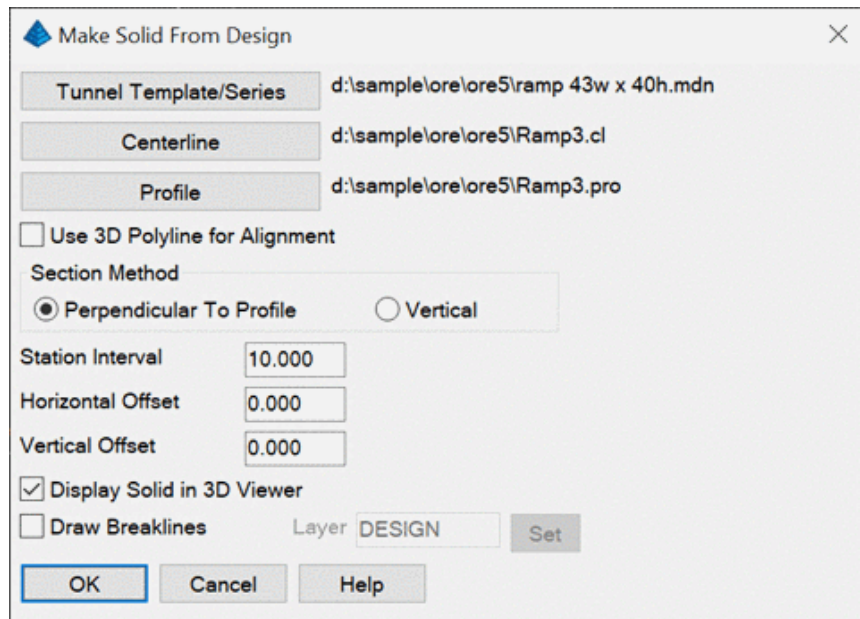
Section Method: controls whether to create the tunnel sections perpendicular to the profile slope or vertical.

Station Interval: sets the interval to create tunnel sections along the alignment.

Horizontal and Vertical Offset: shifts the tunnel template from the alignment such as for a circular template with the profile in the center.

Display Solid in 3D Viewer: automatically runs the View Solid command with the design results.

Draw Breaklines: creates 3D polylines for sections of the design at the station interval.



Prompts

Make Solid From Design dialog

Solid File To Write *Select MDL file*

Pulldown Menu Location: Solid

Keyboard Command: design2solid

Prerequisite: tunnel template and alignment

Make Solid From Entities

This command creates a .mdl graphics files from selected 3D faces. These graphic files are used in 3D viewing commands like 3D Drive Simulation. The Settings > 3D Model Library manages these graphics files. The .mdl file is also used for Solids in the Underground Mining module.

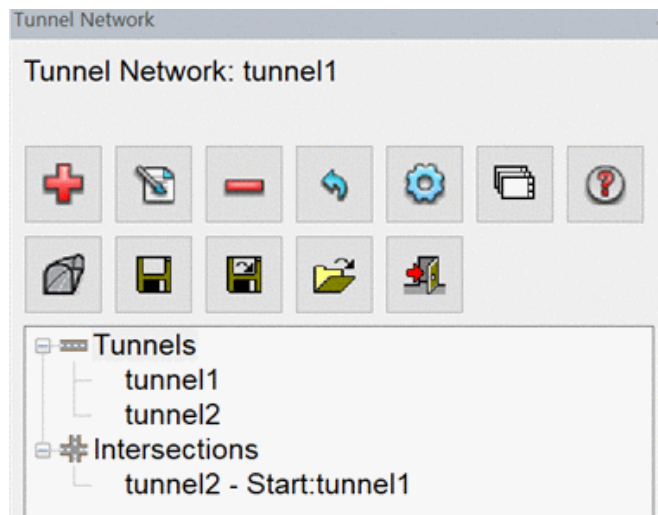
Pulldown Menu Location: Solid

Keyboard Command: dwg2mdl

Prerequisite: 3D faces

Tunnel Network

This command creates a solid model of one or more tunnels. Each tunnel is defined by a centerline, profile and cross section template. This design method is the same as the Make Solid From Design command. The additional ability of Tunnel Network is to process multiple tunnels together. This command automatically finds intersections for tunnel centerlines. At each intersection, the program has controls for how to connect the tunnels.



All the model data is stored in a .TNET file and the program starts by prompting for the TNET file to process. Then the Tunnel Network dialog docks on the side of the screen. The top row of functions are:

Add: To add another tunnel to the model.

Edit: To edit the highlighted tunnel or intersection.

Remove: To remove the highlighted tunnel from the model.

Reset: To reset the highlighted intersection to defaults.

Settings: Tunnel Network settings.

New: Create a new TNET model.

Help: Carlson Manual.

The second row of functions are:

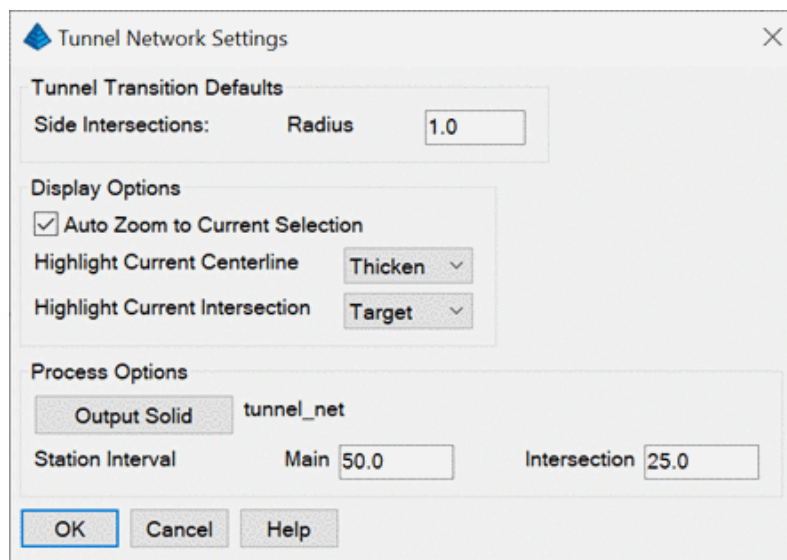
Process: Run the model and create the MDL solid.

Save: Save the current settings to the TNET file.

SaveAs: Save to another TNET file.

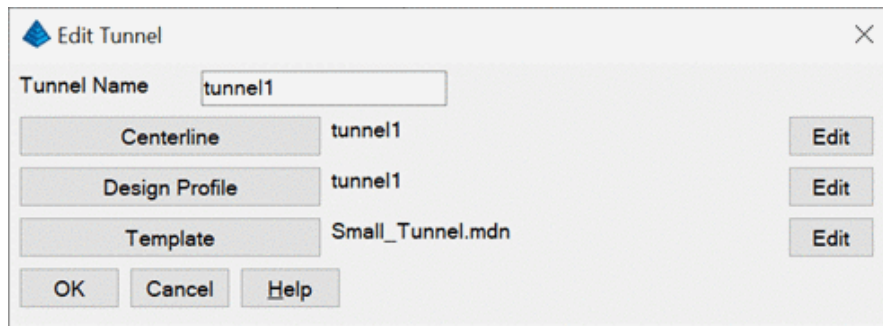
Load: Load another TNET file.

Exit: Close the Tunnel Network program.

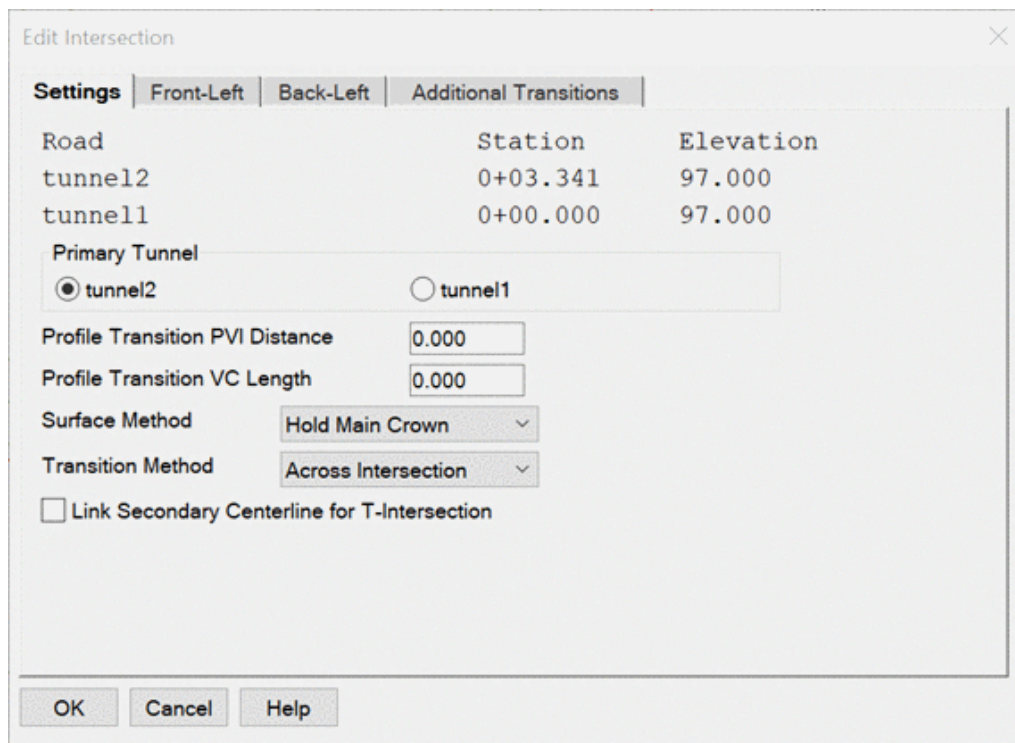


In Tunnel Network Settings, the **Radius** is the default to use for the sides of the tunnels at intersections. The **Display Options** control how to highlight tunnel items in the drawing when they are selected from the Tunnel Network project tree dialog. **Output Solid** sets the name of the MDL to create. The **Station Intervals** control when

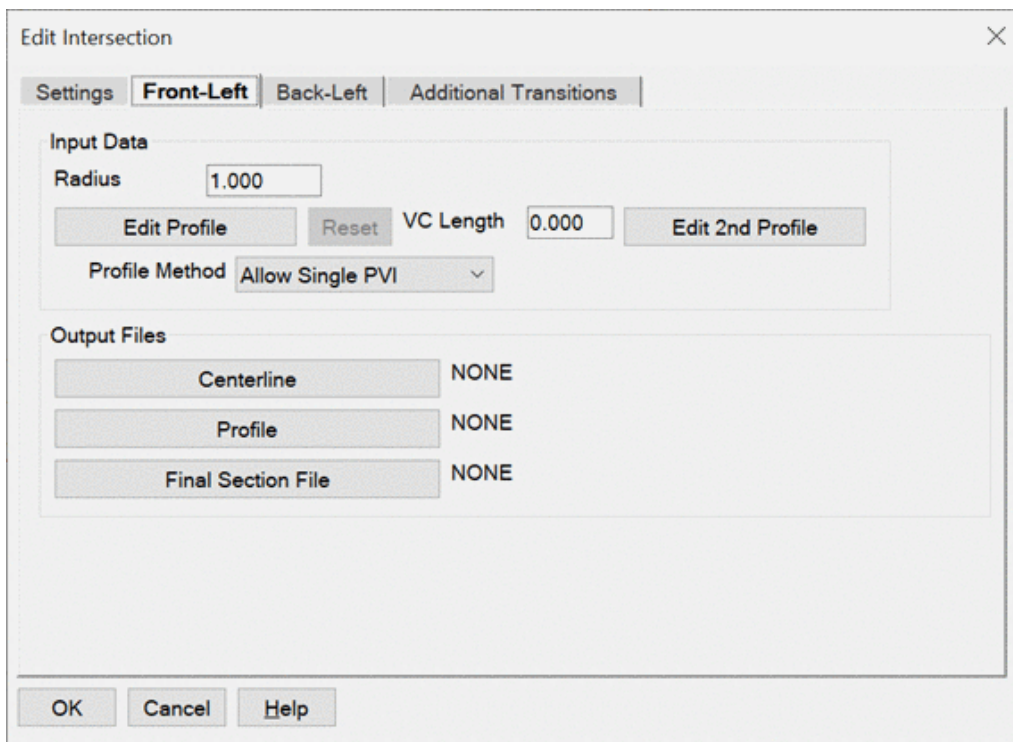
to create sections along the alignment in the model.



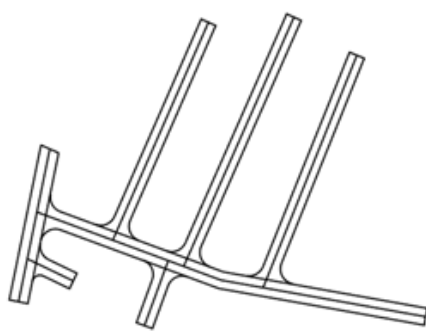
When you add or edit a tunnel, there's a dialog to set the **Tunnel Name**. Each tunnel needs a unique name. The **Centerline** defines the horizontal alignment. The **Profile** defines the vertical alignment. The **Template** defines the tunnel cross section. Besides editing these files within Tunnel Network, there are separate commands for these such as Centerline Editor and Profile Editor.



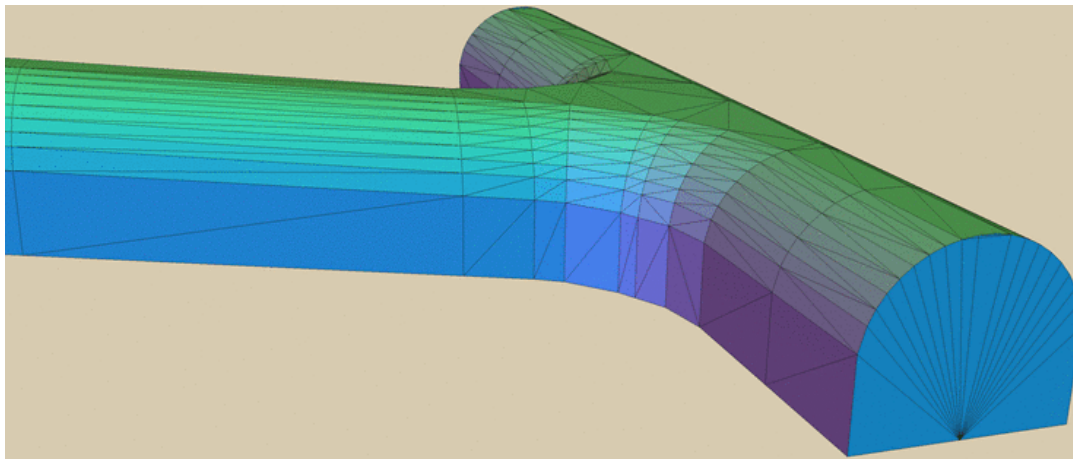
For each intersection, the program calculates the centerline stations and elevation at the intersection. The Primary Tunnel setting controls which profile is held at the intersection. The other profile will be set to match the primary tunnel. There are settings to control the profile transition of the secondary tunnel.



For each corner of the intersection, there are separate tabs for the settings specific to that corner. Each corner is named relative to the primary tunnel direction. For example, Front-Left is the corner along the primary tunnel before entering the intersection and on the left side. The Radius is the fillet radius to transition the outer tunnel side at the intersection. There are setting to control the profile in the corner transition.



When you add tunnels to the model, you can pick the centerline polyline from the drawing and assign a CL file name.



Pick the Process button to build the solid. Then you can use other solids functions such as View Solid.

Pulldown Menu Location: Solid

Keyboard Command: tunnelnet

Prerequisite: solid model file

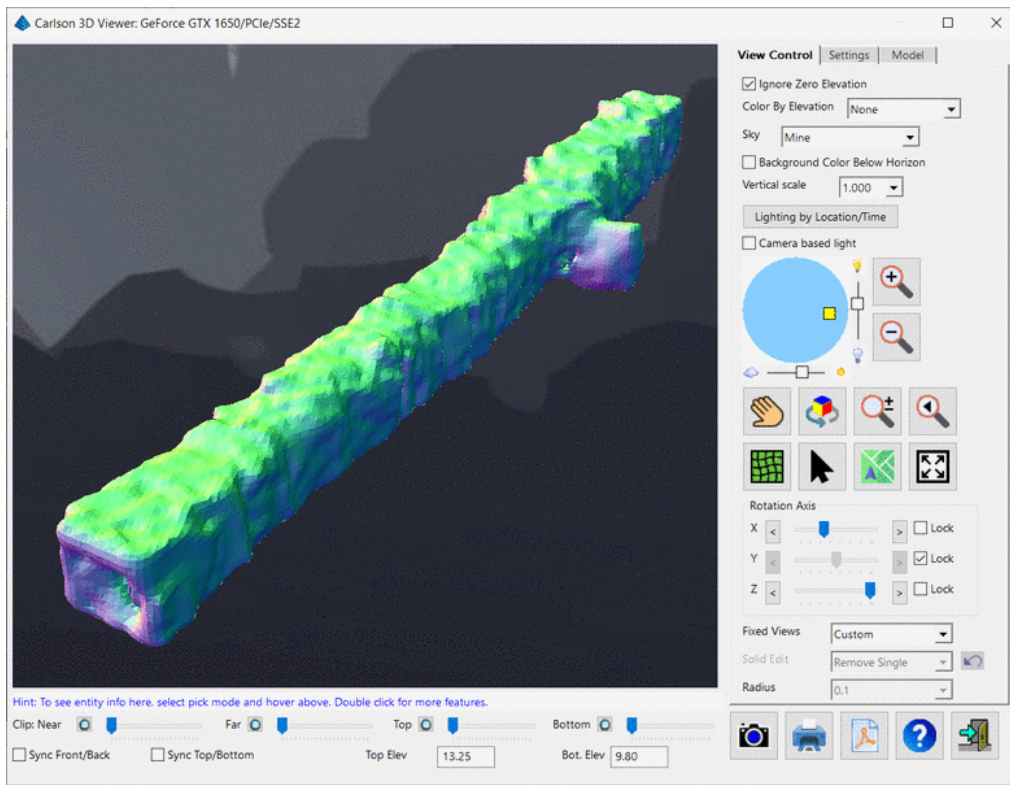
View Solid

This command shows a solid model in the 3D Viewer Window. See the 3D Viewer Window section of the manual for a description of the viewer functions.

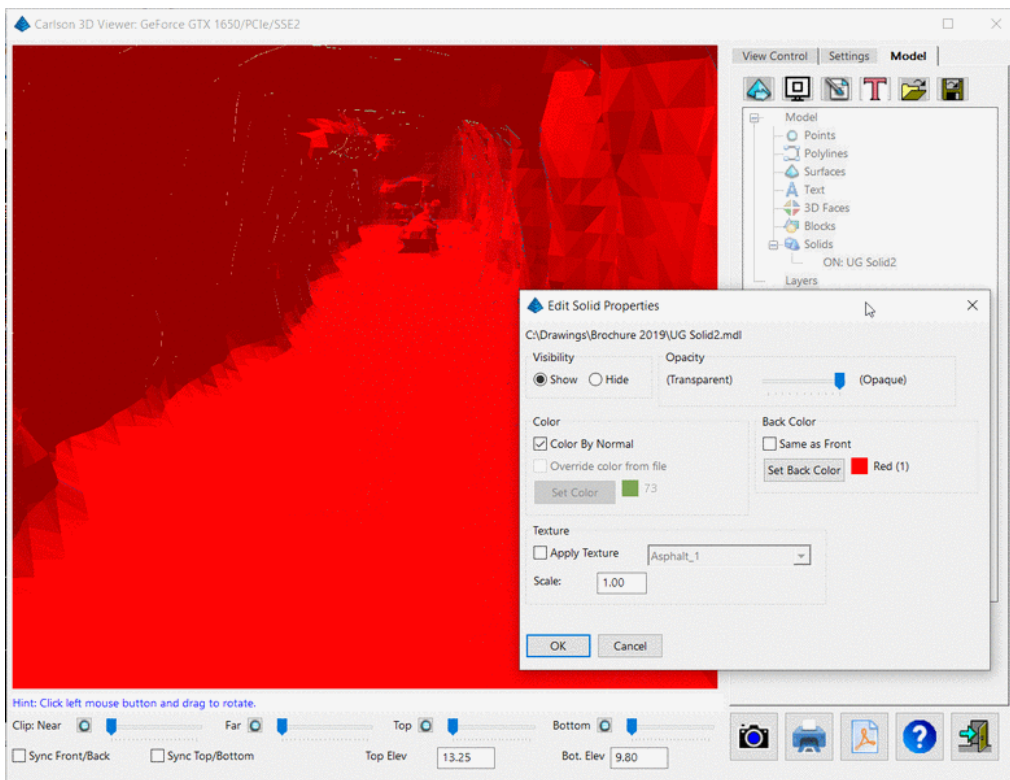
Solid Edit is an extra feature of View Solid that isn't part of the regular 3D Viewer command. To use the Solid



Edit, switch to the Pick Mode by picking this button: The Solid Edit has functions to remove faces from the solid which can be used to clean up a solid. Removing faces will leave holes in the solid. To make the solid watertight and fill in the holes, you can run the Heal Solid command after View Solid. There are two modes to the remove functions. The Remove Single removes only the face that is picked. The Remove Radius removes all faces from the pick point that are within the specified radius.



On the Edit Solid Properties found in the Model tab, there are options to color by Normal, Apply Texture, and to set the Back Color of the faces to a different color than the front. This allows for different color viewing when viewed from the inside of the solid, as seen here.



Pulldown Menu Location: Solid
Keyboard Command: view_solid
Prerequisite: solid model .mdl file

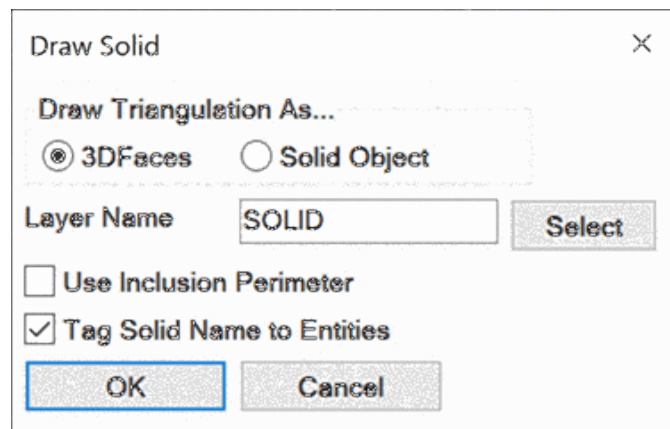
Draw Solid 3D Faces

This command draws a 3D Model from an MDL file as a 3DSolid entity. After selecting the MDL file, the program creates the 3DSolid in the current layer. In AutoCAD, a Solid Entity will be drawn, whereas a Polyface Mesh entity will be drawn in IntelliCAD.

Pulldown Menu Location: Solid
Keyboard Command: mdl2solid
Prerequisite: MDL file

Draw 3D Solid

This command draws a solid model as 3D Faces. The Tag Solid Name To Entities adds XData to the 3D faces with the name of the solid model file that is used by the ID Solid command. To view the solid entities in 3D, use commands like View > 3D Viewer Window.



The Surface Object is a custom object for displaying the solid in the drawing. This custom object also supports the EndPoint snap. The main advantage of the Surface Object is with displaying large solids because the Surface Object is more efficient than drawing individual 3D Faces. The Surface Object requires Carlson to be loaded to display.

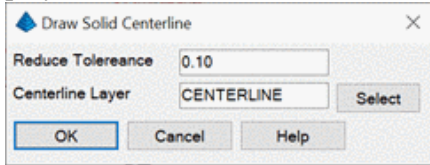
Prompts

Solid File To Read *Select MDL file*
Draw Solid options dialog

Pulldown Menu Location: Solid
Keyboard Command: draw_solid
Prerequisite: solid model file

Draw Solid Centerline

This command draws a 2D alignment polyline at an elevation of zero along the length of the solid. This command applies to solids of tunnels. After selecting the solid file (.MDL), the program prompts for the layer to use for the polyline and the Reduce Tolerance which is used for removing collinear vertices.



Prompts

Solid File To Read *Select MDL file*

Perimeter Layer dialog

Pulldown Menu Location: Solid > Draw Solid

Keyboard Command: solid_cl

Prerequisite: solid model file

Draw Solid Perimeter

This command draws the 2D outline of a solid as a closed polyline at an elevation of zero. After selecting the solid file (.MDL), the program prompts for the layer to use for the polyline.

Prompts

Solid File To Read *Select MDL file*

Perimeter Layer dialog

Pulldown Menu Location: Solid > Draw Solid

Keyboard Command: solid_perim

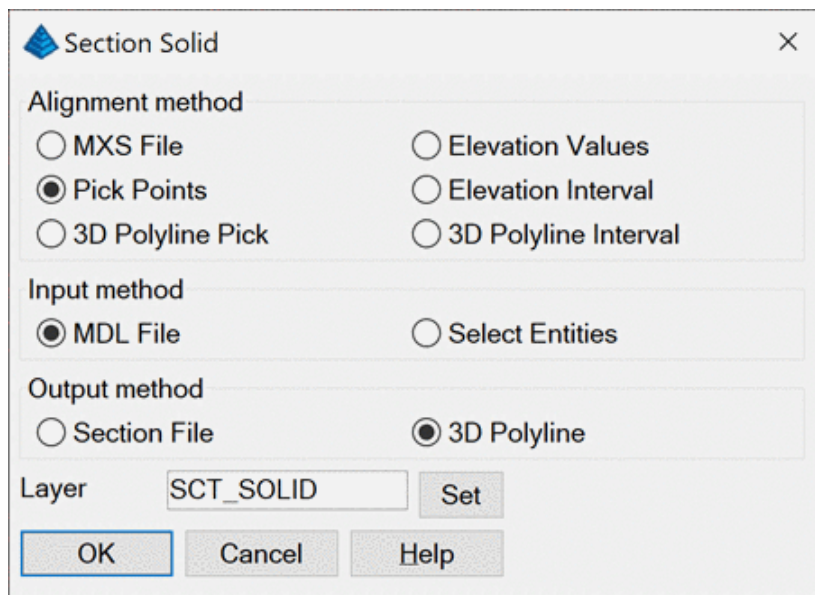
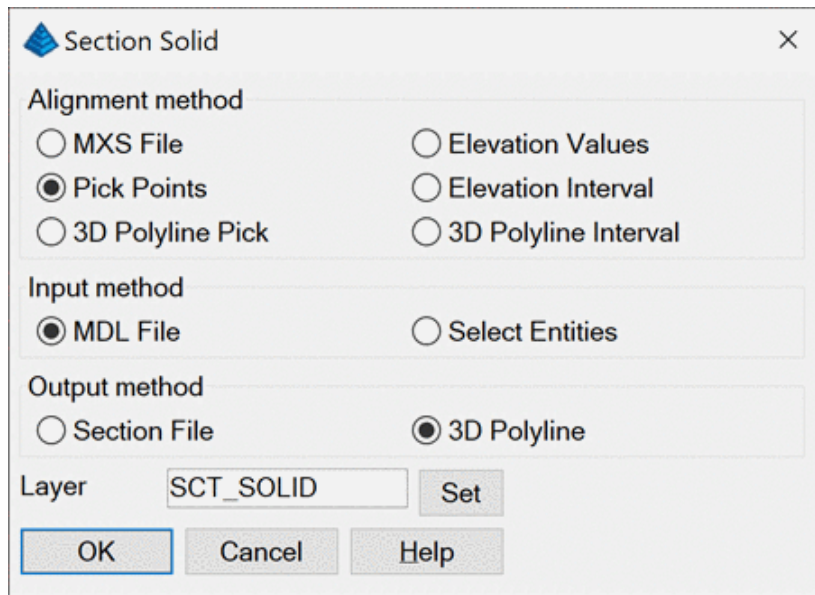
Prerequisite: solid model file

Section Solid

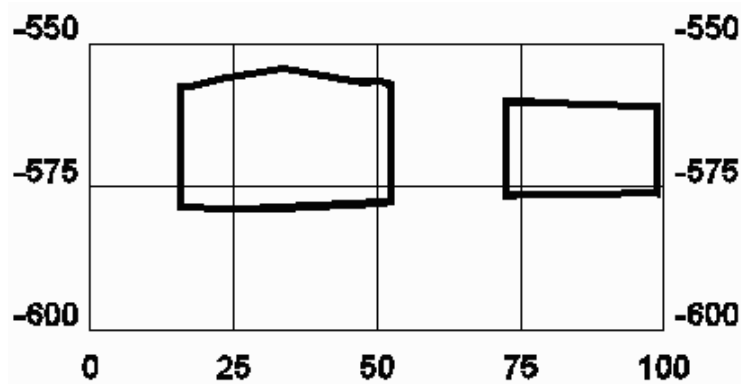
This command creates a cross section of a solid. The cross section is either drawn as a 3D polyline or stored to a SCT file and assigned to a station. Use commands like Draw Section File to view the cross section SCT file.

The section alignment can be defined by **MXS File** for creating sections for multiple stations. Use the **Pick Points** method for a single section. With this single section, the program prompts for two points to define the alignment for the cross section. The **Elevation Values** and **Elevation Interval** methods create horizontal sections of the solid which is like contouring the solid. The **3D Polyline** methods create sections that are perpendicular to the slope of the 3D polyline. The Pick method prompts for stations along the polyline for the sections. The Interval method creates sections at a specified station interval.

The solid model can be defined by a Carlson 3D Model file (MDL) or by 3D Solid entities in the drawing.



Plan view of cross section alignment



Prompts

Solid File To Read *Select MDL file*

Section File To Append/Write *Select SCT file*

Section station <0.0>: *press Enter*

Pick 1st alignment point: *pick a point*

Pick 2nd alignment point: *pick a point*

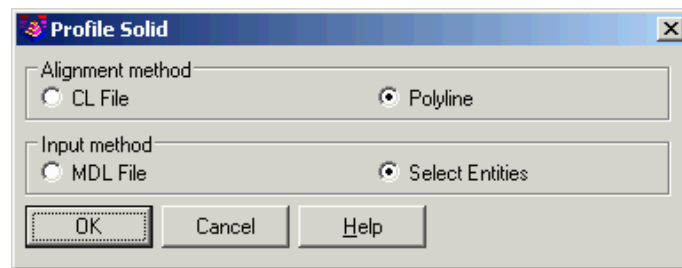
Pulldown Menu Location: Solid in Underground Mining and Section in Civil

Keyboard Command: section_solid

Prerequisite: solid model file or 3D Solid entities

Profile Solid

This command creates a profile of slices through 3D solids. The alignment for the profile can be defined either by centerline file or by a polyline. The solid model can be defined either by a Carlson 3D model file (MDL) or by 3D Solid entities in the drawing.



Prompts

Profile Solid dialog

Profile to Write *Specify a file name*

Select entities for model.

FILter/<Select entities>: *pick 3D solids*

Select polyline centerline: *pick a polyline*

Pulldown Menu Location: Profiles > Create Profile From

Keyboard Command: pro_solid

Prerequisite: CL file or polyline alignment, and 3D Solid or MDL file

Calculate Solid Volumes

This command calculates the volume of a solid model applied to the strata from a Geologic Model file. The volume for each strata from the Geologic Model file is calculated within the solid. Strata attributes and block models within the Geologic Model file are also calculated. The results are reported with the report formatter. Use Define Geologic Model to prepare the PRE file.

Prompts

Solid File To Read *Select MDL file*

Select Geologic Model File To Process *Select PRE file*

Report Formatter dialog

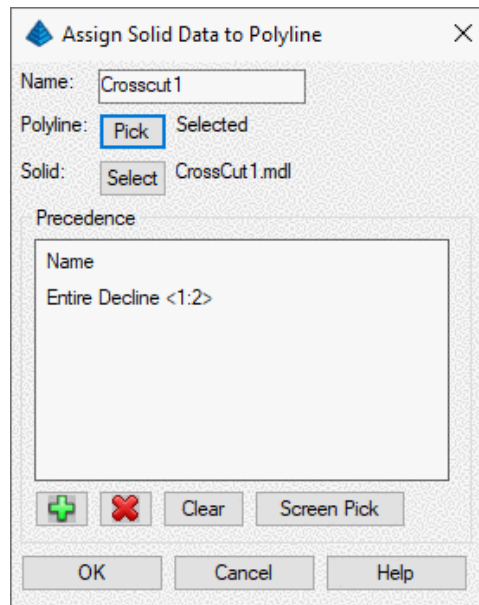
Pulldown Menu Location: Solid

Keyboard Command: calc_solid

Prerequisite: solid model and Geologic Model grid files

Assign Solid Data To Polyline

This command links 3D polylines to design solids (.mdl files) in preparation for the Solid Timing command. When the command is executed, the below dialog will be displayed.



Name: Here you can assign a name to the polyline/solid. This name will be displayed in the Solid Timing command. Note that the name does not need to match the filename for the solid (.mdl file).

Polyline: Clicking the **Pick** button will allow you to select a polyline. Note that only one polyline may be selected. Solids/polylines with pinpoints via the Assign Pinpoint to Solid command.

Solid: Clicking the **Select** button will allow you to select a .mdl file that represents the solid to be mined. Note that the polyline selected must be adjacent to or contained within the solid.

Precedence: Precedence rules may be applied to the solid to ensure timing is not calculated in an illogical or impossible progression. Any solids listed in the Precedence list will be mined prior to the current solid.

Other solids may be added to the list by clicking the green plus icon.

Solids may be removed from the list by clicking the red X icon.

All solids may be removed from the list by clicking the Clear button.

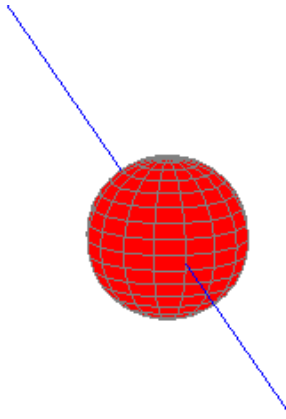
Solids may also be added to the list by clicking the Screen Pick button. With this method, you will be able to select polylines from the drawing area rather than picking the names from a list.

Pulldown Menu Location: Solid

Keyboard Command: ASSIGNPLYSOLID

Add Pinpoint To Solid

This command will add a pinpoint to solid's polyline. Pinpoints are used to subdivide solids for assignment in the Solid Timing command. Pinpoints will appear on the 3D polyline as a sphere, as shown below.



Command: addpinsolid

Select solid polyline: *Here the user will need to select the 3D polyline to add the pinpoint.*

Pick pinpoint or Enter distance or Define size[<Pick>/Enter/Size]:? S *Using the Size option will allow you to change the size of the pinpoint symbol.*

Pinpoint sphere diameter size <10.00>: 15

Pick pinpoint or Enter distance or Define size[<Pick>/Enter/Size]:? P *Using the Pick option will allow you manually select the location of the pinpoint. Using the Enter option will allow you to set the location of the pinpoint by entering the distance from the start of the polyline.*

Pick point on selected polyline: *The Nearest Snap will be activated and you will be able to pick a location on the polyline. The command will end after picking a location.*

Pulldown Location: Solid

Keyboard Command: ADDPINSOLID

Prerequisite: A Polyline with a solid assigned (see Assign Solid to Polyline command)

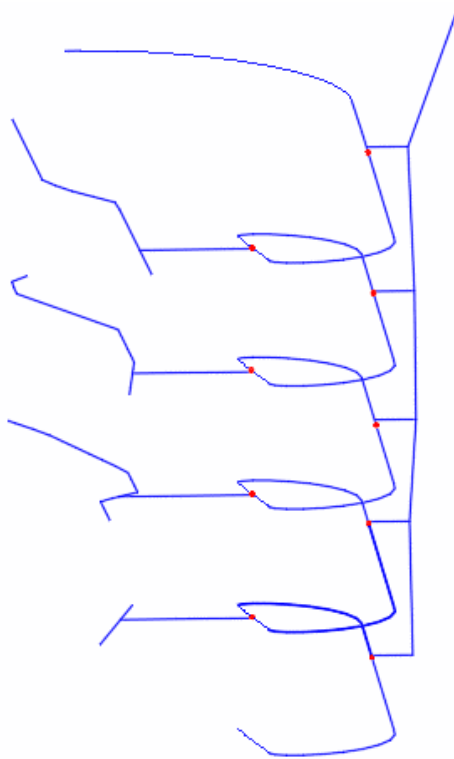
Solid Timing

The Solid Timing command schedules through designed solids (.mdl files) to generate a 3D timing model and a detailed report. Each solid is mined by a single piece of equipment according to a working calendar. As the schedule is calculated, the solids will be compared against a Geologic Model to determine the quality of the material being extracted. The below documentation follows this outline:

1. Required Setup
2. Mine Solid Timing Dialog
3. Reporting Options

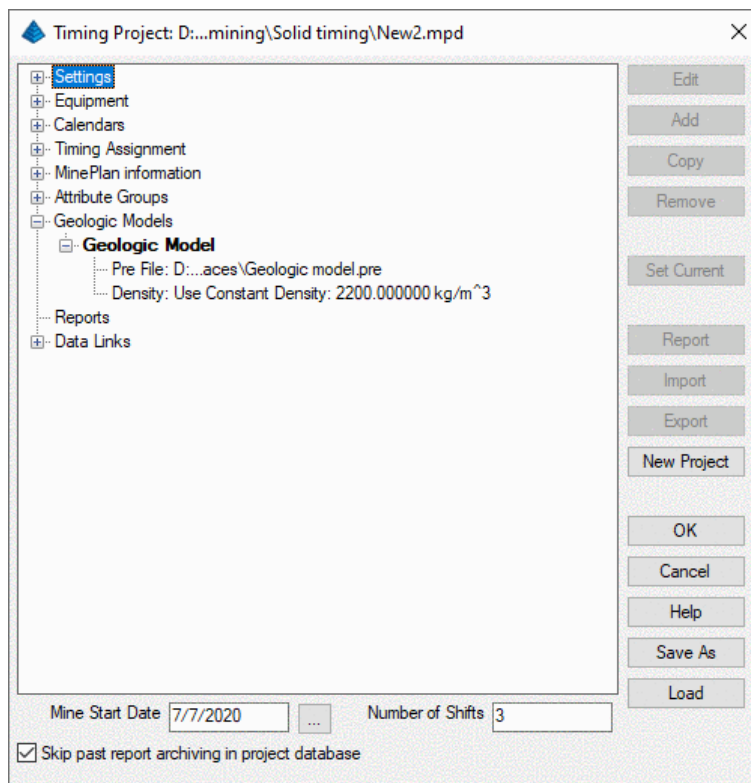
Before the Solid Timing may be executed, proper setup is required. At a bare minimum, this includes drafting polylines that represent the direction/location of the solids, assigning solids to the polylines, and populating the Timing Project Manager.

Polylines may be drafted using any tools you like. The polylines should be contained within or exist along the edge of the solid to be mined. For example, the polyline may represent the geometric centroid of the solid, or they may represent a corner point on the the template of the solid. An example set of polylines is shown below.

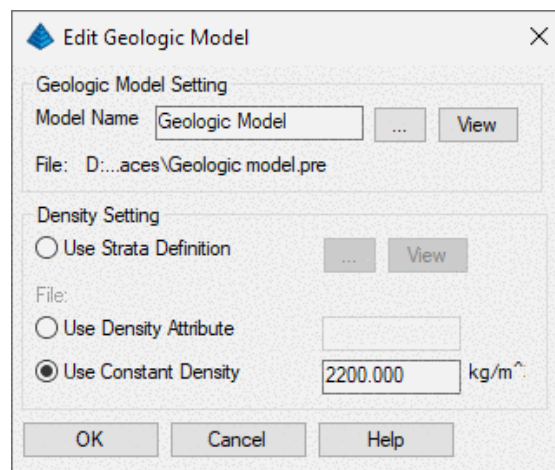


Once the polylines have been drafted, you can assign design solids (.mdl files) to them with the Assign Solid to Polyline command. Note that design solids may be created with a variety of commands found on the Solid Pulldown Menu.

Once the polylines and solids have been defined, you may execute the command. You will first be prompted to define a Timing Project Manager file (.mpd file). This file is described at length here, but has been slightly modified for the application of Solid Timing. Unlike the Surface Equipment Timing and Underground Timing command, the Solid Timing command calculates volumes/qualities directly from a Geologic Model (.pre file) as the timing is executed. Attribute groups are not required, but are still available. An example of the Timing Project manager is shown below.



Note that the Geologic Model branch of the Timing Project Manager is unique to the Solid Timing command. This branch is discussed below, but all other branches of this file are discussed here. It is important to note that in order to calculate the timing, the Equipment, Calendar, and Start Date must be defined. A Geologic Model may be added by double-clicking the Geologic Model branch or by selecting the Branch and clicking the Add Set button. When defining a Geologic Model, the below dialog will appear.



Model Name: The Geologic Model may be assigned any name. You may select a .pre file by clicking the ellipsis button. Once selected, you may view/modify the .pre file by clicking the View button.

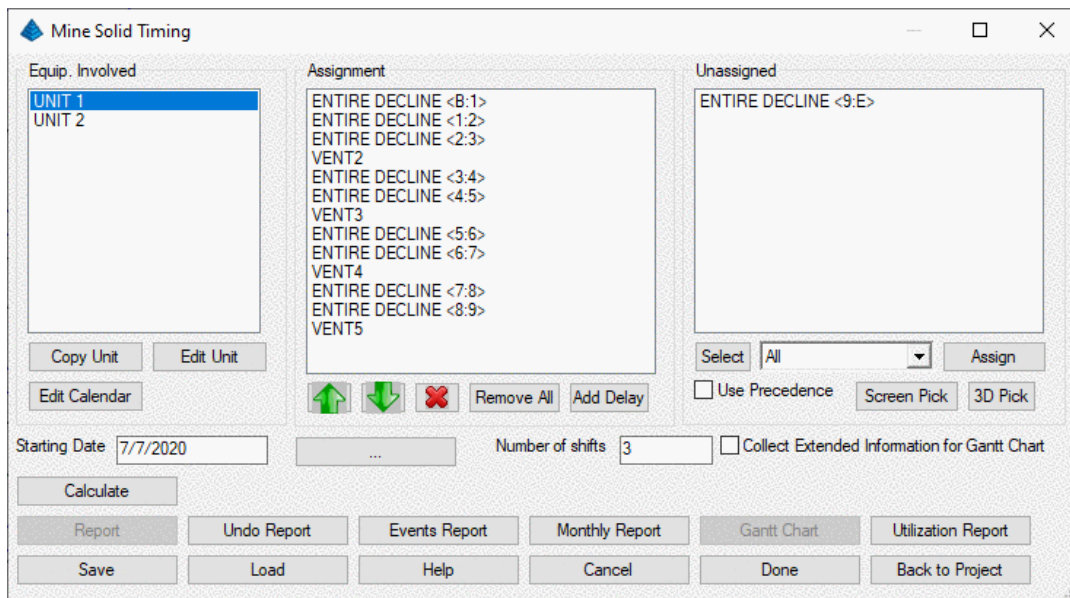
Density Setting: There are three options to specify the density of the materials defined in the Geologic Model.

The **Use Strata Definition** option will read density values from the Strata Definition file (.sdf file). Any strata types

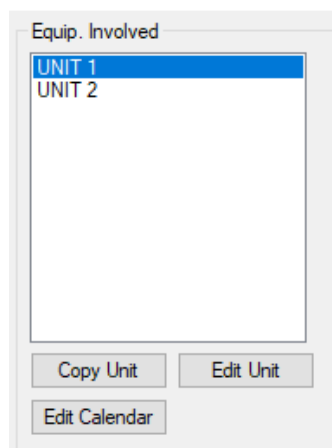
found in the Geologic Model that are not also defined in the Strata Definition will be assigned a default density value. The Strata Definition file may be selected by clicking the ellipsis button and viewed/modified by clicking the View button.

The **Use Density Attribute** will use an attribute found within the Geologic Model to define the density for each strata type. A benefit to this option is that each strata's density may vary across the property. Note that when using this option, you will need to enter the name of the Density attribute exactly as it appears in the Geologic Model. If no density attribute is found when this option is selected, the strata layers will be assigned a default density value.

The **Use Constant Density** option will assign a single density value to all strata types found in the Geologic Model. Once the Timing Project Manager has been defined, you will be prompted to select all of the polylines/solids to use in the timing command. After these entities are selected, the below dialog will appear. This dialog controls the sequencing of the timing. Each section of this dialog is described below.

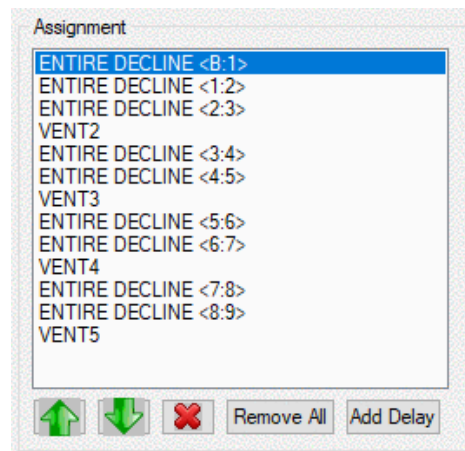


The Equip. Involved column will be automatically populated with equipment from the Timing Project Manager. You may create copies of equipment or edit the equipment definitions with the **Copy Unit** and **Edit Unit** buttons. The **Edit Calendar** button will allow you to modify the current calendar.

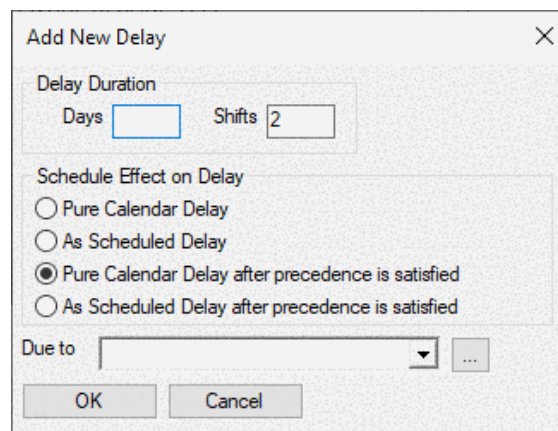


The Assignment Column lists the solids assigned to the currently selected equipment. Selecting a different

unit will update this column with the list of solids assigned to that unit. This column lists solids in the order that they will be mined, with solids at the top of the list being mined first. Solids may be moved up/down in the list by selecting them and clicking the respective **green arrow** icon. Solids may be removed from the list by selecting the solid and clicking the **red X** icon. All solids may be removed from the list by clicking the **Remove All** button.



Delays may be added to the assignment column by clicking the **Add Delay** button. Delays will be added to the assignment column as if they were another solid to be mined. When adding a delay, the below dialog will appear.



Delay Duration: These values control how long the delay lasts. In the above example, the equipment will be delayed for a full shift. Note that partial shift delays are allowed.

Pure Calendar Delay: This option will not force a delay. If the equipment already has scheduled downtime when the delay is encountered, additional time will not be taken off.

As Scheduled Delay: This option will force a delay even if the equipment already has scheduled downtime when the delay is encountered. The delay will wait until the scheduled downtime is complete, then the delay will be applied.

Pure Calendar Delay after precedence is satisfied: This is similar to a Pure Calendar Delay, but will not attempt to apply the delay until precedence rules have been satisfied. Consider the scenario shown in the below

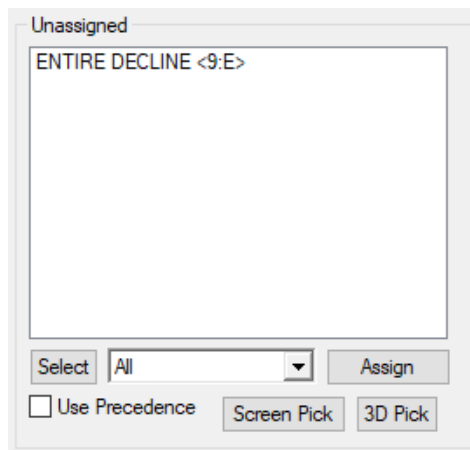
table. Equipment E1 is scheduled to mine Pit A and Pit C. Equipment E2 is scheduled to mine Pit B and Pit D. A precedence rule has been created that does not allow Pit D to be mined until Pit A is complete. A delay of this type is placed on Equipment E2 after Pit B. In this scenario, the finish date of Pit A will first be calculated, then the delay will be applied. If the equipment already has scheduled downtime when the delay is encountered, additional time will not be taken off.

As Scheduled Delay after precedence is satisfied: This option is similar to an As Scheduled Delay, but will not attempt to apply the delay until precedence rules have been satisfied. Consider the below scenario. Equipment E1 is scheduled to mine Pit A and Pit C. Equipment E2 is scheduled to mine Pit B and Pit D. A precedence rule has been created that does not allow Pit D to be mined until Pit A is complete. A delay of this type is placed on Equipment E2 after Pit B. In this scenario, the finish date of Pit A will first be calculated, then the delay will be applied. If the equipment already has scheduled downtime, the delay will be applied in addition to this downtime.

Due To: This dropdown menu lists the Drawing Event delays defined in the Timing Project Manager. The ellipse button will allow you to define a new Drawing Event.

Equipment	E1	E2
Assignment Sequence	Pit A	Pit B
	Pit C	Delay
		Pit D (Precedence rule: Pit A)

The Unassigned Column lists all solids that have not yet been assigned to a piece of equipment.



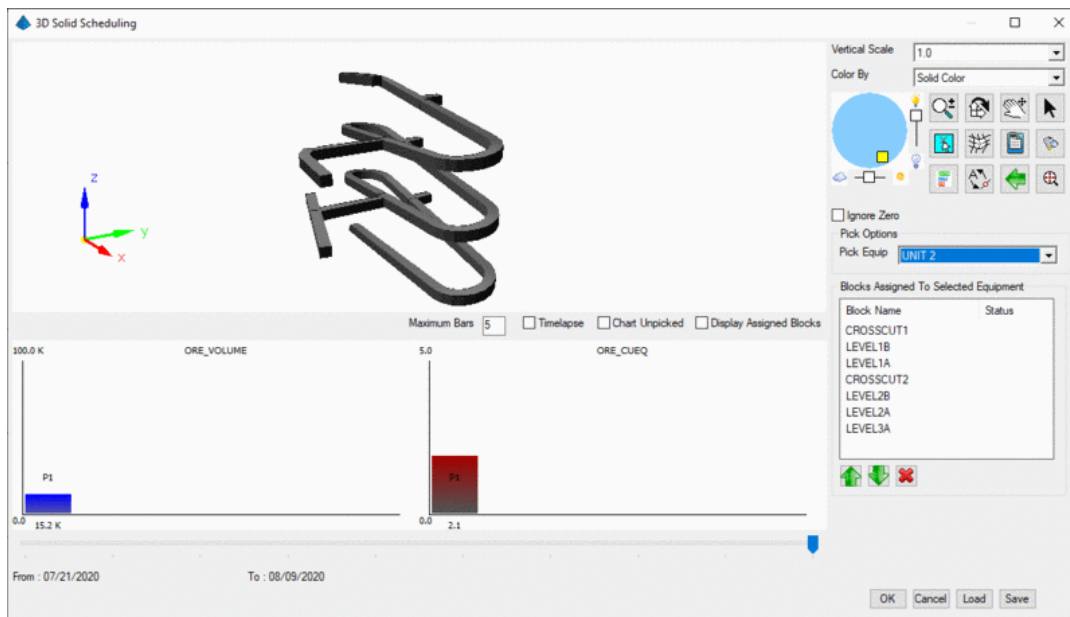
Assign: This button will assign the selected solid(s) to the selected piece of equipment. Solids may also be selected by double-clicking the name in the list.

Select: This button will select the solids according to the drop-down menu to the right of this button. If the drop-down menu is set to NONE, all solids will be unselected. If the drop-down menu is set to ALL, all solids will be selected.

Use Precedence: This checkbox will ensure that precedence is satisfied in the list. For example if Pit1-B1 is assigned to precede Pit1-B2, toggling this option on will reorder the list so that Pit1-B1 is listed before Pit1-B2.

Screen Pick: This button will allow you to sequence the solids by selecting them in the drawing. When activated, the Surface Equipment Timing Dialog will disappear and the drawing area will be visible. The command line will prompt you to click the polyline solid to assign to the currently selected piece of equipment. This option allows you to visually select the solids rather than picking them from a list.

3D Pick: This button will open a new window, as shown below, to show a 3D view of the available solids and the volumes/qualities associated with them. This interactive window allows you to sequence solids in an interactive 3D environment.



Vertical Scale: This value controls the vertical exaggeration of the graphics window.

Color By: This drop-down list controls how benches are colored in the graphics window.

The *Solid Color* option will color each solid with a single color.

The *Normal* option will color each face of the solid according the direction it faces. This adds a more vibrant color variation to the solids.

Control



Action

This control represents the position of the sun in plan view. If the yellow square is in the center of the blue circle, the sun is in a zenith. If the yellow square is near the edge of the circle, the sun is near the horizon. To move the yellow square, simply drag it to a new location, or click on the new location. The slider bars on the sides control the intensity and brightness of the display.

Switches to Dynamic Zoom mode. Click and drag to zoom in and out.



Switches to Rotation mode. When the cursor is placed near the outer edge of the view, a "Z" cursor is presented that permits rotation around the Z-axis. When the cursor is placed further into the interior of the view, an "X,Y" cursor is presented that permits the tilt angle of the view to be adjusted.

Switches to Pan mode. Click and drag to pan.

Switches to Pick mode. Double click a solid to assign it to the currently selected piece of equipment.

Prompts you to select linework such as contours or 3D faces to add to the 3D viewer window. This linework can be used to identify reference points in the drawing to assist in selection.

Toggle shading of Solids and 3D Faces.

Exits the 3D Pick window, calculates the amount of time required to mine the assigned solids, and generates a report. This function is discussed later in this section of the help manual.

Displays a detailed report of the current timing assignment. This is similar to the standard Report function, but provides more detailed information and does not attempt to create a timing map. An example report is shown below.

Task	Date	Eq. name	Start date	Finish date	Days	Pls.
1	PRESTRIP FL	0 AREA 1-9/27/2016	10/2/2016			
2	DRAGLINE	5 0 AREA 1-9/27/2016	10/7/2016			
3	LOADER	12 0 AREA 1-9/27/2016	10/9/2016			
4	PRESTRIP FL	0 AREA 1-9/27/2016	10/7/2016		5.0	
5	DRAGLINE	0 AREA 1-9/27/2016	10/12/2016		-6	
6	LOADER	0 AREA 1-9/27/2016	10/18/2016		-3	
7	PRESTRIP FL	0 AREA 1-9/27/2016	10/9/2016		10.2	
8	DRAGLINE	0 AREA 1-9/27/2016	10/17/2016		3.2	
9	LOADER	0 AREA 1-9/27/2016	10/29/2016		-8	
10	PRESTRIP FL	0 AREA 1-9/27/2016	10/12/2016		12.4	
11	DRAGLINE	0 AREA 1-9/27/2016	10/9/2016		8.2	
12	LOADER	0 AREA 1-9/27/2016	10/29/2016		-7	
13	PRESTRIP FL	0 AREA 1-9/27/2016	10/9/2016		18.8	
14	DRAGLINE	0 AREA 1-9/27/2016	10/29/2016		4.0	
15	LOADER	0 AREA 1-9/27/2016	11/9/2016		1.0	
16	PRESTRIP FL	0 AREA 1-9/27/2016	10/8/2016		22.0	

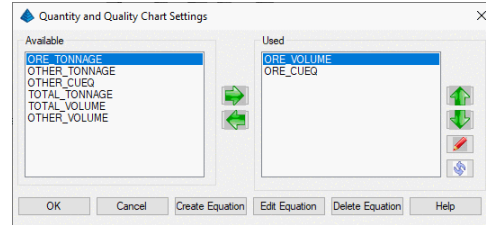
The calculator icon at the top left of the screen will recalculate the timing results of the currently assigned solids. This allows additional solids to be added/removed from the assigned equipment and then quickly recalculate the impact on the scheduling.

The slider bar at the top of dialog represents the timeline of the mining progression. Clicking-and-dragging the slider will update the displayed information.

The spreadsheet report will display solids to be mined in the currently selected time period.



Controls the information displayed on the bar charts. When clicked the below dialog will appear.

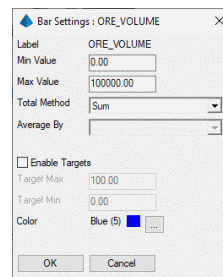


This dialog contains two columns: Available and Used. Only items in the Used column will be shown in the bar chart area below the 3D Pick graphics window. Items may be moved between the two columns by selecting the item and clicking one of the green arrows between the two columns. To the right of the Used column are four icons. The green arrows can be used to move the selected item up and down in the list. The red pencil can be used to edit the selected item as described below. The blue reset icon will clear out all used attributes and return them to the available column.

The attribute listed in the Available column will be automatically generated based off the Geologic Model. The volume, tonnage, and qualities for each strata type will be available.

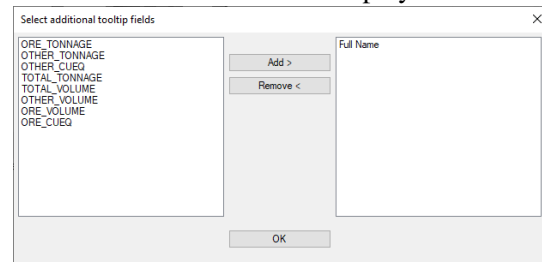
You can use the Create/Edit/Delete Equation buttons to manage equations. These equations can be used to represent values that are not actually in the Geologic Model, but are rather a combination of other values. More information on defining equations is provided here.

When an item is first moved to the Used column, the below dialog will appear to control that item's appearance on the chart, including minimum value to display, maximum value to display, and color. You can also define the method used for calculating the total of the value. For example, values such as volume can be totaled as a Sum, whereas values such as quality can be a Weighted Average that is weighted by the Volume of each solid. You can also enable the Targets, which are simply horizontal lines displayed on the bar charts to represent threshold values.





Controls the attributes that are displayed in the tooltip when the cursor is hovering over a bench. Attributes in the left column will not be displayed.



Undoes the last selection in the 3D pick window.



Zooms to the extents of the items shown in the 3D window.

Ignore zero elev: This option will toggle the visibility of surface features at zero elevation in the graphics window.

Pick Equipment: This drop-down list includes all equipment that have been added to the main Surface Equipment Timing dialog. Solids will be assigned to the selected equipment.

Blocks Assigned to Selected Equipment: This list shows all solids that have been assigned to the selected equipment. The solids are listed in the order they will be mined. The green arrows below the list will move the currently selected solid up and down in the list. The red X will remove the selected solid from the list and make it available for assignment to another piece of equipment.

Maximum Bars: This value controls the maximum number of periods (and thus, the number of vertical bars) that may be shown in each chart when the Timelapse option is enabled.

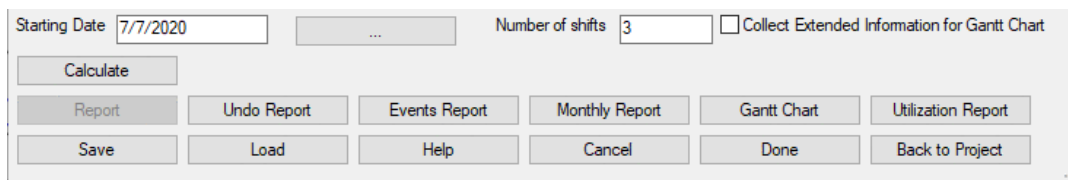
Timelapse: This checkbox will create a new period each time a solid is selected, thus adding another bar to the charts. Once the maximum number of periods is displayed, older periods will be removed from the charts.

Chart Unpicked: This option will reset the charts and only show information pertaining to solids that have not yet been assigned to a piece of equipment.

Display Assigned Blocks: This option will display solids that have already been assigned to a piece of equipment.

Period Slider Bar: The slider bar below the charts is only available when the Timelapse option is disabled. This slider bar allows you to control how many solids are represented in the last vertical bar of the charts. For example, if the slider bar is moved to the right end of the bar, all assigned solids will be represented in the bar chart. If the slider bar is moved closer further to the left, fewer solids will be represented in the bar chart.

These options at the bottom of the Mine Solid Timing dialog control the various reporting options.

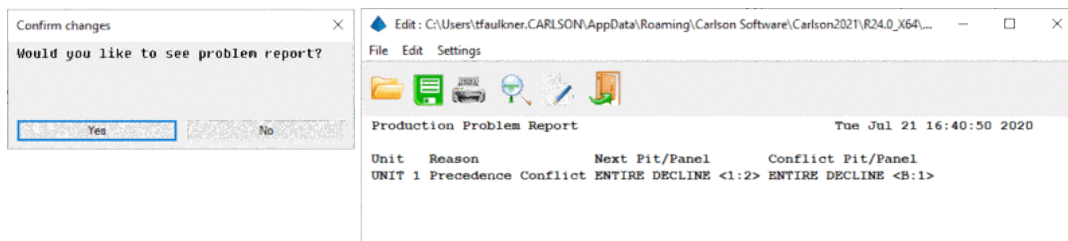


Starting Date: This value sets the date to begin mining operations. The ellipsis button to the right of this field will open a small calendar to assist in date selection. By default, this value will be set the Mine Start Date set in the Timing Project Manager.

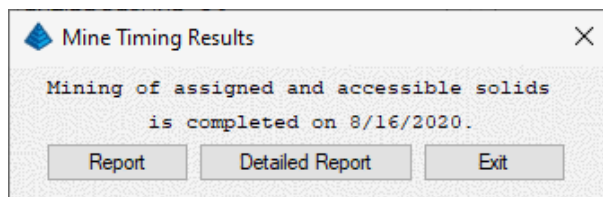
Number of Shifts: This value will set the number of working shifts in a day.

Collect Extended Information for Gantt Chart: When enabled, this option will allow the Gantt Chart to include extra information such as the pit volume, quality, etc.

Calculate: This button will calculate the mine schedule. If any errors are encountered, the below dialog (left) will appear. The problem report generated will appear similar to below (right). In this example, a precedence rule has been violated for the Unit 1. Entire Decline <1:2> cannot be mined until Entire Decline <B:1> has been mined. This will require an edit to the assignment order.



If no problems are encountered, the schedule will be calculated and the below dialog will appear.



Report: This button will generate a very simple report as shown below. Note that this report does not include information about the amount of material mined.

Unit	Days waiting	Solid name	Start date	Finish date
UNIT 1	0.0	ENTIRE DECLINE	7/7/2020	7/11/2020
UNIT 1	0.0	ENTIRE DECLINE	7/11/2020	7/16/2020
UNIT 1	0.0	ENTIRE DECLINE	7/16/2020	7/20/2020
UNIT 1	0.0	ENTIRE DECLINE	7/20/2020	7/25/2020
UNIT 1	0.0	ENTIRE DECLINE	7/25/2020	7/29/2020
UNIT 1	0.0	ENTIRE DECLINE	7/29/2020	8/3/2020
UNIT 1	0.0	ENTIRE DECLINE	8/3/2020	8/6/2020
UNIT 1	0.0	ENTIRE DECLINE	8/6/2020	8/12/2020
UNIT 1	0.0	ENTIRE DECLINE	8/12/2020	8/15/2020
UNIT 1	0.0	VENT2	7/20/2020	7/20/2020
UNIT 1	0.0	VENT3	7/29/2020	7/29/2020
UNIT 1	0.0	VENT4	8/6/2020	8/6/2020
UNIT 1	0.0	VENT5	8/15/2020	8/16/2020
UNIT 2	0.0	LEVEL1A	7/17/2020	7/23/2020

Detailed Report: This button will generate a more detailed report, including information such as the amount of material mined, quality of material, downtime, etc. This report is discussed in more detail later in this help document.

Report: This button (directly below the Calculate button) is only available after calculating the mine schedule. This button will open the Detailed Report dialog. This is the same dialog that will appear after selecting the Detailed Report option after clicking the Calculate button. The Detailed Report is described later in this help document.

Undo Report: This button will delete a previous timing map and clear the calculated schedule. This allows for quick what-if analysis of the scheduling.

Events Report: This button will display the Events Report, which lists all Drawing Events encountered during the mine schedule. Drawing Events are discussed in more detail in the Timing Project Manager. This report is only available after a Detailed Report has been generated.

Monthly Report: This button will generate a monthly report, similar to the Detailed Report, but with less information available and without the ability to set custom time periods for reporting. An example of the Monthly Report is shown below.

Unit	Month	Year	Operating days	Days Weekdays	Worked Saturdays	Worked Sundays	Calendar days	Work shifts	Hour
UNIT 1	July	2020	31.00	31.00	23.00	4.00	31.00	93.00	
UNIT 1	August	2020	31.00	31.00	21.00	5.00	31.00	93.00	
UNIT 2	July	2020	31.00	31.00	23.00	4.00	31.00	93.00	
UNIT 2	August	2020	31.00	31.00	21.00	5.00	31.00	93.00	
Total varies			124.00	124.00	88.00	18.00	124.00	372.00	

Gantt Chart: This button will generate a Gantt chart of the mine schedule. This button is only available after calculating the mine schedule. For more information, see the Gantt Chart section of the help manual.

Utilization Report: This button will generate a Utilization Report of the mine schedule. This report lists each shift for each piece of equipment. Scheduled delays will be reported as such. An example of the Utilization Report is shown below.

Event Date	Shift	Year (start)	Month/Year	Equipment	Shift Hours	Pit	Type	Util. Ratio
7/7/2020	1	2020	07/2020	UNIT 1	8	ENTIRE DECLINE <B:1>	UTILIZATION	1.00
7/7/2020	2	2020	07/2020	UNIT 1	8	ENTIRE DECLINE <B:1>	UTILIZATION	1.00
7/7/2020	3	2020	07/2020	UNIT 1	8	ENTIRE DECLINE <B:1>	UTILIZATION	1.00
7/7/2020	1	2020	07/2020	UNIT 2	8		AVAILABLE	
7/7/2020	2	2020	07/2020	UNIT 2	8		AVAILABLE	
7/7/2020	3	2020	07/2020	UNIT 2	8		AVAILABLE	
7/8/2020	1	2020	07/2020	UNIT 1	8	ENTIRE DECLINE <B:1>	UTILIZATION	1.00
7/8/2020	2	2020	07/2020	UNIT 1	8	ENTIRE DECLINE <B:1>	UTILIZATION	1.00
7/8/2020	3	2020	07/2020	UNIT 1	8	ENTIRE DECLINE <B:1>	UTILIZATION	1.00

Back to Project: This button will return to the Timing Project Manager window.

Detailed Reporting: When a Detailed Report is generated, the below dialog will appear.

Report by: This drop list will determine how the timing map is generated.

If the *Dates* option is selected, the timing map will be colored by date ranges.

If the *Amounts* option is selected, the timing map will be colored by amount intervals.

If the *Equipment* option is selected, the timing map will be colored by equipment.

Dates Table: This button will open the Dates Table, shown below, which is used to define the various time periods used for reporting.

	End Date	Range	Color	Pattern	Scale	Layer	Label
1	2/1/2028	< 2/1/2028	Red	SOLID	Set	250.0 MINETM	Set
2	3/1/2028	2/1/2028 - 3/1/2028	Dark Red	SOLID	Set	250.0 MINETM	Set
3	4/1/2028	3/1/2028 - 4/1/2028	Red-Orange	SOLID	Set	250.0 MINETM	Set
4	5/1/2028	4/1/2028 - 5/1/2028	Orange	SOLID	Set	250.0 MINETM	Set
5	6/1/2028	5/1/2028 - 6/1/2028	Light Orange	SOLID	Set	250.0 MINETM	Set
6	7/1/2028	6/1/2028 - 7/1/2028	Yellow-Orange	SOLID	Set	250.0 MINETM	Set
7	8/1/2028	7/1/2028 - 8/1/2028	Yellow	SOLID	Set	250.0 MINETM	Set
8	9/1/2028	8/1/2028 - 9/1/2028	Light Yellow	SOLID	Set	250.0 MINETM	Set
9	10/1/2028	9/1/2028 - 10/1/2028	Light Green	SOLID	Set	250.0 MINETM	Set
10	11/1/2028	10/1/2028 - 11/1/2028	Green	SOLID	Set	250.0 MINETM	Set
11	12/1/2028	11/1/2028 - 12/1/2028	Dark Green	SOLID	Set	250.0 MINETM	Set
12	1/1/2029	12/1/2028 - 1/1/2029	Light Green	SOLID	Set	250.0 MINETM	Set
13	2/1/2029	1/1/2029 - 2/1/2029	Green	SOLID	Set	250.0 MINETM	Set
14	3/1/2029	2/1/2029 - 3/1/2029	Dark Green	SOLID	Set	250.0 MINETM	Set
15	4/1/2029	3/1/2029 - 4/1/2029	Light Green	SOLID	Set	250.0 MINETM	Set
16	5/1/2029	4/1/2029 - 5/1/2029	Yellow-Green	SOLID	Set	250.0 MINETM	Set

The above dialog can be used to create custom time periods to show in the timing map. The icons at the top left of the dialog will add and delete rows of information.

End Date: This column displays the end date for the row of information.

Range: This column sets the date range of the period. Dates may be preceded by > or < to mark indefinite time periods.

Color: This column sets the color of the time period. Double clicking one of these cells will open the CAD color palette.

Pattern: This column sets the hatch pattern for the timing blocks.

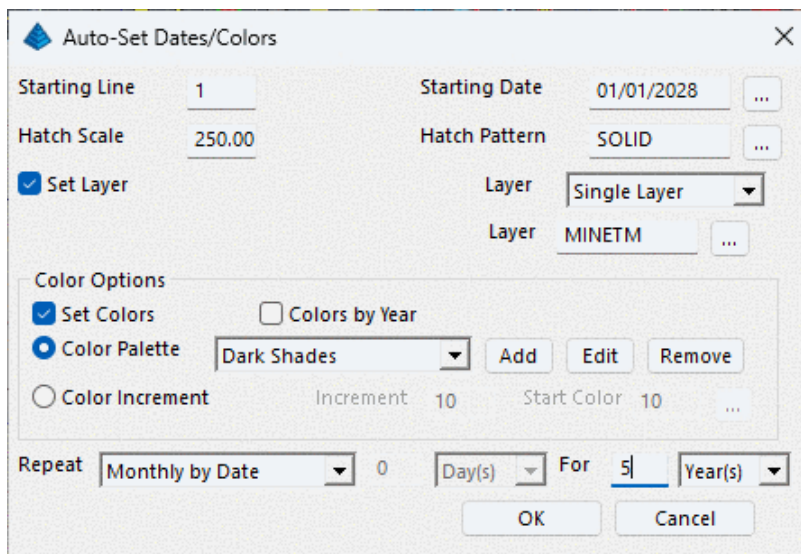
Scale: This column sets the scale of the hatch pattern.

Layer: This column sets the CAD layer on which to draw the timing block.

Label: This column sets a custom label for the timing block.

Clear: This button will clear the table of all information.

Auto Set: This button will open the below dialog, which allows for quick population of the Custom Dates table.



Starting Line: This value will set the row to begin the automatic population of the Custom Date table.

Starting Date: This is the Date From that will be used on the Starting Line. The ellipse button will open a small calendar to assist in date selection.

Hatch Scale: This value will set the hatch scale for the new time periods.

Hatch Pattern: This field will set the hatch pattern for the new time periods. The ellipse button will open the list of available hatch patterns.

Set Same Layer: This option will place all new time periods on the same layer, which is set just to the right of this checkbox. The ellipse button will open the list of existing CAD layers.

Set Colors: This option will set custom colors for the new time periods. The colors are referenced in the CAD color palette, and are numbered 1 through 255.

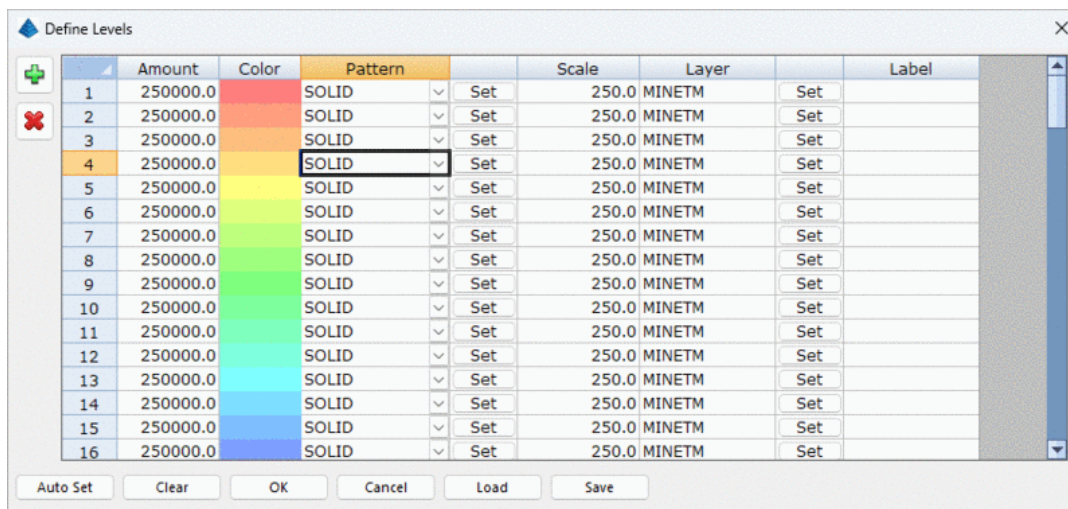
Colors by Year: When this option is enabled, the colors will only increment when the starting date for a row changes. This option is handy for drawing timing maps that have the same color for an entire year, but generate the report on a monthly basis.

Color Increment: This value sets the increment of colors to use for coloring. For example, a value of 10 will increment through colors 10, 20, 30, etc.

Start Color: This value sets the color to use on the Starting Row.

Repeat: This option will determine the length of the time periods. Selecting the *Every* option will allow you to set a custom length for the time periods. The number of time periods to create is set below this dropdown list.

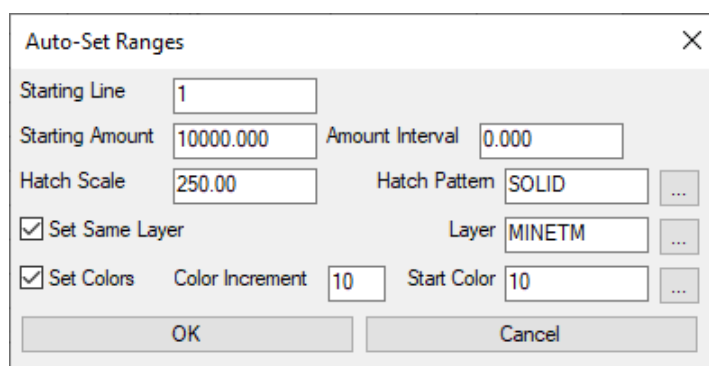
Amounts: This button will open the below dialog, which controls another method of custom coloring of the timing map.



The above dialog is very similar to the Custom Dates table previously discussed. The only difference is that the dates are replaced with the **Amounts** column. These values may represent one of the following values to be mined:

- Total Tons
- Key Material Tons
- Waste Tons
- Total Area
- Mined Area
- Total Volume
- Key Material Volume
- Waste Volume
- User Grid

Auto Set: This button will open the below dialog, which is very similar to the Auto Set dialog used in the Custom Dates table, as previously discussed. This dialog differs in that amounts are specified rather than dates, and the periods cannot be defined to repeat for a specific number of times.



Equipment: This button will open the Equipment Table, shown below, which is used to set the colors for each piece of equipment to be reported. This is like a simplified version of the Dates Table and Amounts Table. Here there is no option to set a report period - only the color, hatch, layer, and label for each piece of equipment.

	Equipment	Color	Pattern	Scale	Layer	Label
1	CM-1	Red	SOLID	100	MINETM	CM1
2	CM-2	Yellow	SOLID	100	MINETM	CM2
3	CM-3	Green	SOLID	100	MINETM	CM3

Draw Period Solids: This option will draw the timing solids in the drawing window. The solids will be colored according to the Report By option previously discussed.

Solid Transparency: This dropdown list will determine the transparency of the timing solids that are drawn in the drawing window.

Draw Legend: This option will draw a color legend of the timing map.

Legend Scale: This option will set the text size of the timing map legend.

Solid Labeling: This dropdown list will determine how the labels are drawn in the timing map.

The *No Block Labels* option will not draw labels.

The *Draw Actual Dates* option will label the starting and ending date of each period.

The *Draw Period Names* option will draw labels defined in the Custom Date Table.

The *Use Custom Names* option will draw labels defined in the Custom Amount Table.

The *Use Custom Text Block* option will draw labels according to the below dialog. When selected, the **Text Block Style**, **Add**, **Edit**, and **Remove** buttons will become available. The Text Block Style dropdown list will list all available text block labels. The Add button will create a new text block. The Edit button will edit the currently selected text block. The Remove button will remove the currently selected text block.

List of pre-defined keys	Buttons	Line definitions for Text Block
ENTWIDTH	<input type="button" value="Add >"/> <input type="button" value="Add Text"/> <input type="button" value="Add Attribute"/> <input type="button" value="Add New Line"/> <input type="button" value="Remove"/>	EQUIPMENT
EQUIPMENT		START
EXTRACT		FINISH
FINISH		
MAREA		
NAME		
OWNER		
PERIOD		
PROPERTY		
RADVANCE		
RECOVER		
BACKPENS		

The left side of the above dialog contains pre-defined keys that may be used in the labels. The right side of the dialog shows how the text block label will appear.

Add: This button will add the selected attribute to the current line. Multiple attributes may be added to a single line.

Add Text: This button will add custom text to the current line. In the above example, "tons" has been added as a suffix to the KEYWEIGHT attribute.

Add Attribute: This button will add any attribute that is not defined in the pre-defined list. This attribute name must match an attribute available in the final text report.

Add New Line: This button will create a new line in the text block. In the above example, three lines have been created.

Remove: This button will remove the selected line from the text block.

Text Size: This value will set the text size of the block labels.

Text Autosize: This option will automatically size the text of the block labels. The text size will vary, as it will be set to 1/6 of the length of the block (e.g. a block that is 600 units long will create a text height of 100 units).

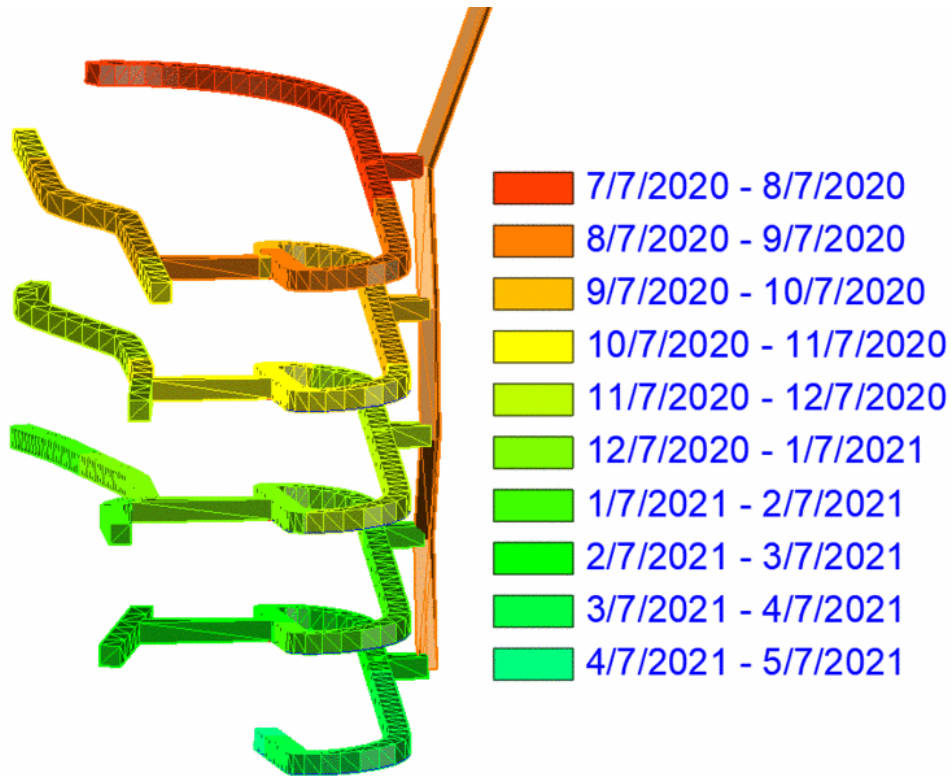
Text Style: This option will set the text style to use for the period labels. The **Select** button will allow you to select one of the existing text styles.

Length-wise labels: This option will rotate the block labels in line with the mining direction of each bench. If not selected, labels will be drawn perpendicular to the mining direction.

Report Only: This option will not create a timing map; only the mine schedule will be reported. When selected, many other options on the dialog will be ghosted out.

Output Timing Viewer File: This option will output a Timing Viewer File of the mine progression. This is a 3D playback that may be controlled with a timeline. You may specify the file name and location by clicking the ellipse button. More information on the Timing Viewer File is available [here](#).

An example report and timing map is shown below.



Edit : C:\Users\tfaulkner.CARLSON\AppData\Roaming\Carlson Software\Carlson2021\R24.0_X64\USER\scadrprt.tmp

File Edit Settings

KEY production report Tue Jul 21 17:25:20 2020

Period	Equipment	Date Start	Date Finish	Year	(start)	KEY (Tons)	Non-Key (Tons)
7/7/2020 - 8/7/2020	UNIT 1	7/7/2020	8/1/2020	2020		7512.9	
7/7/2020 - 8/7/2020	UNIT 1	8/1/2020	8/6/2020	2020		1787.1	
7/7/2020 - 8/7/2020	UNIT 2	8/1/2020	8/2/2020	2020		549.9	
7/7/2020 - 8/7/2020	UNIT 2	8/3/2020	8/6/2020	2020		1237.2	
8/7/2020 - 9/7/2020	UNIT 1	8/7/2020	9/2/2020	2020		7927.0	
8/7/2020 - 9/7/2020	UNIT 2	8/7/2020	8/29/2020	2020		6785.4	
8/7/2020 - 9/7/2020	UNIT 2	9/2/2020	9/6/2020	2020		1346.0	
8/7/2020 - 9/7/2020	UNIT 1	9/2/2020	9/6/2020	2020		1373.0	
9/7/2020 - 10/7/2020	UNIT 2	9/7/2020	9/7/2020	2020		166.5	
9/7/2020 - 10/7/2020	UNIT 1	9/7/2020	9/24/2020	2020		5325.5	
9/7/2020 - 10/7/2020	UNIT 2	9/7/2020	10/6/2020	2020		8860.5	
9/7/2020 - 10/7/2020	UNIT 1	9/25/2020	9/26/2020	2020		549.9	
9/7/2020 - 10/7/2020	UNIT 1	9/26/2020	10/6/2020	2020		3124.6	
10/7/2020 - 11/7/2020	UNIT 2	10/7/2020	10/15/2020	2020		2407.1	
10/7/2020 - 11/7/2020	UNIT 1	10/7/2020	10/26/2020	2020		5836.8	
10/7/2020 - 11/7/2020	UNIT 2	10/15/2020	10/21/2020	2020		2062.5	

There are many attributes available in this report. Most of these attributes are self explanatory, but some are not as clear. The below definitions are therefore provided.

- Calendar Days: Actual period length rounded up (to account for partial days)
- Operating days: Actual period length rounded up, minus days that do not have at least one working shift
- Days: Scheduled Hours divided by Shift Length
- Weekdays Worked: Weekdays with at least a portion of shift worked

- Saturdays Worked: Saturdays with a least a portion of a shift worked
- Sundays Worked: Sundays with a least a portion of a shift worked
- Days Waiting: Number of days equipment cannot work due to a precedence rule
- Work Shifts: Number of shifts when production took place
- Hours Scheduled: Number of hours equipment is scheduled to work (this includes Hours Utilized, Hours Delayed, and Hours Available)
- Hours Down: Number of hours equipment is not scheduled to work (due to downtime scheduled on the calendar)
- Hours Utilized: Number of hours equipment is actually working
- Hours Delayed: Number of hours equipment is delayed due to a Drawing Event or Scheduled Delay (note, this is not the same as Hours Down)
- Hours Available: Number of hours is scheduled to work, but is neither being utilized nor is it actually delayed. If a piece of equipment encounters a delay during the middle of the shift, the partial shift time remaining will be counted as Hours Available (the delay will be applied to the next shift)

The below image helps visualize the organization of the different definitions of Hours.

Hours in a day	Hours Scheduled	Hours Utilized
		Hours Delayed
		Hours Available
	Hours Down	

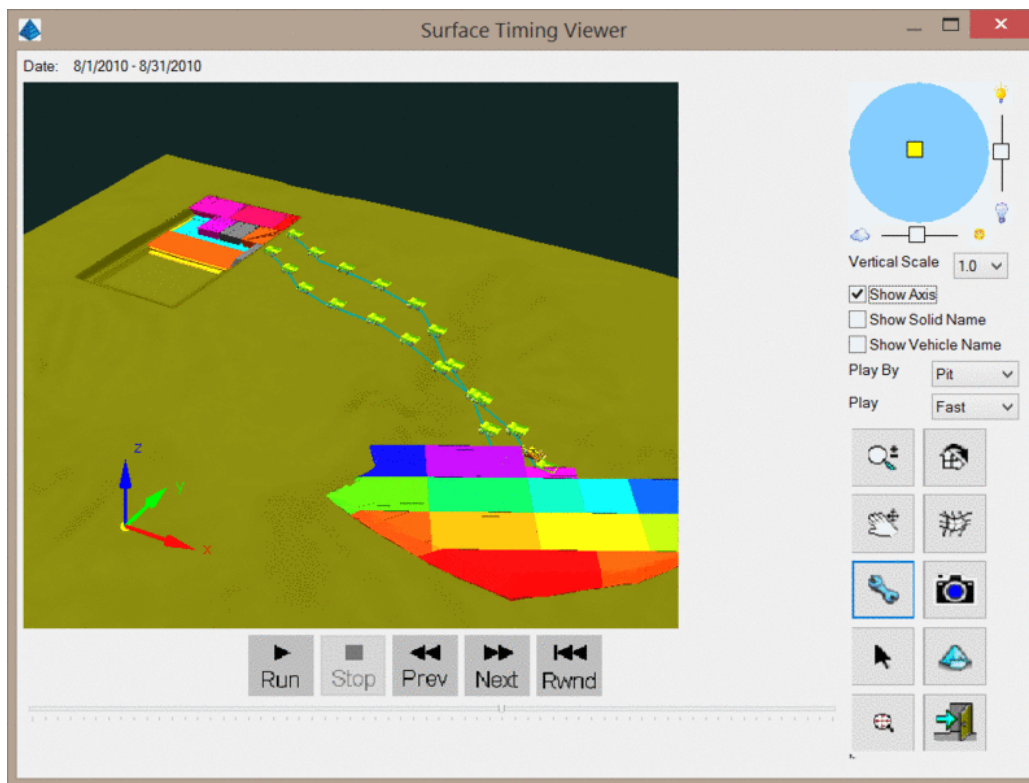
Pulldown Location: Solid

Keyboard Command: UGMINESCHEDULE

View 3D Timing

This command will open a .tv file to playback the progression of the mine. This file can be created from Surface Equipment Timing, Spoil Placement Timing, and Haul Cycle Analysis.

When the command is executed, you will be prompted to select a TIN file representing the topography at the end of the mine life. You will then be prompted to select a .tv file to view, after which the below dialog will appear. It is important to note that in order to view a .tv file, the linework for the spoil piles, haul routes, and the period polylines (output from Surface Equipment Timing as Pits) must be present in the drawing.



Vertical Scale: This value sets the vertical exaggeration for the 3D view.

Show Axis: This toggle controls the display of the X-Y-Z Axis icon in the lower left corner of the 3D view.

Show Solid Name: This toggle controls the display of labels above each solid. Names will only be displayed for solids that are being moved during the current time period.

Show Vehicle Name: This toggle controls the display of labels above each piece of equipment.

Play by: This option controls the playback of the 3D View. The *Pit* option will increment the time period based on when each pit is completely mined out. The *Spoil* option will increment the time period based on when a spoil polyline is completely filled.

Control



Action

This control represents the position of the sun in plan view. If the yellow square is in the center of the blue circle, the sun is in a zenith. If the yellow square is near the edge of the circle, the sun is near the horizon. To move the yellow square, simply drag it to a new location, or click on the new location. The slider bars on the sides control the intensity and brightness of the display.



Switch to Dynamic Zoom mode. Click and drag to zoom in and out.

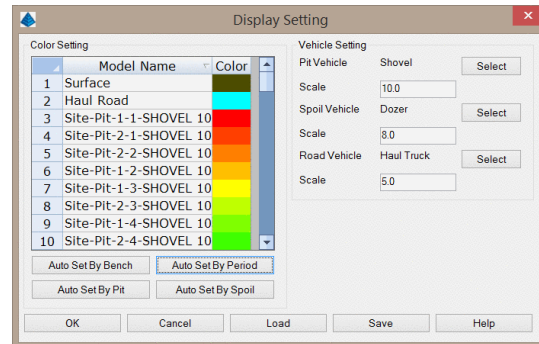
Switch to Rotation mode. When the cursor is placed near the outer edge of the view, a "Z" cursor is presented that permits rotation around the Z-axis. When the cursor is placed further into the interior of the view, an "X,Y" cursor is presented that permits the tilt angle of the view to be adjusted.

Switch to Pan mode. Click and drag to pan.

Toggle shading of 3D Faces and Surfaces.



Controls coloring and appearance of 3D symbols. When clicked, the below dialog will appear.



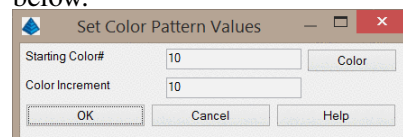
The spreadsheet shown in this dialog lists the name and color of each feature in the 3D window. You can automatically color the solids using one of the Auto Set By... buttons.

Auto Set By Bench: This option will color pits and spoil piles by bench number. For example, all instances of bench 1 will be red, all instances of bench 2 will be orange, etc.

Auto Set By Period: This option will color pits and spoil piles by the time period in which they are modified. For example, all material moved in the first month will be red, all material moved in the second month will be orange, etc.

Auto Set By Pit: This option will color pits and spoil piles by the pit. For example, all material from the first pit mined will be red, all material from the second pit mined will be orange, etc.

Auto Set By Spoil: This option will color pits and spoil piles by the spoil pile. For example, all material that is placed in the first spoil pile will be red, all material placed in the second spoil pile will be orange, etc. All of these buttons will display a similar dialog, shown below.



In this dialog, you can set a **Starting Color #** and a **Color Increment** according to the CAD color palette. The default option of a Starting Color # of 10 and a Color Increment of 10 will produce a rainbow gradient. A Starting Color # of 11 and a Color Increment of 10 will produce a lighter shade of a rainbow gradient.

The **Vehicle Settings** allow you to set different 3D models for the pit equipment, the haulage equipment, and the spoil equipment. The **Select** button will open the 3D Model Library, which includes several pieces of mining equipment. The **Scale** value controls the size of the model in the 3D view.



Takes a screenshot of the 3D Viewer. Several different image file formats are supported including bmp, png, jpg, xpm and gif. The image resolution and color depth may be set with this option.



Switches to Inspection mode. In this mode, hovering the cursor over a solid will display the name of the solid.



Outputs the current time period as a TIN file. With this command, you can progress through the timeline to key points in the mine life and export a surface model.



Exit the command.



Starts playback of the mine timeline.



Stops playback.



Steps back one time period.



Steps forward one time period.



Resets the playback to the beginning of the timeline.



The slider bar shows the current position in the timeline. You can manually click-and-drag the slider bar to progress through the timeline.

Pulldown Menu Location: Reserves/Timing
Keyboard Command: tvviewer

Clear Solid Timing

This command will remove 3D faces and labels generated by the Solid Timing command.

Drop-Down Menu Location: Solid
Keyboard Command: mtime_clear

Import File To Solid

This command creates a solid model in .MDL file format from other solid formats including .STL, STLA, .OBJ, .DXF, .SKP (SketchUp) and .TIN (Carlson Triangulation).

Prompts

Input File To Read *Pick the file to import*
Solid File To Write *Enter MDL file to output*

Pulldown Menu Location: Solid> Solid Utilities

Keyboard Command: import2mdl

Prerequisite: file to import

Export Solid

These commands output a solid model in .MDL file format to other solid formats including .STL, .OBJ and .TXT. The .STL and .OBJ are industry standard formats. The .TXT outputs the points from the solid as space separated X Y Z. The .TXT has options to output the normal vectors at each point and the triangle indices for the three points for each triangle.

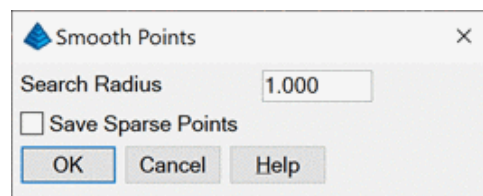
Pulldown Menu Location: Solid> Solid Utilities

Keyboard Command: mdl2stl, mdl2obj, mdl2txt

Prerequisite: Solid MDL file

Smooth Points

This command using a moving least squares method to modify the position of points to make a smoother data set. For each point, the program calculates a least-squares surface using the point plus neighboring points. The Search Radius controls how many points to process when smoothing a point. The Save Sparse Points controls whether to save sparse points which would be thrown out with point smoothing. If points do not have at least 3 neighboring points within the search radius they are discarded. This setting will preserve them.



Prompts

Select points to process.

Select objects: *pick the points to process*

Smooth Points dialog

Pulldown Menu Location: Solid> Solid Utilities

Keyboard Command: smooth_pts

Prerequisite: point entities

Reduce Points

This command erases point entities in the drawing for any points that are under the specified minimum distance of other points. This command is a way to remove extra points from a laser scan that is more dense than needed for solid modeling.

Prompts

Min distance: 1.0

Select points to process.

Select objects: *pick the points to process*

Pulldown Menu Location: Solid> Solid Utilities

Keyboard Command: ptreduce

Prerequisite: point entities

Erase Isolated Points

This command removes points without a specified number of neighbors within a distance. This command is a way to remove isolated points from a laser scan that are from noise or don't belong in the model.

Prompts

Select points to process.

Select objects: *pick the points to process*

Search radius <1.0>: *press Enter*

Min neighbor count <3>: *press Enter*

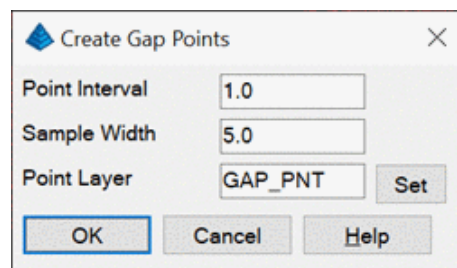
Pulldown Menu Location: Solid> Solid Utilities

Keyboard Command: ptisolate

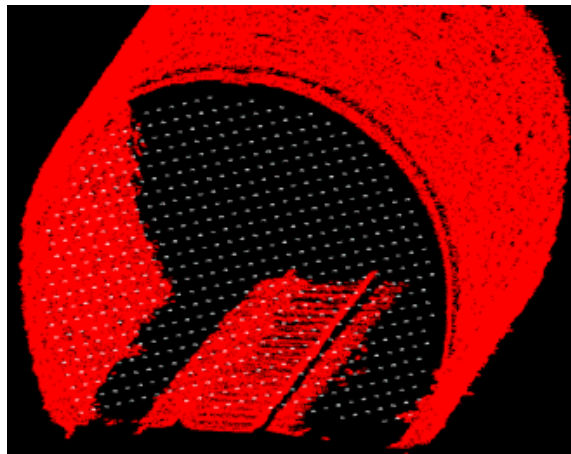
Prerequisite: point entities

Create Gap Points

This command creates points to fill a void in a point cloud such as the opening of a tunnel where there are no points from the scanner. The program prompts for three points around the perimeter of the void area. These points are used to define a 3D plane and the new points are created on this 3D plane. In the dialog, the Point Interval is the distance between the new points to create. The Sample Width is the distance from the 3D plane that the program uses to search for existing points to define the perimeter of the void area. The Point Layer is used for the new points.



The example graphic shows the existing points in red and the new gap points in white.



Prompts

Create Gap Points dialog

Pick 1st alignment point: *pick point*

Pick 2nd alignment point: *pick point*

Pick 3rd alignment point: *pick point*

Select points to process.

Select objects: *pick the existing points around the void area*

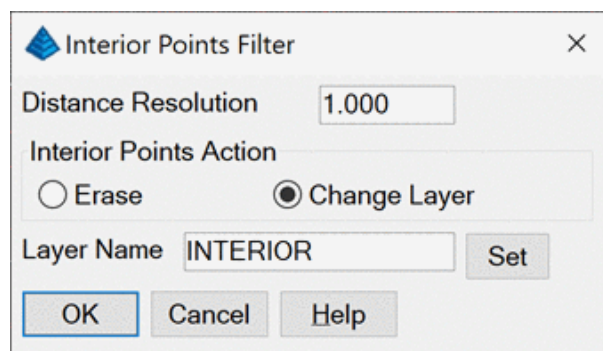
Pulldown Menu Location: Solid > Solid Utilities

Keyboard Command: gap_pts

Prerequisite: points

Interior Points Filter

This command removes wayward interior points in a tunnel or cavity point cloud such as people or equipment picked up in a scan. The routine checks that points fall within a triangulated mesh of varying triangle sizes. Using smaller triangles should capture more detailed exterior surfaces, but are prone to labeling interior points as exterior points. Larger triangles will capture less surface detail but are more likely to drop interior points. The triangle size is set by Distance Resolution. The dialog has options to either erase the interior points or change their layer.



Prompts

Interior Points Filter dialog

Select points to process.

Select objects: *pick the points to process*

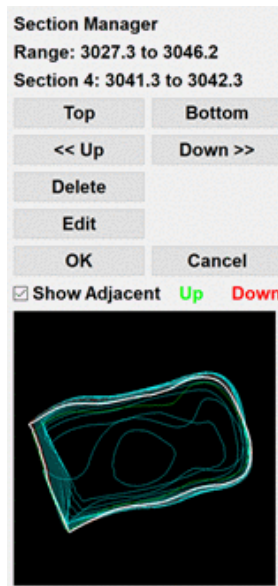
Pulldown Menu Location: Solid > Solid Utilities

Keyboard Command: interior_pts

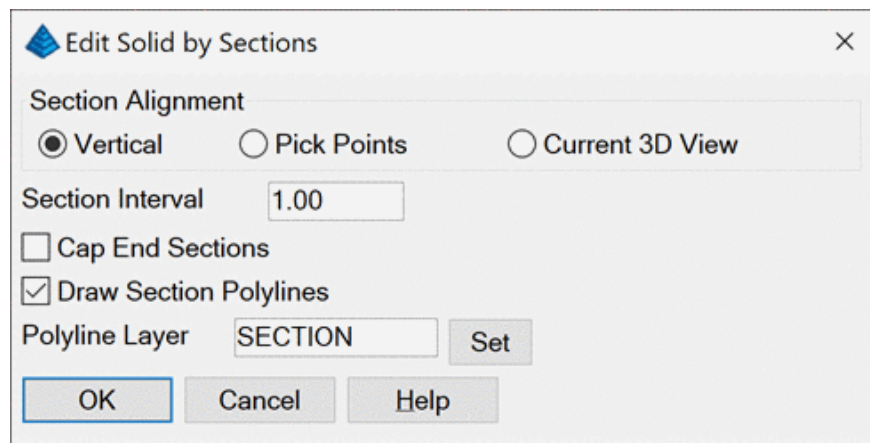
Prerequisite: point entities

Edit Solid

This command edits a solid model (.MDL) by using cross sections. This command is a way to review the solid model in sections and make edits to the sections to correct the model such as for removing nobs.



After selecting the solid model to process and choosing the options in the dialog, the Section Manager dialog docks on the side of the drawing. The Section Manager dialog shows all the cross sections in a graphic window. You can choose the current section by using the Top, Bottom, Up, Down buttons. The current section is highlighted in white. Besides showing the section in the Section Manager graphics windows, the section is also drawn in the drawing window as a polyline. Use the Edit button to edit the current section polyline in the drawing. The edit starts by picking a point on the polyline to begin recreating. Then pick new points for the section polyline. To complete the edit, press Enter at the prompt for new points and then pick a reconnection point on the existing polyline. The Show Adjacent option highlights the section above and below the current section in green and red which helps visualize how the current section transitions to its neighbors. The Delete function removes the current section from the model. When you pick OK on the Section Manager, the program rebuilds the solid model from the sections and prompts for a new MDL file to save the modified model.



Section Alignment: Sets how to section the model. It is important to align the sections with the model to get good sections. For example, with a tunnel the alignment should go along the length of the tunnel so that the sections are like slicing a carrot into discs. The Vertical method slices by elevation range. The Pick Points method prompts for two points and uses the 3D line between these points for the alignment. The Current 3D View method uses the CAD view. This way you can use the CAD 3D view commands to line up a good view for the points.

Section Interval: Sets the distance between sections along the alignment. This distance needs to be big enough to ensure that plenty of points are used for each section, and the distance needs to be small enough to create enough sections to model the solid.

Cap End Sections: This option ends the model at the end cross sections like the model getting sliced at the end sections. Otherwise, the program uses all the model data points at the ends and make a 3D triangulation for the model ends.

Draw Section Polylines: This option draws 3D polylines for the sections on the specified layer.

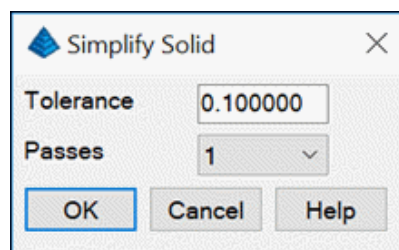
Pulldown Menu Location: Solid

Keyboard Command: edit_solid

Prerequisite: solid model

Simplify Solid

This command reduces the data points in a solid model using the quadric method. The **Tolerance** is the maximum distance the solid is allowed to shift. The program checks all the points in the solid model whether the point can be removed without changing the solid more than the tolerance. After checking all the points and removing points under the tolerance, the program can make another check of the points depending on the **Passes** setting.



Prompts

Solid File To Read: *pick an MDL file*

Simplify Solid dialog: *choose settings*

Solid File To Save: *specify output file*

Pulldown Menu Location: Solid > Solid Utilities

Keyboard Command: simplify_solid

Prerequisite: Solid file (MDL)

Heal Solid

This command will heal solids by filling holes, making the solid watertight.

Pulldown Menu Location: Solid > Solid Utilities

Keyboard Command: heal_solid

Remove Solid Dangles

This command removes triangles that are not connected to the majority of the solid model.

Prompts

Solid File To Read: *pick an MDL file*

Solid File To Save: *specify output file*

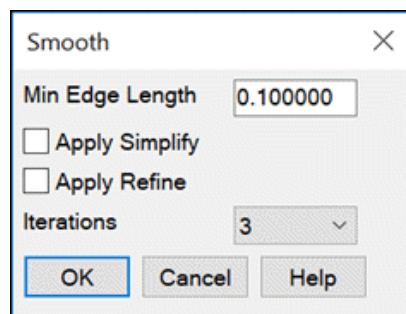
Pulldown Menu Location: Solid > Solid Utilities

Keyboard Command: solid_dangles

Prerequisite: Solid File (MDL)

Smooth Solid

This command smooths the solid by adding points using the Laplacian method. The Min Edge Length prevents adding points that result in a triangle edge smaller than this value. The Apply Simplify option removes points which can give the solid more freedom to smooth. The Apply Refine option adds points to make the triangles more uniform which can improve the smoothing. These Simplify and Refine options are applied before the smoothing. The Iterations is the number of times the smoothing is applied to the solid.



Prompts

Solid File To Read: *pick an MDL file*

Smooth Solid dialog: *choose settings*

Select inclusion perimeter polyline (Enter for None): *press Enter*

Solid File To Save: *specify output file*

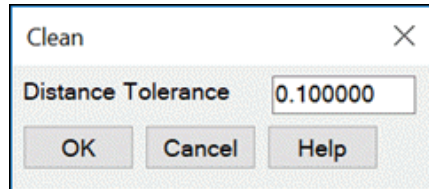
Pulldown Menu Location: Solid> Solid Utilities

Keyboard Command: smooth_solid

Prerequisite: Solid file (MDL)

Clean Solid

This command removes duplicate points and rebuilds the solid. Points closer than the Distance Tolerance are removed.



Prompts

Solid File To Read: *pick an MDL file*

Clean Solid dialog: *choose settings*

Solid File To Save: *specify output file*

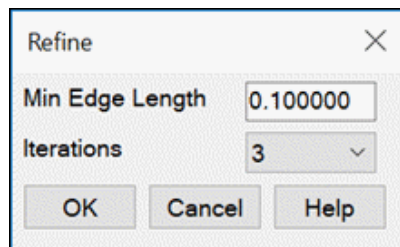
Pulldown Menu Location: Solid> Solid Utilities

Keyboard Command: clean_solid

Prerequisite: Solid file (MDL)

Refine Solid

This command adds points to the solid to make more uniform triangles. The Min Edge Length prevents adding points that result in a triangle edge smaller than this value. The Iterations is the number of times for the program to check the solid triangles.



Prompts

Solid File To Read: *pick an MDL file*

Refine Solid dialog: *choose settings*

Solid File To Save: *specify output file*

Pulldown Menu Location: Solid> Solid Utilities

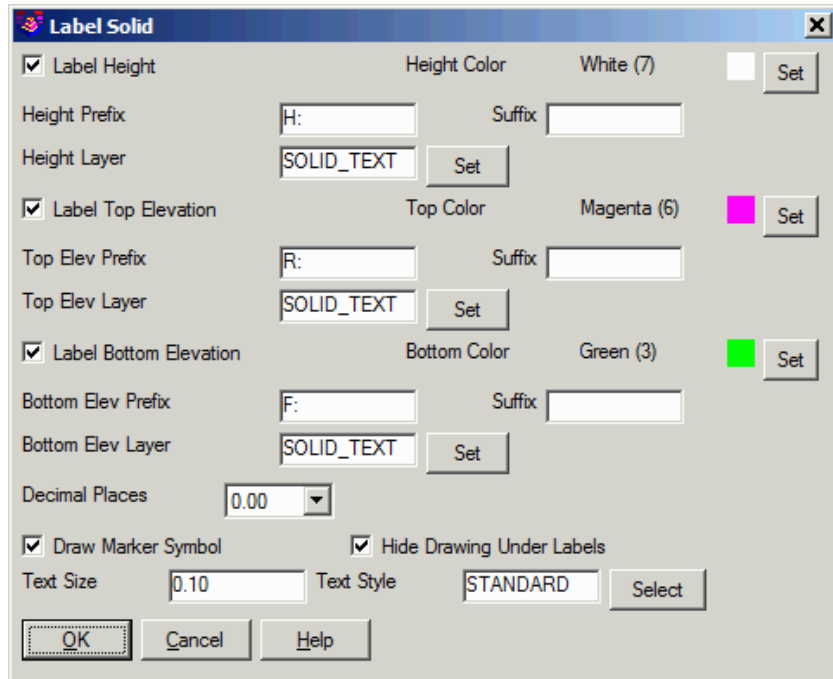
Keyboard Command: refine_solid

Prerequisite: Solid file (MDL)

Label Solid

This command draws labels at picked points for the solid model data. The available labels are the solid model height, top elevation and bottom elevation. For each label, there are controls for the prefix, suffix, color and layer. There are settings that apply to all the labels for the decimal places, size and style. The Draw Marker Symbol option places a symbol at the picked point. The Hide Drawing Under Labels option creates a wipeout under the labels.

× **H:19.84**
R:-558.52
F:-578.36



Prompts

Label Solid options dialog

Solid File To Read *Select MDL file*

Pulldown Menu Location: Solid

Keyboard Command: label_solid

Prerequisite: solid model file

Report Offset Point To Solid

This command calculates the shortest perpendicular distance of a point to the solid. The points can be either individually picked on the screen or read from the vertices of selected entities. The report includes the distance, coordinates of the point and nearest position on the solid, the normal vector from the point to the solid and whether the point is inside or outside the solid.

Prompts

Solid File To Read: *pick an MDL file*

Add from selected entities or pick points [Select/<Pick>]? *press Enter*

Pick point: *screen pick 3D point*

Pick point (Enter to end): *press Enter*

Report Formatter *Choose fields for your report*

Pulldown Menu Location: Solid > Solid Utilities

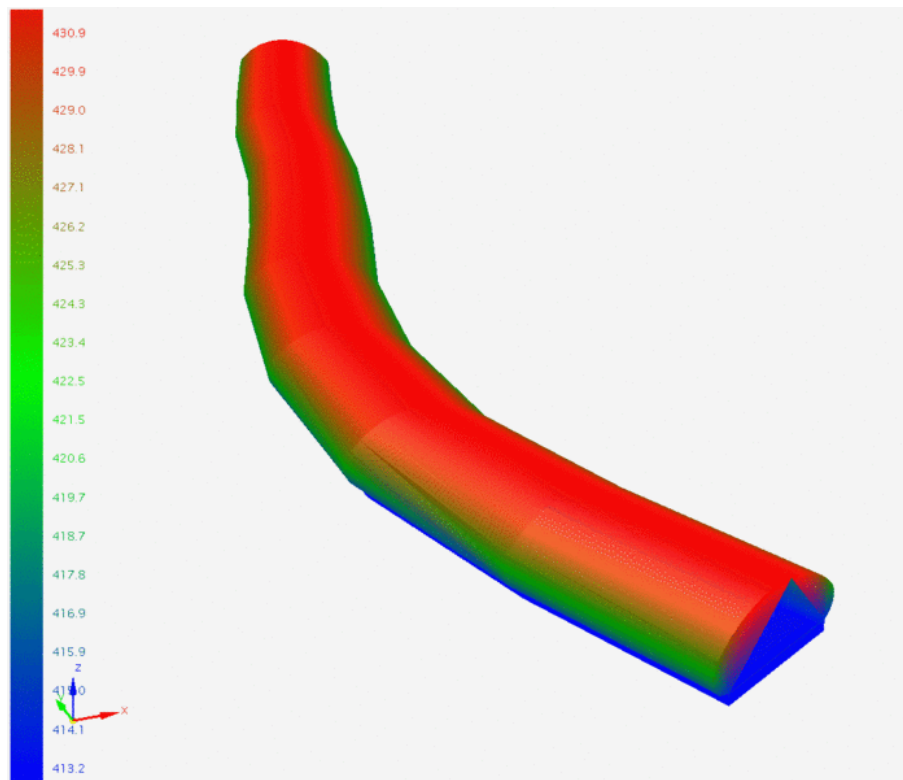
Keyboard Command: solid_pt_report

Prerequisite: solid model entities

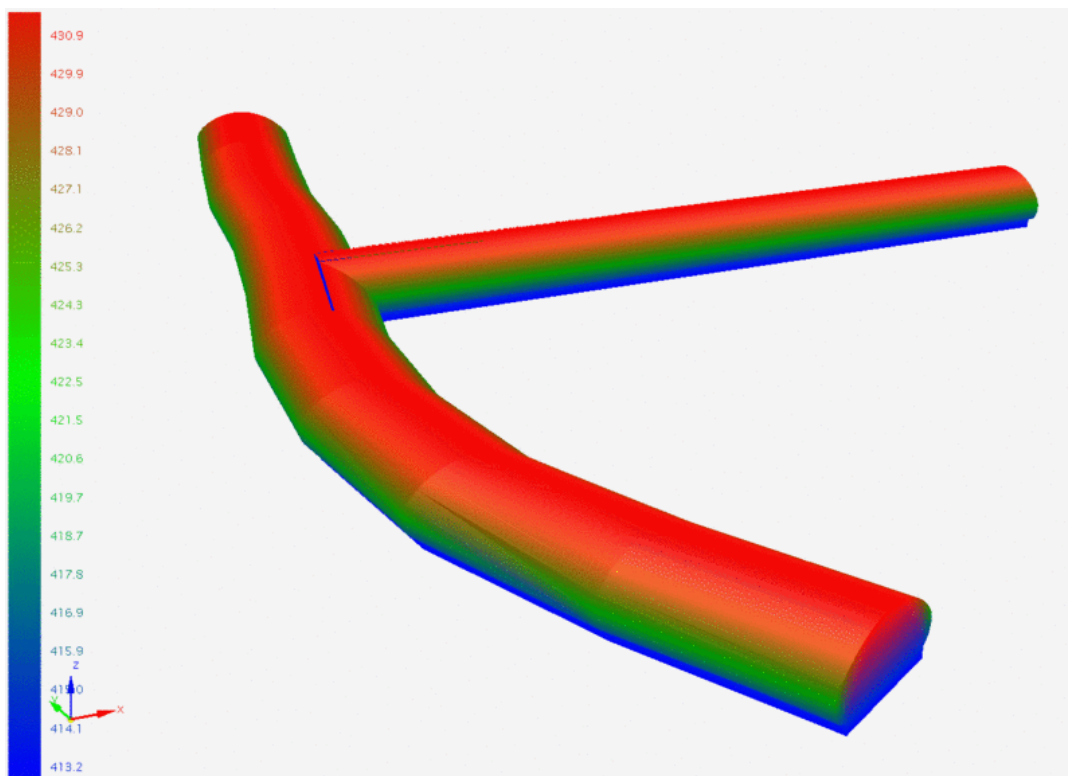
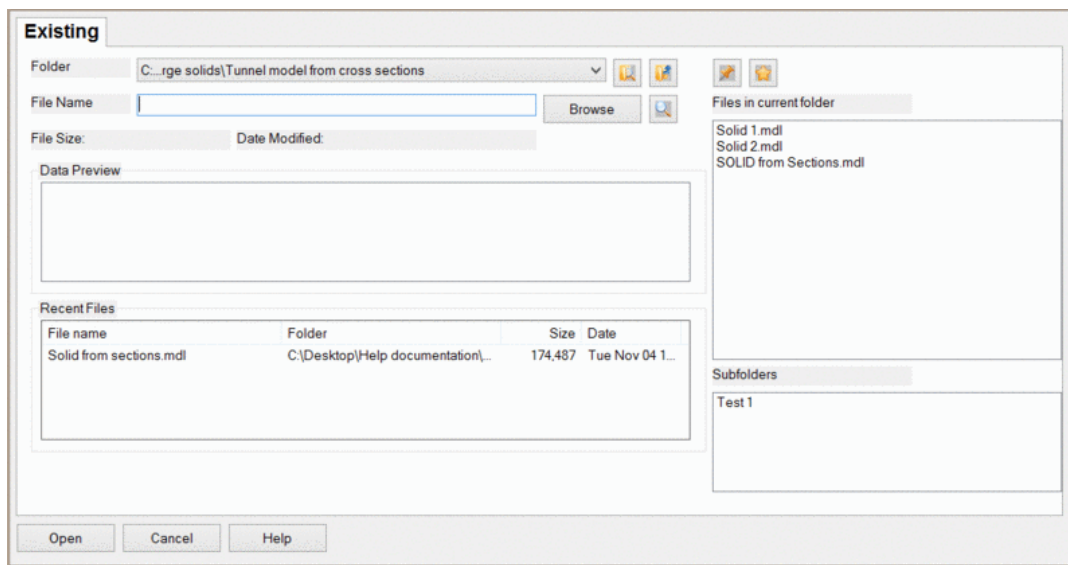
Merge Solids

This command allow you to merge two intersecting/touching solids into a single .mdl file. This command will allow you to select two .mdl files to merge. When prompted, you may assign a name to the merged .mdl file.

The below picture shows two solids, viewed separately.



When the command is executed, the below dialog will appear. The first dialog will prompt for the first .mdl file to load. The second dialog will prompt for the .mdl file to merge with the first.



Keyboard command: merge_solids

Prerequisite: Solid MDL files to merge

Combine Solids

This command puts all the triangles from two solid models into a combined solid model.

Prompts

Select First Solid: *pick an MDL file*
Select Second Solid: *pick an MDL file*
Solid File To Write: *specify output file*

Pulldown Menu Location: Solid> Solid Utilities
Keyboard Command: combine_solids
Prerequisite: 2 MDL files

Add Points To Solid

This command add more points to an existing solid. The points can be either individually picked on the screen or read from the vertices of selected entities. The program finds the nearest position on the solid to the new point and then stitches the point into the solid.

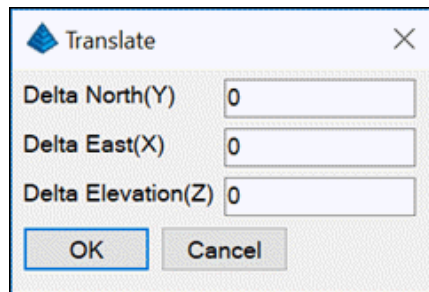
Prompts

Solid File To Read: *pick an MDL file*
Add from selected entities or pick points [Select/<Pick>]? *press Enter*
Pick point: *screen pick 3D point*
Pick point (Enter to end): *press Enter*
Solid File To Write: *pick an MDL file for output*

Pulldown Menu Location: Solid> Solid Utilities
Keyboard Command: add_solid
Prerequisite: Solid file (MDL)

Translate Solid

This command moves a solid by the specified amounts in X, Y and Z.



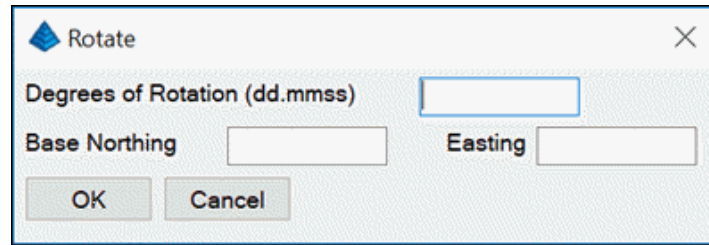
Prompts

Solid File To Read: *pick an MDL file*
Translate dialog: *choose settings*
Solid File To Save: *specify output file*

Pulldown Menu Location: Solid> Solid Utilities
Keyboard Command: translate_solid
Prerequisite: Solid file (MDL)

Rotate Solid

This command rotates a solid in X,Y around the Z axis. The Base coordinate is the pivot point for the rotation.



Prompts

Solid File To Read: *pick an MDL file*

Rotate dialog: *choose settings*

Solid File To Save: *specify output file*

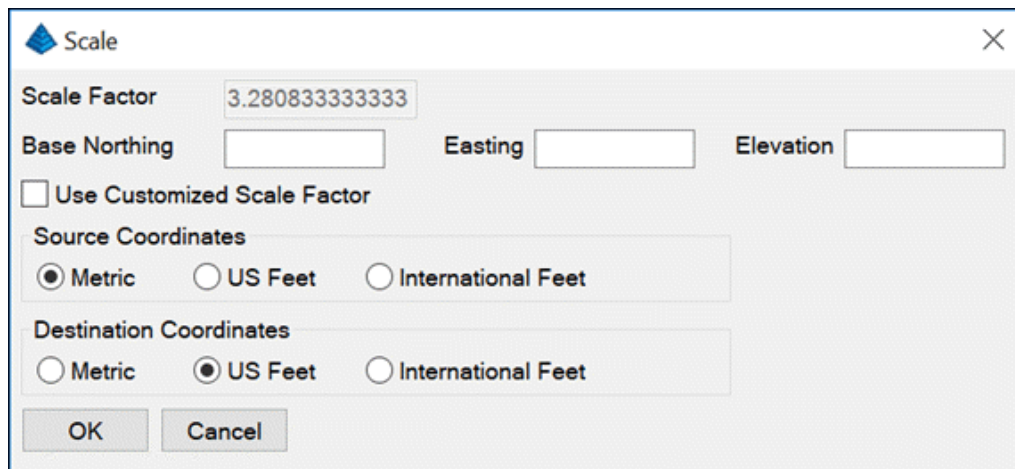
Pulldown Menu Location: Solid> Solid Utilities

Keyboard Command: rotate_solid

Prerequisite: Solid file (MDL)

Scale Solid

This command scales the points in a solid model. The scale factor can be a custom amount or use a factor to scale between meter and feet units. The Base coordinate can be used to scale the difference between the Base coordinate and the solid points.



Prompts

Solid File To Read: *pick an MDL file*

Scale dialog: *choose settings*

Solid File To Save: *specify output file*

Pulldown Menu Location: Solid> Solid Utilities

Keyboard Command: scale_solid

Prerequisite: Solid file (MDL)

Flip Solid Order

This command changes the order between clockwise and counter-clockwise of the three points that define each triangle in a solid model. The order of the triangle points determines which sides of the triangle are front and back. The order is important for 3D Viewing and volume calculations. This routine has two methods. The Auto method looks at the direction of the majority of the triangles and flips any triangles going the opposite of the majority direction. The Flip method reverses the direction for all the triangles.

Prompts

Solid File To Read: *pick an MDL file*

Solid File To Save: *specify output file*

Flip Solid Order dialog: *choose settings*

Pulldown Menu Location: Solid > Solid Utilities

Keyboard Command: order_solid

Prerequisite: Solid File (MDL)

Intersect Solids

This command draws 3D polylines along where two solids intersect each other.

Prompts

1st Solid File To Read: *pick an MDL file*

2nd Solid File To Read: *pick an MDL file*

Intersect Solid dialog: *choose layer*

Pulldown Menu Location: Solid> Solid Utilities

Keyboard Command: intersect_solids

Prerequisite: Solid files (MDL)

Trim Solid

This command trims a solid model by either a closed polyline or station range along an alignment polyline. The resulting trimmed solid is saved to a new MDL file. This command is a way to carve out a subset solid.

For the closed polyline method, the polyline should be entirely inside the limits of the solid.

For the alignment method, the centerline polyline should be a 3D polyline that goes along the full length of the solid. This method applies to solids of tunnels. The program prompts for the starting and ending stations along the polyline. The solid is sliced at these stations and then combined with the original solid faces between these stations to create the new solid.

Prompts

Solid File to Read *Select a .mdl file*
Select trim perimeter polyline: *pick a closed polyline*
Solid File to Write *Specify a .mdl file*

Pulldown Menu Location: Solids
Keyboard Command: trim_solid
Prerequisite: 3D Model file (.mdl) and closed polyline

Color Solid

This command assigns either a color or texture to a solid model file (.MDL). This command prompts for the solid model file and then the color or texture to assign.

Pulldown Menu Location: Solid > Solid Utilities
Keyboard Command: color_solid
Prerequisite: MDL file

Solid Information

This command reports the number of points and faces in a solid model along with the coordinate range.

Prompts

Solid File To Read: *pick an MDL file*
Solid: C:\sample\ore2\ball7.mdl
Min X,Y,Z: 22917.942,15537.403,8670.132
Max X,Y,Z: 23125.785,15671.585,8985.034
Number of Points: 3,436 Faces: 6,906

Pulldown Menu Location: Solid > Solid Utilities
Keyboard Command: solid_stats
Prerequisite: Solid File (MDL)

ID Solid

This command prompts to pick any solid 3D face and then reports the source solid model file. Use the Draw Solid command with the Tag Solid Name option to create 3D faces for this routine.

Prompts

Select solid: *pick a solid 3D Face*
Solid Name: RAMP2
Select solid ([Enter] to End): *press Enter*

Pulldown Menu Location: Solid
Keyboard Command: id_solid
Prerequisite: solid model entities

Triangulate & Contour

At the heart of nearly every land design project is at least one terrain model. These models go by several names and one of the most common is that of a "TIN" or Triangulated Irregular Network; another common name is that of a "DTM" or Digital Terrain Model. Since accurate representations of a surface model are significantly important to most land development projects, having a thorough understanding of the Triangulate & Contour controls is very important.

Surface models are generally comprised of combinations of the following general data types:

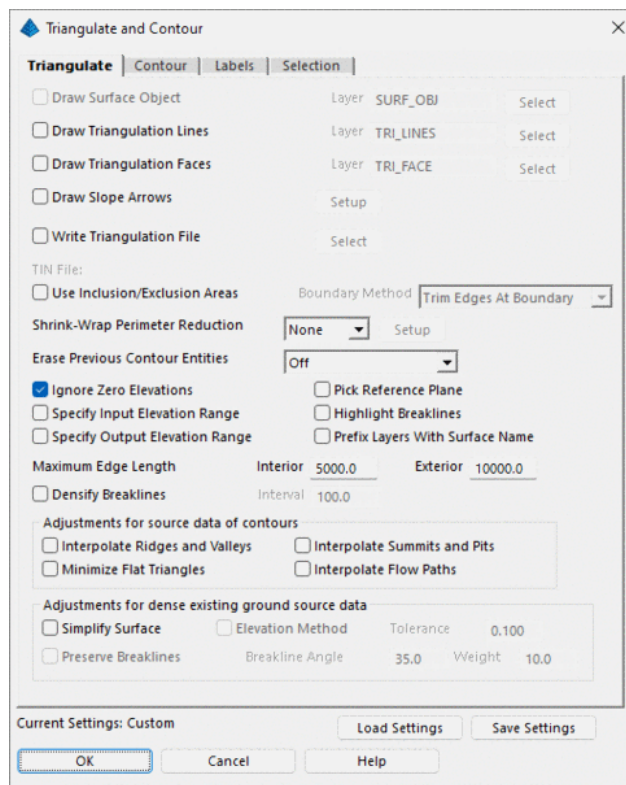
- Points - Most surface models are comprised of points whose coordinates (x,y,z) contribute to the formation of triangular planes that connect three points that are in close proximity to one another. Within Carlson, most points come from the Draw Field to Finish command and/or the Draw-Locate Points command. Points can be selectively filtered from the triangulation engine through the use of the Tag Non-Surface Points command.
- Breaklines - Breaklines (or "fault lines") are used to control the connection sequence between four points which results in two triangles. Common uses of breaklines include ravines, ditches, berms and other areas where distinct grade discontinuity occurs. The "leg" of a triangle can travel along a breakline but cannot cross the breakline. Breaklines must be in the form of 3D polylines or simple lines whose vertices or endpoints define a valid "Z" elevation. A common problem related to breaklines is when two breaklines cross one another in 3D space. In these situations, an impasse results and will result in a "crossing breakline" report. Within Carlson, most breaklines come from the Draw Field to Finish command and/or the 3D Polyline command. Breaklines fall into one of two general categories:
 - "Soft" breaklines - Unless otherwise specified, all breaklines are considered "soft" breakline. The nature of soft breaklines allows a degree of contour smoothing across the breakline itself resulting in a "weathered-" or natural-looking contour.
 - "Hard" breaklines - Breaklines tagged as "hard" breaklines prevent contour smoothing through the breakline. Hard breaklines are generally used to represent man-made terrain breaks that commonly occur during excavation and construction. Breaklines can be changed to hard breaklines through the use of the Tag Hard Breaklines command.

Breaklines and other triangulate-able entities can be selectively filtered from the triangulation engine through the use of the Tag Non-Surface Entities command.

- Inclusions - Inclusions (or "boundaries") are used to identify the entities that can be used for triangulation and multiple inclusion regions can be selected for a given surface model. Entities that fall outside of an inclusion boundary and are not otherwise bound by a different inclusion boundary are ignored by the triangulation engine. Inclusion regions must be in the form of a closed 2D or 3D polylines. Within Carlson, most inclusion polylines come from the Shrinkwrap Entities command.
- Exclusions - Exclusions (or "void regions") are the antithesis of Inclusions and are used to prevent triangulation from occurring between points that are bound by an Inclusion region. Common uses of exclusion regions include building footprints and free-standing water limits (*e.g.* ponds). Entities that fall inside an exclusion region are ignored by the triangulation engine. Exclusion regions must be in the form of a closed 2D or 3D polylines. Within Carlson, most exclusion polylines come from the Boundary Polyline command and/or the 3D Polyline command.

Carlson provides a programming interface for these file types and also offers a third file type (*.GRD) for the representation of terrain data. See the Notes section for additional details.

The **Triangulate** tab provides options and settings that control the creation and analysis of the TIN itself.

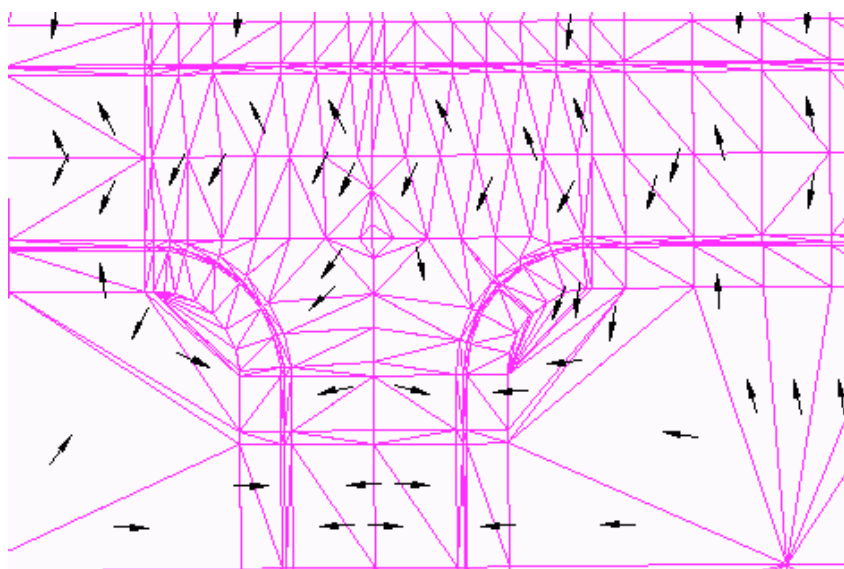


Draw Surface Object: This option draws the triangulation faces as a Carlson Surface Object.

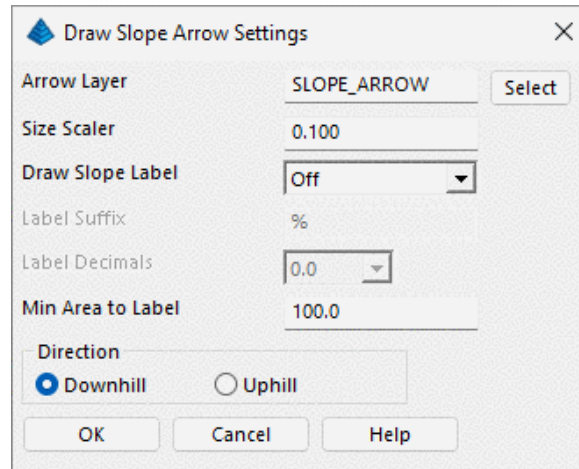
Draw Triangulation Lines: When enabled, the program will draw the triangulation using simple line entities at the appropriate elevation(s). Use the **Select** button or specify the layer for these lines.

Draw Triangulation Faces: When enabled, the program will draw the triangulation using a collection of 3D Face entities. These 3D Faces can then be used rendering routines such as *HIDE* and *SHADE* or in Carlson routines such as 3D Viewer Window, 3D Surface Fly-Over and Slope Zone Analysis. Use the **Select** button or specify the layer for these 3D Faces.

Draw Slope Arrows: When enabled, slope arrows are created within the triangles indicating the downhill dip direction as illustrated below.



Clicking the **Setup** button yields the Draw Slope Arrow Settings dialog box.



Arrow Layer: Indicate the layer to which the slope arrows are to be placed.

Size Scaler: Indicate a positive, non-zero value for the scale factor that should be applied to the slope arrows.

Draw Slope Label: When enabled, the slope value (expressed as a percentage or a ratio) of the triangle is labeled onto the slope arrow. Specify the desired unit suffix (e.g. "%") to apply to the end of the numerical value that is calculated from the TIN triangle(s).

Label Decimals: Indicate the amount of precision that is to be displayed on the slope label.

Min Area to Label: Indicate the smallest allowable triangle size that can be used for the slope percentage labels.

Write Triangulation File: When enabled (strongly suggested), an external surface model file is created which can subsequently be used for volume calculations, the creation of profiles, cross-sections and graded pads. Carlson currently provides two file types to store the DTM data created by the Triangulate & Contour routine:

1. *.TIN - The TIN file format is the default and preferred file format due to its compact file size and organizational efficiency. The Carlson TIN format is governed by Carlson and is in a binary (non-human readable) format.
2. *.FLT - The FLT file format is a legacy ASCII-based (human-readable) file format and is used in some older machine control applications.

Use Inclusion/Exclusion Areas: When enabled, the program will prompt you for inclusion and exclusion polylines and prevents the use of the Shrink-Wrap Perimeter Reduction option. These are used to further control the area of activity for triangulation and contouring. The inclusion and exclusion polylines must be closed polylines and when used, must be drawn before using *Triangulate & Contour*. It is suggested that the height of the **Command:** line display must be set to show at least two lines so that the additional prompts can be easily viewed. Refer to the Notes section for additional information on Inclusion/Exclusion polyline selections.

Boundary Method: This option controls whether edges that cross the inclusion or exclusion perimeter are trimmed or removed. Use the Remove option if you don't want the triangulation to interpolate across the perimeters. Use the Trim option if you want the triangulation to fill the area up to the perimeter.

Shrink-Wrap Perimeter Reduction: This option produces an inferred Inclusion region around the data to be selected and mimics the results of the Shrinkwrap Entities command. Under the Setup, there is an option to draw the shrink-wrap perimeter polyline on a specified layer.

Erase Previous Contour Entities: In the event that a TIN needs to be recreated and Carlson-produced contours are in the drawing, three options exist that allow you to control whether or not the contour data should be removed from the drawing:

- **Off** - All existing Carlson-generated contours are left intact in the drawing. If these contours satisfy all of the triangulation requirements, they can be utilized by the Triangulation algorithm.
- **Current Surface** - Only the Carlson-generated contours that are associated with the active Triangulation file are removed from the drawing.
- **All Contour Entities** - All Carlson-generated contours are removed from the drawing, regardless of the surface model that created them.

Ignore Zero Elevations: When enabled, this option will filter out all data points and entities at an elevation of zero from the triangulation data set.

Pick Reference Plane: When enabled, this option allows you to contour an overhang or cliff by changing the reference plane to a side view. The reference plane can be specified by using the *View>Viewpoint 3D>View* command (see the AutoCAD/IntelliCAD Help menu for additional details) or by specifying three data points on the cliff (two along the bottom and one at the top).

Specify Input/Output Elevation Range: If you would like to manually set the range over which to contour, select either or both of the aforementioned toggles. One controls the triangulation of the source data and the other for the contour output. The program will automatically contour from the lowest elevation in the data set up to the highest at the increment specified in Contour Interval.

Highlight Breaklines: When enabled, this routine highlights breaklines in the triangulation network by drawing the triangulation lines along breaklines in yellow.

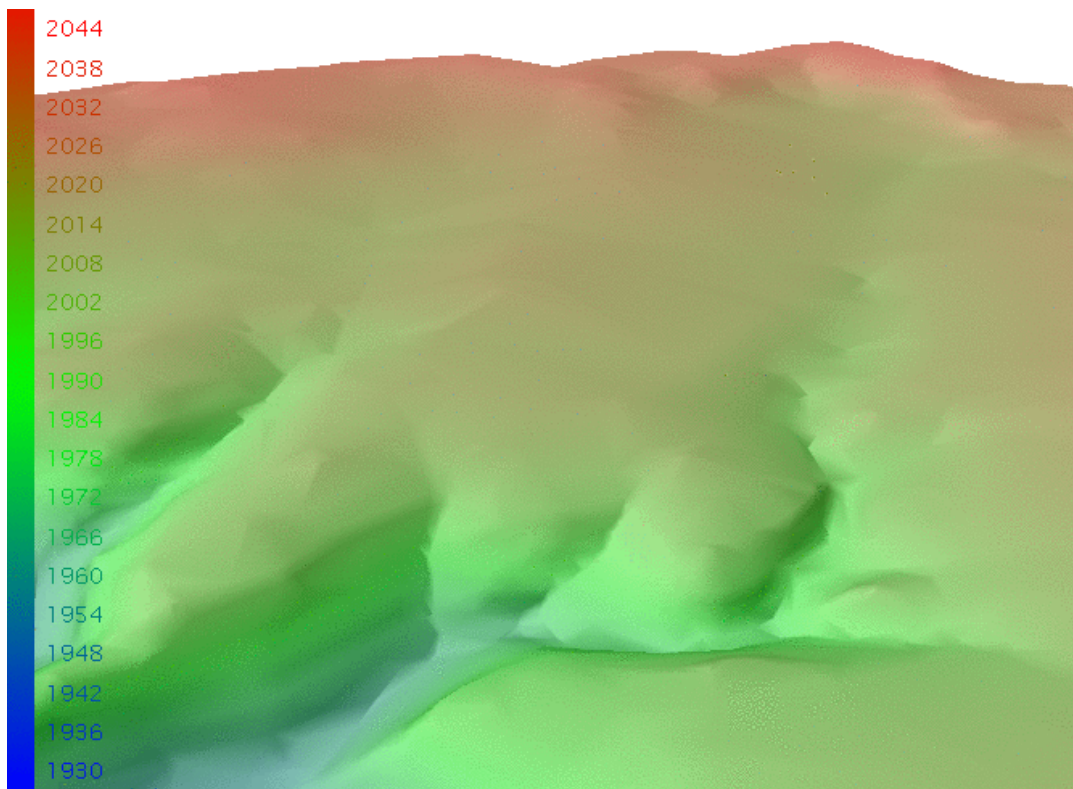
Prefix Layers With Surface Name: This option applies when using Write Triangulation File. The file name is added as a prefix to all the layers created during Triangulate and Contour as a way for layer management to organize all the contouring layers for a surface file.

Maximum Edge Length: Two bounds are provided to limit the length of the "legs" within a triangulation network. Based on the available data, if the edge length of a triangle exceeds the respective bound, the triangle will not be formed:

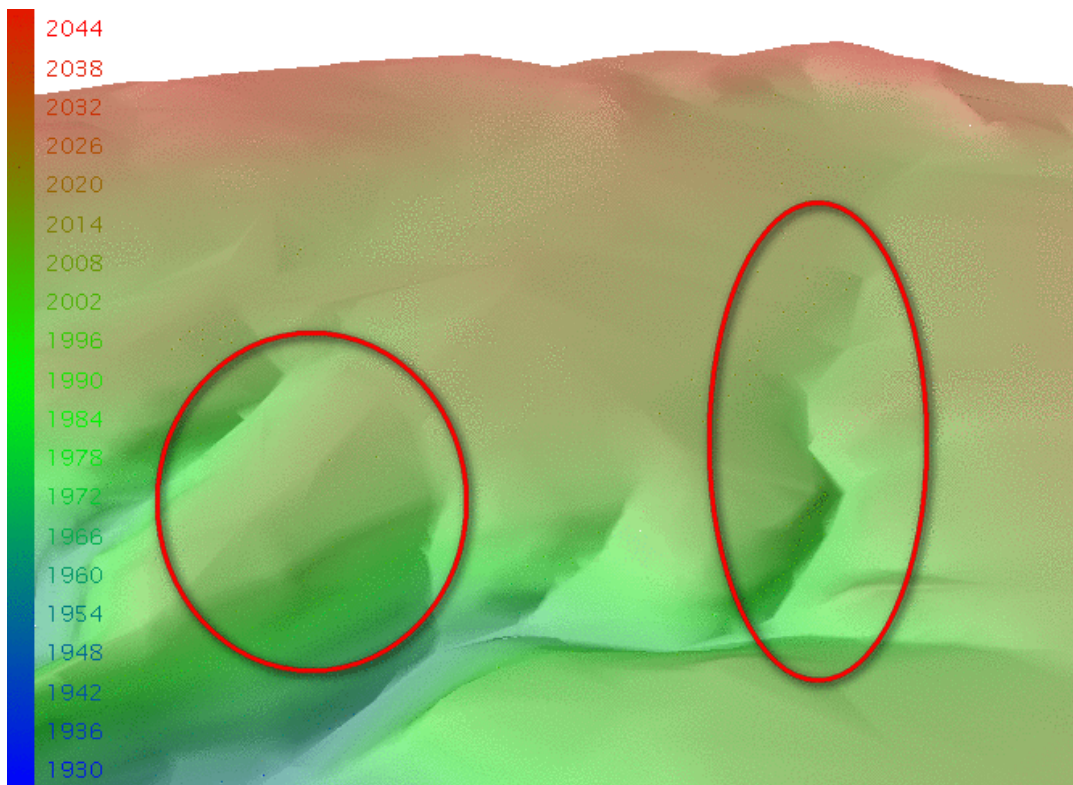
- **Exterior:** This value applies to triangulation lines around the perimeter of the triangulation area.
- **Interior:** This value applies all the other triangulation lines. Generally you would have the Exterior value larger than the Interior value.

Densify Breaklines: This option subdivides linework segments for the input data so that the maximum length of the segments is the specified Interval. This option is similar to the Densify Polyline Vertices command except that the entities are not modified in the drawing. Having shorter breakline segments is often helpful for holding the breaklines in the TIN and making more regularly sized triangles.

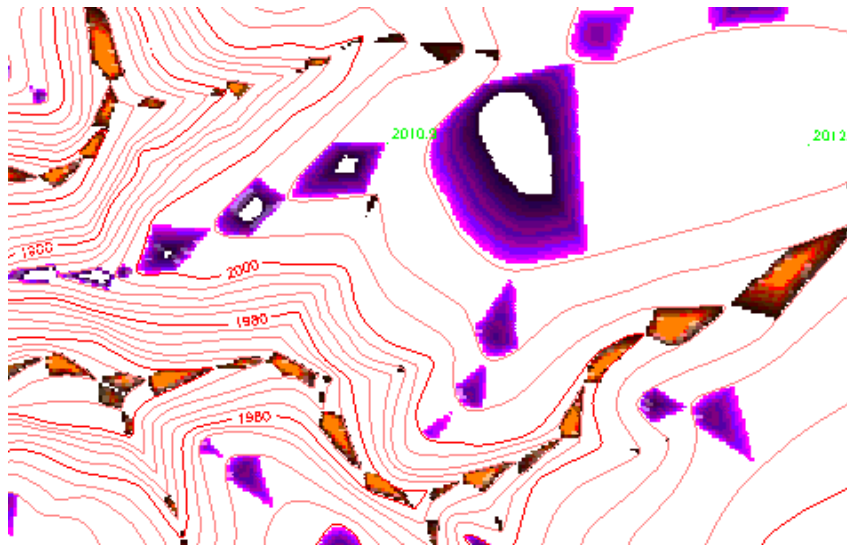
Minimize Flat Triangles: When enabled, this toggle instructs the triangulation "engine" to iterate through the triangulation permutations to minimize the occurrence of "flat" (or more precisely, horizontal) triangles. Flat triangles often occur when creating surface models from contour data. In this scenario, the often used Delaunay triangulation algorithm may produce unrealistic results. The Minimize Flat Triangle option will perform additional permutations of the triangulation network through the use of the Surface Manager > Swap Edge routine in an attempt to maximize the number of "sloped" triangles. Another option that produces similar results is the Interpolate Ridges and Valleys option.



Before: Surface made from an existing contour map with Minimize Flat Triangles disabled.



After: The same surface with Minimize Flat Triangles enabled. Note the better defined ravine and ridge definitions.



Difference: A Cut/Fill Color Map showing the regions of significant triangulation difference between the "Before" scenario and the "After" scenario of "Minimize Flat Triangles."

Interpolate Ridges and Valleys: The purpose of this option is to avoid flat areas by better defining surface ridges and valleys. When enabled, this option inserts breaklines into the drawing which are subsequently used in the triangulation process in an attempt to minimize flat, horizontal triangles.

Interpolate Summits and Pits: When enabled, this option creates additional triangulation in a summit or pit situation to more accurately represent existing ground conditions from a surface model created from contour entities. Since the tops of hills and the bottom of pits are often not shown on existing ground contour maps, this option often helps improve the accuracy of existing terrain conditions.

Interpolate Flow Paths: this function generates 3D data polylines from a contour map that approximate the downhill flow of water through valleys. These polylines are then used when creating a triangulation and can improve the it in these valley regions by giving more definition to these areas. The polylines are created by looking for "sharp" or angled regions of the contours, and then connecting these regions with a smooth curve.

Simplify Surface: When enabled, this option reduces the digital size of a surface without significantly compromising the integrity or accuracy of the surface itself. The most common application to enable this option is when using very large datasets, such as smoothed contours. Its use is less applicable to design surfaces or surfaces based on surveyed points, but it can still be utilized.

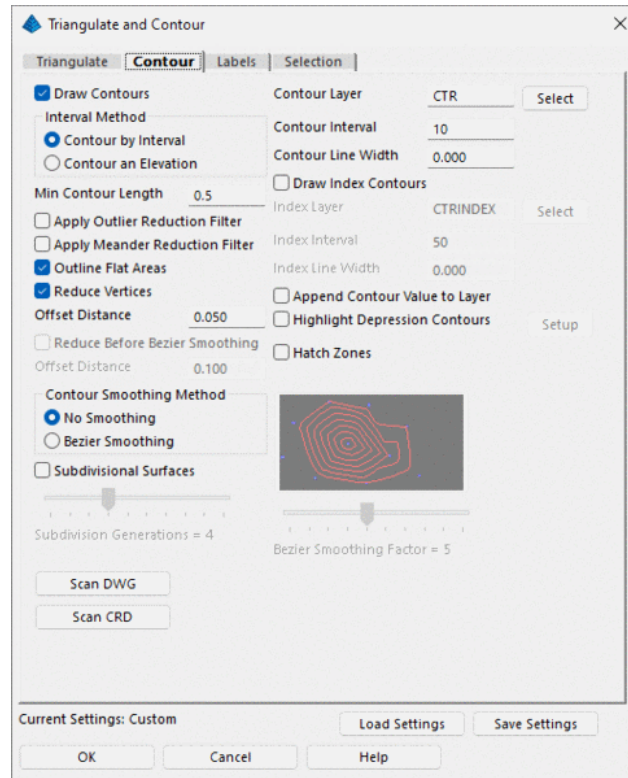
Elevation Method: When enabled, this option reduces the size of the surface file by analyzing the difference in elevation between each vertex of the TIN and the vertices directly surrounding it, assigning a numerical weight or value to each vertex. If it is determined that the calculated weight for a particular vertex is less than the *Tolerance* factor, the vertex is a candidate for removal. The number of vertices removed is directly proportional to the *Tolerance* factor, so the higher the *Tolerance* factor, the more vertices are removed and vice versa.

Preserve Breaklines: When enabled, this option analyzes the TIN by focusing on the edges; calculating the angular difference between adjacent triangular faces. If the angular difference between edges is greater than the specified *Breakline Angle*, it is considered to be a breakline, and it is preserved. If its angular difference is determined to be below the *Breakline Angle*, it becomes a candidate for removal. In that case, the *Weight* factor is applied to the corresponding vertex, adjusting its original value. If the resulting value is still below the *Tolerance*, it is then removed. The number of vertices removed is inversely proportional to the *Weight* factor, so the greater the *Weight* factor. The fewer vertices that are removed, the lower the *Weight* factor, the more vertices that are removed.

A good rule-of-thumb that can be used when deciding whether or not to use these options is:

- If the surface contains no man-made features, use *Simplify Surface* option (with or without the *Elevation Method* option).

- If the surface contains man-made features, such as roads, use both *Simplify Surface* and *Preserve Breaklines*.



Draw Contours: When enabled, the program will draw contour lines using the designated settings after triangulation process is complete. Otherwise, only the designated Triangulation operations are performed. If this option is disabled and contours are subsequently desired, use the Contours from TIN File command.

Interval Method: Indicate the desired elevation(s) for contours to be drawn:

- **Contour by Interval:** Specify the desired interval (*e.g.* every 2 feet) into the *Contour Interval* field.
- **Contour an Elevation:** Specify a desired elevation (*e.g.* a floodplain elevation or other unique elevation of interest) and set the desired value into the *Contour Interval* field.

Contour Layer/Index Layer: Specify the layer to which the contours/index contours are to be drawn.

Contour Interval/Index Interval: Specify the interval to which the contours/index contours are to be drawn. When contouring an elevation, you can enter multiple space-separated values to draw multiple contours. For example, entering "5 8 14" into this field would draw contours at elevations, 5, 8, and 14.

Contour Line Width/Index Line Width: Specify the line width to be applied to the contours/index contours.

Draw Index Contours: When enabled, index (or "major") contours will be created with independent characteristics from the regular contours.

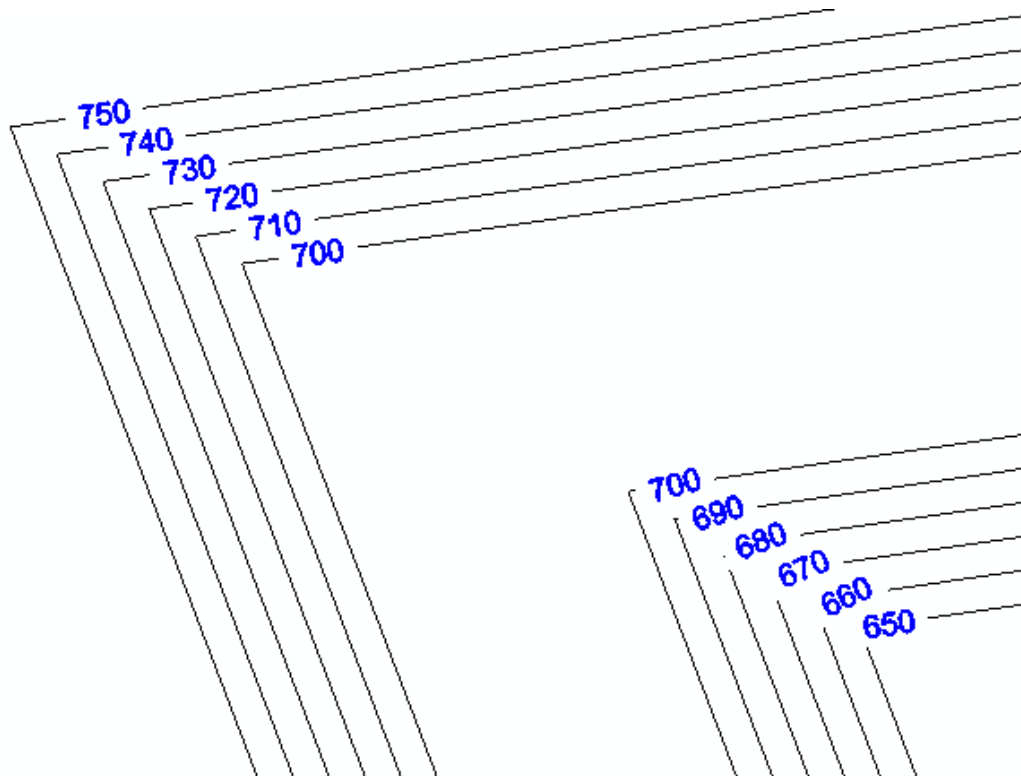
Min Contour Length: Specify the minimum linear threshold that should be used to draw contours.

Apply Outlier Reduction Filter: When enabled, this option attempts to remove "the jaggies" which tend to occur along long, thin triangles.

Apply Meander Reduction Filter: This option smooths contours by removing back-tracking.

Outline Flat Areas: This option will outline flat areas of the surface. This is particularly useful for showing catch-benches that have been designed into a steeply angled portion of the surface. In the below image, a the 700'

contour line is drawn on both sides of the catch bench. If the Outline Flat Areas option were disabled, only one side of the catch bench would be drawn.



Reduce Vertices: When enabled, this option removes extra vertices from the contours using the *Offset Distance* value. The Offset Distance is the maximum amount that the polyline can move horizontally when removing a point. The result of this action is often a significant reduction in vertex locations along the contour resulting in a more efficiently-sized and compact drawing file.

Offset Distance: Specify the maximum allowable distance for shifting the original contour line in order to reduce vertices. The reduced contour will shift no more than this value, at any point, away from the original contour line. A lower value will decrease the number of vertices removed and keep the contour line closer to the original. A higher value will remove more vertices and allows the contour to shift further from the original location.

Reduce Before Bezier Smoothing: When enabled, this option removes extra vertices from the contours before they undergo Bezier Smoothing using the *Offset Distance* value. The Offset Distance is the maximum amount that the polyline can move horizontally when removing a point. Removing points before smoothing gives the Bezier smoothing more freedom to make the contour curvy.

Contour Smoothing Method: Indicate the desired amount of smoothing (often used for existing, natural ground conditions to simulate a "weathered terrain" effect) that should be applied to the contours:

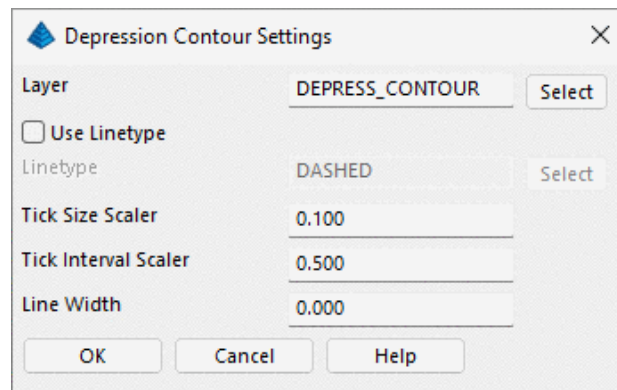
- **No Smoothing:** This option is often used for proposed, man-made surface considerations where the terrain has been shaped with earth-moving equipment. For applications where a "nature-emulated" man-made terrain is desired, refer to the Carlson Natural Regrade documentation.
- **Bezier Smoothing Factor:** This option holds all the contour points calculated from the triangulation and only smooths between the calculated points.
- **Polynomial Smoothing:** This option applies a fifth degree polynomial equation through the contour data points for a smooth transition between the triangulation faces.

Subdivisional Surfaces: When enabled, adjust the horizontal slider to indicate the degree of triangular subdivisions. This causes each triangle in the triangulation surface model to be subdivided into $(x + 1)^2$ triangles, where $x = \text{Subdivision Generations}$. The mathematically generated sub-triangle vertices are raised or lowered to provide smoother contours. More generations increase the smoothness of the contours but incur increased processing time.

Although this algorithm does not produce "crossing contours," it can result in undesired contours in terrain scenarios such as where graded slopes abruptly transition to nearly horizontal slopes (e.g. the sides and bottom of a detention pond).

Bezier Smoothing Factor: Adjust the horizontal slider to obtain a preview of how much smoothing can be expected at each setting. Sliding the bar to the left results in a lower setting which have less looping or less freedom to curve between contour line points. Likewise, moving the slider to the right results in a setting that increases the looping effect. Note that too much smoothing applied in some situations can result in crossing contours.

Highlight Depression Contours: Depression contours are the lowest elevation closed contours that are surrounded by higher closed contours. This option highlights depression contours by either using a separate linetype or by drawing tick marks along the contour. Use the *Setup* button to get this dialog of settings for depression contours.



Linetype: Set the CAD linetype to use for depression contours.

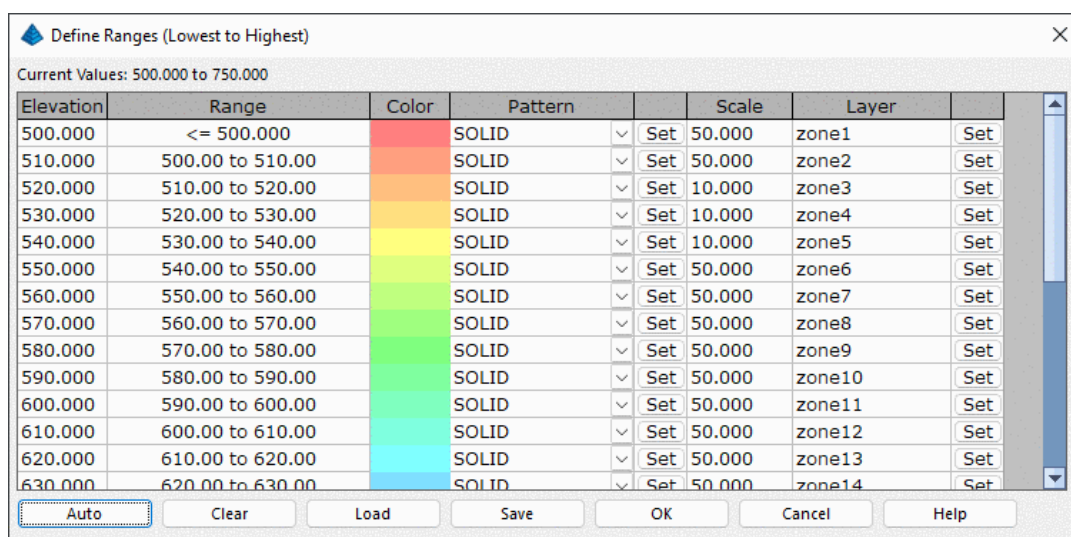
Layer: Specify the layer to which the depression contours are to be drawn.

Tick Size Scaler: Indicate the relative scale factor that should be applied to the depression ticks.

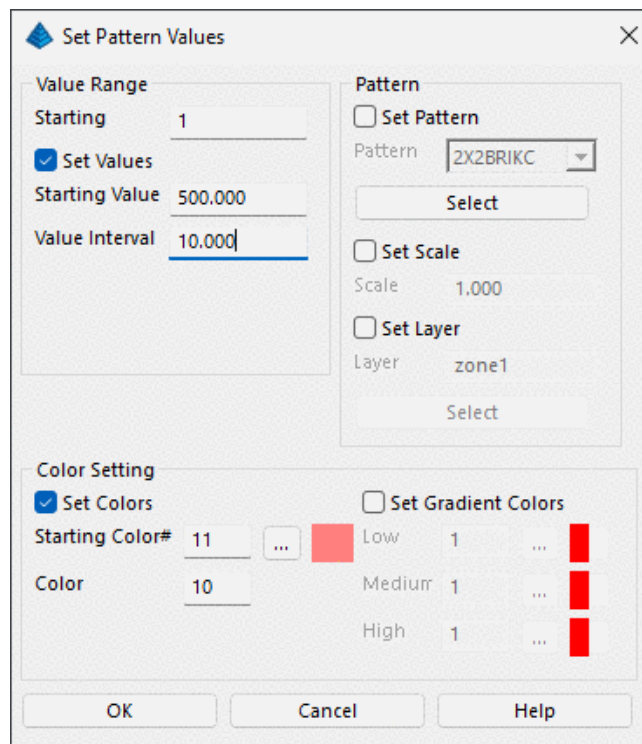
Tick Interval Scaler: Indicate the desired interval scaler which controls the spacing of the depression ticks.

Line Width: Specify the line width to be applied to the depression contours.

Hatch Zones: When enabled, this option will create hatching between the contours based on elevation zones. The following dialog will open allowing the user to specify the hatch type and color for each elevation zone. The entire elevation range of selected data is displayed under Current Values.



Auto: Opens the following dialog, allowing for automatic configuration of the range of elevations in each zone, assigning of colors and hatch patterns, and the scale.



Starting Zone: Sets the zone with which to begin the application of the setting defined in this dialog. For Instance, if the Starting Zone was set to 10, the settings definitions applied here wouldn't affect Zones 1-9, but would start at Zone 10.

Set Values: Enables the Starting Value and Value Interval fields, which allow the user to specify the starting elevation for the given zone and set the zone increment.

Starting Value: Sets the elevation of the beginning zone to define.

Value Interval: Sets the elevation increment for subsequent zones.

Set Pattern: Sets the hatch pattern for the defined zones.

Set Scale: Enables the Scale option.

Scale: Sets the scale for the selected hatch pattern.

Set Colors: Modifies the colors in the table using either Starting Color and Color Increment, or the Gradient option.

Starting Color: Sets the starting color number based on the standard CAD color chart.

Color Increment: Sets the color number to increase for subsequent zones. So if the increment was set to 5, and the starting color was 60, the next color would be 65, 70, and so on.

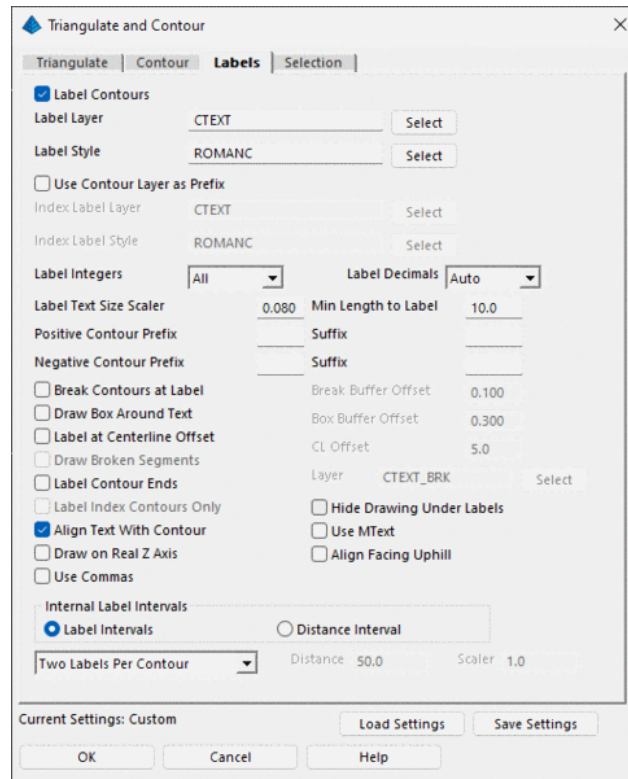
Set Gradient Colors: Sets the colors according to **Low**, **Medium**, and **High** colors. Note that the program will use the Index Colors (total of 256 color options) when setting gradients, which can result in duplicated colors if there are many entries in the table. This option is best used when there are only a small number of hatch zones.

Clear: Clears the all of the Elevation fields in the dialog.

Load: Loads previous settings from a saved .pat file.

Save: Saves the current setting configuration to a .pat file.

Scan DWG/Scan CRD : These buttons report the elevation range which can help with choosing the contour interval.



Label Contours: When enabled, contours will be labeled based on the settings below. If this option is disabled and further contour annotation is desired, utilize the Contour Elevation Label command.

Label Layer: Specify the layer name for intermediate contour labels. To only label index contours, enable the Label Index Contours Only option.

Label Style: Specify the text style that will be used for the contour label text.

Use Contour Layer as Prefix: This option adds the Contour Layer name to the Label Layer name.

Index Label Layer: Specify the layer name for index contour labels.

Index Label Style: Specify the text style that will be used for the index contour label text.

Label Integers controls how many digits to label to the left of the decimal. For example, if all contours are in the 5000's, then setting for three digits would label the 5280 contour as 280.

Label Decimals: Specify the amount of precision to display on the contour labels.

Label Text Size Scaler: Specify a relative text size scale factor to be applied to the label(s).

Min Length to Label: Specify the desired minimum length of contours that should be annotated. In other words, Contours whose length is less than the value will not be labeled.

Positive/Negative Contour Prefix: Indicate a desired string of prefix text (*e.g.* Elev=) that is to precede the positive and/or negative contour elevations, respectively.

Positive/Negative Contour Suffix: Indicate a desired string of suffix text that is to follow the positive and/or negative contour elevations, respectively.

Break Contours at Label: When enabled, the contour lines will be broken and trimmed at the label location for label visibility. As an alternative to physically placing a gap into the contour, consider using the Hide Drawing Under Labels option.

Break Buffer Offset: Specify the offset distance which determines the gap between the end of the trimmed contour line and the beginning or ending of the text.

Draw Box Around Text: When enabled, a rectangle is drawn around the contour elevation labels.

Box Buffer Offset: Specify the offset distance which determines the gap between the box and the beginning or ending of the text.

Label At Centerline Offset: When creating contours and subsequent plan sheets for roads, enable this option to position the labels at a fixed offset from a centerline. The program automatically uses any polylines in the drawing that are tagged as centerlines. To check whether a polyline is a centerline, use the Centerline ID command. To create a centerline polyline from a centerline file, use the Draw Centerline File command.

Draw Broken Segments: When enabled, the segments of contours that have been broken out for label visibility will be redrawn as independent segments. To join these segments back into the contour, use the Join Nearest command.

Layer: Specify the layer that is to receive the newly drawn broken segments.

Label Contour Ends: When enabled, the ends of "open" contours will be labeled.

Label Index Contours Only: When enabled, only the index contours are labeled. This option is active only when Draw Index Contours has been selected in the Contour tab.

Hide Drawing Under Labels: When enabled, a "Wipeout" entity is placed with the annotation label that will create the appearance of trimmed segments at the contour label, even though the contour line is still fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels yet the contour lines themselves remain contiguous. This feature will also hide other entities that are in the immediate vicinity of the contour label.

Align Text with Contour: When enabled, the contour elevation labels will be rotated to align with their respective contour lines.

Use MText: When enabled, contour labels are created using the MText entity type. Otherwise, the standard DText entity type is used.

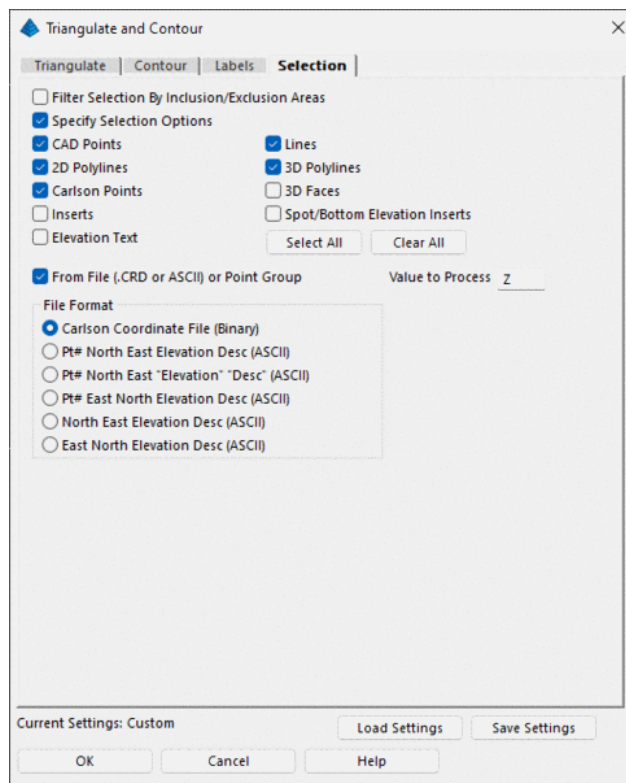
Draw On Real Z Axis: When enabled, the contour labels are placed at the same "Z" (elevation) value of the contour itself. When disabled, the contour labels are placed at a "Z" (elevation) value of 0 (zero).

Align Facing Uphill: When enabled, the contour elevation labels will still be rotated to align with their respective contour lines, but the labels will be placed in such a manner that the top of the text label will always be toward the uphill side of the contour.

Use Commas adds a comma into the labels for the thousands place such as "5,000" instead of "5000".

Internal Label Intervals: Indicate the desired method for contour labels within the contour itself:

- **Label Intervals:** This option will label each contour with a set number of labels.
- **Distance Interval:** This option allows you to specify an interval distance between labels. The distance can be set in drawing units or as a scaler that is multiplied by the drawing horizontal scale.



Filter Selection By Inclusion/Exclusion Areas: This option filters out selected entities from the triangulation that are outside the surface area defined by the inclusion/exclusion perimeter polylines. Otherwise, all the selected entities are used for triangulation and then the triangulation is trimmed at the inclusion/exclusion perimeters. Whether to prompt for inclusion/exclusion perimeters is specified on the Triangulate Tab.

Specify Selection Options: When enabled, indicate the type(s) of entities that are to be used during the triangulation process. This is an excellent method of "filtering out" unwanted entity types or enabling the use of desired entity types.

CAD Points, Lines, 2D Polylines, 3D Polylines, 3DFaces, Elevation Text and Inserts (blocks) are standard CAD entities types.

Carlson Point Inserts refer to Carlson points (such as those placed with the Draw Field to Finish command or which utilize the Carlson "SRVNO*" family of blocks with point number, elevation, and description attributes).

Spot/Bottom Elevation Inserts include text entities that start with 'X'.

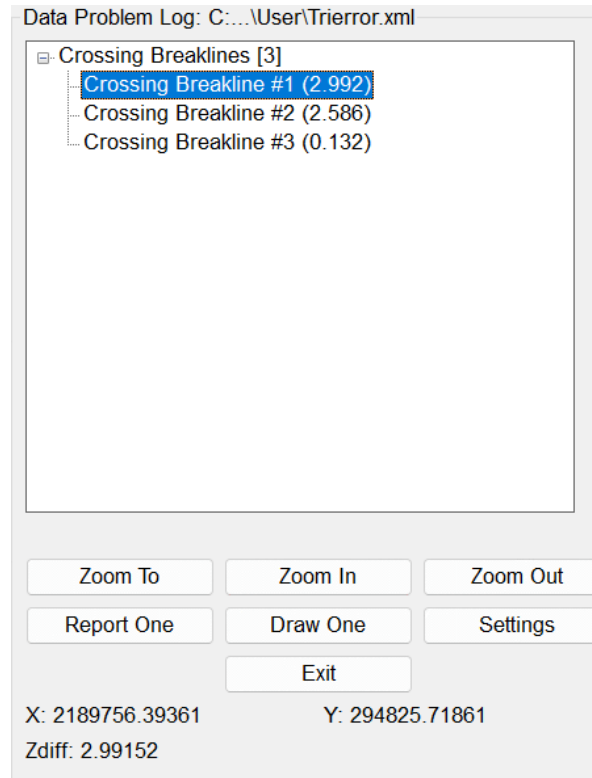
From File: When enabled, allows you to triangulate from the points in an external coordinate (.CRD) or ASCII file. This option also provides access to the use of Point Groups as a data source.

An *Error Log* is generated if the triangulation routine finds vertical conflicts between breaklines or other surface entities and displays the conflicts in a "docked dialog box." Three types of conflicts are reported (each conflict type is listed into its own category):

1. Crossing Breaklines - Indicates the common X,Y location of two breaklines that do not share a common "Z" elevation.
2. Vertical Edges - Indicates that two entities or vertexes of differing elevations have the same x-y location, thus forming a vertical plane to another point.
3. Breakline T-Intersections - Indicates that a third entity is abutting another entity, but the second entity doesn't have a vertex at the point of intersection.

Click the "+" sign beside a category to display the individual conflicts within that category and click the "-" sign

to collapse the list. When a line item error is selected, a highlighted arrow is temporarily placed in the drawing to indicate the exact location of the specific conflict. Zoom functionality allows the user to more closely inspect the specific problem area, and if needed a marker can be drawn or a report generated for an individual conflict or conflicts.

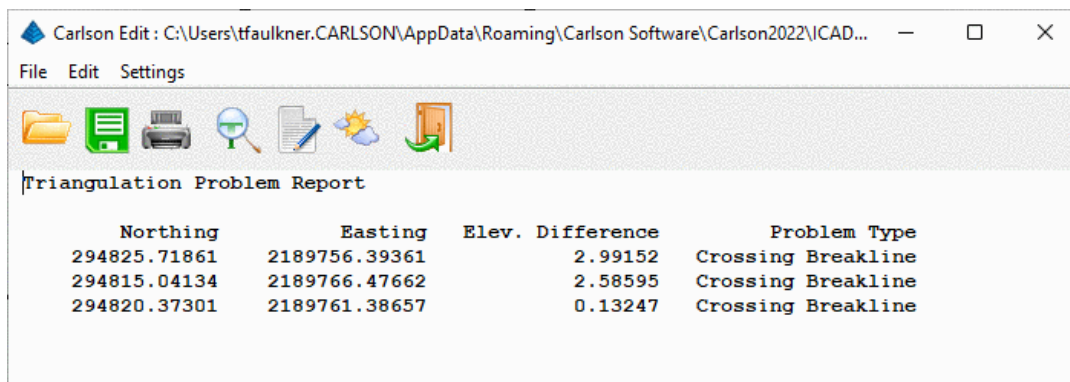


Zoom To: Centers the display on the location of the error without affecting the zoom resolution.

Zoom In: Increases the ability to see detail.

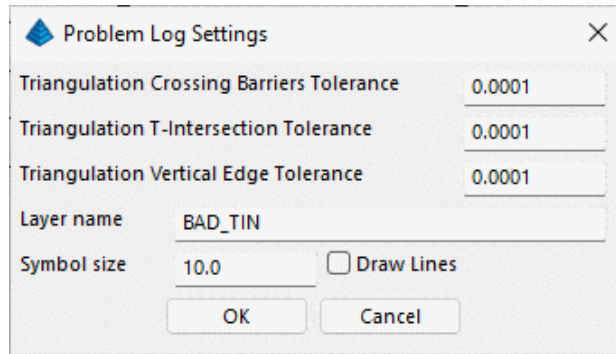
Zoom Out: Decreases the ability to see detail.

Report One/All: This option toggles between "One" and "All" depending whether a single line item conflict or an entire category is selected from the error log. An error report is generated listing the x-y position and the elevation difference of the entities in conflict.



Draw One/All: This option toggles between One and All depending whether a single conflict or a category is selected from the list. This option draws an "X" symbol at each selected conflict.

Settings: Indicate the desired configuration settings for the error log:



Tolerances: Indicate the lowest elevation difference threshold that should be reported for Crossing Breaklines, Vertical Edges and Breakline T-Intersections, respectively.

Layer Name: Specify the layer name for the "X" entities drawn with *Draw One/All* option. This also sets the layer name for the *Draw Lines* option.

In the case of crossing polylines, *Draw Lines* will trace over the polylines responsible for the conflict.

Symbol Size: Specify the size of the "X" symbol that is drawn to delineate the selected errors. This will determine the actual size of the symbol in the drawing. This value is not multiplied by the horizontal drawing scale.

Note:

- When selecting Inclusion/Exclusion polylines, you may select any number of Inclusion polylines and any number of Exclusion polylines. Selecting multiple Inclusion polylines results in "islands" of terrain data within a given TIN file.
- If *Triangulate & Contour* reports zero points found and fails to do anything when you're using Carlson points, then those points are probably located at zero elevation. To fix this problem, make sure that Carlson Point Inserts is toggled on in the Selection tab. This will enable *Triangulate & Contour* to read the elevation from the elevation attribute of the point.
- For those experienced in programming, Carlson offers a DTM API (Application Programming Interface) which provides functions that can be used to access and manipulate information stored within a DTM file.
- In surface situations where a series of rectangular grid cells are desired, explore the Make 3D Grid File command.

Prompts

The following are the most often encountered prompts:

Select the Inclusion perimeter polylines or ENTER for none.

Select entities: Select the desired closed polylines that form the bounding inclusion area(s) of the surface model and press Enter when complete.

Select the Exclusion perimeter polylines or ENTER for none.

Select entities: Select the desired closed polylines that form the regions(s) of the surface model where triangulation should not occur and press Enter when complete.

Select the points and breaklines to Triangulate.

Select entities: Select the desired entities from CAD using standard CAD selection methods and press Enter when complete.

Pull-down Menu Location(s): Surface (Survey, Civil, Hydro, Construction, Field, Natural Regrade), Takeoff > Surface Tools

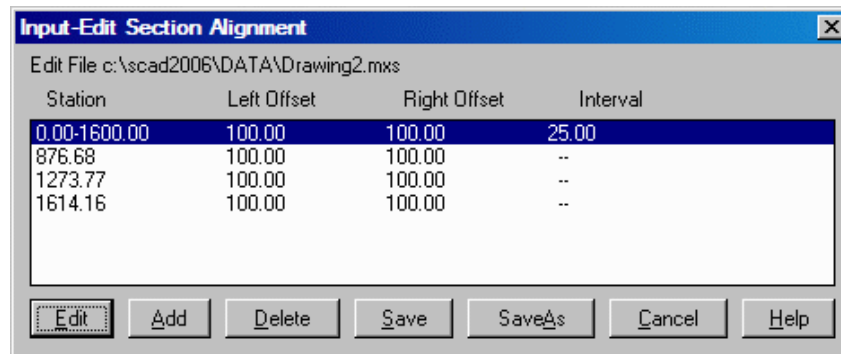
Keyboard Command: tri

Prerequisite: 3D entities in the drawing (defined by the Selection Tab) and/or an external point file

Section Alignment Editor

This command will create or append to a section alignment file which is stored as a Multiple Cross Sections (.MXS) file. This file contains the coordinates that define the center and endpoints of section lines and is a requirement of many section commands such as *Sections from Surface Entities* and *Sections to 3D Polyline*. The section alignment defines the stations along a centerline and how far left and right to create cross sections. This routine starts by asking for a new or existing .MXS file name. Then the centerline is specified by either by choosing a centerline file (.CL file) or selecting a polyline that represents the centerline. Next, the program prompts for the starting station of the centerline. If this is a new section alignment, the Make MXS File Settings dialog appears.

The Section Alignment Editor dialog lists all the section stations and offsets in the alignment of an existing .MXS file.



Dialog if using an existing .MXS file

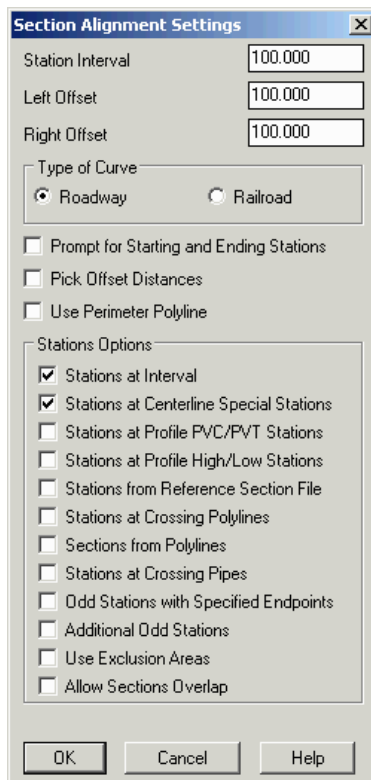
Edit: Allows you to edit the currently highlighted row.

Add: Allows you to add more sections by displaying the Make MXS File Settings dialog (shown below).

Delete: Deletes the currently highlighted row.

Save: Saves the MXS file, exits this dialog and draws the section alignment on the screen using temporary vectors (yellow for left offsets, magenta for right offsets). Any viewport change such as *Redraw* or *Zoom* will cause these vectors to disappear. The draw the section lines with Line entities, use the *Draw Section Alignment* command.

SaveAs: Saves a new MXS file with a user-specified name.



Dialog used for a new section alignment

Station Interval: Enter the station interval for sections.

Right Offset: Enter the width for the sections, right of the centerline. Not available if Pick Offset Distances is checked.

Left Offset: Enter the width for the sections, left of the centerline. Not available if Pick Offset Distances is checked.

Type of Curve: Specify either Roadway or Railroad curve to account for the differences in stationing curves.

Prompt for Starting and Ending Stations: This option allows you to specify the range of stations to process. Otherwise the program will use the full station range of the centerline.

Pick Offset Distances: Allows you to specify the offsets by using the distance between two picked points in the drawing.

Use Perimeter Polyline: Allows you to specify a closed polyline that will be used as the limit of the cross sections. The offsets will be contained within this closed polyline.

Station By Another Reference Centerline: Prompts for a separate CL to use for stationing. The program sets for station to store for the section alignment by locating the position of the original station on the section alignment centerline and then finding the station of this position along the reference centerline.

Stations at Interval: Creates cross sections at the specified interval such as every 25 feet. If the Prompt for Starting and Ending Stations is on, then the program will apply the station interval to the user-specified range of stations. Otherwise the station interval is used along the entire centerline.

Stations at Centerline Special Stations: Creates cross sections at every transition point in the centerline such as the PC, PT, spiral points and end points.

Stations at Profile PVC/PVT Stations: Creates cross sections at profile vertical curve transitions stations. When active, the program will prompt for the profile to process.

Stations at Profile High/Low Stations: Creates cross sections at profile vertical curve high and/or low stations. When active, the program will prompt for the profile to process.

Stations from Reference Section File: Creates cross sections at stations contained in the reference section file. When active, the program will prompt for the section file to process.

Stations at Crossing Polylines: Allows you to select polylines that cross the centerline and creates cross sections at the intersections of these polylines with the centerline.

Sections from Polyline: This method prompts to select polylines that define the section alignments. This method is handy when the section alignments have multiple bends such as for HEC-RAS.

Stations at Crossing Pipes: This method creates cross sections at the stations where pipes intersect.

Odd Stations with Specified Endpoints: Creates cross sections at stations that are entered or at picked points along the centerline. This option also allows you to pick the left and right offset points which do not have to be perpendicular to the centerline.

Additional Odd Stations: Creates cross sections at the specified stations but the offsets are always perpendicular to the centerline with the user-defined default offset distances.

Use Exclusion Areas: This option prompts for selecting closed polylines to use as exclusion areas which are areas to skip for the section surface. The stations and offsets for the exclusion areas are stored in the section alignment file. Then routines like Calculate Section Volumes and Draw Sections will skip over these areas and not calculate volumes or draw section lines in these areas.

Allow Sections Overlap: On the inside of a centerline curve, section lines will cross when the section length is longer than the radius. This option controls whether to shorten the section lines on the inside of the curve or to keep the regular length and let the section lines cross.

Prompts

Specify an MXS file dialog Choose new or existing.

Polyline should have been drawn in direction of increasing stations.

CL File/<Select polyline that represents centerline>: *pick centerline*

Enter Beginning Station of Alignment <0.00>: *press Enter*

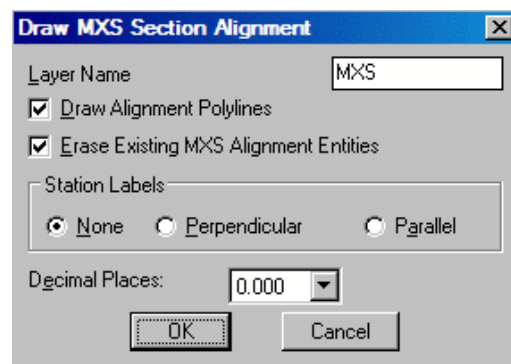
Pulldown Menu Location: Sections

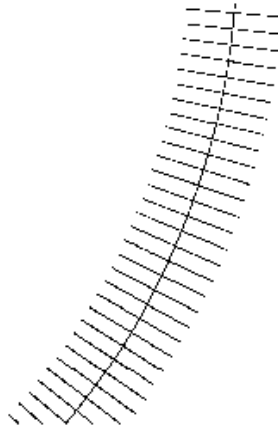
Keyboard Command: editmxs

Prerequisite: A polyline centerline or a centerline .CL file

Draw Section Alignment

This command will draw the location of the cross sections contained in an existing .MXS file. The cross sections stations can also be labeled Perpendicular or Parallel. The main purpose of this routine is to allow you to graphically view the location of the cross sections.





A drawn .MXS file along with the centerline

Pulldown Menu Location: Sections

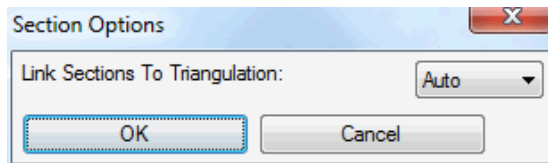
Keyboard Command: drawmxs

Prerequisite: An existing .MXS file

Sections from Grid or Triangulation Surface

This command creates a cross section file (.SCT file) from a surface model that is defined by a 3D rectangular grid file (.GRD file) or a triangulation file (.FLT, .TIN). The grid file can be created in the Civil Design module with the *Make 3D Grid File* routine. The triangulation file can be created with the Write Triangulation File option in the *Triangulate & Contour* command. This command also requires a Section Alignment (.MXS) file to define the alignment and stations of the sections. The number of section points created is displayed at the end of the routine.

When using a triangulation file, there is an option for whether to link the sections to the triangulation. With the link, the section file will get updated in case the triangulation file is updated. When the link option is set to Auto, the update is done automatically. When the link option is set to Prompt, the program will prompt with a dialog for whether to update the sections when a triangulation change is detected.



Prompts

Choose Grid or Triangulation File to process *choose existing .GRD, .FLT, or .TIN file*

Choose MXS File to Process *choose existing .MXS*

Choose Section file to write *enter new file name*

Found 1410 section points.

Pulldown Menu Location: Sections

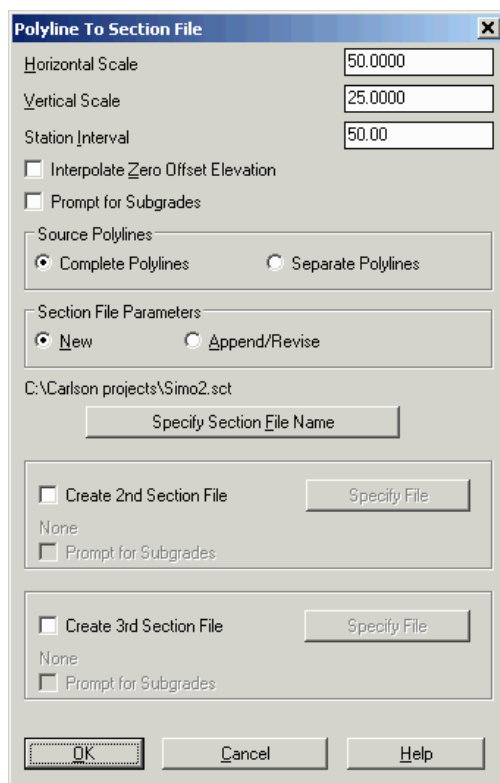
Keyboard Command: sctgrid

Prerequisite: Grid (.GRD) or triangulation (.FLT or .TIN) file, and a cross sections .MXS alignment file

Sections from Polylines

This command allows the user to select a polyline that represents a section in cross section view and writes it to a .SCT file. This can be useful for revising sections or for defining a new one. For example, let's say you have extracted sections from a surface model of the existing ground on a site, and have plotted them using the *Draw Section File* command. Next, the *Polyline by Slope Ratio* command is used to draw the proposed or final grade sections. Now use this command to send the sections to a Section file and compute the earthworks using the *Calculate Sections*

Volume command. After selecting the command, the Polyline to Section File dialog appears.



The first time this command is selected, the output Section file is set to the same name as the current drawing. Select the Specify Section File Name button to specify a different name. The Station Interval edit box allows you to specify the amount that the default station number will be incremented as the station prompt shown below appears. The Interpolate Zero Offset toggle, if on, causes the program to output the elevation of the zero offset to the output .SCT file. A second and a third section file can be specified to process three sections at a time for each station. This allows you to handle both existing and final grades at once. The Prompt for Subgrades option will prompt for selecting subgrade polylines after the surface polyline for that section. For Source Polyline, the Complete Polyline method works with a single polyline that completely define the section for the station. The Separate Polyline methods allows you to select multiple polylines per section in case the section polylines are broken up into separate polyline entities. After selecting the OK button, the prompts below appear.

Prompts

Exit/Pick text/ <Station <0.00>>: *press Enter*

Exit/Pick text/ <Starting elevation of grid <100.00>> 440 This supplies the drawing coordinate to translate the grid from.

[int on] Pick point at starting elevation and zero offset of section ([Enter] for none): *press Enter*

Select station 0.00 1st section polyline: *select a polyline*

Select station 0.00 2nd section polyline: *select another polyline*

Exit/Pick text/ <Station <0.00>>: *E*

Pulldown Menu Location: Sections

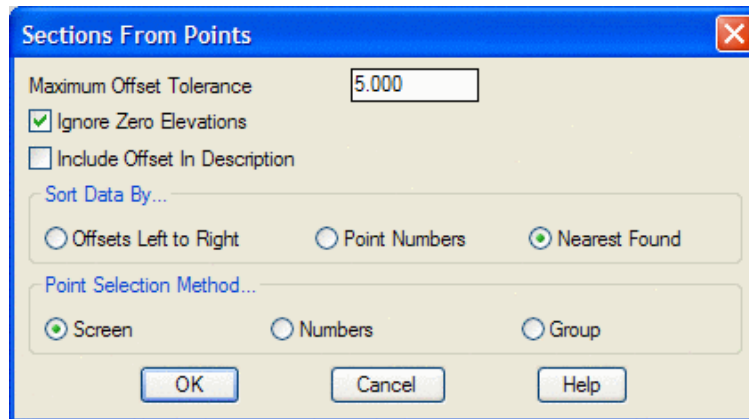
Keyboard Command: scfpl

Prerequisite: Plot the section or profile to write to the .SCT file.

Sections from Points

This command creates an .SCT file from Carlson points in the drawing. An .MXS file is needed to define the centerline and the stations of the cross sections. The offsets for the cross section points are derived from the perpendicular distance between the centerline and the Carlson points. The cross section elevations come directly from the elevations of the points. In order to be included in a cross section, a Carlson point must be within the offset tolerance distance of the cross section line.

The order that the points within the Offset Tolerance at each station are used will of course determine the shape of the cross section. There are three ways for the collected points to be sorted. The Offsets Left to Right option sorts by the distance of each point from the CL. The Point Numbers option ignores that data, and instead sorts the points by their numbers. The Nearest Found option ignores both distance from the CL and point numbers and instead checks the horizontal and vertical proximity of the points to each other and sorts them based on this data. A powerful application of this method would be a survey of a tunnel where the points collected at each station were collected in a random order.



Prompts

Choose MXS File to Process *select file*

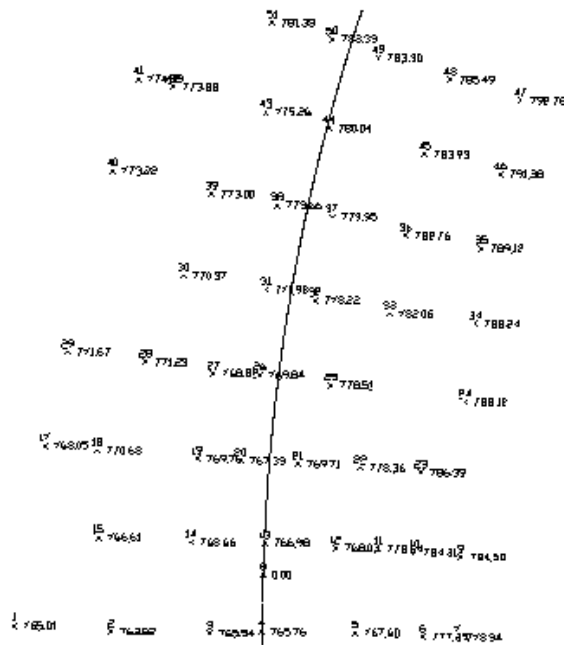
Choose SCT file to Append/Write *select file*

Enter the maximum offset tolerance <1.0>: *press Enter*

Ignore Zero Elevations (<Yes>/No)? *press Enter* This option filters out all Carlson points that have a zero elevation.

Select points along the sections.

Select objects: *pick the Carlson points*



Carlson points for use in creating Section file

Pulldown Menu Location: Sections

Keyboard Command: scpts

Prerequisite: Carlson points and an .MXS file

Section File Editor

This program can be used to enter or edit data stored in a section file (.SCT file), including a real-time graphic window in the Edit mode. The section data consists of stations, offsets, elevations and descriptions. This command also has utilities for translating the offsets and elevations, deleting stations from the file, intersecting the outslopes of one section file with another, combining multiple occurrences of the same station and sorting the stations, offsets and elevations.

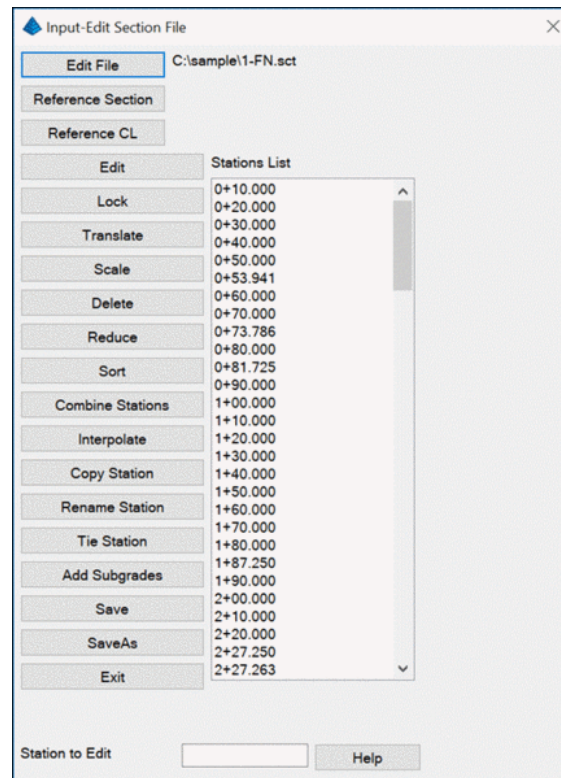
While editing the section file, a second section file can be used as reference. To choose this file, pick the **Reference Section** button. For example, when editing the proposed section file, you can reference and view the ground section file as the second file. Besides showing the reference section in the graphic preview, the program also reports the end areas while editing a section station. Also, the reference section can be used to tie to the catch point. Another option is to set a **Reference CL** which is used to adjust the stations when the centerline has station equations.

The program begins by prompting for a New or Existing section .SCT file to process. The Section file to process dialog appears, allowing you to specify the file that you want to operate on. Use the New option to create a new file. Use the Existing option to edit the offsets and elevations for station/sections that you have already created, or append new stations to a file. The program defaults to a section file with the same name as the drawing or a name that you specified using another section command. You also can choose a 2nd existing .SCT file to reference. After specifying the file name(s), the program displays any stations currently in the file, in the Stations List of the Section File dialog box.


Alternately, when sections are drawn in the drawing, you can double-click on a section polyline to launch Section File for the .SCT file associated with the section polyline.

If you specified a new file, the Stations List box will be blank. To edit and display the offset and elevation data at a station, you can double click on the station in the Stations List box, or input the station in the Station to Edit edit box at the bottom of the dialog. To add a station to a new file or existing file, you must enter the station in the Station to

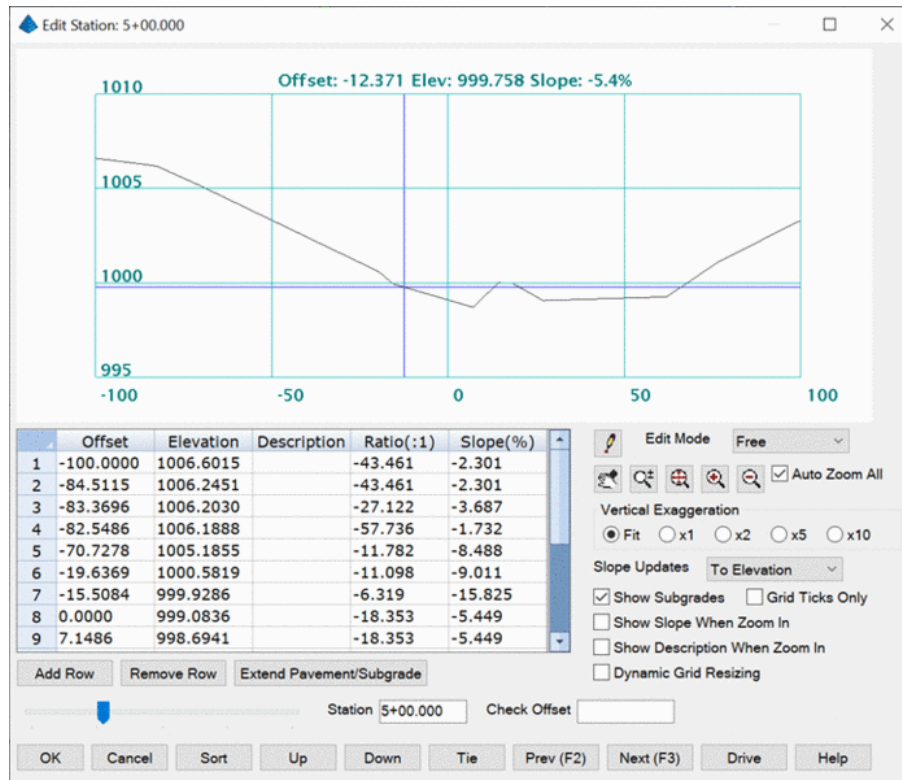
Edit edit box. Stations will present in accordance with the Section-Profile settings in Configure under the Settings pulldown menu (eg. 10+00, 1+000, 1000).



Edit: Opens the Edit Station dialog which shows a graphic of the section on top, a list of the offset-elevation points in the middle, and the function buttons on the bottom. To add an offset point, type in the offset, elevation and optional description in the spreadsheet. Left offsets are entered as negative numbers. You can enter the slope or ratio from the last point and the program will calculate the elevation. To edit an offset point, highlight the point from the list and then edit the values in the Offset, Elev and Desc columns. The highlighted point will be marked by an X in the graphic screen. The Sort button will sort the list of offsets from lowest to highest, left to right. It is recommended that you Sort offsets before doing the Tie command, so that the left-most and right-most offsets appear first and last in the offset list. The Up button will move the highlighted offset point up in the list. Likewise the Down button moves the highlighted offset point down in the list. Prev (F2) and Next (F3) buttons move through the stations and allow you to review and edit stations in forward or reverse order. The scroll bar can also be used to quickly move through stations and then zero in with Prev (F2) or Next (F3). When you edit a slope value in the spreadsheet, then the Slope Updates setting controls which elevation value to update for the section segment.

The section data can be edited directly in the spreadsheet or graphically by picking the Edit Point button with the pencil icon.  To edit graphically, use the click-n-drag method. Start by picking the Edit Point button and then pick the section point to edit in the graphic preview and hold the mouse button down and then drag the mouse to the new position and then release the mouse button. The **Edit Mode** setting governs the click-n-drag operation. The **Free** mode allows the section point to be moved anywhere. There **Hold Slope** modes maintain the slope and moves the point along these slopes. The **Hold Offset** allows changing the section point elevation only. The **Hold Elevation** allows changing the point offset only.

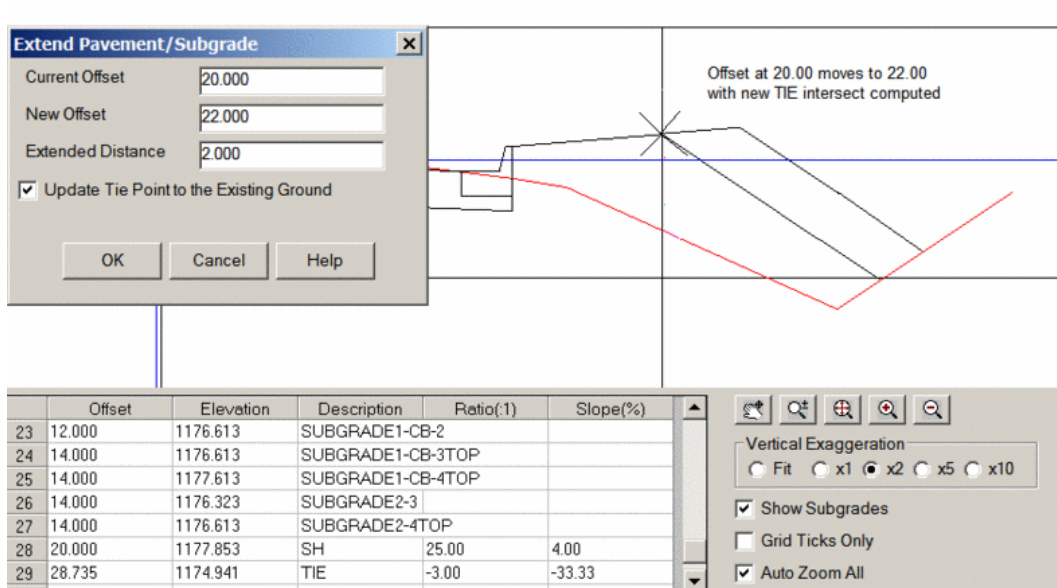
When a Reference Section is set in the main dialog, the Edit Station spreadsheet has a column for Depth. This Depth is the elevation different to the reference section. You can edit the depths to change the elevations for the section.



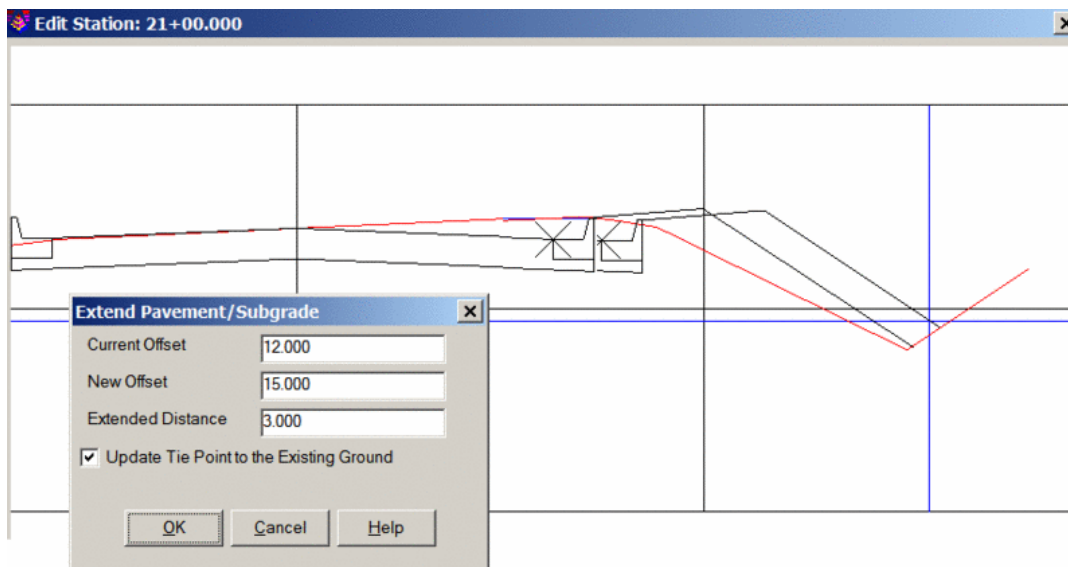
The **Add Row** button inserts an offset line above the currently highlighted row. The **Remove Row** button erases the highlighted offset and elevation from the list. After inputting or editing press the **OK** button to return to the Stations List dialog and keep any changes you have made. Select the **Cancel** button if you want to cancel changes made to the current station.

A real-time report of offset-elevation-slope now displays in the top of the graphic as you move the cursor across the section in the graphic window. The screen defaults to zoom mode where holding down the right-mouse button zooms in and out. You can also switch to pan mode. There are buttons for zoom extents, zoom in and zoom out. If your mouse has a scroll button, you can hold it down to pan and scroll it to zoom in and out. You can also set the **Vertical Exaggeration** ranging from 1X to 10X and including "Fit". **Show Subgrades** has the ability to tie a subgrade into the surface. **Grid Ticks Only** just shows the left and bottom axis lines of the grid with grid tick marks along the axes. With **Auto Zoom All** turned off, you can hold the same view position as you click **Next** and **Previous** and move through the list of stations. The **Description** is used in Draw Section File command to label under the station value and used in the Section Report command. The **Check Offset** field calculates an elevation based on an entered offset.

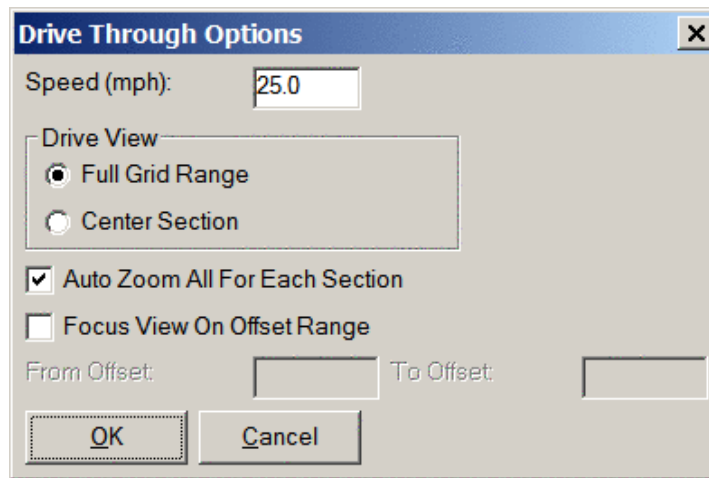
Extend Pavement/Subgrade allows you to move a surface point and shift, in parallel, the associated subgrades and tie points. One application, shown below, is to extend a shoulder point and re-computer the TIE point, all in one clean operation:



Another application of Extend Pavement/Subgrade is to move the curb position and all associated subgrades. The "inside" curb point is at 12.00 units from centerline. If the pavement is extended from 12 to 15 at this station, use of this feature will extend the subgrades, maintain all slopes and re-compute the TIE point, as shown below:

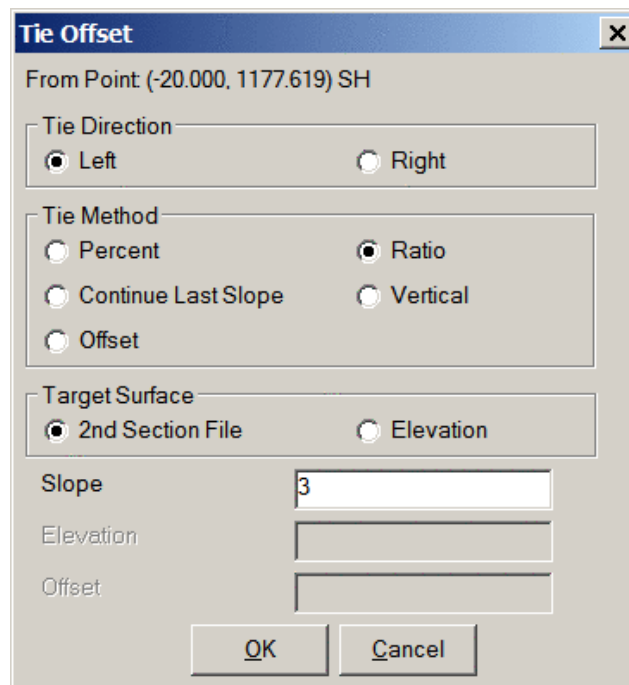


Drive (Edit Station): This function scrolls through the sections at the rate of speed specified by the user in the Speed window. The Drive View options determine whether the sections are displayed using the full width of the graphic window or centered in the window. The combination of Full Grid Range and Auto Zoom All allows the sections to rise and fall with the centerline elevations, as if you were driving an actual road. With Auto Zoom All off, and Full Grid Range on, the grid itself moves up and down at the current position of the first section, as you drive. Focus View On Offset Range allows the user to set the left and right viewing limits of the sections. Section data beyond the specified limits is not displayed.



Elevation Field (Edit Station): Equations (+, -, *, /) can be entered to calculate or adjust an elevation. For instance, to subtract 1.25' from elevation 1926.18, simply enter 1926.18-1.25 and press enter. The new elevation will be calculated and displayed in the viewer window.

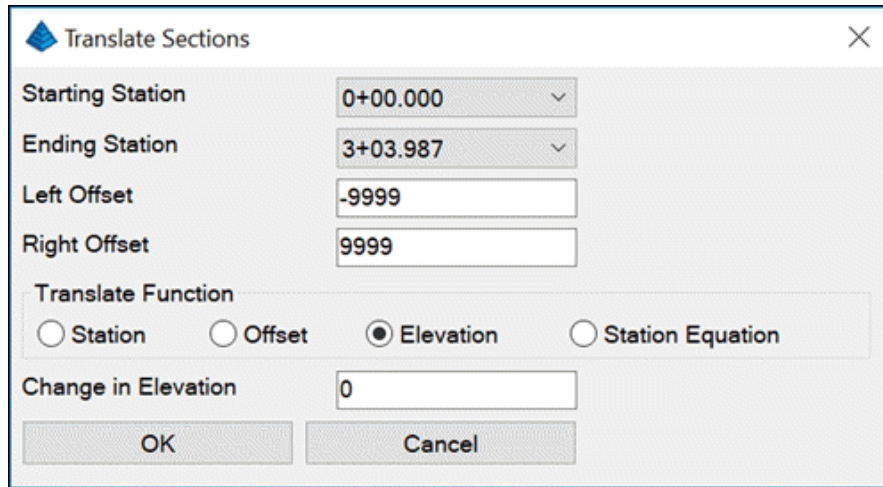
Tie (Edit Station): The Tie button allows you to tie the left and right surface points of the 1st section file into the 2nd section file. It is used for classic outslope intersects from final grade to existing grade. The dialog layout includes an option to tie the section to a specified elevation, in addition to a surface (second section file). A left or right tie direction can also be selected. If a point has been tied in from SH for shoulder at offset -20 at 3:1, a new offset with the description "TIE" is created. If you try another outslope such as 4:1 from the same SH shoulder point, a new "TIE" point is created and the old TIE point is removed automatically.



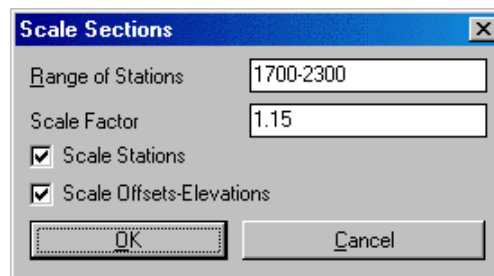
Lock: This function will tag the section file as locked so that no routine can automatically overwrite this file. If a routine attempts to overwrite this section file, the program will stop, report that the file is locked and prompt whether to override the lock.

Translate: Allows you to add or subtract a distance from the offsets to adjust or shift the centerline. You can also adjust the elevations up or down. When using this option, you can choose the range of stations to operate on (starting and ending stations) and the values to adjust the offsets and elevations. If, for example, you want to shift the centerline, but not the elevations, enter the plus or minus amount you want to translate, and when prompted

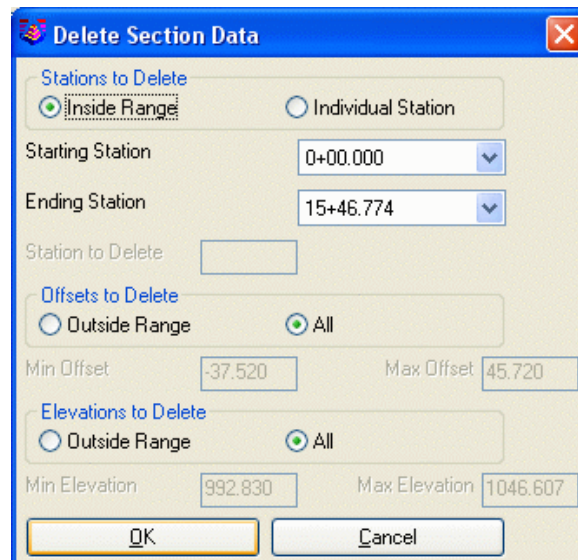
for the elevation enter zero. The Station Equation function removes the station equation adjustments to the station values in order to set the station values to the true centerline distances. To use the Station Equation function, a Reference CL must be set on the main dialog. For elevations, there is an option to use a profile to define the amount of elevation change as the difference between the profile elevation and the section elevation at the zero offset.



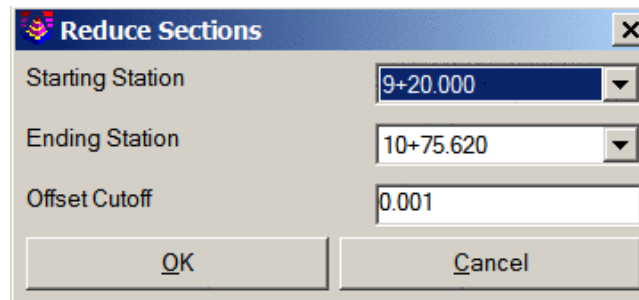
Scale: Allows you to scale the station, offsets and/or elevations by the specified scale factor. This function can be used to convert between English and metric units.



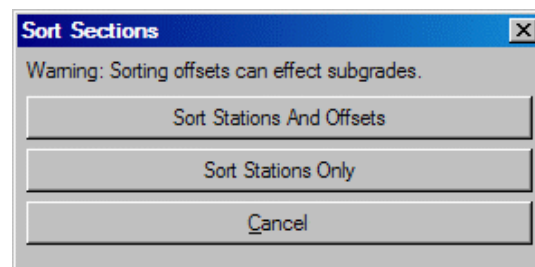
Delete: Allows you to remove a station or range of stations from the Stations List. You can delete a range of stations or an individual station. Also there are options to delete all the data for the selected stations or filter to delete only data that is outside an offset or elevation range. Since the station editor data is stored in memory, if you accidentally delete a range, Quit the editor without saving the stations to disk. Then recall the original file.



Reduce: Allows you to remove offsets from a range of stations by removing vertices in the offsets that are virtually in a straight line. Using an offset cutoff, meaning no offset and elevation moves more than the entered amount (eg. 0.01), excessive numbers of vertices can be eliminated. The command is similar to Reduce Vertices when applied to the plan view.

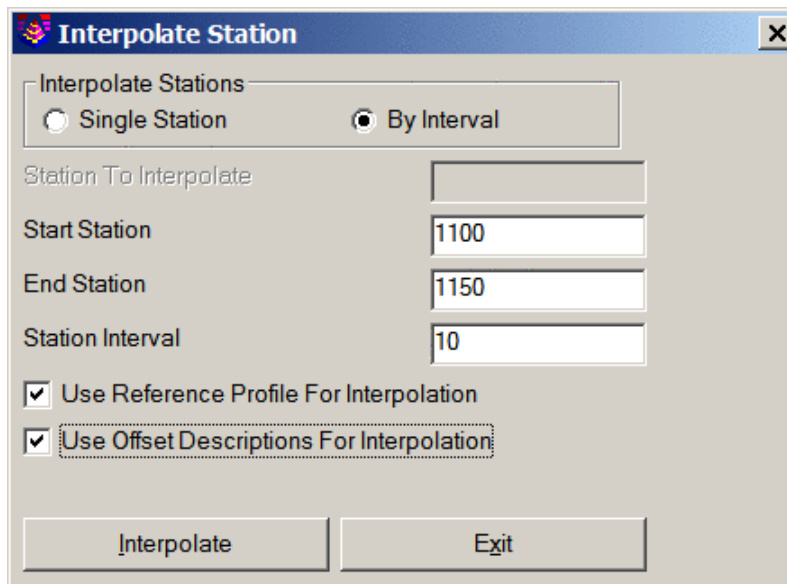


Sort: Allows you to sort the station numbers into ascending order, and sort the offsets and elevations in the individual station records (offsets are sorted from left to right). When sections are derived from the *Sections from Surface Entities* command they are already sorted, but when sections are digitized or input manually they occur in the order that you digitized them. So, for proper plotting and earthworks, you may want to run the Sort option before processing.



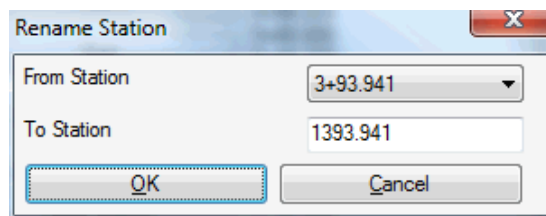
Combine Stations: Used to bring together in one record slot multiple occurrences of the same station number. This can occur when using the Digitize Sections (XSec) command and the section that you are digitizing has match/break lines which forces you to digitize the station in two or more parts.

Interpolate: Allows you to add or overwrite a station between two stations or projecting forward from two stations. You can choose to interpolate a single station or an interval of stations. Specify the two known stations in the Start Station and End Station edit boxes, as well as the interval if using the interval method. The program will do straight line, mathematical interpolations, adding offsets to the interpolated stations to match the totality of offsets in the starting and ending stations. However, if the offsets have descriptions, you can choose to interpolate by description and the program will interpolate by description (eg. EP at 12 on Station 1100 and EP at 15 at station 1150 would lead to EP at 12.6 at 1110). There is also an option to reference a profile, so if station 1100 and 1150 were on either side of a high point at 1125, the interpolated offsets would respect the profile as well as the starting and ending station. Use of this command is often critical to creating accurate digital terrain models of sites for machine control. Select the OK button to execute the function with the current settings or select the Cancel button to abort the process.

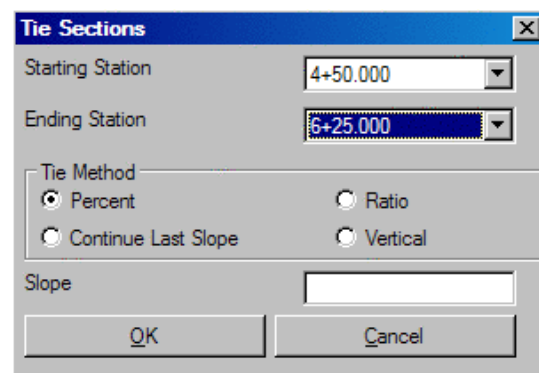


Copy Station: Allows you to copy a station that already exists to a new or existing station number. Choose the existing From Station using the edit pulldown box, then enter the new station number in the To Station edit box. Select the OK button to execute the function with the current settings, or select the Cancel button to abort the process.

Rename Station: Allows you to change the value of a station. In the dialog, select the existing station from the list and enter in the new station value.

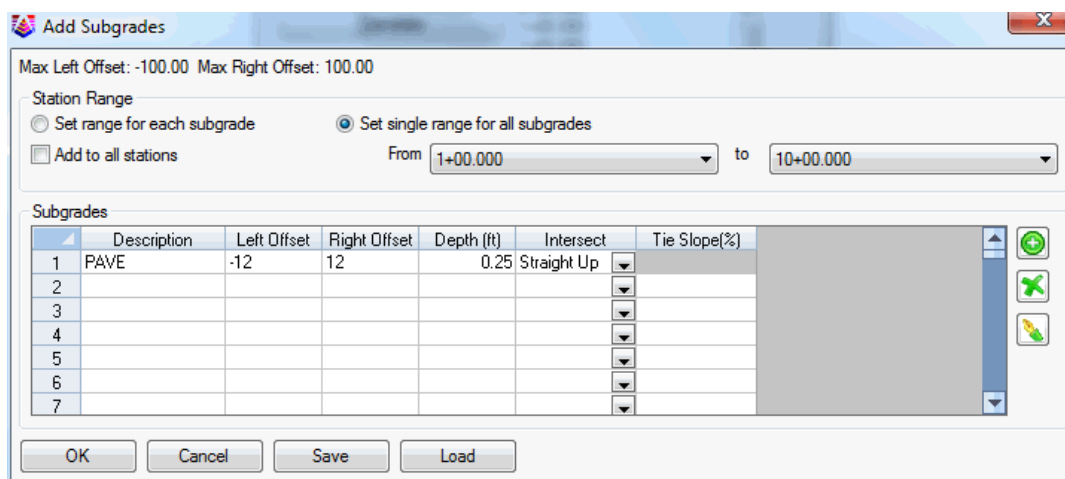


Tie Station: Allows you to tie the outslopes into the reference second section file. This routine first brings up a dialog to specify the range of stations to process. It includes a line to set the slope to tie with. The program will start from the left most offset and use this slope to find the intersection with the reference section file. Then the intersection from the right most offset is calculated with this slope. These intersection points are the tie points. The slope can be defined by percent, ratio, continue the last slope, and vertical.



Add Subgrades: Adds subgrades to the sections with specified depths and offsets. You can add multiple subgrades at a time by filling in the spreadsheet. Each row of the spreadsheet is for a separate subgrade. Each subgrade definition takes a description, left and right offsets, depth and intersection method of either straight up or at a specified slope. The subgrades are added by referencing the existing surface elevation and dropping down the specified depth. The center of the subgrade always drops down vertically. The outside of the subgrade ties in by the specified intersection method. The station range to add the subgrades can be the same of all the subgrades or

specified separately for each subgrade.



Save: Saves the currently loaded section file.

SaveAs: Allows you to save the currently loaded section file as a different file.

Exit: Allows you to exit from the section editor and return to the drawing editor. The program will warn you to save to a file if you have made changes.

Pulldown Menu Location: Sections

Keyboard Command: scted

Prerequisite: None

Draw Section File

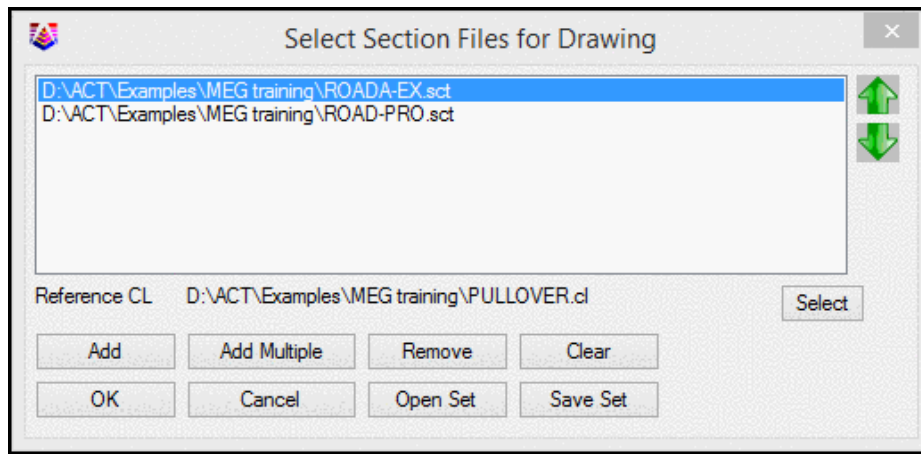
This command generates plots of cross-section data which can be used to further iterate the corridor design or used for construction documentation. The Section files drawn with this command can be created by several methods including the Section File Editor, Digitize Sections, any commands under the Create Sections from... menu, Process Road Design or Road Network commands.

For metric-based projects, please refer to the Drawing Metric Section Sheets section of this document.

The Draw Section File routine will call two primary dialog boxes:

- The first is the Section Files for Drawing dialog box that allows you to specify the Section files (.sct) to be drawn and some general sheet and layer settings.
- The second is the Draw Section File dialog box that allows you to specify various scale, layout and labeling settings.

If the *Type of Plot* option in the Draw Section File dialog box is set to "Sheet," a third Section File Sheet Drafting Parameters dialog will be displayed which provides detailed sheet layout settings.



Add: Specify an unlimited number of Section (.sct) files to plot

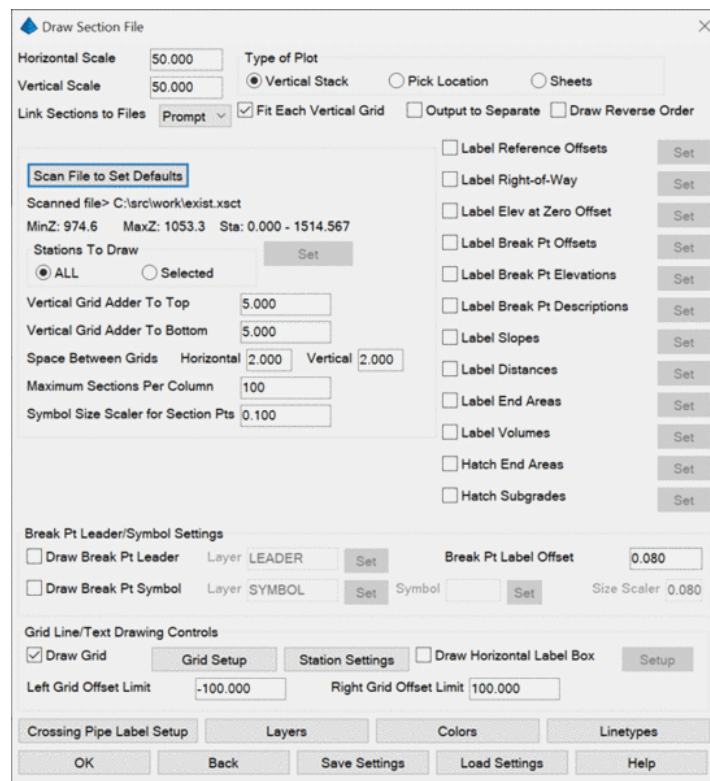
Add Multiple: Add multiple files to plot with a single selection

Remove: Highlight and remove any section file from the selection set

Clear: Clear the selection set (remove all files).

Open Set: Open a preset selection of cross sections(.XST file)

Save Set: Save the current selection set as an .XST file



After specifying the Section Files (.sct) the Draw Section File dialog box opens:

Horizontal Scale: Specify the horizontal scale.

Vertical Scale: Specify the vertical scale. The vertical scale relative to the horizontal scale determines the vertical exaggeration factor.

Link Sections to Files: This setting controls the linkage of the plotted sections to the actual section (.sct) file(s), determining how changes to the file affect the plotted sections.

Type of Plot: Specify the type of Plot you wish to create.

- **Vertical Stack:** Stacks all the cross sections in a vertical row or rows. The cross sections will be drawn in model space and will automatically stack with regard to the parameters set.
- **Pick Location:** allows the user to manually select a location in model space for each individual section.
- **Sheets:** draws the cross sections in either model space or paper space with regard to the settings in the Section File Sheet Drafting Parameters dialog box. When you pick OK on this main dialog, the Sheet Drafting Parameters dialog box appears allowing you to specify all the settings for sheet plotting.

Section File Sheet Drafting Parameters

Choose Space: Indicate whether sheets are to be drawn to **Paper** Space (also known as a Layout) or to **Model** Space. When drawing to Model Space, the Display In Paper Space option will draw the sections in model space and then create layouts with viewports to show the sections.

Layout Name: Indicate the name of the layout to which the first sheet should be drawn.

Use Template Layout: This option allows you to use a layout that exists in the current drawing as the template for layouts created by the command. This option only works for paper space sheet creation.

Add Layout Name To File For Drawing Output: When the Output To Separate Drawing option is active from the

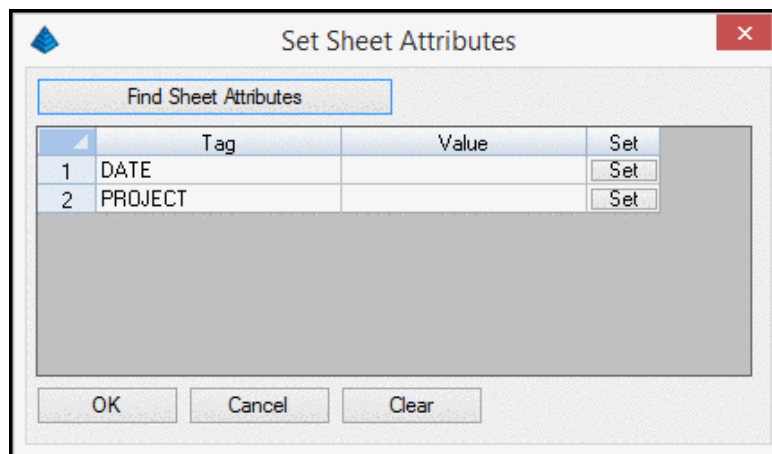
main settings dialog, this option creates separate drawing files for each sheet by appending the layout name to the drawing name.

Tile Sheets: Enabling this option places all sheets in the specified Layout Name. The result is a vertical stack of sheets in the layout. Disabling this option allows additional layouts to be created each containing one sheet. As additional layouts are created, the name of each successive layout is incremented by a value of 1.

Plot at 1:1: Enabling this option draws the sections so that one unit horizontally in the section is equivalent to one plotted unit. The ratio of the Horizontal Scale:Vertical Scale determines the amount of vertical exaggeration.

Block Name: Specify the drawing name that will be inserted for each sheet. The default is SCTSHT1 which is included with Carlson Software and is located in the `%AppData%\Carlson Software\...\Sup\` folder. You can use this or use a sheet block of your own design. The block should be drawn at a 1:1 scale since the program inserts it using the Horizontal Scale setting from the previous dialog. Click the **Set** button to browse/navigate to an alternate drawing file.

Set Sheet Attributes: For grid sheet block names that utilize attributes (useful for items such as sheet numbers, drawn date, drawn by, job name, *etc*), use this command to provide attribute values that will be placed for each sheet block:



Find Sheet Attributes: This routine will scan the Block Name for any attribute definitions and return them to the dialog box so values can be established for each attribute.

Starting Page #: Indicate the starting page number to be applied to the plots through the use of the Set Sheet Attributes command.

Scan Block for Width/Height: Use this routine to scan the specified Block Name for its width and height. These values are populated into the Sheet Width and Sheet Height controls.

Sheet Grid Interval: Indicate the spacing between the grid lines in the sheet block. The routine will not draw the grid lines and uses this information to control the placement of each section onto the sheet.

Vertical Space Between Sheets: Indicate the amount of space that should be placed between sheets when the Tile Sheets option is enabled.

Rows of Sections

Per Sheet: Specify the maximum number of sections that can be stacked on top of each other on a sheet.

Space Between: Specify how much space will be placed between the top of the last section plotted and the bottom of the next section. For U.S. Customary based units, a value of 1 would be a good starting value.

Columns of Sections

Per Sheet: Specify how many columns of sections can be placed on each sheet.

Space Between: Specify the distance between the left edge of one section column edge and the right edge of the next column. This will generally be the area where elevation labels and station circle annotation will be placed. For U.S. Customary based units, a value of 2 would be a good starting value.

Label Grid Zero Offset: Enable this toggle if the zero offset location of each section should be labeled on each section.

Column Order: Controls the station order of the sections for going top to bottom, or bottom to top.

Offset for 1st Section

Horizontal Offset: Specify how far from left edge of the sheet the first section will be placed on to the section sheet. The block SCTSHT1 has a 1" left margin.

Vertical Offset: Specify how far from bottom edge of the sheet the first section will be placed on to the section sheet. The block SCTSHT1 has a 1/2" bottom margin.

Preview: This button allows you to get an approximate idea of what the initial sheet will look like based on the current settings.

Back: This button allows you to return focus to the main dialog and make changes to any previous settings or cancel the routine.

Save Settings: This button allows you to save all the parameters settings to a file so you can easily recall them for another project.

- **Off:** A linkage between the SCT file and the graphical section entities is not formed; you will need to manually re-create section sheets after section design changes.
- **Prompt:** You will be asked whether or not to update the plotted sections when the underlying SCT file is changed.
- **Auto:** The plotted sections will automatically update when the underlying SCT file changes.

Fit Each Vertical Grid: When checked, the grid bottom elevation and grid height are set automatically and you may specify values to add to the top and bottom of each grid (see Vertical Grid Adder to Top and Vertical Grid Adder to Bottom). When not checked, you specify the elevation of the grid bottom and the grid height through the Grid Bottom Elevation and Grid Vertical Height controls, respectively.

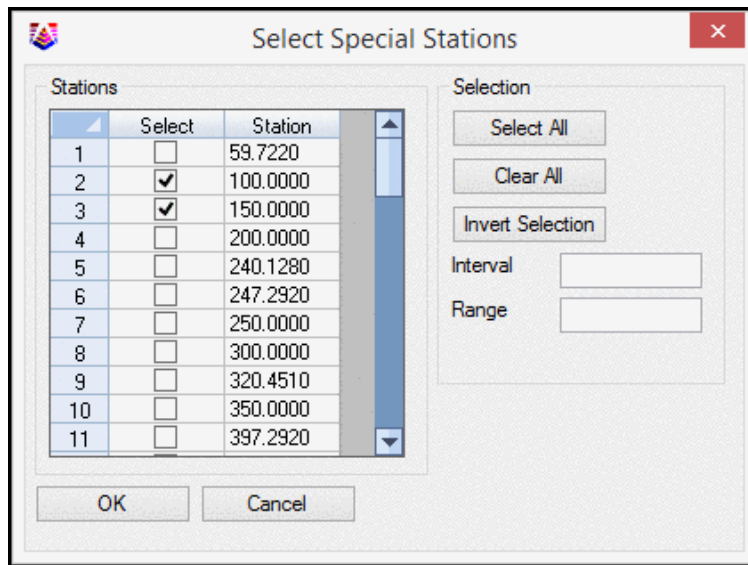
Output to Separate Drawing: When checked, this option will prompt for a New drawing name and location into which all cross sections will be drawn. When using the Vertical Stack method, the program will prompt for whether to output all the sections to the same drawing or create a separate drawing for each station.

Draw Reverse Order: When checked, this option will draw the cross sections in the order of the highest numbered station to the lowest.

Scan File to Set Defaults: This button allows the program to set the minimum and maximum parameters. If you choose this option, the program will automatically set the range of stations, vertical spacing distance, right and left grid distances and starting/datum elevation. This option writes a file called "sectsort.tmp" that is read and used to set the defaults the next time you use the program. Therefore, if you are selecting a different .SCT file to plot you should use this option to update the .TMP file.

Stations to Draw: Select eight All or Selected to specify the range of stations from the file which will be drawn.

- **All:** When selected the program will draw all of the stations in the file.
- **Selected:** This option will allow the user to specify exactly which stations will be drawn via the Set dialog box.



Interval: Specify the interval of stations to draw. For example, perhaps you sampled every 25 feet with the Sections from Surface Model command for more accurate quantities but only want to plot 50 foot stations. ALL is the default value for this field.

Range: Specify a range of stations to include

Vertical Grid Adder to Top: Specify the distance that will be added to the highest elevation of the section for the sheets and pick location options. This option is only available when Fit Each Vertical Grid is checked ON.

Vertical Grid Adder to Bottom: Specify the distance that will be subtracted from the lowest elevation of the section for the sheets and pick location options. This option is only available when Fit Each Vertical Grid is checked ON.

Grid Bottom Elevation: Specify actual bottom elevation for each section grid. This option is only available when Fit Each Vertical Grid is checked OFF.

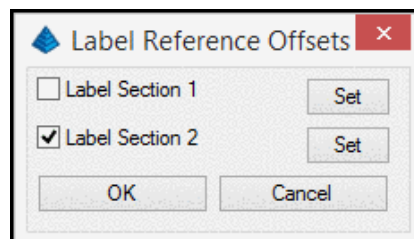
Vertical Grid Height: Specify actual grid height for each section grid. This option is only available when Fit Each Vertical Grid is checked OFF.

Space Between Grids: Specify the **Horizontal** and **Vertical** distance between the sections when they are drawn when the Vertical Stack option is specified.

Maximum Sections Per Column: Sets the maximum number of sections allowed per column when the Vertical Stack option is specified.

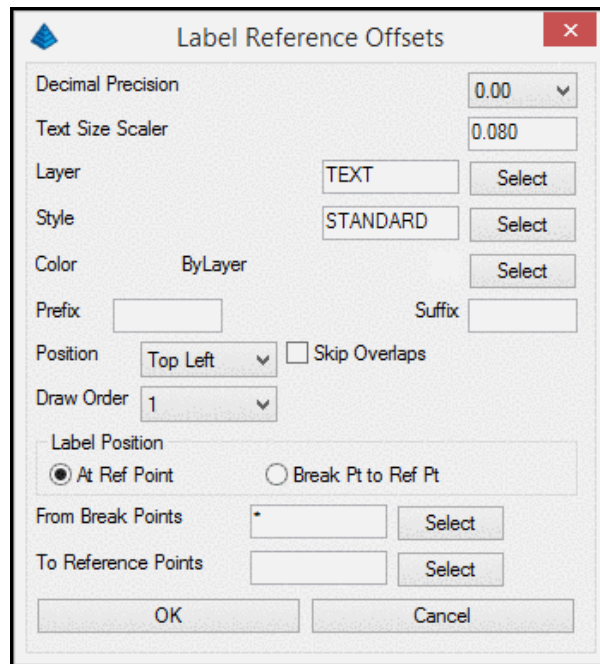
Symbol Size Scaler for Section Pts: Sets the scale as a multiplier of the overall drawing scale for the section points to be drawn.

Label Reference Offsets: When enabled, the offset from selected break points of one section file relative to the position(s) of selected points from another section file can be labeled onto the plots.



Specify which section(s) to draw

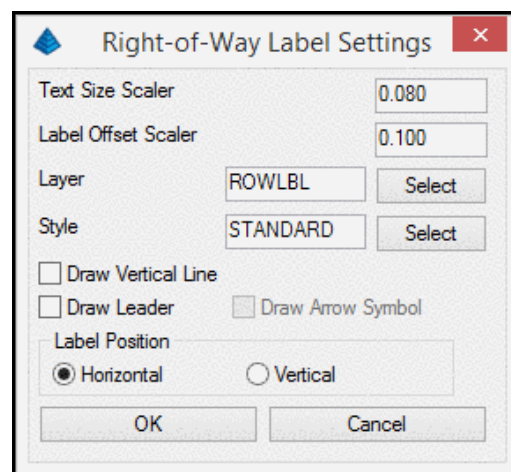
Set: Opens the dialog box for to set the display precision, text size scaler, layer, text style, color, prefix, suffix, position and draw order for these labels.



From Break Points: allows a user based filter to specify break points

To Reference Points: allows the user to control which offsets to label by specifying the reference points from a template (TPL) file.

Label Right of Way: When enabled, this option will label Right of Way points as defined using the Section Points from Right of Way command. Press the **Set** button to the right of this toggle to set the text size and label offset scalers, layer and text style settings.

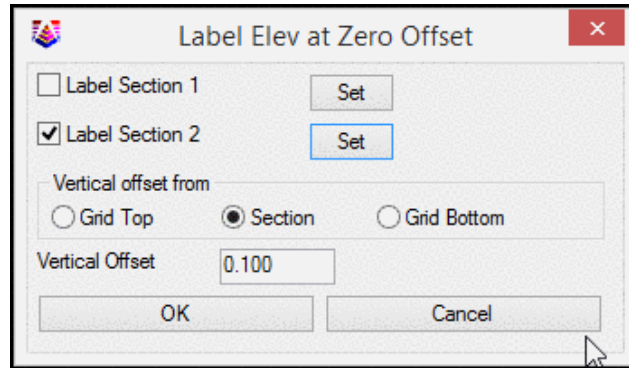


Draw Vertical Line: Places a vertical line, from top to bottom, through the Right-of-Way point.

Draw Leader/Draw Arrow Symbol: When enabled, a short vertical line is drawn, with or without, the arrowhead through the Right-of-Way point.

Label Position: Indicate the desired orientation of the "ROW" text label.

Label Elev at Zero Offset: Will label the section elevation at offset zero.

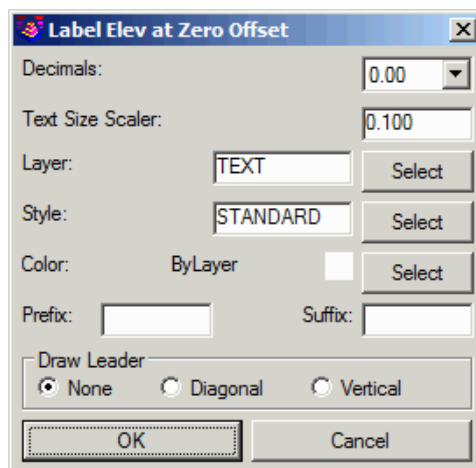


Specify which section(s) to draw

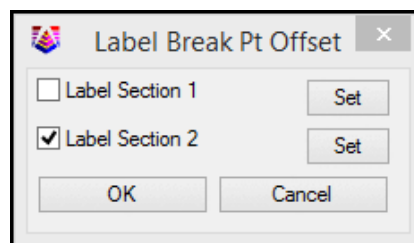
Vertical Offset from: sets the location of the Label relative to the grid or section.

Vertical Offset: controls the offset distance for the label.

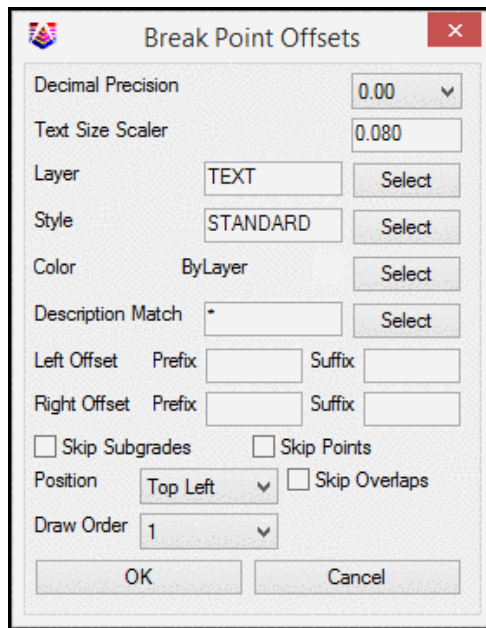
Set: opens the dialog box for to set the display precision, text size scaler, prefix, suffix, color and layer for these labels. The Draw Leader option can be set to None, Diagonal or Vertical.



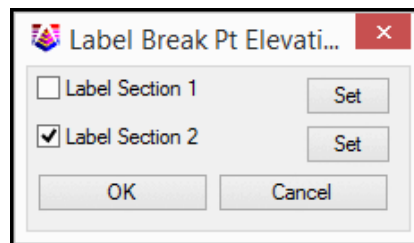
Label Break Pt Offsets: Select the Section(s) to be included in the labeling parameters



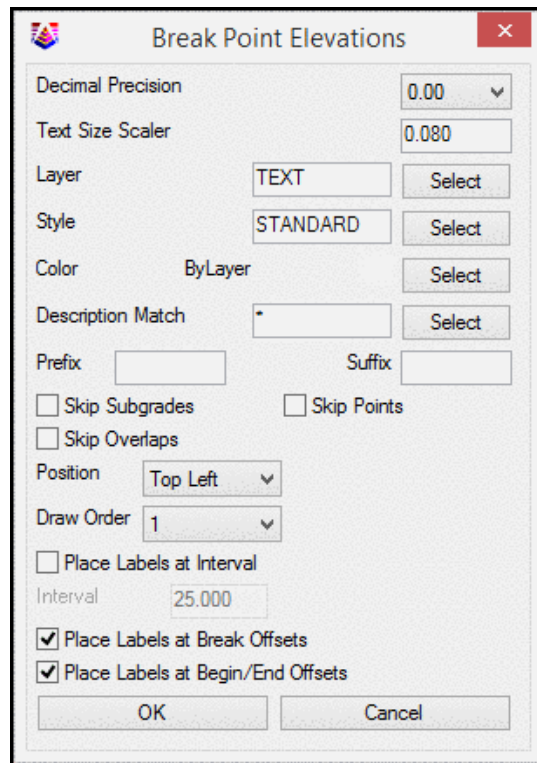
Set: Opens the dialog box for to set the display precision, text size scaler, layer, text style, color, prefix, suffix, and position for these labels. In addition the label position and draw order can be selected.



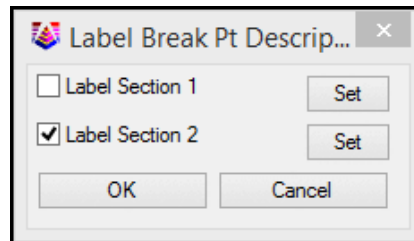
Label Break Pt Elevations: Select the Section(s) to be included in the labeling parameters.



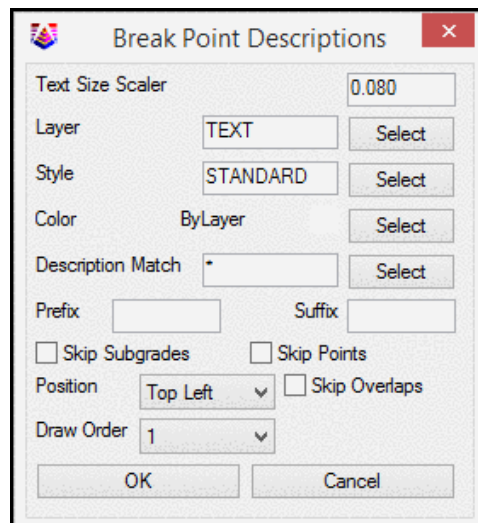
Set: Opens the dialog box to set display precision, text size scaler, Layer, style, color, prefix, suffix, position and draw order for these labels. The Description Match is a way to filter which section points to label. There are also options to toggle whether or not to place labels at the break offsets or Beginning and ending offsets.



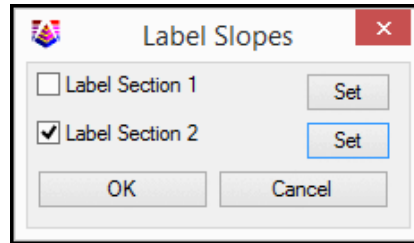
Label Break Pt Descriptions: Select the Section(s) to be included in the labeling parameters.



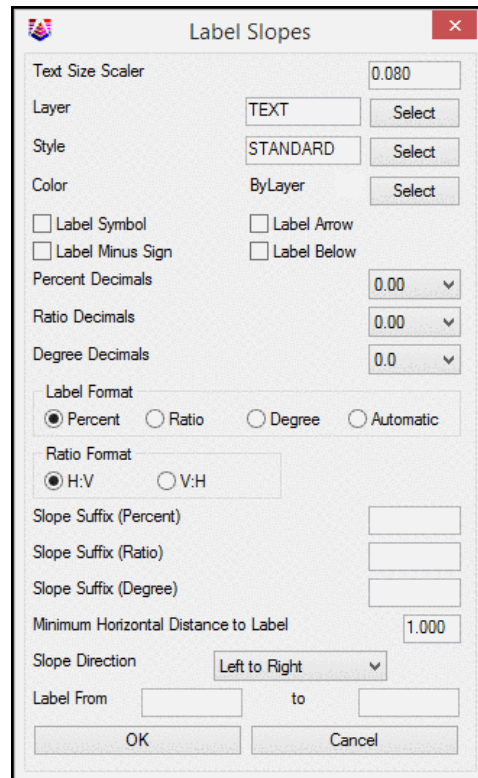
Set: opens the dialog box to set the text size scaler, layer, style, color, prefix, suffix, position and draw order for these labels. The Description Match is a way to filter which section points to label.



Label Slopes: Select the Section(s) to be included in the labeling parameters.



Set: Opens the dialog box to set the text size scaler, layer, style and color for these labels.



Label Symbol: places the % symbol after the slope label

Label Arrow: draws a slope direction arrow next to the label.

Label Minus Sign: places a - in front of negative slope values.

Label Below: places the label below the section line.

Label Format: controls which format the slopes will be labeled in.

Ration Format: Sets either *horizontal to vertical* or *vertical to horizontal* format

Slope suffix (Percent): creates a suffix to the slope label when using the Percent Label Format.

Slope suffix (Ratio): creates a suffix to the slope label when using the Ratio Label Format.

Slope suffix (Degree): creates a suffix to the slope label when using the Degree Label Format.

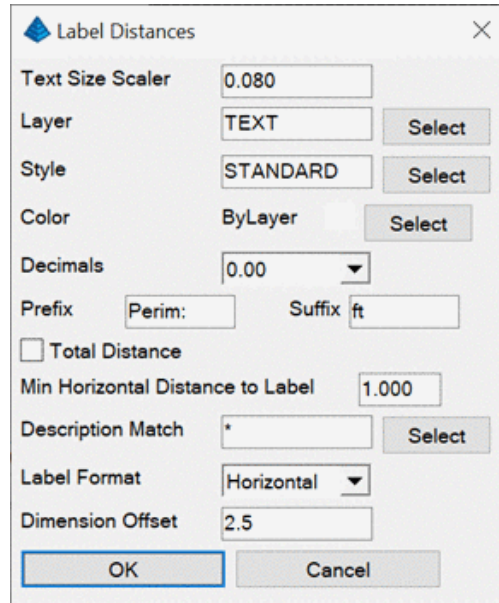
Minimum Horizontal Distance to Label: sets the minimum value of a segment length to label.

Slope Direction: can either be set *left to right* or *right to left*.

Label From: specifies the portions of the section to label a slope. for example, you could specify to only label the

slope between the SW (sidewalk) and SH (shoulder) ID points as defined in the Template file (.tpl) that was used to generate your Section file (.sct).

Label Distances: This option labels the horizontal distances between offset points. There are separate Label Distances settings for each section file. On the first dialog, toggle which section file you want to label distances. Choose the Set button to control the settings for that station. The settings dialog has settings for the distance label size, layer, style, color, decimals and suffix. The **Total Distance** option chooses between labeling just the total distance of the section or individual section segments. The **Description Match** is a way to control which section offset points to label. The **Min Horizontal Distance To Label** setting is a way to filter out short section segments and only label the longer segments. The **Label Format** sets whether to label the distances as a CAD dimension or as regular text rotated by the section slope or horizontal. The **Dimension Offset** controls the vertical distance between the section segment and the label.



Label End Areas: Will label cut and fill end areas on each section. The end areas are either between two sections (existing and design) or with a single section that creates a closed loop like for a tunnel. This dialog box sets the Decimal Precision, text size scaler, layer, style and color of the labels.

The dialog box 'End Area Settings' contains the following fields and options:

- Decimal Precision: 0.0
- Text Size Scaler: 0.080
- Layer: ENDAREA (with Select button)
- Style: STANDARD (with Select button)
- Color: ByLayer (with Select button)
- Cut: Prefix Cut, Suffix
- Fill: Prefix Fill, Suffix
- Skip Zero Labels
- Auto-Center On Section
- Offset: Horizontal 0.000, Vertical 0.000
- Vertical offset from: Top, Bottom
- Use Table
- Title Header: End Areas, Append Station
- Name Header: Material, Value Header: SF
- Include Elevation
- Ground Header: Ground, Final Header: Final
- Include Subgrades
- Subgrade 1: Asphalt, Subgrade 2: Gravel
- Subgrade 3: , Subgrade 4:
- Subgrade 5: , Subgrade 6:
- Buttons: OK, Cancel

Use Table: this option will create a table of the cut/fill values on each section.

Title Header: labels the table with this header. **Append Station** adds the station to the title header.

Cut and Fill label prefix and suffix: allows user defined prefix and suffix added to the label

Auto-Center On Section: this option will center the table or label next to the drawn cross section.

Skip Zero Labels: this option will not label cut or fill values of zero.

Include Elevation: this option will include the centerline elevations in the table. The Ground and Final Headers are user defined.

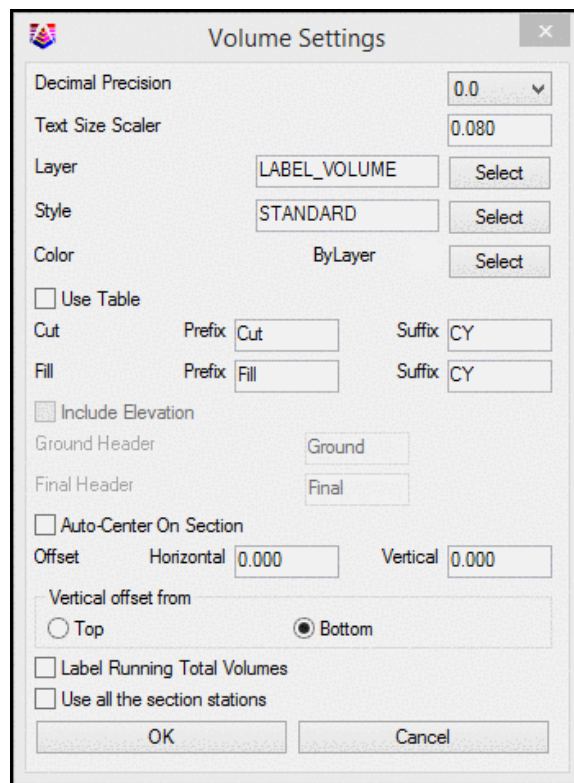
Include Subgrades: labels the end areas of the subgrades with settings for the names of up to 6 subgrades.

Offset Horizontal and Vertical: manually controls the placement of the table or label.

Vertical offset from: specifies where the offset dimensions are to be referenced to.

End Areas	
Material	SF
Cut	387.5
Fill	0.0
Asphalt	2.4
Gravel	8.0
Stone	6.7

Label Volumes: Will label cut and fill volumes on each section. The volumes are measured between the current station and previous station. There are settings to control the format and placement of the labels. This dialog box sets the Decimal Precision, text size scaler, layer, style and color of the labels.



Use Table: option will create a table of the cut/fill values on each section.

Cut and Fill label prefix and suffix: allows user defined prefix and suffix added to the label.

Include elevation: when enabled, this option will include the centerline elevations in the table. The Ground and Final Headers are user defined.

Auto-Center On Section: this option will center the table or label next to the drawn cross section.

Offset Horizontal and Vertical: manually controls the placement of the table or label.

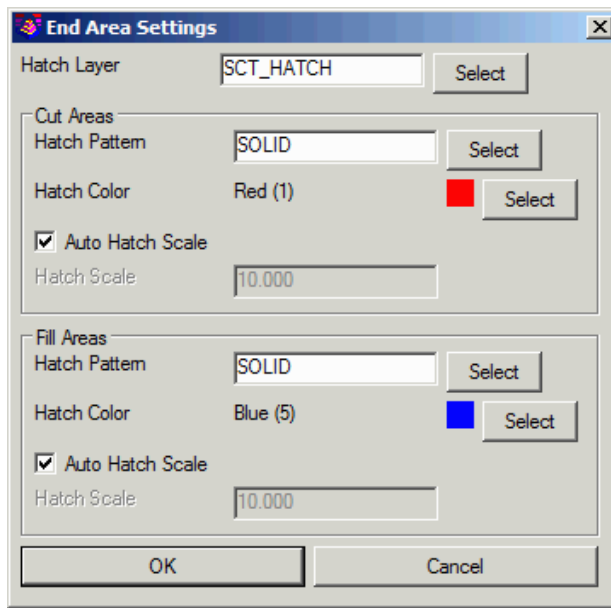
Vertical offset from: specifies where the offset dimensions are to be referenced to.

Label Running total Volumes: will label the accumulative total volumes with each station.

Use all the section stations: chooses between using all the section stations to calculate volumes or only the stations being drawn.

Hatch Subgrades: This option hatches the subgrade areas of design section using the section points with descriptions starting with SUBGRADE. There are settings for up to four different hatch patterns.

Hatch End Areas: This option hatches the cut/fill areas between the first and second section files. The program treats the first section as existing and the second as design for determining cut verses fill. There are separate hatch pattern, color and scale settings for cut and fill.



Draw Break Pt Leader: Enable this option to include a leader with the Label Break Pt Offsets, Label Break Pt Elevations or Label Break Pt Descriptions options. Click the **Set** button to specify the desired layer for the leader.

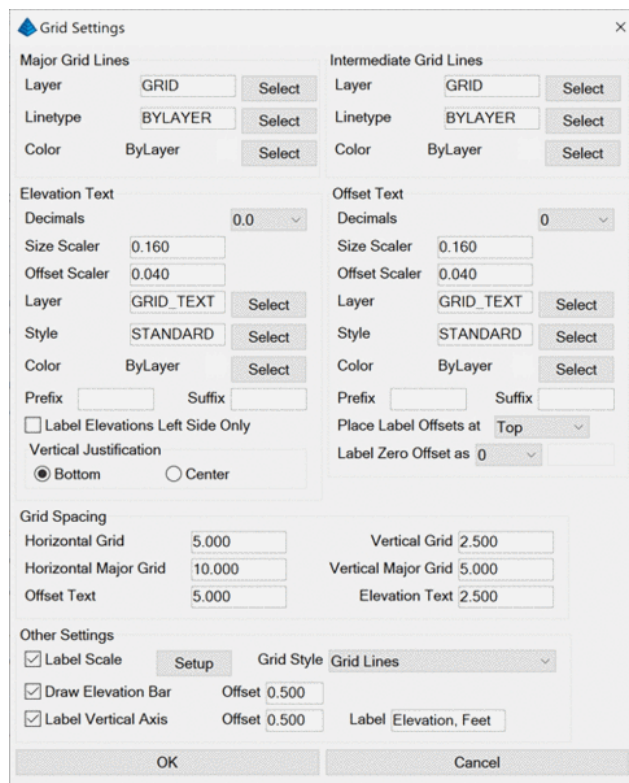
Note:

- When redrawing sections, the program retains any custom edits to label and leader positions.

Draw Break Pt Symbol: Enable this option to include a symbol with the Label Break Pt Offsets, Label Break Pt Elevations or Label Break Pt Descriptions options. Click the **Set** button (to the immediate right of the Layer control) to specify the desired layer for the symbol. Click the **Set** button (to the immediate right of the Symbol control) to specify the desired symbol and indicated the desired Size Scalar.

Break Pt Label Offset: Indicate the desired offset amount from the surface break point to its label.

Draw Grid: When enabled each cross section is drawn on a grid. The **Grid Setup** is used to customize how the grid will be drawn.



Main Grid Lines: Sets the Layer, Linetype and Color for the Major Grid Lines as set below

Intermediate Grid Lines: Sets the Layer, Linetype and Color for the Intermediate Grid Lines. (Those not defined as major)

Elevation Text: this section contains controls for setting the elevation Precision, text Size Scaler, Offset Scaler, Layer, text Style, and color, prefix and suffix for the elevation labels.

Offset Text: this section contains controls for setting the offset Precision, text Size Scaler, Offset Scaler, Layer, text Style, and color, prefix and suffix for the offset labels.

Label Elevations Left Side Only: when enabled, elevations will be labeled on the left side of the sections only.

Vertical Justification: this sets the justification of the elevation text.

Place Label Offsets: **Top** will place the labels at the top of the grid, **Bottom** will place along the bottom of the grid and **None** skips drawing the offset labels.

Grid Spacing: sets the horizontal and vertical spacing for the grid lines as well as the horizontal and vertical Major Grid lines the offset text (grid spacing) and the elevation text.

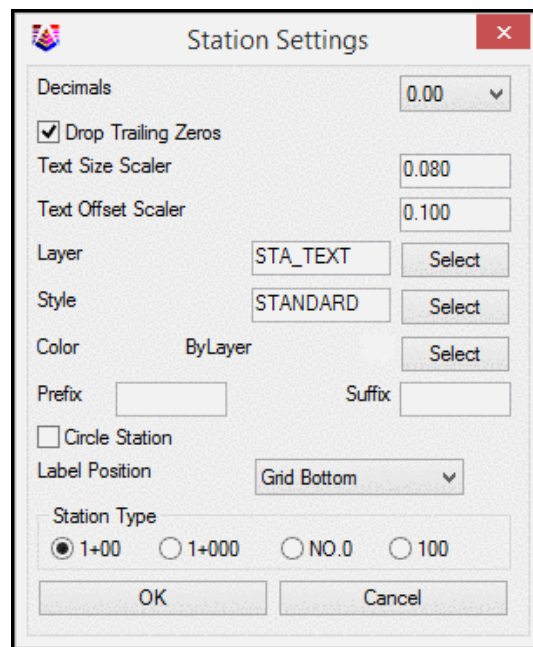
Label Scale: when enabled will label the horizontal and vertical scale.

Grid Style: sets the type of grid lines to be drawn. Grid Lines, Ticks Only, Ticks and Dots, Ticks and Checks, Text Only, Grid Lines and Dots, or Grid Lines Below 1st Section. This last option trims the grid lines by the first section so that the grid lines are drawn only below the section.

Draw Elevation Bar: places the elevations on a bar offset from the grid at a user defined location.

Label Vertical Axis: draws a label vertically along the vertical axis.

The **Station Settings** button displays another dialog box for the station label settings including decimal places, size, layer, style, color, prefix, suffix, format type and position. When the section file has descriptions assigned to the sections, the descriptions are labeled under the station position value. The **Circle Station** option will draw a circle around the station label.



Draw Horizontal Label Box: Enabling this option will draw a table with desired labeling above or below each cross-section. By picking the **Set** button to the right, you can choose the data to be placed in the table. The Elevation, Offset and Description of each point on the cross section can be added to the table. If more than one Section file (.sct) is being drawn on the cross-section, you will also have the option of displaying the elevation difference between sections.

Box Placement: Chooses between drawing the label box above or below the section grid.

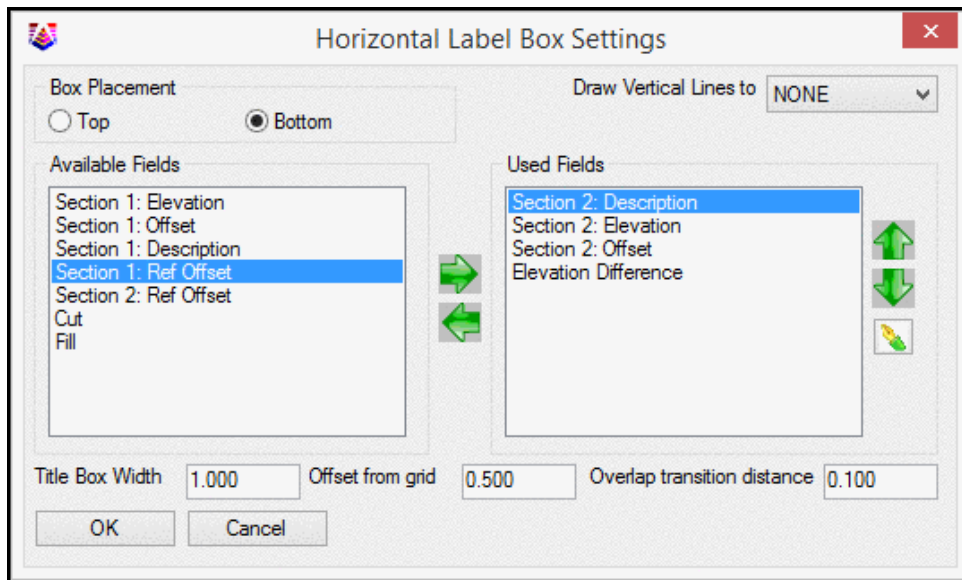
Draw Vertical Line: Sets drawing a vertical line from the section data point to the specified row in horizontal label box.

Title Box Width: Sets the size for the field name in the label box.

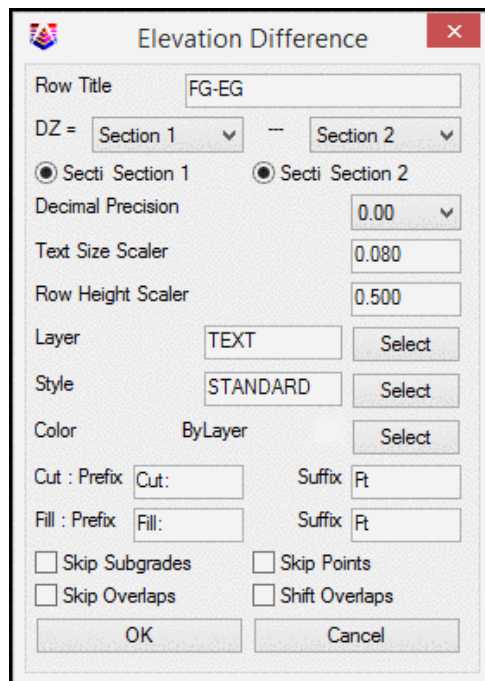
Offset From Grid: Controls the separation distance between the section grid and the label box.

Overlap Transition Distance: When drawing vertical lines and the labels are shifted to prevent overlap, this setting controls the length of the line for shifting over.

Label Elevation at Interval of Top Section: When labeling multiple values such as elevations for different sections, this option uses the stations from the section points of the top section as the stations to label for all the sections.



In the *Draw Horizontal Label Box* dialog, select from the **Available Fields** in the list on the left to populate the list of **Used Fields** on the right side. Once an item has been moved to the list of **Used Fields**, you can double-click on the Field to change settings and format for each Field. An example of the **Elevation Difference** option is shown below:



The **Row Title** for each field can be edited from the default to show a descriptive title. The **DZ** value in the Elevation Difference settings dialog allows you to specify which Section's elevations are to be subtracted from the other. This setting is critical to return the correct cut and fill depth values. In all field settings boxes, you have the ability to skip surface points in order to make the data more legible.

Place Labels at Interval: This option creates labels at the specified offset interval.

Place Labels at Break Offsets: This option creates labels at each section point.

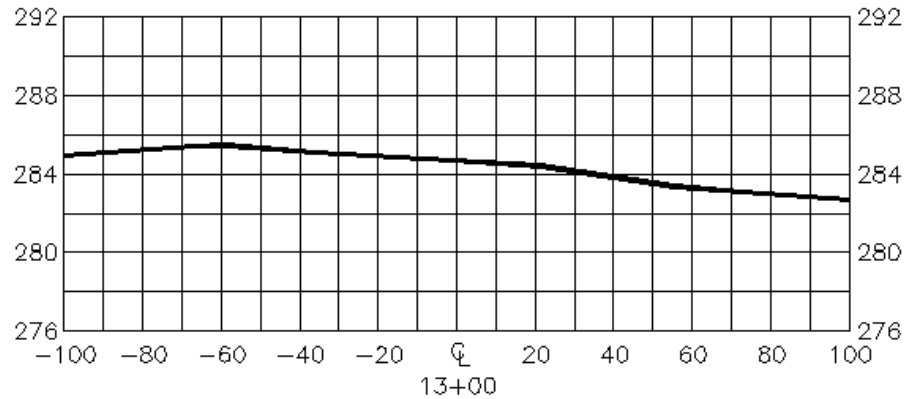
Skip Subgrades: Enable this option to skip all subgrades as may have been defined in Design Template files (.tpl).

Skip Points: Enable this option to skip points in the Section file (.sct) that were created using any of the **Create Section Points...** commands.

Skip Overlaps: Enabling this option will cause any overlapping text in the table to be skipped. Having this option enabled will disable the Shift Overlaps option.

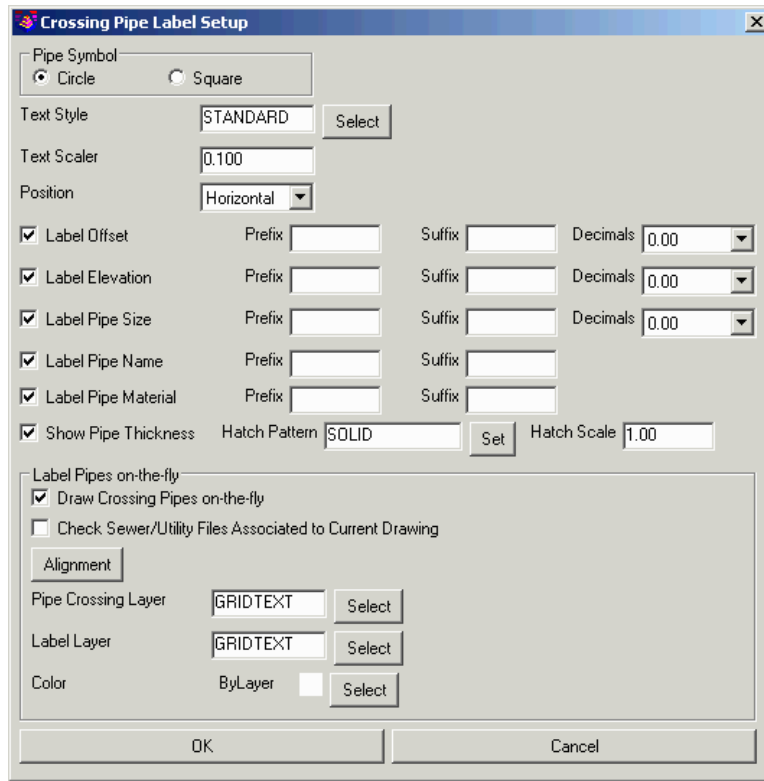
Shift Overlaps: Enabling this option will shift any text in the table to the right so that it does not overlap preceding text.

A sample cross-section with **Horizontal Label Box** is shown below:



EG : Elevation	284.92	285.42	285.48	285.08	284.70	284.42	283.37	283.07	282.70
EG : Offset	-100.00	-66.78	-58.61	-36.29	-1.24	21.45	55.51	72.66	100.00

Crossing Pipe Label Setup



Pipe Symbol: Choose whether to show the Pipe Crossing symbol as a circle or a square. When the sections are drawn with vertical exaggeration by different horizontal and vertical scales, then the circles or squares will be stretched into ellipses or rectangles.

Text Style and Text Scaler: Specify the text style and size of Pipe Crossing labels.

Position: Chooses between creating horizontal or vertical labels.

Label Offset, Label Elevation, Label Pipe Size, Label Pipe Name, Label Pipe Material: Enable any or all of these options to label the distance left or right off the alignment (Label Offset), the invert elevation, pipe size and pipe name of each crossing pipe. Use the optional settings for specifying "Prefix" or "Suffix" text and use Decimals to set precision for each label.

Show Pipe Thickness draws the pipe thickness around the crossing symbols using the specified hatch.

Draw Pipe Crossings on-the-fly: Enable this option to have Crossing Pipes that have been created using a Sewer Network file (.sew) or Draw Pipe 3D Polyline command drawn in cross sections. It is not necessary to enable this option if Pipe Crossings have been saved to a Section file (.sct) using the Section Points from Pipes command.

Check Sewer/Utility Files Associated to Current Drawing: This option looks for crossings with the pipes in .sew and .util files for the current drawing. To review and edit which files are in the current drawing, use the File > Drawing Explorer command. The program will show a list of files to choose from for checking.

Alignment: Pick this button to select either a Centerline file (.cl) or Section Alignment file (.mxs) to scan for Crossing Pipes.

Layer and Color: These settings specify the layer and color of the Pipe Crossing symbol.

Prompts

If the Pick Location option was specified, the program scans the station data and determines the minimum and maximum elevations, and proposes a datum elevation. If you have pre-plotted a grid sheet and want to reference another local grid coordinate, then change the datum elevation appropriately. The Pick Location type of plotting has the following prompts:

Station> 25.000 Min Elev> 1055.301 Max Elev> 1057.068

Change datum elev/<Select point that represents 0 offset elev 1050.0>: Pick a point

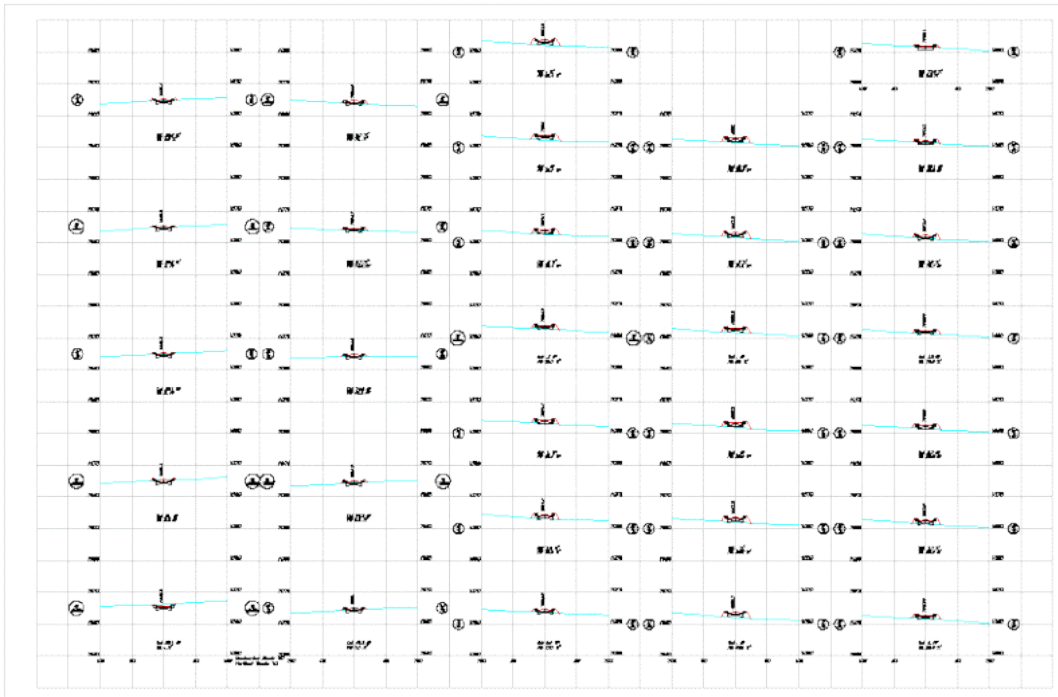
Station> 50.000 Min Elev> 1055.557 Max Elev> 1057.324

Change datum elev/<Select point that represents 0 offset elev 1050.0>: Pick a point

The program continues to prompt until the last station in the range specified is drawn. You can use the Cancel function (the *Esc* key) to stop plotting, if necessary.

If the Sheets option was specified with Model space as the destination, you can choose where to insert the sheet(s):

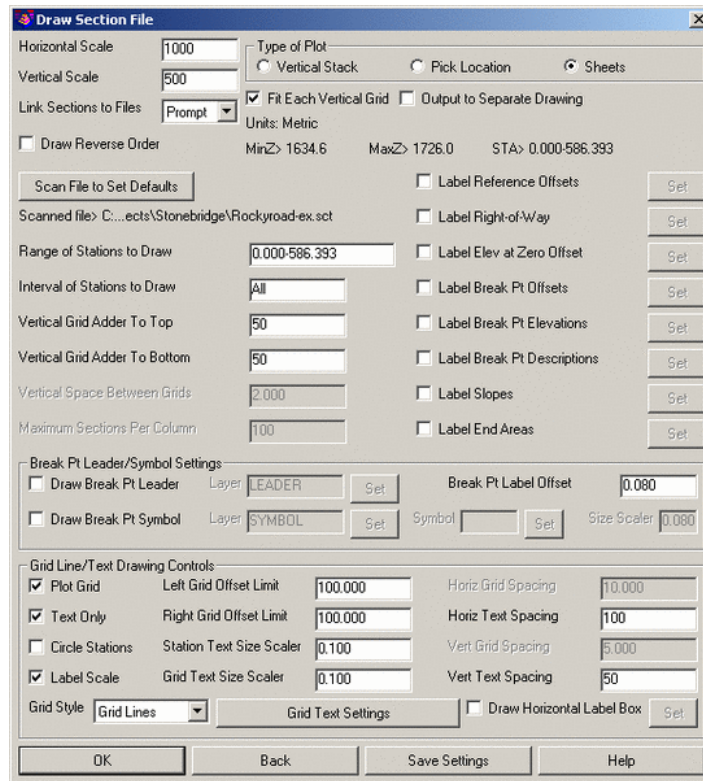
Select Starting Point for Row of Sheets <0.0,0.0>: Pick a point or press Enter to accept the default value specified



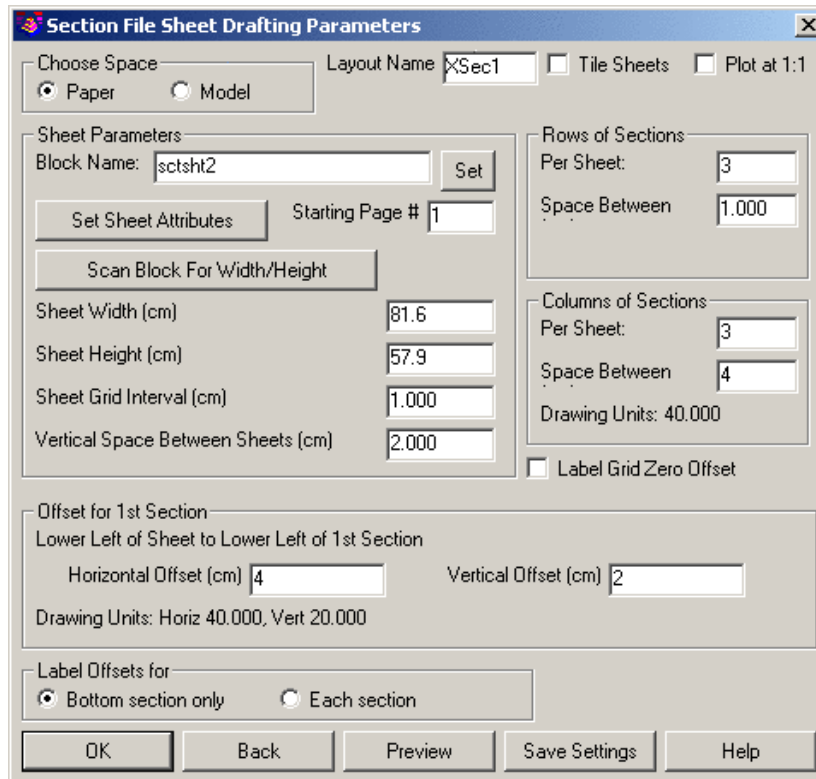
Sheet Sample

Drawing Metric Section Sheets

First, be sure that you are set to metric mode in Drawing Setup under the Settings menu. For our example, assume a 1:1000 horizontal scale. Once set, issue the Draw Section File command and click OK to reach the second dialog. There is a different block name for metric sections called schsht2.dwg which is located in the **%App-Data%\Carlson Software\...\Sup** folder. Begin by setting the parameters for the second dialog as shown.



Second dialog with metric settings

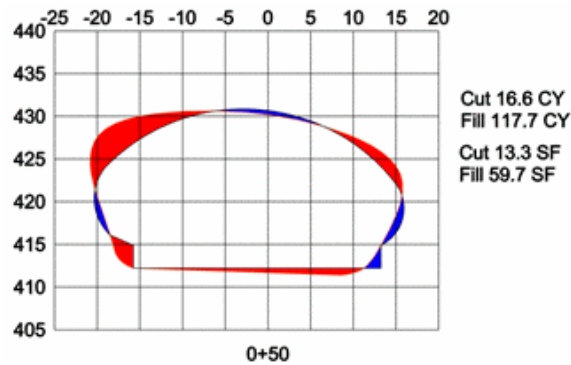


Third dialog with metric settings

Adjust settings as needed to achieve the desired look/layout.

For sections that make a closed loop such as for tunnels, the Label Volumes, Label End Areas and Hatch End Areas will use the difference between the sections. When you have one section for the tunnel design and another section

for the actual, these features will show the over cut and under cut amounts.



Pulldown Menu Location(s): Civil > Sections, Field > Roads

Keyboard Command: drawsct

Prerequisite: A Section (.sct) file

Sections to 3D Polylines

This command creates 3D polylines from a section (.SCT) file. Besides the section file, a centerline polyline, centerline file or section alignment (.MXS) file must be specified to define the plan view location of the 3D polylines. The elevations for the 3D polylines come from the section file. These 3D polylines can then be used by other Carlson routines to create surface models.

In the options, dialog set the range of stations to draw and the Layer Name for the 3D polylines.

The **Draw Method** has four options:

Cross Section: The 3D polylines are drawn as cross-sections perpendicular to the centerline at each station.

Offsets By Description: The 3D polylines are drawn parallel to the centerline by connecting section points when the same description. To use this method, the SCT file must have descriptions on the section points. For example, if the section file has descriptions for each section point then you can draw 3D polylines for EP, SHD, TIE, etc.

Offsets By Value: The 3D polylines are drawn parallel to the centerline at a specified offset value.

Offsets By Sequence: The 3D polylines are drawn parallel to the centerline by connecting section points by their sequence in the section file. For example, a 3D polyline will connect all the 1st section points, another 3D polyline will connect all the 2nd section points, etc. This method requires the sections to have the same number of data points for each station.

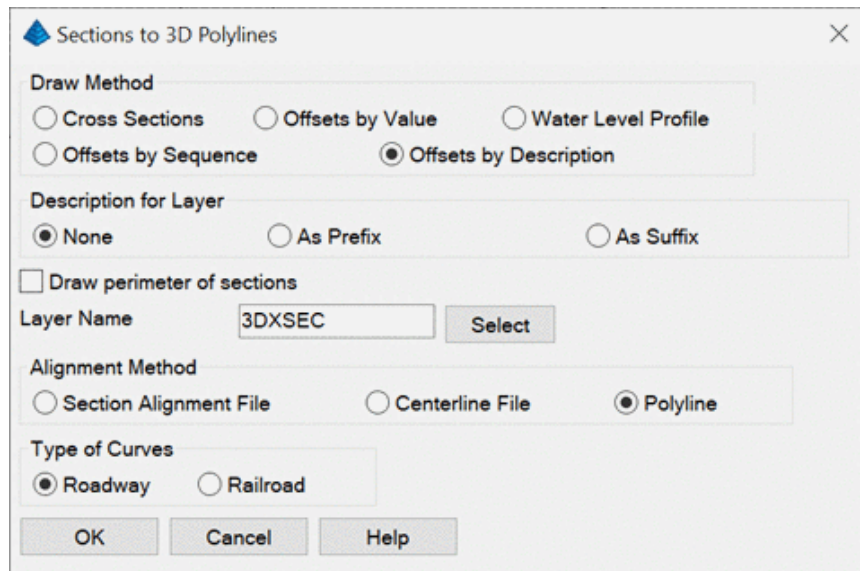
Water Level Profile: This method draws two 3D polylines for the left and right banks of a waterway. The section file represents the cross sections of the waterway. The program also prompts for a profile of the water level. At each section, the program gets the water elevation at that station from the water profile and finds the offset on the section for the left and right banks.

Description for Layer: This option uses the section point descriptions in the layer names for the 3D polylines when using the Offsets By Description method.

Draw perimeter of sections: This option will connect all the left most offsets and right most offsets together with a 3D polyline.

Alignment Method: Chooses how to define the horizontal alignment.

Type of Curves: This option chooses between roadway and railroad methods for stationing along curves.



Prompts

Choose Section File to Process Select the .sct file

Sections to 3D Polylines dialog Choose your settings

Select centerline polyline: *pick the polyline*

Enter the centerline starting station <0.0>: *press Enter*

Use reference profile to interpolate between sections [<Yes>/No]? *N for no.* This option will prompt for a profile to use for interpolating elevations along the 3D polylines between the section stations. This improves the accuracy when the profile goes through vertical curves. Without the profile, the 3D polyline elevations will be straightline interpolated between the sections.

Draw all template ids or specific ids and offsets [All/<Specific>]? *press Enter for Specific*

Enter Offset or Description to draw: *EP*

Pulldown Menu Location: Sections > Section Utilities

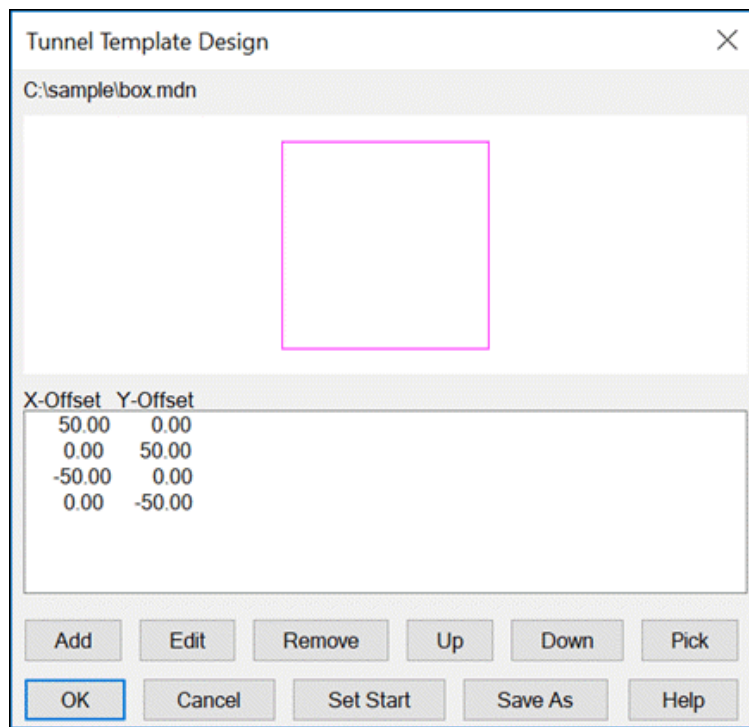
Keyboard Command: scto3dp

Prerequisite: A section (.SCT) file

Design Tunnel Template

This command defines a template to use with the Make Solid From Design command. The template defines the closed cross section for the tunnel. The dialog shows a graphic preview and list of segments. Each segment consists of an offset in X and Y (horizontal and vertical). Use the Add, Edit, Remove, Up and Down buttons to modify the list of segments. The tunnel template definition is stored in a .MDN file.

The template can also be created by first drawing a polyline perimeter and then use the Pick function in this dialog. The program will then use the dimensions of the polyline segments for the template. The starting point of the polyline is used as the alignment point for the tunnel centerline and profile. To change the starting point, highlight the row for the starting segment and pick the Set Start button.



Pulldown Menu Location: Solid

Keyboard Command: tunnel_template

Prerequisite: None

Draw Typical Tunnel Template

This command draws a tunnel template created by the Design Tunnel Template command. The program prompts for the tunnel template file (.MDN), layer name and position in the drawing. Then the tunnel template is drawn as a polyline.

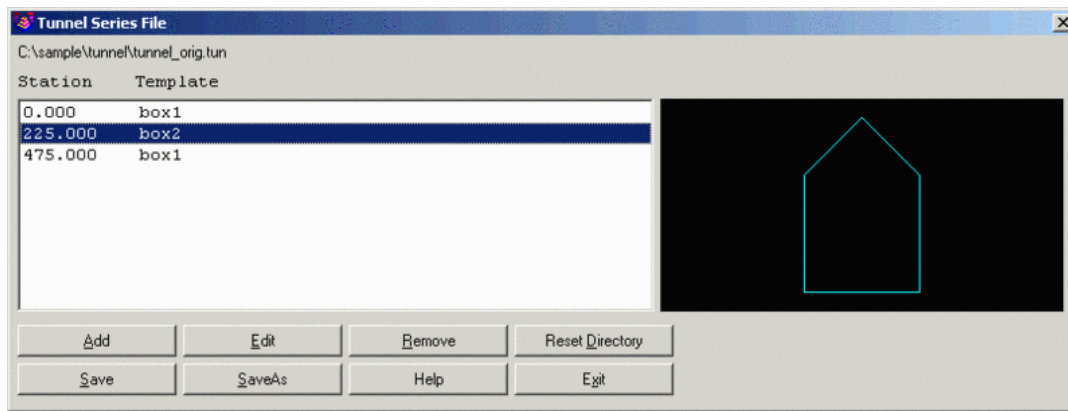
Pulldown Menu Location: Solid

Keyboard Command: draw_tunnel_mdn

Prerequisite: A tunnel template .MDN file

Tunnel Series Editor

This command defines a sequence of tunnel templates to use in the Make Solid From Design command. The tunnel series applies when the tunnel transitions along the alignment. For each tunnel cross section with different dimensions, use Design Tunnel Template to create a .mdn file. Then for the tunnel series, choose the tunnel templates in sequence and assign stations for switching between the templates. In the example dialog, the design starts with template box1, then switches to box2 at station 225 and then switches to box1 at station 475 for the rest of the design. The tunnel series data is stored in a .tun file.



Pulldown Menu Location: Solids
Keyboard Command: tunnel_series
Prerequisite: .mdn files

Tunnel Section Comparison Report

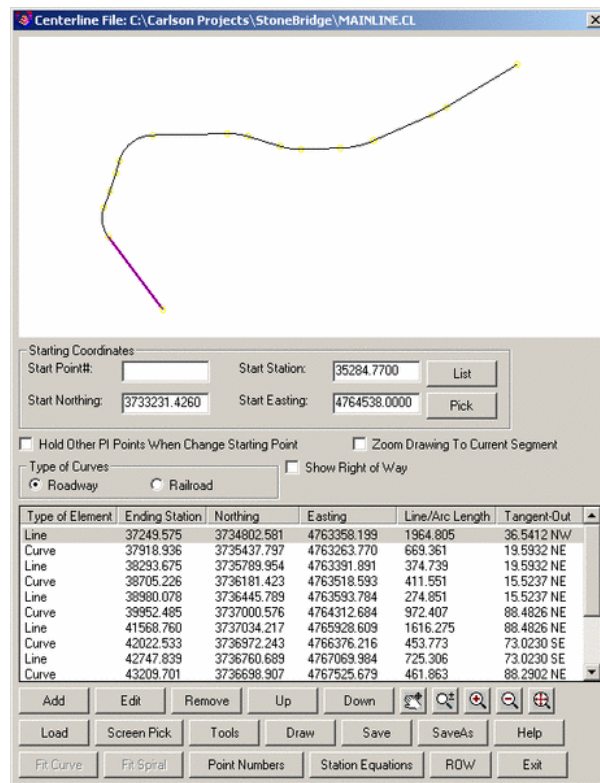
This command calculates the different between two tunnel sections and can be used to compare design sections with as-built sections. The tunnel sections need to make a closed loop. The report includes the end areas and volumes for each section. For the comparison, the report includes the over cut and under cut end areas and volumes. Over cut is area from the 2nd section file that goes outside the 1st section file. Under cut is area from the 1st section file that goes outside the 2nd section file.

Pulldown Menu Location: Solid
Keyboard Command: tunnel_sct
Prerequisite: Two section files (.sct)

Centerline Editor

This command can be used to input a new centerline or edit an existing centerline (.CL) file. It is a dialog-based alternative to Design Centerline and has the advantage of accepting whatever information you have on your centerlines (coordinates, stationing, length of tangents and arcs, *etc*). For creating a new centerline, it is ideal for entering data straight from highway design plans. For editing, this command allows you to change any of the geometric properties of any of the elements of the centerline (lines, curves, spiral-only and symmetrical spiral-curve-spiral elements), including the starting coordinates and station.

Starting this command launches the Centerline Editor main dialog box. To edit an existing Centerline, you can either pick the Load button and pick the .CL file, or pick the Screen Pick button and pick the polyline in the drawing that represents the Centerline. The Centerline is then displayed in the graphics window of the dialog box. The highlighted segment in the text window is also highlighted in the graphics window.



Drag Action (Zoom and Pan): In the graphics window, hold the left mouse button down and move mouse to Pan, roll the wheel to Zoom.

Zoom Drawing To Current Segment: This option zooms the drawing graphics to center on the centerline segment currently highlighted in the dialog.

Hold Other PI Points When Change Starting Point: With this option active, all the existing PI's are held when the starting coordinate is moved. Otherwise, all the PI's are moved by the same amount that the starting point is moved.

Show Right of Way: This option shows any ROW's defined in the centerline in the graphic preview window.

Type of Curves: This setting chooses between roadway and railroad definitions for curve lengths.

Add: Adds a new element after the highlighted element. Prompts you for the type of the element to be added, Line, Curve, Spiral-Only or Spiral-Curve-Spiral.

Edit: Allows you to edit the highlighted segment.

Remove: Removes the highlighted element from the centerline.

Up/Down: Moves elements in the table Up and Down in the list. For example, if this centerline ended with a tangential line from the last curve, then was followed by a non-tangential line at 45d NE, moving the last element up would create a line at 45d after the curve (non-tangential), and the formerly tangential line will remain tangential and therefore continue at NE 45d.

Load: Loads an existing centerline (.CL) file for review or editing. After loading a centerline, the listbox in the dialog shows a list of all the elements in the centerline, identifying them as either a line, curve, spiral only or full spiral-curve-spiral element and reporting the ending station, northing and easting of the element.

Screen Pick: Allows user to pick a CL off the screen in the drawing to load into the editor.

Tools > Reverse: Reverses direction of Centerline.

Tools > Rotate: Rotates the centerline by the specified rotation angle and around the specified pivot point.

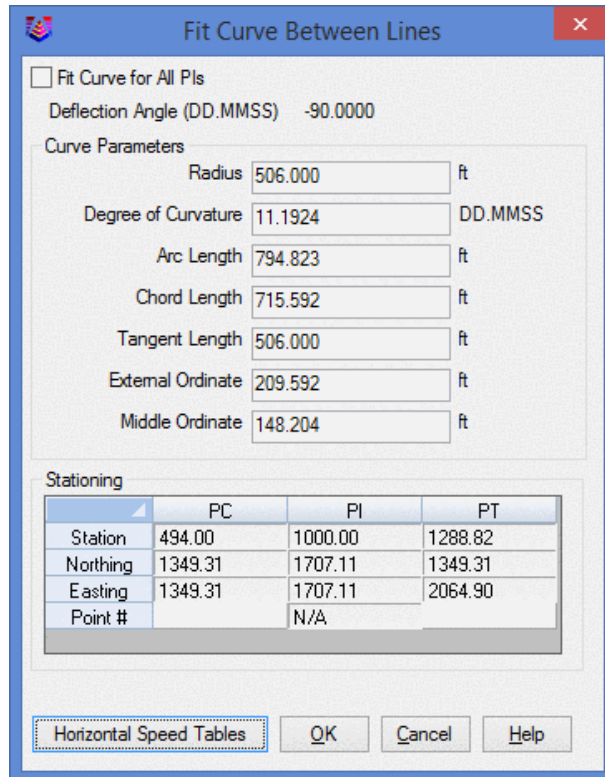
Tools > Scale: Scales the centerline which can be used for changing units between metric and English.

Draw: This button draws the centerline in the drawing on the specified layer.

Save: Saves the currently loaded centerline to a file, or will prompt you for a name if no name has been set.

SaveAs: Prompts you for a file name for the saved file.

Fit Curve: Fits a circular curve element into the centerline after the line element that is currently selected. When all the elements are lines, the program allows you to fit curve for all PIs. The program checks if the radius fits all PIs and will prompt the maximum radius that works for all PIs if the current radius is too big. The Horizontal Speed Tables allows you to pick a speed and a super elevation rate to the minimum radius.

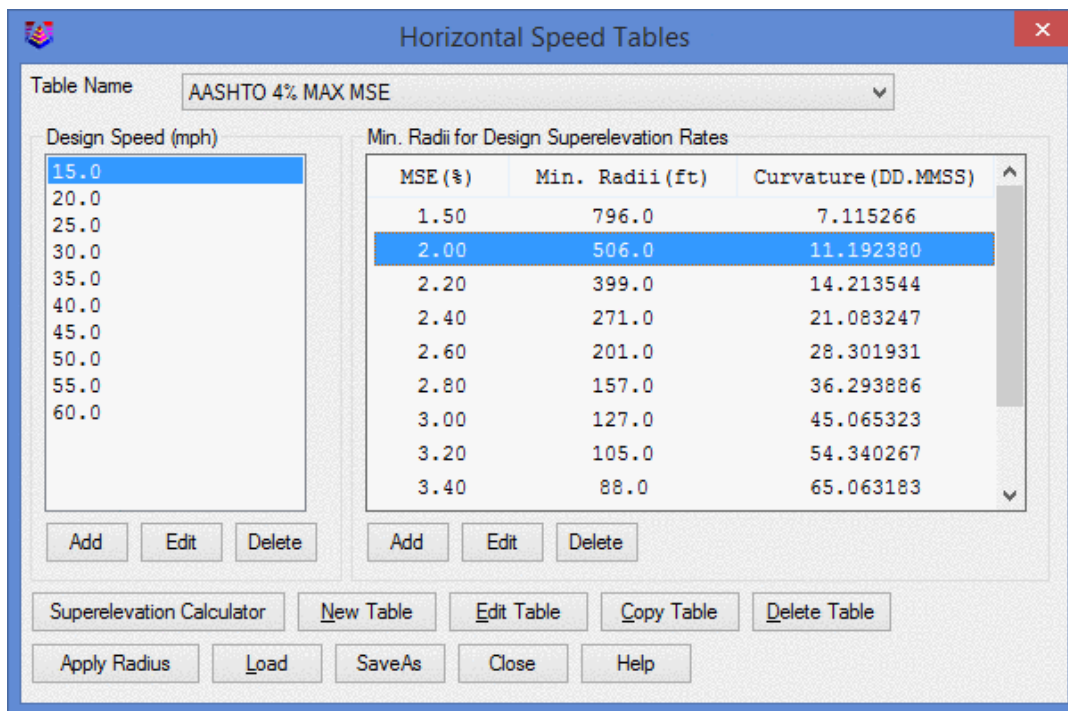


The dialog box titled "Fit Curve Between Lines" contains the following elements:

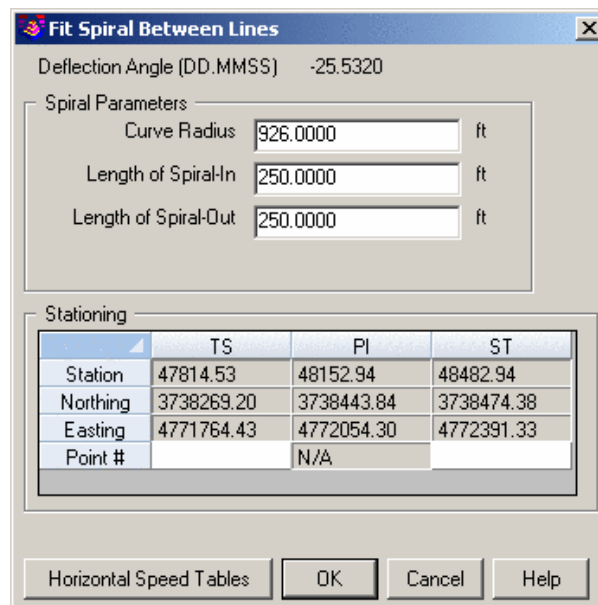
- Fit Curve for All PIs
- Deflection Angle (DD.MMSS) -90.0000
- Curve Parameters section with input fields:
 - Radius: 506.000 ft
 - Degree of Curvature: 11.1924 DD.MMSS
 - Arc Length: 794.823 ft
 - Chord Length: 715.592 ft
 - Tangent Length: 506.000 ft
 - External Ordinate: 209.592 ft
 - Middle Ordinate: 148.204 ft
- Stationing table:

	PC	PI	PT
Station	494.00	1000.00	1288.82
Northing	1349.31	1707.11	1349.31
Easting	1349.31	1707.11	2064.90
Point #		N/A	

Buttons at the bottom: Horizontal Speed Tables (highlighted), OK, Cancel, Help.

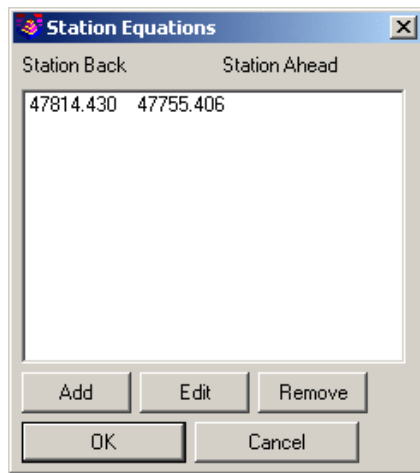


Fit Spiral: Fits a spiral curve element into the centerline after the line element that is currently selected.



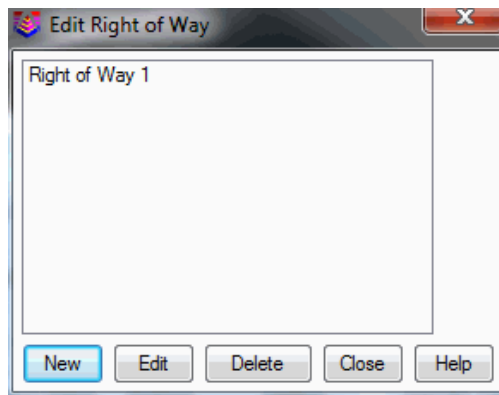
Point Numbers: This will create Carlson points along the elements of the centerline and store them to the current CRD file. The new points will be numbered in sequence beginning with the first available point number in the CRD file.

Station Equations: At any number of locations on a centerline, you can set the back station and forward station for the re-stationing of the centerline. The station equation dialog appears below:

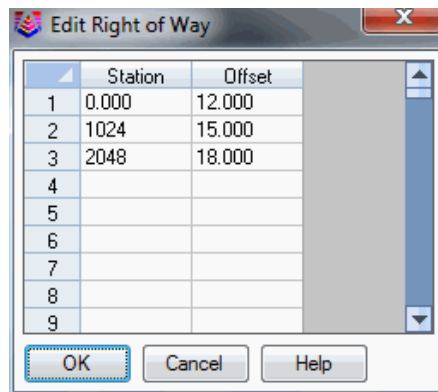


If the Station Back is lower than the Station Ahead, then a "gap" is inserted in the centerline, where the stations jump forward. If the Station Ahead is less than the Station Back, then an overlap occurs, where the common station range is repeated.

ROW: This function edits the right-of-way definitions associated with the centerline. There can be multiple ROW's assigned to the centerline for left and right sides as well as multiple on the same side. The function first shows a list of ROW's for the centerline where you can add, edit or delete.



When you add or edit a ROW, there is a second dialog for entering the stations and offsets that define the ROW relative to the centerline. Use negative offsets for left and positive for right.



Alternatively, the **Enter Right of Way** and **Polyline to Right of Way** commands are other ways to define the ROW's for a centerline.

Exit: Exits this routine, prompting to save changes if necessary.

The dialog for every type of element shows the point ID, the northing, easting and station of the start point of the

element. It then allows the user to modify or define the parameters specific to the type of element. The following are some of the things to remember about data entry in the centerline editor. These are valid for lines, curves and spirals.

- Wherever length of the element is to be entered, entering an expression of the type $123.5 - 93.7$ would evaluate the difference of the values. This is particularly convenient where only the stations of the start and end points of the element are known.
- When the station is specified, the program takes the length of the element as the difference between the station of the start point of the element and the station specified.
- All bearings should be specified by entering the angle between 0 and 90 degrees (in dd.mmss format) and selecting the quadrant.
- When entering the delta angle of a curve, only the absolute value (between 0 and 360 degrees) is to be entered. The direction of the curve is to be explicitly set as right or left, the default being left. All angles are entered in (dd.mmss) format.
- Point numbers, when used, access their coordinates in the current .CRD file. If the point number specified has no coordinates stored in the coordinate file, the point number is remembered for that particular location (say the radius point of a curve or the SC point of a spiral). Then, when the .CL file is saved, the program creates points for that location and stores them to the .CRD file with the specified point number.

The dialog for a Line allows the user to specify the line primarily by its length or station and its bearing. The line can also be defined by its end point number or its coordinates. The bearing of a line can be changed if the Tangential to the Previous Element toggle is not checked. By default, any line which follows a curve element is defaulted to be tangential to it. To use a bearing different than that of the previous element, uncheck this toggle and enter the bearing.

The dialog for the Curve allows the user to define the curve primarily by its radius and delta angle or arc length. The other parameters of the curve that can be edited are the bearing of tangent-out and the "Station to", which also defines the arc length. The curve can also be specified by entering the coordinates or point numbers of its end point (PT) and the radius point. Another way to specify the curve would be to enter the chord length or PT point station and chord bearing. If the central PI point and a point on the forward tangent are known, then the curve can be defined by entering both of these points and at least one other property of the curve (like radius, arc length, delta angle). The point on the forward tangent can be any point that defines the tangent out direction including the next PI point. If only the central PI point is known, then the tangent-out can be entered by bearing instead of by forward tangent point. Central PI and forward tangent points are not displayed from the .CL file. They have to be entered by the user and are valid only for that particular edit session; that is, they are not remembered the next time the file is loaded. Curves are assumed to be tangent to the last element unless the Tangential to the Previous Element checkbox is cleared.

The Curve Edit Mode option defines how the curve is accepted in the centerline. If the Hold PC point is checked on, the radius is taken as fixed and the delta angle of the curve is calculated based on some additional parameter. Hence, the extent of the curve is unlimited. However, if the Hold PI points option is checked on, the bearing of tangent-out of the curve is taken as fixed and the radius is calculated based on some other parameter. In this case, the curve is completely restricted within the central PI point and the bearing of tangent out. Hence, when the Hold PI points option is checked on, the above parameters should also be defined to carry out the calculations.

The dialog for the Spiral-Curve-Spiral element allows the user to define the spiral by entering either the various parameters of the spiral (like the angles and lengths) or the coordinates or point numbers of its defining points: the TS (Tangent-to-Spiral), SC (Spiral-to-Curve), Radius point, CS (Curve-to-Spiral), ST (Spiral-to-Tangent) and end point (optional). While defining the spiral by its geometric properties, the program will accept the data even if the information for the simple curve is given with zero spiral lengths. In this method, however, the central PI point of the spiral MUST be specified (that is, it is always in Hold PI Points mode). The tangent out can be defined by entering bearing or by specifying a point on the forward tangent. This forward tangent point can be the next PI coordinates. The direction of the spiral-in and spiral-out elements would be the same as the direction of the simple curve (left or right). The Spiral Definition setting chooses between Arc definition for clothoid spirals and Chord for 10-chord spirals.

The spiral can be defined by several different parameters and the order that you enter data into the spiral dialog can

be important. There are two main sequences for entering data. The method to use depends on the spiral data that you have. The first method is to enter the radius of the simple curve, the spiral in and out lengths, the tangent bearing out and the PI station. The second method is to make a Line segment coming up to the TS (tangent to spiral) point. This Line segment should be added before creating the Spiral element. Then with the Spiral In point set to the TS point, enter the radius of the simple curve, the spiral in and out lengths, the curve direction (left or right) and the arc length of the simple curve. Then the rest of the spiral points will be calculated.

The Spiral Only element allows for flexible transitions from curve to spiral to curve or line to spiral to curve or between any combination of curve and line elements. The Spiral-Curve-Spiral element, for example, can be entered as Line, Spiral Only, Curve, Spiral Only and Line, producing the same results. You can spiral from tangent to curve, curve to tangent and curve of one radius to curve of another radius. You can also spiral from one endpoint to another endpoint. To define the spiral by sweep angle, use the Delta Angle field. To define the spiral by length, use the Spiral Length field. To define the spiral by end point, fill in the min and max radius fields and then enter either the End Point Pnt# or coordinates and the program will calculate the radius and spiral length to fit that point.

Spiral-Only Element

Spiral-In Northing: 682243.3051
 Spiral-In Easting: 2054761.0326
 Spiral-In Station: 1337.1826

Spiral Length: 175.0000
 Delta (Sweep) Angle (DD.MMSS): 6.160021

Spiral Direction: Left Right
 Element At Spiral Out: Line Curve

Curve Radius At Spiral Out: 800.000
 Min. Radius: 600 Max. Radius: 1100

End Point#:
 Pnt#:
 Northing: 682308.5742
 Easting: 2054598.7600

Tangential to Previous Element

Angle Format: NE SE SW NW AZ Gon

Bearing(dd.mmss): 66.000139
 Deflection Angle(DD.MMSS): 0.000000

Horizontal Speed Table OK Cancel Help

Once all the elements of the centerline are defined, the file can be saved and then plotted using the *Draw Centerline File* command.

Here is an example of a highway interchange ramp that involves a starting tangent and a spiral curve that goes abruptly into a simple curve and then a final tangent. Start by entering a starting Northing and Easting and starting Station. The Start Point# is optional. Then the concept is that you click Add to add each subsequent element (line, curve, spiral-curve-spiral or spiral only):

Starting Coordinates

Start Point#: Start Station:

Start Northing: Start Easting:

Hold Other PI Points When Change Starting Point Zoom Drawing To Current Segment

Type of Curves

Roadway Railroad Show Right of Way

Type of Element	Ending Station	Northing	Easting	Line/Arc Length	Tangent-Out

Line (Tangent) Segment: We want to enter the tangent segment length up to the TS (tangent to spiral). Enter in the length (200.0), bearing (88.0732) and then the bearing quadrant (NW). Since the next spiral-curve-spiral element can be based on a PI station, it is not necessary for this line segment to go up to the TS point. The purpose of this line segment is to establish the tangent-in direction.

Line Element [X]

Start Point#:
 Start Point Easting: 4768889.0000
 Start Point Northing: 3736808.0000
 Start Point Station: 1200.0000

End Point#:

End Point Northing:

End Point Easting:

Length:

Station to:

Tangential to Previous Element

Angle Format

NE SE SW NW AZ

Bearing(dd.mmss):

Deflection Angle(DD.MMSS):

When OK is clicked, the routine will add the Line element as the first in the list of complete centerline elements. Next up is Curve-Spiral-Curve. Click Add.

Spiral-Curve-Spiral Element

Spiral In Point#: Spiral In Point Nothing: 3736814.5419
Spiral In Point Station: 1400.0000 Spiral In Point Easting: 4768683.1070

Spiral Definition:
 Arc Chord Hold PI Point

Data for Simple Curve:
Radius: Curve Direction: Left Right
Degree of Curve (DD.MMSS): Delta Angle:
Arc Length:

Central PI Point:
Point#: Length of Spiral-In: Delta (Sweep) Angle (DD.MMSS):
Northing: Length of Spiral-Out: Delta (Sweep) Angle (DD.MMSS):
Easting: Point on Forward Tangent: Tangent-Out: Angle Format: NE SE SW NW AZ
Station: Point#: Easting: Bearing(DD.MMSS):
Length: Length: Length:

Tangent-to-Spiral Pt Spiral-to-Curve Pt Curve-to-Spiral Pt
Pnt#: Sta:1400.000 Pnt#: Sta:1475.000 Pnt#: Sta:1577.421
Northing: Northing: Northing:
Easting: Easting: Easting:

Spiral-to-Tangent Pt Simple Curve Radius Pt End Point
Pnt#: Sta:1652.421 Pnt#: Pnt#:
Northing: Northing: Northing:
Easting: Easting: Easting:

Horizontal Speed Table OK Cancel Help

Spiral Segment: Though the dialog is complex (for total flexibility), the key on a typical symmetrical spiral curve is to enter four things: (1) the radius of the simple curve, (2) the spiral in and out lengths, and (3) the tangent-out bearing. Everything else will calculate when you press Enter for the PI station.

Curve Segment: Add the next element and select curve. The Curve dialog appears. The key is to enter the Radius Length (255), the Arc Length (150) and the Curve Direction. Everything else will calculate.

Curve Element

PC Station: 1652.4212 Northing: 3736960.5769 Easting: 4768665.1318

Curve Edit Mode:
 Hold PC Point Hold PI Points Curve Direction: Left Right

Tangential to Previous Element

Radius: Station To:
Degree of Curve (DD.MMSS): Chord Length:
Delta Ang.(DD.MMSS): Chord Brg(DD.MMSS):
Arc Length: Format: Use Radial Angle

PT Point Radius Point
Point#: Point#:
Northing: Northing:
Easting: Easting:

Central PI Point Point on Forward Tangent
Point#: List Pick Point#:
Northing: Northing:
Easting: Easting:
Length (from PC): Length (from PT):

Bearing of Tangent-Out(DD.MMSS): Angle Format:

Horizontal Speed Table OK Cancel Help

Final Line Segment: All you need to enter in the final dialog for the line (tangent) segment is its length. All other items will calculate when you press Enter.

Line Element

Start Point#:
 Start Point Easting: 4768812.6754
 Start Point Northing: 3736970.0390
 Start Point Station: 1802.4212

End Point#: List Pick

End Point Northing:

End Point Easting:

Length:

Station to:

Tangential to Previous Element

Angle Format
 NE SE SW NW AZ

Bearing(dd.mmss):

Deflection Angle(DD,MMSS):

OK Cancel Help

The completed centerline will appear as shown in the dialog and each element can be edited. Pick the Save button to store this centerline data to a .CL file.

Centerline File: C:\Carlson projects\Stonebridge\Ramp.cl

Starting Coordinates
 Start Point#: Start Station: List
 Start Northing: Start Easting: Pick

Hold Other PI Points When Change Starting Point Zoom Drawing To Current Segment

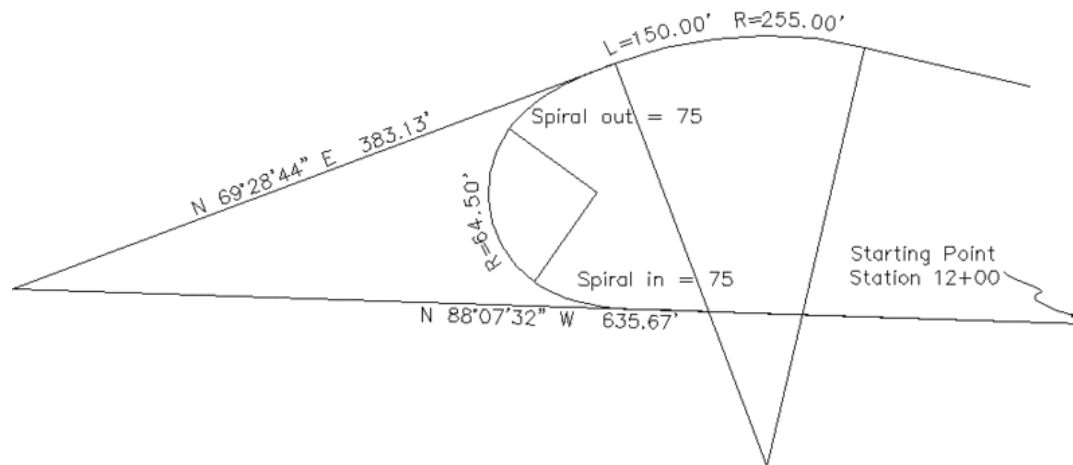
Type of Curves
 Roadway Railroad Show Right of Way

Type of Element	Ending Station	Northing	Easting	Line/Arc Length	Tangent-Out
Line	1400.000	3736814.542	4768889.107	200.000	88.0732 NW
Spiral-Curve-Spiral	1652.421	3736960.577	4768865.132	102.421	69.2844 NE
Curve	1802.421	3736970.039	4768812.675	150.000	76.4904 SE
Line	1902.421	3736947.234	4768910.040	100.000	76.4904 SE

Add Edit Remove Up Down

Load Screen Pick Tools Draw Save SaveAs Help

Fit Curve Fit Spiral Point Numbers Station Equations ROW Exit



Pulldown Menu Location(s): Centerline (Survey, Civil), Roads (Construction, Takeoff)

Keyboard Command: ccredit

Prerequisite: None

Polyline to Centerline File

This command writes a centerline (.CL) file from a polyline in the direction the polyline was drawn. The Northing and Easting for each vertex of the polyline is written to the centerline file and each arc in the polyline becomes a circular curve. After selecting the polyline, the program shows the direction by drawing temporary arrows along the polyline. To reverse the direction of the polyline, there is a keyword option R for Reverse at the Command line. Also, the Reverse Polyline command can be used to switch the direction of a polyline.

For stationing the centerline, there is a Command line prompt for entering the station at the beginning of the polyline and then using the polyline segment lengths for the rest of the centerline stations. Alternatively, there is a keyword option E for Ending to specify the station at the end of the polyline and then back calculating the centerline stations to the beginning using the polyline lengths. Also, the keyword P for Point prompts to pick a reference point along the polyline and enter the station at this reference point.

In addition to being used as roadway/corridor "baselines," a .CL file can also be used as the horizontal control for a Template Point Centerline.

Note: To convert lines and/or arcs into a polyline, use the Entities to Polylines command or the Join Nearest command

Prompts

Centerline file to Write dialog *Enter the .CL file name to create*

Centerline station [Reverse/Ending/Point/<Beginning: 0+00>]: *Press Enter to accept the default station value specified or Type in the beginning station then press Enter*

Select polyline that represents centerline: *Pick the polyline that represents your centerline*

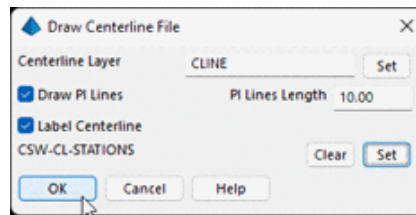
Pulldown Menu Location(s): Civil > Centerline, Survey > Centerline, Field > Roads

Keyboard Command: cpline

Prerequisite: A polyline drawn in the direction of increasing station values

Draw Centerline File

This command reads a centerline (.CL) file and plots it as a 2D polyline in the drawing at the proper coordinates.



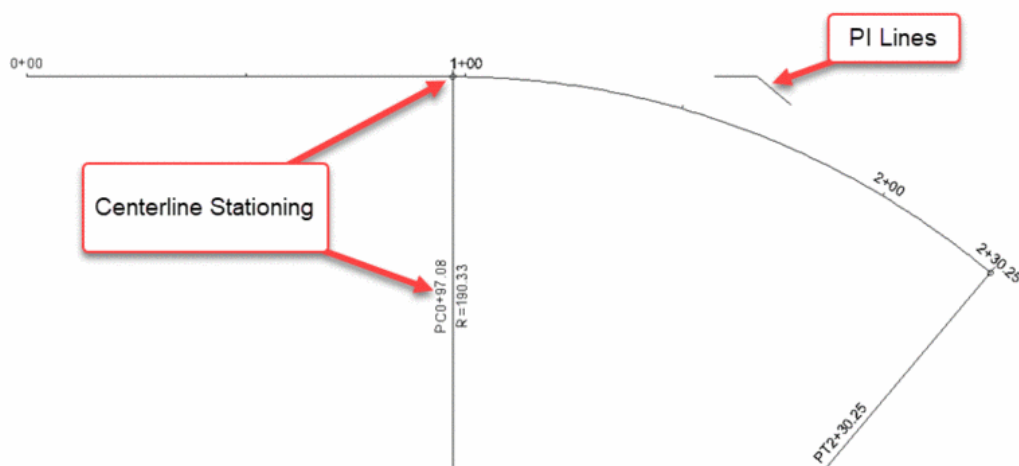
Centerline Layer is the layer name for the polyline to be drawn upon. The **Set** button can be used if you desire to select an existing layer from the current active drawing.

Draw PI Lines draws the straight line segments entering and exiting the PI. The length of the lines is controlled by entering a numerical value in the **PI Lines Length** option.

Label Centerline option draws station labels using a .STA settings file created by the Save Settings function in the Station Polyline/Centerline command.

Clear clears the current .STA file selection

When you click **<OK>**, you are prompted for the file name of the centerline(s) to plot. You can select single centerline (.cl) files individually or, by holding down the **Shift** or **Ctrl** keys, select multiple files at once. All centerline files are drawn with the same settings entered in the above dialog box.



The .CL file can be made with the following commands on the Design menu: *Polyline to CL File*, *Centerline Editor* or *Design Centerline*. Drawing the centerline file is a way to check the .CL file data graphically for correctness. If a spiral exists in the .CL file, the spiral will be represented by polyline segments.

Prompts

Draw Centerline Options dialog

Centerline File to Draw file selection dialog Select the .CL file name(s) to read and plot. Use the CTRL and/or SHIFT keys to select multiple Centerline file(s).

Pulldown Menu Location: Centerline

Keyboard Command: cl2pline

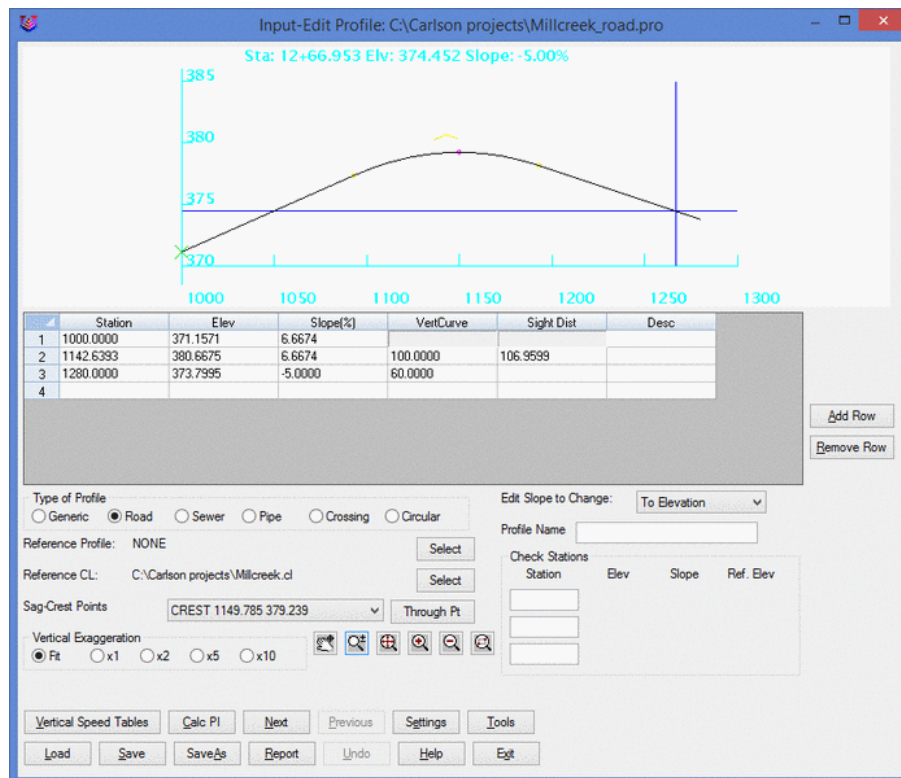
Prerequisite: a centerline file

Profile Editor

Similar to the Road Profile Editor command, this command features a spreadsheet type editor and handles a variety of profile (.PRO) configurations. Besides editing a profile, this routine can be used to just view the contents of a profile.

The command starts by prompting for the profile file to edit. Alternately, you can run Profile Editor by double-clicking on a profile polyline that is drawn on a profile grid.

The opening dialog below shows the layout of this editor. At the top of the dialog, you can dynamically see the profile and vary its appearance by using zoom and pan. The station, elevation and slopes are also shown at the lower left of the dialog which update/track with the movement of the cursor. There are between five and nine possible data fields in a profile depending on the type of profile that has been selected.



Profile Name: This name is optional and often used when multiple profiles are stored in a profile (.PRO) file and graphically generated using the Draw Profile command.

Add Row: Adds a new row into the profile after the current row.

Remove Row: Removes the current row.

Type of Profile: There are 6 types of .pro files and the spreadsheet columns will change to match the data fields for the selected profile type:

- *Generic* - Generic profiles have station, elevation and description fields.
- *Road* - Road profiles include the Generic controls and adds a vertical curve field. For an asymmetrical vertical curve, enter the left and right side values separated by a dash in the spreadsheet cell. For example, a 200' vertical curve with 50' to the left of PVI and 150' to the right would be entered as "50-150".

- *Sewer* - Sewer profiles include the Generic controls and adds step up, pipe size, pipe thickness, manhole elevation and manhole ID fields.
- *Pipe* - Pipe profiles include the Generic controls and adds a pipe size field.
- *Crossing* - Crossing profiles are for pipe crossings along the centerline. Besides station and elevation, the crossing data points also have the pipe size. The crossing elevation is for the bottom elevation of the pipe. The crossing profile data points are not connected.
- *Circular* - Circular profiles are the same as Road profiles except the vertical curve is circular instead of parabolic.

Edit Slope To Change: This setting controls which field to update when the slope is modified in the spreadsheet.

Reference Profile: Selects a reference profile and displays it in the profile graphic view. Also a Depth column is added to the spreadsheet which you can edit to set the profile elevation by depth from the reference profile.

Reference CL: Selects a reference centerline which is used for station equations.

Sag-Crest Points: When editing a road profile, its sag/crest points are shown here.

Through Pt: This button lets user to make the road profile pass through a certain point.

Vertical Exaggeration: Changes the look of the profile.

Edit Slope to Change: When edit the slope value, you can choose a value to change from the following selection: the next elevation, the previous elevation, the next station or the previous station.

Check Stations: Reports profile information at the specified stations. The Check Stations are not stored in the profile; they are merely used as a design/analysis tool for viewing the elevations at certain stations while adjusting the profile data. The **Offset** and **Slope** adjust the profile elevation and report the elevation at the offset.

Speed Tables: This button is enabled only when you edit a road profile. Please refer to the documentation on Road Profile Editor for the information on Vertical Speed Tables.

Next: Used for navigation when editing a .PRO file containing multiple profiles, loads the next profile.

Previous: Used for navigation when editing a .PRO file containing multiple profiles, loads the previous profile.

Load: Used for loading another, existing .PRO file for editing.

Save: Saves the profile using the current profile file name. The current profile file name is displayed in the top title bar of the dialog box.

SaveAs: Allows you to save the profile under a different profile file name.

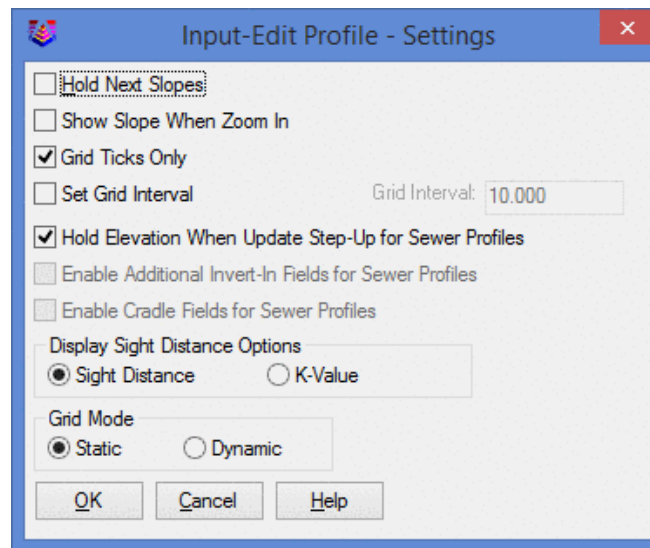
Calc PI: This function calculates a station/elevation point given two existing station/elevation points and slopes from them. The values are entered in this dialog. When you pick Calculate, the program finds the intersection of the grade lines. Then pick OK and the calculated PVI is added to the profile.

Calculate Intersection Point	
Intersection at Station: 1630.000 Elevation: 997.200	
Starting Station	790
Starting Elevation	993
Forward Station	1850
Forward Elevation	999.4
Slope From Starting Position to PI (percent)	0.5
Slope From Forward Position to PI (percent)	-1.0
<input type="button" value="Calculate"/> <input type="button" value="OK"/> <input type="button" value="Cancel"/>	

Report: Creates a report of current profile.

Undo: Reverts the last action in the editor.

Settings: Opens the settings dialog.



Hold Next Slopes: When editing a profile elevation, this option will maintain all the slopes after the edit point by adjusting the elevations. Otherwise, the elevations for the rest of the profile points are held and the slope from the edit profile point to the next profile point is adjusted.

Show Slope When Zoom In: This option allows to display the slopes on the long enough profile segments when zoom in.

Grid Ticks Only: Toggles between displaying the grid and grid ticks in the graphic box.

Set Grid Interval: This option allows you to control the elevation grid spacing in the graphic preview. When this option is off, the program automatically figures the elevation grid interval.

Hold Elevation When Update Step-Up for Sewer Profiles: Toggles between holding the elevation or the pipe slope

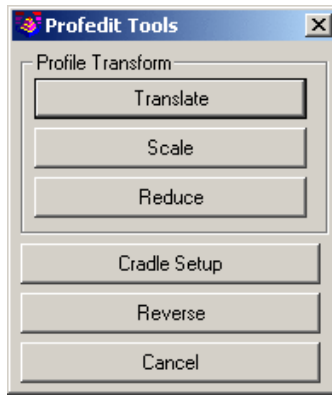
Enable Additional Invert-In Fields for Sewer Profiles: When editing a sewer profile, this option allows you to display an extra invert-in column for in-coming pipes. The invert-in elevations are separated by commas.

Enable Cradle Fields for Sewer Profiles: When editing a sewer profile, this option allows to display cradle above and cradle below columns.

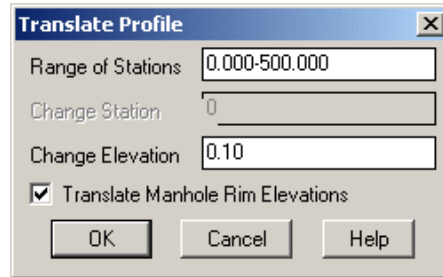
Display Sight Distance Options: Display either K-Value or Sight Distance in the fifth column for road profiles.

Grid Mode: The **Dynamic** option will update the grid interval labels when you zoom in or out of the profile image. The **Static** option will keep the grid interval labels static.

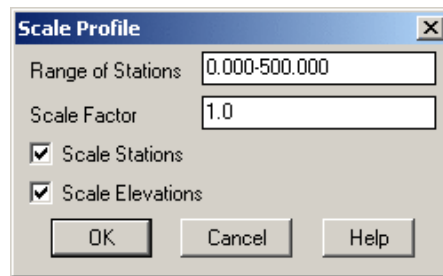
Tools: Opens the Tools dialog.



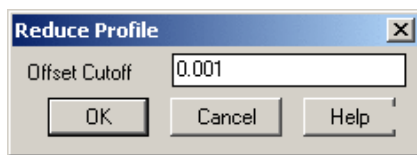
Translate: Globally adds or subtracts value to stations and/or elevations within the specified range of stations. while Scale will apply the specified scale factor to stations and/or elevations within the specified range of stations.



Scale: Applies the specified scale factor to stations and/or elevations within the specified range of stations.

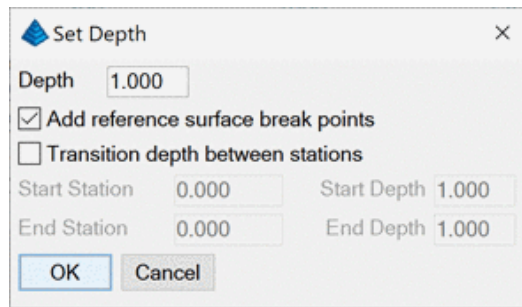


Reduce: Reduces the profile points by the Offset Cutoff value.

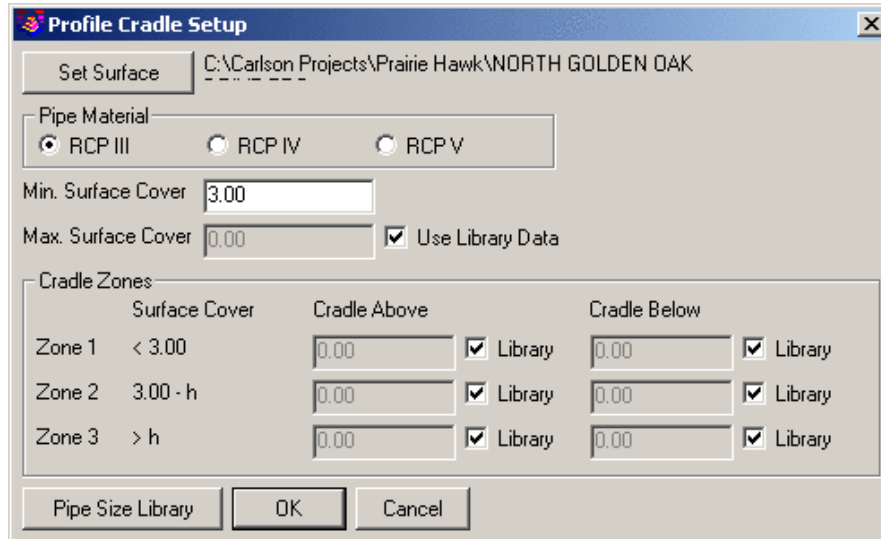


Reverse: Reverses the direction of the stationing for the profile.

Set Depth: When using a Reference Profile, this function sets the profile elevations by a specified depth from the reference profile.



Cradle Setup: Sets up the cradles for sewer profiles. The cradle parameters are different with different pipe sizes and are defined in the Pipe Size Library. You can either use library data or specify new values here.



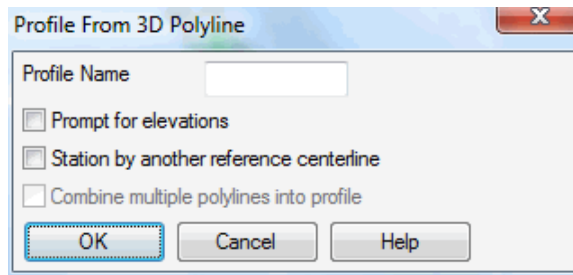
Pulldown Menu Location: Profiles

Keyboard Command: profedit

Prerequisite: None

Profile from 3D Polyline

To create a profile (.PRO), Profile from 3D Polyline uses X-Y distances between the points of a 3D polyline for sequential stations and the Z values at these points for profile elevations. In the options dialog, Profile Name is an optional description for the profile. The Prompt For Elevations option will prompt for the elevation at each polyline vertex to use for the profile instead of using the polyline elevations. The Station By Another Reference Centerline method locates the station for each polyline vertex along a reference centerline and uses this reference station instead of the polyline distance for the profile stationing. The reference centerline can be defined by another polyline or centerline file (.CL). When using the reference centerline, the Combine Multiple Polylines Into Profile option allows you to select multiple 3D polylines and put the data into a single profile. For example, you can use these two options to create a profile of curb elevations with road centerline stationing by selecting multiple 3D curb polylines and the road centerline as the stationing reference.



After the options dialog, the program prompts for the .PRO file to create and then the 3D polyline to process.

Prompts

Profile From 3D Polyline dialog

Profile File to Write dialog Specify a profile file name to create

Select polyline to profile: *pick a 3D polyline*

Created 72 data points for profile C:\sample\abc.pro

The new profile is then stored.

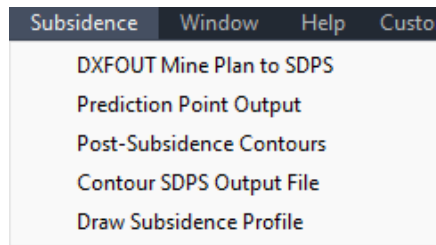
Pulldown Menu Location: Profiles

Keyboard Command: pro3dpl

Prerequisite: A 3D polyline

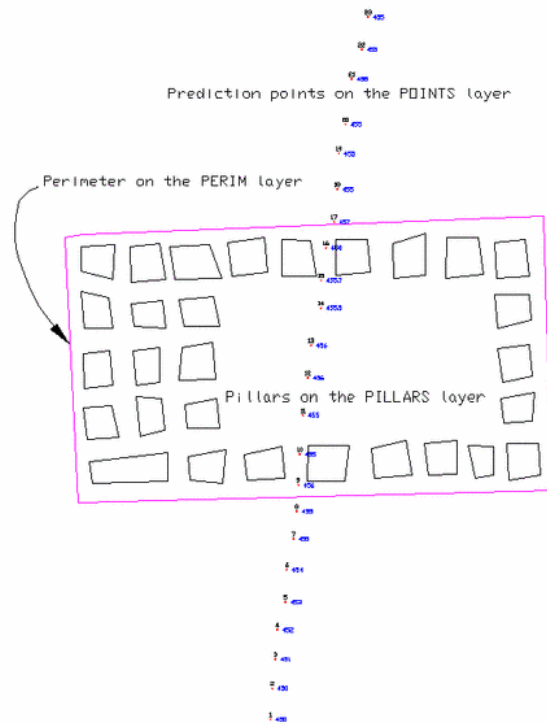
Subsidence Menu

The subsidence routines work with the SDPS (Subsidence Deformation Prediction System) program. SDPS was written by Virginia Tech. Carlson Software is the exclusive reseller for SDPS.



DXFOUT Mine Plan to SDPS

This command creates a DXF file to import the mine plan pillars and perimeter to the Subsidence Deformation Prediction System program. The pillars and perimeter polylines created in Carlson with the layers PILLARS and PERIM. The PERIM layer is written to the DXF file as PANELS for SDPS. This routine provides the user the ability to specify the direction the pillar polygons are drawn. SDPS requires the pillars to be drawn in the counter-clockwise direction. For additional information on this procedure, refer to the SDPS manual.



Prompts

DXF File for SDPS Enter a file name. The default directory is the SDPS directory defined in Configure .

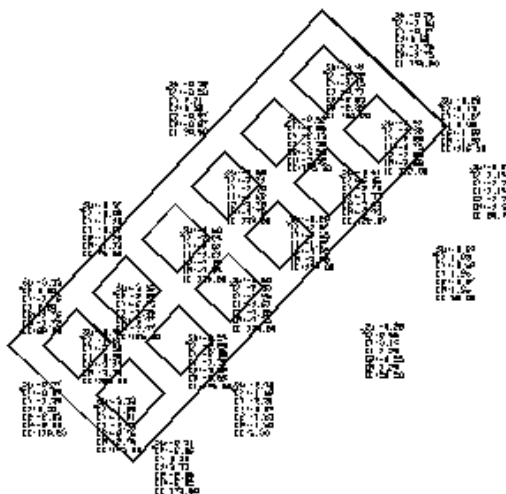
Pulldown Menu Location: Subsidence

Keyboard Command: sdpsout

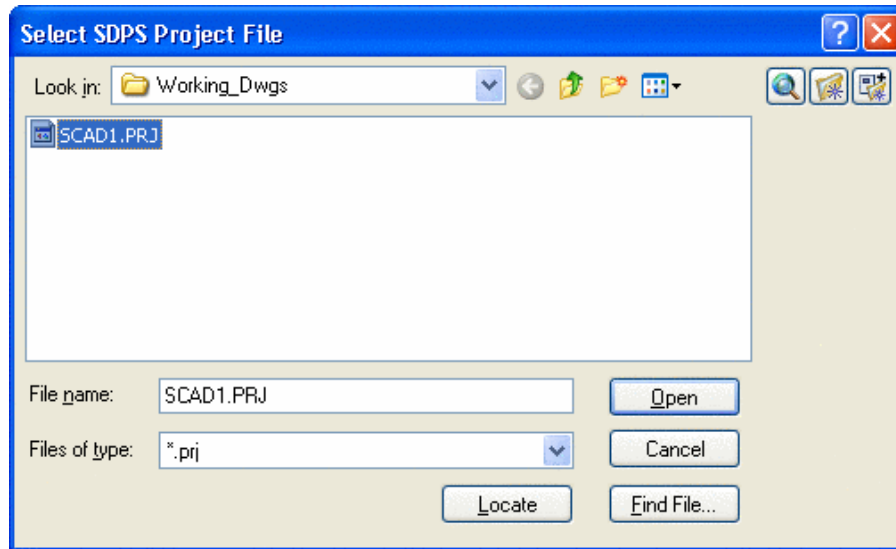
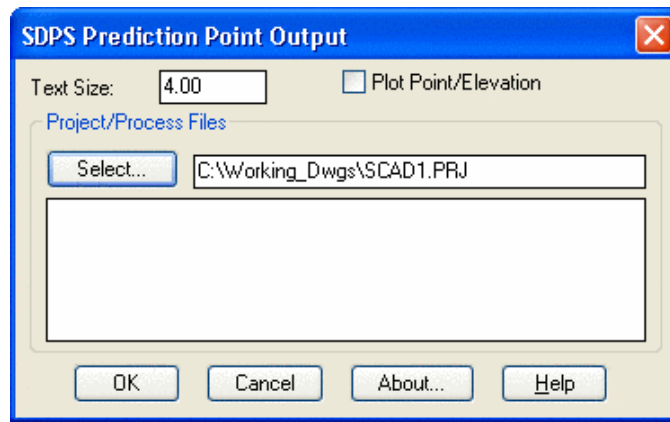
Prerequisite: Polylines in the PILLARS and PERIM layers and prediction points on the POINTS layer, if prediction points are input.

Prediction Point Output

This command draws text at each prediction point of the subsidence, strain and other SDPS deformation output values. From the dialog, click the Select button to choose an SDPS project file. The program then reads the values from all the output .dat files and labels the values next to the prediction points.



Prediction Point Text Output



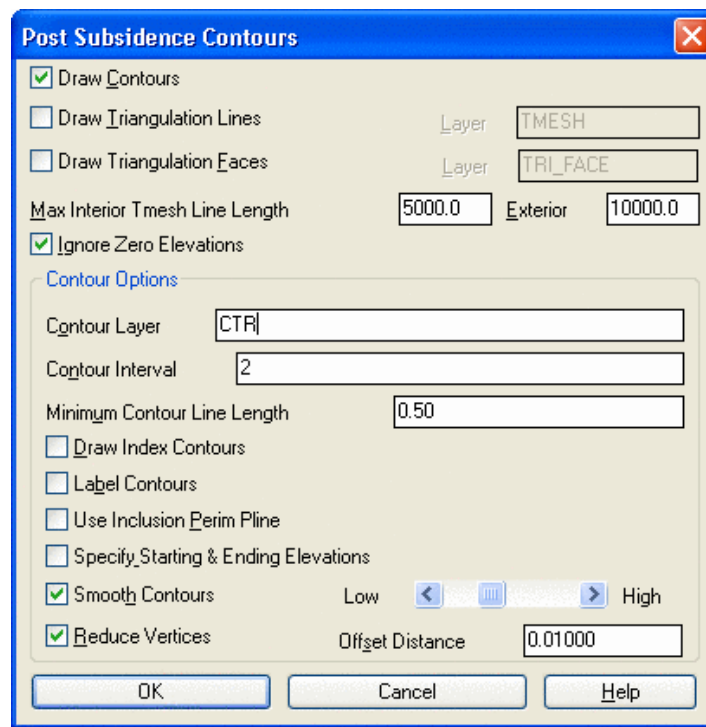
Pulldown Menu Location: Subsidence

Keyboard Command: sdpsout

Prerequisite: SDPS output files

Post-Subsidence Contours

This command creates surface contours after accounting for subsidence. The surface is defined by prediction points which are POINT entities with the elevation of the surface. These points are then exported to the Subsidence Deformation Prediction System program which calculates the subsidence at each point. This subsidence output from SDPS is written to a DAT file in the SDPS directory. Post-Subsidence Contours then subtracts the subsidence values from the file from the elevation of the prediction points.



Prompts

Triangulate & Contour dialog

Select the prediction points.

Select objects: *pick the points*

Select SDPS Output file Choose the subsidence output file.

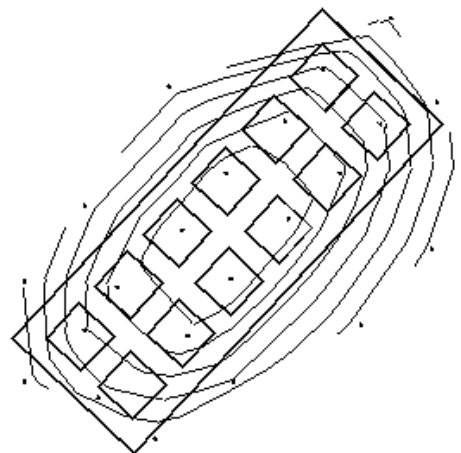
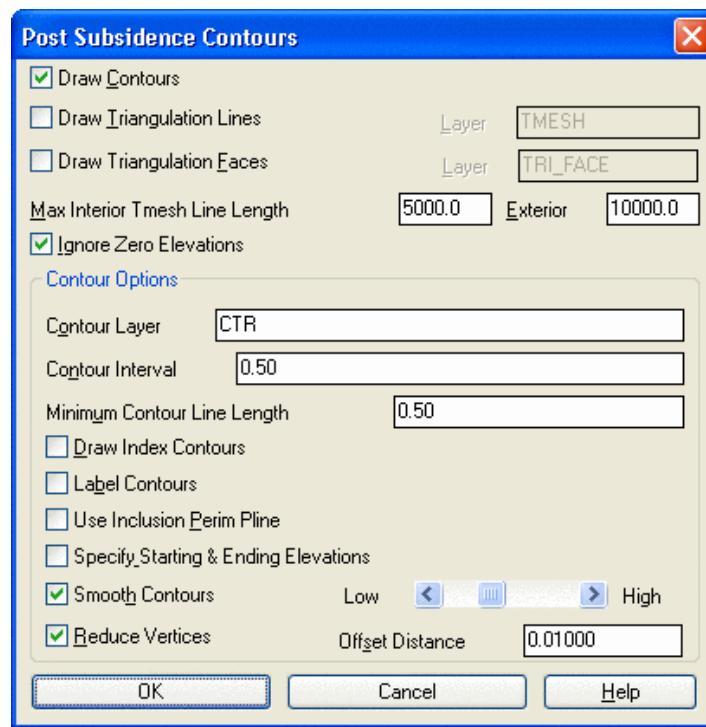
Pulldown Menu Location: Subsidence

Keyboard Command: sdpstr

Prerequisite: prediction points and subsidence output file

Contour SDPS Output File

This command creates contours of the values in SDPS output DAT files. The output files can represent subsidence, strain and any other SDPS deformation output.



Contours of subsidence ".dat" file

Prompts

Triangulate & Contour dialog

Select SDPS Output file Choose the output file.

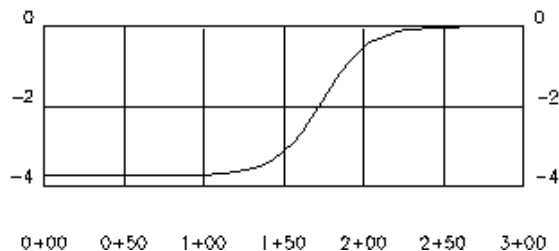
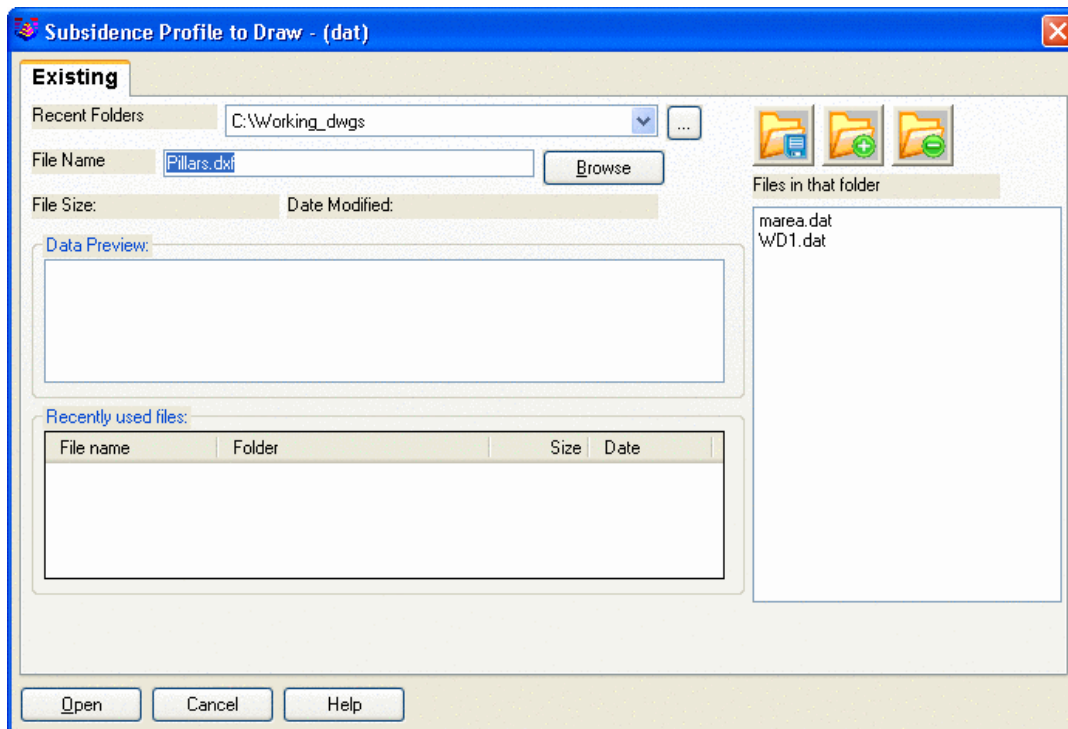
Pulldown Menu Location: Subsidence

Prerequisite: SDPS output file

Keyboard Command: sdpctr2

Draw Subsidence Profile

This command draws a subsidence profile created by Profile Function Formulation in SDPS. The profile data is stored in a .dat file that is created with the Output>Export option in SDPS.



Prompts

Subsidence Profile to Draw Choose the .dat file from SDPS.

Draw Profile dialog You may want to change the vertical scale to a smaller number to exaggerate the vertical.

Bottom elevation of profile grid <-4.0>: *press Enter*

Pick Starting point for axis <5000,5000>: *pick a point in a clear area of the screen*

Pulldown Menu Location: Subsidence

Keyboard Command: sdpsprof

Prerequisite: Subsidence output file

Suggested Standards

Adjacent Mining

Layer: ADJACENT

Color: Blue (5)

Line Type: Continuous

Line Width: 0

Text Height: 20

Text Style: Romanc

Text Font: Romanc

Barrier Lines

Layer: BARRIER
Color: Yellow (2)
Line Type: Dashed
Line Width: 0
Text Height: 20
Text Style: Romanc
Text Font: Romanc

Bench Marks

Layer: BENCHMARKS
Color: Cyan (4)
Text Height: 6.5
Text Style: Romans

—|— BM 2418
| EL. 1065.31
V18—P13

Text Font: Romans

Border

Layer: BORDER
Color: Magenta (6)
Line Type: Continuous
Line Width: 10

Bottom of Coal Elevations

Layer: BOTELX
Block Reference - "ELEV-X" BOTELEV
Color: Cyan (4)
Text Height: 6
Text Style: Romans

Text Font: Romans X 991.48

Coal Sections

Layer: SAMPLE (Circle) COALSECTIONS
Color: Cyan (4)
Text Height: 6
Text Style: Romans
Text Font: Romans

○ TC-65
TR-2
BR-12

Contour Lines

Layer: CONTOURS
Color: Red (1)
Line Type: Continuous
Line Width: 5 ft. intervals = 0, 10 ft. intervals = 2
Text Height: 20
Text Style: Romans
Text Font: Romans

Control Points

Layer: CONTROL
Color: White (7)
Text Height: 8
Text Style: Romanc
Text Font: Romanc

Coordinates

Layer: COORD
Color: Red (1)
Point Mark Ht.: 100
Text Height: 25
Text Style: Romanc
Text Font: Romanc

Drillholes

Layer: DRILLHOLE
Color: 14
Text Height: 15 on Hole I.D., 11.25 on all others
Text Style: Romanc
Text Font: Romanc

Creeks

Layer: CREEKS
Color: Blue (5)
Line Type: Phantom
Line Width: 0
Text Height: 16
Text Style: Romanc
Text Font: Romanc

Falls

Layer: FALLS
Color: Yellow (2)
Text Height: 7.5
Text Style: Romanc
Text Font: Romanc

Fault Lines

Layer: FAULT
Color: Green (3)
Line Type: Continuous
Line Width: 4
Text Height: 10
Text Style: Romanc
Text Font: Romanc

Grid Lines

Layer: GRID
Color: 94
Line Type: Continuous
Line Width: 0
Text Height: 20
Text Style: Romanc
Text Font: Romanc

Legend

Layer: LEGEND

Color: White (7)

Line Type Scale

Command: LTSCALE

Scale: 100

Mine I.D.

Layer: MINEID

Color: White (1)

Text Height: 10

Text Style: Romanc

Text Font: Romanc

Mine Name

Layer: COMPANYNAME

Color: Blue (5)

Text Height: 15

Text Style: Romanc

Text Font: Romanc

Big Mountain
Leaf Mine
M.S.H.A. I.D. NO. 12345
98760

Mining Limits

Layer: LIMITS

Color: 30

Line Type: Border

Line Width: 4

Text Height: 20

Text Style: Romanc

Text Font: Romanc

N - E Line

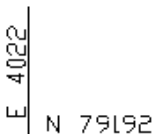
Layer: N-E-LINE (Goes into the current layer)

Color: Cyan (4)

Text Height: 8

Text Style: Romanc

Text Font: Romanc



E 4022
N 79192

North Arrow

Layer: NORTHARROW (Goes into the current layer when inserted)

Color: Varies

Notes

Layer: NOTES

Color: Red (1)

Text Height: 10

Text Style: Romans

Text Font: Romans

Oldworks

Layer: OLDWORKS

Color: Blue (5)

Line Type: Continuous

Line Width: 0

Text Height: 20

Text Style: Romanc

Text Font: Romanc

Outcrop Lines

Layer: CROP

Color: Red (1)

Line Type: Continuous

Line Width: 6

Text Height: 20

Text Style: Romanc

Text Font: Romanc

Outline

Layer: OUTLINE

Color: Blue (5)

Line Type: Continuous

Line Width: 0

Perimeters

Layer: PERIM

Color: Magenta (6)

Line Type: Continuous

Line Width: 0 (Required for extraction ratio calculation with Advanced Mine Module)

Pillars

Layer: PILLARS

Color: White (7)

Line Type: Continuous

Line Width: 0 (Required for extraction ratio calculation with Advanced Mine Module)

Projections

Layer: PROJECTIONS

Color: Green (3) - Belt Entry, Cyan (4) - All Others

Line Type: Continuous

Line Width: 2.5 - Belt Entry, 0 - All Others

Projections Arrows

Layer: PROJVENARROWS

Color: Cyan (4)

Projection Distances

Layer: PROJDISTANCES

Color: Cyan (4)

Text Height: 8

Text Style: Romans

Text Font: Romans

Projection Stoppings

Layer: PROJSTOPPINGS

Color: Cyan (4)

Property Lines

Layer: PROPLINE

Color: Yellow (2)

Line Type: Phantom

Line Width: 5

Text Height: 20

Text Style: Romanc

Text Font: Romanc

Second Mining

Layer: SECONDMINED

Color: Yellow (2)

Line Type: Continuous

Line Width: 0

Section Name

Layer: SECTIONNAME

Color: White (1)

Text Height: 20

Text Style: Romanc

Text Font: Romanc

Section I.D.

Layer: SECTIONID

Color: Cyan (4)

Text Height: 10

Text Style: Romanc

Text Font: Romanc

Spads

Layer: SPAD-NO

Block Reference - "Spad2"

Color: Green (3)

Text Height: 8

Text Style: Romans

Text Font: Romans

Take-Up Dates

Layer: TAKEUPDATE

Color: Cyan (4)

Text Height: 6 on Monthly and 24 on Quarterly (Pillar Section)

Text Style: Romanc

Text Font: Romanc

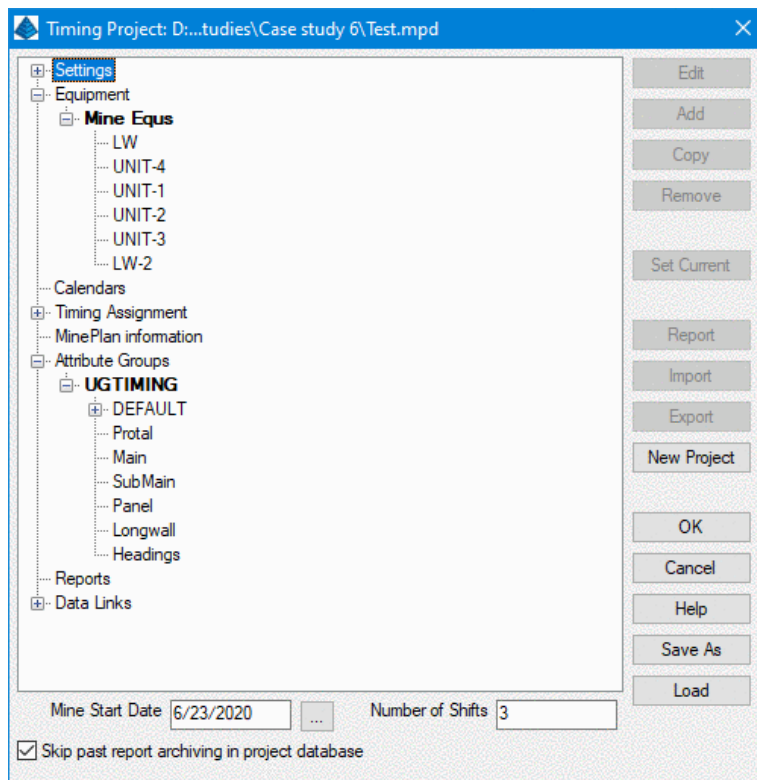
Case Studies

Case Study #6: Underground Mine Layout and Timing

Underground Project Manager

(Underground Mining module > Underground pull-down menu > Timing Project Manager Command)

The Underground Timing Project Manager (MPD) file should be defined before starting with Underground Mine Layout and Timing using the Underground Timing Project Manager command under Underground menu. It allows user to define Equipment, Timing Calendar, Panel Attributes as well as other essential parameters to be used in the timing.



Mine Layout Options

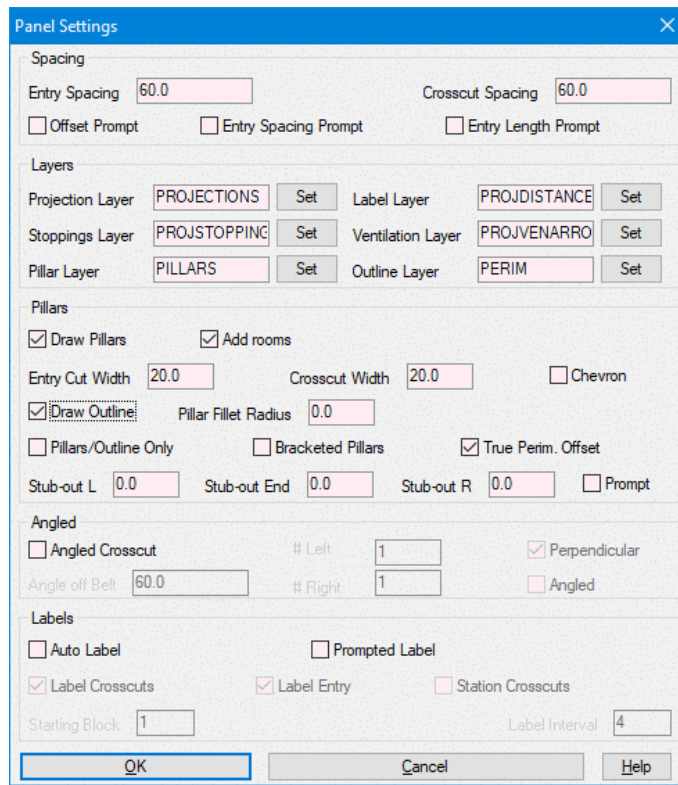
Using the Advanced Projections command in the Underground Mining Module, there are several ways the user can generate mine projections. These different techniques will be based upon personal preference and management's direction. Whether or not to include the centerlines, pillars, and perimeters on the map are up to the user. The Underground Mining Module timing routines require that some information is kept on specific layers. The pillars must be drawn on the PILLARS layer and the perimeter of each section must be drawn on the PERIM layer. These are the defaults set in the Advanced Projections option under the Underground Mining Module.

Choose the Advanced Projections command under Works. The first three prompts are:

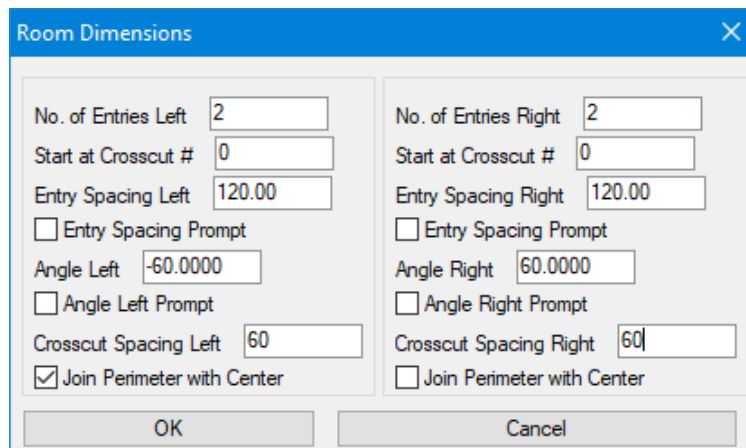
(Underground Mining module > Works pull-down menu > Advance Projections Command)

1. Starting point of the belt
2. Number of entries to the left of the belt
3. The number of entries to the right of the belt

After responding to the initial prompts the Panel Settings dialog box appears. This dialog box allows the user to specify how the mine projections will be drawn. Several different variations are possible. Entry spacing and length can be varied, separately or together.



A nice feature is drawing the rooms on one or both sides of the panel at the same time the panel is drawn. The rooms perimeter on the left or right side can also be joined to the panel perimeter so the timing blocks will match for the panel and the rooms. Join perimeter with center can only be selected once because Carlson doesn't allow you to join all 3 rooms and entry panels together.

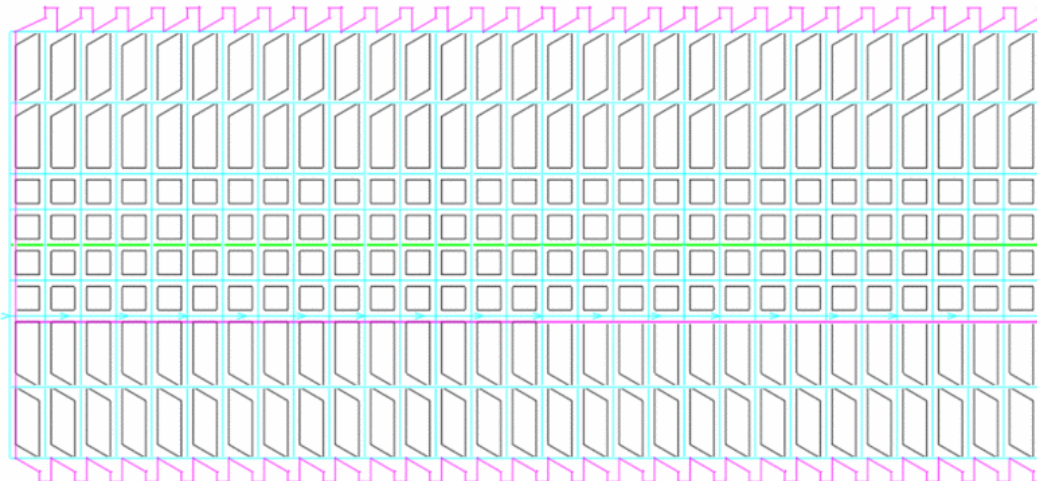


In addition to the configuration of the pillars, the user has the option of what to display, pillars, centerlines, perimeter, as well as chamfer, angled blocks, bracketed pillars, stoppings, and ventilation.

Mine Layout Options



As you can see there are several options to present mine projections. Below is the mine projection with angled room and stub-outs created with above dialogues. The stub-out is automatically added when rooms are created.



Once the panels have been laid out, they have to be "placed", or have extended entity data attached to them, so that they will relate to each other. Carlson has three different methods of placing panels: Place Panel, Pick & Place, and Auto Place by Text. Of these three techniques only Place Panel builds a network topology, or precedence relationship, between the panels as they are placed. With the other two techniques, Pick & Place and Auto Place by Text, there is no relationship built as they place the panels. This allows the user to actually mine through panels that overlay each other, which could be helpful in laying out the mine through overlying strata.

Place Panel

(Underground Mining module > Underground pull-down menu > Place Panel Command)

To place panels, the panels must be previously drawn with the perimeter drawn in the PERIM layer and the pillars, which are optional, drawn in the PILLARS layer. The program subtracts the sum of the areas of the individual pillars in the PILLARS layer from the area of the perimeter drawn in the PERIM layer to calculate the extraction

ratio. The extraction ratio is then applied to the perimeter area to calculate the tonnage in the panel.

When the user selects Place Panel the first prompt is:

Use Quick Place mode (yes/<no>)? Quick place mode is an option that lets the user place panels without entering specific information in the panels other than the panel name. It is a way to get the panels placed quickly. If the user has drawn the pillars and does not intend to retreat mine, the Quick Place mode may be the best way to get the panels placed.

If the user chooses the Quick Place mode the following prompts appear:

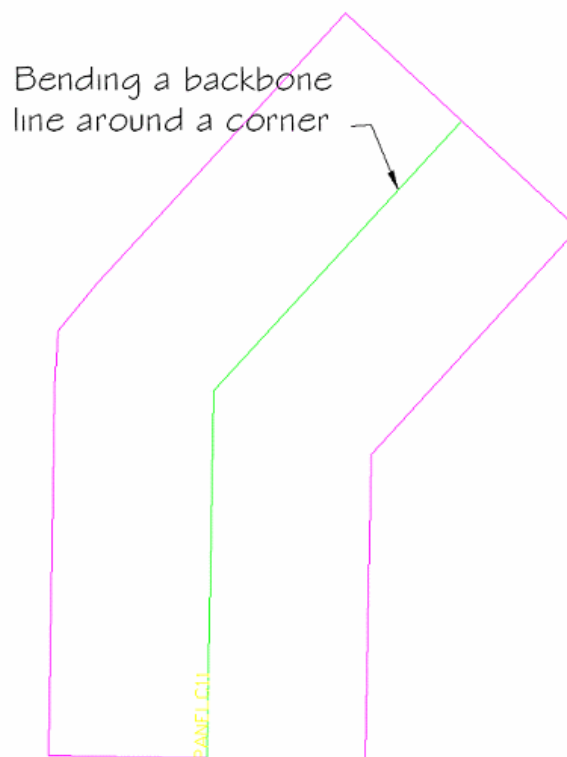
Start new mine plan? (yes/<no>)? The user will always respond with yes for the first panel. Next, as long as the panels being placed are adjacent, the response is negative. Panels that are part of the same mine plan, but are physically separated can be placed as new mine plans and connected later.

The next prompt is: **Direction of panel is defined by (<Perim>,Text, Pick, Segment)?**

- **Perim** - The user defines the starting side of the polygon and the direction of mining is perpendicular to the first side of the polygon. If the panel is laid out using the Advanced Projections in the Underground Mining Module the perimeter is automatically drawn to match up with the requirements for the Perim option.
- **Text** - The direction of mining through the polygon can be set by the direction of the text.
- **Pick** - The user chooses two points to set the direction of mining in the reverse order of mining. The first point is picked inside the polygon, the second point is picked on the beginning side of the polygon. The direction is from the beginning side through the first point picked inside the polygon.
- **Segment** - Pick the starting side of the polygon and any side that parallels the direction of mining of the panel.

When the direction of mining has been set. The program prompts for the name of the panel, offering a name that can be accepted or overwritten. When the name is entered, the program draws the green "backbone" line according to the direction of mining and places the name of the panel at the beginning of the line. The user clicks the right button to complete this process for mining straight through a panel.

To mine around a corner the user left picks the mouse button and repeats the above steps for setting the direction.



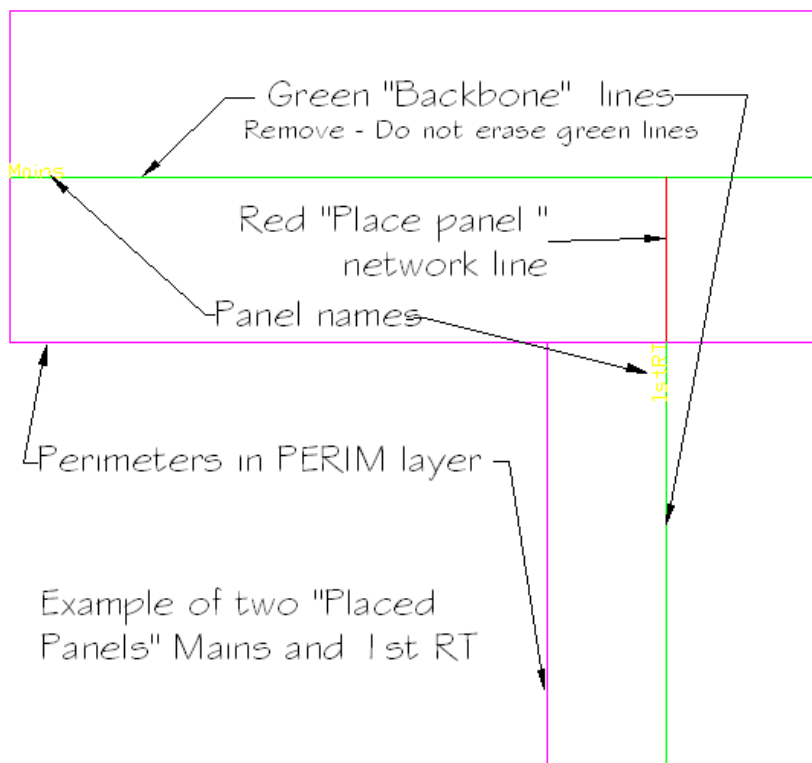
Once the panel has been placed, the program prompts the user for the entry to branch from. The program is asking for the point along the "backbone" line to start the next panel. Once the user selects the starting point for the next panel the process repeats.

Using the Quick Place mode does not give the user access to the Edit Section Data dialog box during the panel placing process. If the pillars are not drawn when using the Quick Place mode the user will have to return to the Edit Section Data dialog box to enter the extraction ratio after placing the panels. Most users prefer to enter this information when the panel is being placed, so they usually do not use the Quick Place mode.

If the user chooses not to use the Quick Place mode there are two additional panel direction options available: Azimuth and Pick internal point.

The screenshot shows the 'Edit Section Data' dialog box with the following fields and values:

- Attr. Group: DEFAULT
- Panel name: ANEL635
- Panel start: [Empty]
- Easting: 3621.1986
- Northing: 5020.1703
- Azimuth: 89°22'38"
- Width: 20
- Length: 60.00
- Buttons: Pick
- Longwall Panel
- Entries: Left: 1, Right: 1
- Width: 20.00
- Spacing: 0.00
- Calculated extraction: 3.00
- Retreat extraction: [Empty]
- Crosscuts: Number: 1
- Width: 20.00
- Spacing: 60.00
- Left Cut Angle: 90.00
- Right Cut Angle: 90.00
- Precedence: Advance, Retreat
- Buttons: Edit, Screen Pick
- Prompt for coal qualities
- Prompt for retreat options
- Owner Name: [Empty]
- Level Layer Suffix: [Empty]
- Advance Difficulty Factor: 1.169
- Retreat Difficulty Factor: 1.0
- Buttons: Select file, Select file
- Buttons: Difficulty Calculator, OK, Cancel



Pick and Place

(Underground Mining module > Underground pull-down menu > Pick and Place Panel Command)

Pick and Place places and times the panel as it is placed, drawing the timing blocks when the panel is placed. In addition to timing as it goes, this option allows the user to test the schedule before placing the panel in the mining sequence, providing real time feedback while scheduling the mine. This technique does not automatically build precedence between panels automatically. Precedence can be added to the panels in the precedence part of the Edit Section Data dialog box. Once all of the individual panels have been placed, the user runs the schedule to get the complete mine timing report.

Define Timing Step

Panel Settings

Panel name: 2 Panel start: Owner Name:

Mine Start Date: 7/24/2020 Number of shifts: 02 Retreat recovery: 0.0000 Entry Width: 20.0

Attr. Group: DEFAULT Panel attributes Level Layer Suffix: Prompt for retreat options:

Advance Diff. Factor: 1.169 Select file Extraction: 0.53627

Retreat Diff. Factor: 1.0 Select file Extraction: 0.00000

Equip. Name: LW FREE

Advance Retreat Development Results

Portion To Mine

Whole perimeter 20.000000

Distance Month

Period To Date

Moving time (shifts):

Test Report Options OK Cancel

Panel Settings

Panel name: 2 Panel start: 7/28/2020 Owner Name: _____

Mine Start Date: 7/28/2020 Number of shifts: 02 Retreat recovery: 0.0000 Entry Width: 20.0

Attr. Group: DEFAULT Panel attributes Level Layer Suffix: _____ Prompt for retreat options

Advance Diff. Factor: 1 Select file Extraction: 0.53627

Retreat Diff. Factor: 1.0 Select file Extraction: 0.00000

Equip. Name: UNIT-1 7/28/2020

Advance Retreat

Portion To Mine

Whole perimeter

Distance: 20.000000

Period: Month

To Date: _____

Moving time (shifts): _____

Development Results

Date Start	7/28/2020
Date Finish	8/16/2020
Crew	UNIT-1
Panel Name	2
KEY (Tons)	114406.983292
Non-Key (Tons)	0.000000
Area mined	875138.310974
Linear adv.	23834.788186
Difficulty	1.000000
Density	80.000000

Buttons: Test Report Options OK Cancel

Similar to Place Panel, Pick & Place Panel has the following options for specifying direction of mining. They are defined above:

- **Perim (default)**
- **Text**
- **Pick**
- **Auto**
- **Segment**

Once the direction has been set, the Define Timing Step dialog box appears.

- **Panel Name** - is a required field. The program will offer a name which can be overwritten.
- **Panel Start** - Is an optional field
- **Owner** - is an optional field, if filled out indicates the property owner for the entire panel and will be available in production reports.
- **Starting Date** - is not optional for the first panel assigned to any unit. After the first panel this field is optional and will be calculated.
- **Number of shifts** - is a required field, and overwrites the setting in the equipment definition file if the shifts are less than the equipment file.
- **Retreat Recovery** - If recovery is different on retreat than advance it can be inserted here.
- **Entry width** - if no pillars are drawn in the perimeter the default is 20 feet. Otherwise, it is picked up from the pillar layout.
- **Panel Attributes** - can be anything the user wants to use such as; ash, sulfur, btu, etc. The user can use fixed values for these or refer to a grid.
- **Prompt for Retreat Options check box** - if this is checked it indicates the shape of the retreat perimeter differs from the advance perimeter. If checked, a separate set of prompts will appear giving the user the option to specify the shape of the retreat perimeter and extraction ratios for the pillars mined inside and outside the advance perimeter.
- **Advance Difficulty Factor** - the difficulty factor used on the advance can differ from that used on the retreat. Normal difficulty is (1), greater than one reduces productivity, less than one speeds up production. The difficulty factor can be a factor or a grid.
- **Retreat Difficulty Factor** - can be specified separate from the advance rate.

- **Equipment Name** - is selected from the defined equipment list.
- **Advance Extraction Ratio** -the extraction ratio is the sum of the mined out area outside the pillars divided by the area of the perimeter. This factor is usually 1.0 for longwall panels, but for continuous miner sections is usually near .50.
- **Retreat Extraction Ratio** - this factor is percentage of the remaining pillars left on the advance. This factor can be up to 100 percent for the remaining pillars, as opposed to a number less than 1 minus the advance extraction ratio.
- **Advance/Retreat radio button** - if the user left clicks in the panel after it has been assigned on the advance to schedule it the radio button will gray out the advance button highlighting retreat button.
- **Portion to mine** - options include:
 - **Whole Perimeter** - to mine the entire perimeter.
 - **Distance** - the distance displayed in the box to the left of the radio button is the distance to the spot where the user left clicked in the perimeter to Pick & Place the panel. This is useful if the user want to know the distance to a point inside the panel. When this option is selected, a pinpoint is placed at the distance selected, allowing the user to schedule other panels.
 - **Moving time (shifts)** - the move time in shifts the panel is delayed prior to starting.
 - **Test** - this option lets the user try the settings and get feedback before applying them.
 - **Report Options** - this button gives the user two reporting options, one for an abbreviated timing report and the other for the full report.
 - **OK** - applies the selection and schedules the panel showing the timing.
 - **Cancel** - breaks out of the procedure and returns the user to the command prompt.

Auto Place Panel by Text

(Underground Mining module > Underground pull-down menu > Auto Place Panel by Text Command)

The Auto Place Panel by Text command gives the user the ability to place panels by drawing text in the panel perimeters on the PANELNM layer. The requirement for drawing the perimeter in the PERIM layer is the same. The prompts are set out below.

Command: auto_place

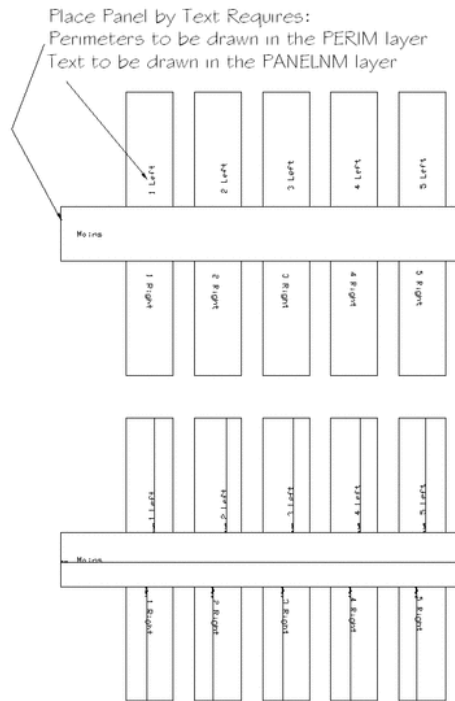
Assign zero retreat extraction (Yes/<No>)?

Advance difficulty <1.0> :

Retreat difficulty <1.0> :

Retreat recovery ratio for remaining pillars <1.0> :

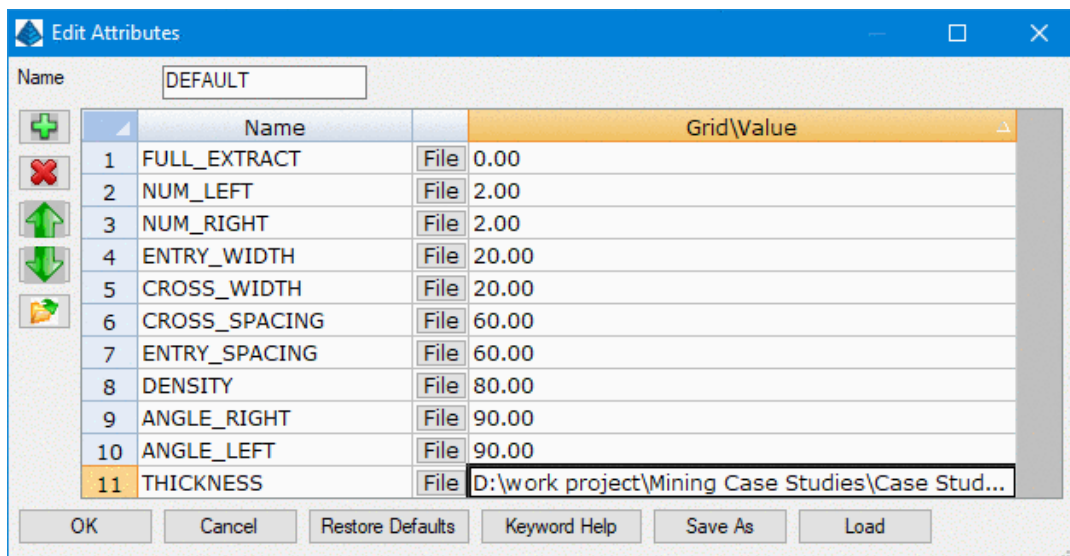
Entry width <20.0> :

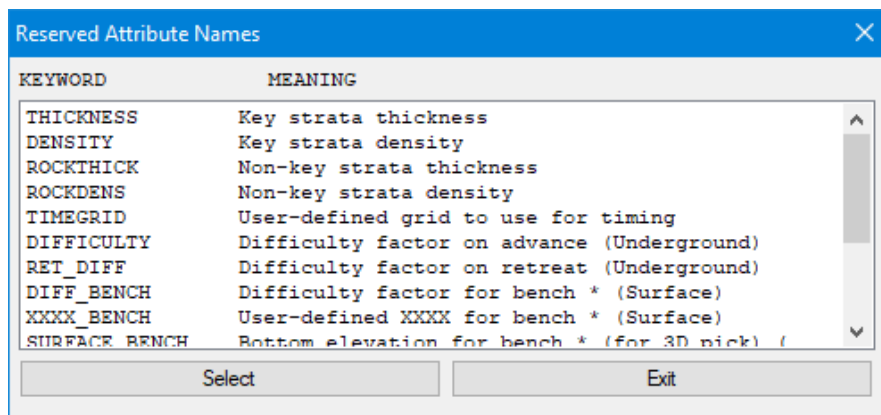


Panel Attributes

(Underground Mining module > Underground pull-down menu > Edit Panel Attributes Command)

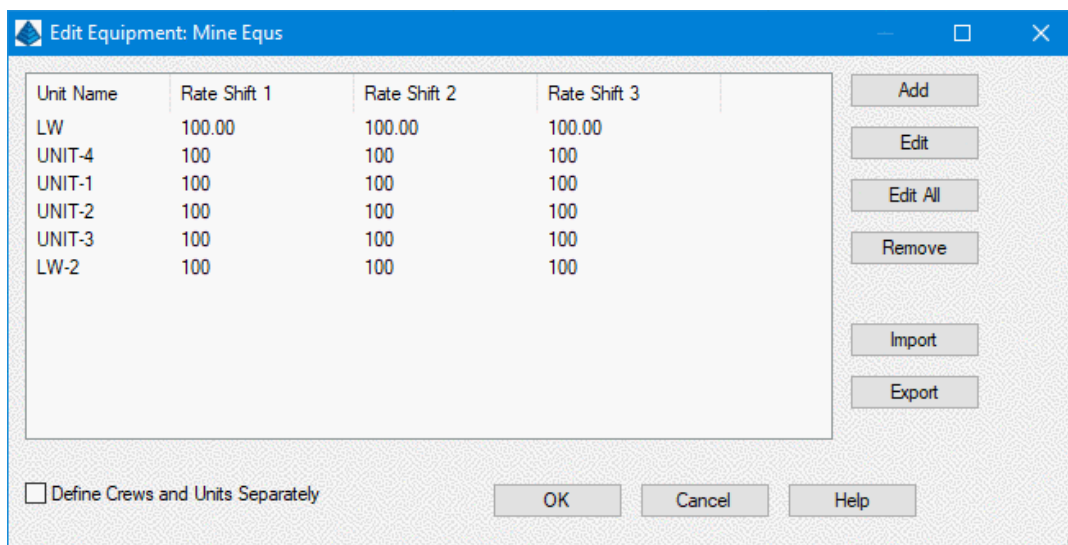
The default panel attributes option lets the user assign attributes by value or grid. Certain key or reserved words are available to the user that Carlson recognizes. Choose the Keywords Help button to see them:





Define Equipment

Define Equipment is where the user adds and edits the production rate of the equipment, it can be accessed from Underground Project Manager. Production rates are input for the entire spread or section. This rate can be modified as a function of time, mining height, or strip bench. The Rate/Shift, Hours/Shift, and Rate/Hour back-calculate, allowing the user the ability to fine tune the production rate. If the toggle to "Define Crews and Units" is on it gives user an ability to define Labor Crews and Units (combination of Crew and Equipment) separately.



Equipment Production Rate

Name:

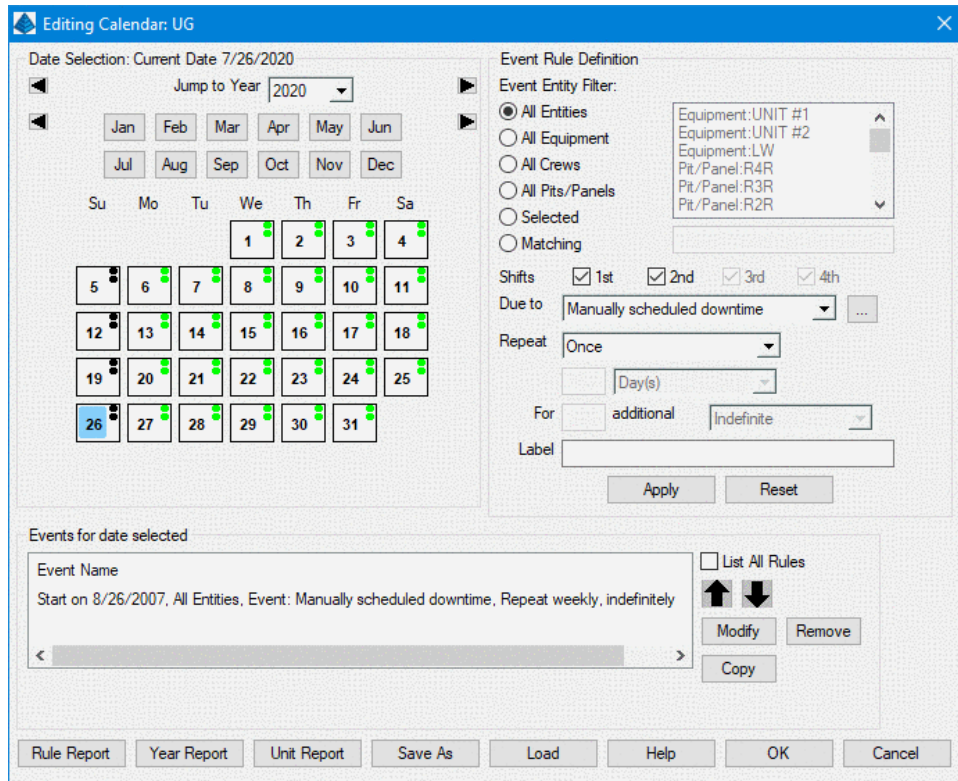
Unit of Production
 Tons Distance Cubic yards Linear ft of Advance

	Shift 1	Shift 2	Shift 3	Shift 4
Advance Rate/shift	<input type="text" value="140.00"/>	<input type="text" value="140.00"/>	<input type="text" value="140.00"/>	<input type="text" value="600.00"/>
Retreat Rate/shift	<input type="text" value="100.00"/>	<input type="text" value="100.00"/>	<input type="text" value="100.00"/>	<input type="text" value="600.00"/>
Hours/shift	<input type="text" value="8.00"/>	<input type="text" value="8.00"/>	<input type="text" value="8.00"/>	<input type="text" value="8.00"/>
Advance Rate/hour	<input type="text" value="17.50"/>	<input type="text" value="17.50"/>	<input type="text" value="17.50"/>	<input type="text" value="75.00"/>
Retreat Rate/hour	<input type="text" value="12.50"/>	<input type="text" value="12.50"/>	<input type="text" value="12.50"/>	<input type="text" value="75.00"/>
Availability	<input type="text" value="1.000"/>	<input type="text" value="1.000"/>	<input type="text" value="1.000"/>	<input type="text" value="1.000"/>
Advance Min. Height	<input type="text" value="7.00"/>	Advance Max. Height		<input type="text" value="9.50"/>
Retreat Min. Height	<input type="text" value="7.00"/>	Retreat Max. Height		<input type="text" value="9.50"/>
Main Maintenance Freq.	<input type="text" value="6500.00"/>	Delay		<input type="text" value="2.00"/> shifts
Routine Maintenance	<input type="text" value="2500.00"/>	Delay		<input type="text" value="5.00"/> shifts
Extraction factor	<input type="text" value="1.0000"/>	Cost per hour		<input type="text" value="0.00"/>

The user can specify the minimum and maximum mining heights. Where mining occurs below the minimum mining height, out of seam dilution is added to make up the desired minimum height. When the maximum mining height is selected key strata is combined with non-key material where the key strata exceeds the maximum mining height. A cost per hour can be input and it will be multiplied by the hours worked. Maintenance frequency is scheduled in the same units as the Rate/Shift. The delays apply to the entire spread or section in shifts for both Routine and Main Maintenance Frequencies. The recovery factor is designed to be used with elliptical cross-sectioned entries developed with boring machines.

Calendar

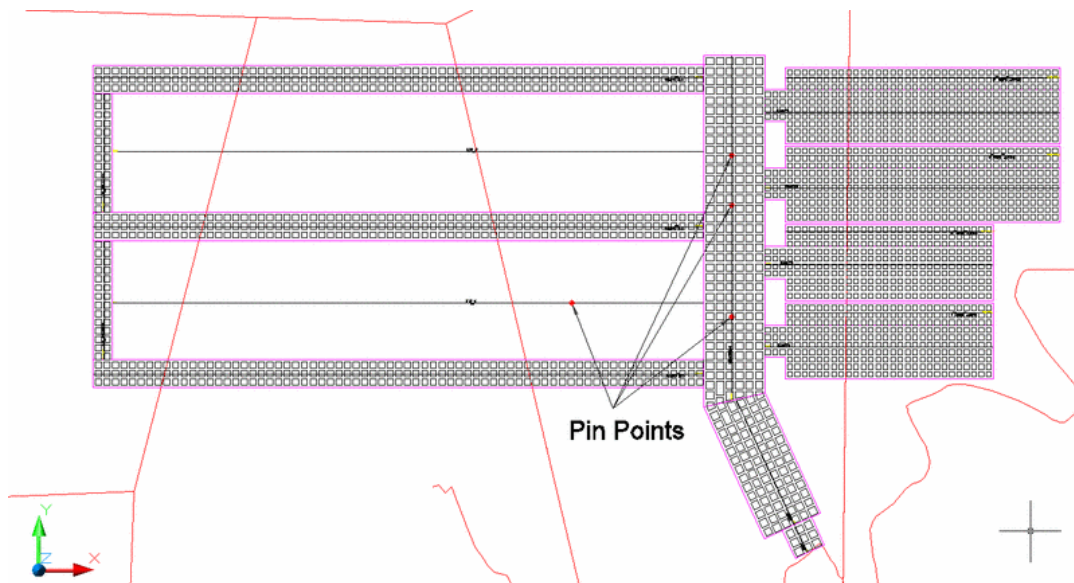
The Editing Calendar is a quick and simple way to schedule a production Equipment or Unit (group of a Equipment and Labor Crew working on it). This should be thought of as an equipment, as opposed to a crew scheduling technique. Equipment needs to be added before the calendar is applied. The calendar can be applied to a single unit or all of the equipment. To apply the calendar to several units select the calendar and the unit and Apply the calendar to the unit. Once the calendar is created two different reports can be generated. The first report is the annual unit calendar and the second is a yearly report of what is working and when.



Pin Points

(Underground Mining module > Underground pull-down menu > Add Pin Point Command)

Pin points are break points set in a panel to indicate where the mining temporarily stops to facilitate mining in other areas. When mining resumes in the panel with the pin point it resumes at the pin point. Pin points allow the user to layout fewer panels.

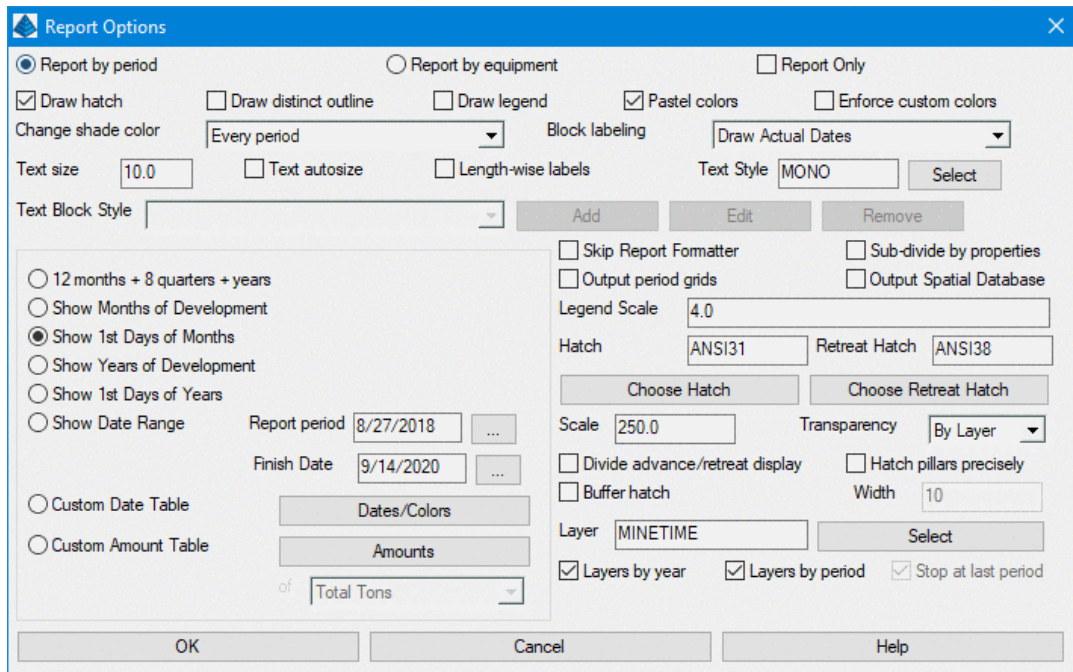
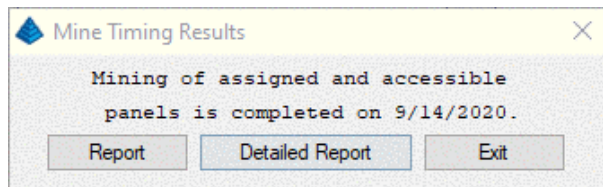
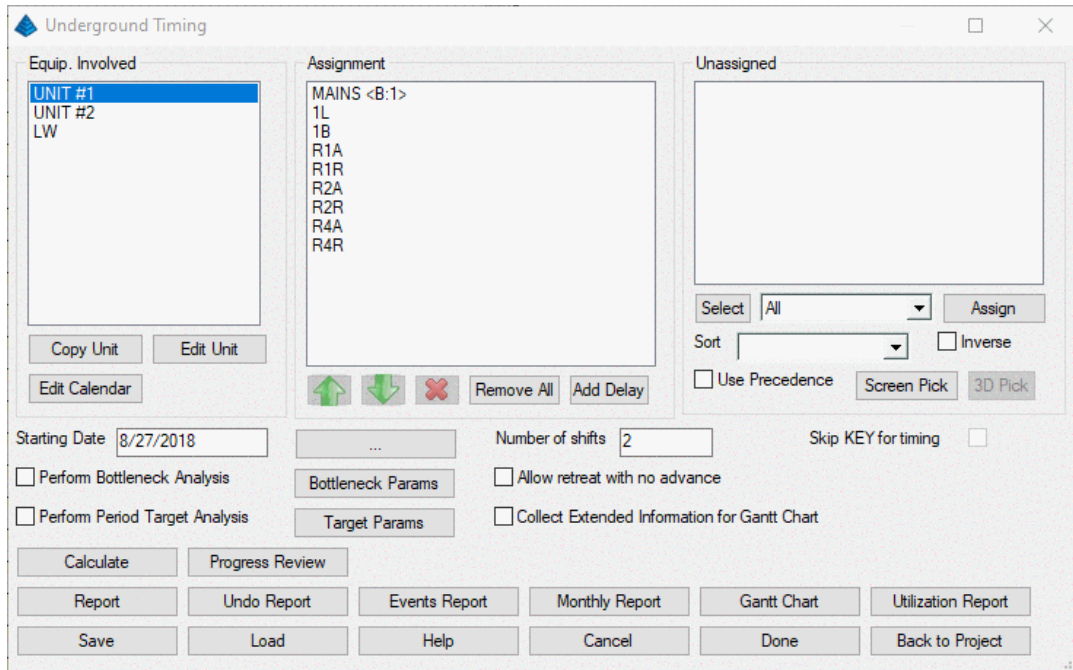


Underground Timing

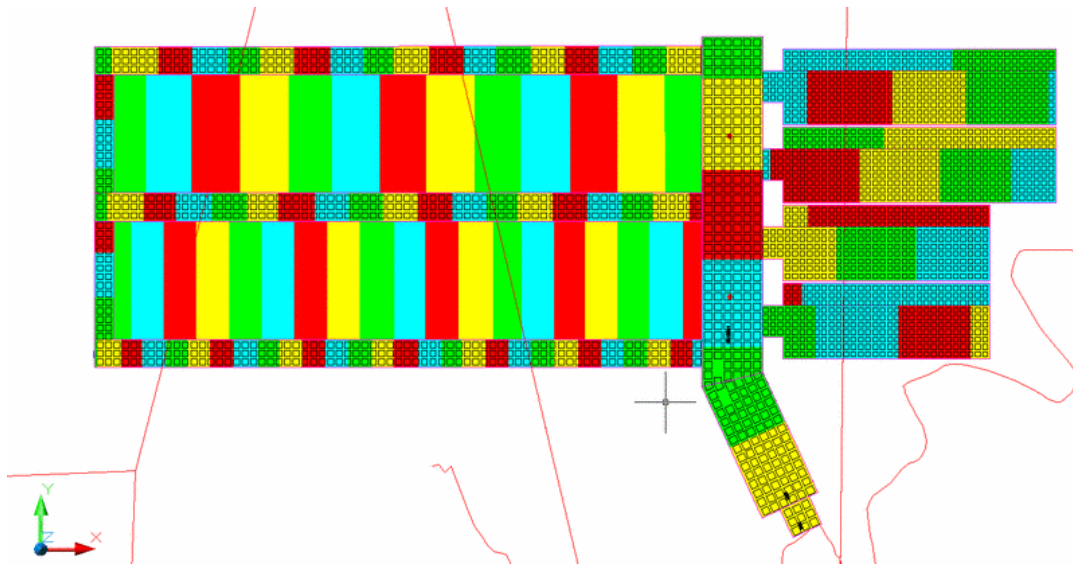
(Underground Mining module > Underground pull-down menu > Underground Timing Command)

The Underground Timing option is selected after the panels have been placed, and the equipment and calendar have been defined. Panels are assigned to equipment in the sequence to be mined. Panels can be selected by name from the unassigned list or screen picked. The unassigned panel list can be sorted by name or direction of mining. Delays can be scheduled between panels in days or shifts. The starting date and number of shifts per day are assigned in the designated boxes. Once the run is set up, left click on the Calculate button. Two reporting options are then available. An abbreviated timing report can be viewed or the detailed timing report can be formatted. The user has a wide

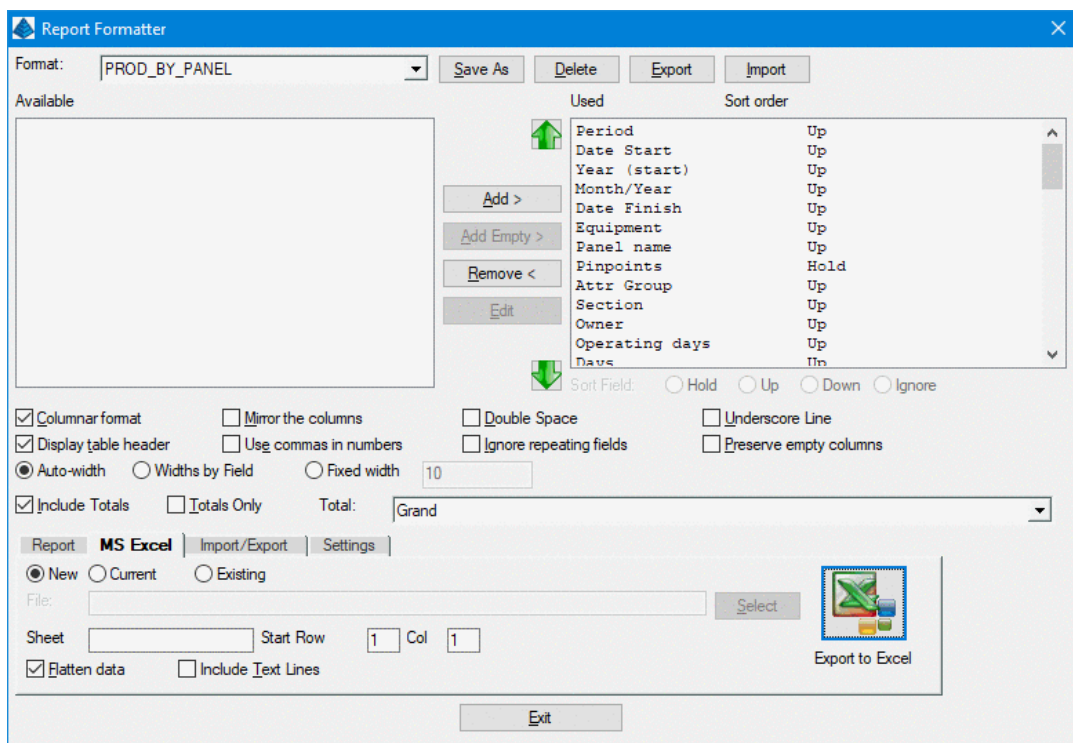
range of timing options that impact the presentation of the timing map.



Once the report options are selected the map is timed out.



When the report times the map it also develops a report. The report formatter allows the user to customize the report. By moving available variables across to the used variables side in the order from top to bottom the user can set the order in the report. This order also drives the sub-totaling option to create various report styles. The report can be displayed on the screen, printed, made part of the drawing, or exported to Excel or Access. The order set up in the report format from top to bottom is the order in the report from left to right. There is a line editor in the screen display that allows the user to add lines and edit the report prior to printing.



This is an example of exporting the report to a spreadsheet. Totals can be exported to spreadsheets as an option. If totals are exported they show up in bold font. This report can also be imported into the Microsoft Access database for additional reporting options.

Period	Date Start	Year (start)	Month/Year	Date Finish	Equipment	Panel name	Pinpoints	Attr Group	Section	Owner	Operating days	Da
8/27/2018 - 9/1/2018	8/27/2018	2018	08/2018	8/31/2018	UNIT #1	MAINS	B:1	DEFAULT	1	UNKNOWN	5.00	
9/1/2018 - 10/1/2018	9/1/2018	2018	09/2018	9/3/2018	UNIT #1	MAINS	B:1	DEFAULT	1	UNKNOWN	1.51	
9/1/2018 - 10/1/2018	9/4/2018	2018	09/2018	9/29/2018	UNIT #1	MAINS	B:1	DEFAULT	2	UNKNOWN	23.49	
10/1/2018 - 11/1/2018	10/1/2018	2018	10/2018	10/29/2018	UNIT #1	MAINS	B:1	DEFAULT	2	UNKNOWN	24.33	
10/1/2018 - 11/1/2018	10/29/2018	2018	10/2018	10/31/2018	UNIT #1	MAINS	B:1	DEFAULT	3	UNKNOWN	2.67	
11/1/2018 - 12/1/2018	11/1/2018	2018	11/2018	11/30/2018	UNIT #1	MAINS	B:1	DEFAULT	3	UNKNOWN	25.99	
12/1/2018 - 1/1/2019	12/1/2018	2018	12/2018	12/12/2018	UNIT #1	MAINS	B:1	DEFAULT	3	UNKNOWN	9.14	
12/1/2018 - 1/1/2019	12/12/2018	2018	12/2018	12/31/2018	UNIT #1	1L	B:E	DEFAULT	1	UNKNOWN	16.81	
12/1/2018 - 1/1/2019	12/12/2018	2018	12/2018	12/31/2018	UNIT #2	MAINS	1:2	DEFAULT	3	UNKNOWN	16.62	
1/1/2019 - 2/1/2019	1/1/2019	2019	01/2019	1/22/2019	UNIT #2	MAINS	1:2	DEFAULT	3	UNKNOWN	18.12	
1/1/2019 - 2/1/2019	1/1/2019	2019	01/2019	1/31/2019	UNIT #1	1L	B:E	DEFAULT	1	UNKNOWN	27.00	
1/1/2019 - 2/1/2019	1/22/2019	2019	01/2019	1/31/2019	UNIT #2	2L	B:E	DEFAULT	1	UNKNOWN	8.88	
2/1/2019 - 3/1/2019	2/1/2019	2019	02/2019	2/28/2019	UNIT #1	1L	B:E	DEFAULT	1	UNKNOWN	24.00	
2/1/2019 - 3/1/2019	2/1/2019	2019	02/2019	2/28/2019	UNIT #2	2L	B:E	DEFAULT	1	UNKNOWN	24.00	
3/1/2019 - 4/1/2019	3/1/2019	2019	03/2019	3/31/2019	UNIT #1	1L	B:E	DEFAULT	1	UNKNOWN	26.00	
3/1/2019 - 4/1/2019	3/1/2019	2019	03/2019	3/31/2019	UNIT #2	2L	B:E	DEFAULT	1	UNKNOWN	26.00	
4/1/2019 - 5/1/2019	4/1/2019	2019	04/2019	4/29/2019	UNIT #1	1L	B:E	DEFAULT	1	UNKNOWN	24.48	
4/1/2019 - 5/1/2019	4/1/2019	2019	04/2019	4/30/2019	UNIT #2	2L	B:E	DEFAULT	1	UNKNOWN	26.00	
4/1/2019 - 5/1/2019	4/29/2019	2019	04/2019	4/30/2019	UNIT #1	1B	B:E	DEFAULT	1	UNKNOWN	1.52	
5/1/2019 - 6/1/2019	5/1/2019	2019	05/2019	5/18/2019	UNIT #1	1B	B:E	DEFAULT	1	UNKNOWN	15.58	
5/1/2019 - 6/1/2019	5/1/2019	2019	05/2019	5/31/2019	UNIT #2	2L	B:E	DEFAULT	1	UNKNOWN	27.00	
5/1/2019 - 6/1/2019	5/19/2019	2019	05/2019	5/31/2019	UNIT #1	R1A	B:E	DEFAULT	1	UNKNOWN	11.42	
6/1/2019 - 7/1/2019	6/1/2019	2019	06/2019	6/10/2019	UNIT #2	2L	B:E	DEFAULT	1	UNKNOWN	7.09	
6/1/2019 - 7/1/2019	6/1/2019	2019	06/2019	6/29/2019	UNIT #1	R1A	B:E	DEFAULT	1	UNKNOWN	25.00	
6/1/2019 - 7/1/2019	6/10/2019	2019	06/2019	6/29/2019	LW	LW1	B:E	DEFAULT	1	UNKNOWN	17.78	
6/1/2019 - 7/1/2019	6/10/2019	2019	06/2019	6/29/2019	UNIT #2	B2	B:E	DEFAULT	1	UNKNOWN	17.09	

=====

Case Study #8, Part 1: Underground Mine Mapping Procedure

Application

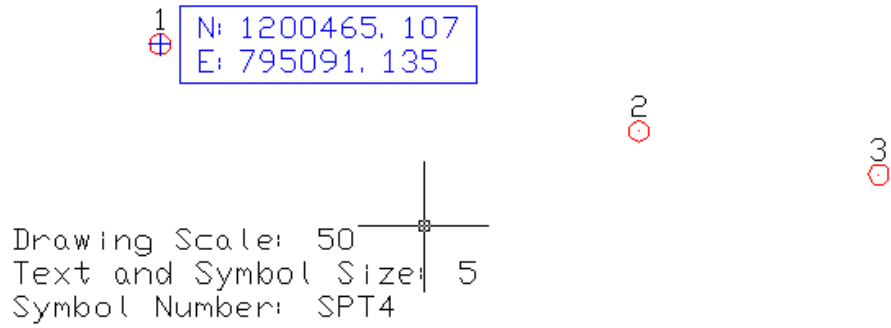
Underground mine mapping requires the entry of traverse notes (for the placement of spads), entry of left and right offsets for locating mine pillars and perimeters, connecting the pillars and perimeters, the placing of special mine symbols and strata thickness measurements, and the drafting of projections for future mining. These are all fully covered in the Mining Module of Carlson. The mapping utilities apply primarily to underground coal mines but also to underground limestone, trona and even salt mines. In fact, any mine that creates pillars for roof support.

Procedure

Here's a quick summary of the mine mapping procedure:

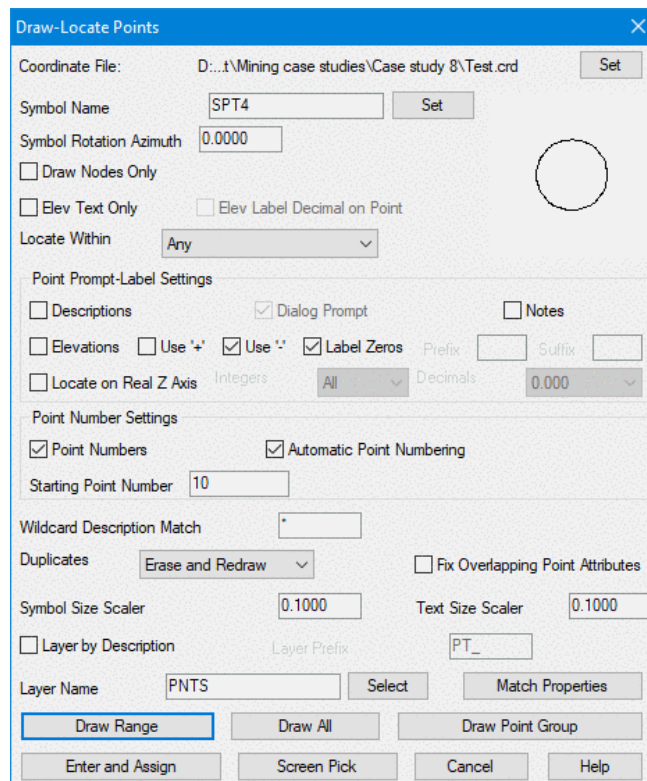
1. Enter the traverse notes and plot the spad points.
2. Draw projections.
3. Enter the mine "offset" notes to pillars and perimeter points.
4. Connect up your pillars and perimeters.
5. Add special mine symbols (stoppings, air flow pointers, etc.).

Let's now look at each of these steps in detail.



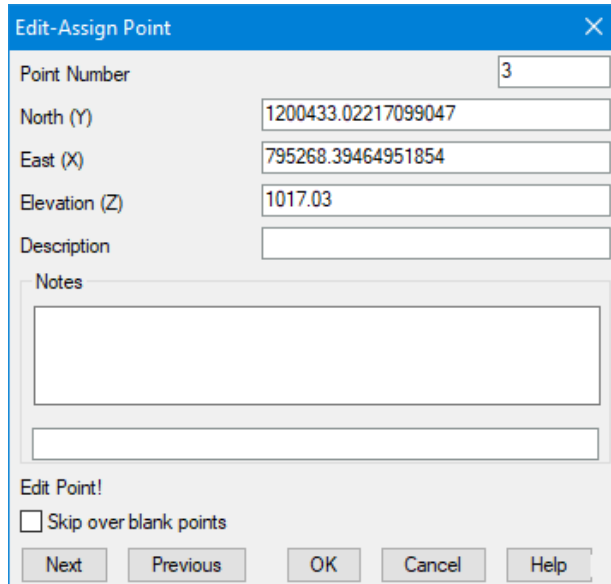
(1) Entering the Traverse Notes

The most direct method of entering traverse notes begins with Draw-Locate Point in the Points Dropdown and continues with Locate by Bearing or Locate by Azimuth. You would Enter coordinates or Pick a Point within Draw/Locate Point. You might choose to enter a specific northing of 1200465.107 and easting of 795091.135. If this is your first point on the job you will be asked to select an existing or make a new .CRD file (coordinate file). You would select New and choose a name, typically equal to the name of the drawing. We will call our file MINE or TEST.



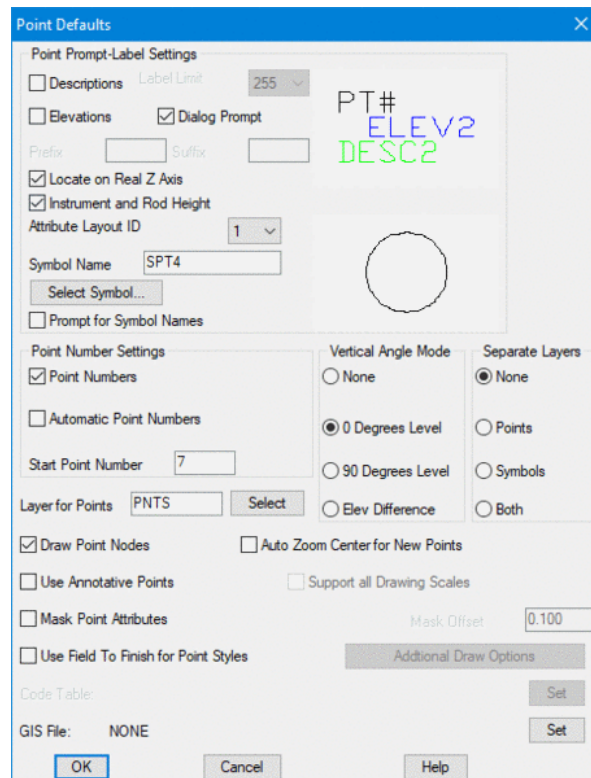
Then if you select LOCATE BY AZIMUTH under the Underground Mining Module > Notes dropdown menu, the program will start at your first point by default, and allow you to enter an azimuth and distance to the next point, such as 100.1535 (100 degrees 15 minutes 35 seconds), and a distance of 120.09 feet. Pressing ENTER will repeat the LOCATE BY AZIMUTH command. Next try defaulting (repeating) the azimuth by pressing ENTER, and input a distance of 60.05. You obtain a drawing as shown above.

Now, let's enter an elevation for all 3 points. Go to Points, Coordinate File Utilities, Edit-Assign Point. The following dialog box appears. Enter in the elevation for Point 1, choose next for points 2 and 3. Point1 elevation is 1016.73, Point 2 is 1016.75 and Point 3 is set to 1017.03.



To enter the additional points. Use the Inverse and Traverse commands from the COGO pulldown in the Survey module.

NOTE:Make sure Instrument Rod and Height Prompting is turned on under Points Pulldown Menu > Point Defaults. This will prompt for the height each time. Next, go to the Survey Module > COGO Pulldown menu and turn on Linework which is "Lines On/Off". This will draw the lines of the traverse. Now Select Inverse from the COGO menu, or type "I" at the command line. Also turn off Automatic Point Numbers for manual naming the points. The point default setting is shown below.



Here are the next four Traverse point entry values.

Command: i, I

Calculate Bearing & Distance from starting point? Traverse/SideShot/Options/Arc/Pick point or point number: 2

PointNo.	Northing(Y)	Easting(X)	Elev(Z)	Description
2	1200443.7177	795209.3048	1016.7500	

Traverse/SideShot/Options/Arc/Pick point or point number: T

Traverse, Line OFF, RAW FILE OFF

Exit/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <7>: 5 for Azimuth

Enter Azimuth (ddd.mmss) <280.1535>: 280.1535

Points/<Distance>: 60.03

Vertical Angle Type (0-3) <2>: 1 (0=None, 1=Vertical(0d level), 2=Zenith(90d level), 3=Elevation Difference)

Enter Vertical Angle (dd.mmss) <0.0000>: 54

Instrument Height <-2.7000>: -2.7 Use negatives to come down from the roof.

Rod-Target Height <-1.9000>: -1.9

Hz Distance > 60.0226

Enter Point Number <4>: press Enter

PointNo.	Northing(Y)	Easting(X)	Elev(Z)	Description
4	1200454.4084	795150.2420	1016.8929	

(next point)

Exit/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <5>: press Enter

Enter Azimuth (ddd.mmss) <280.1535>: 10.135

Points/<Distance>: 59.98

Enter Vertical Angle (dd.mmss) <0.0000>: -.425

Instrument Height <-2.7000>: -2.3

Rod-Target Height <-1.9000>: -2.1

Hz Distance > 59.9753

Enter Point Description <>: press Enter for none

Enter Point Number <5>: press Enter

PointNo.	Northing(Y)	Easting(X)	Elev(Z)	Description
5	1200513.4302	795160.8942	1015.9456	

(next point)

Exit/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <5>: press Enter

Enter Azimuth (ddd.mmss) <10.135>: 280.1535

Points/<Distance>: 60.02

Enter Vertical Angle (dd.mmss) <0.0000>: .23

Instrument Height <-2.3000>: -1.95

Rod-Target Height <-2.1000>: -2.14

Hz Distance > 60.0187

Enter Point Number <6>: press Enter

PointNo.	Northing(Y)	Easting(X)	Elev(Z)	Description
6	1200524.1201	795101.8352	1016.5372	

(next point)

Exit/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <5>: press Enter

Enter Azimuth (ddd.mmss) <280.1535>: 190.1535

Points/<Distance>: 60.01

Enter Vertical Angle (dd.mmss) <0.0000>: .07

Instrument Height <-1.9500>: -1.78

Rod-Target Height <-2.1400>: -1.85

Hz Distance > 60.0099

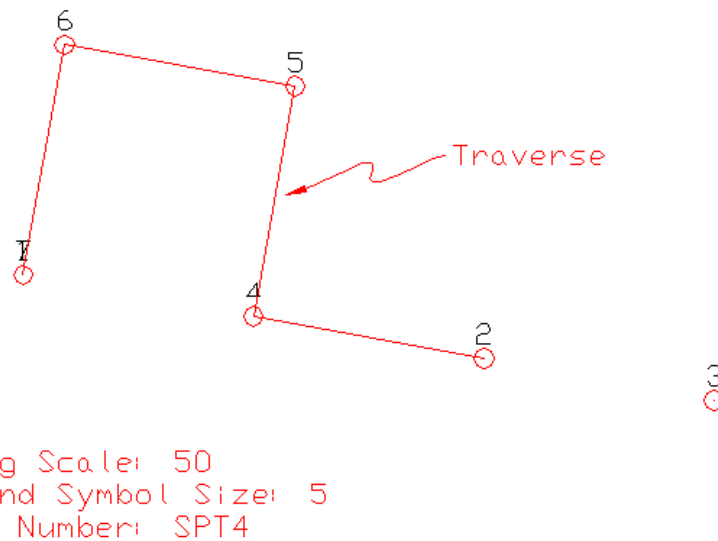
Enter Point Number <7>: *press Enter*

PointNo. Northing(Y) Easting(X) Elev(Z) Description

7 1200465.0698 795091.1468 1016.7293

Exit/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <5>: *press Exit to end*

The following image should appear in the drawing.



(2) Drawing Projections

(Underground Mining module > Works pull-down menu > Basic Projections Command)

There are four types of projections: Basic Projections, Advanced Projections, Projections and Ventilation, and Rooms. Basic Projections will only produce rectangular projections (not angular). This option, however, has fewer prompts and is more automatic. Let's run Basic Projections.

Start pt. of belt entry: *Pick point 4* (You are automatically placed in intersect and node snap mode. All Carlson points have nodes, so you can use the node snap. You can override the pre-set snap by picking the asterisks at the top of the sidebar menu and selecting another choice.)

Command: *panell1, PANEL1*

Command: (intersect/node on)

Start pt. of belt entry: *Pick Point 4* (for point number 4)

<P>ick End Point For Belt or [A] For Azi/Dist: *A*

Azimuth to end of belt [ddd.mmss]: *10.1535*

Distance to end: *540*

Number of entries on left side: *1*

Number on right side: *2*

Entry spacing: *60*

Crosscut spacing: *60*

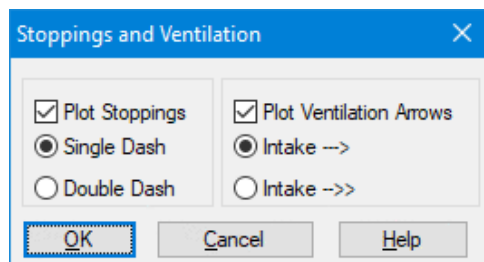
Plot Outer Rib Line y/<n>: *Y*

Cut Width <20>: *press Enter*

Offset to starting rib line (e.g. -10,0,<10>): *press Enter*

Offset to ending rib line (e.g. -10,0,<10>): *press Enter*

Offset xcuts y/<n>: *N*



Pick pt. on xcut for beginning of stopping line: *pick a point as shown on figure*

Draw another stopping line <y>/n: *Y*

Pick pt. on xcut for beginning of stopping line: *pick a point as shown on figure*

Draw another stopping line <y>/n: *Y*

Pick pt. on xcut for beginning of stopping line: *pick a point as shown on figure*

Draw another stopping line <y>/n: *N*

Pick pt. on entry to begin drawing ventilation arrows [nea on]: *pick a point as shown on figure*

Distance between ventilation arrows: *pick a point as shown on figure*

<I>ntake or [R]eturn: *I*

Draw ventilation arrows on another entry <y>/n: *Y*

Pick pt. on entry to begin drawing ventilation arrows [nea on]: *pick a point as shown on figure*

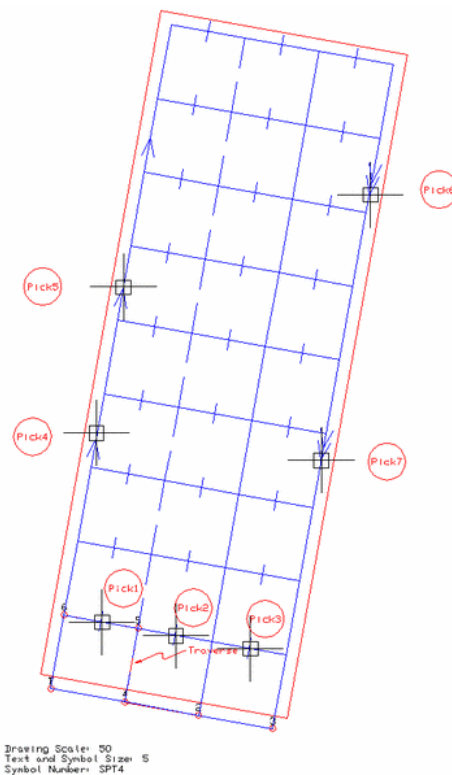
Distance between ventilation arrows: *pick a point as shown on figure*

<I>ntake or [R]eturn: *R*

Draw ventilation arrows on another entry <y>/n: *N*

Project another panel <y>/n: *N*

You will notice how all projection lines are on distinct layers (PROJECTIONS, PROJSTOPPINGS, PROJVENTARROWS).



With the projection shown, it is easy to illustrate rooms. Select Rooms under the Works dropdown.

Command: rooms, ROOMS

Start pt. of rooms: *pick point 1*

End pt. of rooms: *pick point 3*

Distance between room entries: 40
Distance between room crosscuts: 40
Room depth: 240
Rooms on <R>ight or [L]eft side: R

While here, let's select Label Proj. Distances.

Command: LDIST

Belt entry (point to point mode) y/<n>: N

Select line to label distance on:

Pick as many entry and crosscut lines as desired. This Label Proj. Distances routine applies to Basic Projections and Rooms using the pick line option. The pick option places the distance on the picked line segment. For Advanced Projections, most lines are continuous from the start to the end of the projections, so the point-to-point end-point pick mode must be used. This point-to-point mode must also be used on the belt line (from spad 4), since it is one continuous line.

Now let's label our panel and rooms. Select Panel Label Block.

Command: labelplan, LABELPAN

Insertion point: Pick anywhere in the open areas.

Rotation angle: Pick the rotation angle from the insertion point.

SECOND DIMENSION <60'>: *press Enter*

FIRST DIMENSION <60'>: *press Enter*

Now Select the command: Room Label Block

Command: labelrms, LABELRMS

Insertion point: Pick an open spot for the center of the label.

Rotation angle: Pick the rotation angle from the insertion point.

ON ADVANCE OR RETREAT <ADVANCE>: *press Enter*

SECOND DIMENSION <40'>: *press Enter*

FIRST DIMENSION <40'>: *press Enter*

The final projection option we'll run is Advanced Projections. Select it.

Command: panel2, PANEL2 (**Intersect/node on**)

Pick Start Point For Belt: Pick where Pick3 is on the figure below.

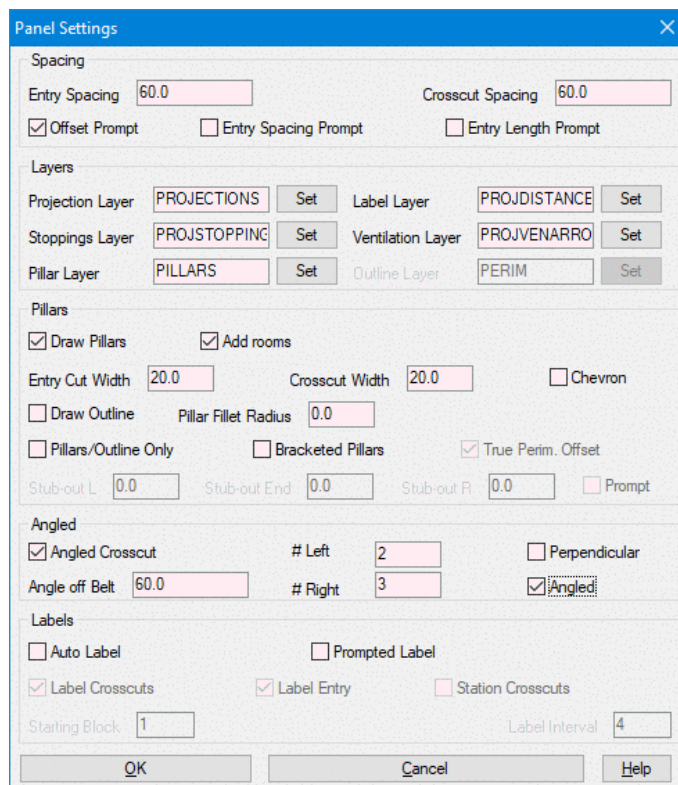
Pick End Point For Belt, or <A> For Azi/Dist: A

Enter Azimuth in ddd.mmss <280.1535>: 280.1535

Enter Distance: 360

How Many Entries Left Of The Belt <0>: 2

How Many Entries Right Of The Belt <0>: 3



Enter Offset For This Heading <0>: *press Enter*

Enter Offset For This Heading <0>: *press Enter*

Enter Offset For This Heading <0>: *press Enter*

Enter Offset For This Heading <0>: *press Enter*

Enter Offset For This Heading <0>: *20*

Use sidebar to select single-dash stoppings.

Pick pt. on xcut for beginning of stopping line: Pick as many xcuts as you would like.

Draw another stopping line <y>/n: *Y* Use sidebar to select single-dash stoppings.

Pick pt. on xcut for beginning of stopping line:

Draw another stopping line <y>/n: *Y* Use sidebar to single select -dash stoppings.

Pick pt. on xcut for beginning of stopping line:

Draw another stopping line <y>/n: *N*

Pick pt. on entry to begin drawing ventilation arrows [nea on]: Pick ventilation arrow location.

Distance between ventilation arrows: Pick interval distance.

<I>ntake or **[R]**eturn: *I*

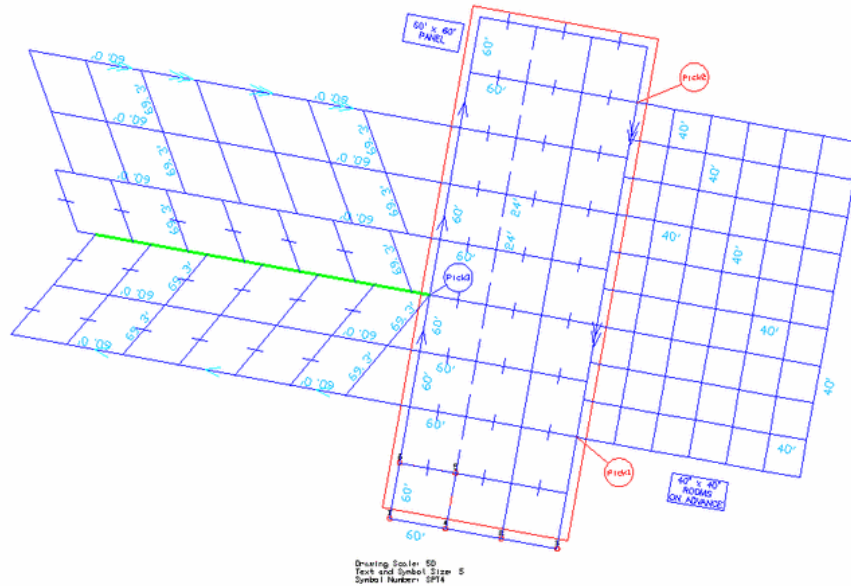
Draw ventilation arrows on another entry <y>/n: *Y*

Pick pt. on entry to begin drawing ventilation arrows [nea on]: Pick ventilation arrow location

Distance between ventilation arrows: Pick interval distance

<I>ntake or **[R]**eturn: *R*

Draw ventilation arrows on another entry <y>/n: *N*



Complete the stopping and ventilation options as desired. A plot similar to the one above is obtained. All entries are continuous lines, while crosscuts are individual lines.

(3) Entering the Offset Notes

Since mining operations have different methods for note taking and posting, there are three distinct routines for entering offset notes: Mine Note Left/Right/Face, Mine Note Auto Left/Right, and Mine Note From CRD File. Each of these routines operate with a similar procedure of first choosing a starting point and direction, and then distancing up and offsetting left and right repeatedly. There are also two setup routines. Mine Note From Face Prompt allows the mine note entry routines to post back from the face, and ASCII File From Notes is an option for keeping a record of the offset notes in an ASCII file. All these routines are located in the Notes dropdown menu.

Before we can begin an example, there needs to be some starting points. Select Locate Point and place one point. Then select Locate By Azimuth and place a point at azimuth 100.0000 and distance 60. Repeat Locate By Azimuth with the same values from the second point.

Now we can enter offset notes. Go to Mine Note Auto Left/Right in the Notes dropdown menu.

Command: note2

Enter Offset File Name <offset.dat>: *press Enter*

Append File [<Yes>/No]? *N* (This appears only if the file already exists.)

Tabular Format–Distance L R Hgt– (<y>/n): *Y*

From Station point[node on]: (795704.0 1.20119e+006) *Pick Point 1*

To Station point (A for Azimuth): *A*

Azimuth of Heading (DD.MMSS) or p for pick <0.0>: *10*

Spad Number, or <ENTER> For None: *press Enter*

Entry Number: *press Enter*

Enter distance from station on centerline (U To Undo, Enter to end): *10*

Enter left offset distance: *10*

Enter right offset distance: *press Enter*

Enter distance from station on centerline (U To Undo, Enter to end): *30*

Enter left offset distance: *C11*

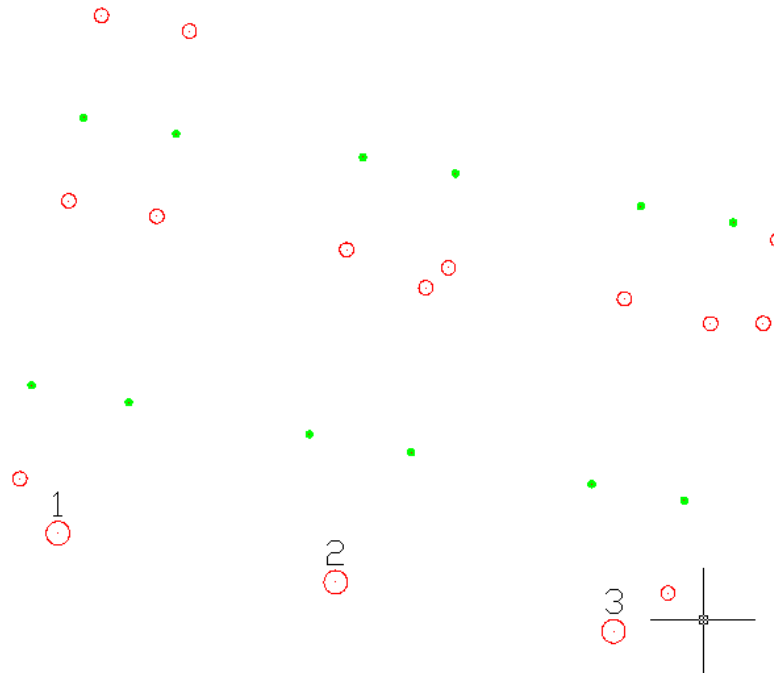
Enter right offset distance: *C10*

Enter distance from station on centerline (U To Undo, Enter to end): *70*

Enter left offset distance: *10*

Enter right offset distance: 9
 Enter distance from station on centerline (U To Undo, Enter to end): 88
 Enter left offset distance: C10
 Enter right offset distance: C10
 Enter distance from station on centerline (U To Undo, Enter to end): 110
 Enter left offset distance: 10
 Enter right offset distance: 9
 Enter distance from station on centerline (U To Undo, Enter to end): *press Enter*
 Another Spad [*<Yes>/No*]?: Y
 From Station point[node on]: (795763.0 1.20118e+006) *pick Point 2*
 Azimuth of Heading (DD.MMSS) or p for pick *<10.0>*: *press Enter*
 Spad Number, or *<ENTER>* For None: *press Enter*
 Entry Number: *press Enter*
 Enter distance from station on centerline (U To Undo, Enter to end): 30
 Enter left offset distance: C11
 Enter right offset distance: C11
 Enter distance from station on centerline (U To Undo, Enter to end): 65
 Enter left offset distance: *press Enter*
 Enter right offset distance: 8
 Enter distance from station on centerline (U To Undo, Enter to end): 70
 Enter left offset distance: 10
 Enter right offset distance: 12
 Enter distance from station on centerline (U To Undo, Enter to end): 90
 Enter left offset distance: C10
 Enter right offset distance: C10
 Enter distance from station on centerline (U To Undo, Enter to end): *press Enter*
 Another Spad [*<Yes>/No*]?: *press Enter*
 From Station point[node on]: (795822.0 1.20117e+006) *pick Point 3*
 Azimuth of Heading (DD.MMSS) or p for pick *<10.0>*: *press Enter*
 Spad Number, or *<ENTER>* For None: *press Enter*
 Entry Number: *press Enter*
 Enter distance from station on centerline (U To Undo, Enter to end): 10
 Enter left offset distance: *press Enter*
 Enter right offset distance: 10
 Enter distance from station on centerline (U To Undo, Enter to end): 30
 Enter left offset distance: C10
 Enter right offset distance: C10
 Enter distance from station on centerline (U To Undo, Enter to end): 68
 Enter left offset distance: *press Enter*
 Enter right offset distance: 9
 Enter distance from station on centerline (U To Undo, Enter to end): 70
 Enter left offset distance: 10
 Enter right offset distance: 20
 Enter distance from station on centerline (U To Undo, Enter to end): 88
 Enter left offset distance: *press Enter*
 Enter right offset distance: 20
 Enter distance from station on centerline (U To Undo, Enter to end): 90
 Enter left offset distance: C10
 Enter right offset distance: C10
 Enter distance from station on centerline (U To Undo, Enter to end): *press Enter*
 Another Spad [*<Yes>/No*]?: N
 Print file containing offset input data [*<Yes>/No*]?: N

This should produce the following layout of points and spads:

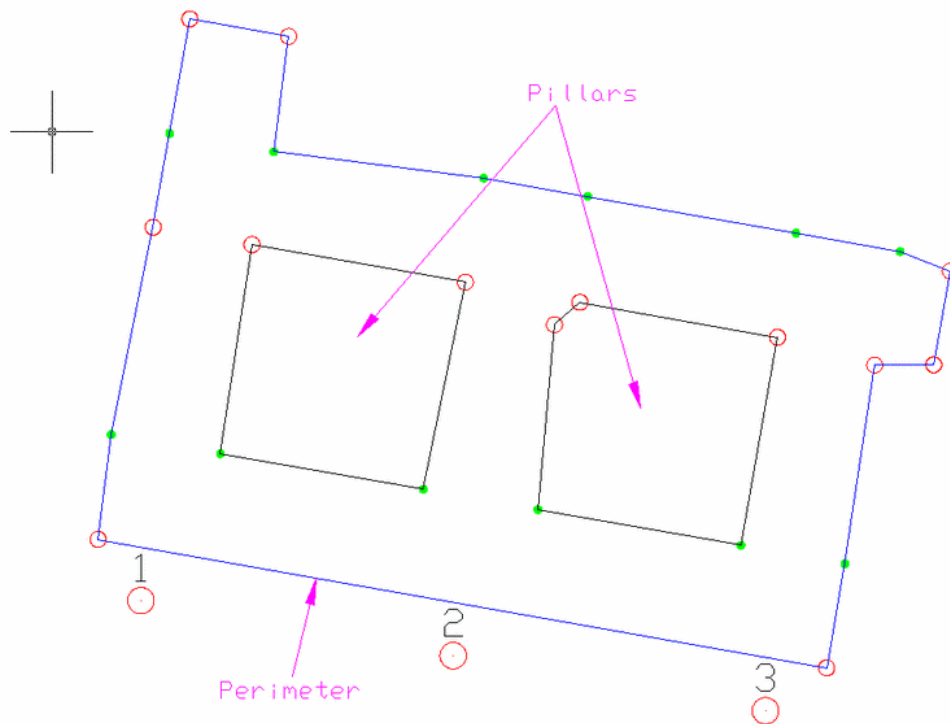


(4) Connecting Pillars and Perimeter

Once the offset notes have been entered, the pillars and perimeters can be drawn by connecting the dots. Pillars are created as closed polylines in the PILLARS layer, and perimeters are created as closed polylines in the PERIM layer.

There are two approaches for connecting the dots. One is to manually connect the pillars and perimeters by simply picking each point in order and using C to close at the end. This is done with the Draw Pillars and Draw Perimeter routines in the Works dropdown. The other method is to use AutoMine Connections which automatically connects up the pillars and perimeter. This routine reads a record file from a mine note entry routine and requires that the corner points (the first points across a crosscut) are coded. If you entered the example offset notes in the previous section, then everything is ready for AutoMine Connections. Select it from the Works dropdown and choose the file "offset". After drawing the pillars and perimeter, AutoMine leaves a pick box hanging off the perimeter so that you may connect it with a previous perimeter. In our sample case, just enter 'C' to close the perimeter.

It is important for quantity calculations that these polylines are closed. To make sure your pillars and perimeter are closed polylines, use the Highlight Unclosed Polyline routine in the Works dropdown.



Case Study #8, Part 2: Computing Tonnage and Acreage

Application

Once the mine map is completed using the techniques described in the previous section, quantities of mined mineral and waste rock may be computed, with industry standard reports. The reports will be generated based on the reporting method selected under the Settings Pulldown Menu > Carlson Configure > Mining Settings.

Procedure

Here's a quick summary of the procedure for computing quantities:

1. Configure the "look" of your coal sections (or other mineral).
2. Locate all coal sections measured in the mines.
3. Compute tonnages by average or modeled grid methods.
4. Draw and compute quantities for pillar cuts on the "retreat".

(1) Configuring the Section Information

Configuring the section information involves assigning each strata a name and a density, and specifying the look of the section for when it is placed in the drawing. This information is stored in a user-specified file and is referenced for locating sections in the drawing and for quantity calculations. If you reconfigure an existing section file, sections placed in the drawing using the previous section file become invalid and must be replaced. The strata in a section can be configured as either individual or composite. With the individual configuration, each strata has its own name and density. With composites, each strata is still named separately but they also are divided into groups that have a group name and density. The principle advantage to composites is that it allows you to enter and list out each strata height and then combine the strata into their corresponding composite category when generating tonnage quantities. First, let's configure an individual section. Select Configure Section Info in the Works dropdown from underground mining module.

Enter Section Configuration Filer Name (MINE.SC)

Mine Name: *Round Mountain*

Enter Abbreviated Strata Name/<ENTER> to End: *C* This name is drawn next to the corresponding height when a section is located in the drawing.

Enter The Full Strata Name <C>: *Coal* This name is used in quantity reports.

Enter Abbreviated Strata Name/<ENTER> to End: *B*

Enter The Full Strata Name <C>: *Bone*

Enter Abbreviated Strata Name/<ENTER> to End: *R*

Enter The Full Strata Name <C>: *Rock*

Enter Abbreviated Strata Name/<ENTER> to End: *press Enter*

Enter Individual densities or Composite densities (I/C) <I>: *press Enter*

Average wt. of C (Coal) [lbs/ft³]: *80*

Average wt. of B (Bone) [lbs/ft³]: *150*

Average wt. of R (Rock) [lbs/ft³]: *150*

Circle the Coal Section (y/<n>)? *Y* This specifies whether a circle is drawn around the section when it is placed in the drawing.

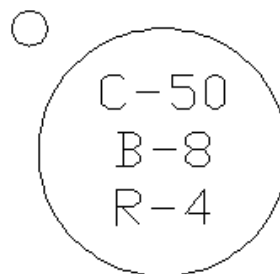
Plot the Numeric Value Only (y/<n>)? *press Enter*

Text Size <6.0>: *press Enter*

Enter thickness in feet or inches [Feet/<Inches>]? *I for Inches*

Prompt for entry width [Yes/<No>]? *Y if Yes Linear Advance will also be reported*

The following figure shows how this section will appear in the drawing.



Now let's make a composite section. Select Configure Section Info in the Works dropdown.

Enter Coal Section Configuration Filer Name: *SECTION2* Or the name of your choice.

Mine Name: *Round Mountain*

Enter Abbreviated Strata Name/<ENTER> to End: *TC*

Enter The Full Strata Name <C>: *Top Coal*

Enter Abbreviated Strata Name/<ENTER> to End: *TR*

Enter The Full Strata Name <C>: *Top Rock*

Enter Abbreviated Strata Name/<ENTER> to End: *BC*

Enter The Full Strata Name <C>: *Bottom Coal*

Enter Abbreviated Strata Name/<ENTER> to End: *BR*

Enter The Full Strata Name <C>: *Bottom Rock*

Enter Abbreviated Strata Name/<ENTER> to End: *press Enter*

Enter Individual densities or Composite densities (I/C) <I>: *C* Define the Composite categories.

Enter Composite Category/<Enter> to END: *Coal*

Average wt. of COAL [lbs/ft³]: *80*

Enter Composite Category/<Enter> to END: *Rock*

Average wt. of ROCK [lbs/ft³]: *150*

Enter Composite Category/<Enter> to END: *press Enter* Assign the strata to a Composite category.

Enter Composite Category for Top Coal TC (COAL ROCK): *coal*

Enter Composite Category for Top Rock TR (COAL ROCK): *rock*

Enter Composite Category for Bottom Coal BC (COAL ROCK): *coal*

Enter Composite Category for Bottom Rock TC (COAL ROCK): *rock*
Circle the Coal Section (y/<n>)? *press Enter*
Plot the Numeric Value Only (y/<n>)? *press Enter*
Text Size <6.0>: *press Enter*
Enter thickness in feet or inches [Feet/<Inches>]? *I for Inches*
Prompt for entry width [Yes/<No>]? *Y if Yes Linear Advance will also be reported*

The following figure shows how this section will appear in the drawing.

```

O
TC-30
TR-5
BC-25
BR-4

```

(2) Locating Sections

To locate section information measured from the mine, select Place Coal Sections from the Works dropdown. Select the desired section configuration file such as SECTION2 which you defined in the previous section.

Pick sample point for coal section: *pick a point on the map.* This is the point where the measurements came from.

Pick Start Point: *pick another point.* This is where the text will be plotted. The text may be placed anywhere you like because the program gets the section measurements from the small circle at the sample point and not from the text.

Pick Alignment Point: *pick a point to align the text*

Now enter your measurements.

How many inches of Top Coal TC: *30*

How many inches of Top Rock TR: *5*

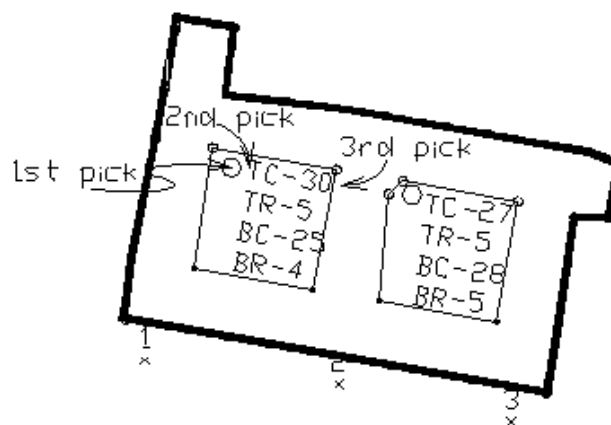
How many inches of Bottom Coal BC: *25*

How many inches of Bottom Rock BR: *4*

Enter the entry width: *20*

Enter Another Section (Y/N) <Y>: *press Enter*

Enter the other section as shown in the next figure.



Placing a section

(3) Compute tonnages by average or modeled grid methods

There are three routines for calculating tonnages: Quantities by Average Method, Quantities by Grid Method and Quantities by Centerline. These methods require section sample points, and pillars and a perimeter defined as closed

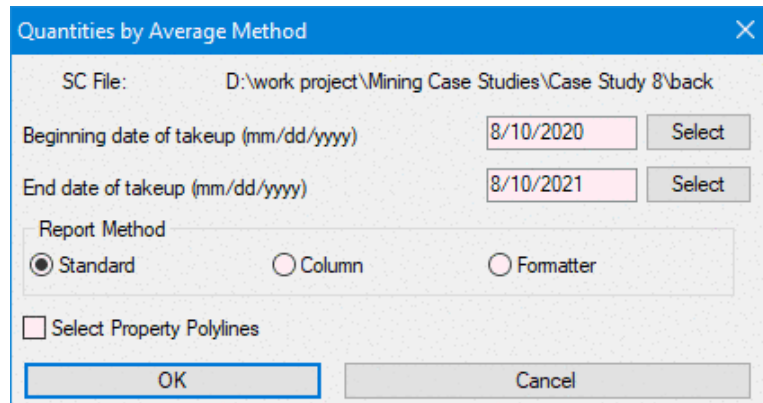
polylines in the PILLARS and PERIM layers respectively. As its name suggests, the average method uses the average values from sample section points to compute its quantities. The grid method actually models the values of the sample points over the mined area which results in more accurate tonnages. For now, let's use the average method because it is much faster and requires fewer sample points.

Select Quantities by Average Method in the Works dropdown.

Select a Mining Project Definition (MPD) File or Create a new one

Select the file that defines the section sample points

Specify the Beginning and Ending dates of take-up and Report Format in the dialog below



Quantities by Average Method

SC File: D:\work project\Mining Case Studies\Case Study 8\back

Beginning date of takeup (mm/dd/yyyy) 8/10/2020 Select

End date of takeup (mm/dd/yyyy) 8/10/2021 Select

Report Method

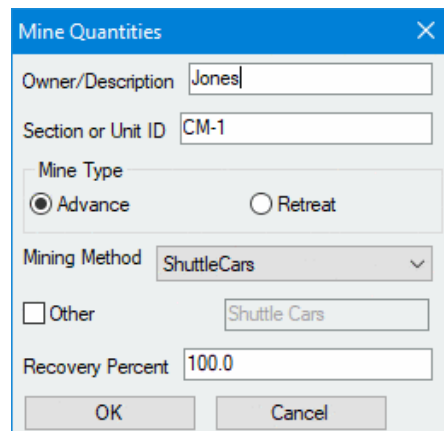
Standard Column Formatter

Select Property Polylines

OK Cancel

Select property polylines or press Enter for none: *press Enter if no property lines defined, if defined program gets Owner Name from here*

Fill in the Following Dialog: *if any field is left blank it will not be reported*



Mine Quantities

Owner/Description Jones

Section or Unit ID CM-1

Mine Type

Advance Retreat

Mining Method ShuttleCars

Other Shuttle Cars

Recovery Percent 100.0

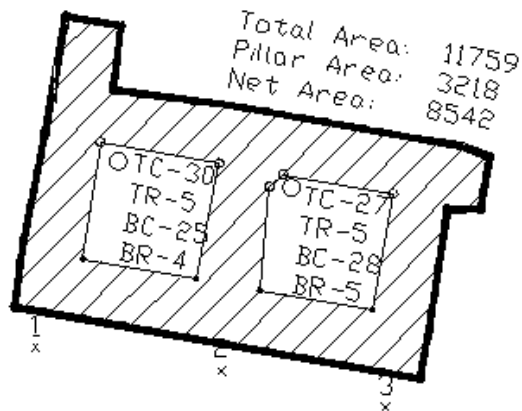
OK Cancel

Select pillars, perimeters, and section sample points.

Select objects: *select the pillars, perimeter, and sample points*

Pick location to draw results or Enter for none: *pick a point above the mine as shown*

Another Area (Y/N) <Y>: N



Quantities by Average Method

A report would be generated based on the Report Format method. If standard method selected following report would be generated

NUMERICAL AVERAGE KEY SECTION METHOD				Mon Aug 10 10:02:19 2020	
Multiple Composites Configuration					
MINE: ROUND MOUNTAIN MINED FROM 8/10/2020 TO 8/10/2021					
AREA NO. 1 DESCRIPTION: Jones					
SECTION/UNIT ID: CM-1 MINE TYPE: Advance MINING METHOD: Shuttle Cars					
GROSS AREA MINED (S.F.): 11760.00 DEPLETED ACRES: 0.270					
AREA OF PILLARS (S.F.): 3217.50 ACRES OF PILLARS: 0.074					
NET AREA MINED (S.F.): 8542.50 NET ACRES MINED: 0.196					
AVERAGE COAL THICKNESS (INCHES): 46.50 (FEET): 3.88					
AVERAGE ROCK THICKNESS (INCHES): 9.50 (FEET): 0.79					
TOTAL MINING HEIGHT (INCHES): 56.00 (FEET): 4.67					
AVERAGE COAL WT. (LBS/CU. FT.): 80.00					
AVERAGE ROCK WT. (LBS/CU. FT.): 150.00					
COAL (TONS): 1324.09					
ROCK (TONS): 507.21					
NON-RECOVERABLE KEY (TONS): 0.00 KEY RECOVERY PERCENT: 100.00%					
TOTAL TONS : 1831.30 PERCENT KEY BY WGT.: 72.30%					
KEY ACRE-FEET: 0.760					
CAVITY ACRE-FEET: 0.915					

Update coal tonnage files (<Y>/N)? Y This option updates data files for tonnage reports.

Enter a mine name: Mine

Enter a panel name: press Enter for none

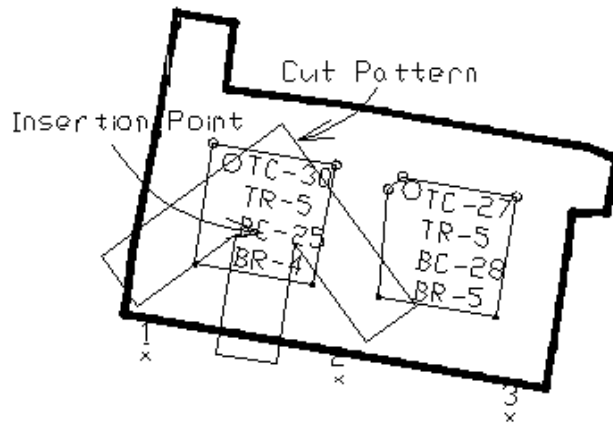
Enter the estimated coal reserves for mine Mine: 300000

(4) Draw and compute volumes for pillar cuts on retreat

One of the final steps in underground mining is cutting into the pillars on the "retreat". The results of these cuts can be quickly drawn on the mine map using the Pillar Cut routine in the Works dropdown. The procedure is simply to pick a cut pattern and then place the pattern inside the pillar to be cut. That's it. Depending on the selected method, this routine will either redraw the pillar's polyline with the cuts removed, or it will create new perimeter polylines inside the pillar in the cut out spaces. The pillars must be closed polylines in the PILLARS layer. The rotation of the placed pattern will follow the current snap. If the cut pattern you need is not already in the table, you can define your own by first drawing a polyline of your pattern and then selecting the user-defined box in the Pillar Cut symbol table.

After redrawing the cut pillars with Pillar Cut, Quantities by Average Method or Quantities by Grid Method can be used to compute the quantities of the cuts. If the Cut option of Pillar Cut was used, then you will get the total

quantities from the area between the pillars plus the pillar cuts. To get the quantities of only the pillar cuts, use the Perim option of Pillar Cut. This will create perimeter polylines inside the pillars. Then use Quantities by Average Method and select all these perimeter polylines. Let's practice pillar cuts.



Locating a pillar cut pattern

Select Pillar Cut from the Works dropdown.

Choose the Cut 14 symbol or the symbol of your choice.

Perimeter layer <PERIM>: *press enter*

Enter the azimuth for the cuts <0.0>: *10.0*

Cut the pillar or create new perimeter? (<Cut>/Perim) *Perim*

Hatch the new perimeter polylines? (Y/<N>) *press Enter*

Select mine pillars Polylines in the PILLARS layer.

Select objects: *select all the pillars*

Pick a point for the symbol: *pick a point as shown in the figure above*

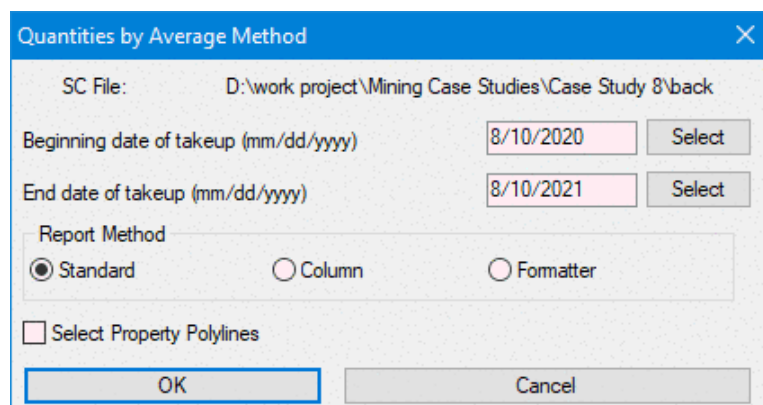
Do another cut [<Yes>/No]? *Y*

Pick a point for the symbol: *pick a point in the other pillars*

Pick a point for the symbol: *press Enter to exit*

Now let's get the quantity mined from the pillar cuts.

Select Quantities by Average Method in the Mining Works dropdown. Choose Section2 or the file that defines the section sample points.



The Cuts "Selection Type" filters out everything except perimeter polylines and sample points. This allows you to select by windowing the area of the pillar cuts. Be sure not to include the main, outside perimeter.

Command: C:QAVG

MINE=Round Moutain

Composite: COAL Density: 80.000000 lbs/ft³

Composite: ROCK Density: 150.000000 lbs/ft³

Which type of selection [<Standard>/Cuts**]? Cuts**

Select objects: *select the perimeter polylines and sample points*

Pick location to draw results or Enter for none: *pick a point above the mine*

Another Area (Y/N) **<Y>:** N

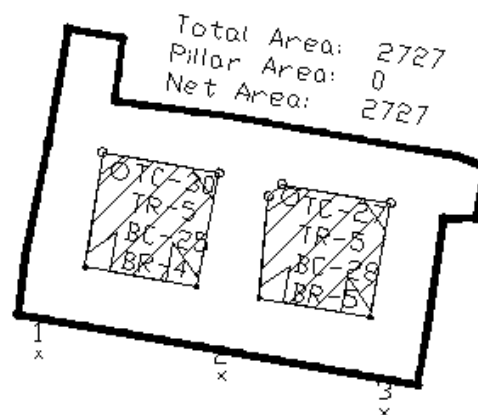
Update coal tonnage files (<Y>/N**)?** Y This option updates data files for tonnage reports.

Enter a mine name: Mine

Enter a panel name: *press Enter for none*

Enter the estimated coal reserves for mine Mine: 300000

This creates the following plot.

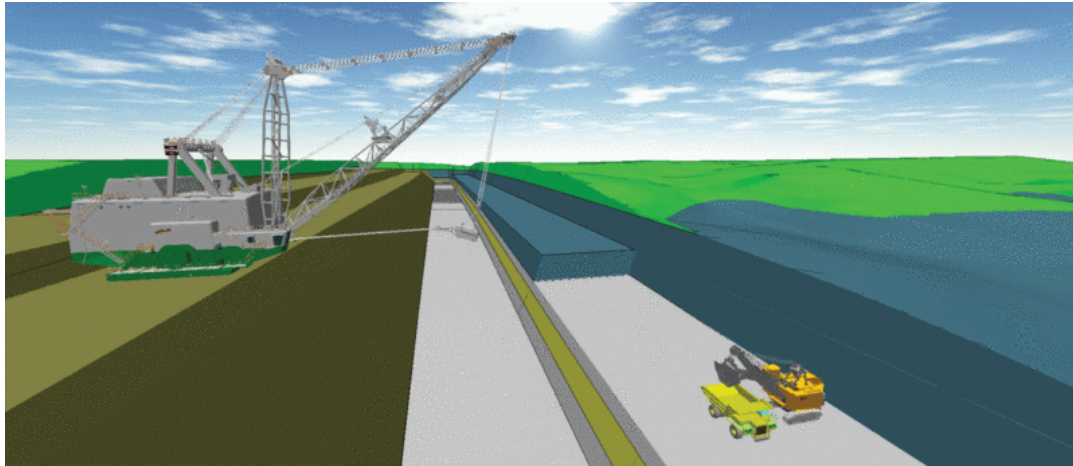


Pillar cut quantities



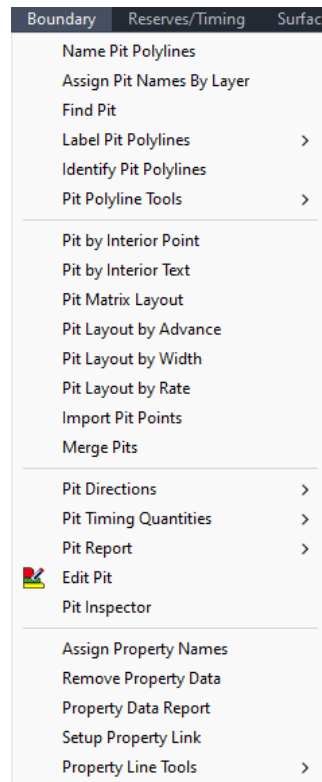
Surface Mining Module 17

The Surface Mining Module consists of various commands for planning and scheduling a surface mining operation. Major features include pit and spoil design, haulage analysis, and mine scheduling. Commands are accessible from the Boundary, Reserves/Timing, Surface, and Spoil pulldown Menus.



Boundary Menu

The Boundary menu has commands for managing pit and property polylines. The property polyline commands are described in the Underground Mining section of the manual.



Name Pit Polylines

This command attaches a pit name and site name to closed polylines that are used as inclusion or exclusion perimeter routines like Surface Mine Reserves and Surface Equipment or Production Timing.

Prompts

Label area names [**<Yes>/No**]? Choose whether or not to label.

Text height **<4.00>**: *press Enter to accept 4 drawing units (ft or m tall) or enter a new text height*

Auto place labels in center [**<Yes>/No**]? *press Enter to accept To manually pick the label position, answer No to this prompt.*

Prompt for inclusion, exclusion or both [**<Inclusion>/Exclusion/Both**]? *Bfor Both.* If you are only choosing inclusion areas, then you can answer Inclusion to this prompt to skip the exclusion polyline prompt and speed up entry. Or you can use Exclusion if you're just going to add exclusion areas to existing inclusion pit areas.

Site name **<Site 1>**: *press Enter to accept Site 1, or type a new site name*

Pit name **<Pit 1>**: *press Enter to accept Pit 1, or type a new pit name*

Select Inclusion perimeter polylines.

Select objects: *pick closed polylines for areas to include in calculations*

Select Exclusion perimeter polylines.

Select objects: *pick closed polylines for areas to exclude from calculations*

Specify another area [**<Yes>/No**]? *press Enter to accept Yes or type N for no* If you do specify another area, then the Site Name remains the same, but the Pit Name is automatically incremented by one for efficient naming.

Pulldown Menu Location: Boundary

Keyboard Command: pitname

Prerequisite: Closed polyline

Assign Pit Names By Layer

When the linework is layerized so that each polyline for pits is placed in the separate layer, this routine may be used to assign pit names by layer name. The user is prompted for site name and pit names are derived from the layer names of each polyline selected.

Prompts

Site name <Site 1>: *Enter to accept or type in a new site name*

Select polylines to set pit name by layer.

Select objects: *pick polylines*

Pulldown Menu Location: Boundary in the Advanced Mine Module

Keyboard Command: layerpit

Find Pit

This command will find a certain pit on screen. It will zoom to the pit extents and center it. The pit line is also dashed or highlighted to distinguish it from surrounding pits.

Prompts

Enter a name of the pit to find: *x-38*

Pulldown Menu Location: Boundary

Keyboard Command: findpit

Label Pit/Site Names

This command labels the pit name and site name that are attached to the selected polyline. The Horizontal and Align placement methods will draw the labels in the center of the polyline. The difference is that the horizontal option will draw the label horizontal to the current twist and the Align method will rotate the label to follow the main direction of the pit. The Pick method prompts you for a center point to pick and an alignment for the text.

Prompts

Text height <4.0>: *press Enter to accept 4.0, or type in a new text height*

Include Site Name in Label [<Yes>/No]? *press Enter for yes or type N for no*

Label placement method [<Align>/Horizontal/Pick]? *press Enter for Align, or type H for horizontal alignment, or P to pick the alignment and location*

Select pit polyline to label: *pick a polyline*

Site Name: Site 1

Pit Name: Pit 1

Pick point for label: *pick a point*

Pick alignment point: *pick a point*

Select pit polyline to label: *press Enter to end*

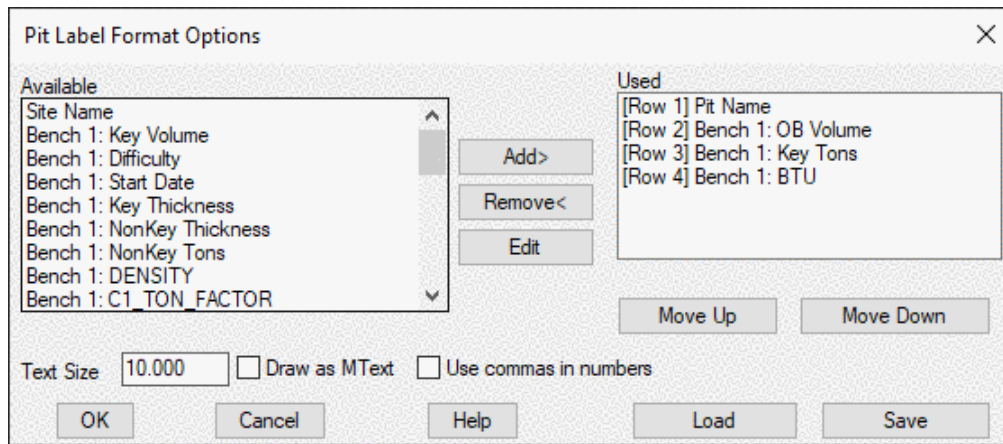
Pulldown Menu Location: Boundary in the Advanced Mine Module under Label Pit Polylines

Keyboard Command: pitlabel

Pit Label Formatter

This command places text inside each pit, labeling values such as name, quantities, and quality. Anything that is stored in the selected pits will be shown under the Available column. The formatter screen allows for placement of text in the pits by row, selecting a justification, text size, layer, and color. The text can either be drawn horizontally,

aligned to a specific angle, or aligned with the long axis of the pit. There must be values stored in the pits and the pits must have direction assigned in order to use this command.



Only items in the Used column will be labeled. Items may be moved to the Used column by double clicking the item or by selecting an item and clicking **Add**.

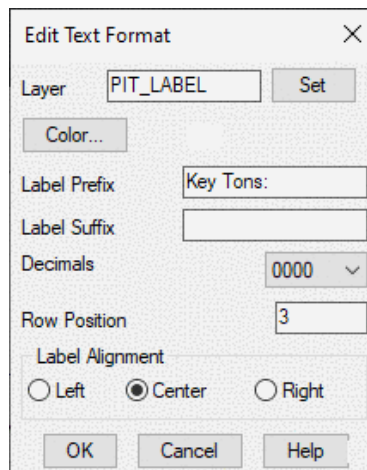
Items may be moved up or down in the Used column with the **Move Up/Down** buttons, but this does not set the row position for the label. If two items are on the same row, the item higher in the list will be labeled first.

Text Size: The size of the label.

Draw as MText: This option will use MText entities for the labels rather than standard Text entities.

Use commas in numbers: This option will insert commas into the numbers.

When an item is added, the below dialog will appear to specify the settings for that item (this is the same as clicking the **Edit** button).



Layer: The layer used for the specific label.

Color: The color of the label.

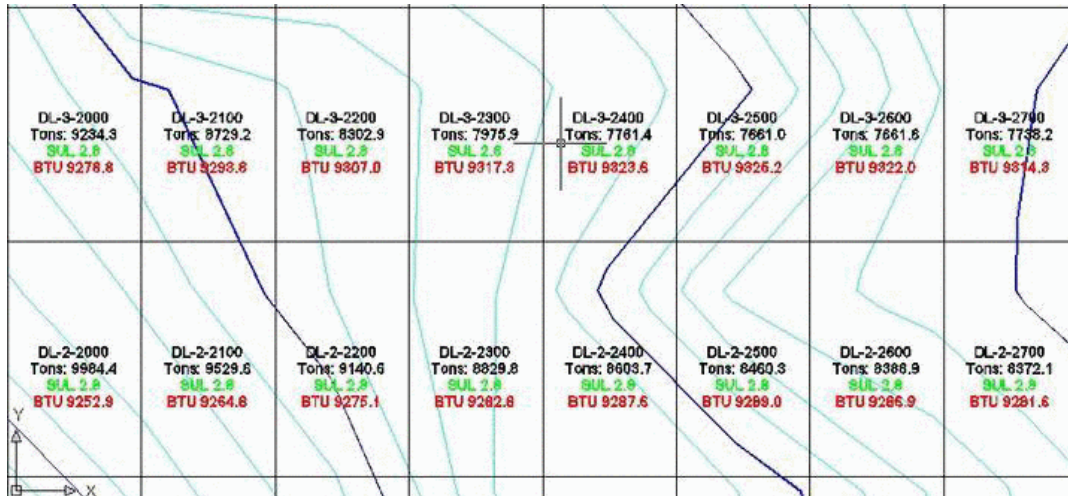
Label Prefix/Suffix: A text prefix/suffix for the label/

Decimals: The number of decimals to use in the label. Note that this will truncate the labels rather than round the value.

Row Position: The row on which the label will be placed.

Label Alignment: Sets the alignment point for the text.

An example set of labeled pits is shown below.



Prompts

Select pit polylines to label.

Select objects: Specify opposite corner: 192 found

Select objects:

Label placement method [**<Align>/Horizontal**]? *H*

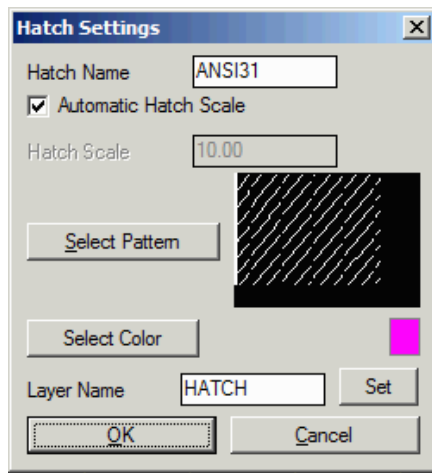
Pulldown Menu Location: Boundary in the Advanced Mine Module under Label Pit Polylines

Keyboard Command: pitlabel2

Prerequisite: There must be quantities assigned to the pits either from Surface Mine Reserves or Import Timing Data. The pits must have direction assigned to be recognized.

Hatch Pits

This command hatches the selected pit areas. A reason to hatch the pits is to visually check that the whole site is covered by pit polylines. In the options dialog, there are settings for the hatch pattern, color, hatch scale and layer name for the hatches to draw.



Prompts

Select pit polylines to hatch.

Select objects: *pick pit polylines*

Hatch Settings dialog

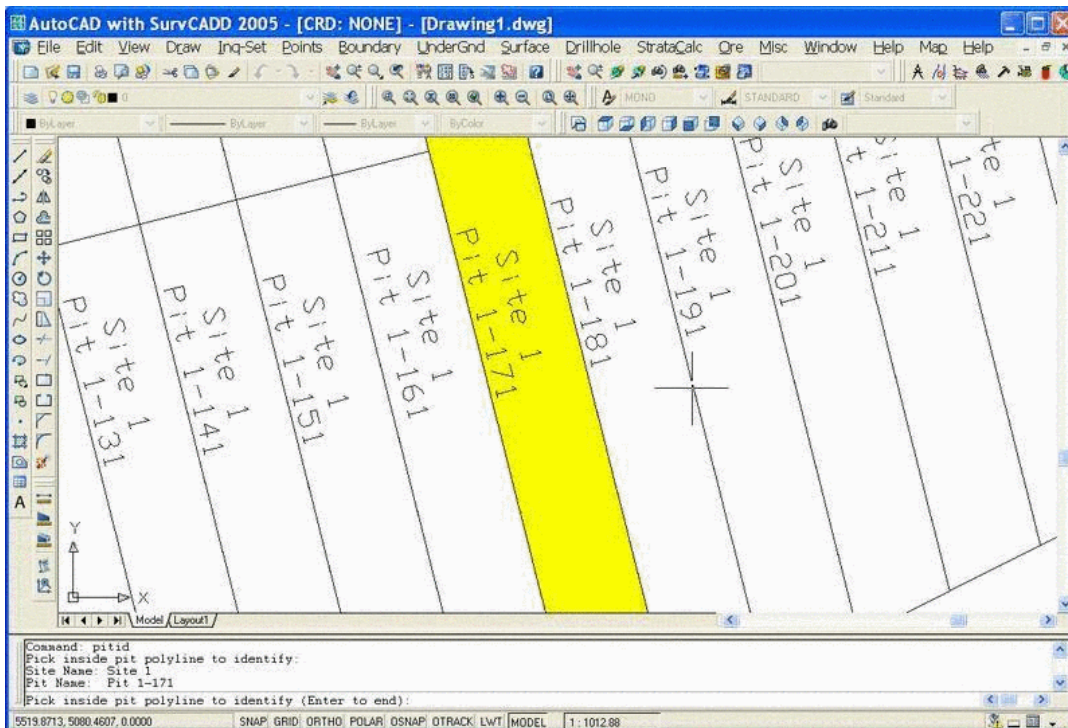
Pulldown Menu Location: Boundary

Keyboard Command: pithatch

Prerequisite: Pit polylines

Identify Pit Polylines

This command reports the pit name and site name that are attached to the selected polyline at the command line and shows the pit graphically with a yellow solid fill.



Prompts

Pick inside pit polyline to identify (Enter to end): *pick inside a polyline* If you pick on or near a common line, one pit is highlighted and you are prompted to Press N for next selection or Enter to accept current, highlighted pit line. The selected pit line is filled with a solid yellow hatch that erases when the next pit is selected, or the command is finished.

Site Name: Site 1

Pit Name: Pit 1

Select pit polyline to identify: *press Enter to end*

Pulldown Menu Location: Boundary in the Advanced Mine Module

Keyboard Command: pitid

Remove Pit Names

This command removes the extended entity data of pit/site names from the selected polylines. For example if you are done calculating for an area, you can use this command to clear the pit names from the polylines in the old area so that these polylines are not picked up as pits anymore by routines such as Surface Mine Reserves and the Timing Routines.

Prompts

Select polylines to remove pit names from.

Select objects: *pick the polylines*

Removed pit names from 11 polylines.

Pulldown Menu Location: Boundary

Keyboard Command: clearpitnm

Check Pit Polyline Overlaps

This command checks whether pit polylines overlap each other. Pits should never overlap since this would lead to extra volumes with the quantities in the overlap area getting reported twice. The program reports any overlaps at the command line.

Prompts

Select pit polylines to check.

Select objects: *pick the polylines*

Pits P-100 and P-101 overlap at 1501884.209,506497.414

Pits P-101 and P-100 overlap at 1504444.952,509533.437

Found 2 overlaps between 56 pits

Pulldown Menu Location: Boundary > Pit Polyline Tools

Keyboard Command: pitcheck

Prerequisite: pit polylines

Pit by Interior Point

This routine is used to create pit polylines from the existing linework by making closed polylines around points picked. The site and pit names are user-specified and the pit name is incremented after each new pit created. This command uses the Boundary Polyline logic, using any linework (open or closed) and searches out from the picked point for the closed perimeter. It draws a new polyline in the selected layer.

Prompts

Label area names (<Yes>/No)? Choose whether or not to label.

Text height <4.00>: *Enter to accept or type new text height*

Auto place labels in center (<Yes>/No)?

Site name <Site 1>:

Pit name <Pit 1>:

Layer name <0>: Enter to accept or type in new layer name.

Pick point inside pit perimeter: *pick inside a loop or linework representing the pit to use as a pit*

Specify another area (<Yes>/No)? If yes is selected, then the Pit Name is automatically incremented by one for efficient entry.

Pull-down Menu Location: Boundary in the Advanced Mine Module

Keyboard Command: pickpit

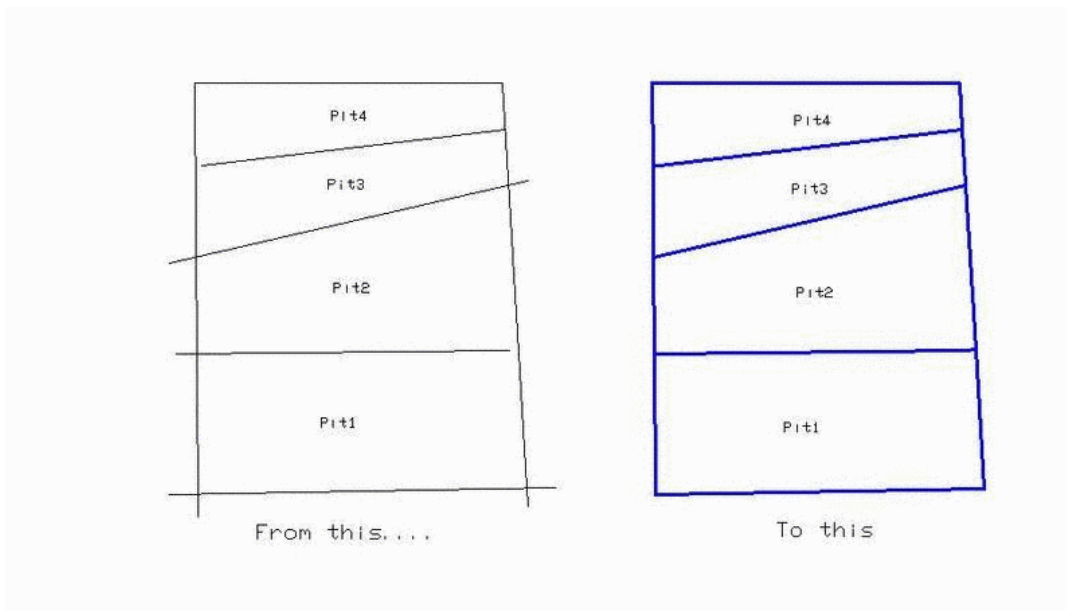
Pit By Interior Text

This command converts lines and polylines that enclose text into Carlson pits, with the text becoming the pit name. It is best used in conjunction with "Sequential Numbers", a command under the "Draw" pull-down menu that allows the user to place text in increasing order. If pits are to be numbered from U10 through U50, Sequential Numbers places this text with a single pick. The text inside the pits does not need to be in the same layer as the polylines/lines making up the pits. Since the text is selected by the user, care should be taken not to select extraneous text such as contour elevations.

The user should also avoid selecting contours and other polylines not associated with the pits. It should also be noted that this command makes entirely new polylines in a new layer from the source lines and polylines. If "Boundary Polylines" was used to make closed polylines, and Sequential Numbers is used to place text, then "Pit By Interior Text" will make a new set of polylines in a new layer (this time as "official" pit polylines), resulting in two sets of identical closed polylines. You should use "Delete Layer" or other Carlson tools such as "Freeze Layer" to delete or hide the duplicated polylines. The command will handle arcs. But note that "Pit By Interior Text" will make polylines out of any combination of polylines and lines enclosing text, even where the lines are shared by two adjacent pits and no duplicated lines exist.

"Sequential Numbers" and "Pit By Interior Text" work together to make pits. The location of the text containing the pit name can be used to assign the direction of mining. Therefore, the very location picked within "Sequential Numbers" can set the direction of mining across the pit. If "Assign Directions" is selected within the "Surface Mining" options, the sub-option "Text" will utilize the text insertion position and start the mining through the pit from the side closest to the insertion point. Note that "Sequential Numbers" will middle-justify the text used for the pit name. To emphasize the linkage with "Sequential Numbers", the prompting sequence shown below includes the prompting for "Sequential Numbers".

Using "Pit Plines from Mineplan" you don't need to use "Boundary Polylines" in advance to create the closed pit polylines. "Pit By Interior Text" makes the new polylines and associates the pits all in one step. This command is a great time saver to convert old mine plans with just linework (sometimes sloppy) and the pit name inside to Carlson pits used in reserves and timing.



Prompts

[Sequential Numbers]

Text Height <8.0>: 50

Pick point at center of label: *pick insertion point for centered text*

Pick point for label alignment: *pick text alignment point*

Number <1>: U10 (or 50 in the second example)

Auto increment labels (Yes/<No>)? Y

Pick point at center of label: *pick next point and U11 (or 51) prints*

Pick point at center of label: *pick next point and U12 (or 52) print, etc.*

[Pit Plines from Mineplan]

Site Name for Pits: Site 1

Layer Name for Pits: PITS

Select pit lines, polylines and text.

Enter snap tolerance <0.0>: This is the size of the gap that the routine can handle to "jump" across and create a closed boundary.

Select objects: *select the text and linework* New pit lines should appear in the selected layer.

Pulldown Menu Location: Boundary

Keyboard Command: minepitlines

Related Commands: Sequential Numbers, Boundary Polylines, Assign Directions

Pit Matrix Layout

The Pit Matrix Layout function will divide a polyline (preferably closed, but not necessary) into surface pits of any arrangement you specify utilizing temporary alignment polylines. The site and pit names are also defined in this routine. This allows for a great deal of imaginative pit configurations and layouts around difficult geometric and individual situations. The routine will insert the specified number of rows and columns of pits in a matrix form directly inside the baselines, and if used, crosslines. The mine boundary will then serve as an inclusion perimeter and create pits in the matrix form only inside of it.

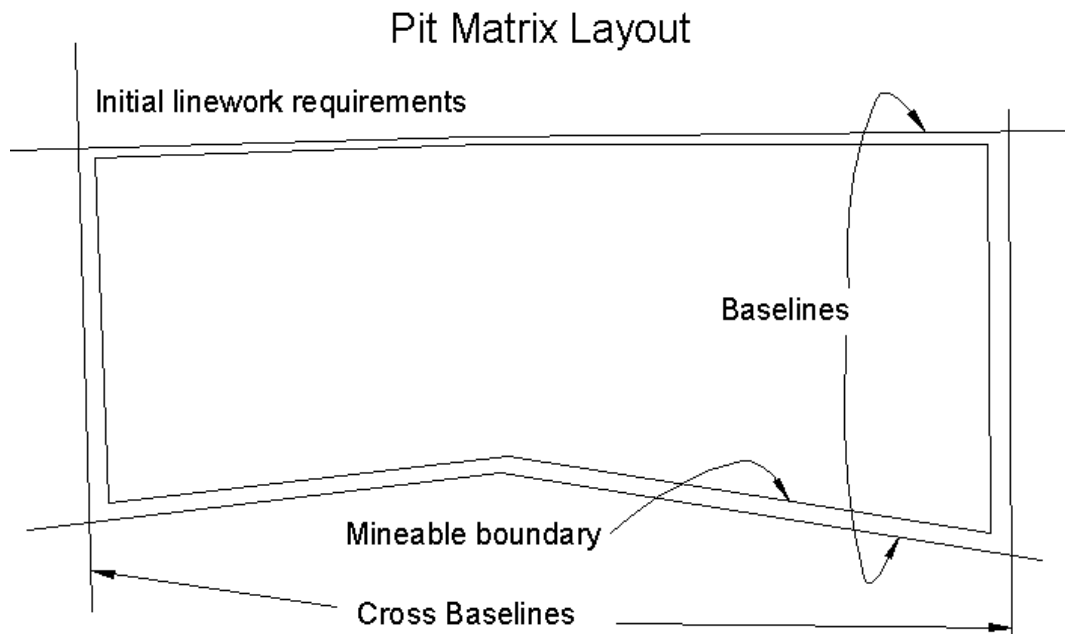
Prompts

Select mine boundaries:

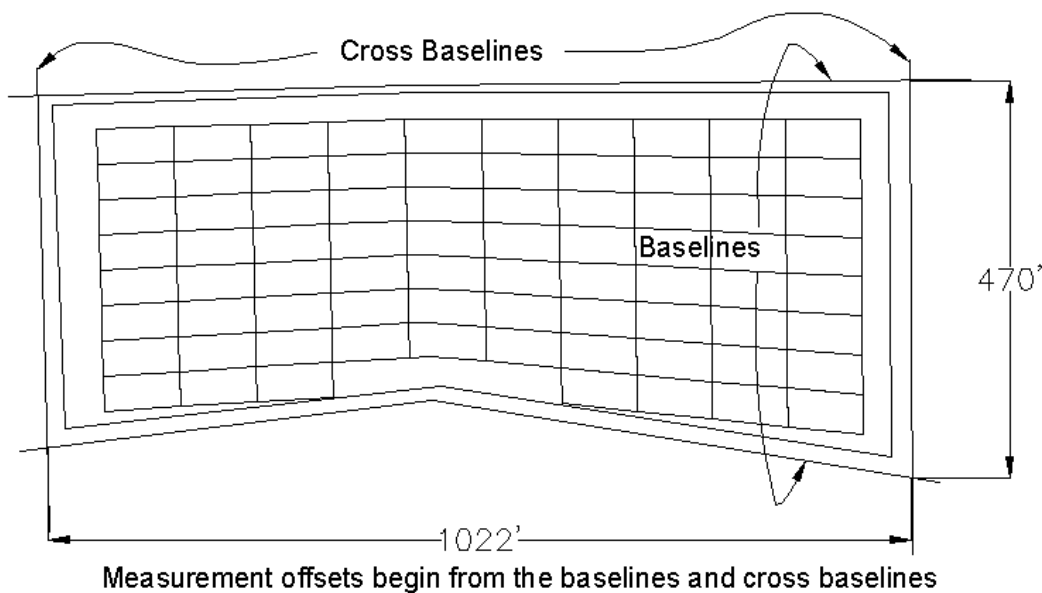
Select TWO baselines: *select objects* The best pit configurations result from baselines with equal number of vertices.

Select TWO crosslines or hit Enter for none: *select objects* Similarly with the baselines, the crosslines should have the same number of vertices.

Note: Baselines are required. Crosslines are not. If just long strips of pits are desired, then just use baselines with no crosslines and enter in 1 for number of pits in the Cutting Across Baselines tab. It is also important to note that the order the base and cross lines are selected will determine where the pit naming will begin. The first baseline selected will be where Pit 1 will start, for example. The same applies to cross lines, for sub pit or block labeling. If there are no crosslines used, then the direction the baselines were drawn will affect the naming of the blocks or sub-pits.



Baselines and crosslines can and probably should be drawn directly on top of the boundary to get exact pit sizing starting at the boundary. They are shown off the boundary line here for clarification of what they should look like. The matrix will begin at the base and cross lines, wherever they are drawn.



With the initial linework completed and knowledge of the general dimensions of the layout, the user can use the

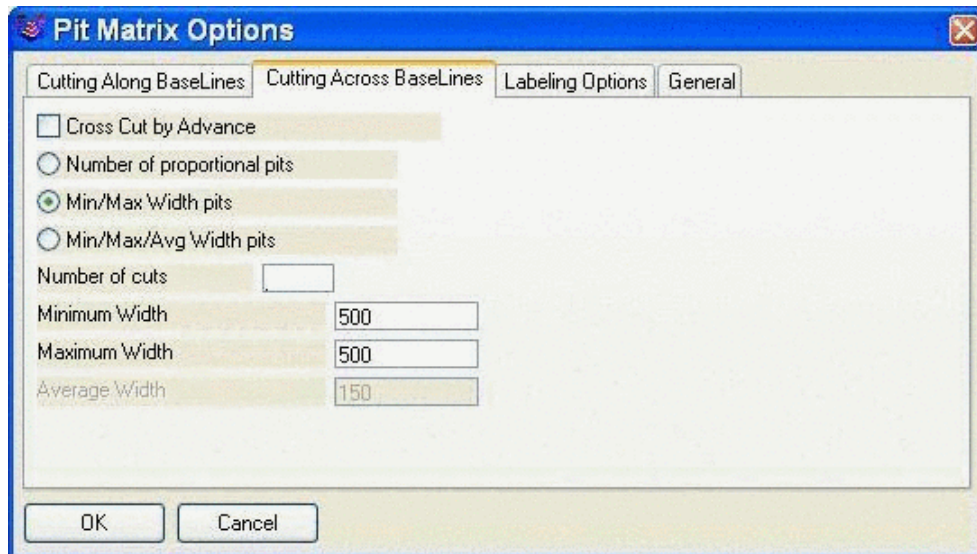
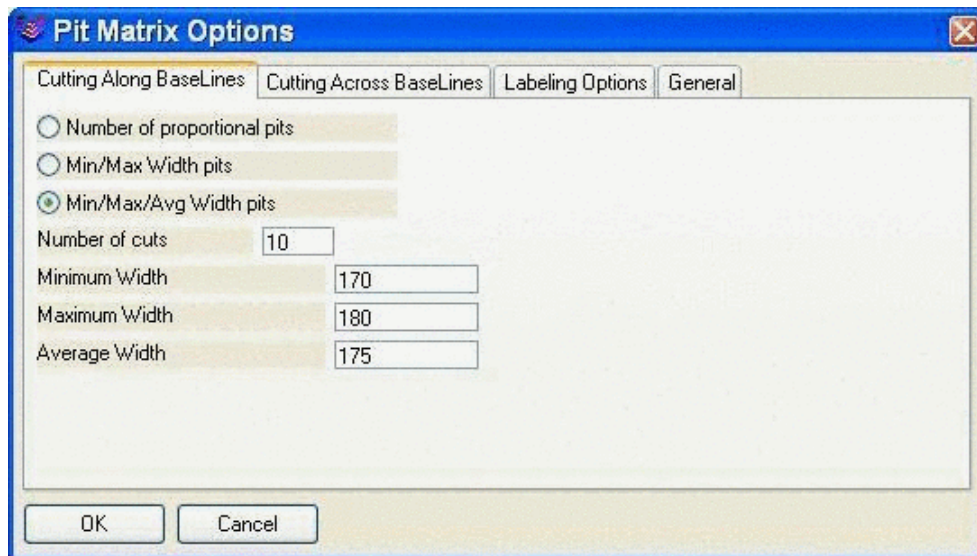
following options.

Number of proportional pits: This will measure the distance between the baselines and divide the pits proportionally, evenly distributing the size of the pits as they are cut along the baselines using the Number of cuts entry. If the Min/Max options are used, Number of cuts will still be applied. For example, if only 10 pits are desired in a certain layout direction, the routine will stop at 10 pits, then a different plan may be applied from there.

Min/Max Width pits: Enter in the minimum and maximum pit widths that will be allowed and the routine will stay within that range to fan pits around corners etc. If the min and max are the same, then all pits will be that width.

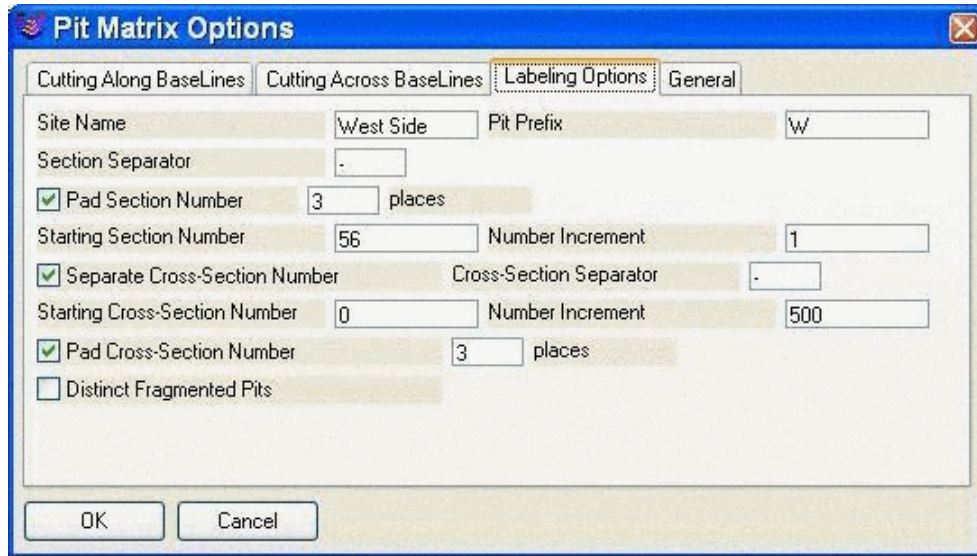
Min/Max/Avg Width Pits: This is very similar to Min/Max Width, except the average value will be the width that it will use unless it has to fan around a corner. Again, if the minimum and maximum and average are the same, say 175, then all pits will be 175 wide.

The Cross Cut by Advance option in the Cutting Across Baselines Tab should be checked if a certain pit or block size is desired, with little flexibility. Everything is grayed out if this is selected and the block sizing will begin at the cross lines.

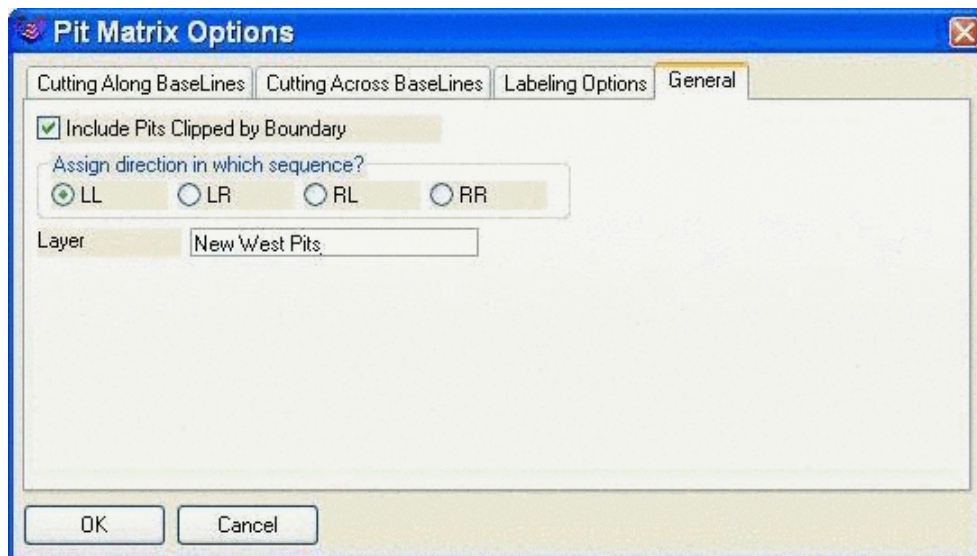


Labeling Options: Consider the pit and block scenario. A pit is defined by the base lines and broken into blocks by the cross lines. For labeling, enter a Site Name and Pit Prefix. Both base lines and cross lines can have a section separator (here a dash -). The total number of places can be set with Pad Section and Cross-Section Number so that a resulting pit number such as 056 (for 3 places) can be obtained. The starting pit (section) and block (cross-section) numbers are set in their own windows. The increment is set for both pit and block, here the pit is set to 1 and the

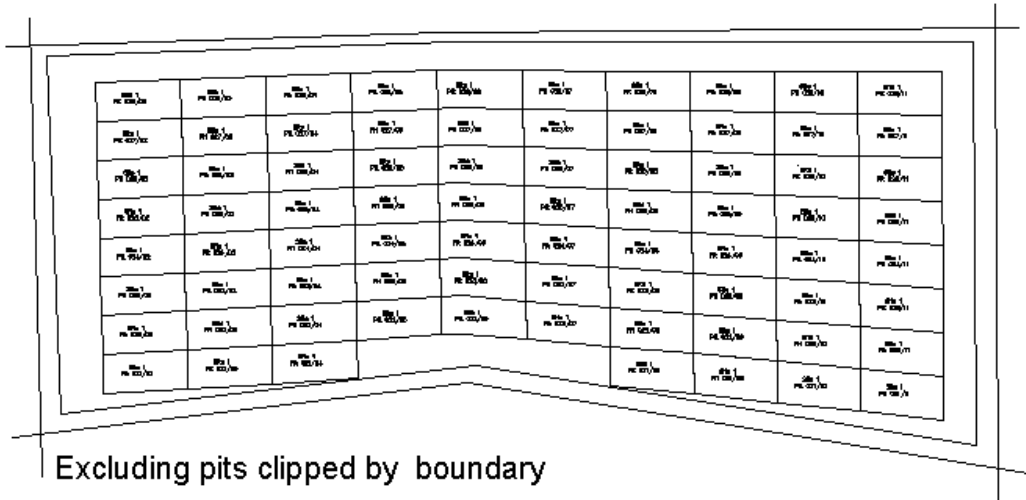
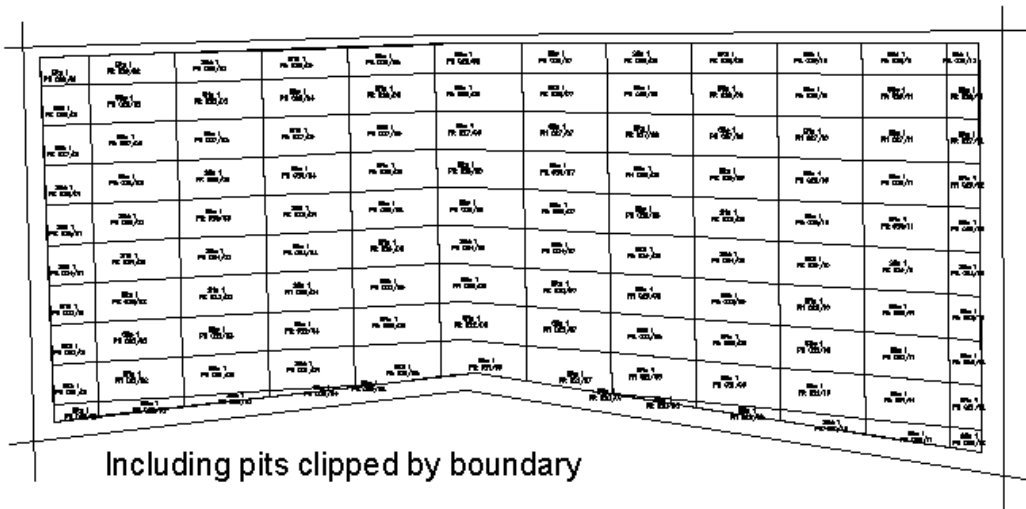
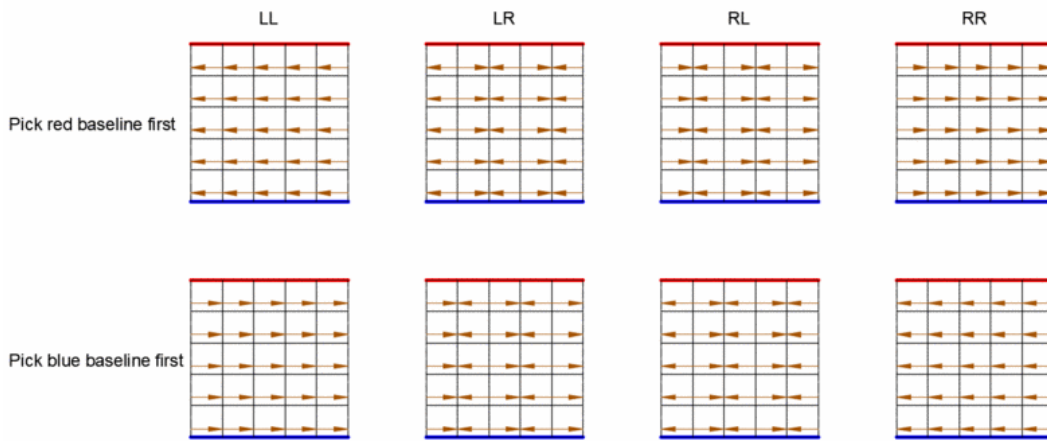
block is set to 500. The first few pits in this example will be named: W-056-000, W-056-500, W-057-000 and so on. Distinct Fragmented pits will give a unique name to a pit that is broken due to a gap in the boundary. For example, if the long dragline pit is broken by a washout of coal, and this option is checked, then the pit will have two names: Pit 56a and Pit 56b. It automatically uses a, b, c, etc. If it isn't selected, then both will be named Pit 56.

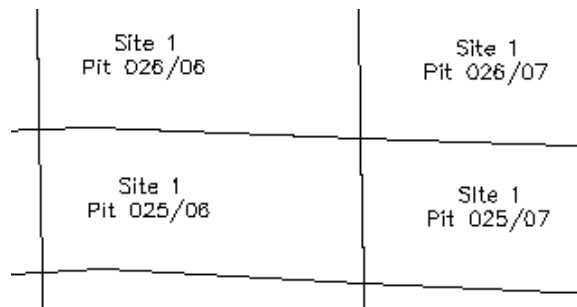


General: The option to include clipped pits (any pit that touches the boundary) is found under the General tab. An example of both options is shown below. The layer of the new pits is set in the Layer box. The pits are assigned direction for timing with the "Assign direction in which sequence?" option. If you imagine looking down the boundary, from the first selected base line to the second, LL will be mining from the left in each pit, to the right. LR will be from the left in the first pit, then back from the right in the second pit etc. RL and RR are just the opposite.

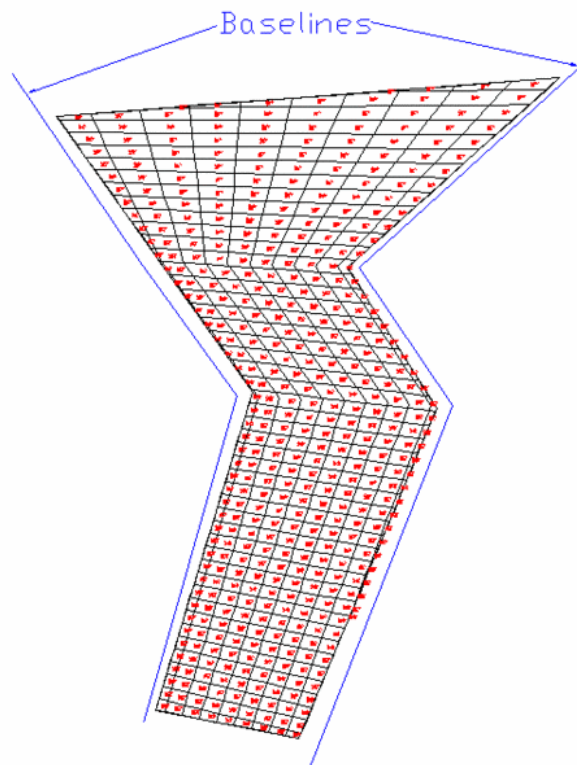


The below image shows an example of the various types of pit directions that may be assigned.





Labeling options allow special separators.



When geometric layouts for the pits become more complicated, Pit Layout by Matrix will allow the user to design the pits to most any situation presented. It is extremely useful for "fanning" of the pit where corners or varied widths are encountered. This example above shows how a matrix of [10 X 50] can be fit inside a varied boundary such as this angular mine boundary.

Prompts

Select mine boundaries: *select the boundary*

Select objects: 1 found

Select TWO baselines: *select the baselines* Pay attention to which is selected first for name direction.

Select objects: 2 found

Select TWO crosslines or hit Enter for none:

Select objects:

Pulldown Menu Location: Boundary

Keyboard Command: cross_pit

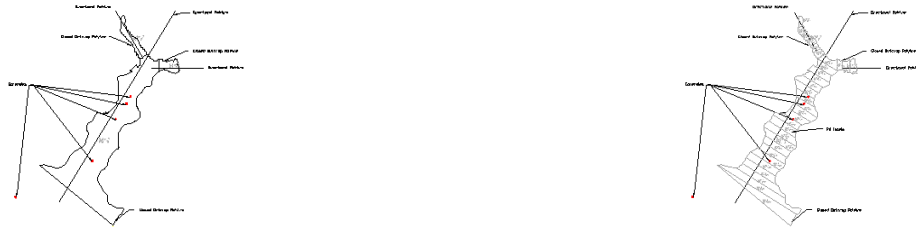
Prerequisite: The routine requires a polyline boundary for the pits to be designed inside of (better if closed), two baselines, one on each side of the boundary, and optionally, two crosslines to further define the extent of the "sides"

of the pits.

Pit Layout by Advance

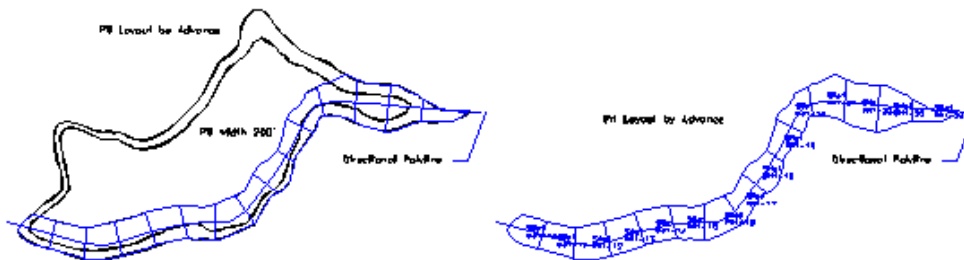
This command creates pits of the desired advance width by intersecting a closed pit polyline and a direction line indicating the direction the pits will be developed. In the first 2 examples below the outcrop has been created as the outer limit of the pit. The first example shows the impact of laying out a straight direction polyline. The direction polyline must enter the pit boundary from outside and exit the boundary to be valid. The program advances along the direction polyline the distance entered for the Cut Advance and intersects for each pit 90 degrees from the direction line. There are two options on where to start the advance. The first is at the end of the direction line and the second is where the direction line intersects the pit boundary. Most of the time the second option is used. The main use for starting the advance at the beginning of the direction line is if all direction lines start at a known "station", then all pit blocks will be uniform, named by station and in line with each other.

Example 1 uses a straight directional polyline and was done in 3 sections. Each pit is drawn 90 degrees from it at the desired Cut Advance.



Example 1

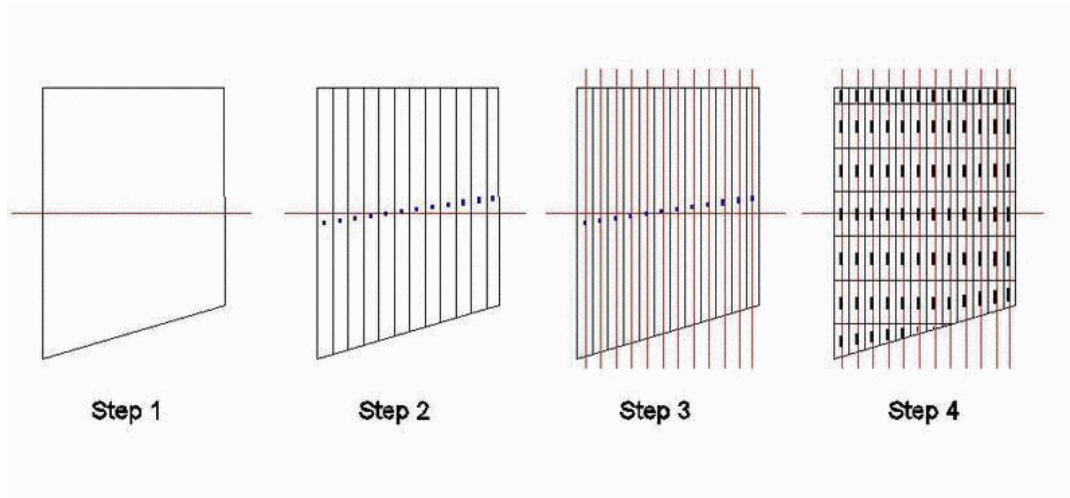
Example 2 uses a bent directional polyline to follow the contour of the outcrop. Here it uses a bent polyline to conform to the contour of the outcrop. The direction polyline cannot contain any arcs, however an arc can be initially drawn for the layout and converted to polyline segments using the command *Remove Polyline Arcs* on the Edit menu. Again, each pit is created by intersecting the direction line at a 90 degree angle. Going around corners will often look better with a smooth line, hugging the inside corner.



Example 2

This third example will go through two iterations of Pit Layout by Advance to first create long strips of pits 175 wide, then break them into blocks that are 500 long, named with the appropriate stationing in the pit name. This is a unique application that is not apparent by just running the command, so a simple case study will follow. Step 1 is to create a mine boundary and draw a direction line through it. Step 2 is the first iteration of Pit Layout by Advance. This example used 175 for the pit width and created long strips of pits with a unique name, such as A-8 and A-9. Step 3 will prepare to break them up into blocks. First, create direction lines through each pit strip. The fastest way to do this is to draw the first direction line through the first strip. Then the ARRAY command will copy that multiple times, placing a direction line down the middle of each pit strip. Running the Pit Layout by Advance will first prompt to select the pit boundaries. Select all of the boundaries at once with a window, fence or by picking. Then it

prompts to select the direction lines, select them all at once. In this type of example, the option to start advance at the beginning of the direction lines is chosen. This gives blocks that are named by station along the direction lines. Notice on the south end where the pit strips are not horizontal. Not using the option to calculate from the starting of the advance line will give pit blocks that are offset and not named by station along the direction centerlines. The pit blocks are created and can be labeled with Label Pit Polylines. Notice the pit names and the stationing of the pit blocks incrementing by 500, which is the length of each block.



A-2-1500	A-3-1500	A-4-1500	A-5-1500	A-6-1500	A-7-1500	A-8-1500	A-9-1500	A-10-1500	A-11-1500	A-12-1500
A-2-1000	A-3-1000	A-4-1000	A-5-1000	A-6-1000	A-7-1000	A-8-1000	A-9-1000	A-10-1000	A-11-1000	A-12-1000

Prompts

Select all pit polylines.

Select objects: *pick pit polylines*

Select all direction polylines.

Select objects: *pick direction polylines*

Enter Cut Advance: *175*

Enter Starting Sub-Pit Number: *1*

Enter Sub-Pit Number Increment (1): *1*

Calculate the advance from the start of the direction line [Yes/<No>]? *N*

The highlighted pit does not have site and pit names assigned

Enter a site name: *Site 2*

Enter a pit name: *A*

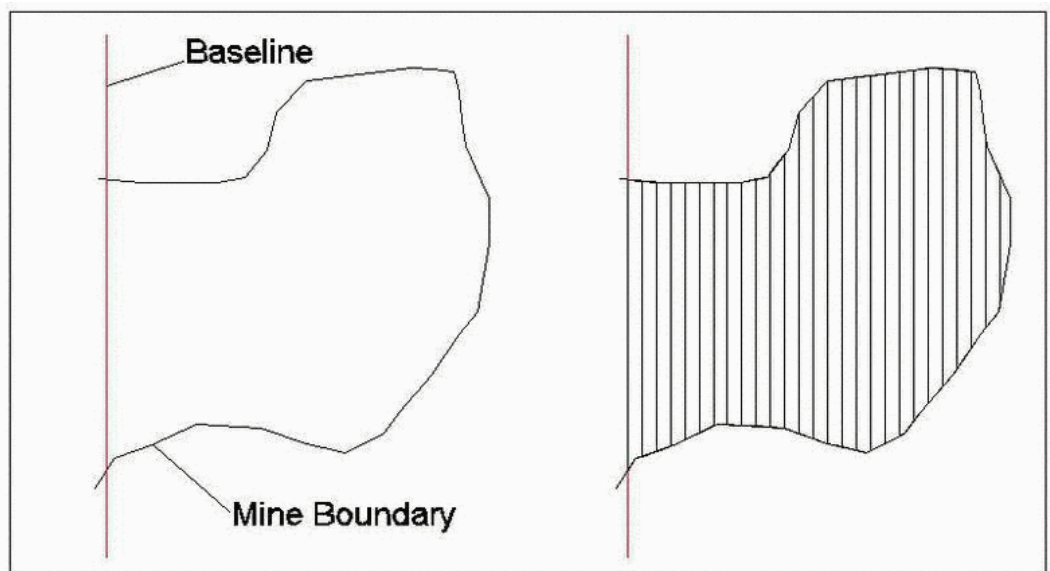
Pulldown Menu Location: *Boundary*

Keyboard Command: *cutpitadv*

Prerequisite: A direction polyline and a closed pit area polyline.

Pit Layout by Width

This command creates closed polylines that represent pit inclusion areas for Surface Reserves and Timing. The pit polylines are created by advancing a baseline polyline by a fixed pit width. The sides and end of the pit polylines are defined by the minable boundary polyline. This polyline should be open at the end of the baseline polyline. Be sure to create the baseline polyline longer than the minable boundary so that the program can always find the intersection of the advancing baseline with the minable boundary. There is an option to automatically assign pit and site names to the polylines. The name of the first pit is set in the dialog shown below. The next pits will be named by incrementing the pit name by one. There is an option to prompt for each pit width or use the same width for all the pits.



Prompts

Pit Sizing dialog Set the pit width, layer and names

Select baseline polyline: *pick the polyline*

Select minable boundary polyline: *pick the polyline*

Pulldown Menu Location: Boundary

Keyboard Command: pitsz2

Prerequisite: One open polyline boundary and one baseline polyline extending across the boundaries open end.

Pit Layout by Rate

This command creates closed polylines that represent pit inclusion areas for Surface Mine Reserves or the Surface Timing routines. The pit polylines are created by advancing a baseline polyline along a minable boundary. The sides and end of the pit polylines are defined by the minable boundary polyline. This polyline should be open at the end of the baseline polyline. Be sure to create the baseline polyline longer than the minable boundary so that the program can find the intersection of the advancing baseline with the minable boundary. The next pits will be named by incrementing the pit name by one. The baseline polyline is advanced a distance such that the pit volume equals the target volume.

The surface model is either represented by contours on screen or a grid file. If neither is available, there is an option to Use drillhole surface elevations in surface model. The Ignore zero elevations will ignore anything on screen with no elevation. When modeling from the drillholes, there are 5 modeling methods available, which are defined elsewhere in detail. If a roof and floor grid exist of the strata to target, then use the Grid File option and these two grids will be prompted for. If Triangulation is the selected modeling method, then the options to Use Triangulation Subdivision (defined in Triangulate and Contour) and Use Global Trend Extrapolation (explained in Make Strata Grids) are active. There is the box to create a new layer name for the new pits. If coming from drillholes, there is the option to choose which strata to include. If Selected is checked, then the following Choose Strata box appears. Holding the CTRL or Shift buttons down allow for multiple strata selection. The grid cell size and resolution are set in the Make 3D Grid window, also only if coming from drillholes.

Pit Layout Parameters

The cubic yards or meters / hour is set here with the shift hours per day and number of days worked per period. This will calculate the Target Pit Size. In this example, 480,000 CY of OB_TOP strata is the targeted pit size. Turning on the Name Pits box activates the Site and First Pit Name windows. The subsequent pit names will be incremented by one from the starting pit name.

Pit Sizing

Source of surface model ...

File Screen

Use drillholes surface elevations in surface model

Ignore zero elevations

Modeling Method

Triangulation Inverse Dist Kriging Polynomial Linear Least Squares Grid File

Use Triangulation Subdivision Use Global Trend Extrapolation

Layer Name

Which strata to include ...

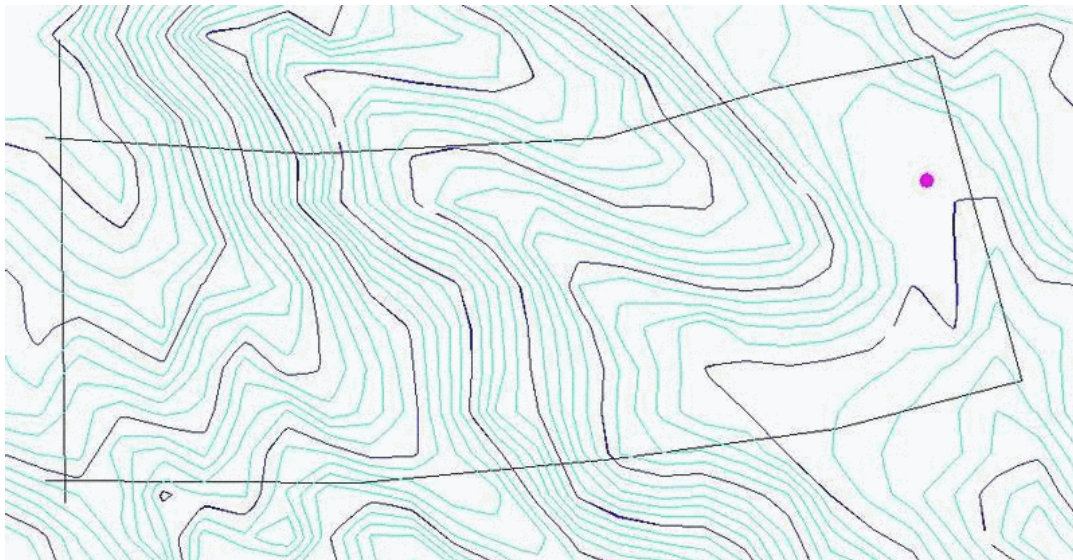
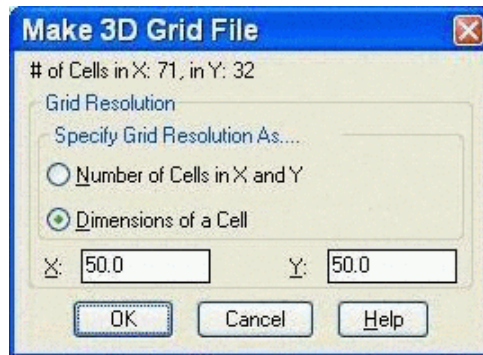
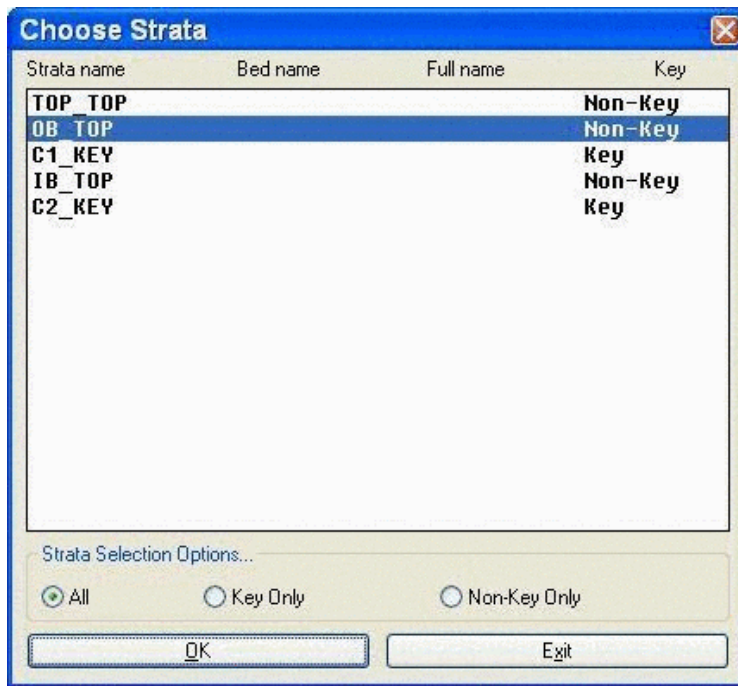
All Selected

Pit Layout Parameters

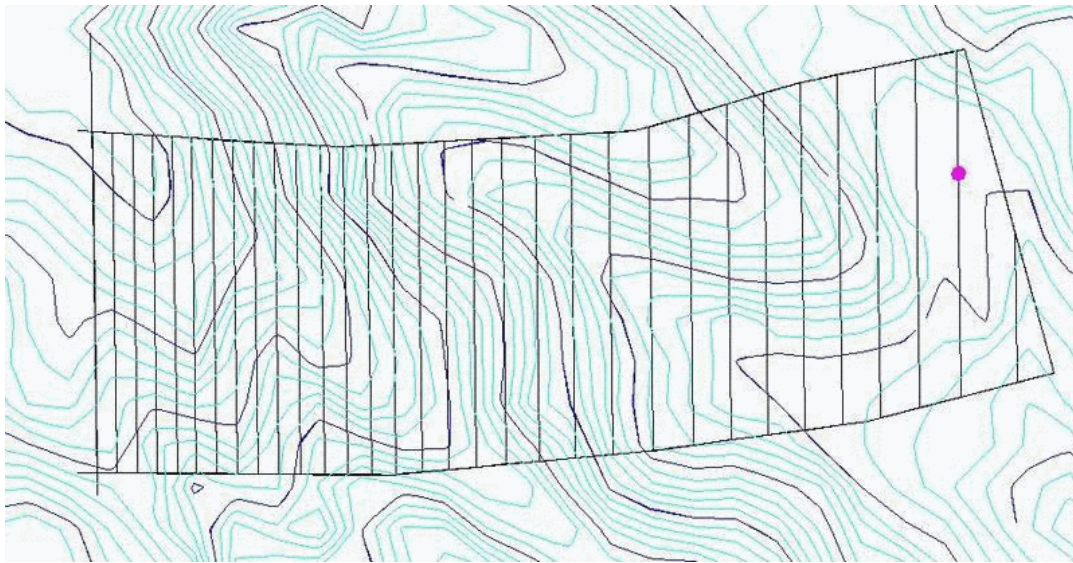
C.Y. Overburden/Hour Shift Hours/Day

No. of Days Target Pit Size (C.Y.)

Name Pits Site Name First Pit Name



Starting linework



Final pit layout. Notice that the pits get wider to the east as the OB.TOP thins and the pits must be wider to mine 480,000 CY.

Prompts

Pit Sizing dialog

Select surface entities & at least 3 drillholes

Select objects: *select the drillhole symbols and the contour polylines*

Select mine boundary polyline: *pick the minable boundary*

Make Grid dialog Choose a grid resolution.

Use inverse distance to which power (First/<Second>/Third/Other)? *press Enter*

Select baseline polyline: *pick the baseline polyline*

Select minable boundary polyline: *pick the minable boundary again*

Pulldown Menu Location: Boundary in the Advanced Mine Module

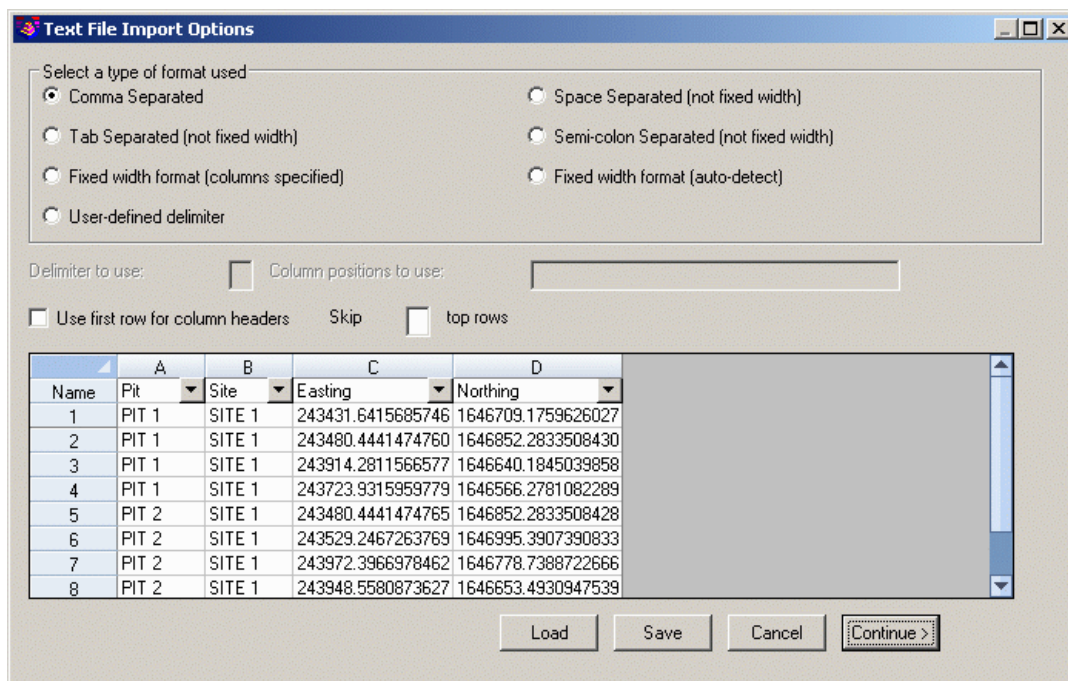
Keyboard Command: pitsz

Import Pit Points

This creates pit polylines using pit names and geometry data stored in a text file. The fields in the text file must contain pit name, site name, easting and northing. Here's a sample text file for two pit polylines:

```
PIT 1,SITE 1,243431.6415685746,1646709.1759626027
PIT 1,SITE 1,243480.4441474760,1646852.2833508430
PIT 1,SITE 1,243914.2811566577,1646640.1845039858
PIT 1,SITE 1,243723.9315959779,1646566.2781082289
PIT 2,SITE 1,243480.4441474765,1646852.2833508428
PIT 2,SITE 1,243529.2467263769,1646995.3907390833
PIT 2,SITE 1,243972.3966978462,1646778.7388722666
PIT 2,SITE 1,243948.5580873627,1646653.4930947539
PIT 2,SITE 1,243914.2811566577,1646640.1845039858
```

After selecting the file to import and the layer for the pit polylines, there is a dialog to set the order of the fields and the delimiter between fields. To import the example, fill out the Names in the top row of the spreadsheet as shown.



Prompts

Import Pit Points dialog

Input Data File Select text file to import

Imported 9 pits.

Pulldown Menu Location: Boundary

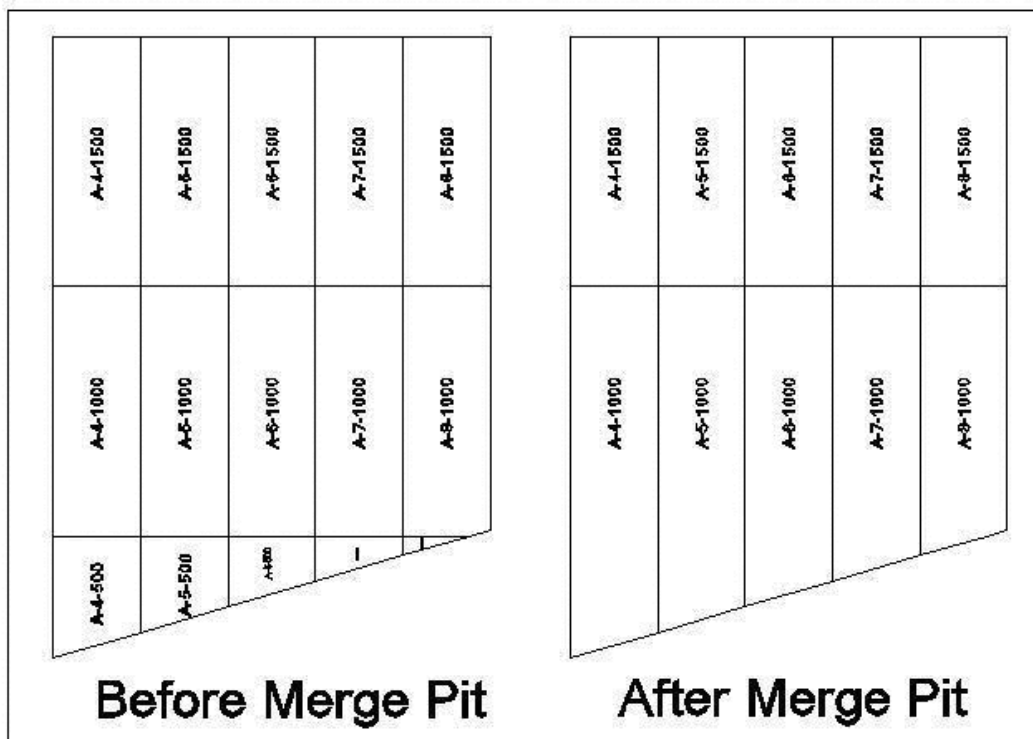
Keyboard Command: importpit2

Prerequisite: Text file with pit data

Merge Pits

This routine takes two adjacent pits or blocks and merges them to create just one. Routines such as Pit Matrix Layout can sometimes leave small irregular shaped pits along complex boundaries. Usually, the user would like to combine or add a small sliver of a pit with an adjacent one so that the volume is reported, but not as its own tiny pit. The volumes are combined if they are stored in the pits as values. This command will also remove precedence rules that would be invalid following the merge. For example, if Pit 2 has a precedence rule that does not allow it to be mined until after Pit 1 has been mined, but Bench 1 is removed by this command, the precedence rule in Pit 2 will also be removed.

There are two methods for selecting the pits to merge. The Pick method prompts to pick points inside the pits to merge. The Size method prompts for a minimum area size and then a selection of pit polylines to process. Then any pits less than the minimum area are merged with a neighboring pit. For the pick method, pick first in the pit you want to keep, then in the pit you want to remove. A new polyline is drawn around both, representing the new pit with the same name as the first pit you picked inside of. The last step is to simply erase the text of the smaller, deleted pit (if they were labeled). If a pit is selected on or near a line, then the pit is highlighted and you are prompted to hit Enter to accept the pit or press N to highlight the other nearby pit.



Prompts

Select pits by individual pick or size filter [**<Pick>/Size**]? *press Enter*

Pick inside 1st pit polyline to merge: *pick a point*

Pick inside 2nd pit polyline to merge: *pick a point*

Created a shrink-wrap polyline successfully.

If the pick is near a pit line, then the following prompts appear:

Pick inside 1st pit polyline to merge (Enter to end):

Press N for next selection or Enter to accept current:

Pick inside 2nd pit polyline to merge:

Press N for next selection or Enter to accept current:

Created a shrink-wrap polyline successfully.

Done.

Pulldown Menu Location: Boundary

Keyboard Command: mergepit

Prerequisite: Two adjacent Carlson named pits that you want to combine into one pit.

Assign Directions

In order to schedule equipment through pits, pits (which are closed polylines) must have three distinct types of Extended Entity Data (EED, extra information) associated with them. They must have (1) a pit name, (2) NonKey Volume, Key Volume and Key Tons and optionally quality information and (3) direction of mining. The Assign Directions command places the direction of mining into the pit itself, where it is permanently stored along with other aspects of the drawing. Directions can vary by bench. The command prompts to assign the same direction to the Whole pit, or by bench. If doing by bench, the routine must be run separately for each bench.

Assign Directions is found within Boundary in the Carlson Surface Mining. There are six methods employed to assign direction: Automatic, Text, Sequence, Linework, Bearing, Azimuth and Gon. The "automatic" method will

mine "longways" across the pit, following the longest axis detected, but may not choose the preferred direction along that axis. The text method finds the side closest to the insertion point of the pit name and will mine from that side perpendicular across the pit away from the text. The sequence method will mine left to right and/or right to left across a series of pits as specified by the user, the linework method will follow a "direction polyline, line or leader" across a pit or series of pits and the Bearing or Azimuth method will mine at defined bearing or azimuth angle. Below is the prompting and results obtained with each method:

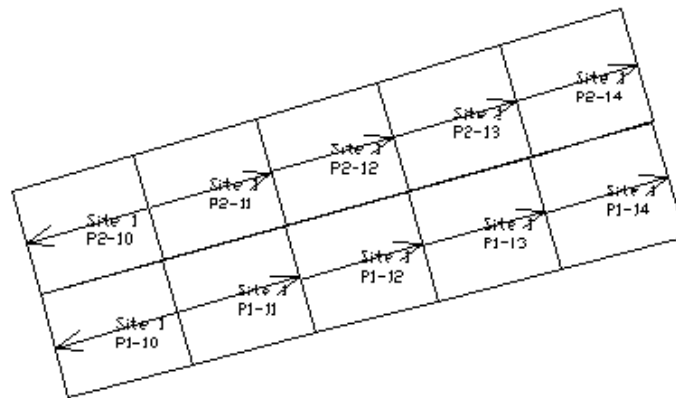
Prompts

Assign direction using which method [/Text/Sequence/Linework/Bearing/AZimuth/Gon]: *A or press Enter*

Select pit polylines to have direction assigned to:

Select objects: *Pick the pits and the direction is assigned as shown below*

In this example, the left most pits were given a direction opposite from the remaining pits. For long-term studies or quick estimates using smaller pits, the direction can be considered a moot point, since you will get where you're going to be whether you mined left to right or right to left through an individual pit. But for larger pits or short-term studies, direction is critical. A pit direction can be easily reversed using the *Reverse Directions* command.



Next we will look at the Text option.

Assign direction using which method [/Text/Sequence/Linework/Bearing/AZimuth/Gon]: *T*

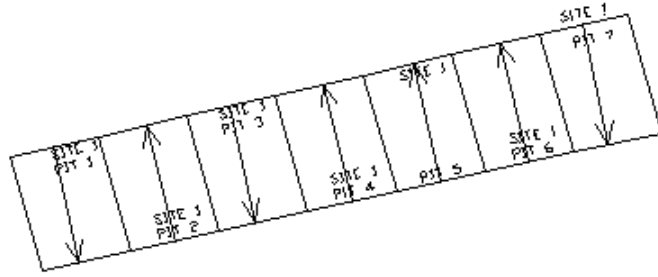
Select pit polylines to have direction assigned to:

Select objects: *pick the pits and the required pit text as shown below*

The highlighted pit has the direction information assigned already.

Would you like to overwrite this (<No>, Yes, None, All): *A for All.* This prompt only occurs when pits are chosen that already have pit direction. Note that the command "Clear Directions" could be used to remove directions prior to using Assign Directions.

The arrows show the resulting directions. Note that this routine looks for the pit name for the direction, not for the "site" name. (All pits have a two-tiered naming convention: site and pit, which can be re-worked as pit and block or any other two-level form, to adapt to company practices.) In Pit 5, for example, the "Site 1" text is ignored and the "Pit 5" text is used for direction. In Pit 7, the Site 1 text is completely outside the pit, but is irrelevant since only the Pit 7 text is used for direction. If pits were named Pit 1, Block U15A, then the insertion position of "Block U15A" would be used for setting directions.



Next we will look at the Sequence option.

Assign direction using which method [`<Auto>/Text/Sequence/Linework/Bearing/AZimuth/Gon`]: *S*

Select pit polylines to have direction as signed to:

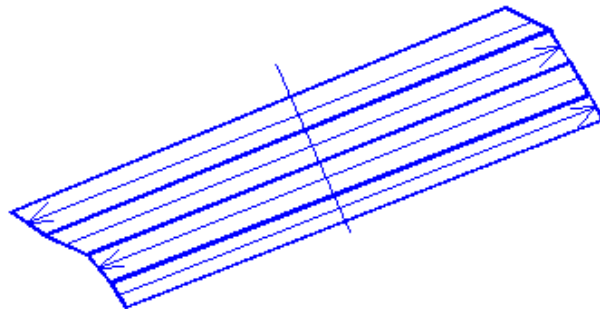
Select objects: *select the pits* (Don't worry if you also select other polylines. It only finds pits.)

Select a direction polyline.

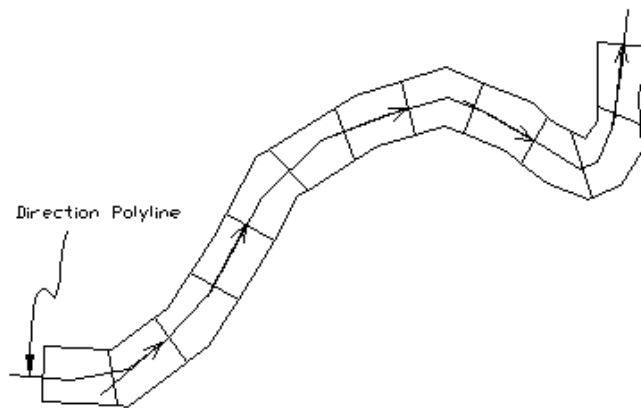
Select objects: *select a polyline that crosses all of the pits* This would be the short south to north polyline in our example.

Assign direction in which sequence (`<LL>`, `LR`, `RL`, `RR`): *LR*

In our example, we selected LR, which causes the pits to be mined left to right on the first pit (with respect to the south-to-north direction polyline—imagine yourself standing at the beginning of the polyline looking down it. LL would mine all pits coming from your left. LR would mine from the Left first, then the Right in the next one and so forth. RR and RL are just the opposite.), then right to left on the second pit, then left to right on the third, etc. An entry of LL would cause the pits to be mined from the left side to the right side. An entry of RR, for example, would mine all the pits from right to left. The entry of RL would mine first right to left, then left to right. The sequence method is ideal for assigning direction to a series of long and narrow pits that have not been broken up into small blocks.



If these same pits were each subdivided into 10 or more blocks, or there are many pits that would not be intersected by the sequence polyline, for example, then the following method, direction by polyline, is most appropriate.



Next we will look at the Polyline option.

Assign direction using which method [`<Auto>/Text/Sequence/Polyline/Bearing/AZimuth`]: *P*

Select pit polylines to have direction assigned to:

Select objects: *select the pits* (Don't worry if you also select other polylines. It only finds pits.)

Select all direction polylines.

Select objects: *In the example above, select the single direction polyline.*

The polyline-based selection is ideal for contour mines or outer pits in a mountaintop removal situation in Appalachia, or for pits that follow sinuous property lines. Most Midwest and western pit mining is linear, but the polyline technique for setting direction is still applicable. Direction polylines cannot have arcs, so if you've used an arc to draw the polyline, use the command *Remove Polyline Arcs* on the Edit menu to remove them.

Last we will look at the Bearing and Azimuth option

Bearing Option:

Assign direction to whole pit or to a specific bench [`<Whole>/Bench`]: *W* or Enter to assign direction to whole pit

Assign direction using which method [`<Auto>/Text/Sequence/Linework/Bearing/AZimuth/Gon`]: *B* for Bearing

Select pit polylines to have direction assigned to:

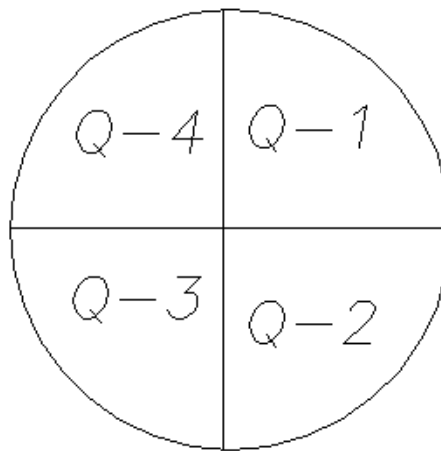
Select objects: *select the pits* (Don't worry if you also select other polylines. It only finds pits.)

Enter Bearing (Qdd.mmss): 145.0000

The highlighted pit has the direction information assigned already.

Would you like to overwrite this [`<No>/Yes/None/All`]: *A*

The Bearing is entered in the format (Qdd.mmss), the following figure shows the quarter numbers and the angle is calculated clockwise.



Azimuth Option:

Assign direction to whole pit or to a specific bench [\langle Whole \rangle /Bench]: W or Enter to assign direction to whole pit

Assign direction using which method [\langle Auto \rangle /Text/Sequence/Linework/Bearing/AZimuth/Gon]: B for Bearing

Select pit polylines to have direction assigned to:

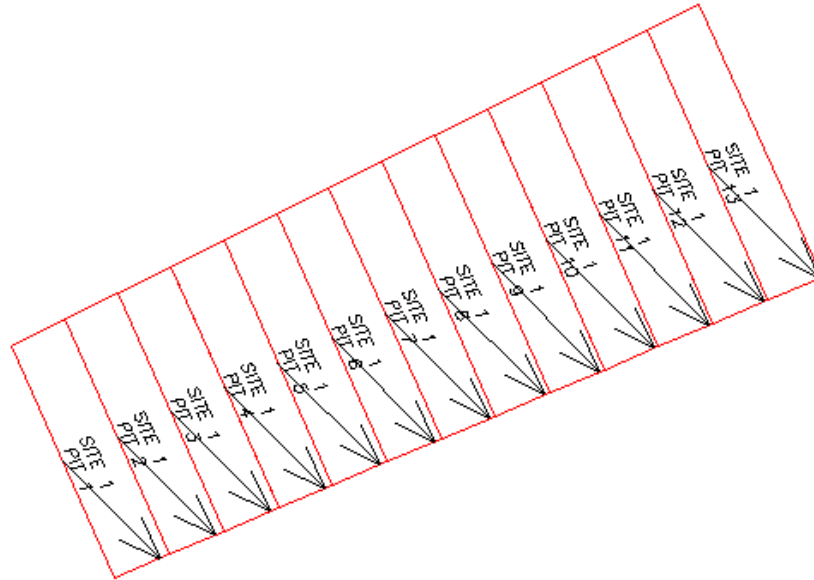
Select objects: *select the pits* (Don't worry if you also select other polylines. It only finds pits.)

Enter Azimuth (ddd.mmss): 135.0000

The highlighted pit has the direction information assigned already.

Would you like to overwrite this [\langle No \rangle /Yes/None/All]: A

The Azimuth angle is calculated from the true north.



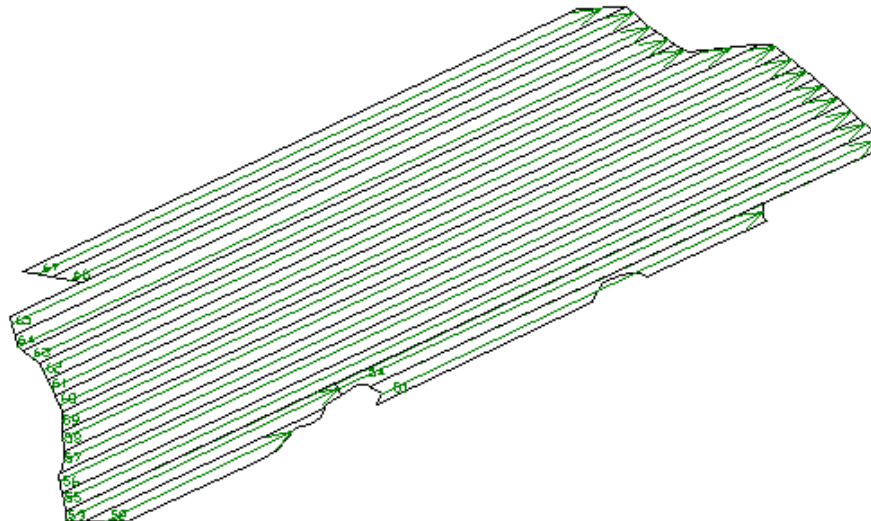
Pulldown Menu Location: Boundary

Keyboard Command: assign_dir

Related Commands: Display Directions, Reverse Directions, Clear Directions, Create Pit Plines from Mineplan

Display Directions

If directions have been assigned to pits, this command will display the directions. It therefore serves two purposes: (1) to verify that directions have, in fact, been assigned previously, and (2) to review the direction of mining. When pit directions are detected, arrows are displayed as shown below. These direction arrows will disappear with any "Zoom" command such as Pan or Window, and will also disappear if a Redraw or Regen is executed. The direction leaders can be used to draw direction arrow entities in the drawing. The sample drawing is a good demonstration of the use of Assign Direction by Text.



Prompts

Display direction to whole pit or to a specific bench [<Whole>/Bench]: W

Draw directions as leaders or temporary arrows [Leaders/<Arrows>]? L for Leaders or A for Arrows

Select pit polylines to have direction displayed:

Select objects: *Pick the pits to review*

Pull-down Menu Location: Boundary

Keyboard Command: display_dir

Related Commands: Assign Directions, Reverse Directions, Clear Directions, Create Pit Plines from Mineplan

Reverse Directions

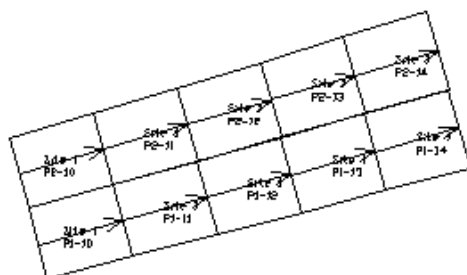
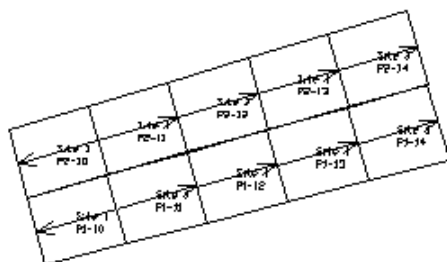
This command reverses the direction of mining within a pit. It is particularly useful in conjunction with the command Assign Directions, <Auto>, since the automatic mode may assign direction to many of the pits opposite from the desired direction. The result is shown below, where the two pit 10s were reversed.

Prompts

Reverse direction to whole pit or to a specific bench [<Whole>/Bench]:

Select pit polylines to have direction reversed:

Select objects:



Pull-down Menu Location: Boundary

Keyboard Command: reverse_dir

Related Commands: Assign Directions, Display Directions, Clear Directions, Create Pit Plines from Mineplan

Clear Directions

This command removes the directions from pits. The Extended Entity Data (EED) designating pit direction is removed from the pit polyline entity. This command has no effect if direction has not been previously assigned to a pit. Keep in mind that it is not necessary to first remove "old" directions before assigning "new" directions. The Assign Direction command will recognize that directions exist and will prompt the user to overwrite the direction for individual pits or for all pits.

Prompts

Clear direction to whole pit or to a specific bench [<Whole>/Bench]:

Select pit polylines to have direction removed:

Select objects: *pick the pit polylines* This can be done individually or with a window.

Pulldown Menu Location: Boundary

Keyboard Command: clear_dir

Related Commands: Assign Directions, Display Directions, Create Pit Plines from Mineplan

Assign Pit Precedence

This command will automatically reassign the precedence for multi-bench pits. If the pits have been edited or cleared of precedence, this command will restore them to the default setting. Bench 2 will always require that Bench 1 gets mined first. Bench 3 will always require that Bench 2 gets mine first and so forth. This is the default setting that should be there, unless the pits have been edited. This will ensure the benches will not under-mine themselves. This command also detects pit overlap and automatically sets precedence accordingly, if the overlap area is greater than the tolerance area.

Prompts

Enter Overlap Area Tolerance (ft²) <0.0>: *Specify the allowed tolerance area*

Select pit polylines to process.

Select objects: Specify opposite corner: 3 found

Select objects:

Done.

Pulldown Menu Location: Boundary > Pit Timing Quantities

Keyboard Command: pitprec

Related Command: Edit Pit

Pit Precedence by Polylines

This command allows you to quickly assign precedence to timing blocks by drafting directional polylines to indicate the direction of precedence. Before the command is executed, a polyline must be drawn on the PRECEDENCE layer, and must be over the pits to be modified. It is important to note that a vertex must exist inside each pit boundary to be affected (a polyline with only 2 vertices will only modify 2 pits).

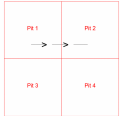
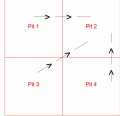
When the command is executed, you will be prompted to select the pits to be modified and the Precedence polylines. You will then be prompted to select the benches to be affected. A brief description of each method is given below.

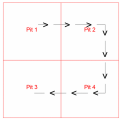
- **Single** - This option will affect a single bench. The pit at the start of the precedence polyline will not have a precedence rule assigned to it.
The second pit along the polyline will be modified so that it cannot be mined until the first pit is mined.
The third pit will be modified so that it cannot be mined until the second pit is mined.
This process repeats down the length of the polyline
- **All** - This option will affect all benches, but each bench is analyzed separately. This produces the same result as using the Single option on each bench separately.
- **Upward Staircase** - This option will affect multiple benches in a staircase method, with higher benches being mined after lower benches.
In this case, the highest bench in the first pit will not have a precedence rule assigned to it.
The second pit will be modified so that the highest bench cannot be mined until the lower bench of the first pit is mined.
The third pit will be modified so that the highest bench of the third pit cannot be mined until the lower bench of the second pit is mined.

- Downward Staircase - This option is the inverse of the Upward Staircase method, with lower benches being mined after higher benches.
 In this case, the lowest bench in the first pit will not have a precedence rule assigned to it.
 The second pit will be modified so that the lowest bench cannot be mined until the higher bench of the first pit is mined.
 The third pit will be modified so that the lowest bench of the third pit cannot be mined until the higher bench of the second pit is mined.

A simple example using three of the four assignment methods is provided in the below table. Here the black polylines with arrows represent the Precedence polylines.

**Drawing - 3 Method
Bench Exam-
ple**

	Single - Bench Upward		Stair- Downward			
	1 Only	case	Stair-	Downward		
	Mine This	Before Mining	Mine This	Before Mining	Mine This	Before Mining
	This	This	This	This	This	This
	Pit1-Bench1	Pit2-Bench1	Pit1-Bench1 Pit1-Bench2 Pit1-Bench3	- Pit2-Bench1 Pit2-Bench2	Pit1-Bench1 Pit1-Bench2 Pit1-Bench3	Pit2-Bench2 Pit2-Bench3 -
	Pit1-Bench1	Pit2-Bench1	Pit1-Bench1 Pit3-Bench1 Pit4-Bench1	- Pit2-Bench1 Pit2-Bench2	Pit1-Bench1 Pit1-Bench2 Pit1-Bench3	Pit2-Bench2 Pit2-Bench3 -
			Pit3-Bench1 Pit3-Bench2 Pit3-Bench3	- Pit2-Bench1 Pit2-Bench2	Pit3-Bench1 Pit3-Bench2 Pit3-Bench3	Pit2-Bench2 Pit2-Bench3 -
			Pit4-Bench1 Pit4-Bench2 Pit4-Bench3	- Pit2-Bench1 Pit2-Bench2	Pit4-Bench1 Pit4-Bench2 Pit4-Bench3	Pit2-Bench2 Pit2-Bench3 -



Pit1-Bench1	Pit2-Bench1	Pit1-Bench1	-	Pit1-Bench1	Pit2-Bench2
Pit3-Bench1	Pit4-Bench1	Pit1-Bench2	Pit2-Bench1	Pit1-Bench2	Pit2-Bench3
Pit4-Bench1	Pit3-Bench1	Pit1-Bench3	Pit2-Bench2	Pit1-Bench3	-
		Pit2-Bench1	-	Pit2-Bench1	Pit4-Bench2
		Pit2-Bench2	Pit4-Bench1	Pit2-Bench2	Pit4-Bench3
		Pit2-Bench3	Pit4-Bench2	Pit2-Bench3	-
		Pit4-Bench1	-	Pit4-Bench1	Pit3-Bench2
		Pit4-Bench2	Pit3-Bench1	Pit4-Bench2	Pit3-Bench3
		Pit4-Bench3	Pit3-Bench2	Pit4-Bench3	-

Pulldown Menu Location: Boundary

Keyboard Command: pit2plprec

Prerequisite: Pit Polylines and directional polyline(s) on the PRECEDENCE layer

Clear Pit Precedence

This command will erase the precedence for multi-bench pits. Removing the precedence is somewhat dangerous, as now Bench 2 can be mined before Bench 1. It is recommended to run Assign Pit Precedence after clearing them, to restore them to the default setting.

Prompts

Select pit polylines to process.

Select objects: Specify opposite corner: 6 found

Select objects:

Done.

Pulldown Menu Location: Boundary > Pit Timing Quantities

Keyboard Command: clear_pitprec

Clear Pit Bench Quantities

This command removes all quantity values and grid paths from the selected pits. There is the option to clear all benches at once, or if Specific is selected, then it will prompt to enter a bench number to clear.

Prompts

Clear all bench quantities or a specific bench [All/<Specific>]? A

Select pit polylines to clear.

Select objects: Specify opposite corner: 5 found, 5 total

Select objects:

Cleared 10 bench quantities.

Pulldown Menu Location: Boundary > Pit Timing Quantities

Keyboard Command: clearpit

Related Commands: Edit Pit, Assign Timing Grids, Import Timing Data

Remove Empty Benches

This routine looks at all the pits and benches and removes the bench completely from the pit if the volume and/or tons are less than the specified tolerance. This is useful to remove very small quantities that have been stored in pits by mistake, or just from calculations on the edges of the data. Running this command will remove the pit bench with volume below the tolerance, so when running timing routines, the pits won't show up. This is useful for the 3D Pick option in Surface Equipment Timing to remove upper benches that are in the pit, but have no volume assigned.

This command will also remove precedence rules that would be invalid following the merge. For example, if Bench 2 has a precedence rule that does not allow it to be mined until after Bench 1 has been mined, but Bench 1 is removed by this command, the precedence rule in Bench 2 will also be removed.

Prompts

Command: emptypit

Key volume tolerance (CY) <0.0>: 100

Non-Key volume tolerance (CY) <0.0>: 100

Key tons tolerance <0.0>: 100

Select pit polylines to remove empty benches.

Select objects: all

7 found

Removed 2 empty benches.

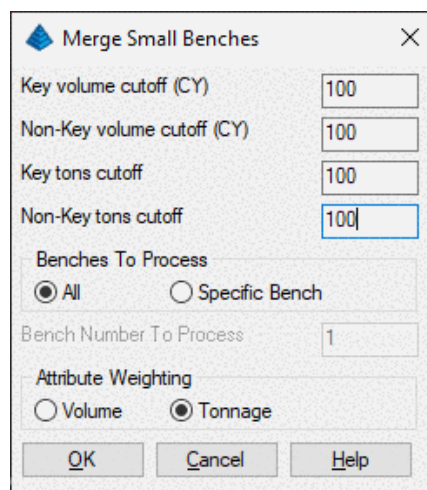
Pulldown Menu Location: Boundary, Pit Timing Quantities

Keyboard Command: emptypit

Prerequisite: Pits with bench quantities assigned

Merge Small Benches

This command combines benches in a pit polyline that have quantities less than a set tolerance. The program prompts for the cutoff tolerances for the bench quantities and then prompts for the pit polylines to process. For each pit polyline, the program checks for a bench with all quantities less than the set tolerances and then moves these quantities to the next bench in the pit polyline and removes the bench with the small quantities.



Volume/tons cutoff: In order for a bench to be merged to a lower bench, all four cutoff values must be greater than

the volumes on the bench being considered. For example, if a bench contains 150 CY of nonkey material and 50 CY of key material, and the cutoff for all values is 100 (as shown above), then the bench would not be merged to a lower bench because the nonkey volume is greater than the cutoff value.

Benches to Process: This option determines if all benches will be considered for merging, or just a specific bench. If all benches are considered, the program will start at the uppermost bench and check to see if it can be merged to a lower bench. The next bench will then be considered, all the way down the pit. Note that if multiple benches can be merged, they will. For example, if both benches 1 and 2 are very small, they could both be merged into bench 3 if their sum is less than the cutoff. If The **Specific Bench** option is selected, then any small benches overlying the specific bench will be considered for merging. For example, if benches 1 and 2 contain small volumes/tonnage, while bench 3 contains significant volumes/tonnage, processing bench 3 would merge both benches 1 and 2 into bench 3.

This command will also remove precedence rules that would be invalid following the merge. For example, if Bench 2 has a precedence rule that does not allow it to be mined until after Bench 1 has been mined, but Bench 1 is removed by this command, the precedence rule in Bench 2 will also be removed.

Attribute Weighting: This option determines if quality values will be averaged based on the volume or the tonnage of the material.

Key attributes will be weighted by either the key tonnage or the key volume.

Nonkey attributes will be weighted by either the nonkey tonnage or the nonkey volume.

Any attributes that include `_TON_FACTOR` or `_VOL_FACTOR` will be handled in such a way that the sum of all similar attributes does not exceed 1.000, as these attributes represent the percentage of each strata contributing to the total volume/tonnage of key or nonkey material in the bench. Any of these attributes ending in "_NK" refer to the nonkey portion of the bench, whereas attributes without this suffix refer to the key portion of the bench.

The below example shows two benches before merging (left) and after merging (right). Notice how different strata were merged into a single bench, but the relative percentage of `TON_FACTOR` and `VOL_FACTOR` has been recalculated so that the sum of all key tonnage/volume percentages does not exceed 1.000, and the sum of all nonkey tonnage/volume percentages does not exceed 1.000. In this example, attributes were weighted by tonnage, and the cutoff values were set to very large values to allow these benches to merge.

	SITE 1-PTT 1-B1		SITE 1-PTT 1-B2	
Bench		1		2
Name	SITE 1-PTT 1-B1		SITE 1-PTT 1-B2	
NonKey Volume (CY)	92534.722	<input type="button" value="File"/>	185069.444	<input type="button" value="File"/>
NonKey Shift Volume (CY)	0.000		0.000	
NonKey Tons	106183.595	<input type="button" value="File"/>	212367.188	<input type="button" value="File"/>
Key Volume (CY)	138802.083	<input type="button" value="File"/>	231336.806	<input type="button" value="File"/>
Key Tons	159275.390	<input type="button" value="File"/>	265458.985	<input type="button" value="File"/>
Difficulty		<input type="button" value="File"/>		<input type="button" value="File"/>
Precedence	▼	<input type="button" value="Edit"/>	SITE 1-PTT 1-B1 ▼	<input type="button" value="Edit"/>
Start Date				
Key Thickness	14.991		24.984	
NonKey Thickness	9.994		19.987	
Area (SF)	250000.000		250000.000	
Area (Acres)	5.739		5.739	
Centroid X	3589.739		3589.739	
Centroid Y	1622.339		1622.339	
	Attributes		Attributes	
DENSITY	80.0000	<input type="button" value="File"/>	85.0000	<input type="button" value="File"/>
BTU	9000.0000	<input type="button" value="File"/>	7000.0000	<input type="button" value="File"/>
C1_TON_FACTOR	1.0000	<input type="button" value="File"/>		<input type="button" value="File"/>
C1_VOL_FACTOR	1.0000	<input type="button" value="File"/>		<input type="button" value="File"/>
DENSITY_NK	155.0000	<input type="button" value="File"/>	150.0000	<input type="button" value="File"/>
OB_TON_FACTOR_NK	1.0000	<input type="button" value="File"/>		<input type="button" value="File"/>
OB_VOL_FACTOR_NK	1.0000	<input type="button" value="File"/>		<input type="button" value="File"/>
C2_TON_FACTOR		<input type="button" value="File"/>	1.0000	<input type="button" value="File"/>
C2_VOL_FACTOR		<input type="button" value="File"/>	1.0000	<input type="button" value="File"/>
IB_TON_FACTOR_NK		<input type="button" value="File"/>	1.0000	<input type="button" value="File"/>
IB_VOL_FACTOR_NK		<input type="button" value="File"/>	1.0000	<input type="button" value="File"/>

	SITE 1-PTT 1-B2	
		2
	SITE 1-PTT 1-B2	
	277604.167	<input type="button" value="File"/>
	0.000	
	318550.750	<input type="button" value="File"/>
	370138.889	<input type="button" value="File"/>
	424734.500	<input type="button" value="File"/>
		<input type="button" value="File"/>
	SITE 1-PTT 1-B1 ▼	<input type="button" value="Edit"/>
	39.975	
	29.981	
	250000.000	
	5.739	
	4487.436	
	1543.727	
	Attributes	
	83.1250	<input type="button" value="File"/>
	7750.0000	<input type="button" value="File"/>
	0.6250	<input type="button" value="File"/>
	0.6250	<input type="button" value="File"/>
	151.6667	<input type="button" value="File"/>
	0.6667	<input type="button" value="File"/>
	0.6667	<input type="button" value="File"/>
	0.3750	<input type="button" value="File"/>
	0.3750	<input type="button" value="File"/>
	0.3333	<input type="button" value="File"/>
	0.3333	<input type="button" value="File"/>



Prompts

Merge Small Benches dialog

Select pit polylines to merge benches.

Select objects: *pick the pit polylines to process*

Pulldown Menu Location: Boundary

Keyboard Command: pitmerge

Prerequisite: Pit polylines with quantities

Assign Pit Attributes

This command assigns the attributes to the pits that are in the current PTA file defined with the Mining Project Manager. The only reason to do this is if more than one PTA file is being used for different areas of the mine, or the attributes have completely changed, and new ones need to be assigned. These may be verified with the Edit Pit command.

Prompts

Enter Bench Number to Assign Qualities To:1

Select pit polylines to have qualities assigned:

Select objects:

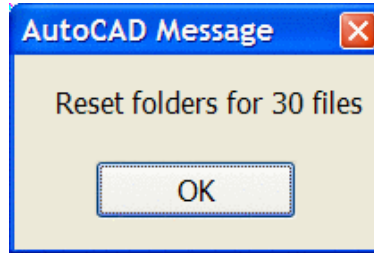
Pulldown Menu Location: Boundary > Pit Timing Quantities

Keyboard Command: setpitattr

Related Commands: Default Pit Attributes, Edit Pit

Reassign Pit Attr Grids Folder

This command will reassign the pit attribute grids folders to a new location if they are moved to a new location or computer. It will prompt for the first grid on the list, then reset all to the same location.



Prompts

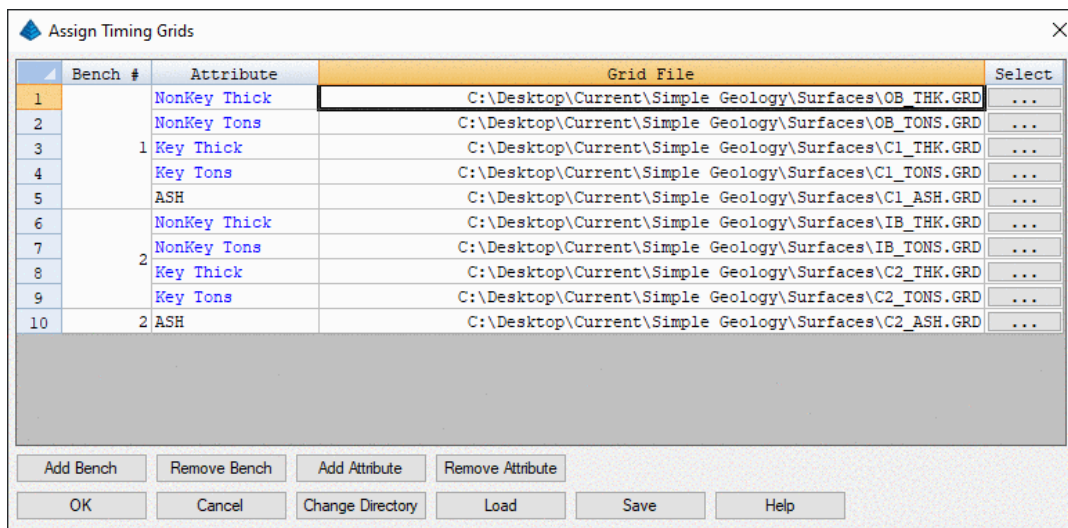
Select the first grid in the list.

Drop-Down Menu Location: Boundary

Keyboard Command: resetpitattr

Assign Timing Grids

This command will assign key thickness, non-key thickness, key tonnage grid files, and non-key tonnage a series of pits or timing blocks. This is an alternate, manual way of assigning grids instead of through the Output Thickness Grids option in the Surface Mine Reserves. These grids are associated with a bench number and stored in the pit polylines. This data is used by the Surface Equipment Timing routine, and these four values are required. Optionally, attribute grids can be assigned here for each bench. You can use the Pit Report or Edit Pit routines to verify which grids are assigned for each bench. The key tonnage grid is tons/ft² (or tonnes/m²). Tonnage grids are easily made with the thickness grid and a density grid in Grid File Utilities. For example: Thickness Grid (feet) x Density (tons/cubic foot) = tons/square ft. The following dialog box allows for interactive entry so all benches can be entered and viewed together as one step. The blue colored text represents required attributes and the black colored text represents optional, user added attributes.



Prompts

Assign Timing Grids Dialog Box.

Select pit polylines to assign grids.

Select objects: *select polylines with pit/site names*

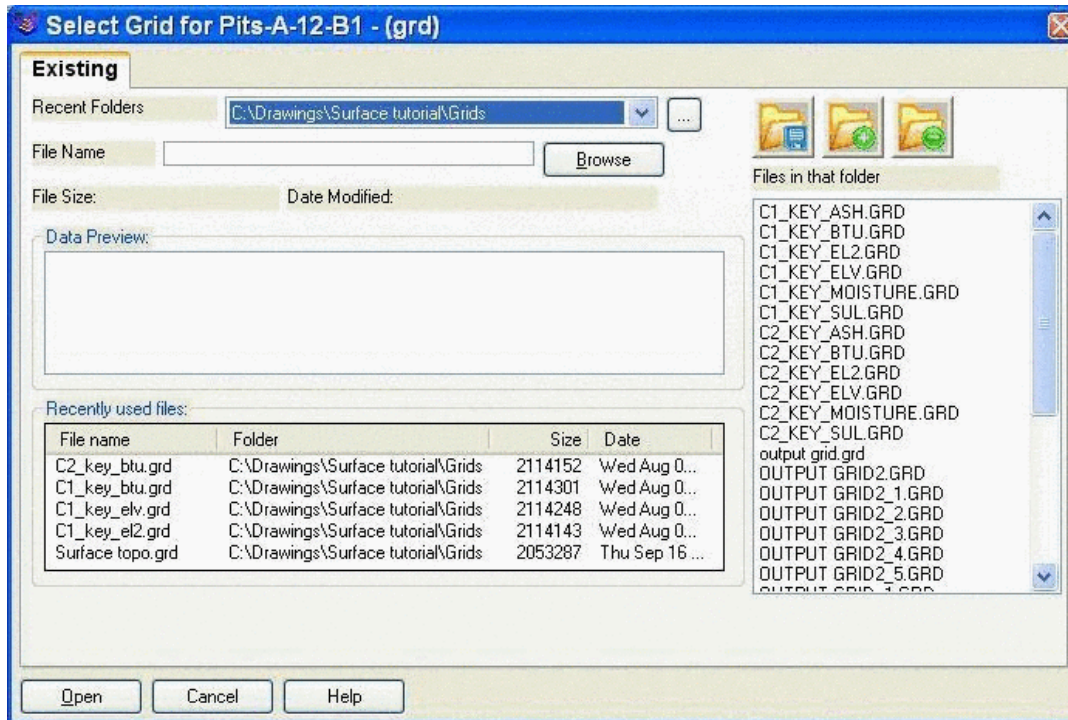
Pulldown Menu Location: Boundary

Keyboard Command: tmgrids

Prerequisite: Key thickness grid, non-key grid, key tons grid, polylines with pit/site names

Reassign Timing Grids Folder

This command reassigns the path of the pit attribute grids stored in the pit perimeters. It is useful when moving the grids folder to a new location, the entire pit attributes do not need to be reassigned, just run this command to point the pit perimeters to the new grid locations. An useful AutoCAD command is XDLIST to view the extended entity data stored within the polyline. This shows the current path of the pit attributes.



Prompts

Select the site/pit perimeter polylines.

Select objects: Specify opposite corner: 10 found

Select objects:

Reset grids for 8 pit perimeters.

Pulldown Menu Location: Boundary > Pit Timing Quantities

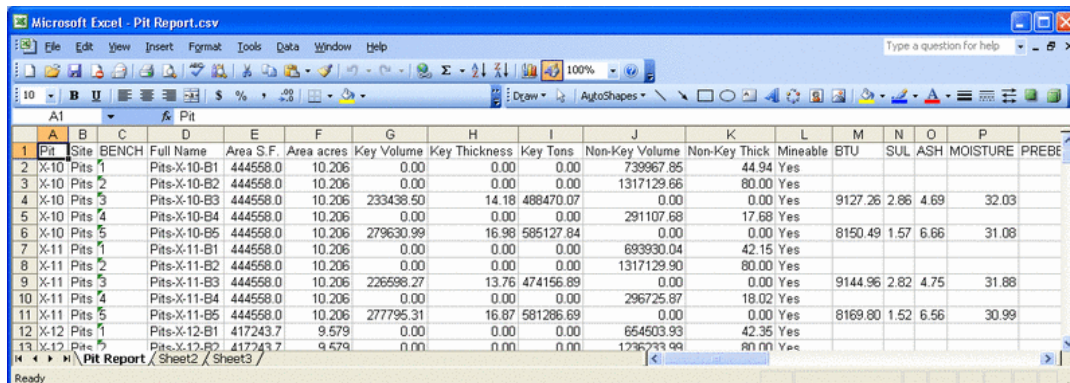
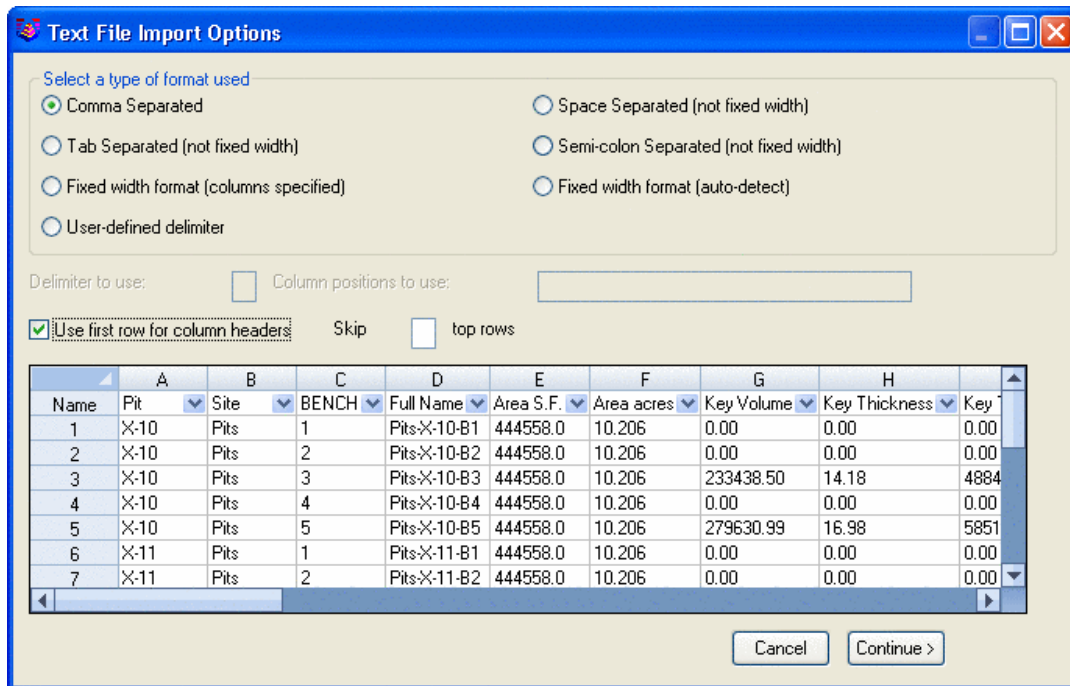
Keyboard Command: reset_tmgrids

Related Commands: Default Pit Attributes, Edit Pit

Import Pit Timing Data

This routine imports timing data values into pits from an external text file. The command can bring in the following pre-assigned parameters: Pit Name, Site Name, Key Volume, Key Tons, Non-Key Volume, Difficulty Factor, Start Date and Bench Number, Precedence and other attributes. Also, any user added quality attribute is inserted with the "Add Attribute" button.

The following example is saved as a comma-delimited file (*.csv), which the routine will bring directly in. It imports files with extensions of TXT, CSV, DAT and ASC. Be sure the Site and Pit names are spelled and identified exactly as they appear in the drawing. Shown below is a sample file in Excel. The program will automatically recognize the data that has been exported by the Pit Report command. Changes and additions can be made to the pit report file, and then reimported to attach the data to the pits.



The Formatter Window is very easy to use. Simply dropdown the arrow in the Name row to define what each column is. This will match the data up with the fields in the pits and import the data.

Use first row for column headers: When enabled, this option will use the first row of the .csv file to set the column headers

Skip _ rows: This option will skip the specified number of rows when importing. This is useful if you have header information in the .csv file

Excel style text quotation: When enabled, the program will look for quotation marks around text in the .csv file and remove them. For example, if a piece of text is found in the .csv file such as "Pit 1", it will only import Pit 1.

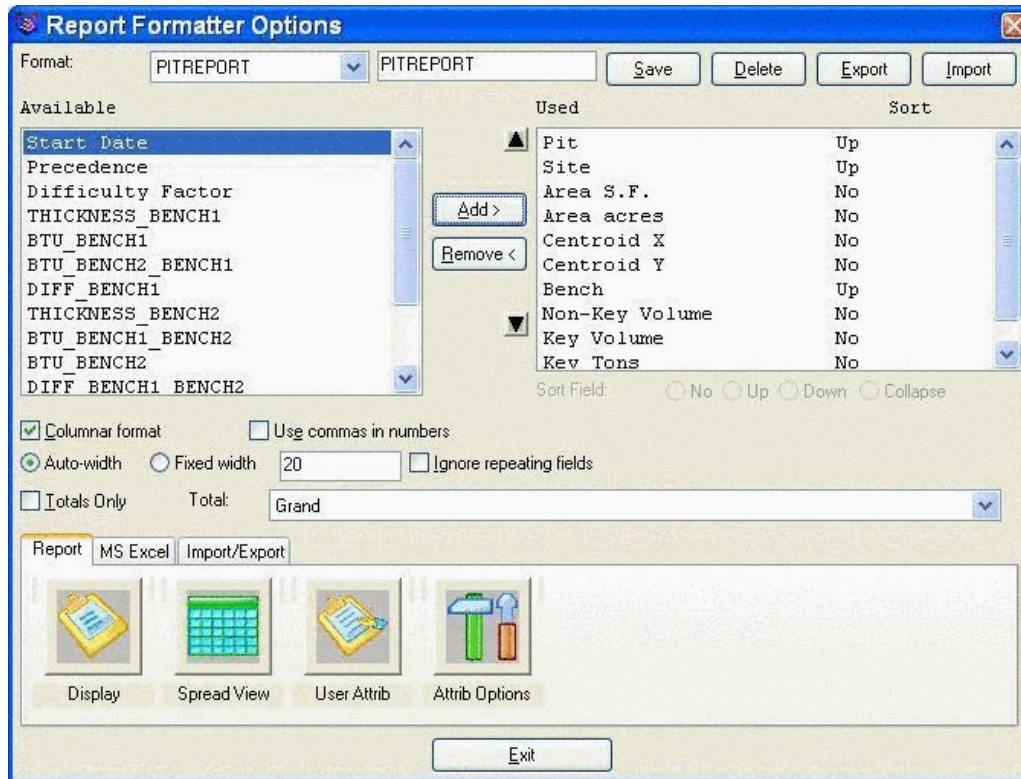
Pulldown Menu Location: Boundary, Pit Timing Quantities

Keyboard Command: tmimport

Prerequisite: A pit data file and pits on screen

Pit Quantities Report

This command will generate a customized report of all data contained in the pit polylines. It is the compliment of the Import Pit Timing Data command. This command uses the Report Formatter which allows you to customize the report fields layout, and to export the data to a spreadsheet or database. There must be data already assigned to the pit polylines before running this routine.



Edit : c:\scad2005\USER\scadrprt.tmp

File Edit Settings

Open Save Print Exit Find Screen

Pit Report

Pit	Site	Area S.F.	Area acres	Centroid X	Centroid Y	Bench	Non-Key Volume	Key Volume	Key Tons	
A-7	Pits 1142515.9	26.229	1458881.98	1974201.50	1	5228517.2	670692.1	737929.0		
A-7	Pits 1142515.9	26.229	1458881.98	1974201.50	2	845509.2	711408.1	777924.7		
A-8	Pits 1130404.7	25.951	1459095.34	1974251.97	1	5325794.2	627756.9	690689.6		
A-8	Pits 1130404.7	25.951	1459095.34	1974251.97	2	842406.0	702974.2	768702.3		
A-9	Pits 1118293.5	25.672	1459308.69	1974302.43	1	5357624.4	596238.7	656011.7		
A-9	Pits 1118293.5	25.672	1459308.69	1974302.43	2	840293.6	693744.7	758609.9		
A-10	Pits 1107149.5	25.417	1459520.65	1974354.85	1	5285103.3	571721.2	629036.2		
A-10	Pits 1107149.5	25.417	1459520.65	1974354.85	2	839970.7	684330.8	748315.7		
A-11	Pits 1102809.3	25.317	1459722.29	1974420.63	1	5362902.0	552629.6	608030.8		
A-11	Pits 1102809.3	25.317	1459722.29	1974420.63	2	845542.1	678528.3	741970.7		
A-12	Pits 1097158.8	25.187	1459917.67	1974494.16	1	5462169.7	533832.7	587349.4		
A-12	Pits 1097158.8	25.187	1459917.67	1974494.16	2	850210.9	671548.6	734338.4		
---- Grand Total -----							37086043.2	7695406.1	8438908.4	

Prompts

Select the site/pit perimeter polylines.

Select objects: *pick the pit polylines*

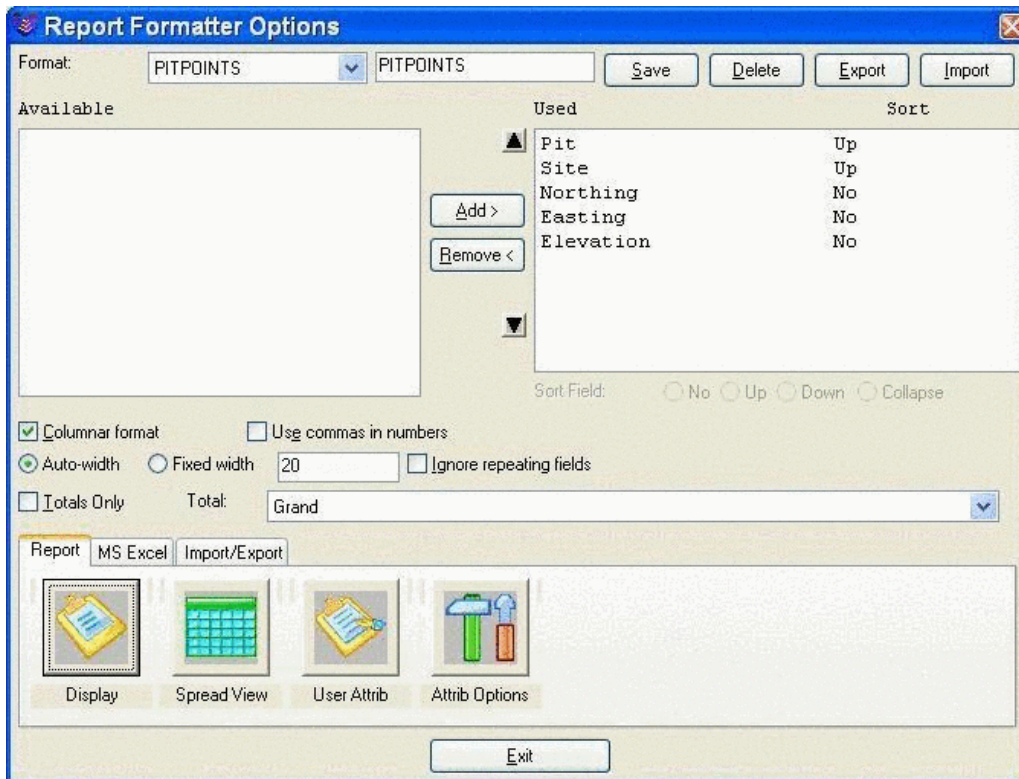
Report Formatter dialog. Choose fields to report then pick Display.

Pulldown Menu Location: Boundary > Pit Report

Keyboard Command: pitreport

Pit Points Report

This command will generate a customized report of the Site Name, Pit Name, Easting, Northing and Elevation of the vertices (corners) of each pit perimeter. This command uses the Report Formatter, so exporting to Excel or Access is seamless.



Pit	Site	Northing	Easting	Elevation
A-1	Pits	1976093.268	1456045.456	0.000
A-1	Pits	1975967.280	1455891.317	0.000
A-1	Pits	1975266.551	1456427.611	0.000
A-1	Pits	1974648.532	1456899.809	0.000
A-1	Pits	1973783.765	1457564.389	0.000
A-1	Pits	1972085.106	1458866.949	0.000
A-1	Pits	1971450.702	1459351.958	0.000
A-1	Pits	1971525.582	1459548.034	0.000
A-1	Pits	1976093.268	1456045.456	0.000
A-2	Pits	1976219.902	1456200.384	0.000
A-2	Pits	1976093.268	1456045.456	0.000
A-2	Pits	1971525.582	1459548.034	0.000
A-2	Pits	1971600.031	1459742.978	0.000
A-2	Pits	1976219.902	1456200.384	0.000
A-3	Pits	1976323.718	1456372.809	0.000
A-3	Pits	1976307.373	1456307.400	0.000
A-3	Pits	1976219.902	1456200.384	0.000
A-3	Pits	1971600.031	1459742.978	0.000
A-3	Pits	1971674.479	1459937.922	0.000
A-3	Pits	1976323.718	1456372.809	0.000
A-4	Pits	1976376.569	1456584.314	0.000
A-4	Pits	1976323.718	1456372.809	0.000

Prompts

Select the site/pit perimeter polylines.

Select objects: Specify opposite corner: 19 found

Select objects: Report Formatter dialog. Choose fields to report then pick Display.

Pulldown Menu Location: Boundary > Pit Report

Keyboard Command: pitreport2

Edit Pit

Edit Pit allows you to review and edit the quantities and attributes stored in each pit which are used for scheduling in the Surface Equipment Timing and Pit Scheduler commands.

Storing Data in Pits

Quantities and attributes may be stored in the pits using the Surface Mine Reserves, Assign Timing Grids, or Import Pit Timing Data commands (explained below). These values are stored in the pits as Extended Entity Data (EED), along with the pit name and the mining direction. In the pits themselves, some pit data (key tonnage, key thickness, nonkey tonnage, and nonkey thickness) may be stored as discrete values or as grid files. The primary difference between these methods is in how Surface Equipment Timing will use the information for scheduling. When data is stored in the pit as a discrete value, it is assumed that the pit is homogenous in regards to that value. When data is stored as a grid file, the value can be treated as variable across the pit. For example, if the OB Volume is represented with a discrete value, it is assumed that the overburden thickness is the same across the entire pit. When mining through the pit, one side of the pit will be mined just as quickly as the other side. If the OB Volume is represented with a grid, however, the true thickness of the overburden will be taken into account during scheduling. This will show how one side of the pit may be mined more quickly/slowly than another side of the pit due to a change in overburden thickness.

1. The Surface Mine Reserves command provides an option to Use Named Pit Polylines and to Store Results in Pits. These options will store all relevant quantities and attributes in the pit on a specific bench as discrete values. Surface Mine Reserves also has an option to Output Thickness Grids. When this option is used, the thickness grids will be stored in the pits in place of the respective discrete values. If the pit needs to be divided into multiple benches, the values for each bench may be calculated simultaneously with the use of the Use Auto-Run option.
2. The Assign Timing Grids command will prompt you for three grids to assign to each bench: a nonkey thickness grid, a key thickness grid, and a key tons grid. The key tons grid should represent tons/square foot or tonnes/square meter.
3. The Import Pit Timing Data command will import pit information from a .csv file.

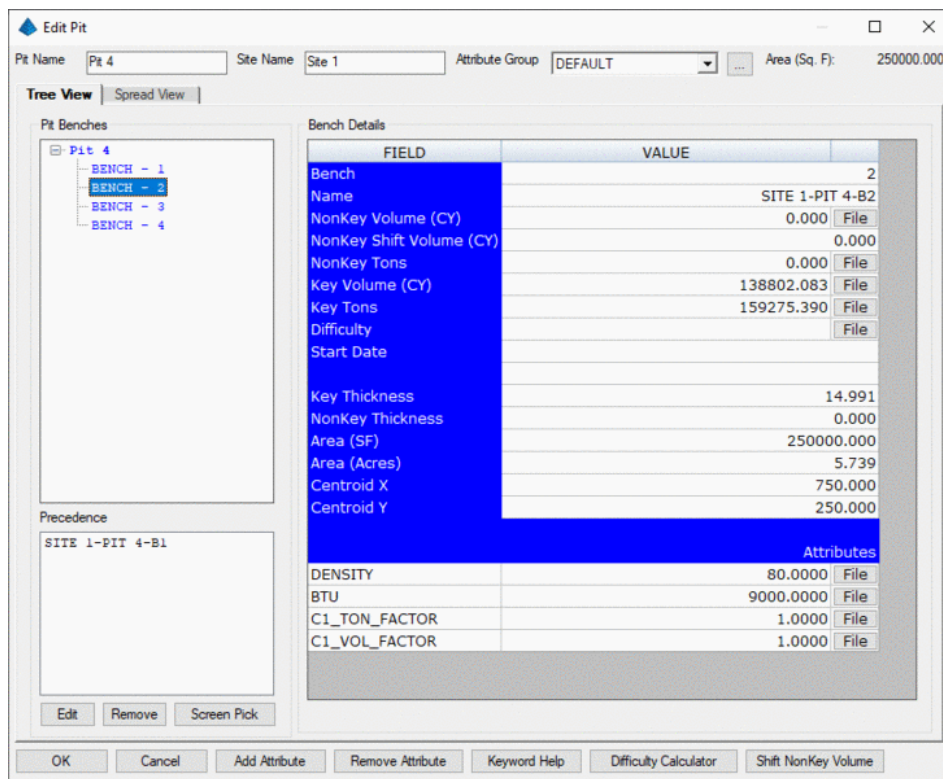
For more information regarding these three commands, see the section of the help manual corresponding to each command.

Edit Pit Dialog

When executing this command, you will be prompted to pick inside a pit polyline. The selected pit will be hatched with a yellow fill and the Edit Pit dialog will appear. If two pit polylines overlap in the location you select, one pit polyline will be highlighted and you will be prompted if the selection is correct. If the program does not detect any named, directional pits with stored quantities, the command will not execute.

Tree View:

The below dialog shows the tree view for a typical pit. In this view, all benches in the pit are listed in the Pit Benches tree view on the left side of the dialog. The information relating to the selected bench will be shown under Bench Details on the right side of the dialog. Most of the values listed under Bench Details may be edited by simply typing a value in the cell of interest or by clicking the File/Edit button to the right of the cell. Precedence rules for the selected bench are shown in the bottom left corner of the dialog.



Spread View:

The below dialog shows the spread view for a typical pit. This view is simply a different way of viewing the pit information shown in the tree view and may be accessed by clicking the Spread View tab near the top of the dialog. In this view, benches are displayed in a spreadsheet for easier side-by-side comparison.

The screenshot shows the 'Edit Pit' dialog box with the 'Spread View' tab selected. The dialog contains a spreadsheet with the following data:

	SITE 1-PIT 4-B1	SITE 1-PIT 4-B2	SITE 1-PIT 4-B3	SITE 1-PIT 4-B4
Bench	1	2	3	4
Name	SITE 1-PIT 4-B1	SITE 1-PIT 4-B2	SITE 1-PIT 4-B3	SITE 1-PIT 4-B4
NonKey Volume (CY)	92534.722 File	0.000 File	185069.444 File	0.000 File
NonKey Shift Volume (CY)	0.000	0.000	0.000	0.000
NonKey Tons	106183.593 File	0.000 File	212367.188 File	0.000 File
Key Volume (CY)	0.000 File	138802.083 File	0.000 File	231336.806 File
Key Tons	0.000 File	159275.390 File	0.000 File	265458.985 File
Difficulty	File	File	File	File
Precedence	▼ Edit SITE 1-PIT 4-B1	▼ Edit SITE 1-PIT 4-B2	▼ Edit SITE 1-PIT 4-B3	▼ Edit
Start Date				
Key Thickness	0.000	14.991	0.000	24.984
NonKey Thickness	9.994	0.000	19.988	0.000
Area (SF)	250000.000	250000.000	250000.000	250000.000
Area (Acres)	5.739	5.739	5.739	5.739
Centroid X	750.000	750.000	750.000	750.000
Centroid Y	250.000	250.000	250.000	250.000
Attributes				
DENSITY_NK	80.0000 File	File	80.0000 File	File
BTU_NK	File	File	File	File
OB_TON_FACTOR_NK	1.0000 File	File	File	File
OB_VOL_FACTOR_NK	1.0000 File	File	File	File
DENSITY	File	80.0000 File	File	80.0000 File
BTU	File	9000.0000 File	File	7000.0000 File
C1_TON_FACTOR	File	1.0000 File	File	File
C1_VOL_FACTOR	File	1.0000 File	File	File
IB_TON_FACTOR_NK	File	File	1.0000 File	File
IB_VOL_FACTOR_NK	File	File	1.0000 File	File
C2_TON_FACTOR	File	File	File	1.0000 File
C2_VOL_FACTOR	File	File	File	1.0000 File

Pit Name and Site Name: These fields set the name of the pit.

Attribute Group: This dropdown menu sets the attribute group for the pit. Attribute groups may be used to assign attribute values/grids to many pits at one time. New Attribute Groups may be defined in the Timing Project Manager.

Bench: This field displays the bench number.

Name: This field displays the full bench name. This is a concatenation of the Pit Name, the Bench Name, and the Bench Number separated by hyphens.

NonKey Volume: This value sets the volume of nonkey material (overburden) in the bench. This value can also be represented by a thickness grid of the nonkey material.

NonKey Shift Volume: This value displays the volume of nonkey material that has been shifted into the current bench from another bench. Nonkey material may be shifted between benches with the use of the Shift OB Volume button at the bottom of the dialog.

NonKey Tons: This value sets the tonnage of nonkey material in the bench. This value can also be represented by a grid of tons/square foot or tonnes/square meter.

Key Volume: This value sets the volume of key material in the bench. This value can also be represented by a thickness grid of the key material.

Key Tons: This value sets the tonnage of key material in the bench. This value can also be represented by a grid of tons/square foot or tonnes/square meter.

Difficulty: This value sets the difficulty factor associated with the bench. The difficulty factor temporarily modifies the equipment mining rate set in the Timing Project Manager. The adjusted mining rate is the quotient of the original mining rate and the difficulty factor. A difficulty factor of 1.0, or if the difficulty factor is not specified, will maintain the original mining rate. A difficulty factor of 2.0 will double the amount of time required to mine the bench (original mining rate is divided by 2.0). A difficulty factor of 0.5 will half the amount of time required to mine the bench (original mining rate is divided by 0.5). To quickly calculate the difficulty factor, you may want to use the Difficulty Calculator at the bottom of the dialog. The difficulty factor may also be represented by a grid file.

Here it is important to note the difficulty value set in this dialog may be overridden according to a 4-level hierarchy.

1. The manual entry of the value or grid file set here is at the bottom of the hierarchy.
2. The second level is specifying the difficulty within an Attribute Group in the Timing Project Manager. There, an attribute named DIFF_BENCH* (where * is a bench number) may also be represented by a discrete value or a grid file. When this DIFF_BENCH* attribute is added to the attribute group, it will actually create a new attribute in the pit itself called DIFF.
3. The DIFF attribute in the pit is the third level in the hierarchy. This allows you to use a generic difficulty for the entire attribute group, but still manually set the difficulty for individual benches.
4. The fourth and highest level in the hierarchy is layerized text in the drawing. The text should be the difficulty factor (no prefixes or suffixes) and should be placed on a layer named DIFF_BENCH* (where * is a bench number). For example, if the text "1.2" on the DIFF_BENCH1 layer is placed inside a pit polyline, the difficulty factor for the first bench in that pit will be set to 1.2. This difficulty factor will override all others.

Start Date: This field sets the earliest start date for the pit. This can be used to prevent mining an area due to waiting for permit approval, waiting to finalize the purchase of new land, etc. Most of the time this field will be left blank, however.

Key Thickness: This value displays the average key thickness of the bench.

NonKey Thickness: This value displays the average nonkey thickness of the bench.

Area: These values display the area of the bench.

Centroid X and Centroid Y: These values display the coordinates of the geometric center of the pit.

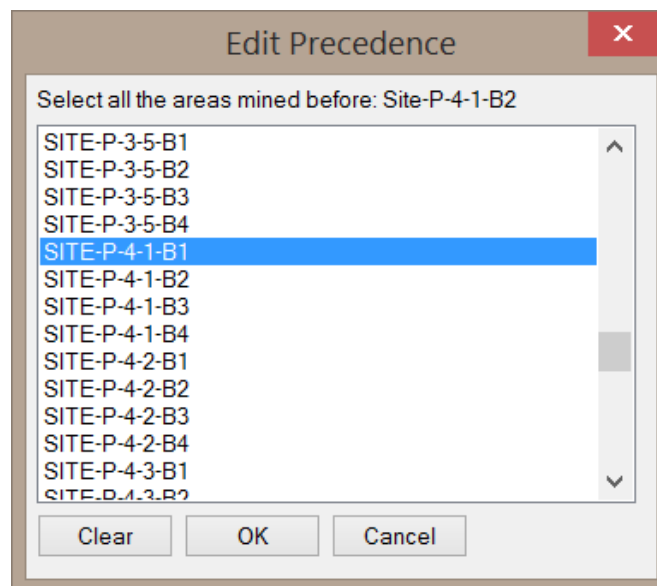
Other Attributes: The bottom of the Bench Details list leaves spaces for a variety of other attributes. Most of these attributes are used to define qualities of the bench. For example, if a bench contains an attribute for Moisture, this attribute will apply to the entire bench. If two strata are added to the same bench, this Moisture attribute will relate to both strata. If you are storing values in the pits using Surface Mine Reserves, the weight-averaging of the attribute will be applied automatically based on the volume and the density of the material. Here it is important to note that the weight averaging of attributes will not account for null values. For example, if Strata A has a Moisture attribute of 0.3, but Strata B does not have a Moisture attribute defined, then the combined Moisture content for the two stratum will be 0.3. To properly weight attributes, it is necessary to define an attribute value for each strata.

Some attributes may be shown with a _NK suffix. This suffix will be applied to differentiate between key attributes and nonkey attributes. For example, if Strata A is key and has a Sulfur value of 0.1 and Strata B is nonkey and has a Sulfur value of 0.15, these two Sulfur values will be handled separately. The Sulfur value for Strata A will be stored as "Sulfur" with a value of 0.1 whereas the Sulfur value for Strata B will be stored as "Sulfur_NK" with a value of 0.15.

Some attributes may be in the form of XXXX_VOL_FACTOR or XXXX_TON_FACTOR where XXXX is a strata name. This is used to define ratios of material when more than one key strata or more than one nonkey strata are added to the same bench. For example, if strata A and B are both key strata, an attribute for B_TON_FACTOR will be shown in the list. If this value is 0.4, then 40% of the key tonnage in the bench comes from strata B. If the attribute A_VOL_FACTOR is 0.56, then 56% of the key volume comes from strata A. For nonkey strata, a suffix of "_NK" will be applied to these attributes.

Precedence: This section of the dialog controls precedence rules for scheduling. The precedence list is simply a list of other benches that need to be mined before the current bench can be mined. Basic top-down precedence rules will be automatically entered to prevent mining a lower bench before the upper bench has been mined.

Edit: This button will open a list of all other benches found in the drawing, as shown below. Selecting benches of interest and clicking OK will add the benches to the precedence list. Multiple benches may be selected by clicking and dragging in the list or by holding the CTRL key while clicking.



Remove: This button will remove the highlighted bench from the precedence list.

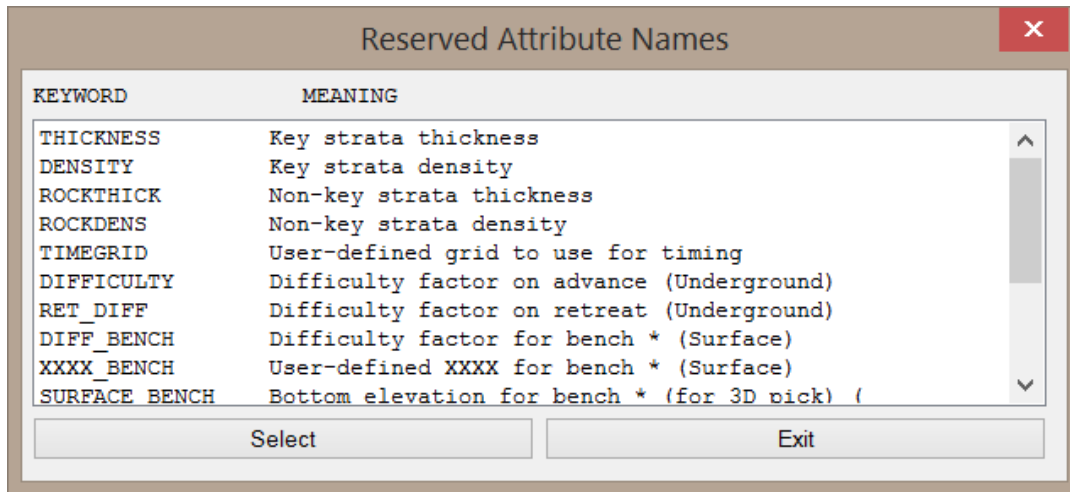
Screen Pick: This button will allow you to add benches to the precedence list by picking them in the drawing. As soon as you click inside a pit polyline, the uppermost bench for that pit will be added to the precedence list. Clicking in the pit polyline multiple times will add lower benches.

Add Attribute: This button will add a new row to the Attributes list.

Remove Attribute: This button will remove the currently selected attribute from the Attribute list.

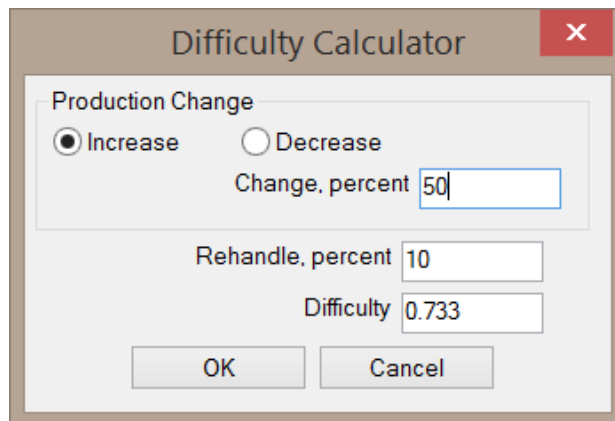
Keyword Help: This button will open the list of keyword attributes, as shown below. These keyword attributes may also be added to an Attribute Group in the Timing Project Manager. When an attribute with the _BENCH* suffix (where * is a bench number) is added to the Timing Project Manager, it will also appear in the Edit Pit dialog.

Manually editing the value in the Edit Pit dialog will override the value set in the Attribute Group. Attributes that are added to the Attribute Group that do not have the `_BENCH*` suffix, however, will not appear in the Edit Pit dialog. Note that some of these keywords are only intended for use with the Underground Mining Module. For more information on the specifics of each keyword, see the section of the help manual corresponding to the Timing Project Manager.

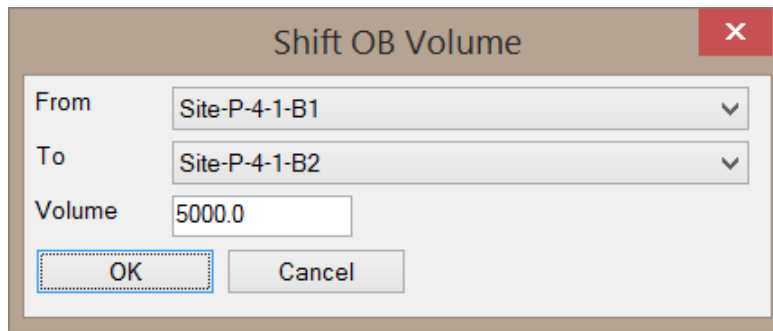


Double clicking one of the keywords in the above dialog will add it to the Attributes list on a new row.

Difficulty Calculator: This button will open the difficulty calculator as shown below. This dialog will calculate the difficulty factor based on an increase/decrease in the mining rate and a percentage of rehandle. For example, if the mining rate is expected to increase by 50% and 10% of the material will need to be rehandled, the calculated difficulty factor will be 0.733.



Shift OB Volume: This button will open the Shift OB Volume dialog as shown below. This will move nonkey volume from one bench to another. This dialog will not allow you to move more material than what exists in the source bench.



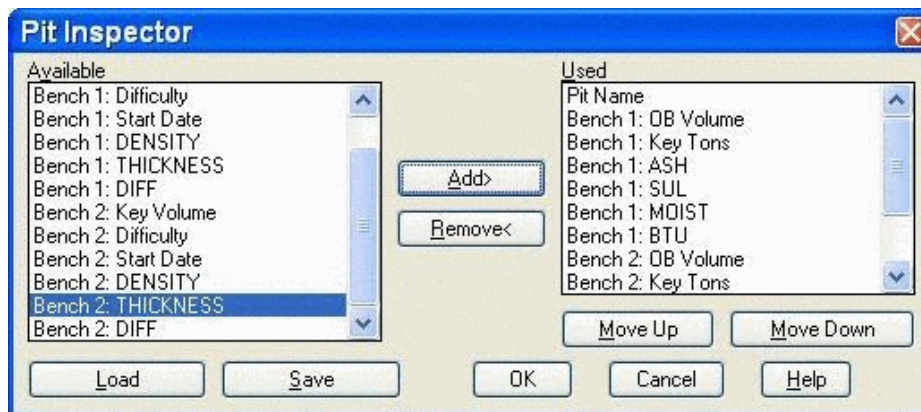
Pulldown Menu Location: Surface Mining Module > Boundary

Keyboard Command: editpit

Related Commands: Surface Mine Reserves, Timing Project Manager, Surface Equipment Timing

Pit Inspector

This command follows the "inspector" theme found throughout Carlson. Any variable or attribute will be displayed within the inspector window as the cursor is moved from pit to pit. When each pits information is being displayed, the pit perimeter is dashed to highlight the pit. If a grid is stored in the pit, then just the grid name path is displayed or labeled, not the value. Picking with the mouse inside a pit will bring up the Edit Pit dialog box where changes may be made if necessary. Options will activate the Pit Inspector window to add or edit the items. Label will allow for labeling the Used items on the screen. The text alignment and height is prompted for at the command line.





Prompts

Options/Label/<Pick To Edit>: L (for label)

Pick starting point for label:

Pick label alignment point:

Text height <50.00>:

Options/Label/<Pick To Edit>:O (brings up the settings window again)

Drop-Down Menu Location: Boundary

Keyboard Command: pitinspect

Files:

Assign Property Names

In order to evaluate the quantity and/or quality of ore mined on the particular property, Carlson employs the use of closed polylines representing the property boundary. This function is used for assignment and modification of the owner and property ID names to these polylines. The following commands support property lines:

Quantities by Average
Quantities by Grid
Surface Equipment Timing
Surface Production Timing
Underground Timing
Surface Mine Reserves
Reserve Classification

This command simply prompts to select the polyline and type in the owner and ID. Property polylines supersede the Boundary Enclosure command for the purposes of calculation, since the use of property polylines does not require actual subdivision of the mine plan into properties. Instead the subdivision is performed internally "on-the-fly" by the cutting of a given polyline (such as whole mine property or monthly production pit polyline) against an underlying set of property lines.

In its current implementation the use of property lines has the following limitations: property lines should be closed polylines and make sure that they do not overlap other property lines. If a portion of a pit or panel is outside of all property lines, then in the report, the owner will be reported as UNKNOWN.

Prompts

Select property polyline:

Property Owner Name<>: *Federal*

Property ID<>: *#J1267*

Pulldown Menu Location: Boundary

Keyboard Command: property_line

Remove Property Data

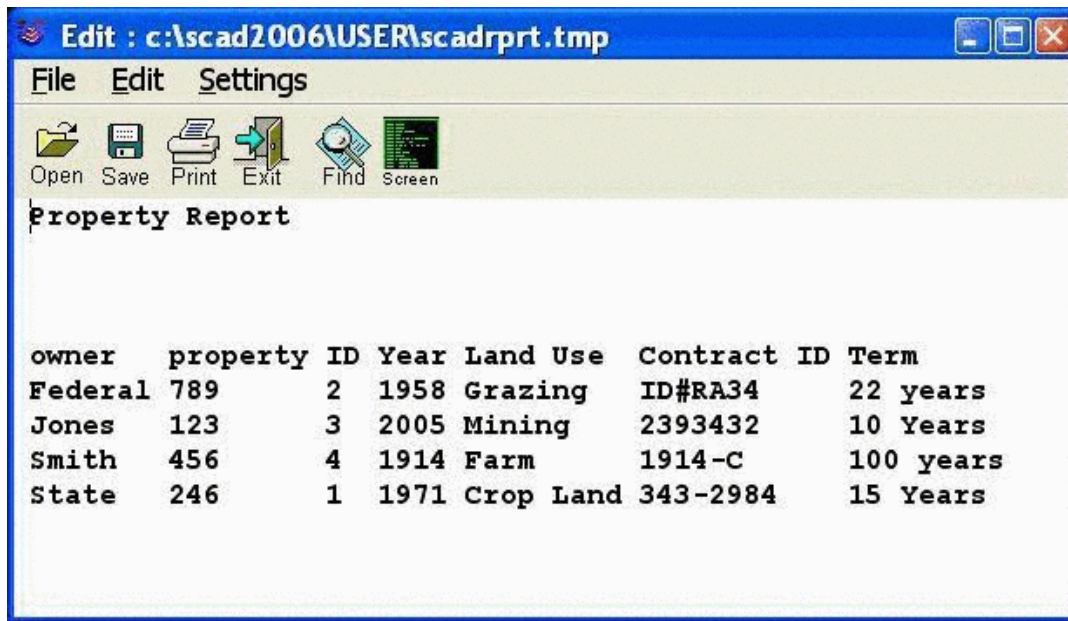
When property information attached to the polyline is no longer needed, it may be removed by using this command. Multiple polylines may be processed at once.

Pulldown Menu Location: Boundary

Keyboard Command: clearprop

Property Data Report

This command uses the report formatter to create a report showing all property data associated with the selected property lines. They must be linked to the MPD file where all of the property data is stored.

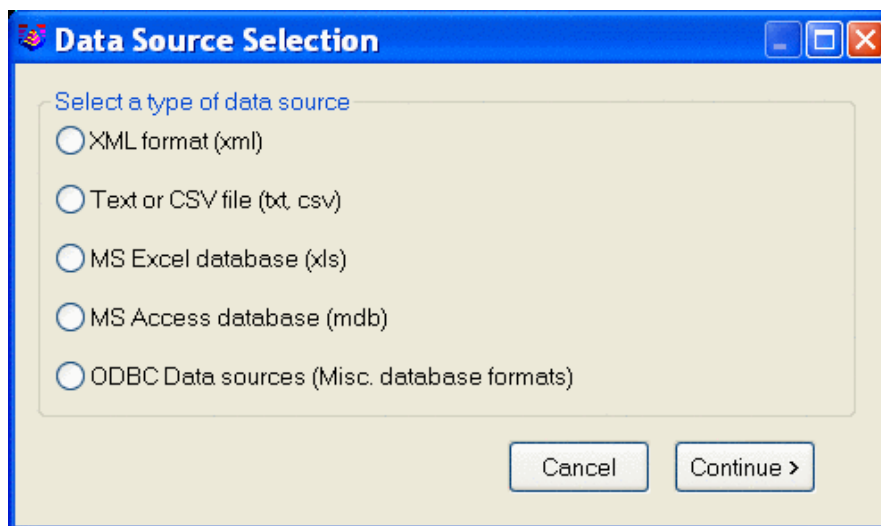


Pulldown Menu Location: Boundary, Property Line Tools

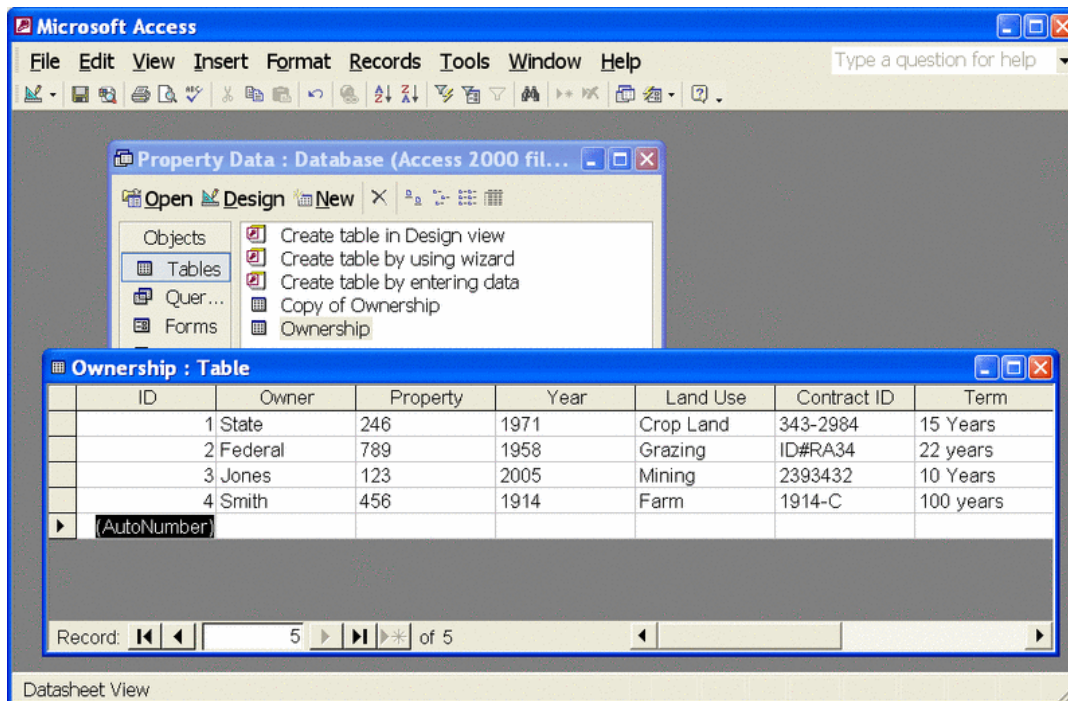
Keyboard Command: reportprops

Setup Property Link

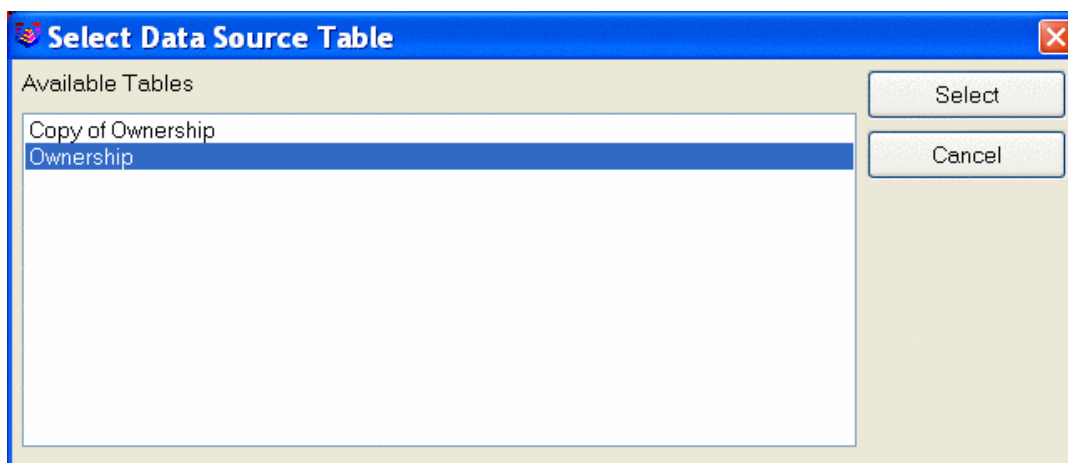
This command creates the link between the property lines in the drawing and the database. The data can come from an XML file, a TXT or CSV file, an XLS file, an Access MDB file, or miscellaneous ODBC data sources.



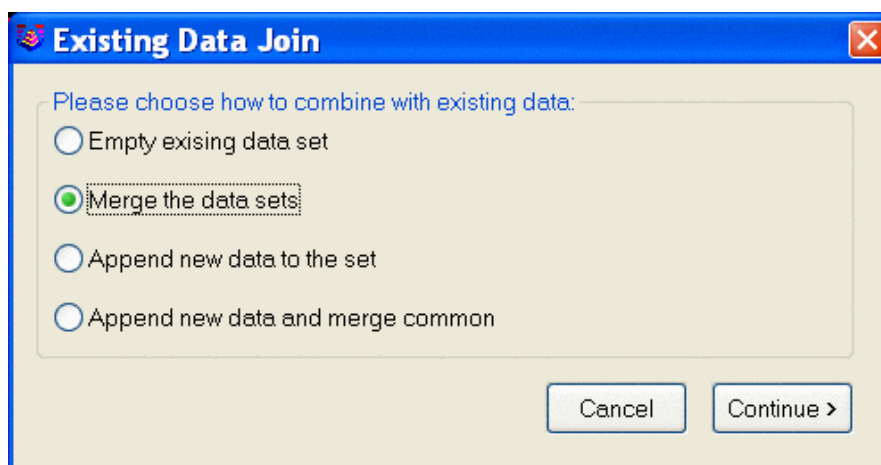
After choosing which format it is, select the Continue button.



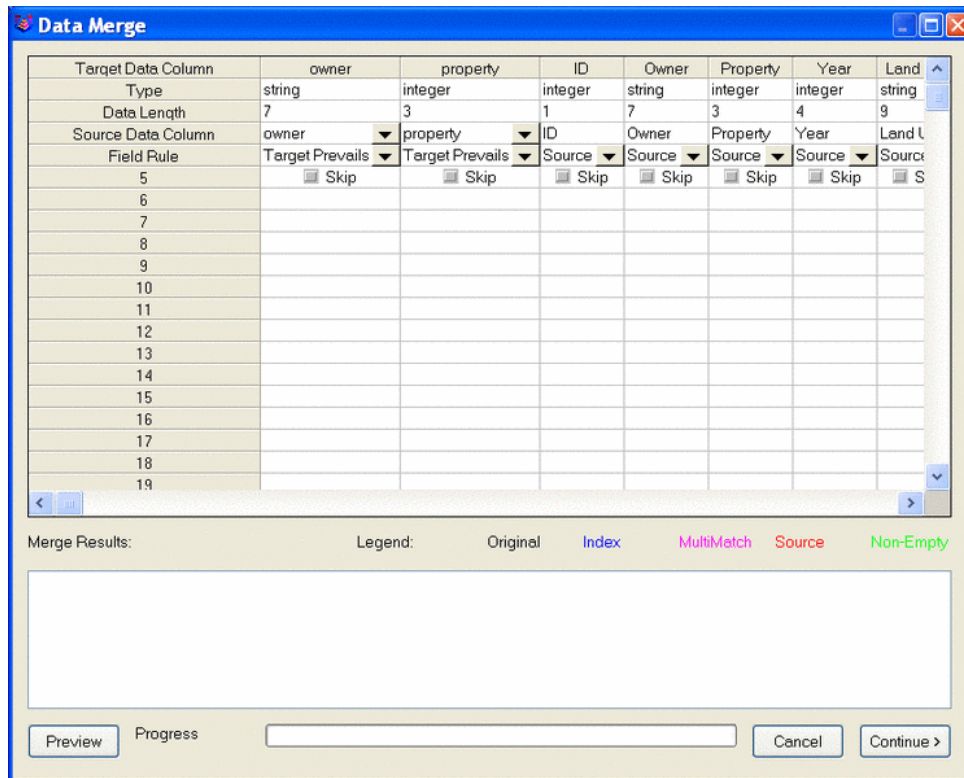
This Access database shows some data entered in for four different properties.



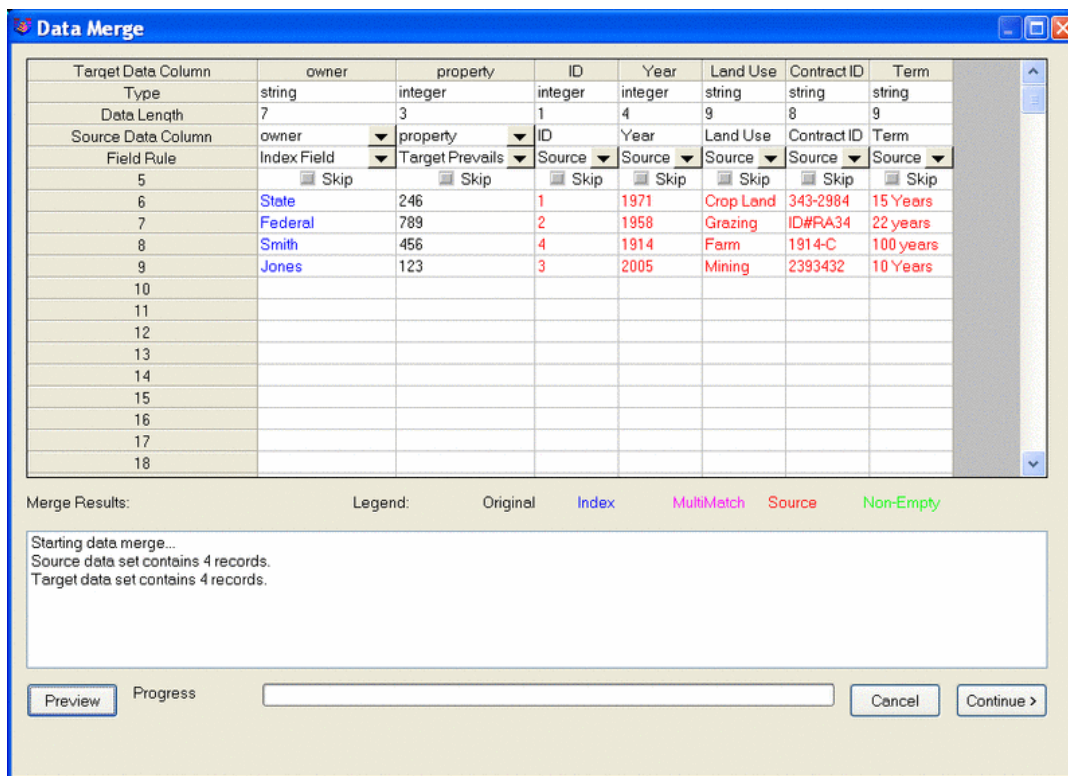
After choosing the source file, this selection window appears if it is from an Access database. Choose the table the data is stored in.



There are then four choices on how to join the data. The first is to empty the existing data set which would delete the property and ownership line data. The second is to merge the data, using both data sets. The third option is to append all to the end of the data, and finally, the last option is to append any new data, and merge the common data. The next Data Merge table displays all of the available fields. Notice in this example, there are two Owners and two Properties. In this case, the Merge option is the way to go.



Choosing Index Field on the first Owner gives the following result. The data is now merged and ready to be used for reporting and reserve runs. Any time these property lines are used in reserves, all of this data is available for reporting. Choose Continue will process the data and return to the command line.



Pulldown Menu Location: Boundary, Property Line Tools

Keyboard Command: setuppropdblclick

Property Names By Text

This command converts a large number of closed polylines into property lines by using text label placed inside of the polyline. You are prompted to select closed polylines to become property lines and then the program tries to locate a text label inside each closed polyline that has the same layer as that closed polyline. The text is used as the owner name and the property ID is set to blank.

When you have text labels to start with but no closed polylines, use the Boundary Polyline command under Draw menu to create closed polylines from appropriate linework by using the text insertion point.

Pulldown Menu Location: Boundary, Property Line Tools

Keyboard Command: grabprop

Identify Property Lines

This command is one of several property line tools available to identify and trouble shoot property lines. To identify a property line, simply click inside the polyline and the name of the owner and property ID will be displayed in the command text window.

Prompts

Pick on or inside of property polyline:

Owner: State

Property: 459a

Pick on or inside of property polyline:

Owner: Jones

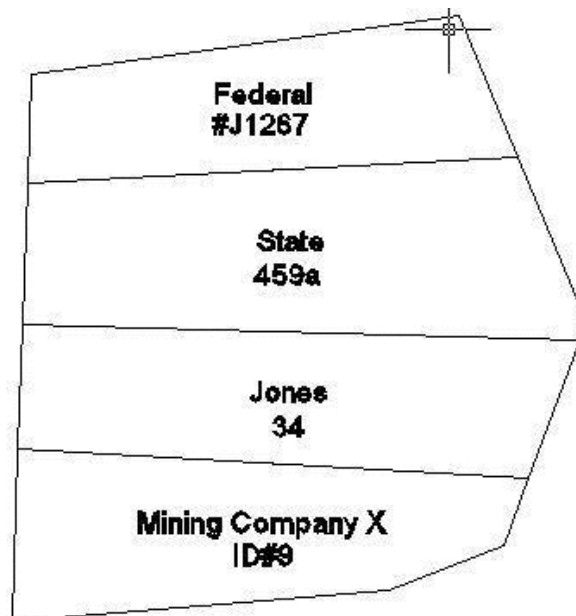
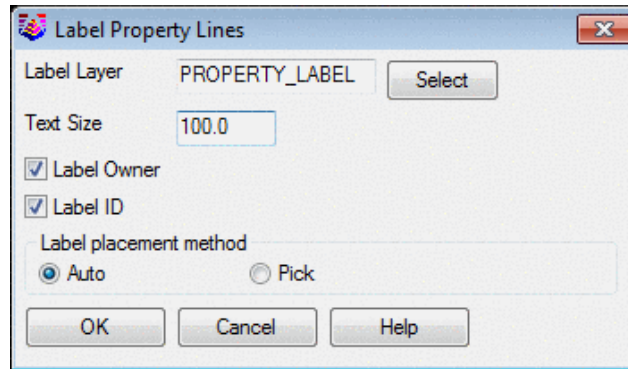
Property: 34

Pulldown Menu Location: Boundary, Property Line Tools

Keyboard Command: idprop

Label Property Lines

This command labels property lines in the drawing. The dialog contains option set the label layer and text size in feet or meters. There are options to label the owner, ID or both. When the Automatic mode is selected, the label is placed at the center of the property line. If the placement method is Pick, then each property line text location will need to be prompted for.



Prompts

Select all property polylines to label:

Select objects:

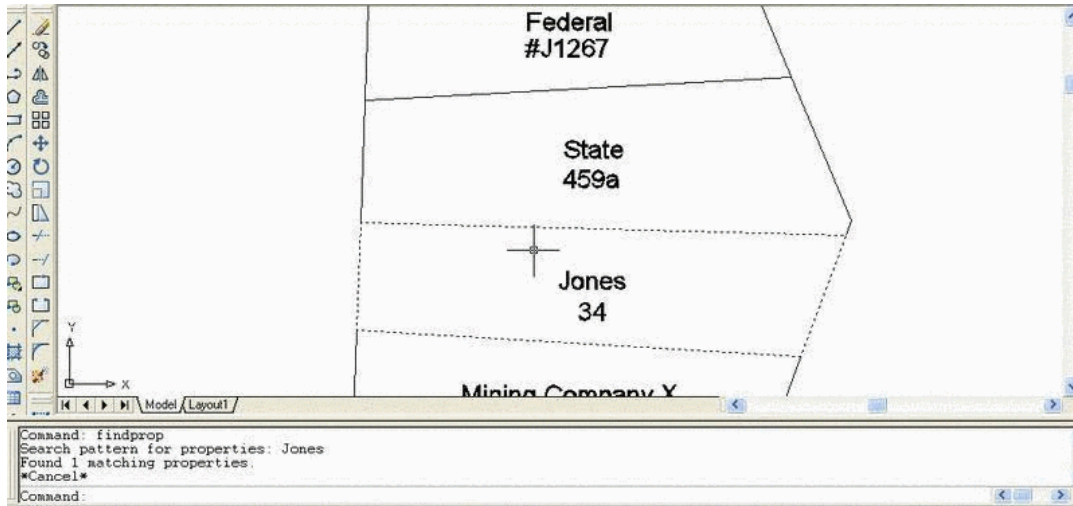
Dialog Box.

Pulldown Menu Location: Boundary in Surface Mining and Underground Mining Modules

Keyboard Command: labelprop

Find Property Lines

This command locates property lines by matching the user provided search pattern against owner name and property ID of all property lines in the drawing. All matching property lines will be highlighted. The wildcard character (*) can be used in the search pattern.

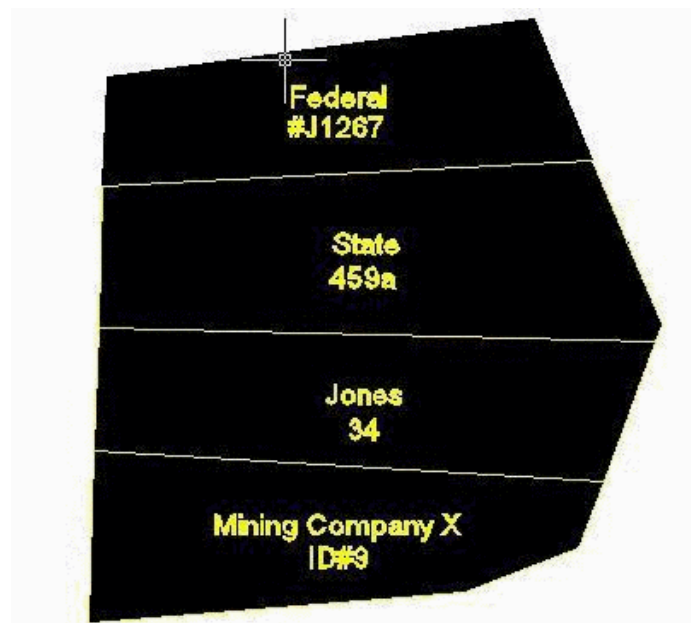


Pulldown Menu Location: Boundary, Property Line tools

Keyboard Command: findprop

Show All Property Lines

This function displays all property lines in the drawing either by highlighting them or by using solid fill. The latter option works for detection of the unassigned properties totally surrounded by the other properties. It is also convenient for location of tiny gaps between properties shown in the timing report as small areas with an UNKNOWN owner.



Prompts

Solid-fill the properties [Yes/<No>]? Y

Pulldown Menu Location: Boundary, Property Line Tools

Keyboard Command: showprop

Check Property Polyline Overlaps

This command checks whether property polylines overlap each other. Properties should never overlap since this would lead to extra volumes with the quantities in the overlap area getting reported twice. The program reports any overlaps at the command line.

Prompts

Select pit polylines to check.

Select objects: *pick the polylines*

Found 0 overlaps between 56 properties

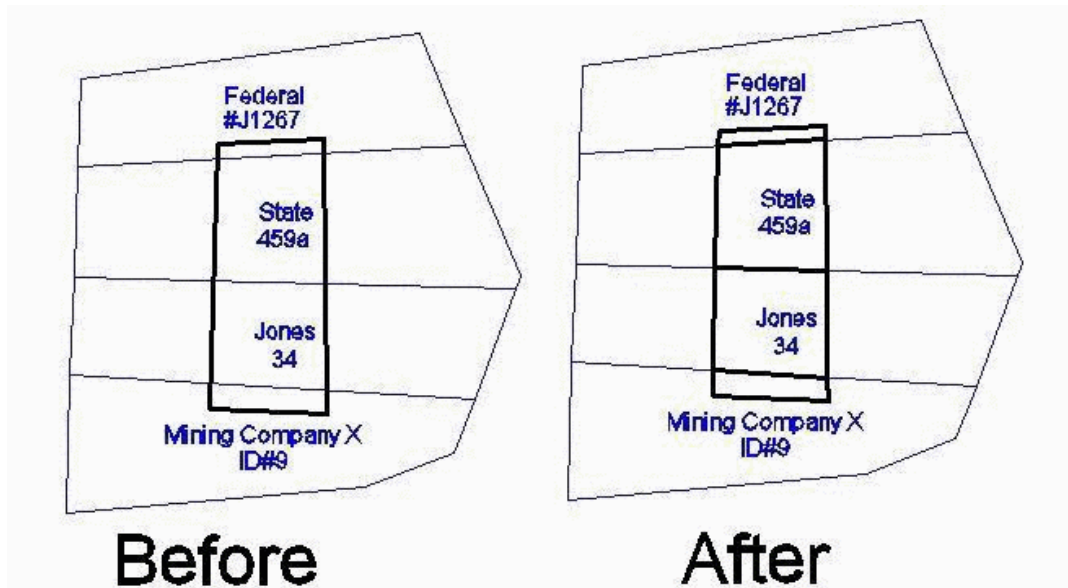
Pulldown Menu Location: Boundary > Property Line Tools

Keyboard Command: propcheck

Prerequisite: property polylines

Break Polyline by Property

This command breaks a closed polyline into separate closed perimeters where they cross property lines. This is useful when smaller, individual perimeters are desired for pits or panels.



Prompts

Select polyline to divide by owners:

Pulldown Menu Location: Boundary

Keyboard Command: ownerdivide

Extract Centroid Data

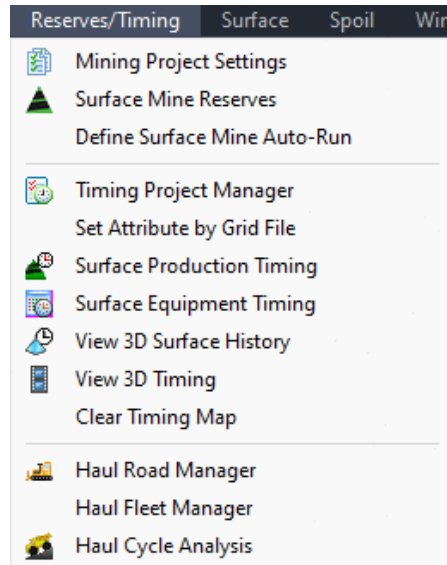
This command is used to convert property information imported from ArcInfo using shape files into property lines. It searches for polyline centroids, extracts the necessary information stored there and attaches it to the property line. This will only work in AutoCAD MAP.

Pulldown Menu Location: Boundary, Property Line Tools

Keyboard Command: import_shape

Reserves Timing Menu

The Reserves/Timing menu has commands for calculating mining quantities and surface timing. The Mining Project Manager and Surface Mine Reserves commands are described in the Geology section of the manual.



Mining Project Settings

This command sets the current file names used for several Carlson definition files. For example, this command assigns the Drillhole Configuration file (.ch file) to be used for drillhole placement in other routines such as Place Drillhole and Drillhole Data Sheet. The Mining Project Manager lets you set up different groups of definitions. For example, if you work with two mines, then you can have two sets of strata definitions (strata1.sdf and strata2.sdf), two sets of attribute definitions (attrib1.atr and attrib2.atr), etc. When switching between the projects, you can load the .mpj file to instantly update all of your relevant settings.

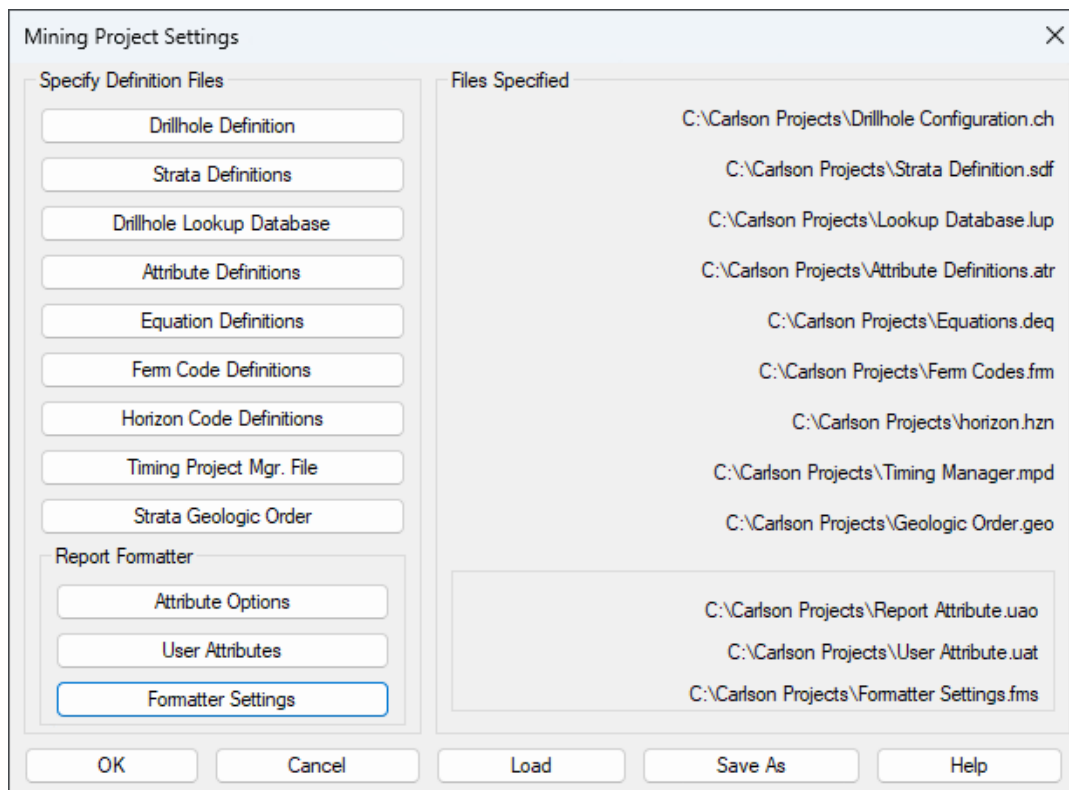
Whenever a command is executed that creates one of the below files, it will be automatically added to the Mining Project Manager. However, you can also specify the file names directly from this command.

All the file names in Mining Project Manager are saved with the drawing in the drawing .INI file. You can also save this set of file names to a mining project file (.mpj) by picking the Save button. The Load button will then recall the set of file names from the selected .MPJ file.

Following is a list of commands that will create the files shown in the Mining Project Manager. The extensions of the files are shown in the dialog window below.

1. Drillhole Definition: Define Drillhole
2. Strata Definitions: Define Strata/Bed

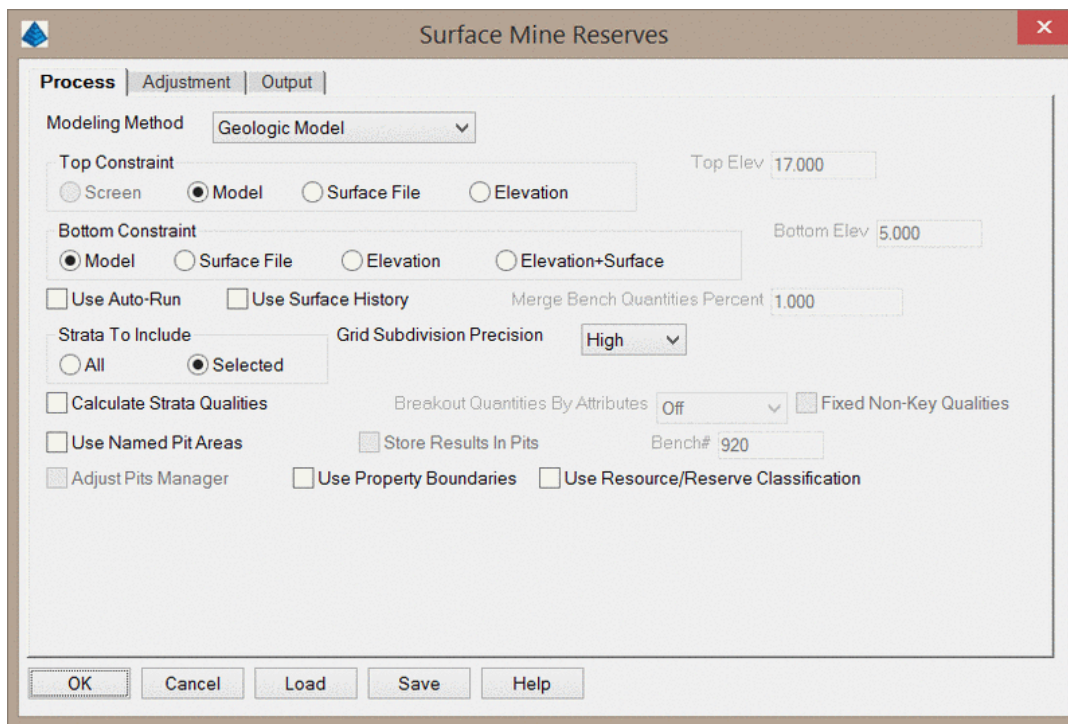
3. Drillhole Lookup Database: Define Lookup Database
4. Attribute Definitions: Define Attributes
5. Equation Definitions: Define Equations
6. Ferm Code Definitions: Define Ferm Codes
7. Horizon Code Definitions: Define Horizon Codes
8. Timing Project Mgr. File: Underground Timing, Surface Equipment Timing, Solids Timing
9. Strata Geologic Order: Define Geologic Order
10. Attribute Options: In the Report Formatter, Attribute Options
11. User Attributes: In the Report Formatter, User Attributes.
12. Formatter Settings: In the Report Formatter, Export



Pulldown Menu Location: Drillhole
Keyboard Command: mproject

Surface Mine Reserves

This command calculates quantities and qualities directly from drillholes or from predefined Geologic/Mining Models. Additional outputs include stripping ratio contours, composite thickness grids, and more. The dialog is divided into three tabs - Process, Adjustment, and Output - as shown below. It is important to note that if an option is grayed out (not editable) it will not affect the calculations. Some options cannot be used together, and so the unnecessary options will be grayed out to avoid confusion.



Modeling Method: This dropdown menu sets the modeling method to be used for calculations.

The first seven options (Triangulation, Inverse Distance, Kriging, Polynomial, Linear Least Squares, ABOS Method, and Nearest Neighbor) will calculate the reserves from drillholes, strata polylines, and pit/channel samples using the selected modeling method. This option will create grids for calculations, but the grids will not be saved. For more information, see the section of the help manual corresponding to the Make Strata Grids command.

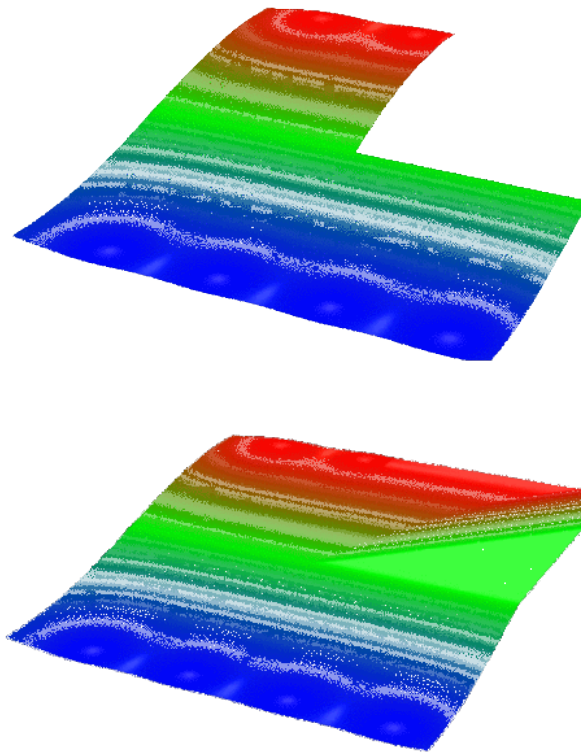
The Block Model option will also calculate the reserves from drillholes, strata polylines, and pit/channel samples by first creating a block model. This block model will not be saved.

The Geologic Model and Mining Model options will calculate the reserves from pre-calculated grid files stored in a .pre or .mmd file. It is highly recommended that you use either a Geologic Model or Mining Model for calculating reserves, as the other options will need to generate grids on-the-fly every time the command is run. Using a Geologic/Mining Model will greatly increase calculation time and reproducibility of the results.

The only difference between a Geologic Model and a Mining Model is the file extension (however, a Mining Model must be based on elevation grids whereas a Geologic Model may be based on elevations or thickness grids). The distinction is made purely for organizational purposes in that the Geologic Model can be used to represent the true geology while the Mining Model can be used to represent the strata as they will be mined. For example, some strata layers may need to be composited in the Mining Model. For more information on creating Geologic/Mining Models, see the sections of the help manual corresponding to the Define Geologic Model, Define Mining Model, and Geologic to Mining Model commands.

Here it is important to note that this command will temporarily extrapolate grid files in the Geologic/Mining Model. This replaces all null values with the nearest known data point. These changes are not saved to the .grd file itself however. When grids are extrapolated, you will receive a notification of which grids were temporarily modified for the calculation. This can increase the amount of time required to complete the calculation, and for this reason, it is recommended that you manually extrapolate grid files using either Grid File Utilities, or the Extrapolate function in Define Geologic Model. When the program automatically extrapolates surface, it will hold the values at the edge of the grid and extrapolate horizontally. An example of a surface file with null values, and the resulting grid that

would be temporarily created for this calculation is shown below.



Top Constraint: This option sets the upper limit for the reserve calculation. All calculations will be based on a layering of strata grids. This upper limit will omit any strata information that may exist above it. Four options are available for use:

The Screen option will prompt you for 3D entities (usually elevation contours) to create a grid file. This grid file will not be saved, however.

The Model option will use the surface file specified in the Geologic/Mining Model.

The Surface File option will prompt you for an existing grid or TIN file.

The Elevation option will use a flat elevation as set in the **Top Elev** field.

Bottom Constraint: This option sets the lower limit for the reserve calculation. This lower limit will omit any strata information that may exist below it. Four options are available for use:

The Strata Model option will use the bottom-most elevation grid file specified in the Geologic/Mining Model.

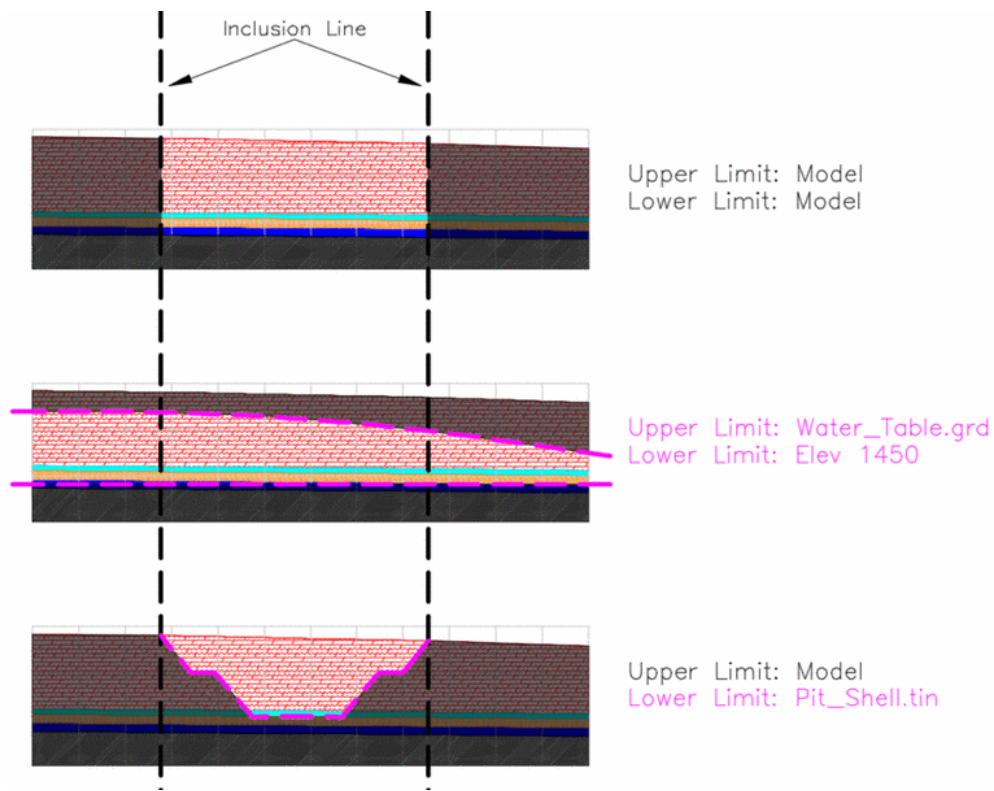
The Surface File option will prompt you for an existing grid or TIN file. This is most commonly used when calculating the reserves within a pit shell.

The Elevation option will use a flat elevation as set in the **Bottom Elev** field.

The Elevation+Surface option will allow you to define both an elevation and a surface file to limit the calculation. For example, you can constraint the calculation with a pit shell surface, but also with a bench elevation.

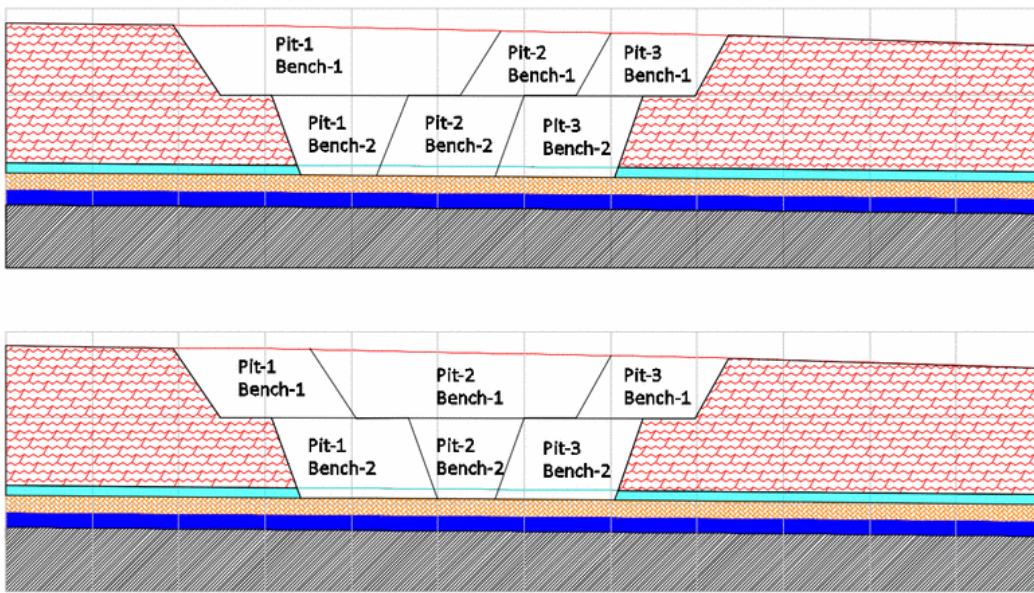
Top Elev and Bottom Elev: These fields will set the elevation limits for the reserve calculation.

Whenever selecting the upper and lower limits of the reserve calculation, it is important to visualize the options you are selecting. The below illustrations show some of the ways that the upper and lower limits can be used to constrain the calculation.



Use Auto-Run: This option will allow you to divide the reserves into benches according to a Surface Mine Reserves Autorun file (.sma file). Dividing reserves into separate benches is useful when using the Store Results in Pits option as it allows for more selective mining with the Surface Equipment Timing command. If this option is not used, results can still be stored in pits, but all data will be stored on a single bench. For more information, see the section of the help manual corresponding to the Define Surface Mine Auto-Run command.

Use Surface History: This option will calculate reserves according to a Surface History file (.gsq file). This file is essentially a list of grid or TIN files and can be created manually or with the Design Bench Pit command. When this option is used, reserves will be divided according to the volumes between the surface files (i.e. the first volume is calculated between the first and second surface file, the second volume is calculated between the second and third surface file, etc.). The below images show an example of how the reserves will be divided into three pits and two benches using a surface history file. The ordering of the surfaces in the Surface History file will have a significant impact on the calculations as it determines the cut angles to be used. For example, the first image shown below cuts out Pit 1 first, whereas the second image cuts out Pit 2 first. For more information on the Surface History file, see the section of the help manual corresponding to the View 3D Surface History command.



Merge Bench Quantities Percent: This option is only available when using a Surface History File. When active, this value is used as a tolerance for reporting volumes. For example, if this value is set to 1%, any volumes less than 1% of the total bench volume will be merged into the previous bench.

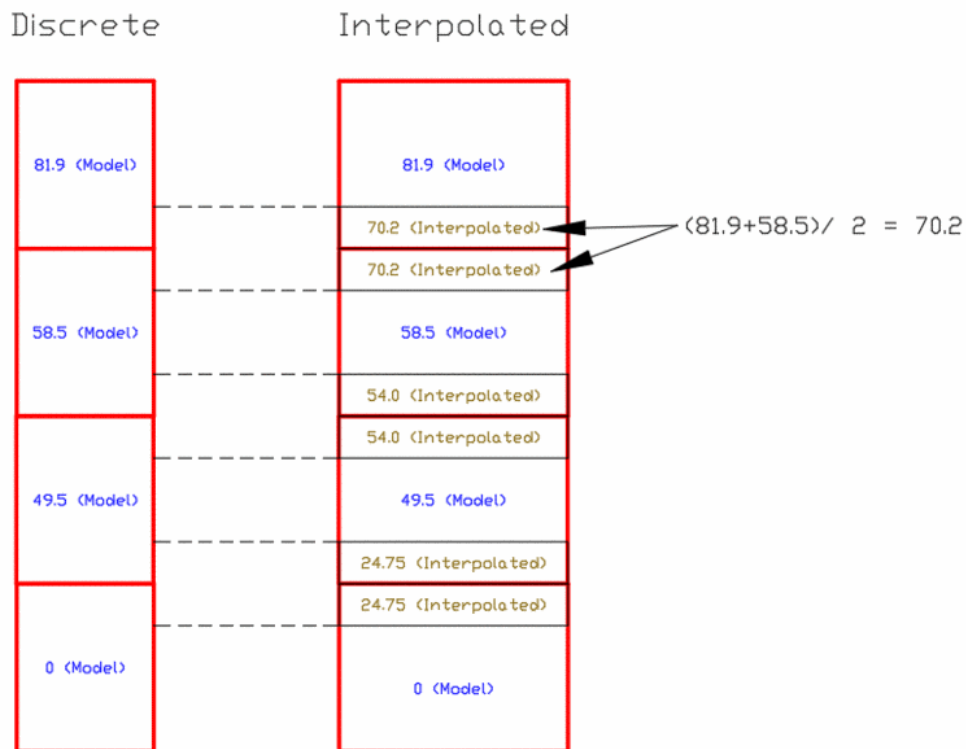
Strata to Include: This option determines which strata layers will be calculated for the Report. If the Selected option is chosen, you will be prompted with a dialog for which strata to include. If you are not using the Auto-Run option, the Selected option can be useful for storing specific strata layers onto a bench.

Grid Subdivision Precision: This option controls how grid cells will be subdivided along angled borders for more precise calculations. For the most accurate results, use the High option. Using the Low option will decrease accuracy in exchange for improved calculation time. For grid volume calculations, the program subdivides grid cells that cross a boundary. This setting controls how many times the cell is subdivided. Subdividing more increases the volume accuracy but takes longer. The default for this setting is High. It only makes sense to set it to Low when you're in a hurry and you've got a huge model and a lots of pits. Otherwise, the routine should be fast anyway that you might as well use High and get full precision.

Calculate Strata Qualities: This option controls if strata qualities will be calculated. These are the user defined attributes such as BTU, Ash, Moisture, etc. Non-user defined attributes such as thickness, area, etc. will always be calculated. When qualities are merged for reporting, by default they will be weight averaged according to the strata volume and density. Here it is important to note that the weight averaging of attributes will not account for null values. For example, if Strata A has a Moisture attribute of 0.3, but Strata B does not have a Moisture attribute defined, then the combined Moisture content for the two stratum will be 0.3. To properly weight attributes, it is necessary to define an attribute value for each strata. Besides reporting the averaged quality, the program also reports the minimum and maximum quality values. When qualities have not been defined for all strata, you may find the Fixed Non-Key Qualities option useful.

Breakout Quantities by Attributes: This option can only be used when a .blk file is used (either generated on-the-fly from drillholes or when it is stored in a Geologic/Mining Model). When enabled, you will be prompted for a Grade Parameter File (.gpf file) which defines the various grades of the material. This option will report the quantities of material according to this file. For example, if the .gpf file relating to a limestone project defines Grade A material as having 90-100% Calcium, Grade B as having 80-90% Calcium, etc., then the report will calculate the tonnage, volume, and average Calcium content for each grade individually.

The Off option will not breakout quantities by attribute. You will not be prompted for a .gpf file with this option. The Interpolated option will actually interpolate block values between data points, thus allowing for variation of grade within each block. The Discrete option will treat block values as discrete for the entire block. This prevents any variation of grade within individual blocks. A comparison of the Discrete and Interpolated options is shown below. Notice that in the Discrete method, each red block is homogeneous. There is no variation of quality within the block. For the interpolated option, however, each red block is subdivided. Half of the block retains the modeled value, while the upper and lower fourth of each block is a calculated average.



Fixed Non-Key Qualities: This option will prompt you to enter a fixed value for each strata attribute found in the drillholes or in the Geologic/Mining Model. These values will be used for all non-key strata. This option is useful for properly weighting attributes due to dilution with non-key material. For example if you have defined an attribute for Sulfur content in your key strata, but not your non-key strata, this option will allow you to quickly assign a Sulfur value to the non-key material for attribute weighting. Using this option for proper attribute weighting is faster than modifying the drillholes or Geologic/Mining Model with these attributes. Here it is important to note that if a non-key strata attribute has been defined in the Geologic/Mining Model, this option will overwrite that value.

Use Named Pit Areas: This option will restrict the calculation to use only closed polylines that have been tagged as pits. Polylines may be tagged as pits using one of the various pit tagging commands in the Surface Mining Module > Boundary Pulldown Menu. These commands include Name Pit Polylines, Assign Pit Names By Layer, Pit by Interior Point, Pit by Interior Text, Pit Matrix Layout, Pit Layout by Advance, Pit Layout by Width, and Pit Layout by Rate. When this option is used, reserves will be split into their respective pits. When this option is not used, you may still use pit polylines (or other, untagged polylines) as inclusion areas, but the reserves will not be divided by pit.

Store Results in Pits: This option is only available when the Use Named Pit Areas option is used. This option will store the total non-key volume, key volume, key tons and all quality attributes in the pit polyline as extended entity data. These quantities and attributes can then be used by the Surface Equipment Timing command. Besides the

quantity and attribute values, a Bench number is also stored with the quantities for sequencing each bench. Here it is useful to note that all the material stored on a single bench must be mined together. If two types of material are to be mined separately, they should be placed on separate benches with the Use Auto-Run option or by repeating the Surface Mine Reserves command and incrementing the Bench Number while selecting specific strata.

Rather than storing discrete values in the pits, the non-key volume, key volume, and key tonnage may be stored in the pits as grid files. If the Output Thickness Grids option on the Output tab is used, these three values will be stored in the pits as grids rather than discrete values. Note that you may also store grids in the pits with the Surface Mining Module > Boundary Pulldown Menu > Pit Timing Quantities > Assign Timing Grids command. When this command is run, the grids will overwrite the discrete values.

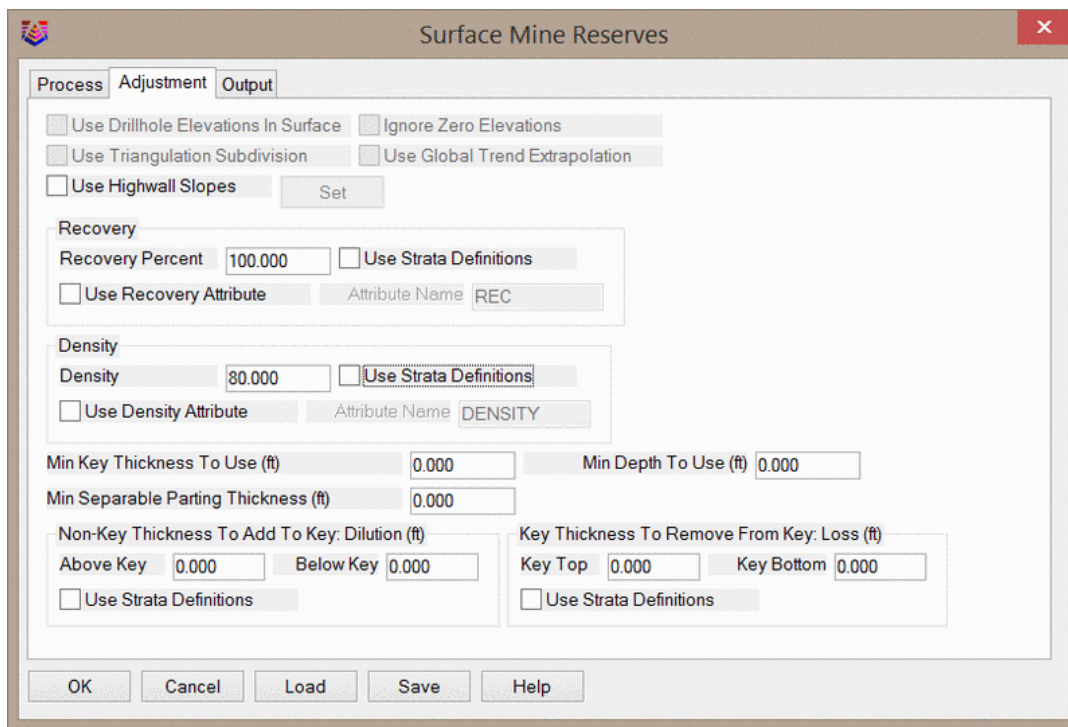
Bench #: This value will set the bench number for storing results in pit polylines. Note that any reserves calculated will be stored together on this bench number. Note that benches must be integer values - alphanumeric bench numbers are not supported.

Adjust Pits Manager: This option will allow you to adjust the pit boundaries after the calculation is complete, then reprocess to check the updated results. This can be useful when trying to target a specific volume/tonnage to store in a pit. When this option is enabled, the below Pits Manager dialog will be docked to the bottom of the screen. You can then modify the pit polylines with standard CAD commands (move, grip edit, etc.). Clicking the Process button will then rerun the calculation and display the results. Clicking the Report button will send the results to the Report Formatter. Clicking the Exit button will close the Pits Manager without generating a report.

Process	Pit	Strata	Pit Area S.F.	Total C.Y.	COAL Tons	BTU
Report	Pit 1	C1_TOP	410462.8	1242447.5		
	Pit 1	C1_KEY	410462.8	101102.0	120109.2	12007.713
	Pit 1	C2_TOP	410462.8	325767.2		
	Pit 1	C2_KEY	410462.8	91401.2	108584.6	11492.317
	Pit 2	C1_TOP	233272.6	961928.5		

Use Property Boundaries: This option will divide the reserves according to Property Boundaries, which are closed polylines that have been tagged using the Assign Property Names or Property Names by Text command (available in Surface Mining Module > Boundary Pulldown Menu and Underground Mining Module > Property Pulldown Menu). As long as an inclusion polyline is selected, these property boundaries will be automatically selected when the calculation is performed, even if the property boundaries are on a frozen layer. If an area lies outside of a tagged property boundary, the reported field for Owner will simply say "Unknown" rather than using one of the tagged owner names. It is important to note that "island" property boundaries are supported - for example, if Property A is completely inside of Property B, the program will not double-count the volume/tonnage in Property A.

Use Reserve Classification: This option will divide the reserves according to a Reserve Classification file (.rsv file) into Measured, Indicated, Inferred, and Hypothetical categories. This file can be generated with the Geology Module > Grids Pulldown Menu > Reserve Classification command. When this option is used, the Report Formatter will include a new attribute called "Reserve Class."



Use Drillhole Elevations in Surface: When calculating the reserves directly from drillholes, this option determines if the drillhole collar elevations will be used to generate the topographic surface.

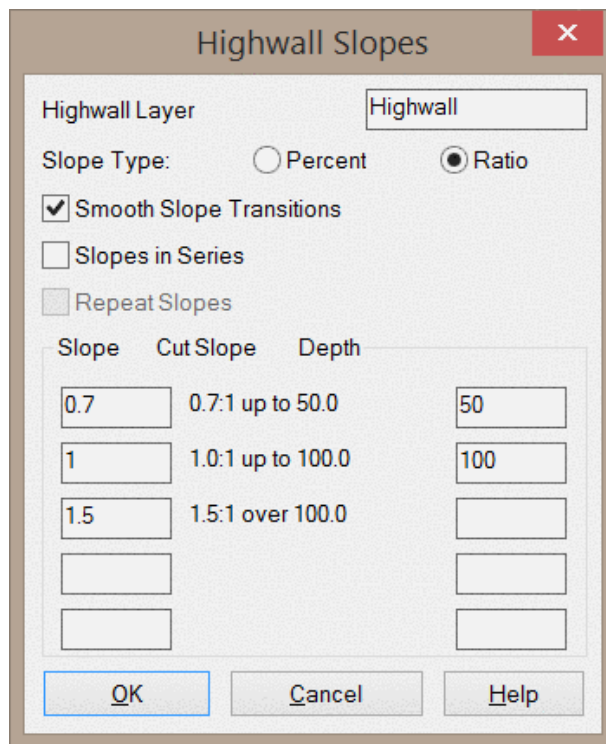
Ignore Zero Elevations: When calculating reserves directly from drillholes, this option will determine if entities at zero elevation will be used to generate the topographic surface. Generally, this option is useful in that it allows you to broadly select entities in the drawing without manually filtering out zero elevation entities.

Use Triangulation Subdivision: This option only applies when using the Triangulation or Polynomial modeling methods to calculate reserves directly from drillholes. When enabled, this option will subdivide triangles to create smoother surfaces.

Use Global Trend Extrapolation: This option only applies when using the Triangulation or Polynomial modeling methods to calculate reserves directly from drillholes. When enabled, this option finds the average slope and direction of the interpolated data and applies this slope to extrapolate to the full extents of the grid.

Use Highwall Slopes: This option will apply highwall slopes to the inclusion polylines to simulate a more realistically shaped pit. This option is limited in that the inclusion polylines selected for reserve calculation must represent the toe of the highwall. This option will project highwalls upwards until it reaches the topography. This option does not account for benches, and so more complex calculations will need to be made using a Surface History File generated from the Design Bench Pit command.

Set (Slopes): This button will open the dialog shown below, which controls the highwall slopes to use.



Highwall Layer: This field sets the layer to use for the highwall linework. This linework will be drawn as a 3D polyline where the highwall projection meets the surface.

Slope Type: This option controls if the highwall slope is expressed as a percent or as a ratio.

Smooth Slope Transitions: This option will gradually transition between slope angles as depth increases around the pit. In the above example, the pit will have an overall slope ratio of 1.0 where the cut depth is between 50 and 100', and a slope ratio of 1.5 for cut depths greater than 100'. In areas where the cut depth is exactly 100', this option will smoothly transition from a slope ratio of 1.0 to 1.5 rather than abruptly changing the slope angle.

Slope in Series: This option will use all slope angles specified in the dialog. Using the above dialog as an example, the cut angle from 0-50 feet (or meters) will always be a 0.7 ratio. After this first 50 feet (or meters) of cut, the slope will be a 1.0 ratio for the next 50 feet (or meters). Any remaining cut will use a 1.5 ratio.

Repeat Slopes: This option will repeat the slopes with specified depths until the projection meets the surface topography. In the above example, the program will repeat cut ratios of 0.7 and 1.0 until it reaches the surface. Since no cut depth has been specified for the 1.5 cut ratio, this slope angle will not be used with this option.

Recovery Percent: This value will set the overall recovery percent for key strata. This is applied after adjustments for non-key dilution and key loss. The volume of material that is not recovered will be reported as non-key material.

Use Strata Definitions: This option will use the recovery percent specified in the current Strata Definition file. This allows you to set a recovery for each strata individually.

Use Recovery Attribute: This option will set the recovery percent according to an attribute defined in the drillholes or in the Geologic/Mining Model. This allows you to use a grid file to set the recovery percent, thus allowing the recovery of each strata to vary.

Attribute Name: This field sets the recovery attribute name to search for in the drillholes or Geologic/Mining Model. This attribute name must exactly match the attribute in the drillholes/model, or else the recovery percent will not be applied.

Density: This value sets the density to use for all strata. Density is always expressed in lbs/ft³ or kg/m³.

Use Strata Definitions: This option will use the density specified in the current Strata Definition file. This allows you to set a density for each strata individually.

Use Density Attribute: This option will set the density according to an attribute in the drillholes or in the Geologic/Mining Model. This allows you to use a grid file to set the density, thus allowing for a varying density within each individual strata.

Attribute Name: This field sets the attribute name to use for density in the drillholes or Geologic/Mining Model. This attribute name must exactly match the attribute in the drillholes/model, or else the density will not be applied.

Min Key Thickness to Use: This value sets a minimum key thickness for reporting. Any areas where the key strata is less thick than this value, the strata will be reported as non-key discard material.

Min Depth to Use: This value sets a minimum depth of cover for key strata for reporting. Any areas of key strata that have less than the specified thickness of cover will be reported as non-key discard material. An example use of this option is accounting for oxidized coal that is (or is close to) outcropping.

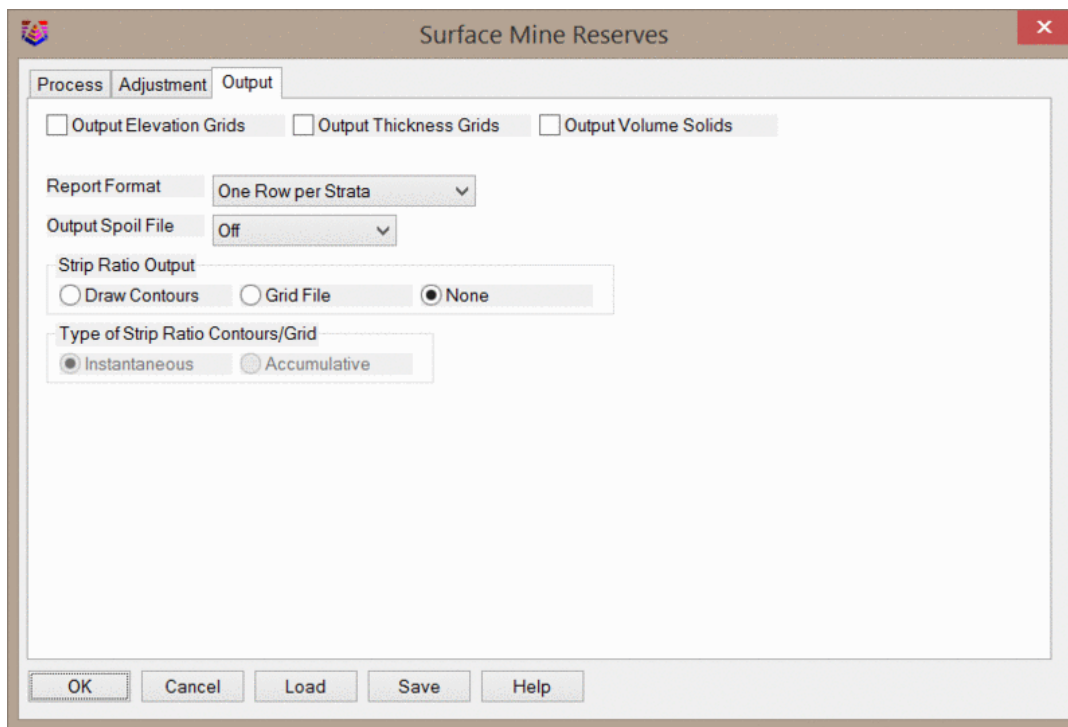
Min Separable Parting Thickness: This value will be used as a tolerance for calculating volume of non-key material between key strata based on thickness. For example, if this value is set to 0.5, any non-key material sandwiched between key strata that is less than 0.5 feet (or meters) thick will be reported as key material. Areas where the non-key strata is greater than 0.5 thick will not be affected.

Non-Key Thickness To Add To Key (Dilution): Above Key/Below Key: These values set the amount of non-key thickness above/below each key strata that will be mined with the key strata. This diluted non-key material will not be mixed with the key strata, but will instead be reported as another key strata layer. The final report, however, can be formatted to account for the dilution of strata qualities. Note that dilution will be accounted for determining the Stripping Ratio in the final report and in grid/contour output.

Use Strata Definitions: This option will use the Dilution values set in the Strata Definition File. This allows you to set the Dilution for each strata individually.

Key Thickness To Remove From Key (Loss): Above Key/Below Key: These values specify the amount of key thickness above/below the key strata that will be subtracted from the key. This amount will be subtracted from the key quantities and reported as non-key material. Note that loss will be accounted for determining the Stripping Ratio in the final report and in grid/contour output.

Use Strata Definitions: This option will use the Loss values set in the Strata Definition File. This allows you to set the Loss for each strata individually.



Output Elevation Grids: This option will create grid files for the bottom elevation of each strata. Each grid will need to be named separately, and so it is recommended to instead use the Strata Grids Autorun to make these grids quickly.

Output Thickness Grids: This option will create composite grid files for the total key thickness, non-key thickness, and key tonnage (expressed in tons/sq ft or tonnes/sq m). Each grid file will need to be named separately. Note that when the option to Store Results in Pits is used, these grids will be referenced in the pit polylines. When this option is used, you will also be prompted to Divide Bench by Thickness. This division will create three more grid files that represent the divided thickness.

Output Volume Solids: This option will create solids of the mined bench according to the pit names. These solids will be automatically saved in a PIT_MODEL folder as .mdl files. This PIT_MODEL folder will be automatically created in the current project folder.

Report Format: This option controls the format of the report.

The One Row per Strata option will report each strata layer on a separate row. This default option is the most commonly used report format.

The All Strata on Same Row option will report all strata layers on a single row. This creates many extra columns for the report, but allows for easier comparison of pit volumes.

The Group Key/Non-Key Pairs option will place overburden strata on the same row as the key strata. This option searches for similar naming in the strata names. For example, the strata COAL_KEY and COAL_OB will be grouped together due to the similarity of the strata names.

Output Spoil File: This option allows you to create a Spoil Source File (.spo file) for use with the Spoil Placement Timing command. This file includes the volume and centroid location of waste material to be placed in spoil piles.

The Off option will not create a Spoil Source File.

The Non-Key Only will only include non-key strata in the Spoil Source File.

The All Strata option will include all strata in the Spoil Source File.

Strip Ratio Output: This option allows you to output stripping ratio as contours or a grid file. The units of stripping ratio will default to $(\text{yd}^3 \text{ waste})/(\text{ton key})$ or $(\text{m}^3 \text{ waste})/(\text{tonne key})$, but different units may be selected in Carlson Configure.

The None option will not output the stripping ratio as contours or a grid file. Note that when this option is used, the stripping ratio will still be calculated in the final report.

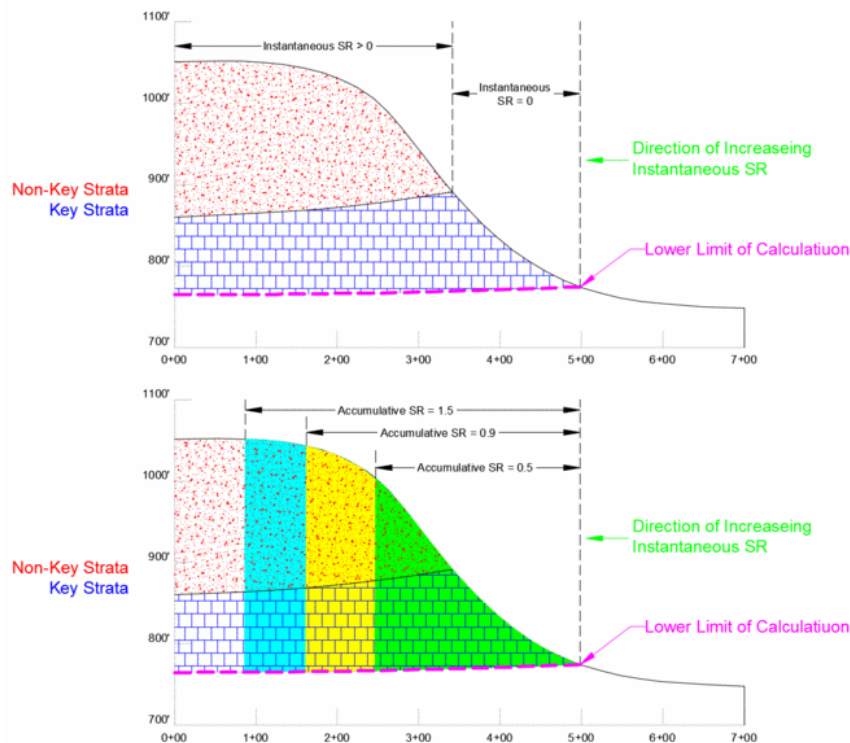
The Draw Contours option will output contour lines of the stripping ratio. This option will use settings similar to those used in the Contour from Grid File command.

The Grid File option will create a grid file that represents the stripping ratio.

Type of Strip Ratio Contours/Grid: This option controls how the stripping ratio is expressed when it is output to contours or a grid file

The Instantaneous option will calculate the stripping ratio for each individual grid node. This stripping ratio will account for all non-key and all key material within the calculation limits - any non-key material existing below the last key strata will also be used for the calculation of the stripping ratio. When the reserves are divided into benches, the stripping ratio will be reported by-bench.

The Accumulative option is intended for use in situations where the key strata outcrops and is relatively flat. The below images illustrate the way this form of stripping ratio is calculated based on direction. The program will first determine the lowest Instantaneous stripping ratio in the calculation area. The program will then determine in which direction the stripping ratio increases the least. If mining were to start in the area of lowest stripping ratio and then move one grid node in the direction of least increasing stripping ratio, the Accumulative stripping ratio would be based on all key and non-key strata between the start point and the next grid node location. In other words, imagine that mining starts at the outcrop of the key material. As mining progresses into the hillside, the accumulative stripping ratio is the overall stripping ratio of all material mined up to the current position.



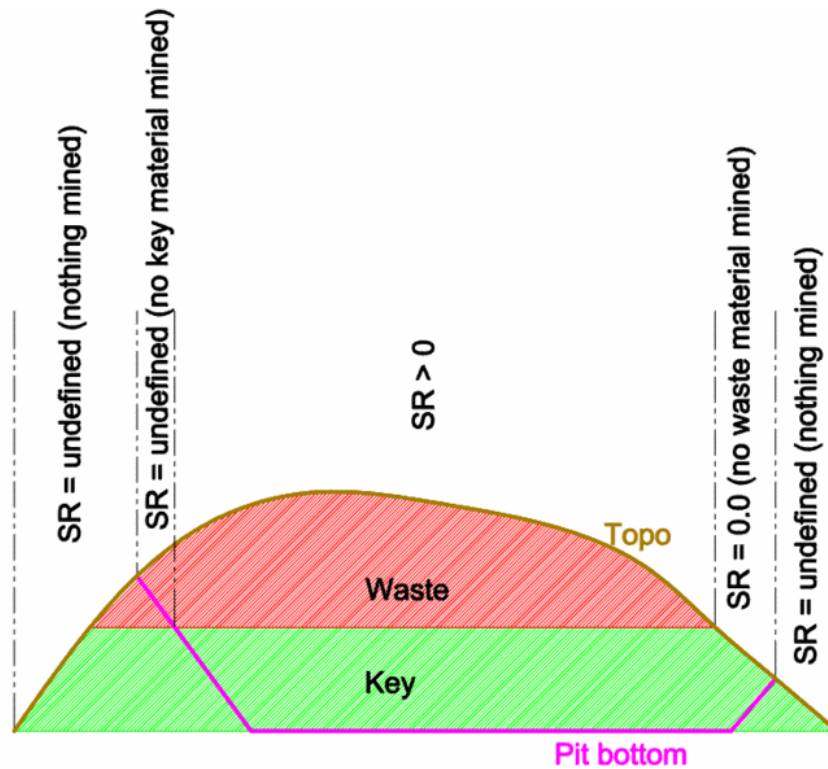
There are some situations in which a stripping ratio cannot be calculated, as shown in the below illustration.

For areas in which neither waste or key material are included, the stripping ratio will be undefined - if a stripping ratio grid is created, it will be null in this area.

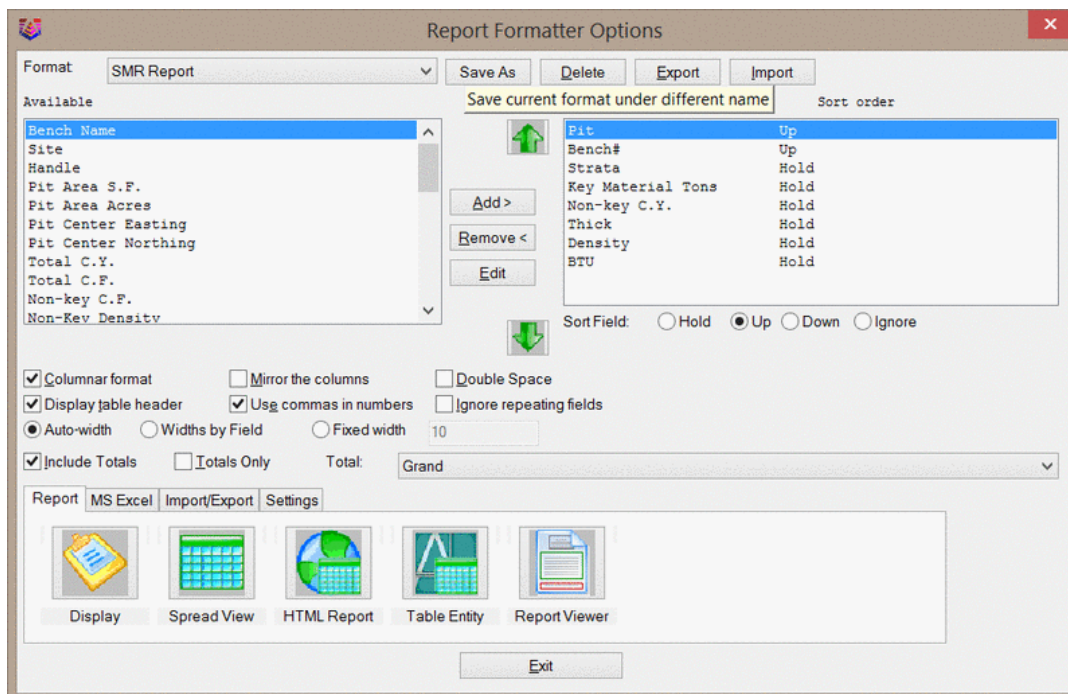
For areas in which only waste material is included, the stripping ratio will also be undefined- if a stripping ratio grid is created, it will be null in this area.

For areas in which no waste material is included, the stripping ratio will be zero.

For areas in which waste material is included, but only a very small portion of key material is included, the stripping ratio can approach an undefined value, which may appear as spikes in the stripping ratio grid file. This is normal for areas with a very small amount of key material. However, the stripping ratio output grid will not exceed a value of 1000. Spikes in the stripping ratio grid can be further filtered with Grid File Utilities.



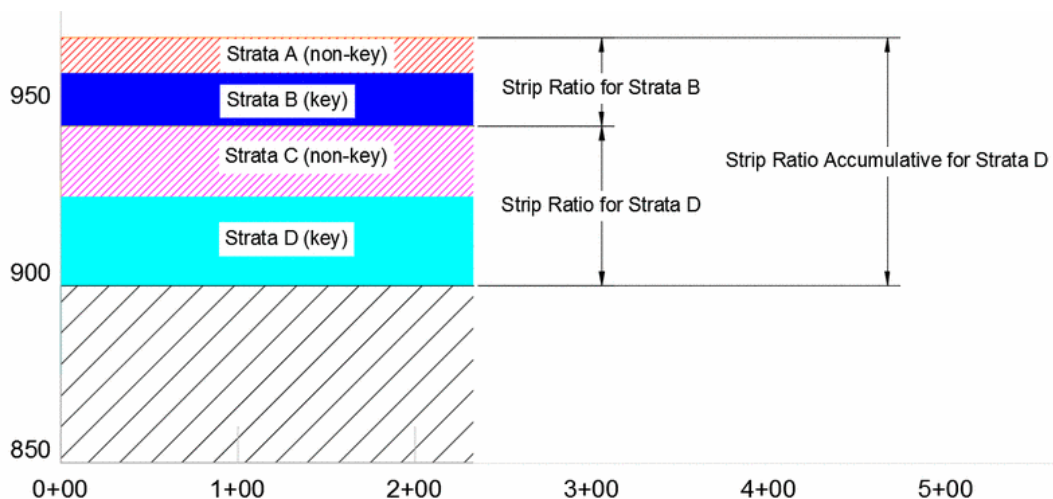
The Report Formatter displays all information calculated with this command, as shown below. For general information on using the Report Formatter, see the corresponding of the help manual.



Here it is important to note that when a parent seam has been configured to split into two child seams according to the strata definition file, the program will automatically report the splits. For example, if seam A splits into B and C, the program will report tonnage for seam A where it exists as a parent as well as seams B and C where they exist as child seams.

As an additional means of dividing reserves, Cut Sets defined by the GIS Module > GIS Tools Pulldown Menu > Polygon Processor command will be considered when the reserves are calculated. If a Cut Set has been defined for wetland areas with a name of "Wetlands", an additional reporting field will be available in the report format called PP_Wetland. This will show the amount of strata inside the wetland area. Any strata outside the wetland area will be reported separately.

The report formatter will include two forms of stripping ratio: "Strip Ratio" and "Strip Ratio Accumulative". The latter term should not be confused with the Accumulative Stripping Ratio for the stripping ratio grid/contour output. Consider four strata layers: A, B, C, and D where stratum A and C are non-key while stratum B and D are key. When these four stratum are calculated on a single bench, the report will include a "Strip Ratio" and "Strip Ratio Accumulative" for stratum B and D. Both attributes reported for strata B report the stripping ratio as if only strata A and B were mined. The "Strip Ratio" for strata D will report the stripping ratio as if only strata C and D were mined. The "Strip Ratio Accumulative" reported for strata D will report the stripping ratio as if all strata above it were also mined (stratum A and B will be included). The below image is an illustration of this concept.



Miscellaneous note on block models: When the program detects a Geologic Model containing one or more Block Models (.blk files), the thickness of the strata containing the block model will be compared to the thickness of the block model itself. Recall that each strata in a Geologic Model is defined by the difference between two elevation grids (or a thickness grid). Also, a block model will be defined with upper and lower limits (also two elevation grids). If the strata thickness is greater than the block model thickness, the program will extend the block model thickness an extra half-block height on the top and bottom of the model.

Carlson block models will always have top/bottom layers of blocks with half the height of other blocks (for more information on this, see the section of the help manual corresponding to the Make Block Model command). To account for the full block size, surface mine reserves applies the above logic. The idea is that whenever the true strata thickness is thicker than the block model thickness, the extra top/bottom block height needs to be accounted for.

Two types of thickness values can be reported for each strata: Thick Geologic and Thick Mined. Thick Geologic is the full Geologic Thickness, as if no constraints were applied to the model. Thick Mined, however, reports the thickness with all constraints applied (such as a pit shell that only cuts through a portion of the strata). When reporting Pit Volumes, it is recommended to report Thick Mined, as it will reflect the thickness extracted only within the pit shell. It is worth mentioning that when average thickness is calculated, an area-based weight is applied rather than a volume-based weighting.

Two types of areas are included in the report: Area and Pit Area. The Pit Area is the area of the inclusion line used for the report. The Area, however, is not a measurement, but rather a calculation based on the volume and the full Geologic Thickness. The Area is calculated as Volume divided by Thick Geologic.

Prompts

Surface Mine Reserves dialog

Select surface entities and at least 3 drillholes. (Unless using a Geologic Model File PRE.)

Select objects: *select the drillhole symbols and surface entities.* Surface entities can include points, lines, and polylines.

Select the Inclusion perimeter polylines and ENTER for none:

Select objects: *select the polylines or named pit polylines.* The area within these polylines will be included in the calculations. They must be closed polylines.

Select the Exclusion perimeter polylines and ENTER for none:

Select objects: *select the polylines.* The area within these polylines will be excluded from the calculations. They must be closed polylines.

Make Grid File Set grid resolution

Triangulating points ... 49

Pass > 6 NULL Z values left > 0

Processing cell 2500 ...

Finished strata Y2

The above four steps are repeated for each strata.

Report Formatter

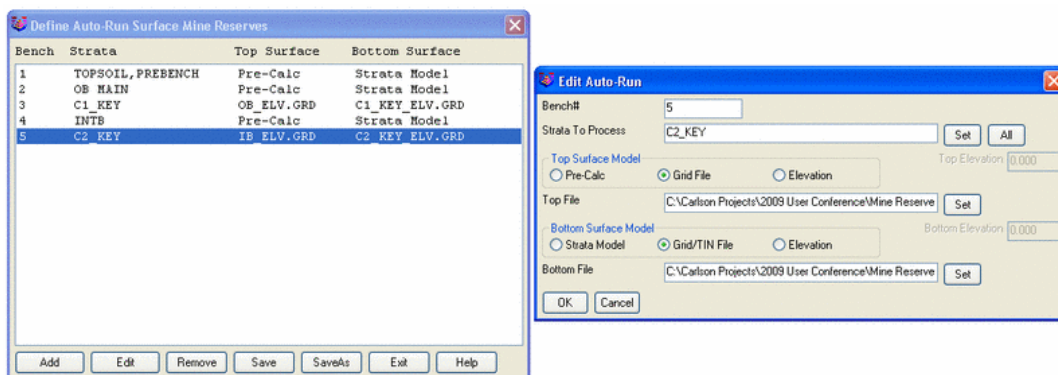
Pulldown Menu Location: Geology Module > StrataCalc and Surface Mining Module > Reserves/Timing

Keyboard Command: mtntop

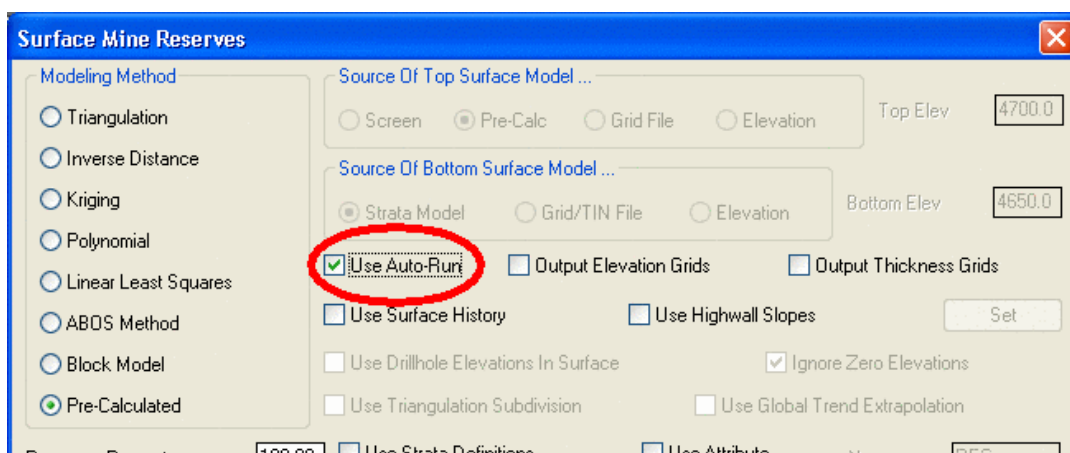
Define Surface Mine Auto-Run

This command is the setup routine to create the autorun file for batch processing of multiple benches with one run of the Surface Mine Reserves command. Without this predefined autorun file, then the Surface Mine Reserves command needs to be run separately for each bench. With this Autorun file, all benches are predefined by the Geologic Model file, a top and bottom Grid or TIN, or an Elevation.

The steps necessary to create or edit the file are detailed below. Choosing Add or Edit will bring up the Edit Auto-Run window. This is where the Bench# is set. The Strata to Process can either be set to one, multiple, or all to define the benches. If coming out of the Geologic Model file, then the elevation grids defined in there will define the benches here. Alternately, the benches can be other grid or TIN surfaces, or flat elevations, and they will be intersected by the seams that appear in the Geologic Model file.



To utilize this Autorun file, turn on the option in the Surface Mine Reserves window for "Use Auto-Run" as circled below. Once this is on, the program will prompt to select the *.SMA file this command created.



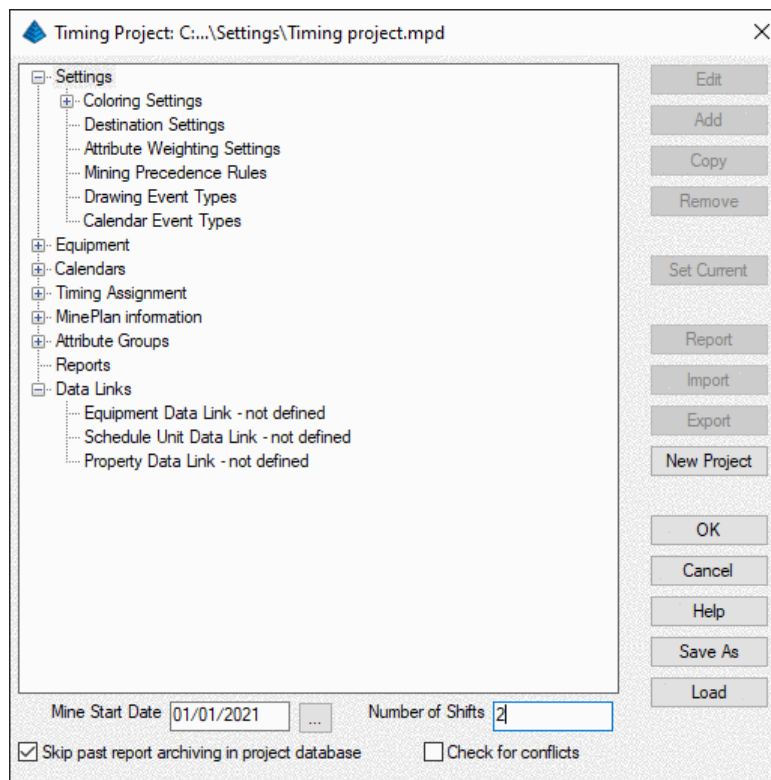
Pull down Menu Location: StrataCalc

Keyboard Command: mntop_atorun

Prerequisite: Grids and PRE Calculated Grids file, named pits

Timing Project Manager

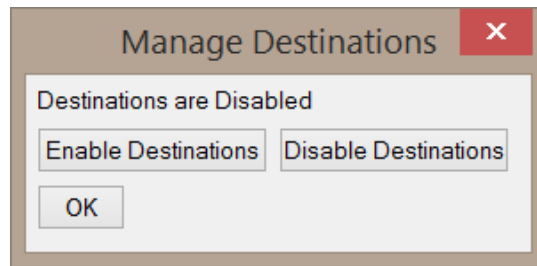
The Timing Project Manager (Surface Project Manager or Underground Project Manager) applies to both surface and underground mining for file organization and selection. All files and most settings and configurations used in the mine scheduling are found in the manager. Each topic is described below in detail. Note that some topics listed below may refer to panels for Underground Timing or Pits for Surface Equipment Timing.



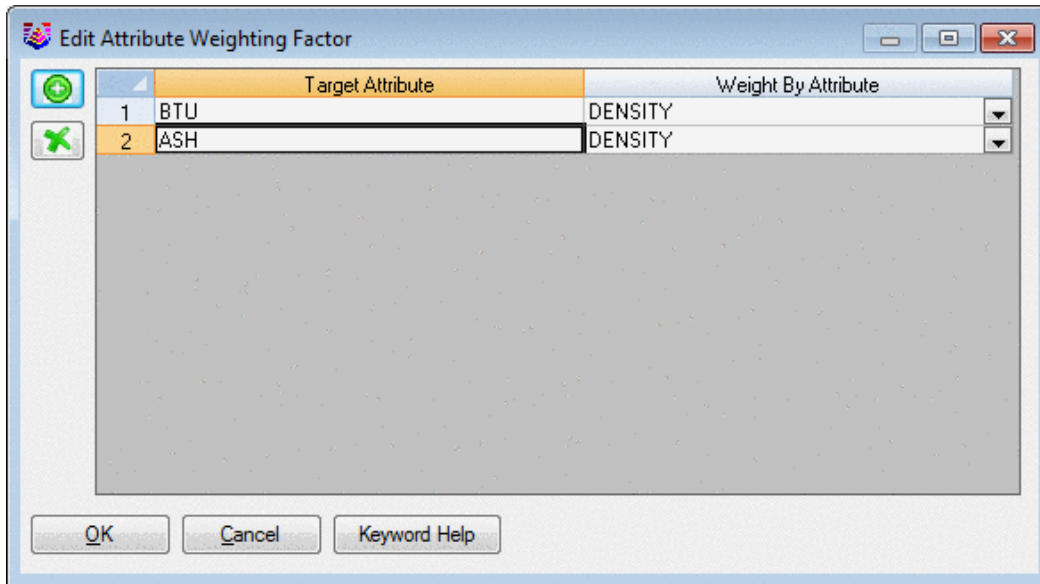
Settings

- **Coloring Settings:** These settings are from the Report Options screen from the equipment timing command. Defaults can be set here before running the actual schedule. Explanation of this dialog is found in the help documentation for Underground Timing or Surface Equipment Timing.
- **Destination Settings:** This gives the ability to assign destinations to pits, which can then be reported separately in the timing report. For example, if the material from some pits need to be sent to the waste pile while the material from other pits need to be sent to the processing plant, adding destinations attributes will provide an additional category for reporting. To assign destinations to pits, you must open the Timing Project Manager through the Surface Equipment Timing command and select pit polylines. When editing destination settings, the below dialog will appear. Clicking the **Enable Destinations** button will simply add a DESTINATION attribute to each bench in each pit polyline. This Destination can be modified manually with the Edit Pit

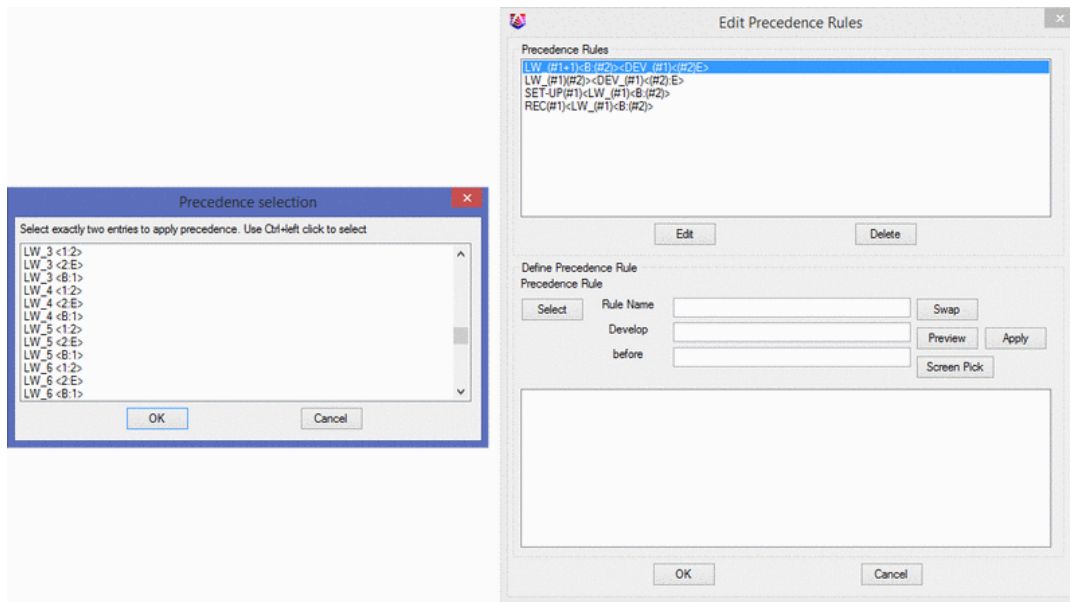
command or according to a grade parameter file via the 3D Pick Function within Surface Equipment Timing. Clicking the **Disable Destinations** button will remove this DESTINATION attribute from the pit polylines.



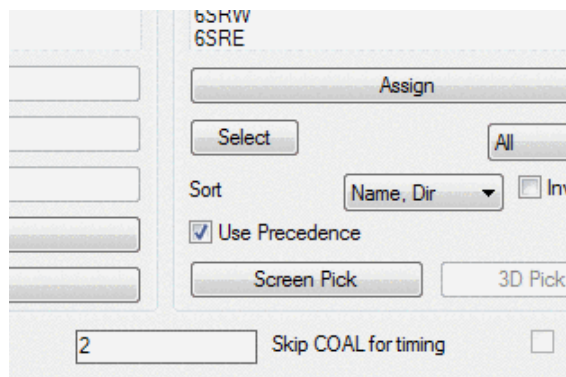
- **Attribute Weighting Settings:** This gives the ability to weight average attributes by other attributes, both calculated and internal. Add new lines with the + button. Enter in the Attribute, or pick it from the Keyword Help button. Then dropdown the Weight By Attribute. This will show up in the Report Formatter once the schedule is ran.



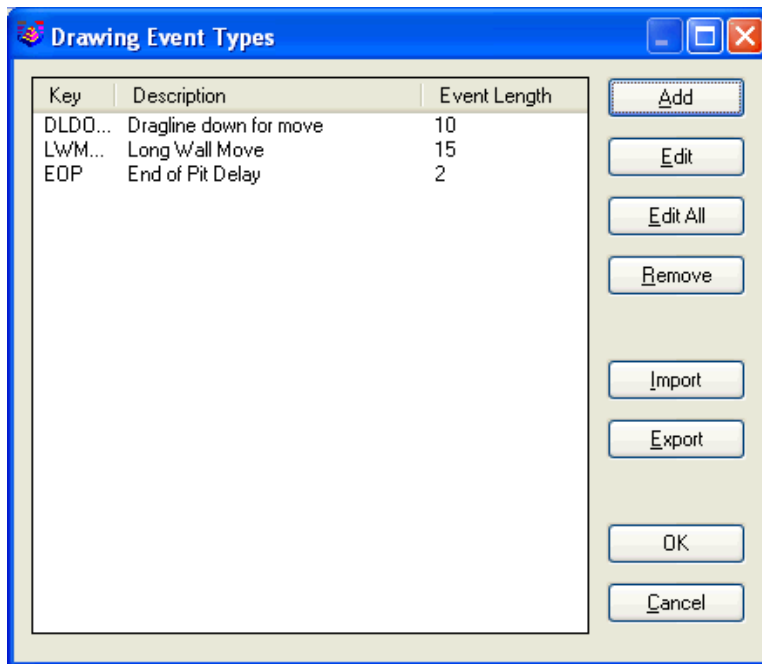
- **Mining Precedence Rules:** This allows for automatic precedence using an user-defined sequence. When using commands such as AutoPlace Panel by Text, it does not set precedence. Using this command makes sure panels do not start before they are exposed or available to be mined. Each rule is shown at the top. Start by choosing the Select button and select two entries that are compared. Then choose which one goes first, if it is wrong, use the Swap button. Give it a name (optional) and hit Apply. The Preview button will show the order in the lower window. The Screen Pick button allows for choosing precedence on a screen. When using Screen Pick start by picking on or inside of first panel polyline and then pick on or inside of second panel polyline.



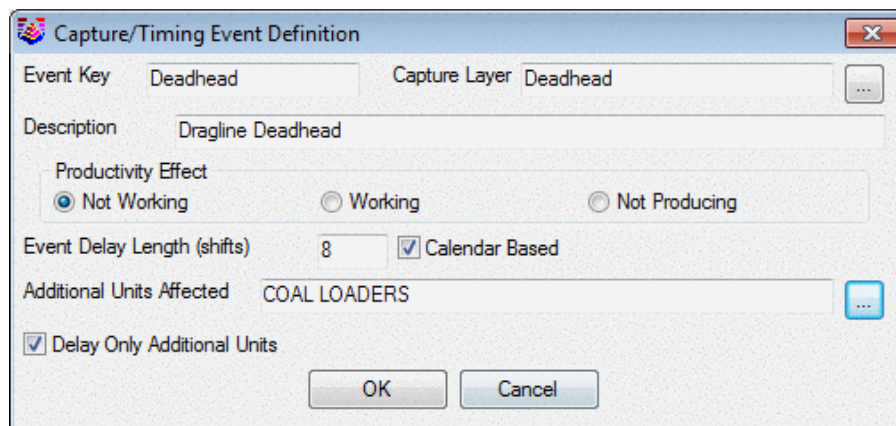
To use this precedence, turn on the Use Precedence option in the Sequence Window, as seen here and the blocks and panels will reorganize to fit the order.



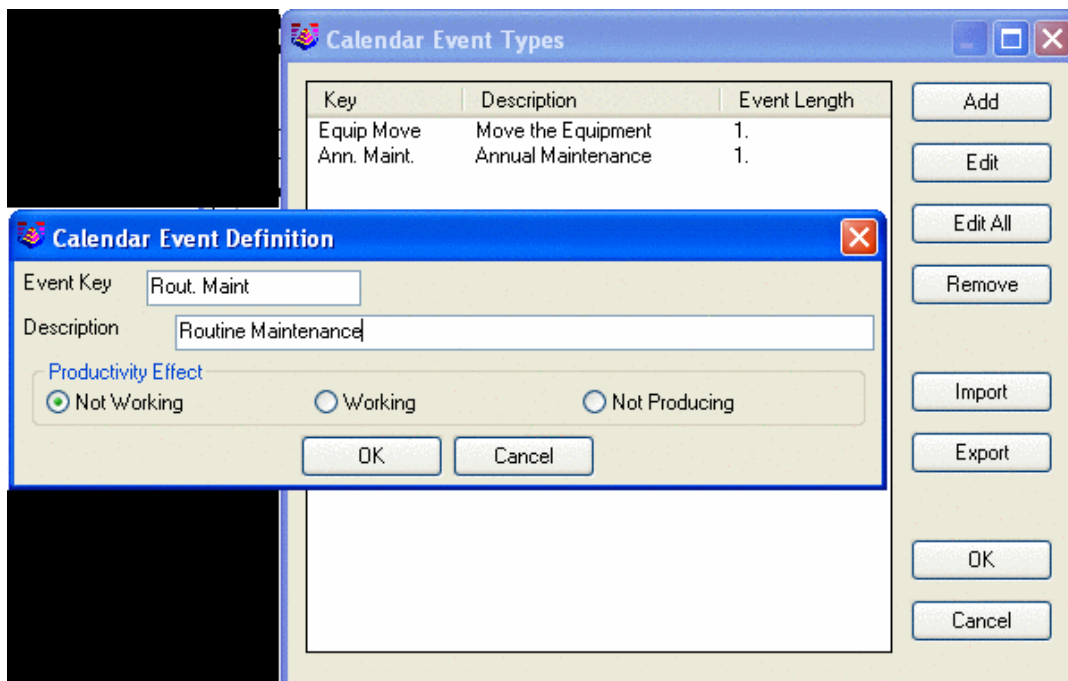
- Drawing Event Types:** Drawing events are entities in the drawing at a specific location that will be used for a marker in the schedule as an option for a delay. This event can be used to delay equipment, switch crews, move other equipment, etc. When the schedule crosses over the insertion point of the drawing event (usually text) the event will trigger the delay or effect. The text/drawing entity in the drawing should be inserted to the position where the delay will occur and in the AutoCAD layer defined in the Capture Layer window. The Event Key and description can be used in the reporting of the schedule to create additional reported items.



There are options to set the layer, give it a full description and internal event key name. The Productivity can be set to not working, working (as normal), and not producing (working, but not moving any material). The event delay length is set here by number of shifts. Calendar based will determine if the shift is off in the calendar. If the box is checked, and that shift is off in the calendar already, then it will not take off any more. If it is off, then it will apply to only the next working shift(s) for not working or producing. Additional Units can be used to also set other units off during the same period. For example, if a longwall is down due to a move, then the CM working with it can also be down for the same time. There is also an option to Delay Only Additional Units. This can be used to set one unit down, when a different unit crosses the drawing event. This allows for clever equipment delays and precedence by using different drawing events.

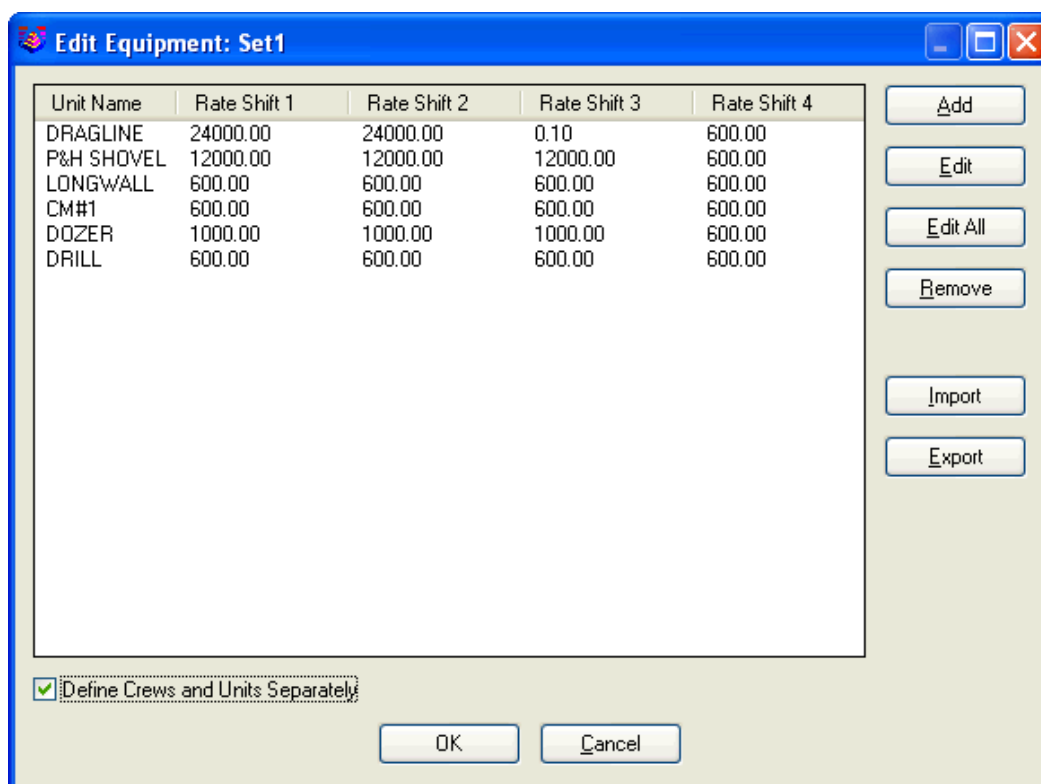


- Calendar Event Types:**Calendar events are named events that will determine equipment down time. The default event in the calendar is Manual. The other events are set under the Calendar Event Types and Calendar Event Definition. Calendar events can be any event that will cause a delay, such as a routine maintenance, a long wall move, or a dragline dead head. The Productivity Effect allows the option of Not Working at all, Working as normal, or Not Producing (yet still working with crews, using hours, but not mining).



Equipment

- **Equipment Set:** The productivity of the unit (crew) may vary from shift to shift, up to 4 shifts a day. The number of shifts is set in the Timing window where the sequence of pits or panels is set. The average productivity rate is used for scheduling, however the production rate for the particular pit or panel may be adjusted by setting the difficulty factor or adding delays for that pit or panel. The first window is the edit equipment screen with a columnar display of equipment. Crews can be defined to work with a specific unit, or can switch from unit to unit based on the schedule.



Add and Edit will bring up the next screen for detailed entry.

The Unit of the Production value is defined by a rate unit setting. Equipment may mine either tons, tonnes, CY or CM, Distance and Linear Advance. The difference between distance and linear foot of advance settings is that latter one is a combined length of all pathways/entries mined (underground only), and distance is just distance moved, of a longwall for example.

Enter the Advance Rate/shift or the Advance Rate/hour and the other will be automatically calculated based on the Hours/shift. The Retreat Rate/shift is for Distance and Linear ft of Advance in underground equipment only. The availability value of less than 1.0 will reduce effective production rate of the equipment. 0.94 is a 94% productivity of the full shift.

Underground units may be assigned an Advance minimum and/or maximum height, so that the extra rock will have to be mined or correspondingly unmined coal will be left in the seam if the maximum miner height is less than combined coal and rock parting thicknesses. These settings affect the underground mine timing only. The maintenance settings provide the ability to schedule a delay for routine or major maintenance/repair of the equipment based on the number of shifts worked. Add in the length of the delay and the number of shifts to determine the frequency.

If operational cost per hour is specified, the total cost will appear in the production report.

The extraction (recovery) factor is used to adjust the amount of material mined from given area to account for certain technical limitations of the equipment such as inability to mine out corners, or not cleaning the top and bottom of ore. This machine will always use that recovery rate.

Advanced Options:

Define Production Change

PH SHOVEL

Date Specific Difficulty		
	End Date	Difficulty
1	11/1/8	1.000
2	4/1/9	1.200
3	11/1/9	1.000
4	4/1/10	1.200
5	11/1/10	1.000
6	4/1/11	1.200
7	11/1/11	1.000
8	4/1/12	1.200
9		
10		
11		
12		
13		
14		
15		
16		

Thickness Specific Difficulty and Rehandle			
	Thickness	Difficulty	Rehandle
1	0	1.000	
2	10	1.050	
3	25	1.100	
4	50	1.150	
5	75	1.170	
6	90	1.200	
7	100	1.500	
8			
9			
10			
11			
12			
13			
14			
15			
16			

Bench Specific Difficulty		
	Bench	Difficulty
1	1	1.000
2	2	1.100
3	3	1.150
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		

Reset OK Cancel Load Save

Under the Advanced options the variation of equipment-related Difficulty Factor with time, depth or bench number may be specified. Resulting from the advanced options, the difficulty factor is a product of coefficients calculated for given date, thickness and bench number. The final difficulty factor used in calculations is a product of location-specific and equipment-specific difficulty factor.

In the Period End Date column, the difficulty for the date is for next, later date, or last one if no later entry exists. For the thickness column the value of difficulty factor between two entries in the table is a linear approximation. The rehandle value is calculated in the same fashion and passed over to the report, not being used in calculations, but can be used in the report in equations to calculate the total amount mined. The difficulty will modify the rate, and the rehandle is reported. These are not necessarily always the same, linear factors. The Bench Specific Difficulty will be used for surface equipment mining on that bench number, and use that difficulty factor to change the rate of the equipment.

Selecting Edit All from the first screen brings up this editor where all equipment may be viewed and edited. Both the first screen and this one have the import and export buttons.

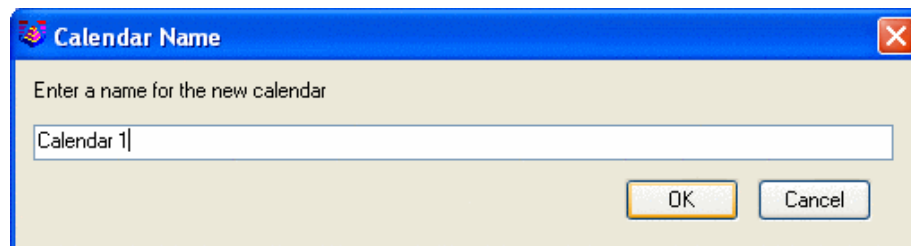
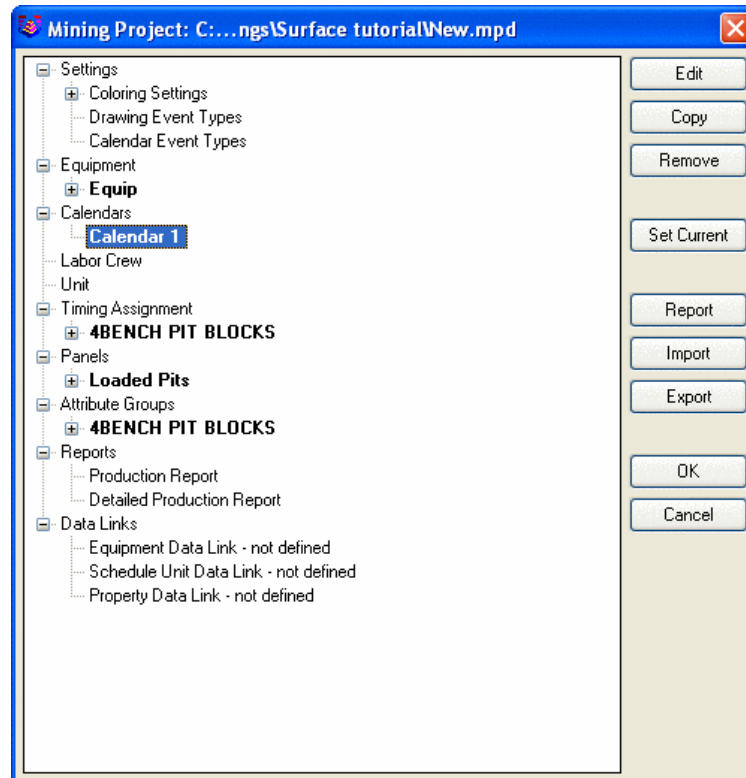
Edit Equipment: Set1

	Unit Name	Rate Shift 1	Rate Shift 2	Rate Shift 3	Rate Shift 4	min height	max height
1	DRAGLINE	24000.00	24000.00	0.10	600.00	0.00	98765432
2	P&H SHOVE	12000.00	12000.00	12000.00	600.00	0.00	98765432
3	LONGWALL	600.00	600.00	600.00	600.00	0.00	98765432
4	CM#1	600.00	600.00	600.00	600.00	0.00	98765432
5	DOZER	1000.00	1000.00	1000.00	600.00	0.00	98765432
6	DRILL	600.00	600.00	600.00	600.00	0.00	98765432

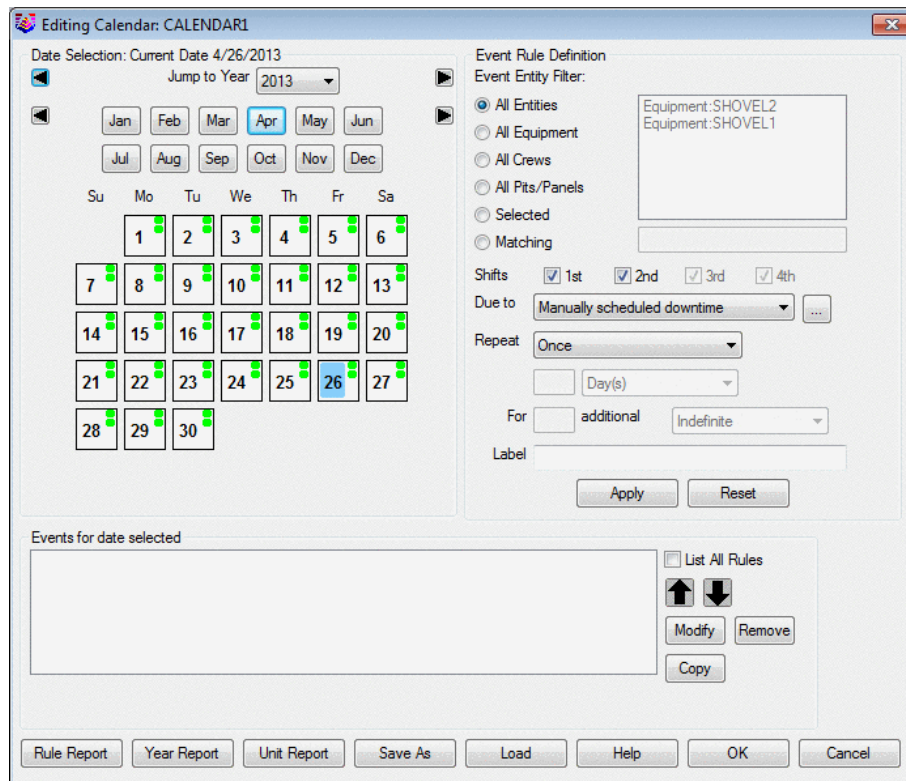
Import Export OK Cancel

Calendars

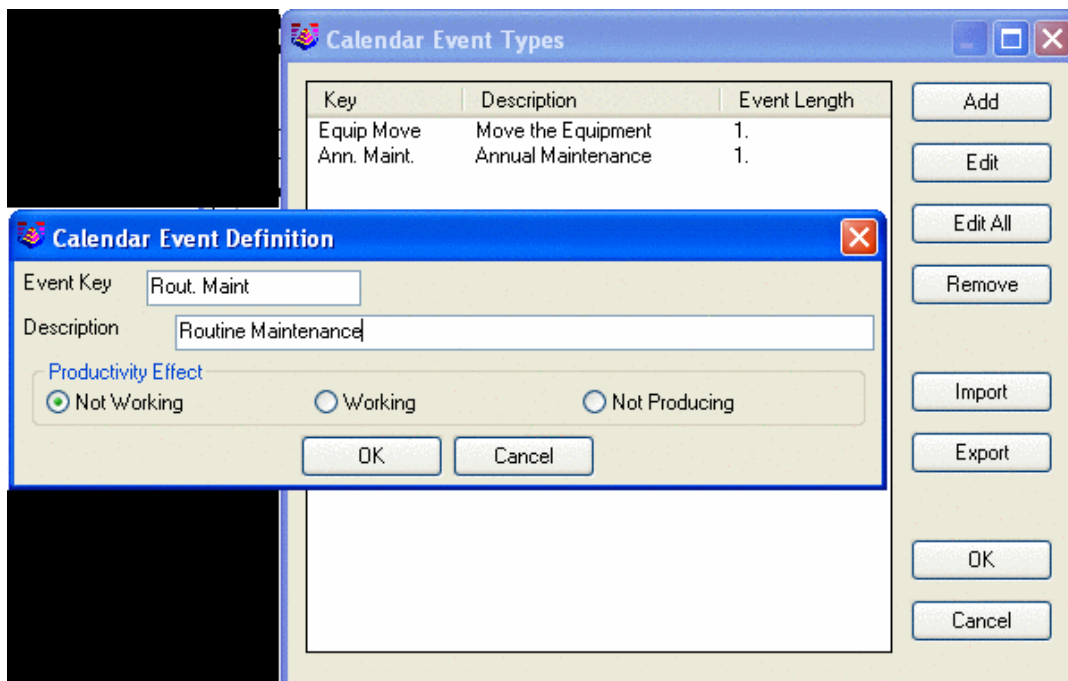
- New or Existing Calendar: Highlight the Calendar tree, and the New Calendar button is activated if no calendars are present. If some are present, the edit button will also be active for editing a calendar. If it is a new calendar, the Calendar Name box appears to enter a name.



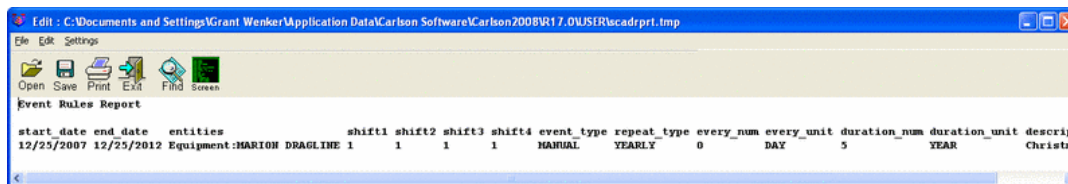
The Equipment Calendar allows for entity production down time. By default, entities are working every day, every shift. Entities include equipment, crews, pits and panels. Days and shifts defined in this calendar as down-time are taken into account during scheduling routines. The assignment pattern is very flexible: it works for a particular crew or all crews, for a particular shift or the whole day and allows replication of the defined behavior over period of time as desired. The calendar will clean out, or purge a schedule for a crew which is no longer present.



- **Date Selection:** The calendar should initiate on the current day. Use the left/right arrows to scroll through the months and years. The current or selected day is shown in blue. The shifts appear on the right of each day. There can be up to 4 shifts. If the shift is green, then all entities are working; if it is yellow, then some are working and some are down. If the shifts are black, then all entities are down on that day and shift.
- **Event Rule Definition:** This section is where the event down times are created and edited. Apply adds the event, Reset clears it for the next assignment.
- **Event Entity Filter:** The event list contains the Equipment, Pits or Panels and Crews. The choices for filtering are All Entities, All Equipment, All Crews, All Pits/Panels, Selected (highlight the desired Entity), and Matching (highlight the Entity to find the match).
- **Shifts:** The number of shifts checked here will display next to each day.
- **Due to/Calendar Events:** The options for down time are selected here. The default event is Manual. The other events are set under the Calendar Event Types, activated by clicking the button next to the "Due to" window. Calendar events can be anything that will cause a delay, such as a routine maintenance, a long wall move, or a dragline dead head. The Productivity Effect allows the option of Not Working at all, Working as normal, or Not Producing (yet still working).



- **Repeat for/Label:** This setting is how often the down time will repeat. This can be from a day, month, year or indefinite. This event can have a name, which is entered in the Label window. The calendar will refer to this label when encountering and reporting this event.
- **Events for Date Selected/All Defined Events:** This window displays the events for the highlighted day. If the List All Rules is on, then all the Events will be displayed here, no matter which day is highlighted.
- **Rule Report:** This report displays all of the rules defined for down time within the calendar.



- **Year Report:** This reports all days for the selected calendar year.
- **Unit Report:** This reports all equipment and dates defined in the calendar. See example below.

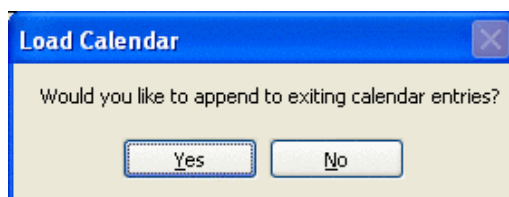
File Edit Settings

Open Save Print Exit Find Screen

Year Unit Report

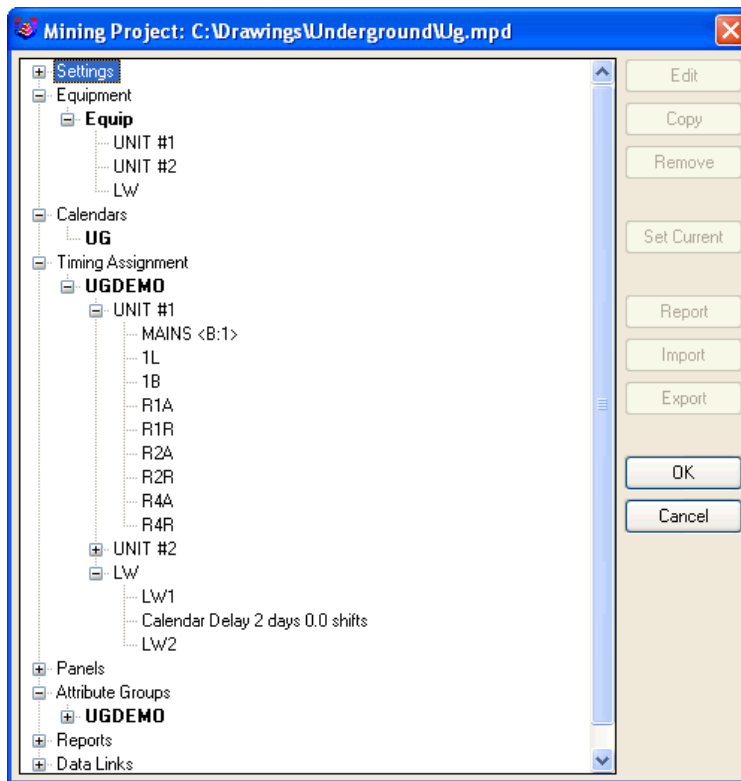
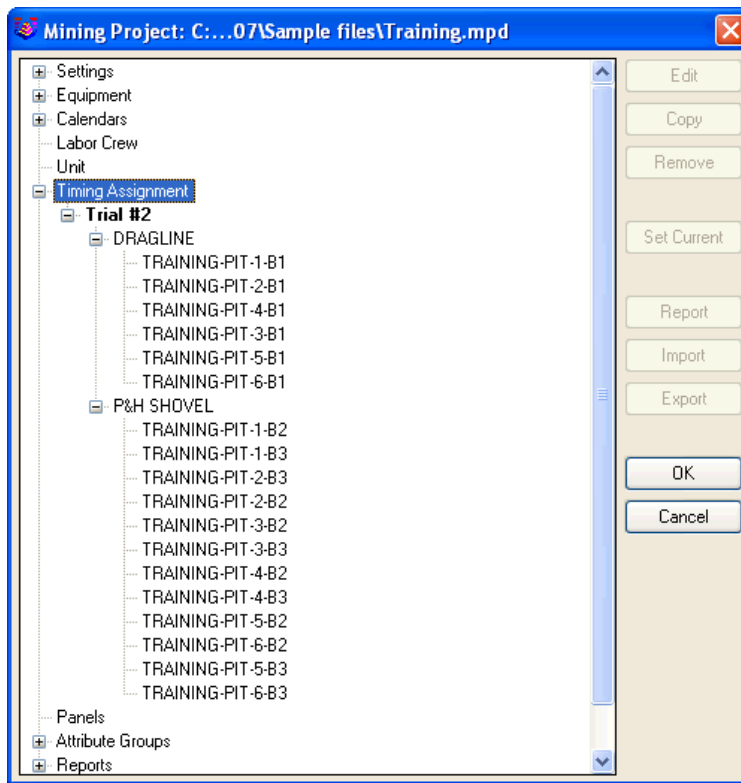
Unit	Month	Year	Operating days	Days	Weekdays Worked	Saturdays Worked	Sundays Worked	Calendar days	Work shifts	Hours Utilized	Scheduled hours
D11 DOZER	January	2008	31	31	23	4	4	31.0	93.0	0.00	743.
D11 DOZER	February	2008	29	29	21	4	4	29.0	87.0	0.00	695.
D11 DOZER	March	2008	31	31	21	5	5	31.0	93.0	0.00	743.
D11 DOZER	April	2008	30	30	22	4	4	30.0	90.0	0.00	719.
D11 DOZER	May	2008	31	31	22	5	4	31.0	93.0	0.00	743.
D11 DOZER	June	2008	30	30	21	4	5	30.0	90.0	0.00	719.
D11 DOZER	July	2008	31	31	23	4	4	31.0	93.0	0.00	743.
D11 DOZER	August	2008	31	31	21	5	5	31.0	93.0	0.00	743.
D11 DOZER	September	2008	30	30	22	4	4	30.0	90.0	0.00	719.
D11 DOZER	October	2008	31	31	23	4	4	31.0	93.0	0.00	743.
D11 DOZER	November	2008	30	30	20	5	5	30.0	90.0	0.00	719.
D11 DOZER	December	2008	31	31	23	4	4	31.0	93.0	0.00	743.
MARION DRAGLINE	January	2008	31	31	23	4	4	31.0	93.0	0.00	743.
MARION DRAGLINE	February	2008	29	29	21	4	4	29.0	87.0	0.00	695.
MARION DRAGLINE	March	2008	31	31	21	5	5	31.0	93.0	0.00	743.
MARION DRAGLINE	April	2008	30	30	22	4	4	30.0	90.0	0.00	719.
MARION DRAGLINE	May	2008	31	31	22	5	4	31.0	93.0	0.00	743.
MARION DRAGLINE	June	2008	30	30	21	4	5	30.0	90.0	0.00	719.
MARION DRAGLINE	July	2008	31	31	23	4	4	31.0	93.0	0.00	743.
MARION DRAGLINE	August	2008	31	31	21	5	5	31.0	93.0	0.00	743.
MARION DRAGLINE	September	2008	30	30	22	4	4	30.0	90.0	0.00	719.
MARION DRAGLINE	October	2008	31	31	23	4	4	31.0	93.0	0.00	743.
MARION DRAGLINE	November	2008	30	30	20	5	5	30.0	90.0	0.00	719.
MARION DRAGLINE	December	2008	30	30	22	4	4	31.0	90.0	0.00	719.
PH SHOVEL	January	2008	31	31	23	4	4	31.0	93.0	0.00	743.
PH SHOVEL	February	2008	29	29	21	4	4	29.0	87.0	0.00	695.

- **SaveAs/Load:** This saves the calendar, optionally as a new name. Load will open a new one, and give the option to append to the existing, open calendar.



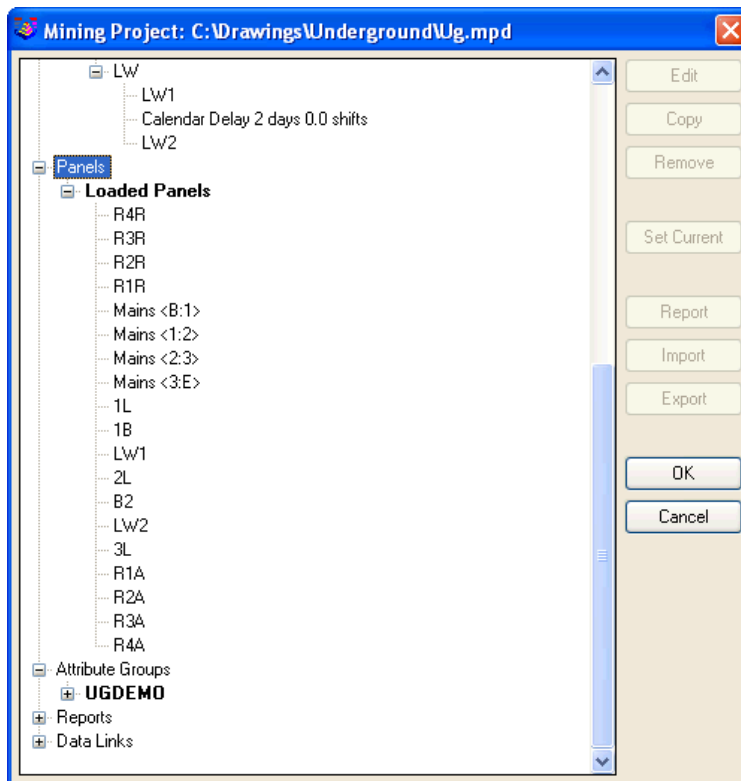
Timing Assignment

The Timing Assignment allows for setting the current sequence. The Assignment is similar to the TIM file in the timing routines. The equipment used and which panels or pits they are mining are shown. The Set Current button controls which one will be loaded and used. Shown here is both a surface mine and an underground mine example.



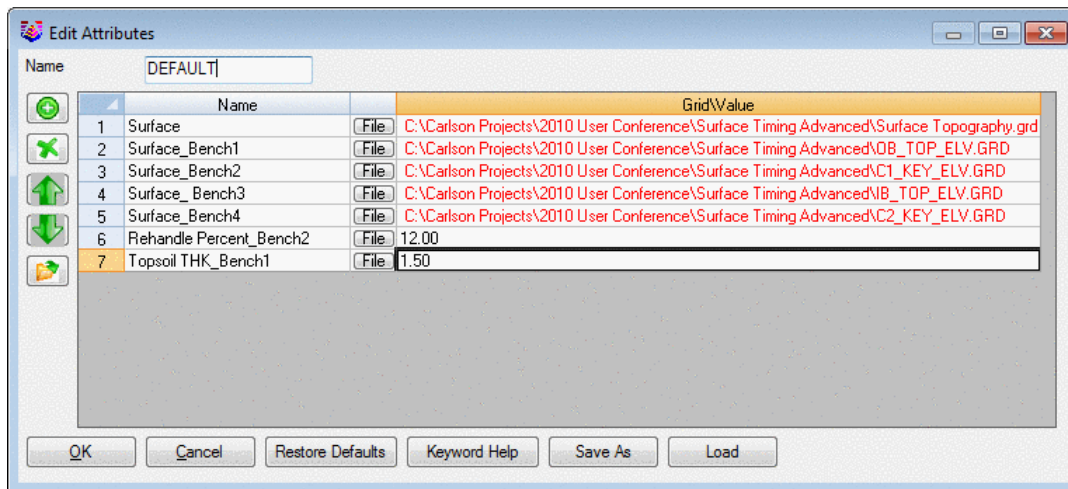
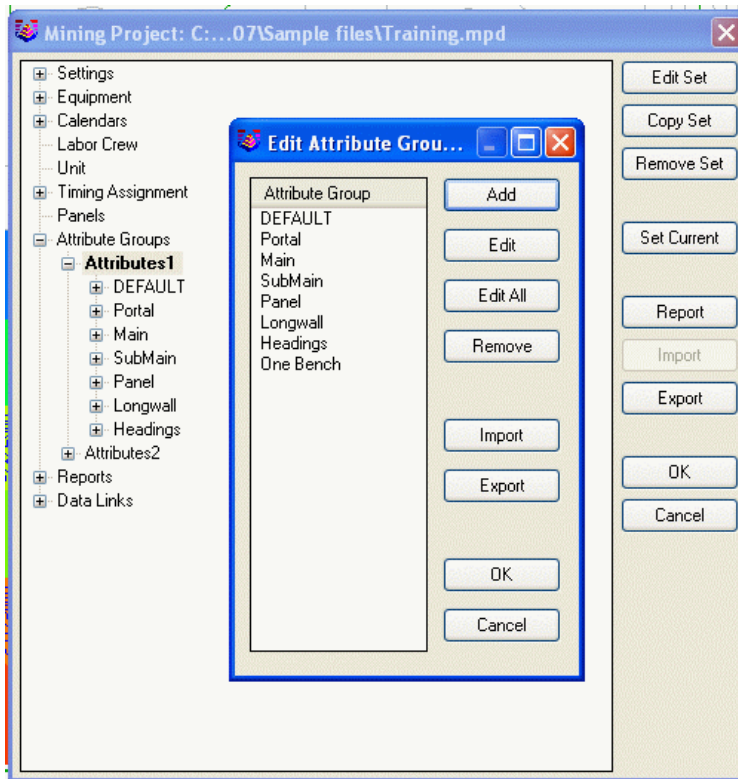
Panels

This displays the selected underground panels in the mine plan assignment. An example is shown here.

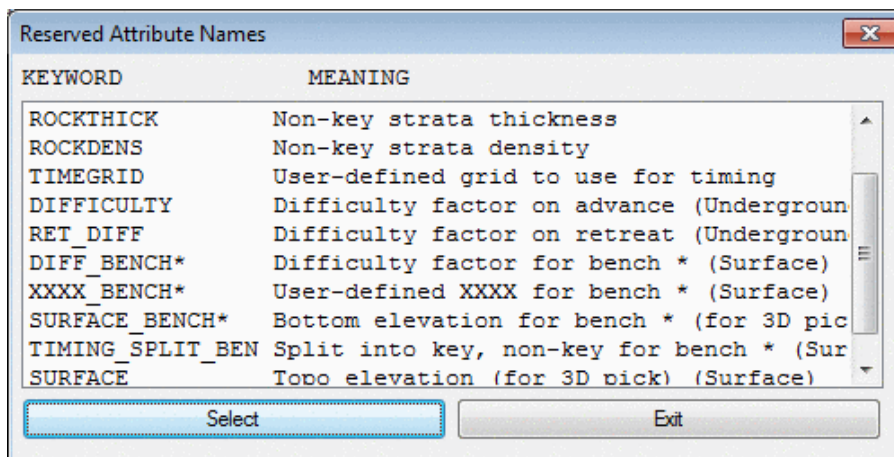


Attribute Groups

Attribute groups are Pit or Panel Attributes assigned to the pits and panels for timing. When timing the pits or panels, there are additional attributes that will be calculated and reported in addition to the Non-Key and Key quantities. If these are defaults that should be applied to all the panels or pits, then they should be entered here. Any quality values, density, or difficulty attributes can be defined here either just as a value, or as a grid file with varying values. The Reserved Attribute Names window displays the reserved words that will be recognized in the timing routines, and how they should be entered in. Selecting Keyword Help brings up the Reserved Attribute Names. If the grid name is red, as shown here, then the file cannot be found. They need to be colored black for successful use.



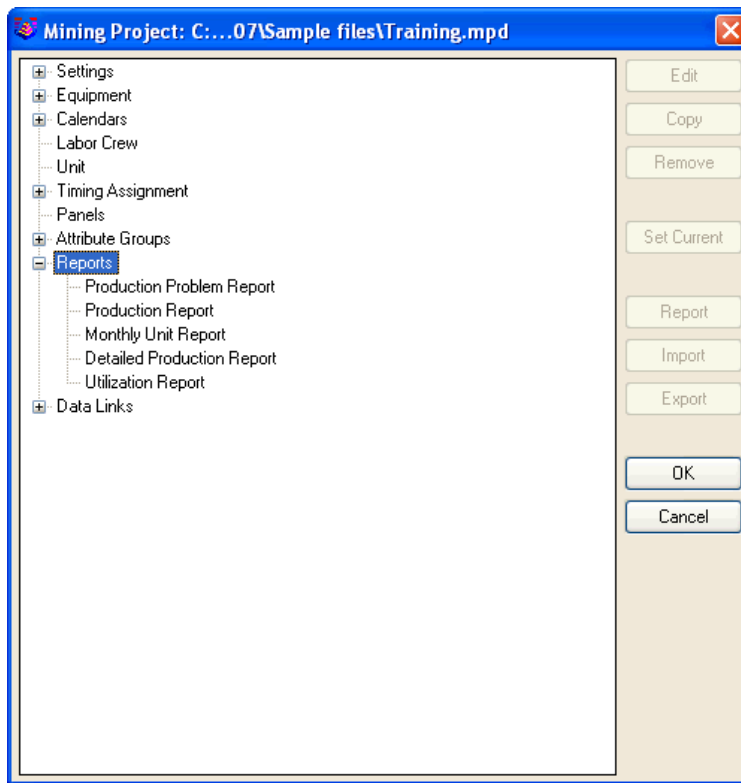
The Reserved Attribute Names are defined in a little more detail here:



- **THICKNESS:** Key strata thickness: Will report any other thickness values, this is used mostly for underground timing.
- **DENSITY:** Key strata density: Will report Key density, is used mostly in underground timing to calculate the tons. In surface timing, the tons are already in the pits.
- **ROCKTHICK:** Non-key strata thickness: Reports the Non-Key thickness mostly for underground timing.
- **ROCKDENS:** Non-key strata density: Reports the Non-Key density mostly for underground timing.
- **TIMEGRID:** User-defined grid to use for timing: This will be any additional grid the user would like to add for reporting.
- **DIFICULTY:** Difficulty factor on advance just for underground timing: This will alter the underground equipment rate on advance.
- **RET_DIFF:** Difficulty factor on retreat for underground timing: This will alter the underground equipment rate on retreat.
- **DIFF_BENCH*:** Difficulty factor for bench * for surface timing: This will speed up or slow down the equipment as it mines the specified bench. If a value is above 1, such as 1.2, then it will mine 20% slower at that point in the grid, or everywhere if it is set to value. If it is less than 1, then it will mine faster at that point. Difficulty is calculated by the equation: $DIFF=100/(100-\%reduction)$.
- **XXX_BENCH*:** This is the dominate attribute that is most widely used in this command for surface pits. All quality grids will be defined in this fashion, for each bench. The example above, BTU_BENCH1, is defined in this way. All quality parameters need to be entered as the NAME_BENCH*.
- **SURFACE:** This attribute is used for the 3D Pick window to drape the pit blocks onto the ground surface.
- **SURFACE_BENCH*:** This attribute defines the bottom elevation grid for each bench. It is used to display the blocks in the 3D Pick window properly, so the benches are represented accurately.
- **TIMING_SPLIT:** This will globally split all blocks that contain both Key and Nonkey material into two blocks that may be scheduled separately, by either the same unit, or different units. The split is into two portions - key and non-key. Entering a non-zero value will cause a split. Additionally three values are supported as pertains to precedence. Enter in one of these values in for the attribute to split it.
 1 = Both OB and KEY needs to be completed to satisfy precedence
 2 = OB portion needs to be completed to satisfy precedence
 3 = KEY portion needs to be completed to satisfy precedence
- **TIMING_SPLIT_BENCH*:** This attribute is the same as TIMING_SPLIT, but is only applied to the specified bench number.

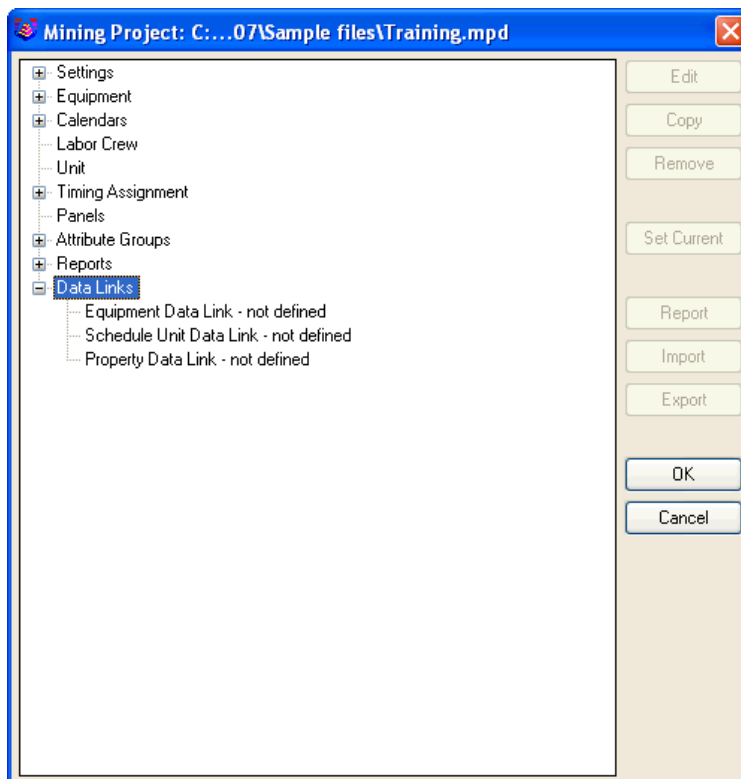
Reports

Some of the reports generated in the scheduling will appear here for other options to report. These are also accessed in the calendar or after a schedule is run.



Data Links

This shows the link of the data when it is linked to an external source for reporting.



Skip Past Report Archiving in Project Database: By default, all reports generated with timing commands will be saved in a backup of the Timing Project Manager as an .mrd file (the file name will be the same as the .mpd file,

just with a different extension). If many reports are generated, this archive can become quite large, and may even slow down processing time when running the timing command. When this option is enabled, no reports will be archived, preventing potential slow-down from large .mrd files. It is recommended to enable this option when you have no need for old timing reports.

Check for Conflicts: When enabled, this option will check for any conflicts in the pits. If a conflict is detected, you will be prompted to display a report of the conflicts when you start the Surface Equipment Timing command.

Conflict checks include:

1. Pits with duplicate names
2. Empty pits (no tonnage/volume in the pit)
3. Pits with invalid direction assignment
4. Pits with invalid precedence (can occur if a precedence rule is assigned, but the preceding pit is deleted or not selected for timing). For example, If Pit A must be mined prior to Pit B, but Pit B is later deleted/not selected for timing.
5. Pits outside the bounds of Attribute Group Grids

Pull-Down Menu Location: Reserves/Timing in Surface Mining and Underground in Underground Mining

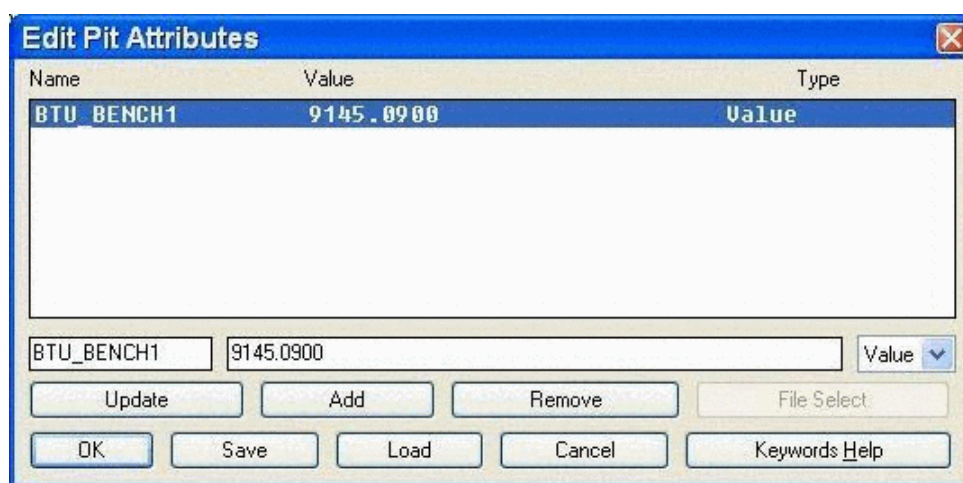
Keyboard Command: stime_project or utime_project

Set Attribute By Grid File

This command assigns an attribute grid value such as thickness or BTU to a panel or pit area. The program prompts for the attribute name and the grid file that represents the attribute. Then multiple panel/pit perimeter polylines can be selected and the program calculates the average grid value in each panel/pit polyline and stores this value. This command works similar for both surface pits or underground panels. This is the surface pit description.

Another method for using grids is to define the attribute as the actual grid file name instead of the average value. Then the timing routines will calculate attribute values for each timing block from the grid value. The advantage to using Set Attribute by Grid File instead of using the actual grid file is speed. Timing runs a bit slower when reading the grids.

The average value is stored in each pit and can be verified with the Edit Pit command. If the attribute exists there already, it will not overwrite it. Choose the Attributes button and the attribute should be there, as shown in the Edit Pit window. It matches the first BTU value calculated in the example.



Prompts

Type of polylines [Underground/<Surface>]? S

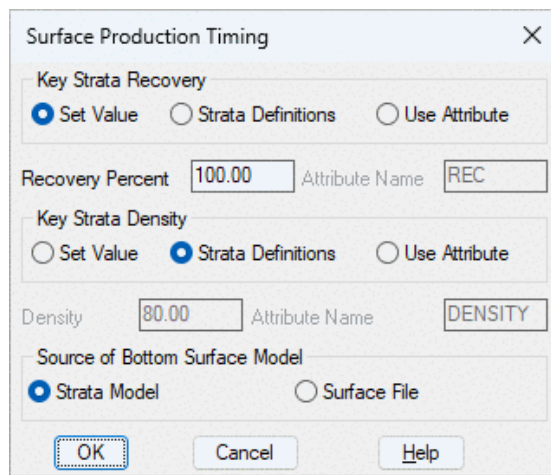
Pit bench number <1>: 1
Attribute Name: BTU
Reading cell> 194032
Pass> 1 Null Z values left> 0
Select pit polylines.
Select objects: 3 found, 3 total
Select objects:
Pre-processing grid cells
Processing cells ...
Average BTU = 9145.09
Pre-processing grid cells
Processing cells ...
Average BTU = 9145.92
Pre-processing grid cells
Processing cells ...
Average BTU = 9149.74
Pulldown Menu Location: AdvMine
Keyboard Command: gridattr

Surface Production Timing

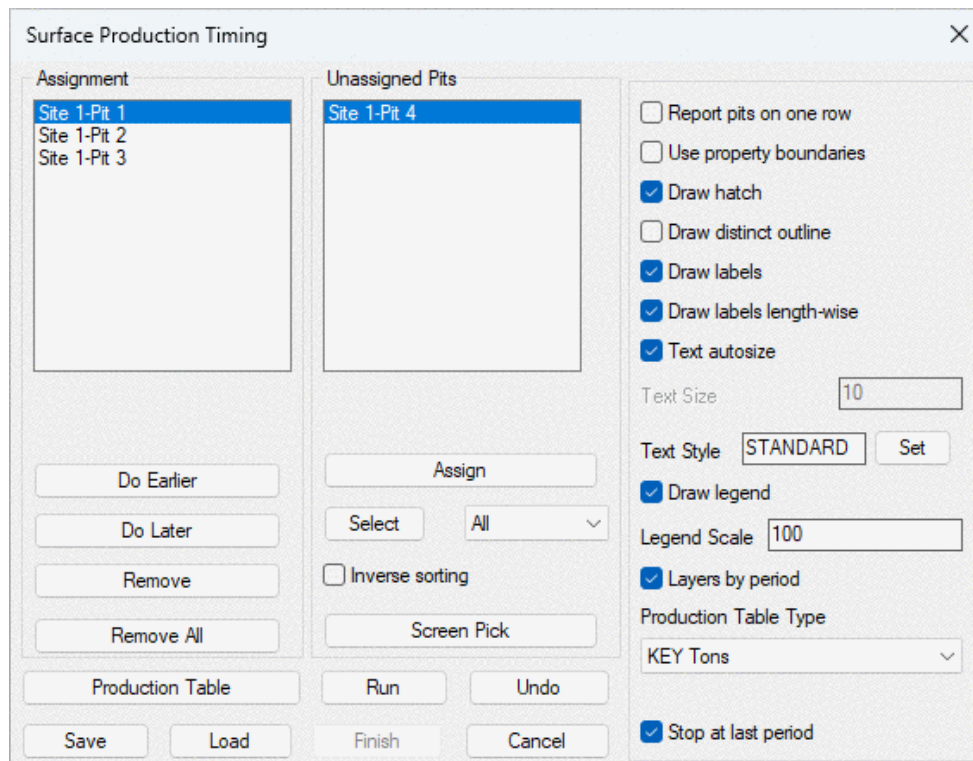
This command is used for production based timing of a mineplan to see the mine progress when defining production by time period. It is sometimes referred to a pre-scheduler, prior to running Surface Equipment Timing to get an estimate of mine progression. There are 3 steps needed to prepare for Production Timing as shown in the flow chart below. First, the mining model needs to be set up in the Geologic Model file (PRE). This could also be the geologic model. Next, there must be named, Carlson pit polylines representing the mine plan. Finally, the pits lines must have direction assigned for mining. Once these 3 items are created, the command may be run. This command does not use highwall angles or laybacks. It loads the mining model and vertically intersects it with the directioned pits to obtain timing blocks.

Make Geologic Model File —> Create Named Carlson Pits —> Assign Directions to Pits —> Surface Production Timing

The first step is to choose the Geologic Model file to process. After it is selected, the first dialog box appears with some initial settings. The Key Strata Recovery can be entered in the window with Set Value, or can be set By Strata Definitions, where it will refer to the Strata Definition File created with the Define Strata command, or the recovery can be an attribute grid surface found in the geologic model, with the attribute name of REC, which may be customized. The grid would need to contain values from 0-100 for recovery. Each strata can have it's own recovery with the second two options. The Key Strata Density is set in the same way. Either by entering it here, or as defined in the SDF file. The units are either pounds/cubic foot or kg/cubic meter, depending on US units or metric units in the drawing. Finally, the Source of Bottom Surface Model can be set here. If Strata Model is chosen, then the routine will use all seams from the surface topography down to the lowest strata surface in the Geologic Model file. If Surface File is selected, then volumes will only be calculated from the topography down to that bottom surface grid or TIN file, which could be a flat bench elevation for example, or a complex benched pit with many levels. A file-select window will appear next, to choose the grid or TIN if that option is used.



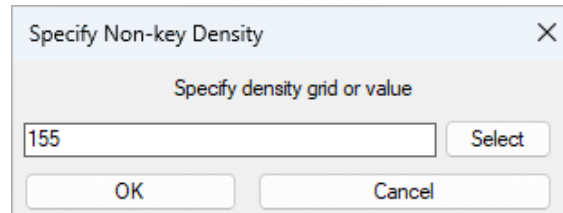
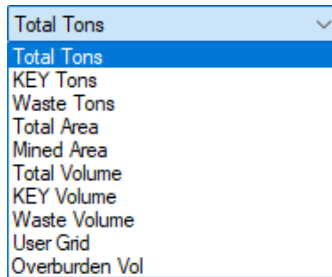
Next, after selecting the pits and the mining project file, the Surface Production Timing window appears as follows.



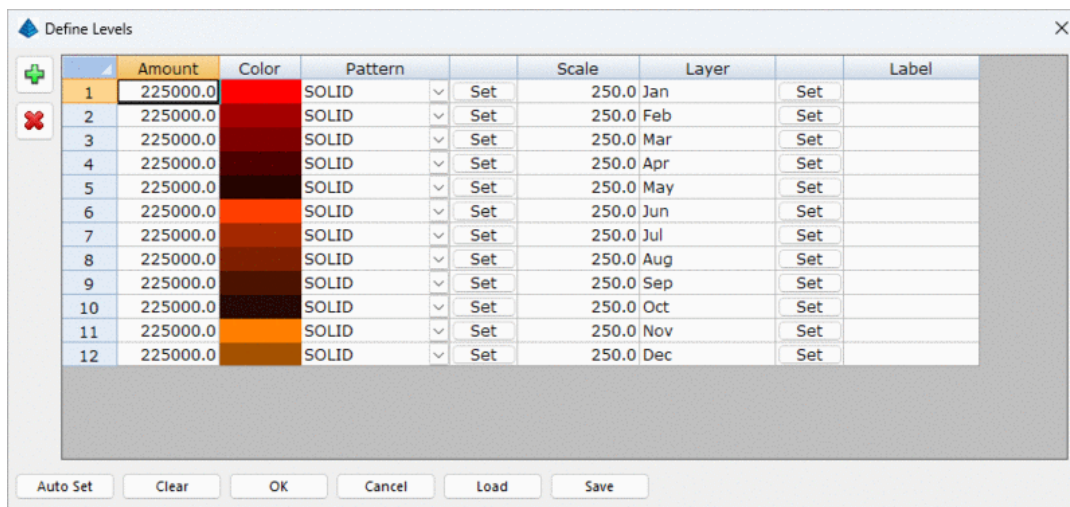
The first step is to select pits and move them over to the Assignment box on the left with the Assign button. Following is a detailed description of each item in the dialog box.

- **Do Earlier / Do Later:** These buttons move the pits up or down in the Assignment window.
- **Remove, Remove All:** These buttons will move either the selected pit, or all pits from the Assignment window to the Unassigned Pits window
- **Assign:** This button will move the highlighted pits from the Unassigned window to the Assignment window.
- **Select All:** This is a quick way to highlight all pits in the Unassigned Pits.
- **Inverse sorting:** If this is checked, then the pits will appear in reverse order in the Unassigned Pits window, otherwise they are in alphabetical and numeric order.

- **Screen Pick:** Allows for manual selection of pits in plan view with a cross-hairs. Simply place the cursor in the pits and pick in the order to mine. Hit enter when done. The selected pits should appear in the Assignment window in the order they were picked.
- **Report pits on one row:** This option is for report formatting. If this is checked, the report will have one row per time period and all strata quantities and qualities will follow on that row, each in their own column. If this is not checked, then each strata will be in its own row in the report, with its quality to follow. This option is usually not selected.
- **Use property boundaries:** There must be named property boundaries on screen for this option. Production timing will automatically detect them and break out the production by owner and property if desired.
- **Draw hatch:** One of the main items of output from this command. It draws the periods as blocks of solid fill or any hatch pattern inside the pits in plan view. They are colored by period. As they are being drawn, the direction and progress of the mineplan can be seen for review.
- **Draw distinct outline:** This is very similar to the timing blocks, except that they are just closed polylines with no fill or hatch. They are in their own color, and optionally, layer. They can be used later for further reserve calculations, or to save the mineplan without hatch or fill.
- **Draw labels:** This option places the period name in the block or outline. It is either the user defined label, or will just use "Period1", "Period2" etc.
- **Draw labels length-wise:** Selecting this option orients the text lengthwise to the long axis of the block outline. Otherwise it will be placed horizontally.
- **Text autosize:** If this is selected, the text will be automatically sized to fit the size of the block outline. Otherwise, the text might be too large for some of the smaller blocks.
- **Text Size:** Enter in the text size for labels. It will use this if Text Autosize is not selected.
- **Text Style:** Enter the AutoCAD Text Style for labels and legend.
- **Draw Legend:** Select this option to draw a legend of the timing blocks showing the color and the name of the period. It will ask to pick the legend position.
- **Legend Scale:** Enter in a legend scale size to size the legend.
- **Layers by Period:** This option will create a new layer for each time period and draw the blocks and outlines in those layers.
- **Production Table Type:** This is a very important setting that needs to be selected for the target material. There are 10 choices to use for production targeting. Most common are Key Tons and Overburden Volume, but others are available. Total Tons converts all strata to tonnages and mines them accordingly. The NonKey density is set in a window that appears as soon as Run is selected. Key Tons are tons of the combined key material, which name is set in Configure /Mining. Waste Tons will combine all NonKey strata, calculate the tons for it and target that tonnage for production. Mined Area will target a defined area in square feet or meters for production. Total Volume will combine all strata, both Key and NonKey and target the total CY or CM. Key Volume or Coal Volume will target the CY or CM of all Key strata in the selected Geologic Model file. Waste Volume combines all NonKey strata and targets the CY or CM of them. User Grid will take any grid and target total quantities for production. An example of this would be a power plant that wants to target total BTU. The grid to select would be total BTU/area. Finally, the Overburden Volume will take just the first NonKey strata in the Geologic Model and target that, but still report any additional NonKey seams below it.



- **Stop at Last Period:** If this is checked, then the routine will stop after the last period entered in the Production Table. If it is not checked, then it will continue with the last target amount all the way through the last selected pit polyline.
- **Production Table:** This is the screen to set the production amount and time period or date. The first column is the amount to target. This is what you have selected under Production Table Type. The second column is the color for that period. The next column is the hatch pattern for the block. If it is a hatch pattern other than SOLID, then it must have a scale factor, set in the next column. the AutoCAD Layer is set in the next column and finally, the Label is set in the last column. This can be anything from dates, to owners and areas. If the Labels are left blank, then it will fill in labels such as Period3, Period4, or month or year, etc. The Clear button will clear the entire screen. There are Save and Load buttons for easy retrieval of the CQT files.



- **Auto-set:** This button will open the below dialog, which can be used to automatically populate the table with

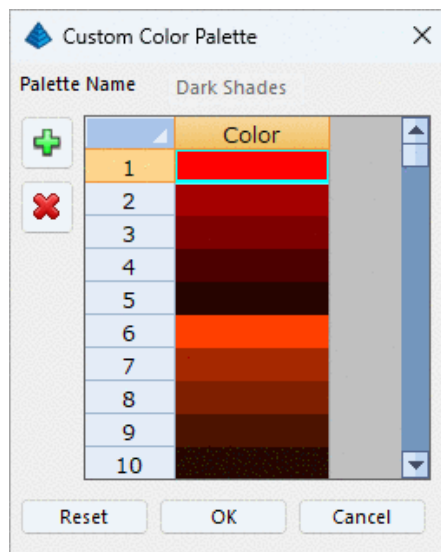
values.

The screenshot shows the 'Auto-Set Amounts/Colors' dialog box. It features the following fields and options:

- Starting Line:** 1
- Starting Amount:** 225000.000
- Amount Interval:** 0.000
- Hatch Scale:** 250.00
- Hatch Pattern:** SOLID
- Set Layer:** (checked)
- Layer (top):** Single Layer
- Number of Rows to Create:** 12
- Layer (bottom):** MINETM
- Color Options:**
 - Set Colors:** (checked)
 - Color Palette:** Dark Shades (selected), with Add, Edit, and Remove buttons.
 - Color Increment:** (unselected), with Increment: 10 and Start Color: 10.

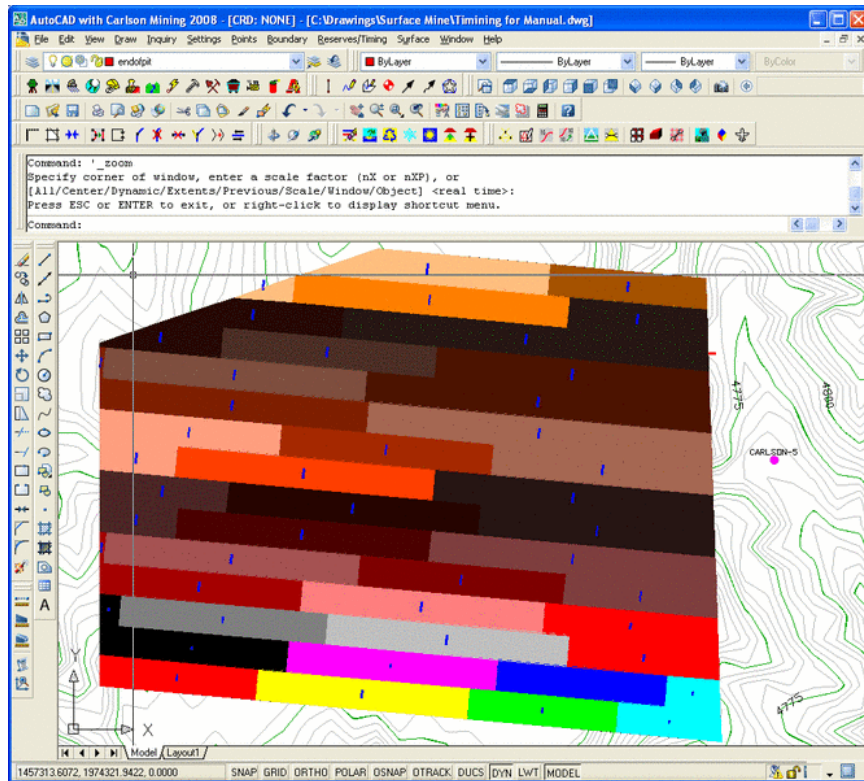
Buttons: OK, Cancel

- **Starting Line:** This value will set the row to begin the automatic population of the Custom Date table.
- **Starting Amount:** This is the initial amount used in the table.
- **Amount Interval:** This value will be added to the Starting Amount in each additional row. If the Starting Amount needs to be applied to all entries in the table, you can set this value to zero.
- **Hatch Scale:** This value will set the hatch scale for the new periods.
- **Hatch Pattern:** This field will set the hatch pattern for the new periods. The ellipsis button will open the list of available hatch patterns.
- **Set Layer:** This option will overwrite the Layer names when enabled.
- **Layer by:** This drop list determines how the layers will be populated. If the *Single Layer* option is selected, all layers will be the same, according to the **Layer** text box.
- **Set Colors:** This option will set custom colors for the new periods. There are two options for setting colors - by using a Color Palette, or by using a Color Increment.
- **Color Palette:** This option will use a predefined color palette to color the periods. When the end of the palette is reached, the colors will start over. Palettes may be added, edited or removed using the corresponding buttons. An example color palette is shown below. Colors may be added or removed from the palette by clicking the Plus or X icons, and colors may be modified by double-clicking them. Any of the predefined palettes may not have their name modified. If you modify the colors in a predefined palette, you may click the **Reset** button to return to the default colors.



- **Color Increment:** This option will color the periods by using a **Start Color** and a **Color Increment**. All colors in the CAD color palette range from 1 to 255. You may want to inspect the color palette when using this option to relate colors back to their numeric value. For example, a Start Color of 10 and a Color Increment 10 will increment through colors 10, 20, 30, etc.
- **Undo:** This will undo a previous run, removing the colored blocks and outlines.
- **Run:** This is the button to start the actual timing and sequencing of the pits. After the Production Table and all the settings are good, then choose Run.
- **Finish:** After the blocks are drawn and the Surface Production Timing window comes back, selecting the Finish button will start the calculations and generate the quantities and qualities in the Report Formatter. Here items are selected to appear in the report and exported to a file, such as Excel or Access.

A finished map with the blocks drawn on it and the corresponding report are shown here. In this example, the amount was set to 225,000, then 250,000 tons. In the report, notice the accuracy of the tons. Most are within a couple of hundred tons.



Surface Mine Reserves

Pre-Calc Definition: C:\Drawings\Surface Tutorial\MODEL.pre
 Key strata percent recovery: 100.00%
 Minimum depth to use: 0.00 ft
 Minimum key thickness to use: 0.00 ft
 Minimum minable parting thickness to use: 0.00 ft
 Non-Key thickness Above key to add to key: 0.00 ft
 Non-Key thickness Below key to add to key: 0.00 ft

Period:	Strata	Coal Tons	Waste C.Y.	Density	Acres	Thick	MinThick	MaxThick	Strip ratio	BTU
January	OVERBURDEN		1,050,798.9		4.716	138.12	125.02	155.22		
	C1	97,578.2		80.00	4.716	11.88	10.60	12.99	10.8	9,317.107
	PARTING		130,149.4		4.716	17.11	15.78	18.24		
	C2	127,629.8		80.00	4.716	15.53	15.01	16.06	1.0	8,181.919
---- Total for Period: January ----		225,208.0	1,180,948.3	80.00		182.63	166.41	202.51	5.2	8,673.773
February	OVERBURDEN		719,373.1		4.450	100.20	2.02	128.85		
	C1	107,218.6		80.00	4.649	13.24	12.76	14.47	6.7	9,251.266
	PARTING		136,755.9		4.929	17.20	17.26	18.12		
	C2	146,425.0		80.00	6.369	13.20	15.10	17.35	0.9	8,162.589
---- Total for Period: February ----		253,643.6	856,128.9	80.00		143.83	47.13	178.79	3.4	8,622.788
March	OVERBURDEN		965,368.4		4.465	134.03	124.46	144.49		
	C1	111,460.0		80.00	4.465	14.33	13.12	14.96	8.7	9,019.724
	PARTING		125,034.3		4.465	17.47	17.02	18.58		
	C2	138,597.2		80.00	4.465	17.82	16.65	18.51	0.9	8,092.148
---- Total for Period: March ----		250,057.2	1,091,202.6	80.00		183.64	171.25	196.53	4.4	8,505.604
April	OVERBURDEN		833,533.3		4.741	108.98	90.15	135.40		
	C1	100,130.7		80.00	4.741	12.12	9.73	13.52	8.3	9,093.651
	PARTING		145,877.8		4.741	19.07	17.97	21.69		
	C2	149,958.4		80.00	4.741	18.15	14.39	19.15	1.0	8,186.345
---- Total for Period: April ----		250,089.1	979,411.1	80.00		158.34	132.24	189.77	3.9	8,549.612
May	Strata	Coal Tons	Waste C.Y.	Density	Acres	Thick	MinThick	MaxThick	Strip ratio	BTU

Pulldown Menu Location: Surface Mine Module - Reserves/Timing

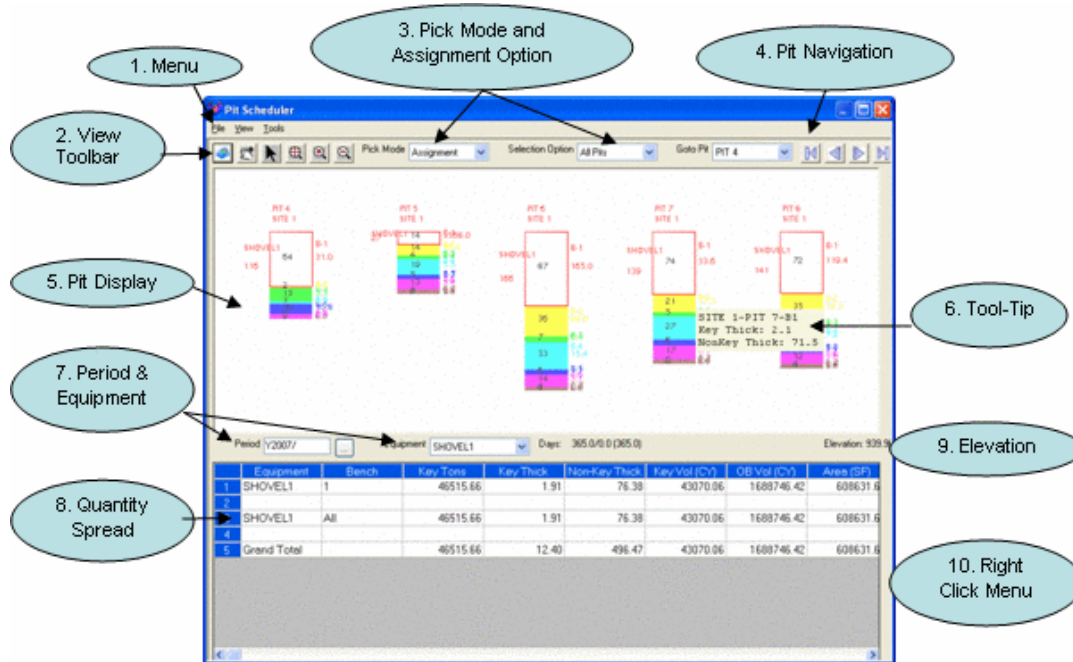
Keyboard Command: calcplan

Prerequisite: Timing pit polylines

Pit Scheduler

Pit Scheduler is a short-range surface mine scheduling command and is located under Reserves/Timing dropdown menu in Carlson Surface Mining. It displays selected pits in a cross section view with benches and allows user to assign equipments to pit benches by time period. The planner can simply pick on a bench to mine/un-mine it and the updated quantities will appear in the quantity spreadsheet. These quantities can be reported using report formatter at any time.

Interface Components

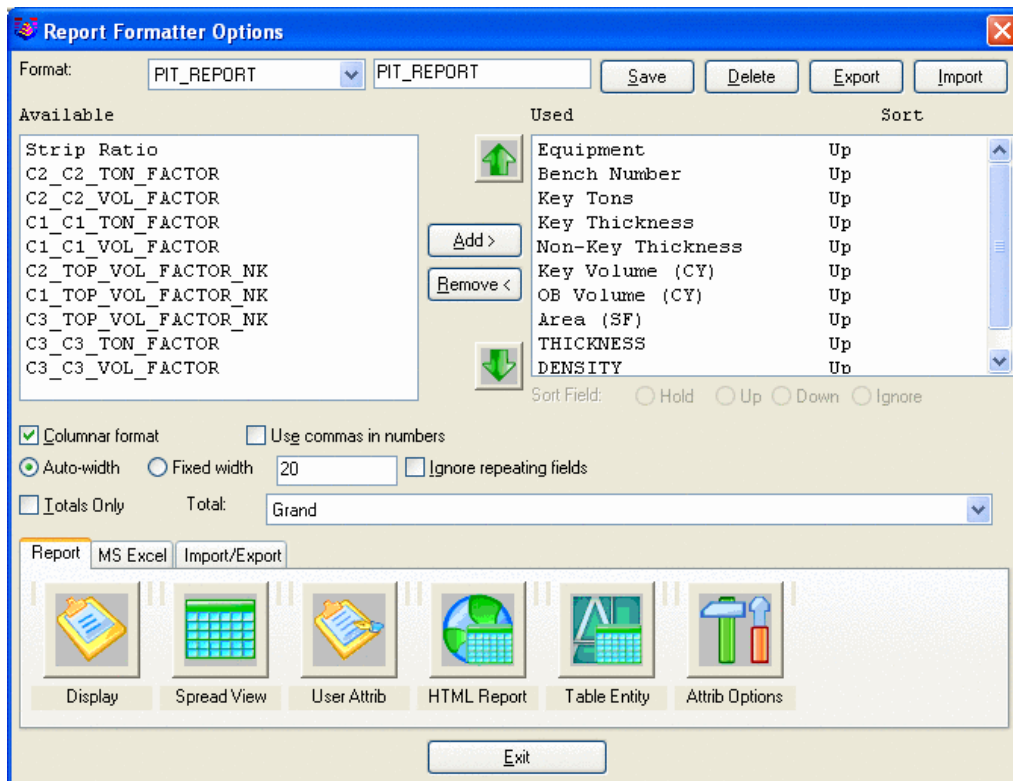


1. Menu

The Pit Scheduler has three main dropdown menus.

a. File Menu

- i. Read Pit Blocks: This command can be used to read in the new pits in the view to work on.
- ii. Report: This command can be used to report the currently assigned quantities.



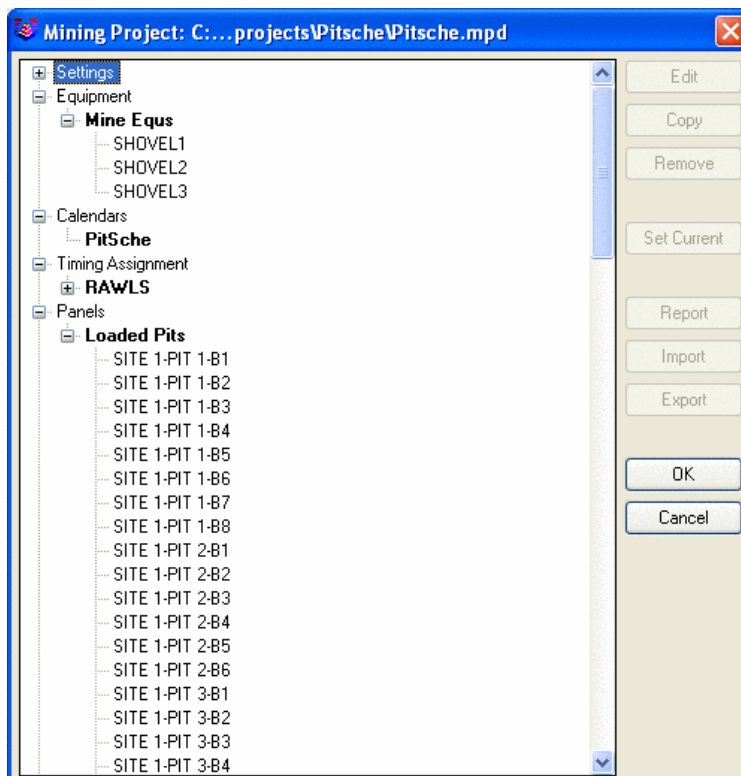
iii. Export to Bitmap: This command can be used to save the current pit display in to a bitmap file.

iv. Save and Exit: This command saves all the assignment and exits from the Pit Scheduler.

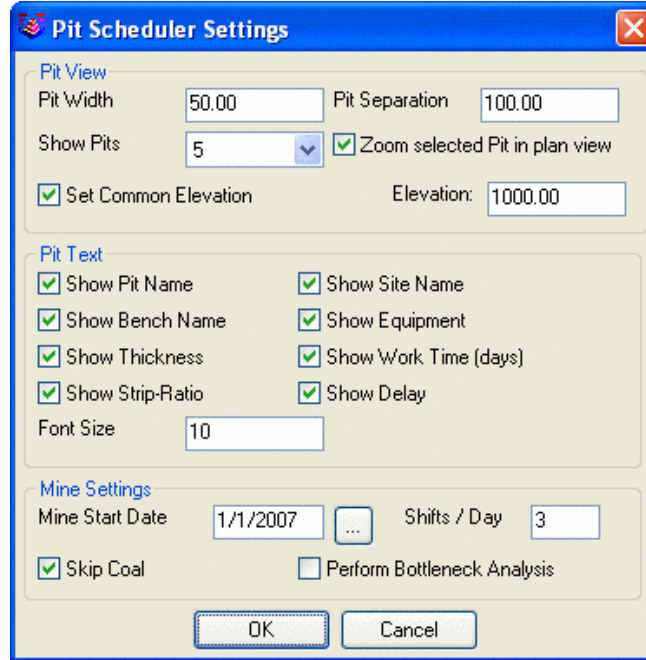
v. Exit: This will exit from the Pit Scheduler without saving any assignments.

b. View Menu

i. Project Manager: This command brings up the Surface Project Manager that can be used to define equipments, timing calendar, and other components to be used in scheduling and to view assignments and loaded pit names.



ii. Settings: This command shows the settings for Pit View, Pit Text and Mining.



- Pit View: These settings can be used to modify the pit display to view it more efficiently. The "Pit Width" and "Pit Separation" can be set for better viewing the pits based on their thickness. The pit width is the width of the pit column and pit separation is the separation between two pit columns. The "Show Pit" allows user specify the number of pits to view in current view. If "Zoom selected Pit in plan view" is on as soon as used clicks on a pit it will be highlighted in the AutoCAD drawing plan view and the view will be zoomed in to current pit. The "Set Common Elevation" option allows pits to be hanged from a common elevation specified in the "Elevation" edit box, if this option is turned off the program looks for the "SURFACE" attribute for the pit and reads the elevation from there.

- Pit Text: The Pit Text settings can be used to display different text in the pit display for a pit column.

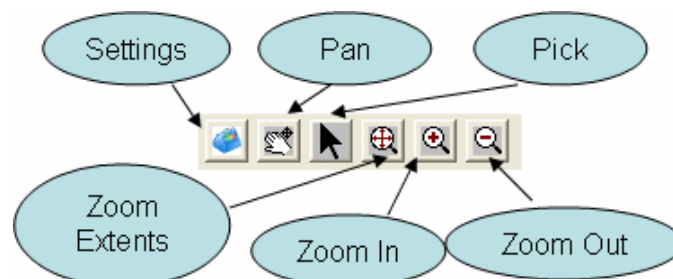
- Mine Settings: These settings are used in timing the pits similar to used in Equipment Timing.

c. Tools Menu

i. Clear Assignments: This command clears all the assignments for the current equipment selected in "Equipment" dropdown list. This will allow user to select the assignments for equipment from the beginning.

ii. Clear All Assignments: This command will clear all the assignments for all the equipments and bring the mine to the starting point, and allows user to start the scheduling from the beginning.

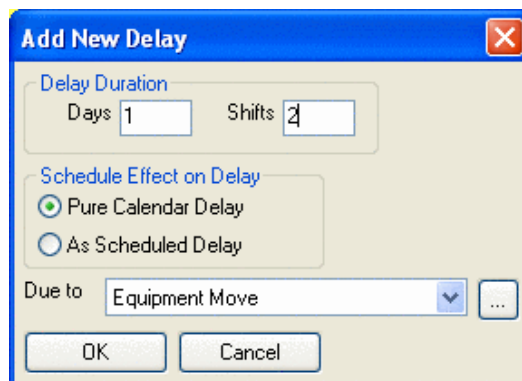
2. View Toolbar



3. Pick Mode and Selection Option

a. Pick Mode: is used to set the mode for selection of pit benches. There are three pick modes:

- i. Assignment: This mode is used to assign the current equipment selected in the "Equipment" dropdown list to the selected bench/benches based on the "Selection Option".
- ii. Waste Bench: This mode is used to specify whether a bench will be mined or treated as waste bench. The waste bench quantities will not be added to the quantities as it will not be mined.
- iii. Add Delay: This mode allows user to add delays associated with a bench. If this mode is selected than based on the selection option delay will be added to the bench/benches and mining will be delayed by delay duration. (Look in Surface Equipment Timing for more information on adding delays)

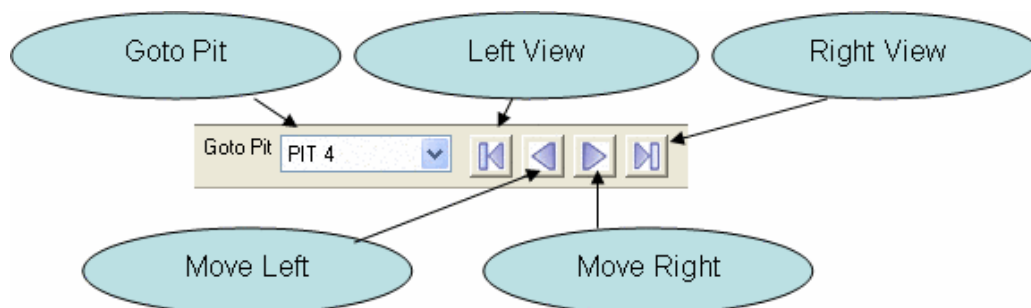


b. Selection Option: is used to apply the pick mode to the benches. There are five selection options user can choose from.

- i. Individual Pit: the pick mode will be applied to the selected bench in selected pit.
- ii. Current View: the pick mode will be applied to the selected bench for all the pits in the current view.
- iii. All Pits: the pick mode will be applied to the selected bench for all the pits loaded in the Pit Scheduler.
- iv. Pits to the Right: the pick mode will be applied to the selected bench for selected pit and to all the pits on right side.
- v. Pits to the Left: the pick mode will be applied to the selected bench for selected pit and to all the pits on left side.

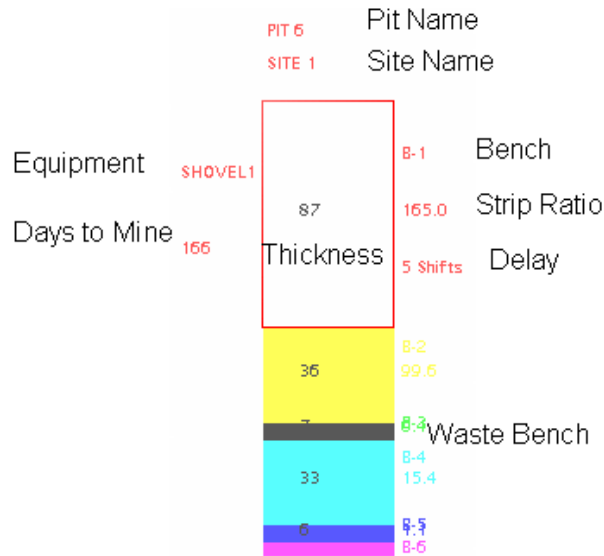
4. Pit Navigation

The Pit Navigation can be used to scroll through the loaded pits. The "Goto" pit allows user to select the starting pit for the view. The Left or Right View scrolls the view to left or right respectively by the number of pits selected in "Show Pits" in the Settings dialog under View Menu. The Move Left or Right scrolls the view by a single pit.



5. Pit Display

The Pit Display shows the pits with benches in a plan view based on the view and text settings. Following figure shows a typical Pit with all text options turned on.



6. Tool-Tip

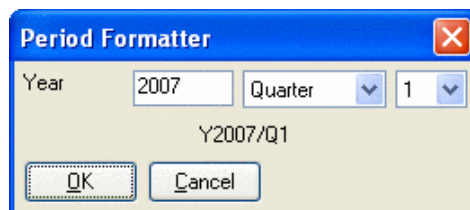
A Tool-Tip window is shown with the full name of bench (Site-Pit-Bench) and Key and Non-Key thickness when user cursor over a bench.

```
SITE 1-PIT 6-B1
Key Thick: 0.5
NonKey Thick: 86.0
```

Period: Y2007/Q1 ... Equipment: SHOVEL2 Days: 69.0/21.0 (90.0)

7. Period & Equipment

a. Period: can be used to view the number of days used and available for the selected equipment. The Period string can be formatted using the Period Formatter dialog or can directly be typed in the edit box. The Period string must be in the specified format. As soon as the user updates the Period and hit enter the "Days:" label is updated to show "Used Days/Available Days (Period Days)".



b. Equipment: This dropdown lists all the defined equipments in the Project Manager. The current selected equipment is used for assignments.

8. Quantity Spread

The quantity spread shows the quantities and qualities for the equipment selected in "Equipment" dropdown for each bench it is assigned to. The spread shows Equipment, Bench, Key-Tons, Key-Thickness, Non-Key Thickness, Key Volume, OB Volume, Area, Acres, and Strip Ratio columns followed by the pit attribute values. It is important to note that the Strip Ratio will always be reported as units of nonkey volume to key tonnage (yd³/ton or m³/tonne).

	Equipment	Bench	Key Tons	Key Thick	Non-Key Thick	Key Vol (CY)	OB Vol (CY)	Area (SF)	Coal Acres	Strip Ratio
1	SHOVEL1	1	46515.66	1.91	76.38	43070.06	1688746.42	608631.60	13.97	39.21
2	SHOVEL1	2	35676.67	1.46	21.73	33033.95	470988.28	608631.60	13.97	14.26
3										
4	SHOVEL1	All	82192.33	1.68	49.05	76104.01	2159734.70	1217263.19	27.94	28.36
5										
6	Grand Total		82192.33	21.88	637.69	76104.01	2159734.70	1217263.19	27.94	28.36

9. Elevation

The current elevation of cursor is show on the "Elevation" label. If the "SURFACE" attribute is defined for the pit it will show the actual elevation for that pit and its benches.

10. Right Click Menu

A Right Click menu pops-up when user right click on a Pit Column. This menu lists Edit Pit followed by the full name of benches in that pit. The thickness of a bench can be very small that it becomes hard to pick that bench using left click, in that case right click menu can be used to pick that particular bench. It shows the mined benches with a check mark. The "Edit Pit" command can be used to edit the pit values and attributes. (For more information refer to "Edit Pit" command under Boundary dropdown menu of Carlson Surface Mining).

Pulldown Menu Location: Reserves/Timing in Surface Mining

Keyboard Command: pit_scheduler

Prerequisite: Pits with assigned quantities

Surface Equipment Timing

Surface Equipment Timing schedules through distinct pits on a bench-by-bench basis. Each bench is mined by a single piece of equipment according to a working calendar. This is unique compared to the Surface Production Timing command in that it requires the use of grids/quantities stored in pits, adheres to a work calendar, and allows for varying rates of production for each piece of equipment. This command requires careful preparation and organization. The following documentation explains the details of this command according the below outline:

1. Required Setup
2. Surface Equipment Timing Dialog
3. Reporting Options

Before Surface Equipment Timing may be executed, proper setup is required. This includes 1) creating pits/benches with quantities and mining direction and 2) defining the project working parameters via the Timing Project Manager.

1) Create pits/benches

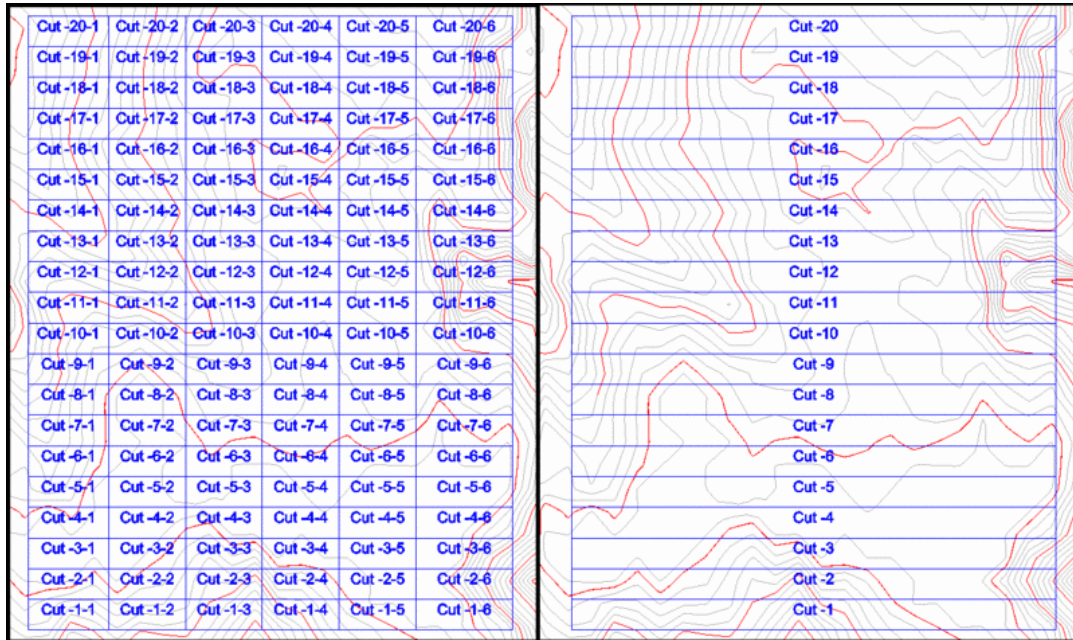
In order to create pits/benches ready for timing, the following steps must be taken.

1. Create Pits

As previously mentioned, the Surface Equipment Timing command schedules through pits on a bench-by-bench basis. These benches are the base unit used for scheduling, making proper layout of pits/benches key to effective scheduling. It is helpful to first define the scope of the planning process (long-range vs. short-range and schedule flexibility required for equipment). It is important to note that only one piece of equipment may work on one bench at a time, and that partial mining of a bench is not allowed; once a piece of equipment begins mining a bench, it must finish that bench before it can start on another bench.

Short range planning can require more flexibility in the sequencing of pits/benches, which requires pits/benches to be divided into smaller distinct units. Long range planning, however, often requires less

detail, and can therefore consider larger areas to be mined as a single unit. Consider the below image, which shows how an area of land may be divided into pit outlines for different planning scenarios. The pits shown on the right do not allow for much flexibility in planning. If only one piece of equipment is working, at least one bench of each long pit must be fully mined before mining another bench. The pits on the left, however, allow for a greater amount of flexibility. In this scenario, a piece of equipment may mine Bench 1 of Cut 1-1, then mine Bench 1 of Cut 1-2, followed by Bench 2 of Cut 1-1, etc.



Pits are simply closed polylines that are tagged with a pit name and additional information. To create pits from linework, one of the below commands may be used. All of these commands are found under the Boundary Pulldown Menu of the Surface Mining Module. Detailed information about each command is listed in the corresponding section of the help manual.

- Name Pit Polyline
- Assign Pit Names By Layer
- Pit by Interior Point
- Pit by Interior Text
- Pit Matrix Layout
- Pit Layout by Advance
- Pit Layout by Width
- Pit Layout by Rate
- Import Pit Points

2. Store Quantities/Qualities to Pits by Bench

Once the pits have been created, the quantity and quality of material to be mined must be calculated within those boundaries. This information is stored within each pit in distinct benches. This is done by one of the below commands. The Surface Mine Reserves command is found under the Reserves/Timing Pulldown Menu, while the other two are found under the Boundary Pulldown Menu. Detailed information about each command is listed in the respective location of the help manual.

- Surface Mine Reserves
- Assign Timing Grids
- Import Pit Timing Data

3. Assign Mining Direction

Before pits/benches may be scheduled, a mining direction must be assigned. Some of the pit layout tools will assign mining directions automatically. Pits without a mining direction may be assigned direction via the Assign Directions command. Mining direction may vary by bench. It is important to note that the mining direction does not control the sequencing of pits. For example, a long row of pits may be sequenced to mine the west end first and progress to the east, but mining of each individual pit does not have to progress from west to east.

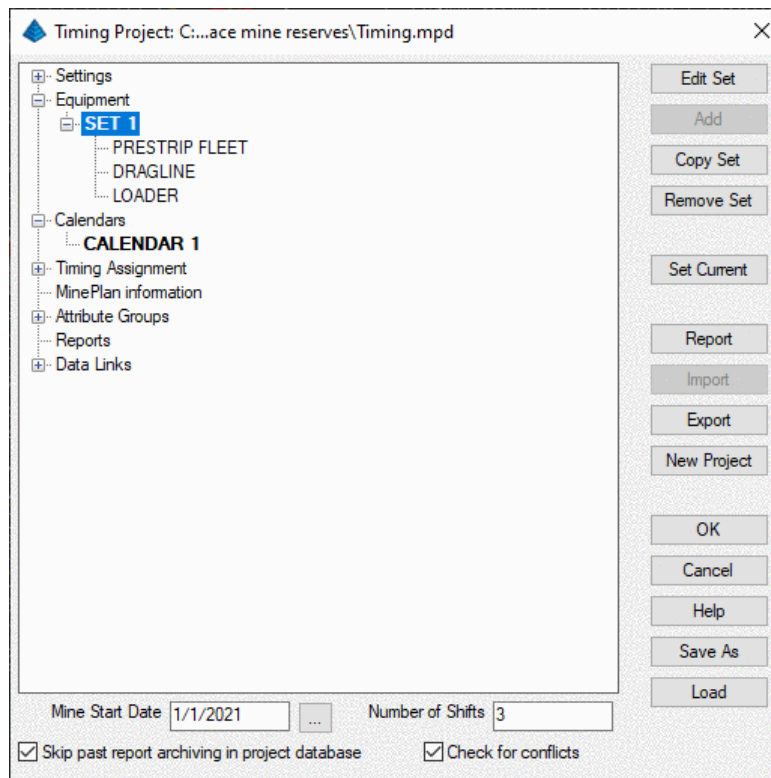
4. Optional - Verify Pits

Once pits have been named, assigned quantities/qualities by bench, and assigned a mining direction, they may be verified that they are ready for timing via one of the following commands, all of which are found under the Boundary Pulldown Menu.

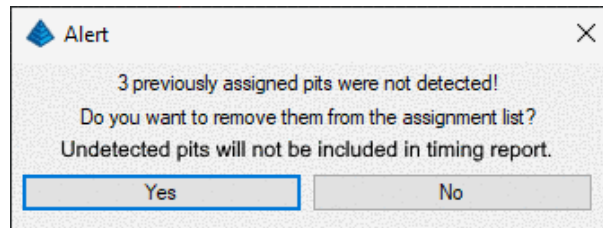
- Identify Pit Polylines
- Edit Pit
- Pit Quantities Report
- Pit Points Report
- Pit Inspector

2) Timing Project Manager

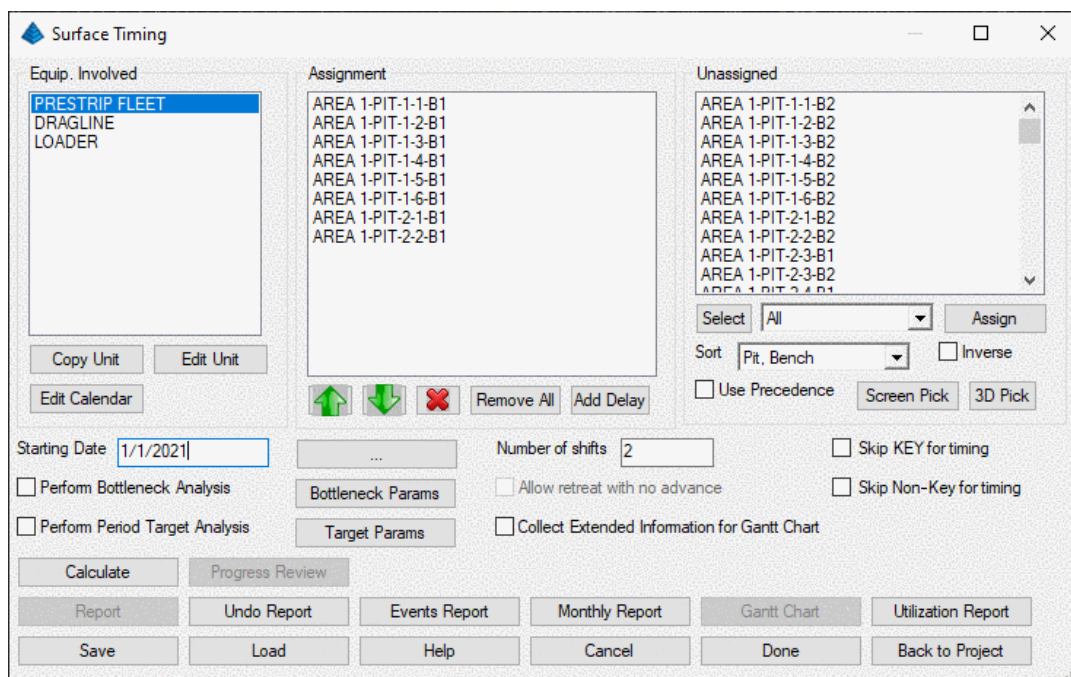
Working parameters are defined in the Timing Project Manager. This includes information such as the number of working units, equipment working rates, number of shifts, working calendars, etc. Each time the Surface Equipment Timing command is executed, the current Timing Project Manager settings will be displayed before continuing. An example of the Timing Project Manager is shown below. Detailed information about the Timing Project Manager is found elsewhere in the help manual.



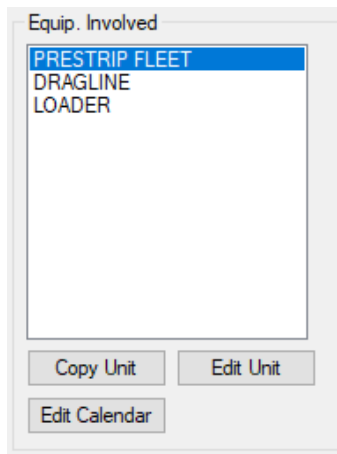
After clicking OK on the timing project, the program will check to see if any pits previously assigned to a piece of equipment were omitted from the timing. If this occurs, the below message box will appear asking if you want to remove the omitted pits from the assignment list. Any undetected pits will not be accounted for in the timing report.



Once the pits/benches and working parameters have been defined, the Surface Equipment Timing command may be used. After accepting the settings in the Timing Project Manager, the Surface Equipment Timing Dialog will appear as below. The primary purpose of this dialog to move pits from the Unassigned Column to the Assignment column in the order in which they will be extracted. Each piece of equipment has its own assignment list.



The Equip. Involved Column lists all pieces of equipment to be used for the mine scheduling. Benches must be assigned to a specific piece of equipment in order to be mined.

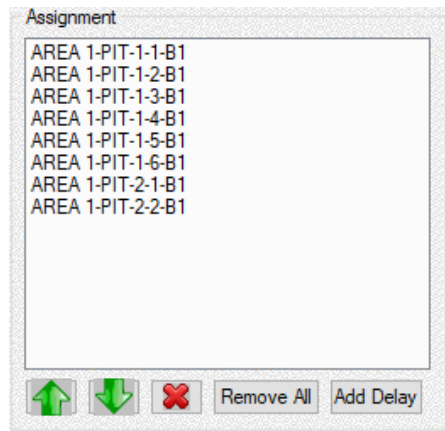


Copy Unit: This button will create a copy of the currently select unit.

Edit Unit: This button will allow you to edit the specifications of the currently selected equipment. More information on defining equipment specifications is available in the Timing Project Manager section of the help manual.

Edit Calendar: This button will allow you to edit the current work calendar. More information on defining the work calendar is available in the Timing Project Manager section of the help manual.

The Assignment Column lists the benches assigned to the currently selected equipment. Selecting a different unit will update this column with the list of benches assigned to that unit. This column lists benches in the order that they will be mined, with benches at the top of the list being mined first.

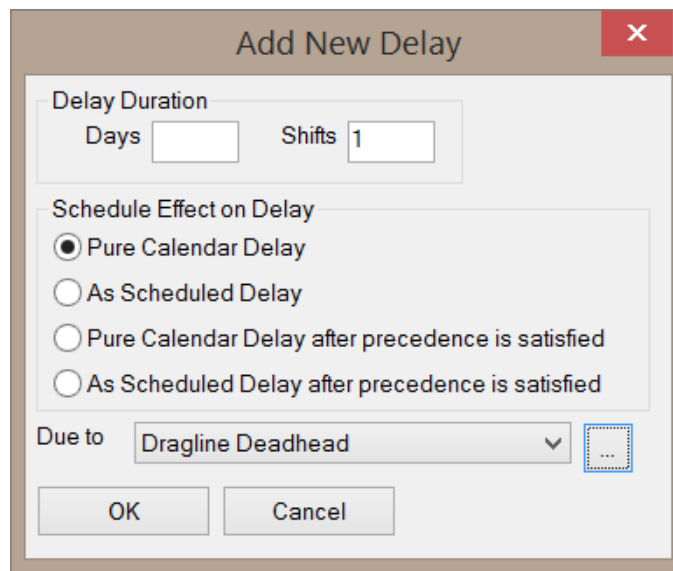


The green **Up/Down Arrow Icons** will move the currently selected bench(es) upward or down in the list, causing it to be mined earlier or later in the sequence.

The red **X icon** will remove the currently selected bench(es) from the list, allowing it to be re-assigned to another piece of equipment.

Remove All: This button will remove all assignments from the currently selected unit.

Add Delay: This button will open the dialog shown below, which controls how delays are handled. A delay will appear in the Assignment Column as if it were another bench to be mined, and may be moved up or down in the sequence.



Delay Duration: These values control how long the delay lasts. In the above example, the equipment will be delayed for a full shift. Note that partial shift delays are allowed.

Pure Calendar Delay: This option will not force a delay. If the equipment already has scheduled downtime when the delay is encountered, additional time will not be taken off.

As Scheduled Delay: This option will force a delay even if the equipment already has scheduled downtime when the delay is encountered. The delay will wait until the scheduled downtime is complete, then the delay will be applied.

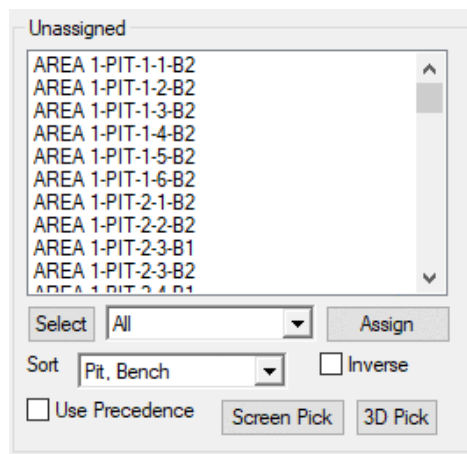
Pure Calendar Delay after precedence is satisfied: This is similar to a Pure Calendar Delay, but will not attempt to apply the delay until precedence rules have been satisfied. Consider the scenario shown in the below table. Equipment E1 is scheduled to mine Pit A and Pit C. Equipment E2 is scheduled to mine Pit B and Pit D. A precedence rule has been created that does not allow Pit D to be mined until Pit A is complete. A delay of this type is placed on Equipment E2 after Pit B. In this scenario, the finish date of Pit A will first be calculated, then the delay will be applied. If the equipment already has scheduled downtime when the delay is encountered, additional time will not be taken off.

As Scheduled Delay after precedence is satisfied: This option is similar to an As Scheduled Delay, but will not attempt to apply the delay until precedence rules have been satisfied. Consider the below scenario. Equipment E1 is scheduled to mine Pit A and Pit C. Equipment E2 is scheduled to mine Pit B and Pit D. A precedence rule has been created that does not allow Pit D to be mined until Pit A is complete. A delay of this type is placed on Equipment E2 after Pit B. In this scenario, the finish date of Pit A will first be calculated, then the delay will be applied. If the equipment already has scheduled downtime, the delay will be applied in addition to this downtime.

Due To: This dropdown menu lists the Drawing Event delays defined in the Timing Project Manager. The ellipsis button will allow you to define a new Drawing Event.

Equipment	E1	E2
Assignment Sequence	Pit A	Pit B
	Pit C	Delay
		Pit D (Precedence rule: Pit A)

The Unassigned Column lists all benches that have not yet been assigned to a piece of equipment.



Assign: This button will assign the selected bench(es) to the selected piece of equipment.

Select: This button will select all benches according to the drop-down menu to the right of this button. If the drop-down menu is set to NONE, all benches will be unselected. If the drop-down menu is set to ALL, all benches will be selected. If the drop-down menu is set to Bench 1, Bench 1 from all pits in the list will be selected.

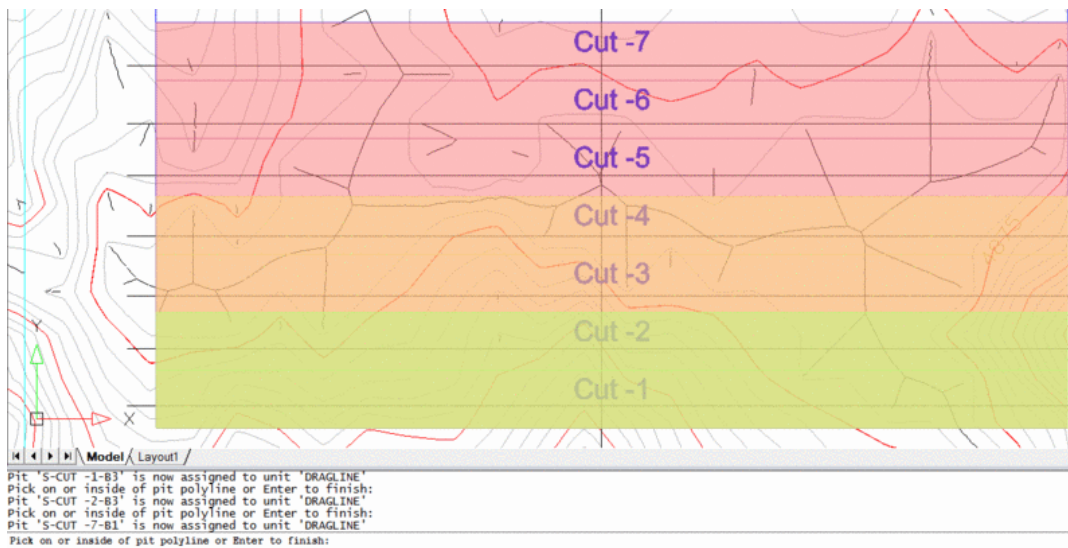
Sort: This drop-down list will sort the benches according to a variety of options.

- Pit, Bench: this option will sort the list in order of increasing pits, then in order of increasing benches (for example Pit1-B1; Pit1-B2; Pit2-B1; Pit2-B2; etc.)
- Bench, Pit: this option will sort the list in order of increasing bench, then in order of increasing pit number (for example Pit1-B1; Pit2-B1; Pit1-B2; Pit2-B2; etc.)
- X-Bench Staircase: this option will sort the benches in a staircase method according to the number of benches available. A three-bench scenario will sort the benches in a staircase-fashion such as Pit1-B1; Pit2-B1; Pit1-B2; Pit3-B1; Pit2-B2; Pit1-B3; Pit4-B1; Pit3-B2; Pit2-B3 etc.

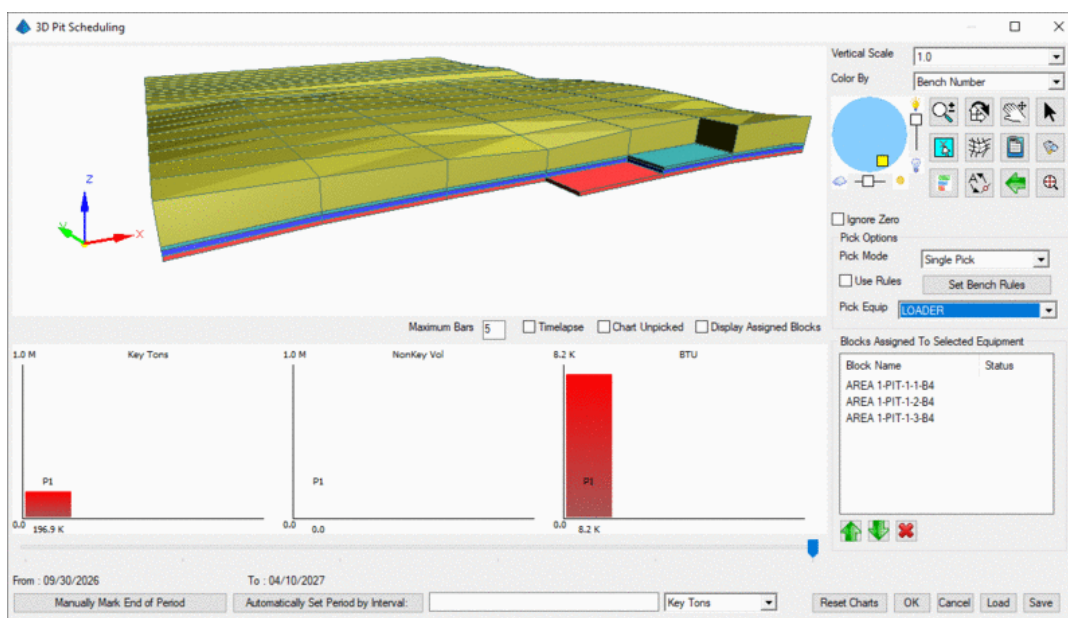
Inverse: This checkbox will invert the sorting order of the Unassigned Column.

Use Precedence: This checkbox will ensure that precedence is satisfied in the list. For example if Pit1-B1 is assigned to precede Pit1-B2, toggling this option on will reorder the list so that Pit1-B1 is listed before Pit1-B2.

Screen Pick: This button will allow you to sequence the pits by selecting them in the drawing. When activated, the Surface Equipment Timing Dialog will disappear and the drawing area will be visible. The command line will prompt you to click inside the pit to assign to the currently selected piece of equipment. Pits/benches that have already been assigned to a piece of equipment will be filled with a transparent color. Clicking inside a pit will assign the top available bench in that pit to the selected equipment. Clicking inside the pit a second time will assign the next available bench and the fill color will change to indicate that the second available bench has been assigned. Pressing ENTER after selecting the benches will return to the Surface Equipment Timing dialog. An example of the Screen Pick method of selection is shown below.



3D Pick: This button will open a new window, as shown below, to show a 3D view of the available pits and the volumes/qualities associated with them. This interactive window allows you to sequence pits in a 3D environment.



The benches shown in the 3D graphics window will be shown as flat benches by default. However, benches may be associated with elevation surfaces to show realistic dimensions via the Timing Project Manager attribute groups. More information on this feature is available in the corresponding section of the help manual.

Vertical Scale: This value controls the vertical exaggeration of the graphics window.

Color By: This drop-down list controls how benches are colored in the graphics window.

The *Bench Number* option will color each bench level with a color specified in the **Set Bench Rules** options.

The *Grade* option will color each bench according to a grade parameter file. Any attribute in the pit may be used for grade categorization.

The *Destination* option will color each bench according to its assigned destination. Note that destinations must be enabled in the Timing Project Manager in order for this option to be made available.

Control

Action



This control represents the position of the sun in plan view. If the yellow square is in the center of the blue circle, the sun is in a zenith. If the yellow square is near the edge of the circle, the sun is near the horizon. To move the yellow square, simply drag it to a new location, or click on the new location. The slider bars on the sides control the intensity and brightness of the display.

Switches to Dynamic Zoom mode. Click and drag to zoom in and out.

Switches to Rotation mode. When the cursor is placed near the outer edge of the view, a "Z" cursor is presented that permits rotation around the Z-axis. When the cursor is placed further into the interior of the view, an "X,Y" cursor is presented that permits the tilt angle of the view to be adjusted.

Switches to Pan mode. Click and drag to pan.

Switches to Pick mode. When active, double-clicking benches in the graphics window assign benches to the equipment or destination based on the **Pick Action** option.

Prompts you to select linework such as contours or 3D faces to add to the 3D viewer window. This linework can be used to identify reference points in the drawing to assist in pit selection.

Toggle shading of Pits and 3D Faces.

Exits the 3D Pick window, calculates the amount of time required to mine the assigned benches, and generates a report. This function is discussed later in this section of the help manual.



Displays a detailed report of the current timing assignment. This is similar to the standard Report function, but provides more detailed information and does not attempt to create a timing map. An example report is shown below.

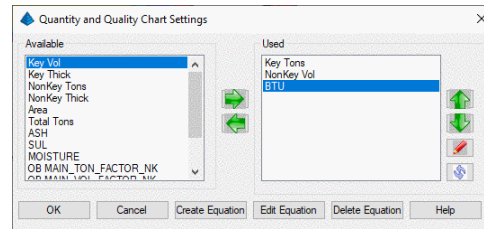
Unit	Days	Pit name	Start date	Finish date	Days pit
1	PRESTRIP FL	0 AREA 1-PIT-9/27/2016	10/1/2016		
2	DOAGLINE	6.8 AREA 1-PIT-10/2/2016	10/7/2016		
3	LOADS	10.8 AREA 1-PIT-10/7/2016	10/19/2016		
4	PRESTRIP FL	0 AREA 1-PIT-10/9/2016	10/7/2016		
5	DOAGLINE	0 AREA 1-PIT-10/7/2016	10/12/2016		-4
6	LOADS	0 AREA 1-PIT-10/19/2016	10/19/2016		-9
7	PRESTRIP FL	0 AREA 1-PIT-10/7/2016	10/9/2016		10.3
8	DOAGLINE	0 AREA 1-PIT-10/12/2016	10/17/2016		3.2
9	LOADS	0 AREA 1-PIT-10/18/2016	10/28/2016		6
10	PRESTRIP FL	0 AREA 1-PIT-10/9/2016	10/12/2016		12.4
11	DOAGLINE	0 AREA 1-PIT-10/17/2016	10/29/2016		5.2
12	LOADS	0 AREA 1-PIT-10/28/2016	10/29/2016		0
13	PRESTRIP FL	0 AREA 1-PIT-10/12/2016	10/19/2016		15.8
14	DOAGLINE	0 AREA 1-PIT-10/29/2016	10/29/2016		4.0
15	LOADS	0 AREA 1-PIT-10/29/2016	11/9/2016		1.0
16	PRESTRIP FL	0 AREA 1-PIT-10/19/2016	10/25/2016		22.0

The calculator icon at the top left of the screen will recalculate the timing results of the currently assigned benches. This allows additional pits to be added/removed from the assigned equipment and then quickly recalculate the impact on the scheduling.

The slider bar at the top of dialog represents the timeline of the mining progression. Clicking-and-dragging the slider will update the displayed information. The spreadsheet report will display benches to be mined in the currently selected time period.

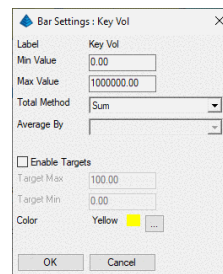


Controls the information displayed on the bar charts. When clicked the below dialog will appear.



This dialog contains two columns: Available and Used. Only items in the Used column will be shown in the Quantity and Qualities chart. Items may be moved between the two columns by selecting the item and clicking one of the green arrows between the two columns. To the right of the Used column are four icons. The green arrows can be used to move the selected item up and down in the list. The red pencil can be used to edit the selected item as described below. The blue reset icon will clear out all used attributes and return them to the available column.

When an item is first moved to the Used column, the below dialog will appear to control that item's appearance on the chart, including minimum value to display, maximum value to display, and color. If destinations have been enabled in the pits, you will be able to define charts for each grade of material according to the grade parameter file used to color the pits by Grade.



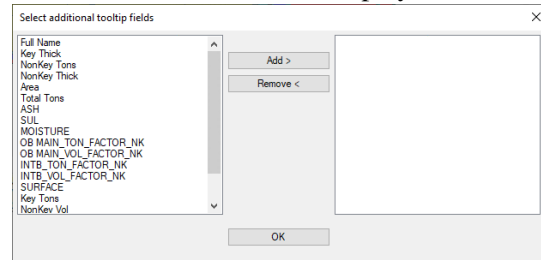
Total Method: This drop list controls how the attribute will be totaled. For volumes/tonnages, it is most sensible to simply Sum the parts to calculate the total, whereas quality attributes often need to be calculated as a Simple Average or a Weighted Average. When using the Weighted Average option, you will be able to use another attribute for weighting.

Enable Targets: This option will simply draw two horizontal lines across the bar charts, according to the Target Max and Target Min values.

Create/Edit Equation: These buttons allow you to create/edit new equations to display in the bar charts (and the tooltip). These equations are the same as those available in the Define Equations dialog.



Controls the attributes that are displayed in the tooltip when the cursor is hovering over a bench. Attributes in the left column will not be displayed.



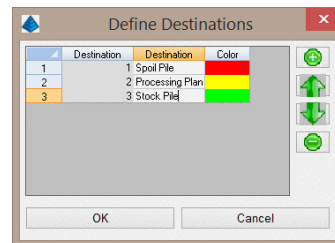
Undoes the last pit(s) selected in the 3D pick window.



Zooms to the extents of the items shown in the 3D window.



This icon will only be shown when destinations have been enabled in the Timing Project Manager. When clicked, the below dialog will appear. Here you can define various destinations and assign a color. Destinations may be added/removed with the plus/minus icons, and reordered with the up/down arrows.



This icon will only be shown when destinations have been enabled in the Timing Project Manager. Clicking this button will prompt you for a grade parameter file. Benches will then be automatically assigned to their various destinations based on the grade of the material. Please note that the color assigned to the grade category in the grade parameter file must match the color of a pit destination. If no match is found, the bench will be assigned to the first destination in the above list.

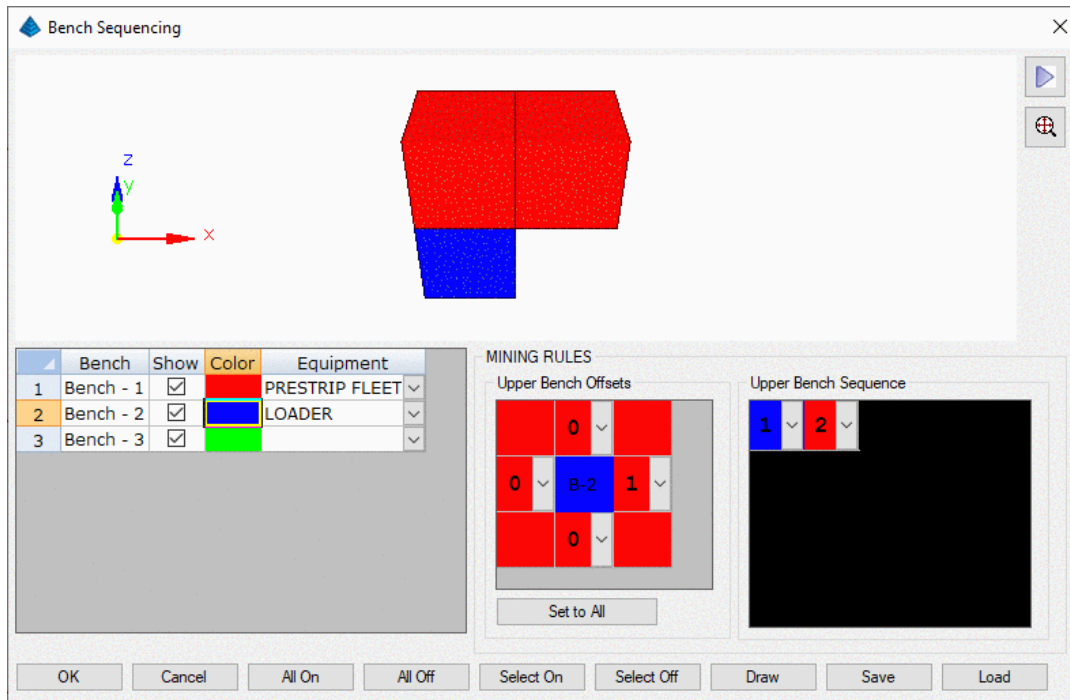


Ignore zero elev: This option will toggle the visibility of surface features at zero elevation in the graphics window.

Pick Mode: This drop-down list controls how benches are assigned to equipment. When the Single Pick option is selected, benches will be selected one at a time by double-clicking on each bench. When the Multi Pick option is selected, multiple pits may be assigned by double-clicking the first pit in a line to be mined, then double-clicking the last pit to be mined. Once two pits have been selected, they may be assigned to the equipment by right-clicking anywhere in the graphics window. Additional lines of pits may be added with additional double-clicks.

Use Rules: This option will apply the bench rules when selecting benches. These option can be defined with the Turn Benches On/Off icon.

Set Bench Rules: This button will open the Bench Sequencing dialog, shown below. This is intended to assist with sequencing when using the Single Pick option of selecting benches. When a bench is selected for mining, other benches will be automatically selected based on these bench rules. Note that in order to use the Bench Rules, pits must be oriented in the N-E-S-W directions. If the pits are not naturally oriented this way, the Twist Screen command may be used (prior to executing the Surface Equipment Timing command) to align the pits in the N-E-S-W directions.



The Graphics window at the top of this dialog will show an example of the bench rules to be applied. The List of Benches on the left side of this dialog controls various options for the benches. The Mining Rules on the right side of this dialog control the automatic sequencing of the benches.

Bench Column: This column lists the benches in order from top to bottom.

Show Column: This column controls if benches are shown in the graphics window.

Color Column: This column controls the color of each bench. Double-clicking one of the color cells will open the CAD color palette for color selection.

Equipment Column: This column controls which piece of equipment the bench will be assigned to. If no equipment is specified, the benches will be assigned to the current equipment selected on the 3D pick dialog.

Upper Bench Offsets: This graphic controls how many benches should be developed in addition to the bench that is actually selected in the 3D Pick dialog. The center square represents the bench that is selected in the 3D pick window. The numbers on each side of this square control how many upper-level benches must be sequenced in addition to the selected bench. In the above image, anytime a bench on level-2 is selected for sequencing, the bench on level-1 just to the east will also be sequenced. These benches will be assigned to the equipment specified in the Equipment Column.

Upper Bench Sequence: This graphic controls actual sequencing of the automatically sequenced benches. In the above image, the upper bench (red) will be mined first, then the level 2 bench will be mined.

Set to All: This button will apply the current bench sequencing rule to all other benches. For example, if a rule is defined for level 2 to mine one bench to the east on level 1, clicking this button will apply a similar rule to lower benches.

All On/Off: These buttons will toggle the "Show" option for all benches.

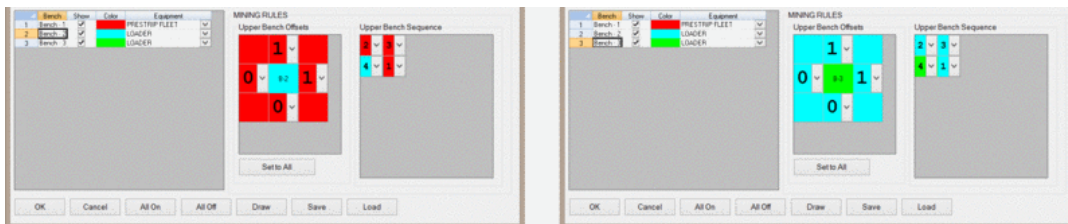
Select On/Off: These buttons will toggle the "Show" option for the currently selected benches.

Draw: This button will draw a preview of the selection in CAD. Picking a bench will fill in that bench with a yellow hatch. All other benches to be automatically sequenced along with this bench will be outlined with the same Bench Color. In the below example, Pit 1-3 Bench 2 has been picked for scheduling, and Pit 1-4 Bench 1 has been outlined to show that it will be automatically sequenced.

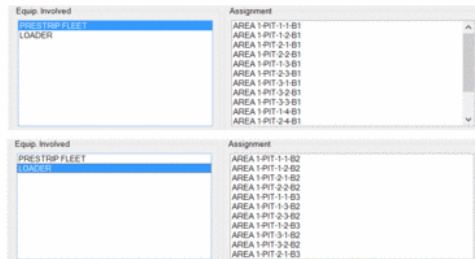
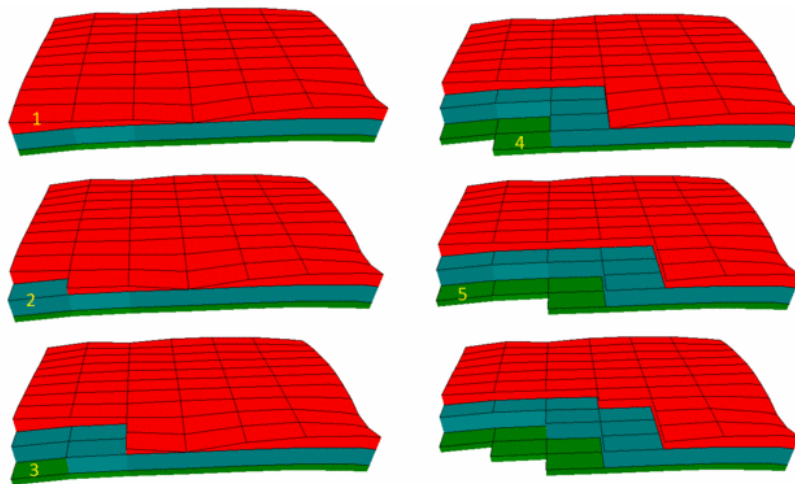
Pit-2-1	Pit-2-2	Pit-2-3	Pit-2-4	Pit-2-5
			Pit-1-4	Pit-1-5

Bench Sequence Rules Example:

The below images show the bench rules for a 3-bench example. In this example, whenever a level-2 bench is selected, up to 3 level-1 benches will also be sequenced. Whenever a level-3 bench is selected, up to 3 level-2 benches will also be sequenced. These rules compound, meaning that whenever a level 3-bench is selected, up to 8 level-1 benches may also be sequenced.



Using these bench rules, 5 benches were manually picked as shown below. The yellow number, indicating the order of selection, lies on top of the bench that was actually selected; all other benches were automatically assigned to the appropriate equipment. The final sequence applied to each piece of equipment is also shown below this progression. This allows a total of 26 benches to be sequenced by selecting only 5 benches. This can save tremendous amounts of time when working with multi-bench pits.



Pick Action: This option will only be shown when pit destinations have been enabled in the Timing Project Manager.

The *Set Equipment* option will assign benches to the current equipment.

The *Set Destination* option will assign the destination of the selected bench.

Pick Destination: This option will only be shown when pit destinations have been enabled in the Timing Project Manager and when the Pick Action is set to *Set Destination*. When shown, any benches selected will be assigned to the currently selected destination.

Pick Equipment: This drop-down list includes all equipment that have been added to the main Surface Equipment Timing dialog. Benches will be assigned to the selected equipment.

Blocks Assigned To Selected Equipment column: This list shows all benches that have been assigned to the selected equipment. The benches are listed in the order they will be mined. The green arrows to the right of this list will move the currently selected bench up and down in the list. The green X to the right of this list will remove the selected bench from the list and make it available for assignment to another piece of equipment.

Bar Charts: The charts at the bottom of the dialog show the quantities and qualities of the pits assigned to the current piece of equipment. As benches are assigned to a piece of equipment, the charts will dynamically update. The appearance of each chart is controlled with the Chart Settings icon, as discussed above. Each chart may show multiple vertical bars, with each bar representing a period of one or more pits. Periods may be defined manually or automatically. It is important to note, however, that the periods that define these charts are only temporary tools for inspection - these periods will not be used in the final timing report. It is important to note that the bar charts are intended to be a short-term inspection tool. The bar charts will not be saved when you exit the 3D Pick window.

Maximum Bars: This value controls the maximum number of periods (and thus, the number of vertical bars) that may be shown in each chart.

Timelapse: This checkbox will create a new period each time a bench is selected, thus adding another bar to the charts. Once the maximum number of periods is displayed, older periods will be removed from the charts. You will need to click the Reset Charts button before this option can be disabled.

Chart Unpicked: This option will reset the charts and only show information pertaining to benches that have not yet been assigned to a piece of equipment.

Display Assigned Blocks: This option will display timing blocks that have already been assigned to a piece of equipment. The blocks will display with a transparent gray color to indicate they have been assigned. When this option is enabled, selecting a pit in the "Blocks Assigned to Selected Equipment" list will highlight that block with a yellow color. With this option, you can quickly review the progression of blocks that have already been assigned to a piece of equipment.

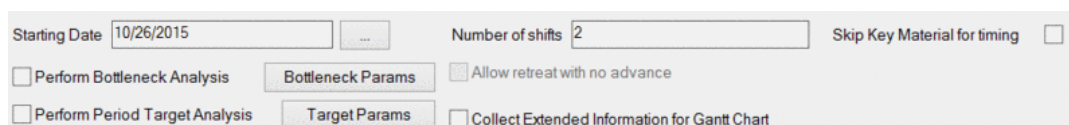
Period Slider Bar: The slider bar below the charts is only available when the Timelapse option is disabled. This slider bar allows you to control how many benches are represented in the last vertical bar of the charts. For example, if the slider bar is moved to the right end of the bar, all assigned pits will be represented in the last vertical bar chart. If the slider bar is moved closer further to the left, then fewer pits will be represented in the last vertical bar chart.

Manually Mark End of Period: This option is only available when the Timelapse option is disabled. The slider above this button controls which benches are in the current period. When this button is clicked, the current period will end and a new vertical bar will be added to the charts. You may then repeat the process to create additional periods. This allows you to compare attributes for multiple periods, thus showing, how the quality of the material mined will change over time. When manually marking the ends of the periods, none of the benches from a previous period will be considered in the new period.

Automatically Set Period by Interval: This button will define periods according to a set value. The pulldown menu to the right of this field controls which attribute is to be used for the definition of each period. For example, if this option is set to mark periods at 10,000 Key Ton intervals, the program will determine how many benches are needed to hit that target. Each time the value is exceeded, a new vertical bar will be added to the charts. It is often useful to reset the charts before using this option.

Reset Charts: This button will clear the charts.

Additional scheduling options are shown at the bottom of the Surface Equipment Timing dialog.



The screenshot shows a software interface for scheduling. It includes a 'Starting Date' field with the value '10/26/2015' and an ellipsis button to its right. Next to it is a 'Number of shifts' field with the value '2'. To the right of that is a checkbox labeled 'Skip Key Material for timing'. Below these are two rows of options. The first row has a checkbox for 'Perform Bottleneck Analysis', a 'Bottleneck Params' button, and a checkbox for 'Allow retreat with no advance'. The second row has a checkbox for 'Perform Period Target Analysis', a 'Target Params' button, and a checkbox for 'Collect Extended Information for Gantt Chart'.

Starting Date: This value sets the date to begin mining operations. The ellipsis button to the right of this field will open a small calendar to assist in date selection. By default, this value will be set the Mine Start Date set in the Timing Project Manager.

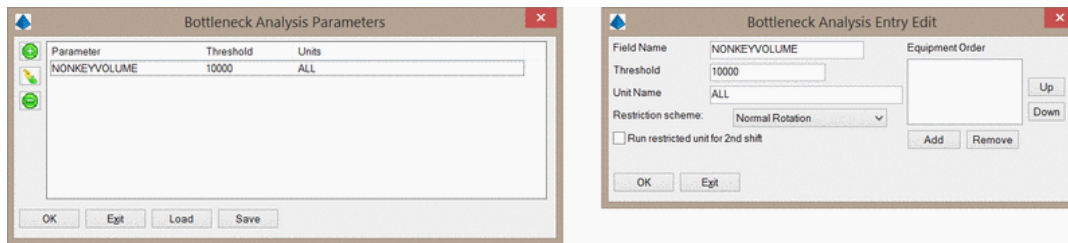
Number of Shifts: This value will set the number of working shifts in a day.

Skip Key for timing: This checkbox controls if key material is used for the timing. When this option is on, the key material (be it coal or some other resource) will not be included in the scheduling. Instead, it will be assumed that all key material will be mined quickly enough so that it never delays the removal of waste material.

Skip Non-Key for timing: This checkbox controls if Non-Key material is used for the timing. When this option is on, the Non-Key material will not be included in the scheduling. Instead, it will be assumed that all Non-Key material will be mined quickly enough so that it never delays the removal of Key material.

Perform Bottleneck Analysis: This option will perform a bottleneck analysis by limiting the total production from several pieces of equipment.

Bottleneck Params: This button will open the below dialog (left), which controls the Bottleneck Analysis.



The above dialog (left) lists all bottleneck parameters. The green plus icon to the left of the list will add a new parameter to analyze. The green ink quill icon will edit the selected parameter. The green minus icon will remove the selected parameter. When a new parameter is added, or a current parameter is edited, the dialog shown above (right) will open.

Field Name: This field sets the value to be used for bottleneck analysis, such as NONKEYVOLUME. This name must EXACTLY match an attribute used in the Detailed Report. A partial list of available names is given below. Note that user-defined attributes are not shown below. A full list of available attributes may be found in the Detailed Report > Report Formatter > Settings Tab > Attrib Options.

- KEYVOLUME
- NONKEYVOLUME
- KEYWEIGHT
- NONKEYWEIGHT
- WEIGHT
- etc.

Threshold: This value is the limit to be placed on the Field Name. In the above dialog, a maximum of 10,000 cubic yards of waste may be mined in a single shift.

Unit Name: This value sets the equipment to be limited by the Field Name. In the above example, the 10,000 cubic yard threshold is applied to all units. If only one or two units are listed, then the combined production of those units may not exceed the threshold value. The order in which units are listed will be used in the Restriction Scheme option.

Restriction Scheme: This drop-down list controls how units are scaled back due to bottlenecks.

The *Normal Rotation* option will alternate production through all units.

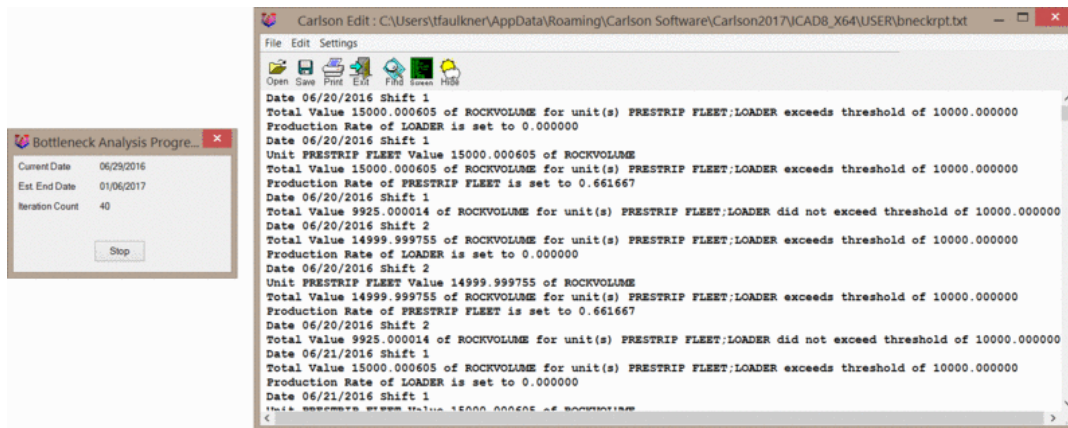
The *Hold Lead, rotate rest* option will operate the first unit listed in the Unit Name field at full production and all other units will be rotated through production.

The *Hold Tail, rotate rest* option will operate the last unit listed in the Unit Name field at full production and all other units will be rotated through production.

The *Fixed order option* maintains the order of unit preference and reduces production beginning with the tail unit until it is completely idled, then the next unit in the preference order is reduced until the objective is met.

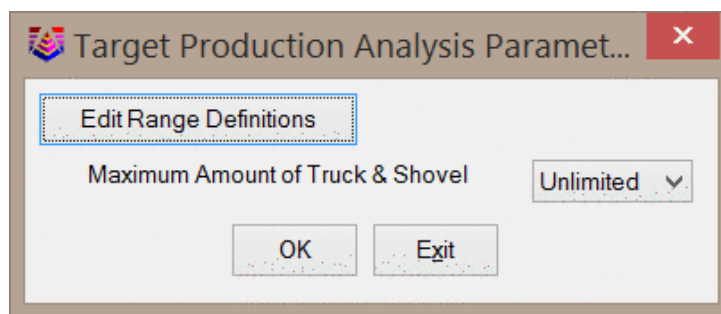
Bottleneck Example: Consider a PRESTRIP FLEET with a production capacity of 15,000 cubic yards per shift and a LOADER with a production of 12,000 cubic yards per shift. All the material excavated must be loaded and hauled away. Only 10 trucks are available to haul material, each with a hauling capacity of 1,000 cubic yards per shift. In this case, the trucks are the bottleneck in the mining process, limiting the overall production to 10,000 cubic yards per shift.

Using the Hold Last, rotate rest option for the Restriction Scheme, the PRESTRIP FLEET will be scaled back as necessary to ensure the LOADER is always operating at full capacity. When the schedule is calculated, the production capacity of each shift is checked, as indicated in the timing window shown below (left). After the schedule is checked for bottlenecks, a report of the shifts that require units to be scaled back will be generated, as shown below (right).



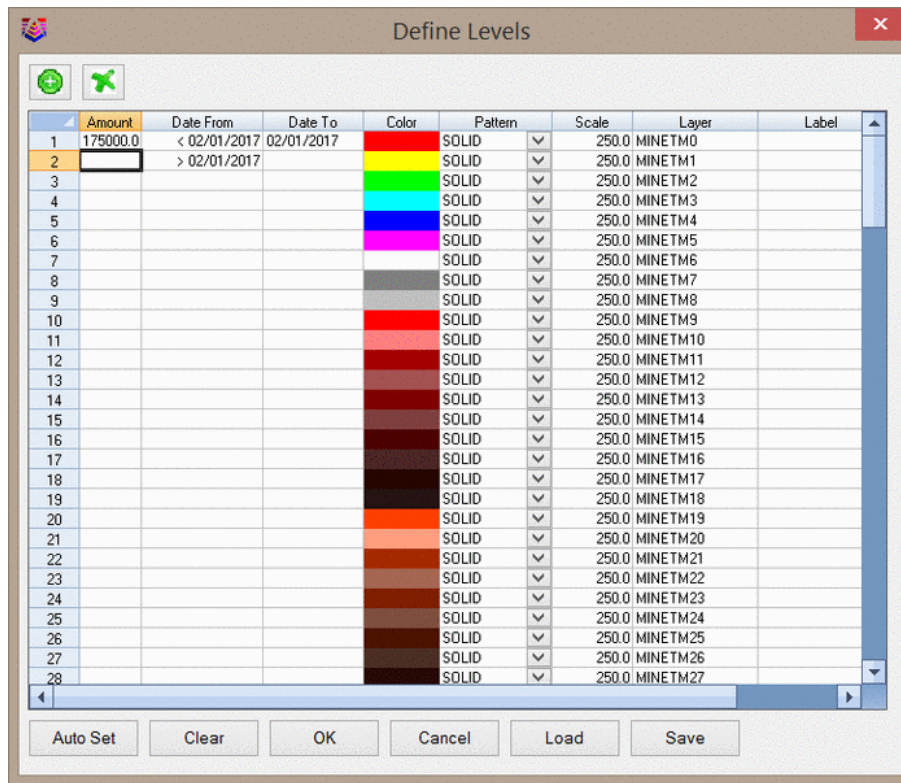
Perform Period Target Analysis: This option will perform a Period Target Analysis, which will add extra truck and shovel units for overburden removal as needed in order to ensure production targets are met. It is important to note that this option can only be used to increase overburden removal - additional units will not be added to increase the production rate of key material. To use this function, a piece of equipment named TRUCK&SHOVEL must be defined in the Timing Project Manager.

Target Params: This button will open the dialog shown below.



Maximum Amount of Truck and Shovel: This drop-down list will set the maximum number of Truck and Shovel units to be added to meet the production target.

Edit Range Definitions: This button will open the below dialog. In this dialog, the AMOUNT column represents the amount of Key material (in tons/tonnes) to be uncovered during the specified time period. This dialog is similar to the Amounts table, which is discussed in the Reporting Options section of this document. In the below example, a target production of 175,000 tons of key material must be uncovered by 02/01/2017.



Target Production Analysis example:

Consider the dialog shown above and the below reports. In the first report, only 146,000 tons of coal are uncovered. This is accomplished by mining three pits to completion and partially mining a fourth pit. If the target production shown above is applied, however, truck and shovel units are used to remove additional overburden in order to uncover the targeted 175,000 tons of coal.

Without Target Production Analysis

Period	Pit Name	Rock (c.y.)	Key Material (tons)	PRESTRIP FLEET Volume
01/01/2017 - 02/01/2017	Pit-1-1	296,296	44,212	296,296
01/01/2017 - 02/01/2017	Pit-1-2	296,296	46,654	296,296
01/01/2017 - 02/01/2017	Pit-1-3	296,296	48,792	296,296
01/01/2017 - 02/01/2017	Pit-1-4	41,111	7,227	41,111
Total		930,000	146,886	930,000

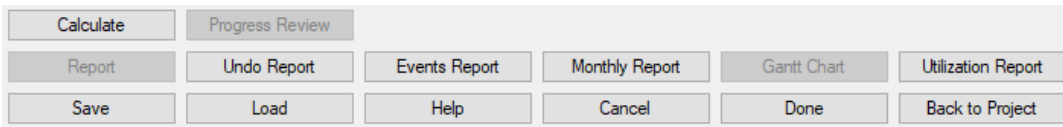
With Target Production Analysis

Period	Pit Name	Rock (c.y.)	Key Material (tons)	PRESTRIP FLEET Volume	TRUCK&SHOVEL Volume
01/01/2017 - 02/01/2017	Pit-1-1	296,296	44,212	252,818	43,478
01/01/2017 - 02/01/2017	Pit-1-2	296,296	46,654	252,818	43,478
01/01/2017 - 02/01/2017	Pit-1-3	296,296	48,792	252,818	43,478
01/01/2017 - 02/01/2017	Pit-1-4	201,046	35,345	171,545	29,501
Total		1,089,935	175,003	930,000	159,935

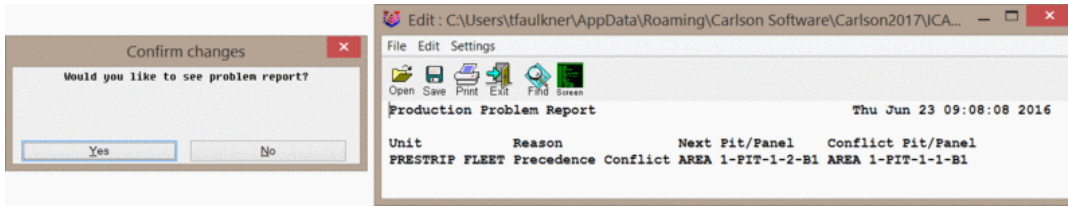
Allow Retreat with No Advance: This option is only used for the Underground Timing command. It will be disabled in the Surface Equipment Timing command.

Collect Extended Information for Gantt Chart: When enabled this option will allow the Gantt Chart to include extra information such as the pit volume, quality, etc.

The bottom of the Surface Equipment Timing dialog, shown below, has several buttons for generating reports.



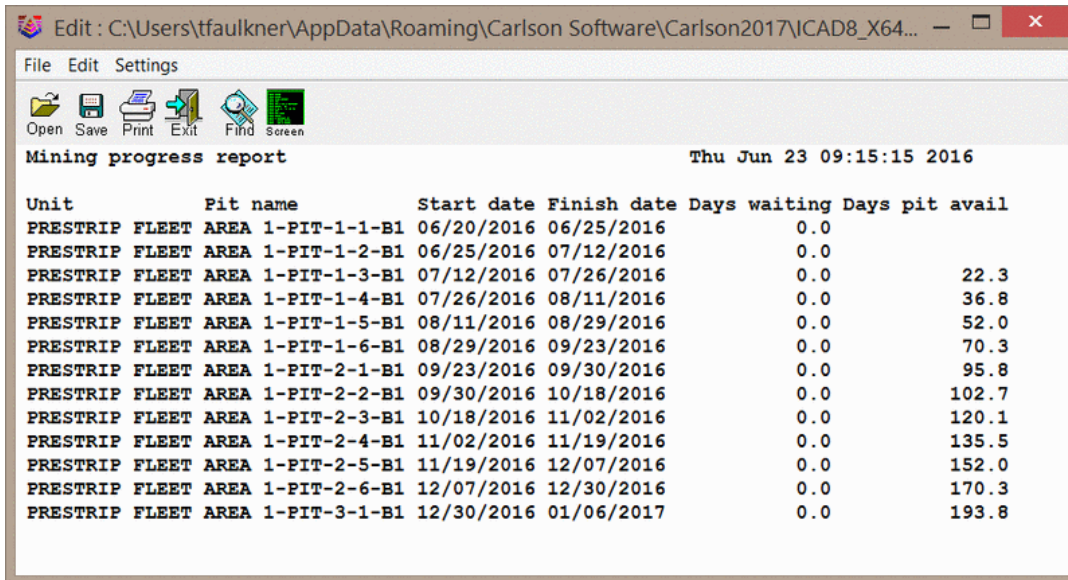
Calculate: This button will calculate the mine schedule. If any errors are encountered, the below dialog (left) will appear. The problem report generated will appear similar to below (right). In this example, a precedence rule has been violated for the PRESTRIP FLEET equipment. Pit-1-2-B1 cannot be mined until Pit-1-1-B1 has been mined. This will require an edit to the assignment order.



If no problems are encountered, the schedule will be calculated and the below dialog will appear.

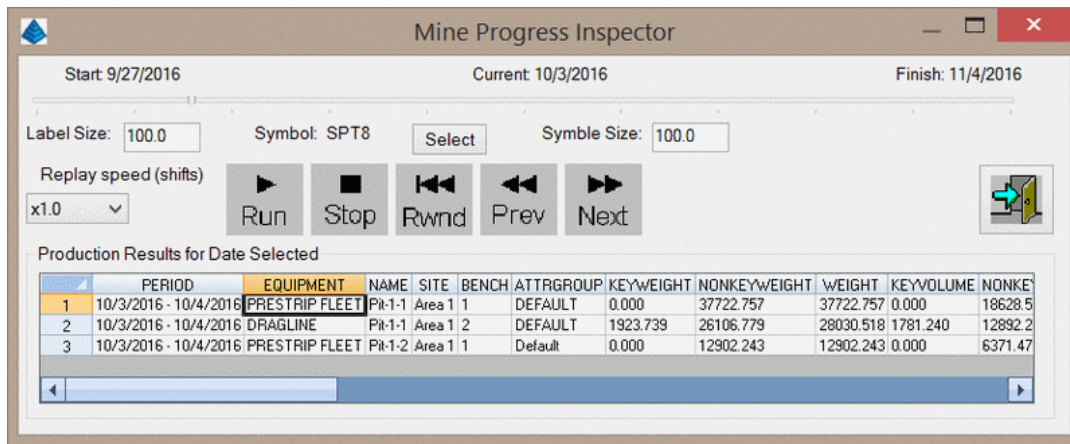


Report: This button will generate a very simple report as shown below. Note that this report does not include information about the amount of material mined. The Days Waiting in this report refer to the number of days in which a pit is waiting to be excavated due to precedence rules, or due to a scheduled delay. The Days Pit Avail refers to the number of days the pit is ready for mining (no waiting needed due to precedence)



Detailed Report: This button will generate a more detailed report, including information such as the amount of material mined, quality of material, downtime, etc. This report is discussed in more detail later in this help document.

Progress Review: This button will open the Progress Inspector shown below. The Progress Inspector will only be generated with information after the mine schedule has been calculated.



The Progress Inspector allows you to step through the day-by-day schedule of each piece of equipment. The slider bar at the top of the window may be used to cycle through the dates quickly. Various information about each piece of equipment is shown in the spreadsheet view. As you progress through each day with the below controls, a symbol and label will appear on the drawing to indicate the location of the equipment. The size of the labels/symbols may be set with the **Label Size** and **Symbol Size**. A different symbol may be selected by clicking the **Select** button.

Run: This button steps through the overall mine progression day-by-day. You may modify the playback speed by changing the **Replay speed** option.

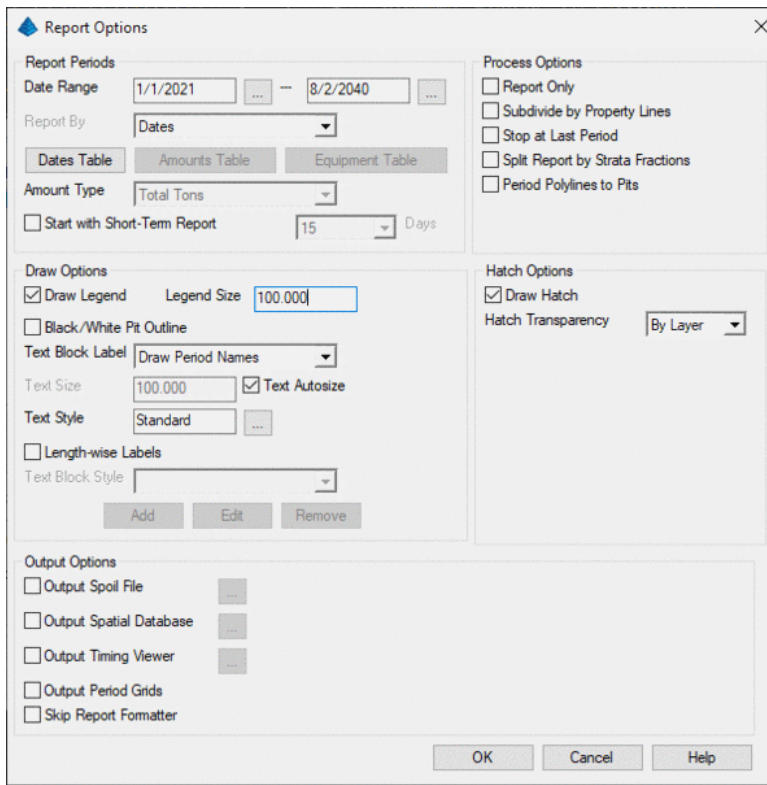
Stop: This button stops the action of the Run button.

Rwnd: This button returns the inspector to the first working day of the schedule.

Prev: This button steps to the previous day in the mine schedule.

Next: This button steps to the next day in the mine schedule.

Report: This button is only available after calculating the mine schedule. This button will open the Detailed Report dialog. This is the same dialog that will appear after selecting the Detailed Report option after clicking the Calculate button.



Date Range: This option will set this range of dates to report.

Report by: This drop list will determine how the timing map is generated.

If the *Dates* option is selected, the timing map will be colored by date ranges.

If the *Amounts* option is selected, the timing map will be colored by amount intervals.

If the *Equipment* option is selected, the timing map will be colored by equipment.

Dates Table: This button will open the Dates Table, shown below, which is used to define the various time periods used for reporting.

	End Date	Range	Color	Pattern	Scale	Layer	Label
1	2/1/2028	< 2/1/2028	Red	SOLID	Set	250.0 MINETM	Set
2	3/1/2028	2/1/2028 - 3/1/2028	Dark Red	SOLID	Set	250.0 MINETM	Set
3	4/1/2028	3/1/2028 - 4/1/2028	Red-Orange	SOLID	Set	250.0 MINETM	Set
4	5/1/2028	4/1/2028 - 5/1/2028	Orange	SOLID	Set	250.0 MINETM	Set
5	6/1/2028	5/1/2028 - 6/1/2028	Dark Orange	SOLID	Set	250.0 MINETM	Set
6	7/1/2028	6/1/2028 - 7/1/2028	Orange	SOLID	Set	250.0 MINETM	Set
7	8/1/2028	7/1/2028 - 8/1/2028	Dark Orange	SOLID	Set	250.0 MINETM	Set
8	9/1/2028	8/1/2028 - 9/1/2028	Orange	SOLID	Set	250.0 MINETM	Set
9	10/1/2028	9/1/2028 - 10/1/2028	Dark Orange	SOLID	Set	250.0 MINETM	Set
10	11/1/2028	10/1/2028 - 11/1/2028	Orange	SOLID	Set	250.0 MINETM	Set
11	12/1/2028	11/1/2028 - 12/1/2028	Dark Orange	SOLID	Set	250.0 MINETM	Set
12	1/1/2029	12/1/2028 - 1/1/2029	Orange	SOLID	Set	250.0 MINETM	Set
13	2/1/2029	1/1/2029 - 2/1/2029	Dark Orange	SOLID	Set	250.0 MINETM	Set
14	3/1/2029	2/1/2029 - 3/1/2029	Orange	SOLID	Set	250.0 MINETM	Set
15	4/1/2029	3/1/2029 - 4/1/2029	Dark Orange	SOLID	Set	250.0 MINETM	Set
16	5/1/2029	4/1/2029 - 5/1/2029	Yellow	SOLID	Set	250.0 MINETM	Set

The above dialog can be used to create custom time periods to show in the timing map. The green and red icons at the top of the dialog will add and delete rows of information.

End Date: This column sets the ending date of the period.

Range: This column is automatically generated, and simply identifies the Date Range for the period. Dates may be preceded by > or < to mark indefinite time periods.

Color: This column sets the color of the time period. Double clicking one of these cells will open the CAD color palette.

Pattern: This column sets the hatch pattern for the timing blocks. The Set button to the right of this column will open a dialog of available hatch patterns.

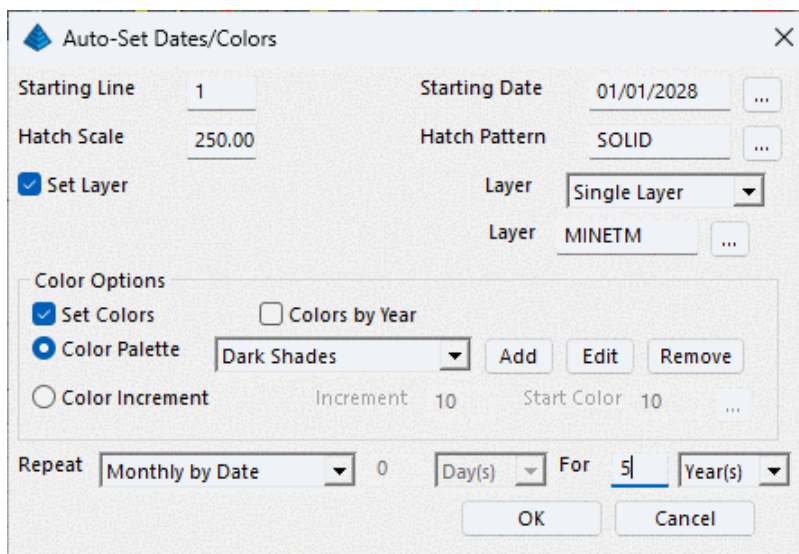
Scale: This column sets the scale of the hatch pattern.

Layer: This column sets the CAD layer on which to draw the timing block.

Label: This column sets a custom label for the timing block.

Clear: This button will clear the table of all information.

Auto Set: This button will open the below dialog, which allows for quick population of the Custom Dates table.



Starting Line: This value will set the row to begin the automatic population of the Custom Date table.

Starting Date: This is the Date From that will be used on the Starting Line. The ellipsis button will open a small calendar to assist in date selection.

Hatch Scale: This value will set the hatch scale for the new time periods.

Hatch Pattern: This field will set the hatch pattern for the new time periods. The ellipsis button will open the list of available hatch patterns.

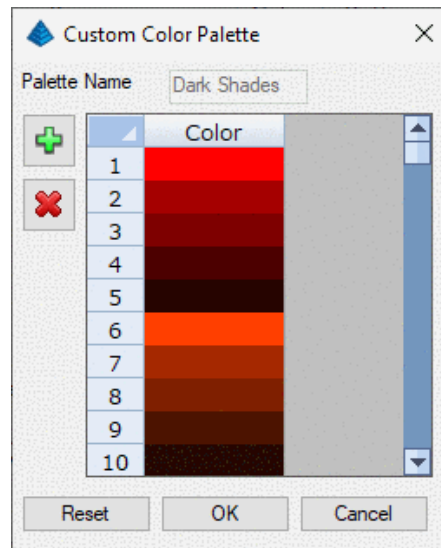
Set Layer: This option will overwrite the Layer names when enabled.

Layer by: This drop list determines how the layers will be populated. If the *Single Layer* option is selected, all layers will be the same, according to the **Layer** text box. If the *Year* option is selected, the layer will be set by the year of the time period.

Set Colors: This option will set custom colors for the new time periods. There are two options for setting colors - by using a Color Palette, or by using a Color Increment.

Colors by Year: When this option is enabled, the colors will only increment when the starting date for a row changes. This option is handy for drawing timing maps that have the same color for an entire year, but generate the report on a monthly basis.

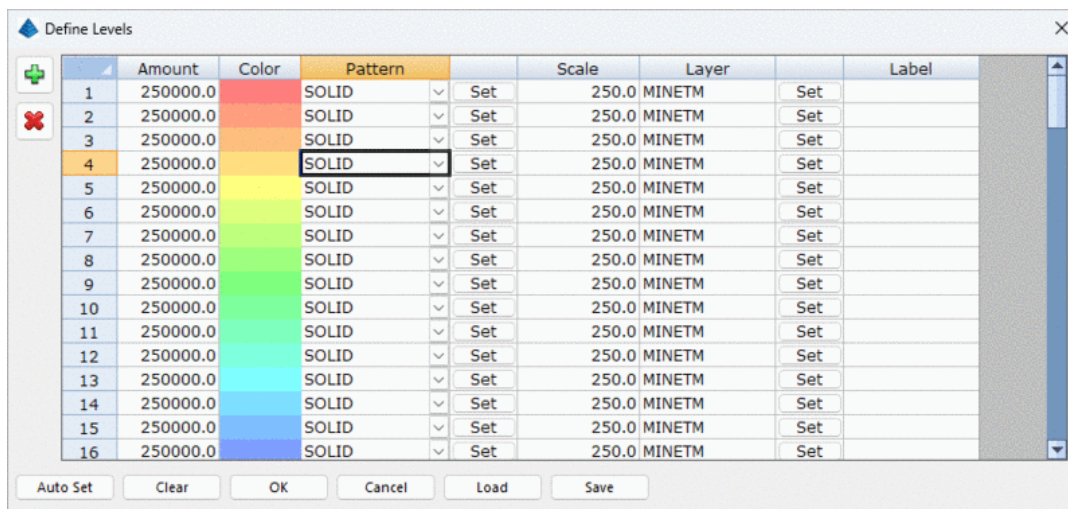
Color Palette: This option will use a predefined color palette to color the time periods. When the end of the palette is reached, the colors will start over. Palettes may be added, edited or remove used the corresponding buttons. An example color palette is shown below. Colors may be added or removed from the palette by clicking the Plus or X icons, and colors may be modified by double-clicking them. Any of the predefined palettes may not have their name modified. If you modify the colors in a predefined palette, you may click the **Reset** button to return to the default colors.



Color Increment: This option will color the periods by using a **Start Color** and a **Color Increment**. All colors in the CAD color palette range from 1 to 255. You may want to inspect the color palette when using this option to relate colors back to their numeric value. For example, a Start Color of 10 and a Color Increment 10 will increment through colors 10, 20, 30, etc.

Repeat: This option will determine the length of the time periods. Selecting the *Every* option will allow you to set a custom length for the time periods. You can also specify the length of time to define. For example, you could set the date ranges on a weekly basis for 2 years.

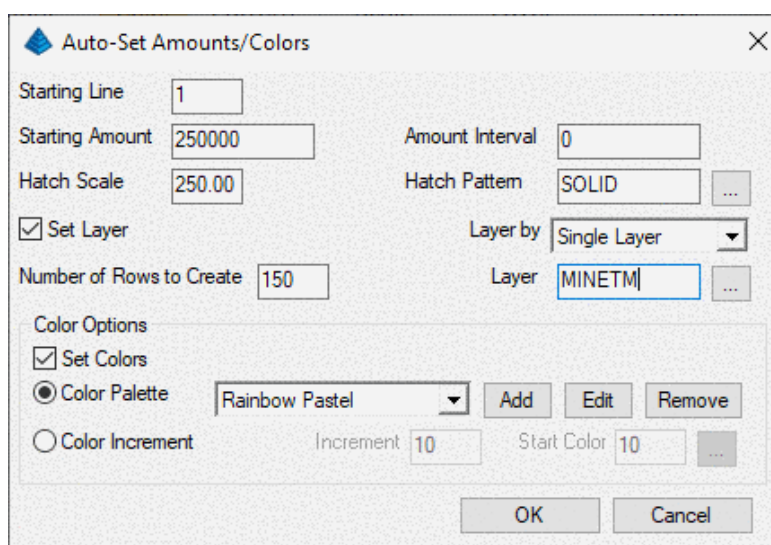
Amounts Table: This button will open the Amounts Table, shown below, which is used to define the various amount intervals used for reporting.



The above dialog is very similar to the Custom Dates table previously discussed. The primary difference is that the dates are replaced with the **Amounts** column. These values may represent one of the following values to be mined, set according to the **Amount Type**:

- Total Tons
- Key Material Tons
- Waste Tons
- Total Area
- Mined Area
- Total Volume
- Key Material Volume
- Waste Volume
- User Grid

Auto Set: This button will open the below dialog, which is very similar to the Auto Set dialog used in the Custom Dates table, as previously discussed. This dialog differs in that amounts are specified rather than dates, and the periods are set to repeat for a set number of times, rather than being based on the dates.



Equipment Table: This button will open the Equipment Table, shown below, which is used to set the colors for each piece of equipment to be reported. This is like a simplified version of the Dates Table and Amounts Table. Here there is no option to set a report period - only the color, hatch, layer, and label for each piece of equipment.

	Equipment	Color	Pattern	Scale	Layer	Label
1	PRESTRIP FLEET	Red	SOLID	100	MINETM	Prestrip
2	DRAGLINE	Yellow	SOLID	100	MINETM	Dragline
3	LOADER	Green	SOLID	100	MINETM	Loader

Start with short-term report: This option will include a shift-by-shift report for the beginning of the timing report. The drop list specifies the number of days to be reported in this manner. After this number of days, the program stops reporting shift-by-shift and reverts to reporting based on the defined periods.

Report Only: This option will not create a timing map; only the mine schedule will be reported. When selected, many other options on the dialog will be ghosted out.

Subdivide by Property Lines: This option will sub-divide the report by property owner.

Stop at last period: This option is only available when the Custom Amount Table report option is selected. This option will end the schedule at the last entered user row in the Custom Amount Table.

Split report by strata fraction: This option will utilize the tonnage and volume factors stored as pit attributes and report out the volume and tons of each named strata (rather than simply reporting Key and Non-key values).

Period Polylines to Pits: This option will generate new Pit Polylines from the period boundaries. The pit names will match the name of the period used to create them.

Draw Legend: This option will draw a color legend of the timing map. The **Legend Size** can be used to make the legend larger or smaller.

Black/White Pit Outline: This option will draw the boundaries of the timing periods with a white/black color as opposed to matching the color of the hatch.

Text Block Label: This drop list will determine how the labels are drawn in the timing map.

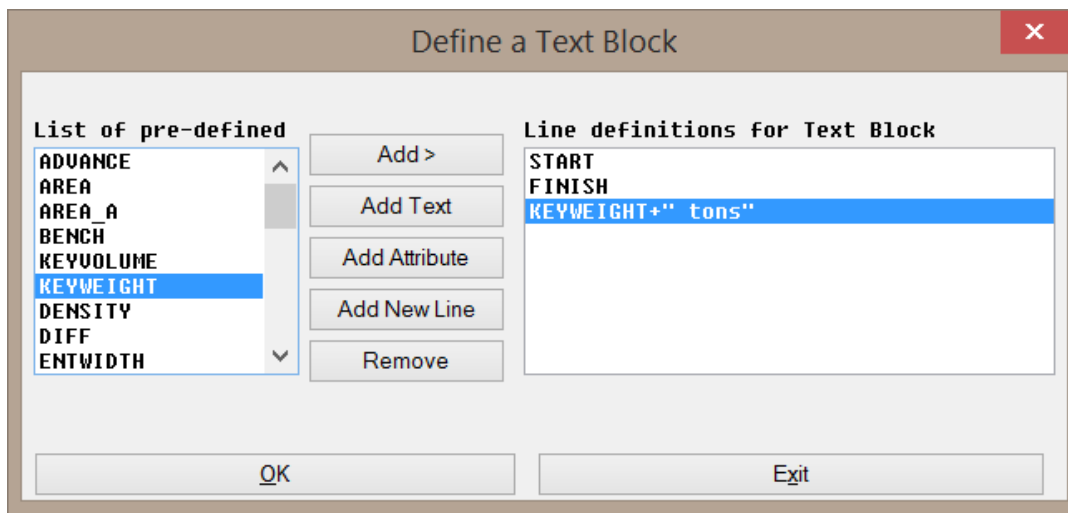
The *No Block Labels* option will not draw labels.

The *Draw Actual Dates* option will label the starting and ending date of each period.

The *Draw Period Names* option will draw labels defined in the Custom Date Table.

The *Use Custom Names* option will draw labels defined in the Custom Amount Table.

The *Use Custom Text Block* option will draw labels according to the below dialog. When selected, the **Text Block Style**, **Add**, **Edit**, and **Remove** buttons will become available. The Text Block Style dropdown list will list all available text block labels. The Add button will create a new text block. The Edit button will edit the currently selected text block. The Remove button will remove the currently selected text block.



The left side of the above dialog contains pre-defined keys that may be used in the labels. The right side of the dialog shows how the text block label will appear.

Add: This button will add the selected attribute to the current line. Multiple attributes may be added to a single line.

Add Text: This button will add custom text to the current line. In the above example, "tons" has been added as a suffix to the KEYWEIGHT attribute.

Add Attribute: This button will add any attribute that is not defined in the pre-defined list. This attribute name must match an attribute available in the final text report.

Add New Line: This button will create a new line in the text block. In the above example, three lines have been created.

Remove: This button will remove the selected line from the text block.

Text Size: This value will set the text size of the block labels.

Text Autosize: This option will automatically size the text of the block labels.

Text Style: This option will set the text style to use for the period labels. The **Select** button will allow you to select one of the existing text styles.

Length-wise labels: This option will rotate the block labels in line with the mining direction of each bench. If not selected, labels will be drawn perpendicular to the mining direction.

Draw Hatch: This option will create boundaries for the timing periods and fill them in with the specified hatch pattern/color.

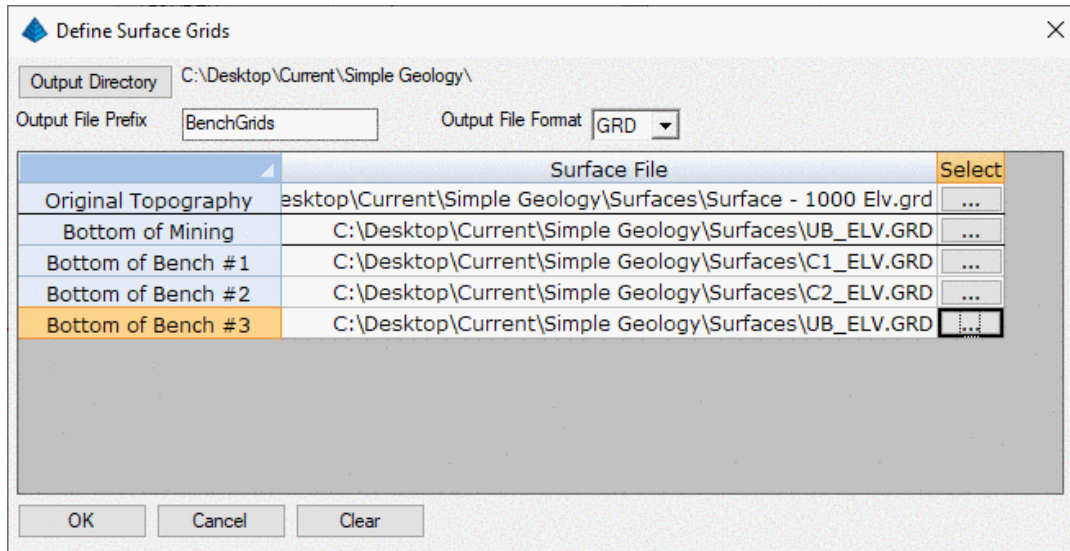
Hatch Transparency: This drop list sets the transparency of the hatches.

Output Spoil File: This option will create a .spo file to be used with the spoil timing commands found under the Spoil Pulldown Menu. The .spo file includes the amount of waste material excavated from each pit, the date the pit was excavated, etc. You may specify the .spo file by clicking the ellipsis button to the right of the checkbox.

Output Spatial Database: This option will output the report information to an SQLITE database. The database will retain the coordinate system of the drawing for linkage with GIS software packages. You may specify the file by clicking the ellipsis button to the right of the checkbox.

Output Timing Viewer: This option will output a .tv file and solids of each timing block. The .tv file may be reviewed to show a 3D playback of the timing, very similar to the Period Grids, but with the use of solids (.mdl files) rather than surfaces. You may specify the file by clicking the ellipsis button to the right of the checkbox. When using this option, you will need to specify the elevation of each bench using the dialog shown below.

Output Period Grids: This option will create a grid or TIN file of the updated surface at the end of each period. When the report is generated, the below dialog will appear.



The surfaces in the above dialog will be automatically set according to the files found in the Attribute groups of the Timing Project Manager. If these surfaces have not been assigned in the Timing Project Manager, they will need to be manually selected. It is important to note that the output surfaces will display near-vertical highwalls when the mining face advances. Each surface will be colored by the period/disturbance area that matches the color displayed on the timing map for each period.

Output Directory: This button will set the folder in which to save the output surfaces.

Output File Prefix: This button will set the prefix for each surface file to be saved.

Output File Format: This option determines if the output surfaces will be grids or TIN's.

Original Topography: This surface file defines the existing surface topography.

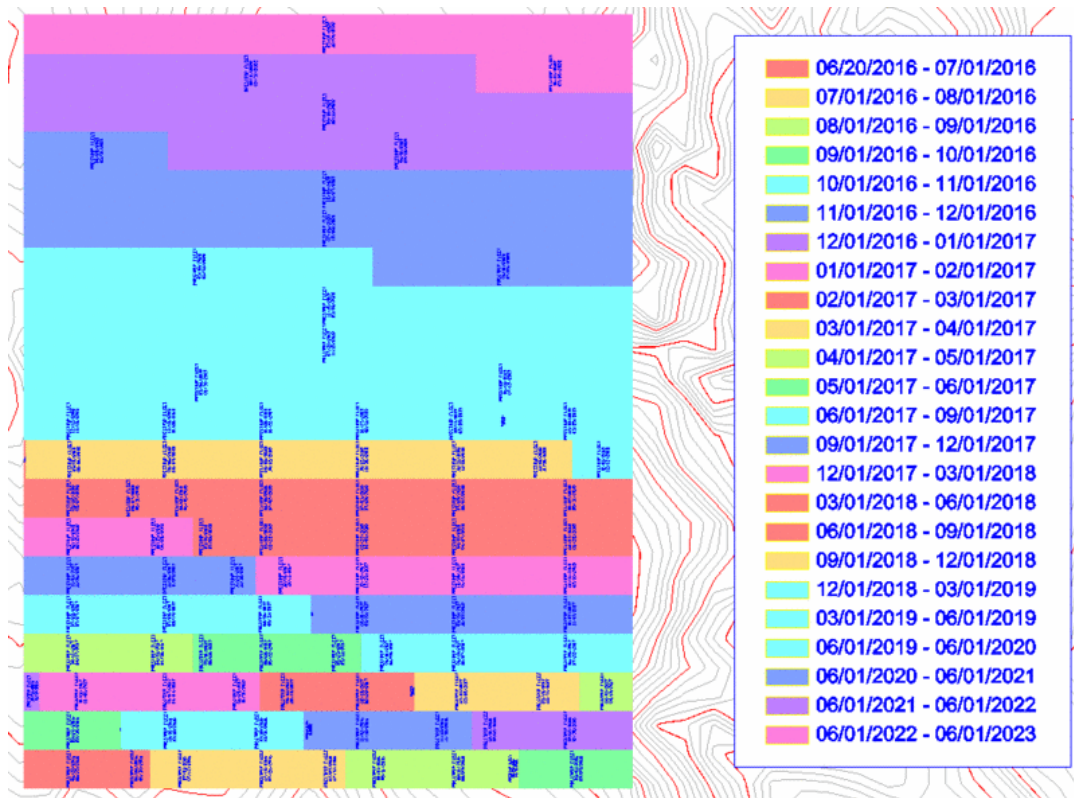
Bottom of Mining: This surface file defines the ultimate pit floor.

Bottom of Bench# : Each of these surfaces define the bottom of each bench.

Skip Report Formatter: This option will skip the report formatter. Rather than customizing the format of the report, the information will be automatically sent to a text file preview.

Examples of a timing map and a generated text report are shown below. Note that there are MANY ways that a

timing map and report may be formatted. The Report Formatter is used to generate the text report, but this dialog is not discussed at length here.



Key Material production report

Fri Jun 24 11:18:04 2016

Period	Pit Name	Equipment	Rock (c.y.)	Key Material (tons)	Key thickness	Strip ratio	Work shifts
06/20/2016 - 07/01/2016	Pit-1-1	PRESTRIP FLEET	174087.404	0.000	0.000	999.990	11.606
06/20/2016 - 07/01/2016	Pit-1-2	PRESTRIP FLEET	155912.595	35020.573	27.871	4.452	10.394
07/01/2016 - 08/01/2016	Pit-1-2	PRESTRIP FLEET	340416.957	76463.334	27.871	4.452	22.694
07/01/2016 - 08/01/2016	Pit-1-3	PRESTRIP FLEET	434376.592	115979.669	28.995	3.745	28.958
07/01/2016 - 08/01/2016	Pit-1-4	PRESTRIP FLEET	155206.452	41807.002	30.670	3.712	10.347
08/01/2016 - 09/01/2016	Pit-1-4	PRESTRIP FLEET	300240.273	80873.864	30.670	3.712	20.016
08/01/2016 - 09/01/2016	Pit-1-5	PRESTRIP FLEET	547677.202	128430.887	32.108	4.264	36.512
08/01/2016 - 09/01/2016	Pit-1-6	PRESTRIP FLEET	82082.523	18614.698	33.157	4.410	5.472
09/01/2016 - 10/01/2016	Pit-1-6	PRESTRIP FLEET	683558.597	155017.617	33.157	4.410	45.571
09/01/2016 - 10/01/2016	Pit-2-1	PRESTRIP FLEET	208597.055	0.000	0.000	999.990	13.906
09/01/2016 - 10/01/2016	Pit-2-2	PRESTRIP FLEET	7844.350	1684.265	27.986	4.657	0.523
10/01/2016 - 11/01/2016	Pit-2-2	PRESTRIP FLEET	513520.582	110258.269	27.986	4.657	34.235
10/01/2016 - 11/01/2016	Pit-2-3	PRESTRIP FLEET	416479.417	105521.581	29.147	3.947	27.765
11/01/2016 - 12/01/2016	Pit-2-3	PRESTRIP FLEET	43684.723	11068.209	29.147	3.947	2.912
11/01/2016 - 12/01/2016	Pit-2-4	PRESTRIP FLEET	496324.038	123584.364	30.896	4.016	33.088
11/01/2016 - 12/01/2016	Pit-2-5	PRESTRIP FLEET	359991.240	85043.359	32.405	4.233	23.999
12/01/2016 - 01/01/2017	Pit-2-5	PRESTRIP FLEET	188692.037	44576.098	32.405	4.233	12.579
12/01/2016 - 01/01/2017	Pit-2-6	PRESTRIP FLEET	705897.297	173445.404	33.121	4.070	47.060
12/01/2016 - 01/01/2017	Pit-3-1	PRESTRIP FLEET	35410.665	0.000	0.000	999.990	2.361
01/01/2017 - 02/01/2017	Pit-3-1	PRESTRIP FLEET	176813.269	0.000	0.000	999.990	11.788
01/01/2017 - 02/01/2017	Pit-3-2	PRESTRIP FLEET	529950.265	112312.183	28.078	4.719	35.330
01/01/2017 - 02/01/2017	Pit-3-3	PRESTRIP FLEET	223236.467	52636.365	29.248	4.241	14.882
02/01/2017 - 03/01/2017	Pit-3-3	PRESTRIP FLEET	272939.698	64355.765	29.248	4.241	18.196
02/01/2017 - 03/01/2017	Pit-3-4	PRESTRIP FLEET	537388.929	124065.383	31.016	4.331	35.826

There are many attributes available in this report. Most of these attributes are self explanatory, but some are not as clear. The below definitions are therefore provided.

- Difficulty Factor: This mining rate factor represents the difficulty of mining a specific area. A value of 1.0 means that mining progresses at a normal rate. This value is the product of the Equipment Difficulty and the

- Location Difficulty.
- **Equipment Difficulty:** This value is the product of the optional difficulty rates set in the equipment definitions. This can include
 - Date Difficulty
 - Thickness Difficulty
 - Bench Difficulty
 Note that pits have a dedicated Difficulty attribute that will be used as an additional multiplier for the Equipment Difficulty. This is not the same as specifying difficulty with the DIFF_BENCHX attribute.
 - **Location Difficulty:** This value is a difficulty factor determined by a hierarchy of the following three options. If the high priority item is detected, it will be used instead of the other two options. If the high priority option is not detected, the medium priority item will be used. If neither of these items are detected, the low priority item will be used. The low priority item defaults to a value of 1.0, so if no location difficulty is defined, the mining rate will not be affected.
 - Highest priority: text in the drawing on the DIFF_BENCHX layer (where X is a bench number)
 - Medium priority: DIFFICULTY attribute value/grid in the Timing Project Manager Attribute Groups
 - Lowest priority: Difficulty value assigned to the specific pit (set with the EDITPIT command)
 - **Calendar Days:** Actual period length between start date and end date (allows for partial days). Note that a day is not necessarily a 24 hour period, but is instead defined by the equipment definition. For example a piece of equipment working two 8-hour shifts means that one calendar day is equivalent to 16 hours.
 - **Operating days:** Scheduled shifts divided by number of shifts per day (as defined in equipment definition)
 - **Days:** Scheduled Hours divided by day length in hours (as defined in equipment definition). Note that if all shifts are the same length, this attribute will be the same as the Operating Days.
 - **Weekdays Worked:** Shifts scheduled on weekdays divided by number of shifts per day
 - **Saturdays Worked:** Shifts scheduled on Saturdays divided by number of shifts per day
 - **Sundays Worked:** Shifts scheduled on Sundays divided by number of shifts per day
 - **Days Waiting:** Number of days equipment cannot work due to a precedence rule
 - **Work Shifts:** Number of scheduled work shifts. A shift that is not scheduled to work (due to a Calendar Delay) will not contribute to this value.
 - **Hours Scheduled:** Number of hours equipment is scheduled to work (this includes Hours Utilized, Hours Delayed, and Hours Available)
 - **Hours Down:** Number of hours equipment is not scheduled to work (due to downtime scheduled on the calendar)
 - **Hours Utilized:** Number of hours equipment is actually working
 - **Hours Delayed:** Number of hours equipment is delayed due to a Drawing Event or Scheduled Delay (note, this is not the same as Hours Down caused by a Calendar Delay). Note that a Scheduled Delay will appear on the Assignment List for each piece of equipment
 - **Hours Available:** Number of hours is scheduled to work, but is neither being utilized nor is it actually delayed. If a piece of equipment encounters a delay during the middle of the shift, the partial shift time remaining will be counted as Hours Available (the delay will be applied to the next shift)

The below image helps visualize the organization of the different definitions of Hours.

Hours in a day	Hours Scheduled	Hours Utilized
		Hours Delayed
	Hours Down	Hours Available

Undo Report: This button will delete a previous timing map and clear the calculated schedule. This allows for quick what-if analysis of the scheduling.

Events Report: This button will display the Events Report, which lists all Drawing Events encountered during the mine schedule. Drawing Events are discussed in more detail in the Timing Project Manager. This report is only available after a Detailed Report has been generated. An example of the Events Report is shown below, in which four drawing events were encountered.

Event Key	Event Layer	Description	Period	Unit	Pit/Panel Name	Northing	Easting	Event Value	Station	Event Date	Year (start)	Month/Year
Deadhead	Delay1	Dragline at end of strip	08/01/2016 - 09/01/2016	DRAGLINE	Pit-1-6	1970979.55	1460575.17	Delay1	654.59	08/01/2016	2016	08/2016
Deadhead	Delay1	Dragline at end of strip	09/01/2016 - 10/01/2016	DRAGLINE	Pit-2-6	1971179.55	1460575.17	Delay1	654.59	09/16/2016	2016	09/2016
Deadhead	Delay1	Dragline at end of strip	10/01/2016 - 11/01/2016	DRAGLINE	Pit-3-6	1971379.55	1460575.17	Delay1	654.59	10/31/2016	2016	10/2016
Deadhead	Delay1	Dragline at end of strip	12/01/2016 - 01/01/2017	DRAGLINE	Pit-4-6	1971579.55	1460575.17	Delay1	654.59	12/16/2016	2016	12/2016

Monthly Report: This button will generate a monthly report, similar to the Detailed Report, but with less information available and without the ability to set custom time periods for reporting. An example of the Monthly Report is shown below.

Unit	Month	Year	Operating days	Days	Weekdays Worked	Saturdays Worked	Sundays Worked	Calendar days	Work shifts	Hours Utilized	Scheduled hours	Hours Availa
PRESTRIP FLEET	June	2016	30.00	30.00	22.00	4.00	4.00	30.00	60.00	263.99	719.99	456
PRESTRIP FLEET	July	2016	31.00	31.00	21.00	5.00	5.00	31.00	62.00	743.99	743.99	0
PRESTRIP FLEET	August	2016	29.00	29.00	21.00	4.00	4.00	31.00	58.00	695.99	695.99	0
PRESTRIP FLEET	September	2016	30.00	30.00	22.00	4.00	4.00	30.00	60.00	719.99	719.99	0
PRESTRIP FLEET	October	2016	31.00	31.00	21.00	5.00	5.00	31.00	62.00	743.99	743.99	0
PRESTRIP FLEET	November	2016	30.00	30.00	22.00	4.00	4.00	30.00	60.00	719.99	719.99	0
PRESTRIP FLEET	December	2016	31.00	31.00	22.00	4.00	4.00	31.00	62.00	743.99	743.99	0
PRESTRIP FLEET	January	2017	31.00	31.00	22.00	4.00	5.00	31.00	62.00	743.99	743.99	0
PRESTRIP FLEET	February	2017	28.00	28.00	20.00	4.00	4.00	28.00	56.00	671.99	671.99	0
PRESTRIP FLEET	March	2017	31.00	31.00	23.00	4.00	4.00	31.00	62.00	743.99	743.99	0
PRESTRIP FLEET	April	2017	30.00	30.00	20.00	5.00	5.00	30.00	60.00	719.99	719.99	0
PRESTRIP FLEET	May	2017	31.00	31.00	23.00	4.00	4.00	31.00	62.00	743.99	743.99	0
PRESTRIP FLEET	June	2017	30.00	30.00	22.00	4.00	4.00	30.00	60.00	719.99	719.99	0
PRESTRIP FLEET	July	2017	31.00	31.00	21.00	5.00	5.00	31.00	62.00	743.99	743.99	0
PRESTRIP FLEET	August	2017	31.00	31.00	23.00	4.00	4.00	31.00	62.00	743.99	743.99	0

Gantt Chart: This button will generate a Gantt chart of the mine schedule. This button is only available after calculating the mine schedule. For more information, see the Gantt Chart section of the help manual.

Utilization Report: This button will generate a Utilization Report of the mine schedule. This report lists each shift for each piece of equipment. Scheduled delays will be reported as such. An example of the Utilization Report is shown below.

Event Date	Shift	Equipment	Shift Hours	Util.	Ratio	Type	Calendar Rule
06/20/2016	1	PRESTRIP FLEET	12	1.00	1.00	UTILIZATION	
06/20/2016	2	PRESTRIP FLEET	12	1.00	1.00	UTILIZATION	
06/21/2016	1	PRESTRIP FLEET	12	1.00	1.00	UTILIZATION	
06/21/2016	2	PRESTRIP FLEET	12	1.00	1.00	UTILIZATION	
06/22/2016	1	PRESTRIP FLEET	12	1.00	1.00	UTILIZATION	
06/22/2016	2	PRESTRIP FLEET	12	1.00	1.00	UTILIZATION	
06/23/2016	1	PRESTRIP FLEET	12	1.00	1.00	UTILIZATION	
06/23/2016	2	PRESTRIP FLEET	12	1.00	1.00	UTILIZATION	
06/24/2016	1	PRESTRIP FLEET	12	1.00	1.00	UTILIZATION	
06/24/2016	2	PRESTRIP FLEET	12	1.00	1.00	UTILIZATION	

Back to Project: This button will return to the Timing Project Manager window.

Prompts

Select all pit polylines.

Select objects: *Pick all polylines* **Pull-Down Menu Location:** Reserves/Timing

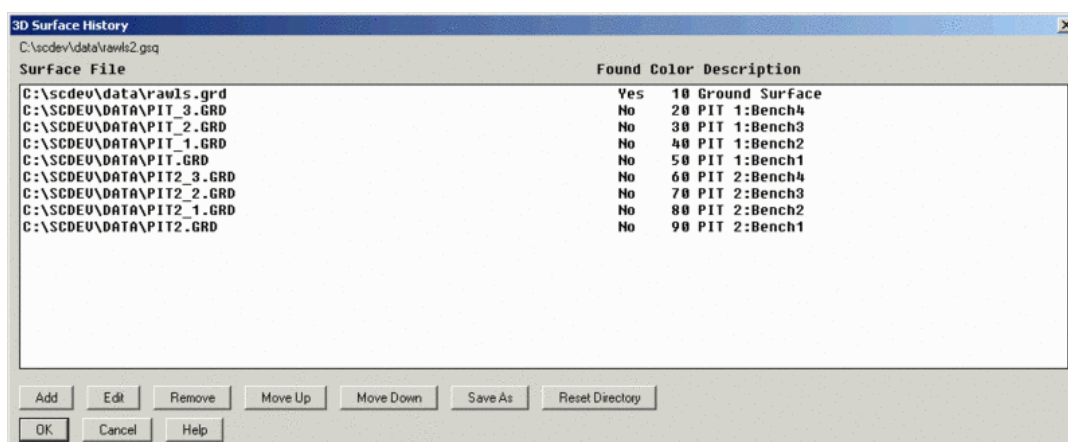
Keyboard Command: timepit

Related Commands: Surface Mine Reserves, Assign Directions, Surface Project Manager, Assign Timing Grids, Import Timing Data, Define Pit Attributes

View 3D Surface History

This command allows you to modify and playback the surface sequence history generated by several Carlson commands such as Design Bench Pit. The surfaces can be either grid or triangulation files. The command initially displays a list of surface files in the sequence. You can add more surfaces, modify surface colors or change the surface order. The Report function creates a report of the files and their parameters. The Add and Edit functions have a dialog to set the surface file name, description and color for the step. There are also settings for the optional pit and perimeter polylines. The pit polyline is the base of the pit where the cut slopes start. The perimeter polyline is the limit of the pit where the cut slope ties into the ground surface. When the Surface History has the pit and perimeter polylines assigned, then the Surface Mine Reserves command has an option to use the Surface History to report strata quantities for each step in the history.

When editing is complete, pick the OK button which brings up the playback 3D viewer window.



In the right side of the Viewer window, there are buttons allowing you to move from step to step in a surface sequence. On each step the surface displayed has elevations of the surface assigned for that step and the color of the last surface when the elevation at a particular location changed. For example the unaffected regions will have the color assigned to original ground surface. The Run Speed controls the rate of showing the surface with the Run button.

The blue circle icon with yellow dot represents the sky and sun position. You can move the yellow dot to change the shading for the sun. There are sliders to control the brightness of the ambient and sun lighting. The Display Axis toggle shows an X/Y/Z axis icon in the scene.

The zoom button activates zoom mode, where if you hold left mouse button and drag the mouse pointer up or down the picture zooms out or in correspondingly. In the rotate mode, the horizontal drag rotates the X axis (around Y), the vertical drag rotates the Y axis around X. The circular movement along the periphery of the picture rotates picture around the Z axis.

When pan button used, the picture will pan if mouse is dragged with the left button pressed.

The shade button turns the shading of the surface on or off.

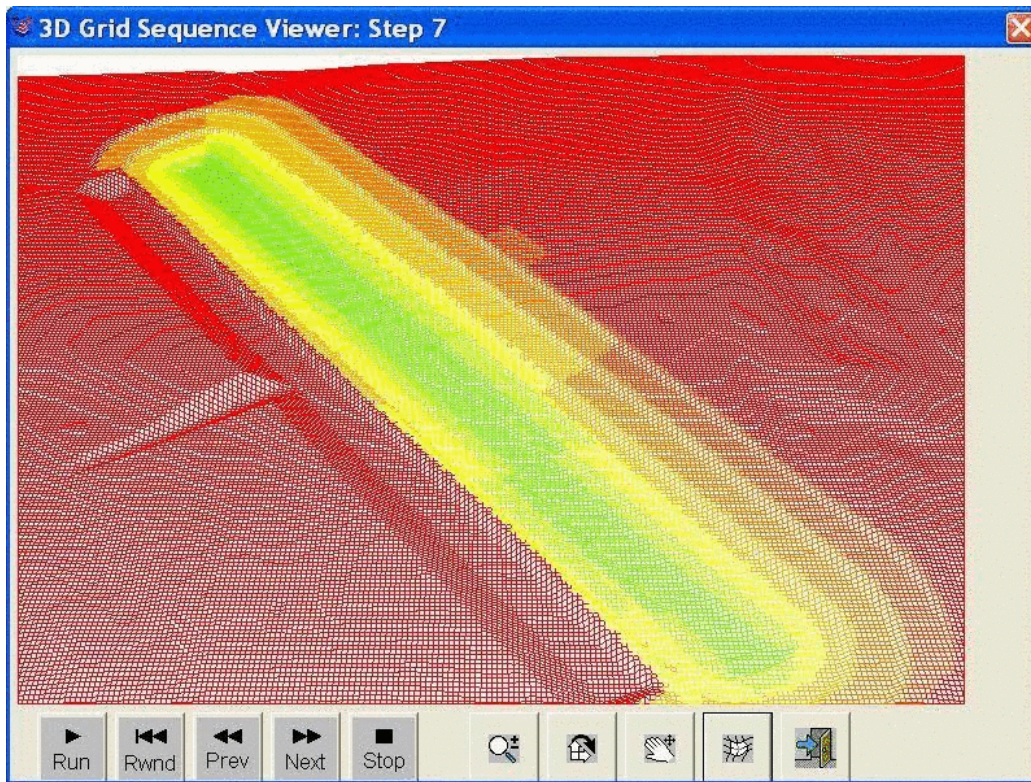
The camera button saves an image file of the current view.

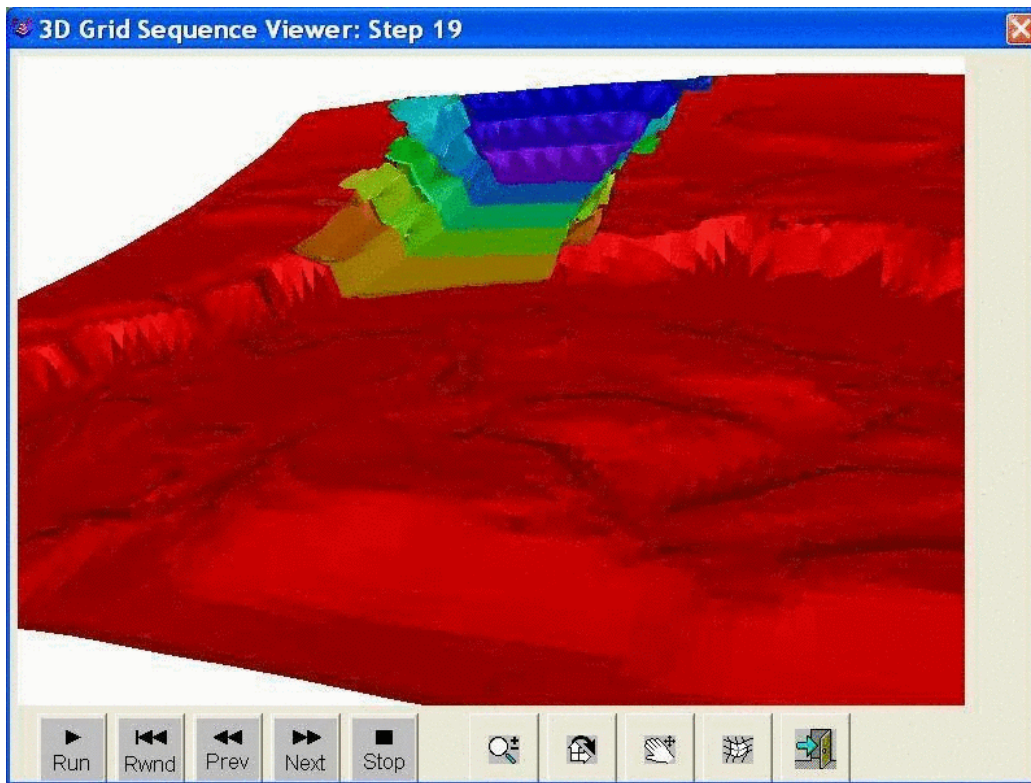
This GSQ file can be used in Surface Mine Reserves to automatically calculate the quantities by bench for the pits with highwalls.

Prompts

Surface Sequence File file selection dialog Choose the .gsq file to update.

Here are a couple of examples:





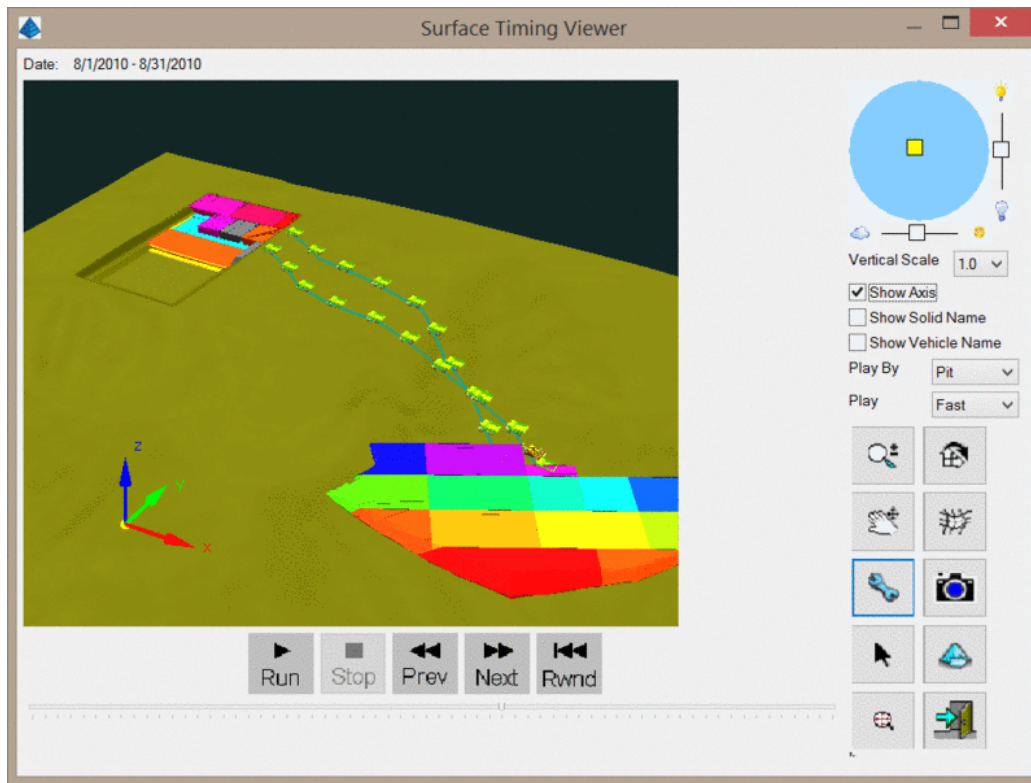
Pulldown Menu Location: Surface

Keyboard Command: gridstep

View 3D Timing

This command will open a .tv file to playback the progression of the mine. This file can be created from Surface Equipment Timing, Spoil Placement Timing, and Haul Cycle Analysis.

When the command is executed, you will be prompted to select a TIN file representing the topography at the end of the mine life. You will then be prompted to select a .tv file to view, after which the below dialog will appear. It is important to note that in order to view a .tv file, the linework for the spoil piles, haul routes, and the period polylines (output from Surface Equipment Timing as Pits) must be present in the drawing.



Vertical Scale: This value sets the vertical exaggeration for the 3D view.

Show Axis: This toggle controls the display of the X-Y-Z Axis icon in the lower left corner of the 3D view.

Show Solid Name: This toggle controls the display of labels above each solid. Names will only be displayed for solids that are being moved during the current time period.

Show Vehicle Name: This toggle controls the display of labels above each piece of equipment.

Play by: This option controls the playback of the 3D View. The *Pit* option will increment the time period based on when each pit is completely mined out. The *Spoil* option will increment the time period based on when a spoil polyline is completely filled.

Control



Action

This control represents the position of the sun in plan view. If the yellow square is in the center of the blue circle, the sun is in a zenith. If the yellow square is near the edge of the circle, the sun is near the horizon. To move the yellow square, simply drag it to a new location, or click on the new location. The slider bars on the sides control the intensity and brightness of the display.



Switch to Dynamic Zoom mode. Click and drag to zoom in and out.

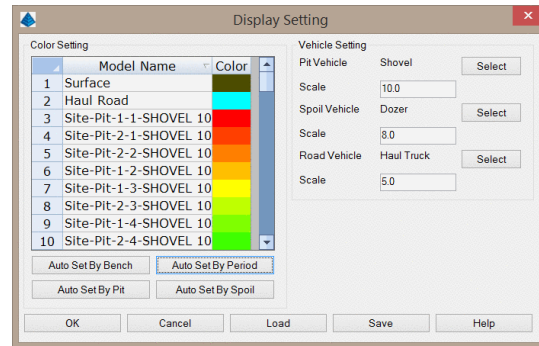
Switch to Rotation mode. When the cursor is placed near the outer edge of the view, a "Z" cursor is presented that permits rotation around the Z-axis. When the cursor is placed further into the interior of the view, an "X,Y" cursor is presented that permits the tilt angle of the view to be adjusted.

Switch to Pan mode. Click and drag to pan.

Toggle shading of 3D Faces and Surfaces.



Controls coloring and appearance of 3D symbols. When clicked, the below dialog will appear.



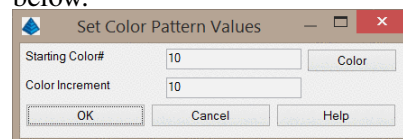
The spreadsheet shown in this dialog lists the name and color of each feature in the 3D window. You can automatically color the solids using one of the Auto Set By... buttons.

Auto Set By Bench: This option will color pits and spoil piles by bench number. For example, all instances of bench 1 will be red, all instances of bench 2 will be orange, etc.

Auto Set By Period: This option will color pits and spoil piles by the time period in which they are modified. For example, all material moved in the first month will be red, all material moved in the second month will be orange, etc.

Auto Set By Pit: This option will color pits and spoil piles by the pit. For example, all material from the first pit mined will be red, all material from the second pit mined will be orange, etc.

Auto Set By Spoil: This option will color pits and spoil piles by the spoil pile. For example, all material that is placed in the first spoil pile will be red, all material placed in the second spoil pile will be orange, etc. All of these buttons will display a similar dialog, shown below.



In this dialog, you can set a **Starting Color #** and a **Color Increment** according to the CAD color palette. The default option of a Starting Color # of 10 and a Color Increment of 10 will produce a rainbow gradient. A Starting Color # of 11 and a Color Increment of 10 will produce a lighter shade of a rainbow gradient.

The **Vehicle Settings** allow you to set different 3D models for the pit equipment, the haulage equipment, and the spoil equipment. The **Select** button will open the 3D Model Library, which includes several pieces of mining equipment. The **Scale** value controls the size of the model in the 3D view.



Takes a screenshot of the 3D Viewer. Several different image file formats are supported including bmp, png, jpg, xpm and gif. The image resolution and color depth may be set with this option.



Switches to Inspection mode. In this mode, hovering the cursor over a solid will display the name of the solid.



Outputs the current time period as a TIN file. With this command, you can progress through the timeline to key points in the mine life and export a surface model.



Exit the command.



Starts playback of the mine timeline.



Stops playback.



Steps back one time period.



Steps forward one time period.



Resets the playback to the beginning of the timeline.



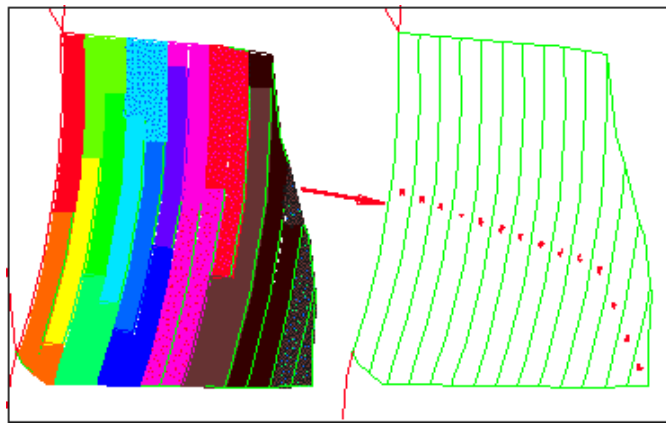
The slider bar shows the current position in the timeline. You can manually click-and-drag the slider bar to progress through the timeline.

Pulldown Menu Location: Reserves/Timing

Keyboard Command: tvviewer

Clear Timing Map

Removing the coloring after the timing routines are completed may be a tedious task. This routine removes and erases all text labels and hatching from surface pits and underground panels for a specific bench or mining direction in one step. It is a quick and efficient procedure, even for objects on many layers. Items are permanently removed from the drawing screen. It supports both surface and underground mining.



Prompts

For Surface Timing:

Enter Bench Number to Remove coloring for (Bench Number/<All>):

Select parts of mineplan to have timing coloring removed:

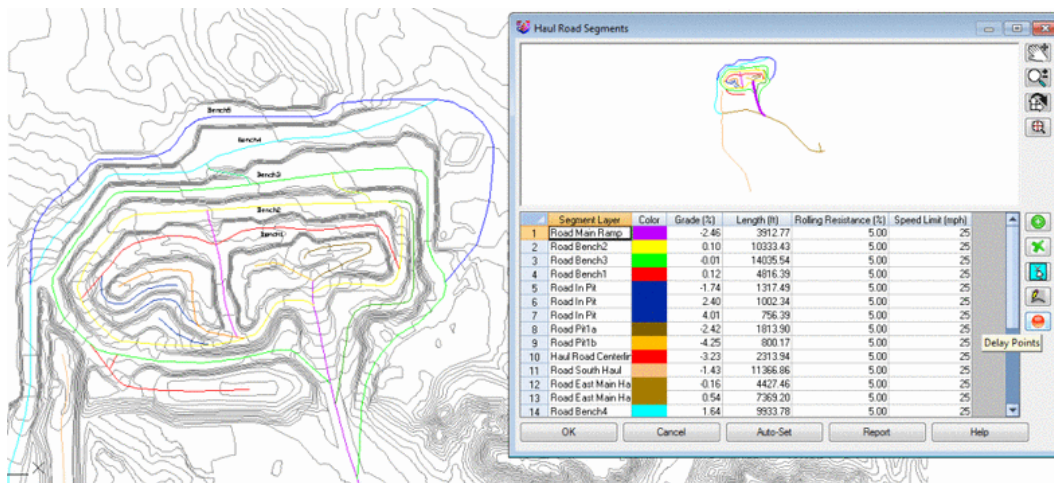
(Bench Number/<All>): *press Enter*

Pulldown Menu Location: Surface

Keyboard Command: utime_clear, stime_clear

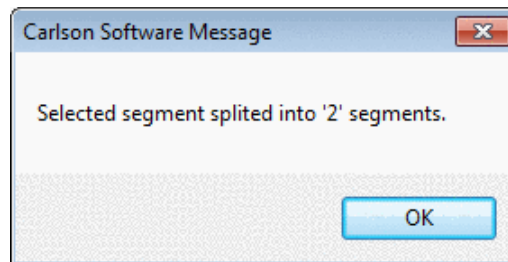
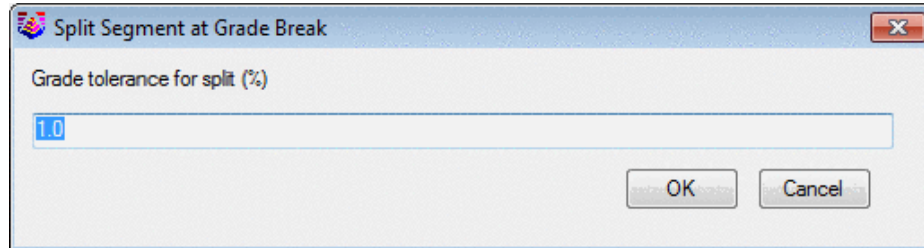
Haul Road Manager

This command builds and links the haul road network, recognizing 3D polylines and adding them to the network. All possible roads must be drawn as 3D polylines in CAD, either draped onto the surface model, or snapped to contours. Individual layers can be used for clarity to see the roads in the manager by color and layer. There aren't any tagged segments to select the first time this command is run, so the preview image will be blank. The segments must be added by picking the green "+" button to bring them into the editor.

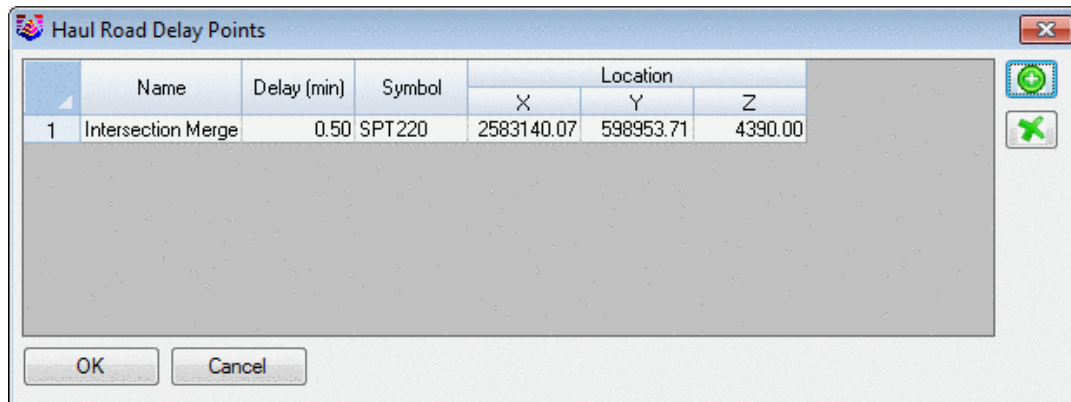


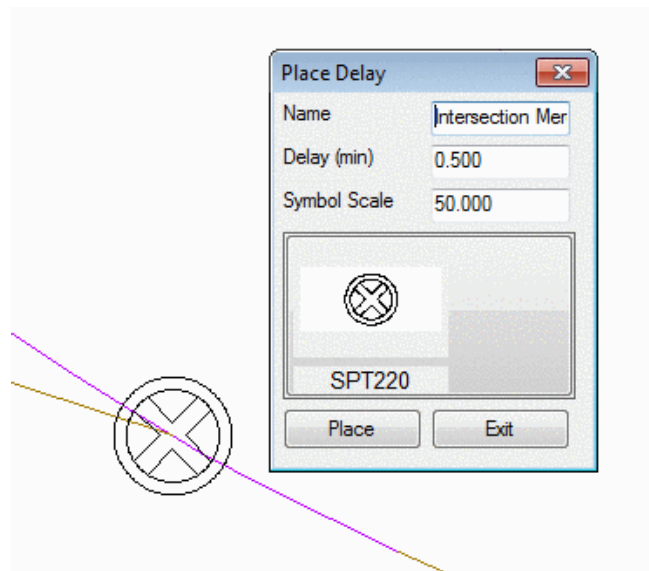
- **Preview Window Display:** The upper preview window has four buttons for Pan, Zoom Realtime, Rotate and Zoom Extents.
- **Add Segments:** Selecting the green "+" button will prompt for selecting the 3D polylines in CAD to add to the haul road network. This must be done initially if there are no segments found.
- **Remove Segment:** This green "X" button will remove the highlighted segment from the manager.

- **Pick Segments:** The Select from Screen, Pick Segments button is used to pick already named haul road segments, and add them to the manager. They must be already tagged with the Add Segments for them to be recognized.
- **Split at Grade Break:** Selecting this button will split the haul roads into smaller segments, based on the percent grade tolerance so that each segment is within that slope tolerance. This isn't a requirement, but the idea is to have segments with constant grade as the truck speed is depended on the grade of the road, and this will divide the road into smaller segments if desired. This will also break the 3D polylines into the smaller segments, so it is recommended to Save the drawing as a new name before breaking, so that there is a back up to go back to if necessary. There are two dialogs that appear from this command. It will only break the highlighted segments, so it is easy to control which roads are being divided.



- **Delay Points:** These are point symbols inserted into the drawing the will trigger a delay event for the specified number of minutes. Examples would be a stop sign at an intersection, or a one lane bridge where trucks have to stop. Any symbol from all of the libraries can be inserted to represent the location. These delays will be accounted for when the trucks pass over that spot.

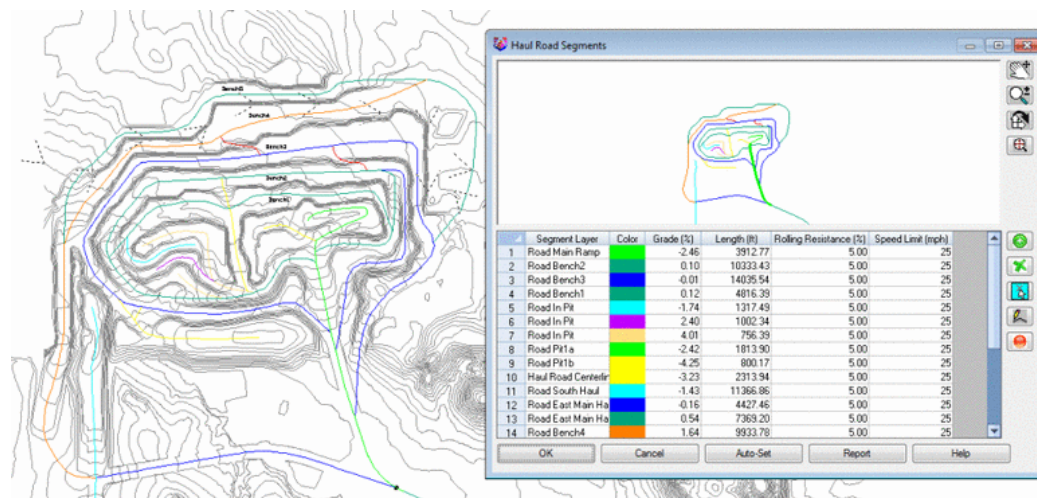




- AutoSet:** The AutoSet Segment Parameters based on Grade option colors all the haul roads based on their percent slope. This is an option to graphically see the slope of each road, simply by the color it is drawn in. This will also change the color of the 3D polylines in the plan view. They will stay on their original layers, so they can be set back to color ByLayer if desired. The slope interval was set here to a 1 or 2%, starting at -5%, as an example.

	If Grade is	Grade (%)	Color	Rolling Resistance (%)	Speed Limit (mph)
1	<	-5.00	Red	5.00	25
2	<	-3.00	Yellow	5.00	25
3	<	-2.00	Green	5.00	25
4	<	-1.00	Cyan	5.00	25
5	<	0.00	Blue	5.00	25
6	<	1.00	Black	5.00	25
7	<	2.00	Orange	5.00	25
8	<	3.00	Purple	5.00	25
9	<	5.00	Yellow-Orange	5.00	25

Notice in the image below, that the new colors are set by the percent slope of the road. These colors are now different than the initial image where they were colored by the layer.



Now that the haul roads are defined in the manager, they can be used in the haul road processing commands.

Haul Fleet Manager

This command manages the haul trucks that can be used in the Haul Cycle Analysis command. The main dialog of this command is shown below.

24 trucks are included in the program (more can be added manually), The default values are shown in yellow on this initial screen, and in the Edit Performance section, and are the same for each truck. All other values are specific to the truck type. Therefore, all truck specifications should be verified before implementing Haul Cycle Analysis.

- **Model #:** This is a customizable name that can be used to identify the truck model
- **Amount:** Haul Cycle Analysis allows you to calculate the potential production from a pre-defined fleet of trucks. Entering this value will determine the fleet to be considered when calculating production based on a given fleet
- **Loading Spot Time (sec):** This is the time in seconds, on average, the truck takes to position itself at the loading point
- **Loading Time (sec):** This is the time in seconds, on average, required to load the truck
- **Dumping Spot Time (sec):** This is the time in seconds, on average, the truck takes to position itself at the dumping point
- **Dumping Time (sec):** This is the time in seconds, on average, the truck requires to dump its load
- **Load Capacity (Payload):** This is the weight of the load the truck will haul

Trucks can be manually added or removed from the Haul Fleet Manager using the green Plus Button or the green Exit button, respectively. Both buttons are shown below.

Individual trucks can also be saved or loaded from a .hrt or .xml file using the green Save button or the Folder button. Both are shown below.

Selecting a truck and clicking 'Edit Performance' will open the dialog shown below. Yellow fields are default values, while the other fields are truck specific parameters.

An explanation of how each of these variables is implemented in the performance of the truck is discussed in more detail in the help documentation of the Haul Cycle Analysis command.

- **Traction Coefficient:** This value determines the force of friction that will act to prevent the truck tires from slipping
- **Normal Payload:** This value is dynamically linked to the Load Capacity of the truck shown on the main dialog of the Haul Fleet Manager; this value is dynamically linked to the Loaded Truck Weight and the Empty Truck Weight
- **Gross Vehicle Weight:** This value is NOT always a simple addition of the weight of the truck and the weight of the load. Instead, it is the intended operating weight of the truck as provided by the truck manufacturer. In many cases, however, the loaded truck weight will match this value
- **Max. Accel.:** This is the maximum allowable acceleration of the truck
- **Brake Reliance:** This is essentially a measure of confidence that can be placed on the defined traction coefficient to slow the truck in braking situations. For example, the default traction coefficient is 0.36 and the default brake reliance is 0.7. If the truck needs to come to an immediate halt, the maximum braking force is

determined in part by the traction coefficient. However if you are only 70% confident in the traction of the road, the maximum braking force would be calculated as if the traction coefficient were $0.36 \times 0.7 = 0.252$

- **Driver Perception Response Time (DPRT):** This is the usual time it will take for a truck driver to respond to a need to stop. Using the default value as an example, it would take a driver 2.5 seconds to actually press on the brake pedal once a threat presents itself.
- **Loaded Weight:** This is the actual operating weight of the truck when it carries a normal payload
- **Empty Weight:** This is the actual weight of the truck itself; this does not include the weight of the payload
- **Front Axle Weight Factor:** This is the portion of the truck weight that is supported by the front axle
- **Rear Axle Weight Factor:** This is the portion of the truck weight that is supported by the rear axle. This value and the Front Axle Weight Factor should sum to 1.0
- **Max. Decel.:** This is the maximum allowable deceleration of the truck
- **Brake System Response Time (BSRT):** When a truck brake is activated, the full braking force will not be immediately applied; the braking force will instead increase gradually. This value is the delay between the initial depression of the brake pedal and when the full braking force is applied

The Mine Speed Guidelines Table allows you to enter in the maximum allowable operating speed for a given Total Resistance. The default value is 50, and should be edited based on the mine guidelines. This table can be saved to and loaded from a .htr file. This dialog is shown below.

Mine Speed Guidelines Table

Truck Model: CAT_769D

Mine Speed Limits and Corresponding Equipment Usage

	Total Resistance	Loaded Oper. Speed	Empty Oper. Speed
1	-30.00	50.00	50.00
2	-25.00	50.00	50.00
3	-20.00	50.00	50.00
4	-15.00	50.00	50.00
5	-10.00	50.00	50.00
6	-5.00	50.00	50.00
7	.00	50.00	50.00
8	5.00	50.00	50.00
9	10.00	50.00	50.00

Buttons: OK, Cancel, Load, Save, Help

- **Total Resistance:** This is the force that works either for or against the truck's movement, expressed as a percentage of the truck weight. This is a combination of the grade of the road and the rolling resistance of the tires on the road. For example, a 5% rolling resistance on an uphill slope of 7% would equate to a total resistance of 12%
- **Operating Speed:** This is the maximum allowable speed of the truck. Unless another constraint requires the truck to move at a slower speed, the truck will accelerate until it reaches this speed

This table provides information on the rimpull force that will be exerted to maintain various speeds of the truck at normal operating conditions.

This information will be used when calculating the acceleration of the truck for various conditions. This table can be saved to and loaded from a .htp file.

This table allows you to set speed limits for both loaded and empty hauling conditions on downhill segments. The downgrade is the slope of the road expressed as a percentage. The dialog is shown below:

This table can be saved to and loaded from a .tdt file.

- Thank you to the late Peter Nahan of Goldcorp for his suggestions, references and help in developing this command.

Pull-Down Menu Location: Reserves/Timing in Surface Mining

Keyboard Command: haul_fleet

Haul Cycle Analysis

This command is used for detailed analysis of haul routes using various haul trucks to determine either 1) the required truck fleet required to meet a target production or 2) the attainable production based on a given haul fleet. This command requires the user to have executed the Haul Road Manager to define the haulage route. Although optional, it is recommended to define and customize truck parameters using the Haul Fleet Manager command to ensure the most accurate results. The first section of this document discusses the basic use of this routine, while the second section discusses the actual calculations that are performed to analyze the haul cycle.

First, you will be prompted to select the start and ending points of the haul road to be considered. The dialog that appears will show a spreadsheet view of all potential routes between the starting and ending point which will be sorted by distance in increasing order. The dialog will change slightly depending on the 'Calculate' option selected (either Fleet Size or Production).

	Haul	Length	#	Cycle time	Fleet Size	Fleet Capacity	Loaded Haul	Empty Haul
1	Road # 1	6.50	20	24.54	16	15.13	9.16	8.46
2	Road # 2	6.54	24	24.64	16	15.19	9.16	8.55
3	Road # 6	6.54	24	24.64	16	15.19	9.26	8.46
4	Road # 3	6.54	18	24.64	16	15.19	9.16	8.56
5	Road # 1	6.54	18	24.49	16	15.10	9.11	8.46
6	Road # 7	6.58	28	24.73	16	15.25	9.26	8.55
7	Road # 8	6.58	22	24.73	16	15.25	9.26	8.56
8	Road # 1	6.58	22	24.58	16	15.16	9.11	8.55

- **Preview Window:** highlights the currently selected haul route. This window works very similar to the 3D

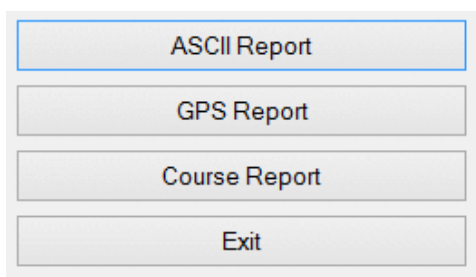
Viewer Window. Using the four icons next to this screen, you can pan, zoom, rotate, and zoom to the extents of the haul route.

- **Production Data Section:** allows you to choose the type and amount of haul trucks to be used for the cycle analysis. The 'Assign Single Truck Type' option will allow you select one of the trucks defined in the Haul Fleet Manager library. The 'Assign All Fleet Trucks' option will use the types and amounts of trucks defined in the same library. Clicking the 'Fleet' button will allow you to edit this library. Using the 'Assign All Fleet Trucks' option will only allow you calculate the attainable production rate. For more information on the Haul Fleet Manager, see the appropriate section of the help manual.
- **Truck Starts option:** allows you to specify if the selected starting point represents the loading or the dumping point of the route.
- **Use Density:** This option allows you specify the density of the material being hauled. When used, the tonnage of the material hauled will be included with the Process Spoil Timing option.
- **Calculate option:** changes the routine to calculate either the required fleet size based on a production target, or the attainable production target based on the haul fleet.
- **Production Rate:** This value can be set when calculating the required fleet size. This sets the target production the haul fleet must meet.
- **Number of trucks used:** This value can be set when calculating the attainable production when using the 'Assign Single Truck Type' option.
- **Estimated Queuing time:** This the estimated time that the truck must wait in line at either the loading or the dumping point per cycle (25 second wait at the loading point and 15 seconds at the dumping point would be a total Estimated Queuing time of 40 seconds).
- **Minutes per production hour:** This is the expected working time per hour. This is essentially a measure of work efficiency, and should therefore account for variable delays due to factors such as:
 - Excavating equipment delays
 - Personnel delays
 - Climate delays
- **Truck Filling Efficiency:** This is the fill percentage of the truck. A 90% filling efficiency applied to a truck that is intended to carry 100 tons will only carry 90 tons per cycle.
- **Truck Availability X Utilization:** This value is multiplied by the Minutes per Production Hour to determine the total working time of the truck. It is intended to account for fixed delays and mechanical availability of the truck. For a truck listed as working 50 minutes per production hour, an Availability X Utilization value of 90% will result in only 45 working minutes per hour.
- **Use Report Formatter:** This option will allow you use the Report Formatter for the Cycle Report and the Cycle Detail Report to customize the reported information.

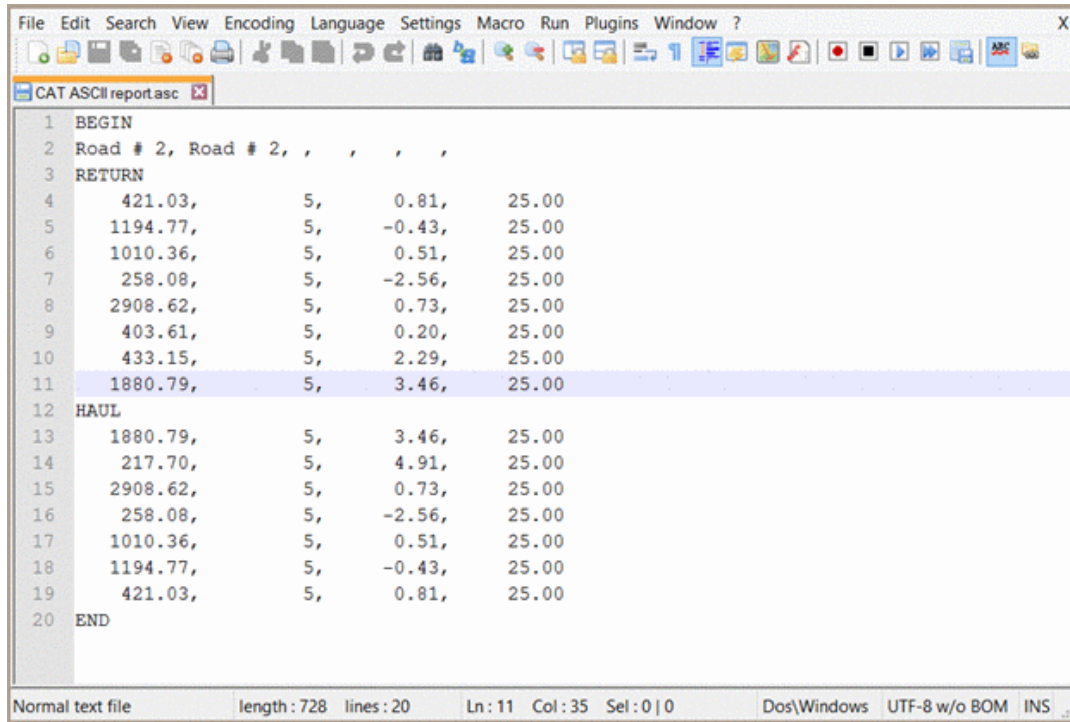
Reports

The Haul Cycle Analysis command is very quick and dynamic. Whenever a value is edited, the results in the spread sheet view will be automatically updated. These results can be reported in a variety of formats as discussed below.

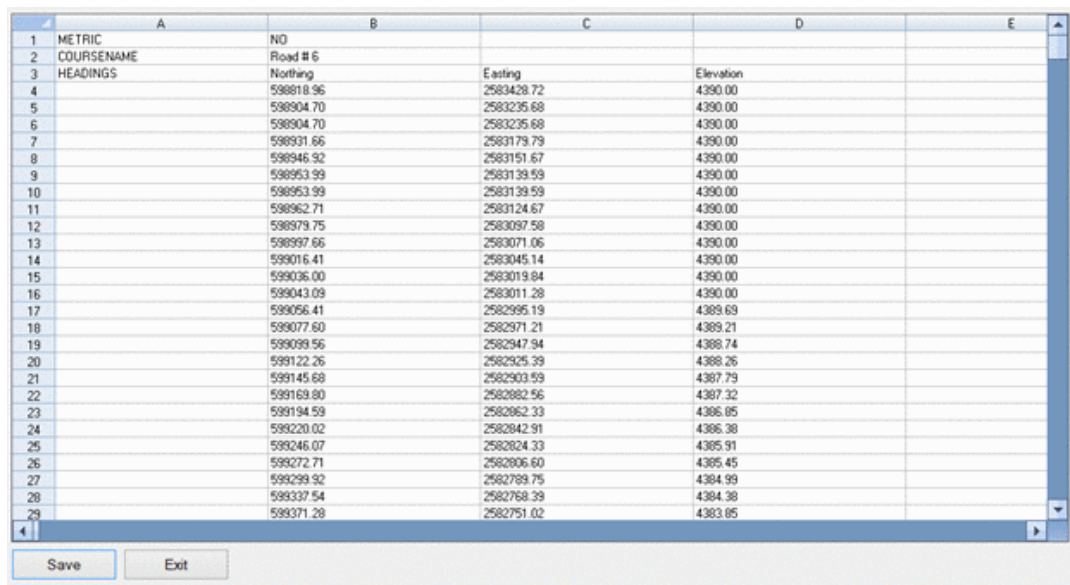
CAT FPC Reports: This option will allow you to export one of three report types: ASCII, GPS, and Course.



The ASCII report will export the segment distance, the rolling resistance, the slope, and the speed of the truck along each segment into an .asc file as shown below:



The GPS report will export the coordinates and elevation of the haul road at each vertex representing the road. This report can be saved as an Excel spreadsheet (.xls file) as shown below:



The Course Report is reported in a spreadsheet with three tabs: General Info, Haul, and Return. This file can also be saved as an Excel spreadsheet, as shown below:

	A	B	C	D	E
1	..BEGINCOURSE	Road # 1			
2	Distance (m)	Rolling Resistance %	% Grade	Speed Limit (kph)	Comments
3	100	4	0	100	Dump
4	352	4	-10	100	Ramp
5	200	4	0	100	Bridge
6	100	4	-5	40	Corner
7	1150	4	0	100	Along Highway
8	50	5	0	15	Around Loader
9	Total Length:	1952			
10					
11					
12					
13					
14					
15					
16					
17					
18					
19					
20					
21					
22					
23					
24					
25					
26					
27					
28					
29					

Cycle Report: This report gives basic information about the haul cycle.

Cycle Detail Report: This report provides more detailed information about the haul cycle, which in addition to the information in the standard report, provides information about the transition of speed along each segment, the rimpull applied on each road, and detailed information about the sub-segments which are used in the calculations.

```

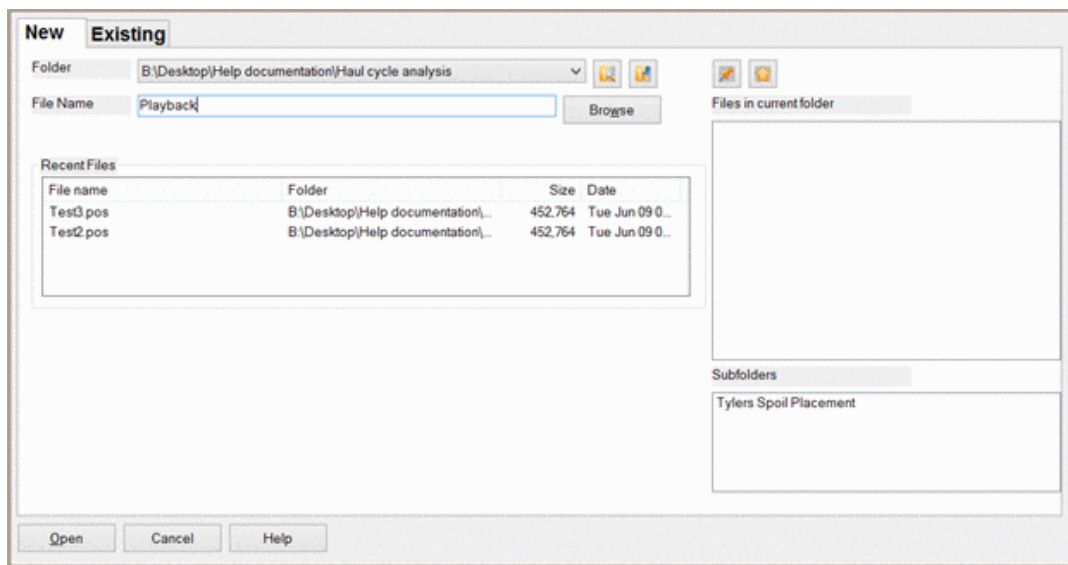
Haul Cycle Detail Report                               Mod Jun 10 09:47:42 2015
-----
HAULAGE DISTANCES
Haul Road: Road # 1
Depart: 1.95 km
Return: 1.95 km
Total : 3.90 km

Truck Model: CAT_769C

UNIT TIMES (min.)          PRODUCTION DETAILS
Loading: 2.92              Fleet Size: 1
Loaded Haul: 0.00         Haul truck model: CAT_769C
Dump Spotting: 1.00      Unit truck capacity: 40.00 tonnes
Dumping: 1.00            Truck filling efficiency: 100.00 %
Empty Haul: 0.00         Working minutes per production hour: 50.00 min.
Shovel Spotting: 1.00    Production cycle time: 6.92 min.
Dwelling time: 1.00      TRUCKS OPERATING: 1.00
CYCLE TOTAL: 6.92        Truck availability / utilization: 80.00 %
                          Production rate: 231.33 tonnes/hr
-----
HAULAGE SEGMENT DATA
Segment          Segment          Basic Information
#              Direction  Queuey  Status  Length  Grade  RollResist  TotalResist  CycleDuty  AveSpeed  Time  UserSet  Rimpull  Downgrade  Slowdown  Final  Initial  Speed  Maximum  Final  Rimpull  Grade  Fc
#              #              #              #              m        %        %           %           %           kph      min  kph      kph      kph      kph      kph      kph      kph      kph      kph      kph      %
-----
1  Depart  FALSE  Empty  |  50.00  0.00  0.00  |  0.00  0.00  1.00  0.00  |  15.00  25.00  |  44.00  15.00  15.00  |  0.00  0.00  0.00  |  -0.00  0.00
2  Depart  FALSE  Empty  |  1150.00  0.00  0.00  |  0.00  0.00  1.00  0.00  |  100.00  25.00  |  44.00  100.00  25.00  |  0.00  0.00  0.00  |  0.00  0.00
3  Depart  FALSE  Empty  |  100.12  0.00  0.00  |  0.00  0.00  1.00  0.00  |  40.00  25.00  |  44.00  40.00  25.00  |  0.00  0.00  0.00  |  0.00  0.00
4  Depart  FALSE  Empty  |  200.00  0.00  0.00  |  0.00  0.00  1.00  0.00  |  100.00  25.00  |  44.00  100.00  25.00  |  0.00  0.00  0.00  |  0.00  0.00
5  Depart  FALSE  Empty  |  351.75  0.00  0.00  |  0.00  0.00  1.00  0.00  |  100.00  25.00  |  44.00  100.00  25.00  |  0.00  0.00  0.00  |  0.00  0.00
6  Depart  FALSE  Empty  |  100.00  0.00  0.00  |  0.00  0.00  1.00  0.00  |  100.00  25.00  |  44.00  100.00  25.00  |  0.00  0.00  0.00  |  0.00  0.00
-----
7  Return  FALSE  Loaded  |  100.00  0.00  0.00  |  0.00  0.00  1.00  0.00  |  100.00  25.00  |  44.00  100.00  25.00  |  0.00  0.00  0.00  |  0.00  0.00
8  Return  FALSE  Loaded  |  351.75  0.00  0.00  |  0.00  0.00  1.00  0.00  |  100.00  25.00  |  44.00  100.00  25.00  |  0.00  0.00  0.00  |  0.00  0.00
9  Return  FALSE  Loaded  |  200.00  0.00  0.00  |  0.00  0.00  1.00  0.00  |  100.00  25.00  |  44.00  100.00  25.00  |  0.00  0.00  0.00  |  0.00  0.00
10 Return  FALSE  Loaded  |  100.12  0.00  0.00  |  0.00  0.00  1.00  0.00  |  40.00  25.00  |  44.00  40.00  25.00  |  0.00  0.00  0.00  |  -0.00  0.00
11 Return  FALSE  Loaded  |  1150.00  0.00  0.00  |  0.00  0.00  1.00  0.00  |  100.00  25.00  |  44.00  100.00  25.00  |  0.00  0.00  0.00  |  0.00  0.00
12 Return  FALSE  Loaded  |  50.00  0.00  0.00  |  0.00  0.00  1.00  0.00  |  15.00  25.00  |  44.00  15.00  15.00  |  0.00  0.00  0.00  |  -0.00  0.00
-----
| 3903.74 | 0.00  1.00  0.00 | | |

```

Create Flyby Log: This option will create a .pos file which can be reviewed with the Surface 3D Flyover command in the Civil module. You will be prompted to enter a road width, then you will be allowed to specify the file name.



Process Spoil Timing: This command will perform a timing analysis on an exported .sph file from Spoil Placement Timing. This will give a more detailed analysis of the haul cycle during each time period of the spoil placement. An example report is provided below:

ID	Volume (C.M.)	Source Name	Destination Name	Starting Date	End Date	Fleet Size	Total Days	Haul Cycle Time (min)	Production Rate (tonnes/Hr)	Loaded Haul Distance (km)	Empty Haul Distance (km)	Total Distance (km)	Source Centroid X	Source Centroid Y	Source Centroid Z
1	13,438	Site-Pit 2-14-B1-Site 1-Pile 1-B1	Site-Pit 2-14-B1-Site 1-Pile 1-B1	01/01/2011	01/01/2011	8	0.56	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
2	16,980	Site-Pit 2-10-B1-Site 1-Pile 1-B1	Site-Pit 2-10-B1-Site 1-Pile 1-B1	01/01/2011	01/02/2011	8	0.71	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
3	8,966	Site-Pit 2-6-B1-4-Site 1-Pile 1-B1	Site-Pit 2-6-B1-4-Site 1-Pile 1-B1	01/02/2011	01/03/2011	8	0.37	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
4	45	Site-Pit 2-2-B1-4-Site 1-Pile 1-B1	Site-Pit 2-2-B1-4-Site 1-Pile 1-B1	01/03/2011	01/03/2011	8	0.00	12.26	997.15	0.67	0.67	1.33	0.00	0.00	0.00
5	221	Site-Pit 2-13-B1-Site 1-Pile 1-B1	Site-Pit 2-13-B1-Site 1-Pile 1-B1	01/03/2011	01/03/2011	8	0.01	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
6	45	Site-Pit 2-17-B1-Site 1-Pile 1-B1	Site-Pit 2-17-B1-Site 1-Pile 1-B1	01/03/2011	01/03/2011	8	0.00	12.26	1000.09	0.67	0.67	1.33	0.00	0.00	0.00
7	180,463	Site-Pit 2-20-B2-Site 1-Pile 1-B1	Site-Pit 2-20-B2-Site 1-Pile 1-B1	01/03/2011	01/10/2011	8	7.52	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
8	154,015	Site-Pit 2-19-B2-Site 1-Pile 1-B1	Site-Pit 2-19-B2-Site 1-Pile 1-B1	01/10/2011	01/16/2011	8	6.42	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
9	230,481	Site-Pit 2-18-B2-Site 1-Pile 1-B1	Site-Pit 2-18-B2-Site 1-Pile 1-B1	01/16/2011	01/26/2011	8	9.60	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
10	183,925	Site-Pit 2-15-B2-Site 1-Pile 1-B1	Site-Pit 2-15-B2-Site 1-Pile 1-B1	01/26/2011	02/03/2011	8	7.66	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
11	308,709	Site-Pit 2-14-B2-Site 1-Pile 1-B1	Site-Pit 2-14-B2-Site 1-Pile 1-B1	02/03/2011	02/16/2011	8	12.86	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
12	245,029	Site-Pit 2-11-B2-Site 1-Pile 1-B1	Site-Pit 2-11-B2-Site 1-Pile 1-B1	02/16/2011	02/26/2011	8	10.21	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
13	284,507	Site-Pit 2-10-B2-Site 1-Pile 1-B1	Site-Pit 2-10-B2-Site 1-Pile 1-B1	02/26/2011	03/10/2011	8	11.85	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
14	222,740	Site-Pit 2-7-B2-4-Site 1-Pile 1-B1	Site-Pit 2-7-B2-4-Site 1-Pile 1-B1	03/10/2011	03/19/2011	8	9.28	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
15	167,181	Site-Pit 2-6-B2-4-Site 1-Pile 1-B1	Site-Pit 2-6-B2-4-Site 1-Pile 1-B1	03/19/2011	03/26/2011	8	6.97	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
16	142,625	Site-Pit 2-3-B2-4-Site 1-Pile 1-B1	Site-Pit 2-3-B2-4-Site 1-Pile 1-B1	03/26/2011	04/01/2011	8	5.94	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
17	127,774	Site-Pit 2-2-B2-4-Site 1-Pile 1-B1	Site-Pit 2-2-B2-4-Site 1-Pile 1-B1	04/01/2011	04/06/2011	8	5.32	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
18	106,873	Site-Pit 2-1-B2-4-Site 1-Pile 1-B1	Site-Pit 2-1-B2-4-Site 1-Pile 1-B1	04/06/2011	04/11/2011	8	4.45	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
19	154,002	Site-Pit 2-4-B2-4-Site 1-Pile 1-B1	Site-Pit 2-4-B2-4-Site 1-Pile 1-B1	04/11/2011	04/17/2011	8	6.42	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
20	163,642	Site-Pit 2-5-B2-4-Site 1-Pile 1-B1	Site-Pit 2-5-B2-4-Site 1-Pile 1-B1	04/17/2011	04/24/2011	8	6.82	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00
21	175,398	Site-Pit 2-8-B2-4-Site 1-Pile 1-B1	Site-Pit 2-8-B2-4-Site 1-Pile 1-B1	04/24/2011	05/01/2011	8	7.31	12.26	1000.00	0.67	0.67	1.33	0.00	0.00	0.00

It is worth mentioned the way that some of the values included in the report are calculated. These are provided below

- *HaulCycle-Lift (Source):* This value is the change in elevation from the pit centroid to the start of the road
- *HaulCycle-Lift (Road):* This value is the change in elevation from the start of the road to the end of the road
- *HaulCycle-Lift (Destination):* This value is the change in elevation from the end of the road to the centroid of the destination
- *HaulCycle-Lead (Source):* This value is the horizontal distance from the pit centroid to the start of the road, plus half the width and half the length of the pit. The half width/length of the pit is included to account for additional ramp length required to travel out of the pit.
- *HaulCycle-Lead (Road):* This value is the horizontal distance of the road.
- *HaulCycle-Lead (Source):* This is the value is the horizontal distance from the end of the road to the centroid of the spoil pile, plus half the width and half the length of the pile. The width/length of the pile is included to account for additional ramp length required to travel to the top of the pile.

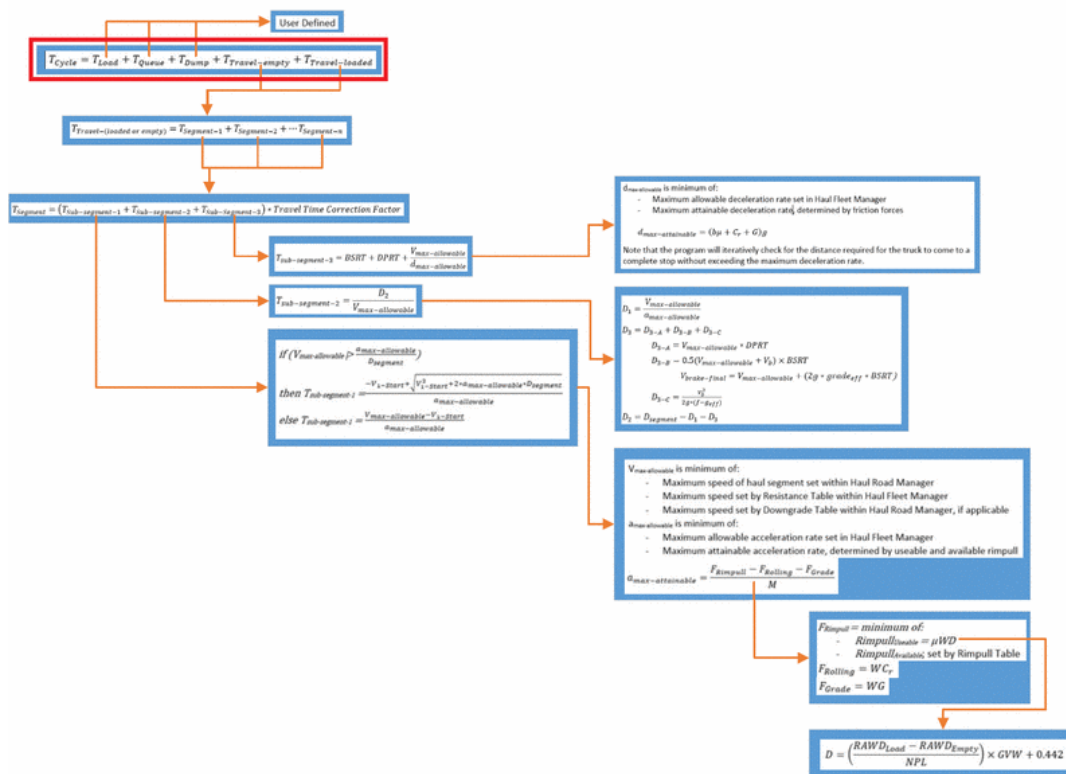
Calculation Methods

A good planner will be cautious about blindly accepting calculations. For this reason, the following section details the logic and calculations used in Haul Cycle Analysis so that the user can fully understand how the routine works.

As previously stated, this routine seeks to calculate either the attainable production based on the truck fleet or the fleet size required to meet a target production. Actually determining these values is rather simple once to the true cycle time has been calculated. However, calculation of the cycle time considers many factors. The first sub-section of this section details how the program calculates cycle time. The following sub-section details how the attainable production and required fleet size are calculated based on the calculated cycle time.

Calculation of Cycle Time

The diagram shown below is visual outline of the equations used to calculate the haul cycle time. This is provided to allow for a better understanding of how Haul Cycle Analysis uses input variables to determine various operating parameters. Note that this diagram is simplified, and does not detail all of the logic used in the actual routine.



The total haul cycle time can be calculated with the following equation:

$$T_{Cycle} = T_{Load} + T_{Travel-loaded} + T_{Queue} + T_{Dump} + T_{Travel-empty} \text{ where}$$

- T_{Cycle} is the total cycle time
- T_{Load} is the time required to load the truck
- $T_{Travel-loaded}$ is the time required for the truck to travel from the loading point to the dumping point
- T_{Queue} is the total estimated queuing time (sum of queuing time at both the loading point and the dumping point)
- T_{Dump} is the time required for the truck to dump its payload

- TTravel-empty is the time required for the truck to return to the loading point from the dumping point

Three variables, TLoad, TDump, and TQueue, are manually entered. The remaining variables, TTravel-loaded and TTravel-empty, are calculated similarly to one another, but with different values based on the road profile and truck specifications for loaded and empty scenarios. TTravel-loaded and TTravel-empty are calculated with the following formula: $T_{Travel-(loaded\ or\ empty)} = T_{Segment-1} + T_{Segment-2} + \dots + T_{Segment-n}$

where

TSegment-1 is the travel time of the first segment of the haul route

TSegment-2 is the travel time of the second segment of the haul route

Etc.

The travel time for a single haul segment will be further divided into three sub-segments according to the following chart.

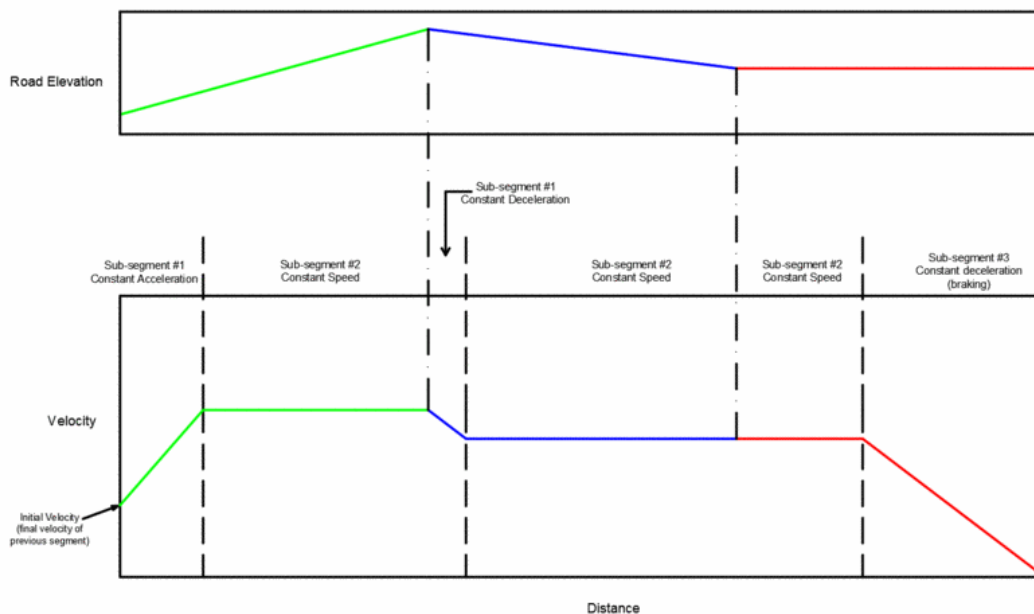
Sub-segment #1: the truck is accelerating/decelerating at a constant rate

Sub-segment #2: the truck is traveling at a constant velocity

Sub-segment #3: the truck is braking. Note that sub-segment #3 is only used when the truck comes to a complete stop.

In the below picture:

- The top portion of colored lines represent the road profile, broken into three haul segments
- The lower portion of colored lines represent the velocity of the truck along the haul road
- Haul Segment 1 (green line)
 1. Sub-segment #1 shows the truck accelerating as it travels uphill to meet the new road speed limit
 2. Sub-segment #2 shows the truck traveling at a constant velocity once it reaches the road speed limit
- Haul Segment 2 (blue line)
 1. Sub-segment #1 shows the truck decelerating as it starts to travel downhill to meet the new road speed limit
 2. Sub-segment #2 shows the truck traveling at a constant velocity once it reaches the road speed limit
- Haul Segment 3 (red line)
 1. Sub-segment #2 shows the truck traveling at a constant velocity. It is traveling at the speed limit when it approaches Haul Segment 3.
 2. Sub-segment #3 shows the truck braking as the truck comes to a complete stop.



The total travel time for a haul segment is simply a sum of the sub-segment travel times, as shown below:

$$T_{\text{Segment}} = (T_{\text{Sub-segment-1}} + T_{\text{Sub-segment-2}} + T_{\text{Sub-segment-3}})$$

The travel time of each sub-segment is calculated using the following formulae:

$$T_{\text{sub-segment-1}} = \frac{V_{\text{max-allowable}} - V_1}{a_{\text{max-allowable}}}$$

$$T_{\text{sub-segment-2}} = \frac{D_2}{V_{\text{max-allowable}}}$$

$$T_{\text{sub-segment-3}} = \text{BSRT} + \text{DPRT} + \frac{V_{\text{end}} - V_{\text{max-allowable}}}{a_{\text{max-allowable}}}$$

where

- TSub-segment-1 is the travel time of sub-segment #1
- Vmax-allowable is the maximum allowable truck velocity along the haul segment
- V1-Start is the truck velocity at the start of sub-segment #1, equal to zero at the beginning of the haul route
- amax-allowable is the maximum allowable acceleration of the truck on sub-segment #1
- TSub-segment-2 is the travel time of sub-segment #2
- D2 is the distance of sub-segment #2
- TSub-segment-3 is the travel time of sub-segment #3
- BSRT is the Brake System Response Time
- DPRT is the Driver Perception Response Time
- dmax-allowable is the maximum allowable deceleration rate of the truck

Sub-Segment #1 Supplement

Vmax-allowable is minimum of:

- Maximum speed of haul segment set within Haul Road Manager
- Maximum speed set by Resistance Table within Haul Fleet Manager
- Maximum speed set by Downgrade Table within Haul Road Manager, if applicable

amax-allowable is minimum of:

- Maximum allowable acceleration rate set in Haul Fleet Manager
- Maximum attainable acceleration rate, determined by useable and available rimpull

Note that a truck's velocity does not necessarily have to increase during sub-segment #1. If a truck enters a haul segment at a speed which exceeds the maximum allowable velocity for the haul segment, then sub-segment #1 will refer to a decelerating situation.

Sub-Segment #2 Supplement

In the final report, sub-segment distances will be reported as D1, D2, and D3, which refer to the distances of the respective sub-segments. The distance for each sub-segment is calculated as shown below. Here it is also important to note that the sub-segment #3 is further divided into three sections which represent A) the time required for the driver to realize that a stop is required, B) the time from when the driver first applies the brake to the time when the brake system applies the full braking force, and C) the time required for the truck to stop once the full braking force is applied.

$$D_1 = \frac{V_{max-allowable}}{a_{max-allowable}}$$

$$D_3 = D_{3-A} + D_{3-B} + D_{3-C}$$

$$D_{3-A} = V_{max-allowable} * DPRT$$

$$D_{3-B} = 0.5(V_{max-allowable} + V_b) \times BSRT$$

$$V_{brake-final} = V_{max-allowable} + (2g * grade_{eff} * BSRT)$$

$$D_{3-C} = \frac{V_b^2}{2g*(f-g_{eff})}$$

$$D_2 = D_{segment} - D_1 - D_3$$

Note that in many situations one or two of these sub-segments will not be used in the final calculations. For example, if a truck enters a new haul segment and does not need to accelerate/decelerate to meet the speed limit, the distance and travel time for sub-segments #1 and #3 will be zero. As another example, sub-segment #3 will only be used when the truck is braking to a complete stop.

where

- D1 is the distance of sub-segment #1
- D2 is the distance of sub-segment #2
- D3 is the distance of sub-segment #3
- D3-A is the distance of sub-segment #3, section A
- D3-B is the distance of sub-segment #3, section B
- D3-C is the distance of sub-segment #3, section C
- DSegment is the length of the current haul segment
- Vb is the truck velocity at the end of the Brake System Response Time
- g is the acceleration due to gravity
- f is the coefficient of friction available for braking (
- geff is the effective grade, which is a sum of the road grade and the rolling resistance of the road on the tires

Sub-segment #3 Supplement

dmax-allowable is minimum of:

- Maximum allowable deceleration rate set in Haul Fleet Manager
- Maximum attainable deceleration rate, determined by friction forces

$$d_{max-attainable} = (b\mu + C_r + G)g$$

Note that the program will iteratively check for the distance required for the truck to come to a complete stop without exceeding the maximum deceleration rate.

Max Truck Acceleration/Deceleration

Both sub-segments #1 and #3 must check for the maximum allowable acceleration/deceleration rate, and one of the rates to be considered is the rate limited by friction, also referred to as the useable rimpull. The below equations show how the acceleration/deceleration rates are calculated.

As previously shown, Haul Cycle Analysis analyzes truck movement by constant acceleration/deceleration along sub-segments and sections of sub-segments. For this reason, the acceleration of the truck can be defined by scalar quantities in Newton's Second Law of motion, shown below and simplified:

$$F = Ma_{max-attainable} = F_{Rimpull} - F_{Rolling} - F_{Grade}$$

$$a_{max-attainable} = \frac{F_{Rimpull} - F_{Rolling} - F_{Grade}}{M}$$

where

F is the sum of all forces acting on the truck

M is the mass of the truck

amax-attainable is the maximum possible acceleration of the truck

FRimpull is the effective rimpull force (minimum of available rimpull and useable rimpull force)

FRolling is the force of rolling resistance, which is set within the Haul Road Manager for each haul segment

FGrade is the grade resistance force, defined as positive for uphill and negative for downhill movement

FRimpull

$$F_{Rimpull} = \text{minimum of:}$$

- $Rimpull_{Useable} = \mu WD$
- $Rimpull_{Available}$; set by Rimpull Table

Available Rimpull is determined from standard Rimpull Curves provided by truck manufacturers. This data is entered into the Haul Fleet Manager in a tabular format in the Rimpull Table. From the entered values, the program will determine the available rimpull at various operating speeds.

Useable rimpull is the maximum force the truck can apply before the tires start to slip on the road. This value is calculated with the following equation

$$Rimpull_{Useable} = \mu WD$$

where

- RimpullUseable is the useable rimpull force
- μ is the traction coefficient between the road and the tires, which is set in the Haul Fleet Manager
- W is the truck weight
- D is the ratio of the truck weight on the rear axle

Furthermore, D can be calculated with the following equation

$$D = \left(\frac{RAWD_{Load} - RAWDE_{Empty}}{NPL} \right) \times GVW + 0.442$$

where

RAWDLoad - drive (rear) axle weight distribution percentage in loaded condition, expressed as a decimal

RAWDEEmpty - drive (rear) axle weight distribution percentage in empty condition, expressed as a decimal

NPL - nominal payload

GVW - gross vehicle weight; note that will not always equal the loaded truck weight

F_{Rolling}

$$F_{Rolling} = WC_r$$

Where

Cr is the haul segment rolling resistance, expressed as a decimal; this is set in the Haul Road Manager

F_{Grade}

$$F_{Grade} = WG$$

Although this calculation does not perfectly calculate the resistive/assistive force due to the grade of the road, the difference between the calculated value and the 'true' value is considered insignificant. Once the maximum possible acceleration of the truck has been calculated, this value is compared to the maximum acceleration limit set in the Haul Fleet Manager. The minimum of these two values is used for the calculation of the distance and travel time of sub-segment #1 for each haul segment.

Maximum Deceleration

When a truck slows down, it is limited to using the lesser of two possible deceleration rates. One rate is defined directly in the Haul Fleet Manager. The other rate is calculated based on Newton's Second Law using the below formulae:

$$F_{Available} = Md_{max-attainable} = F_{Retard} + F_{Rolling} + F_{Grade} = b\mu Wg + C_r Wg + GWg$$

$$d_{max-attainable} = (b\mu Wg + C_r Wg + GWg) / M = (b\mu + C_r + G)g$$

where

- F_{Available} is the total available retarding force
- d_{max-attainable} is the maximum attainable retarding deceleration rate
- M is the mass of the truck
- F_{Retard} is the retarding force
- F_{Rolling} is the rolling resistance force
- F_{Grade} is the grade resistance force
- b is the braking reliance on traction
- μ is the traction coefficient
- Cr is the rolling resistance coefficient
- G is the road grade; uphill is positive, downhill is negative
- g is the acceleration due to gravity
- W is the truck weight

Calculation of Required Fleet Size and Attainable Production

Once the total cycle time has been calculated, the calculation of the required fleet size/attainable production is very simple. The calculations for these values are shown below

$$\text{Attainable Production} = \sum N_{\text{trucks}-i} \frac{NPL_i * \text{Mins}_{\text{hr}} * \text{Fill} * \text{Avail_Util}}{T_{\text{Cycle}-i}}$$
$$\text{Required Fleet Size} = \frac{\text{Target Production Rate}}{\left(\frac{NPL * \text{Mins}_{\text{hr}} * \text{Fill} * \text{Avail_Util}}{T_{\text{Cycle}}} \right)}$$

Where

- NTrucks-i is the number of trucks of a specific model
- NPLi is the nominal payload of a specific truck model
- Minshr is the minutes working per production hour
- Fill is the truck filling efficiency
- Avail_Util is the truck availability multiplied by the truck utilization
- TCycle-i is the truck cycle time for a specific truck model

Note that the total attainable production is the sum of each attainable production rate of each truck in the haul fleet.

References:

Parreira, Julianna. "An Interactive Simulation Model to Compare an Autonomous Haulage Truck System with a Manually-Operated System." Diss. THE UNIVERSITY OF BRITISH COLUMBIA (VANCOUVER), 2013. Print.

U.S. Department of Labor. MSHA Haul Road Inspection Handbook. June 1999. Handbook Number PH99-I-4.

Prompts

Delay: Intersection Merge (will be a pre-named delay name)

[near on] Pick start point on Haul Road: select where the truck will begin, usually where it gets loaded at.

[near on] Pick end point on Haul Road: select where the truck will end, usually where it unloads/dumps at.

Pull-Down Menu Location: Reserves/Timing in Surface Mining

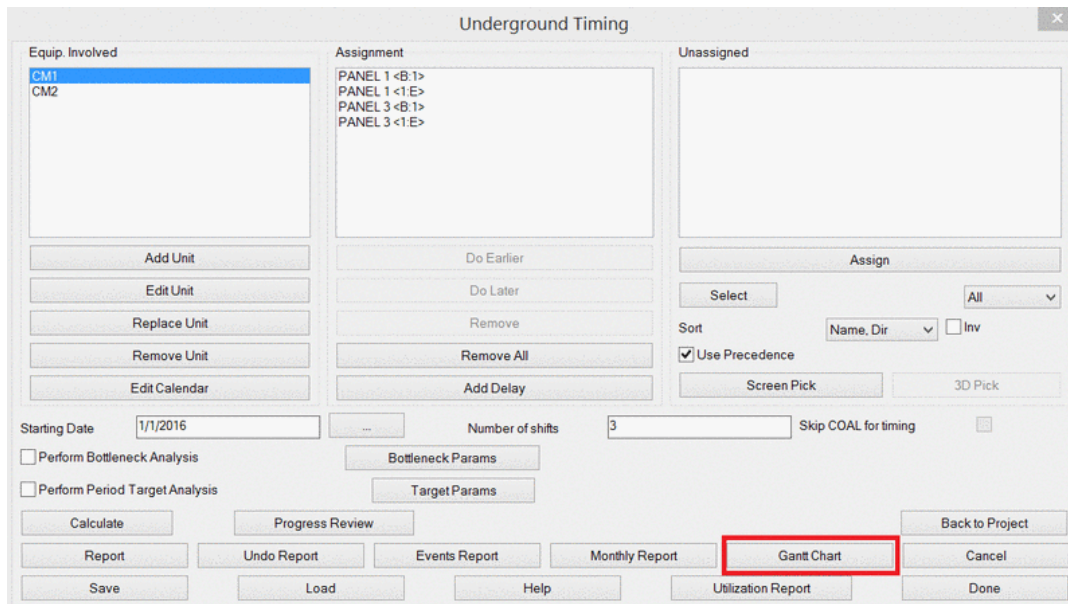
Keyboard Command: haul_cycle

Gantt Charts

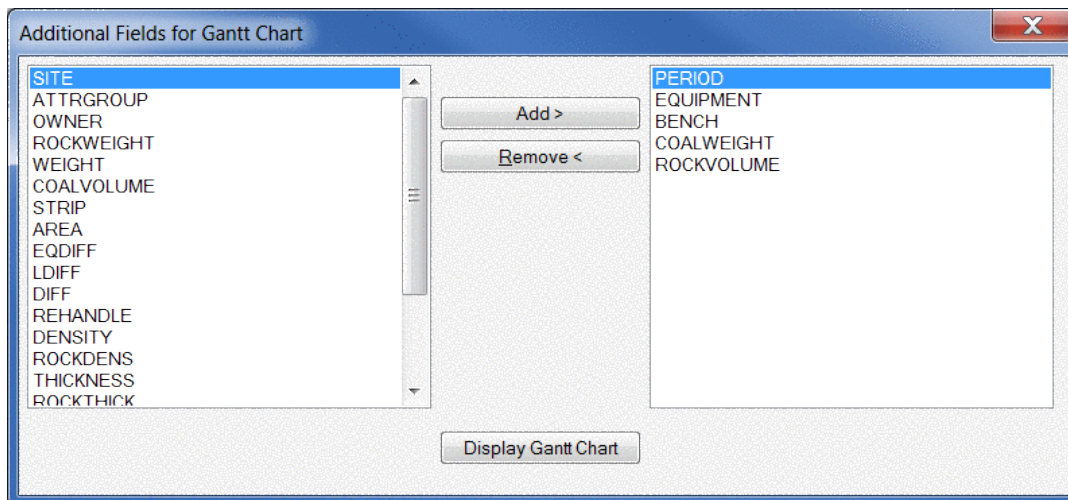
Gantt Charts are used to show the starting and ending dates for scheduled events, making it simpler to understand the progress of the project. These charts can be created for scheduling commands including Underground Timing,

Surface Equipment Timing, and Spoil Placement Timing. Underground Timing and Surface Equipment Timing are very similar commands, and thus Gantt charts are made in the same way for both commands. The below example shows Gantt charts used with these commands. Gantt charts for Spoil Placement Timing appear on a slightly different dialog, but all functionality is the same.

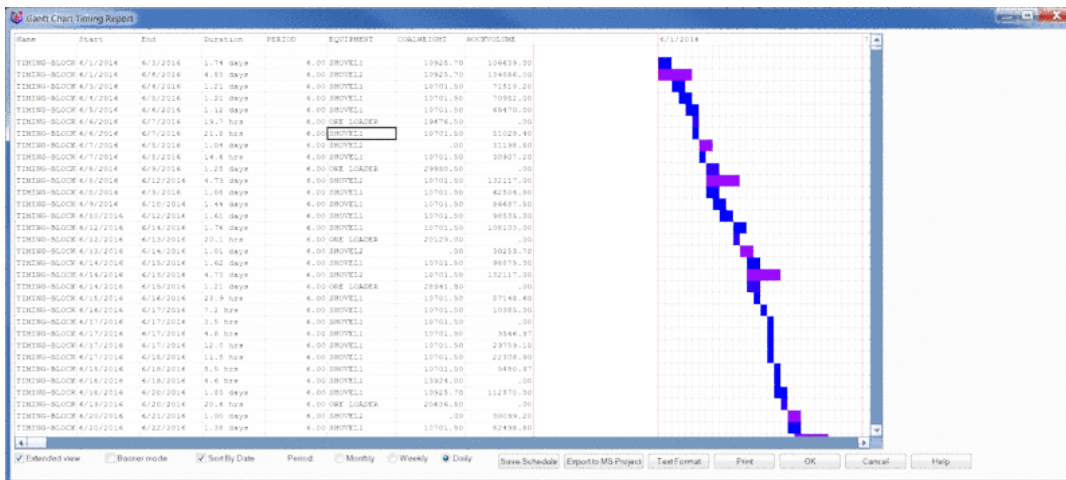
In the Underground and Surface Equipment Timing dialog shown below, panels have been assigned to equipment. Clicking **Calculate** will make the **Gantt Chart** button available, as outlined in red in the below image.



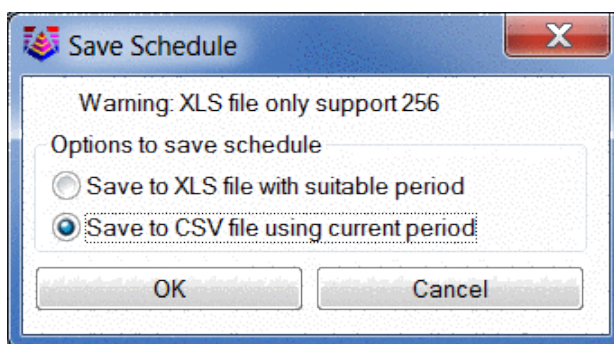
Clicking the **Gantt Chart** button will open the dialog for Additional Fields for Gantt Chart shown below. This screen allows to Add> items from left to right to include in the chart report.



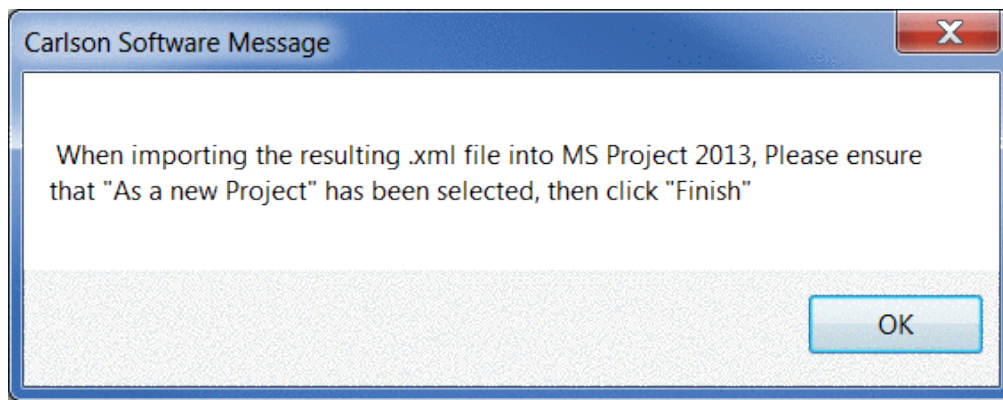
After adding the selected attributes, choosing the Display Gantt Chart brings up the report screen shown here.



- **Extended View:** This option will toggle the first four columns of the Gantt chart. If this option is turned on, the Gantt chart will be shown as above. If this option is turned off, the first four columns of the Gantt chart will not be shown.
- **Banner Mode:** This option changes how the chart is printed. When this option is turned on, the chart will be printed on multiple pages. This maximizes the chart on each page and makes it more readable, which is very helpful when printing a complex schedule. When this option is turned off, the chart will be scaled to fit on a single sheet. For complex schedules, this can result in an unreadable chart. However, you can change the size of the print paper if you would like to keep the chart on a single sheet.
- **Sort By Date:** This option sorts the rows by the dates of the schedule. This is useful to see all the equipment grouped together in the chart.
- **Period:** This option sets the time duration of each cell of the Gantt chart. In the above example, the **Daily** option is used. In this case, each cell of the Gantt chart represents one day.
- **Save Schedule:** This option will allow you to save the Gantt chart to an Excel spreadsheet that is limited to 256 fields, or to a CSV file, which is unlimited.



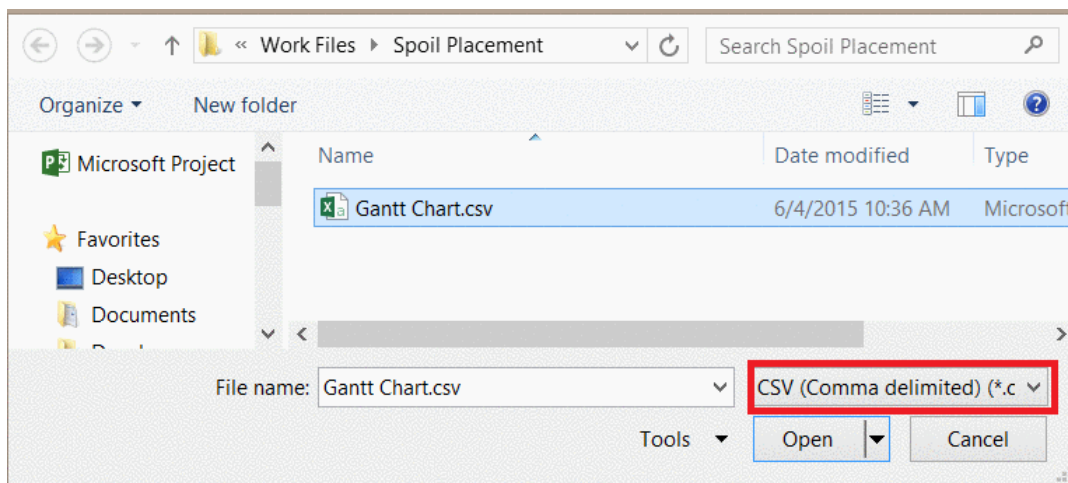
- **Export to MS Project:** This option will allow you to export the Gantt chart to an XML file, which can be imported into Microsoft Project. A detailed description of this process is provided further down in this document.



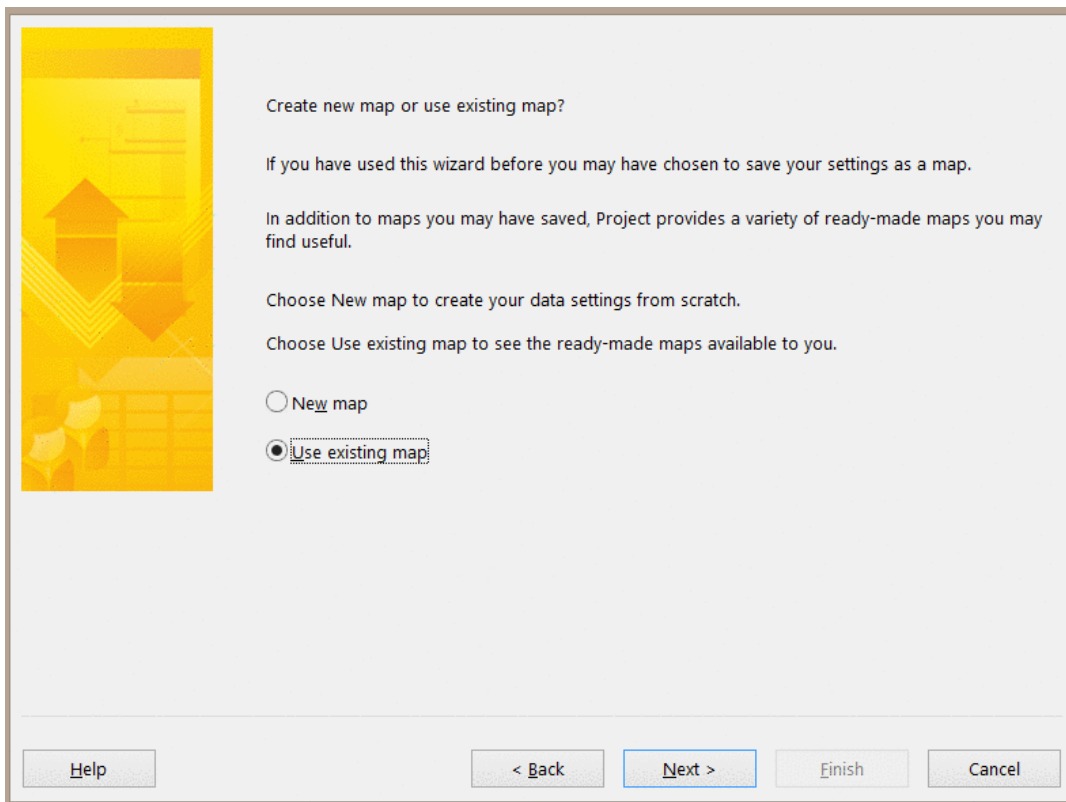
- **Text Format:** This option allows you to set the format of the text of the exported chart.
- **Print:** This option allows you to print the Gantt chart to any of the printers installed on your computer, or to save it as PDF file.

Opening CSV file with MS Project

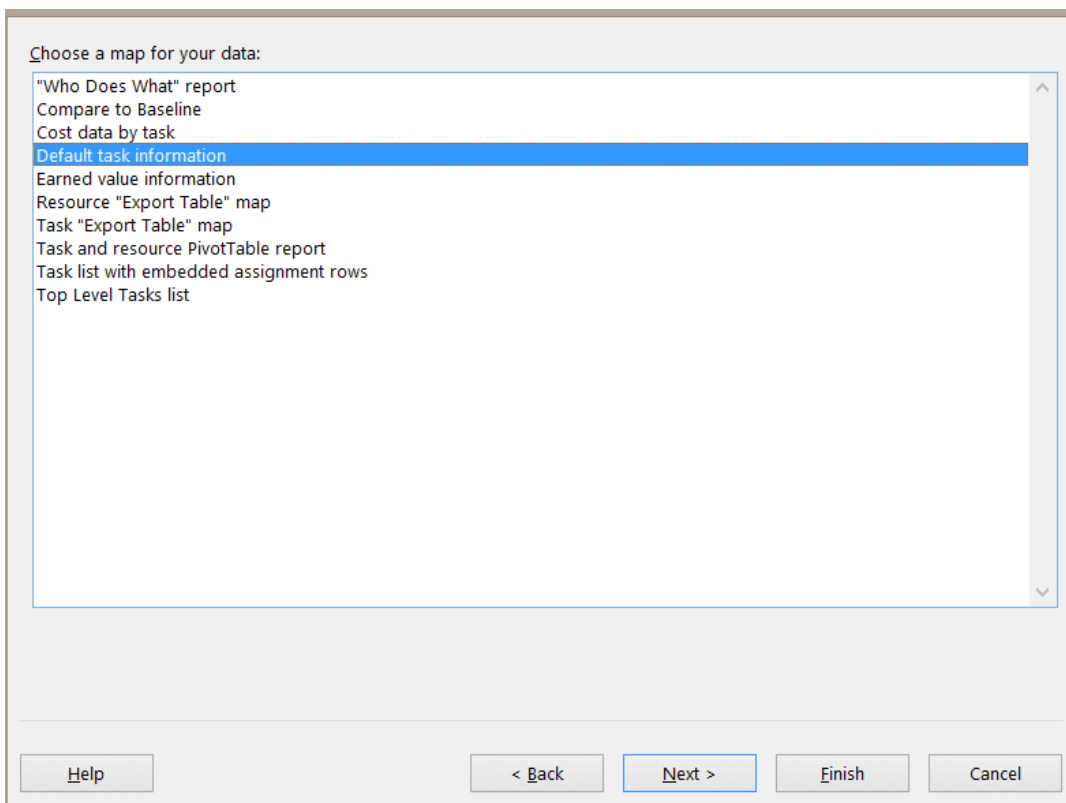
This section describes how to open a CSV file with MS Project. Before opening the .csv file, you will need to enable Legacy Formats. This can be done via File Menu > Options > Trust Center > Trust Center Settings > Legacy Formats > Allow loading files with legacy or non-default file formats. Note that this refers to Microsoft Project 2013. With this setting enabled, you may open an existing file and specify the format as a .csv as shown below:



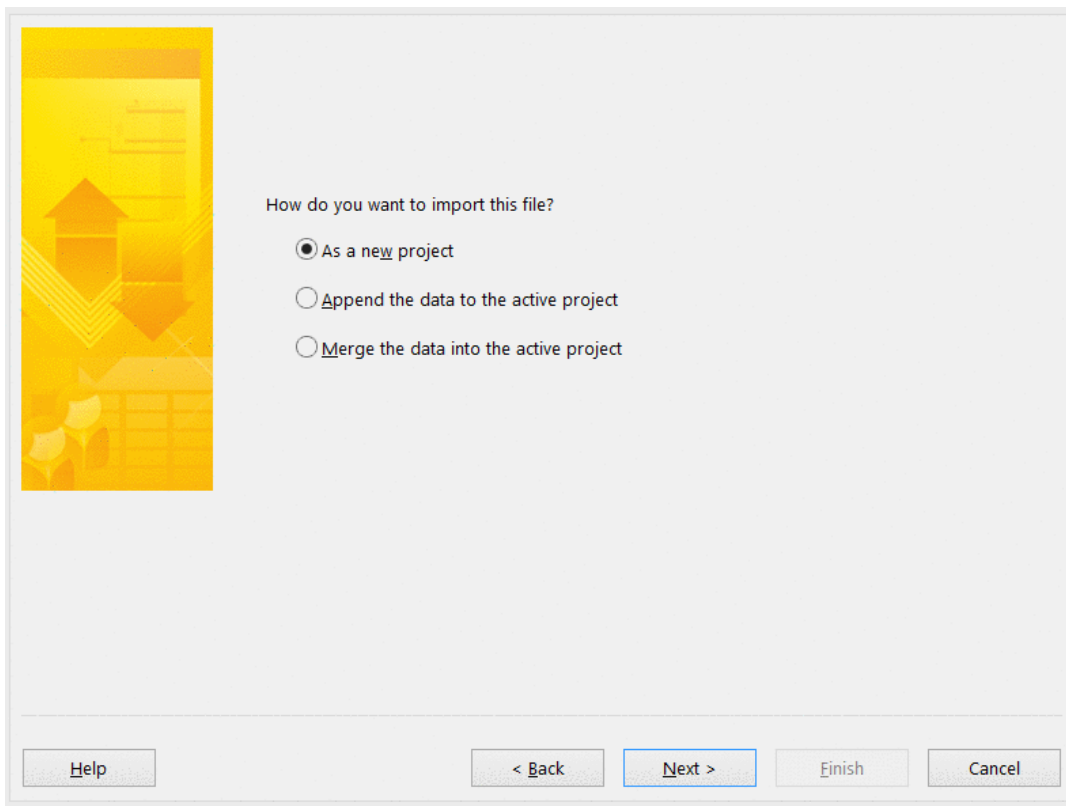
This will open the Import Wizard. On the **Map** dialog, select **Use existing map** and click **Next**.



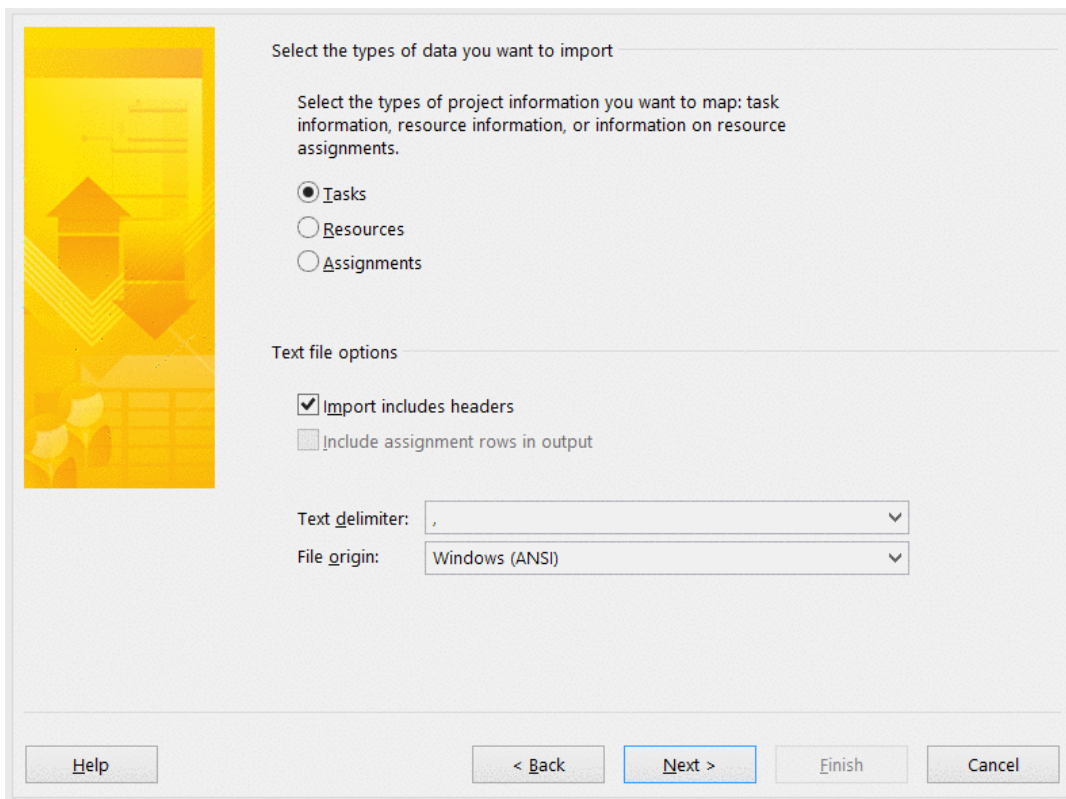
On the **Map Selection** dialog, select **Default Task Information** and click **Next**.



On the **Import Mode** dialog, select **As a new project** and click **Next**.

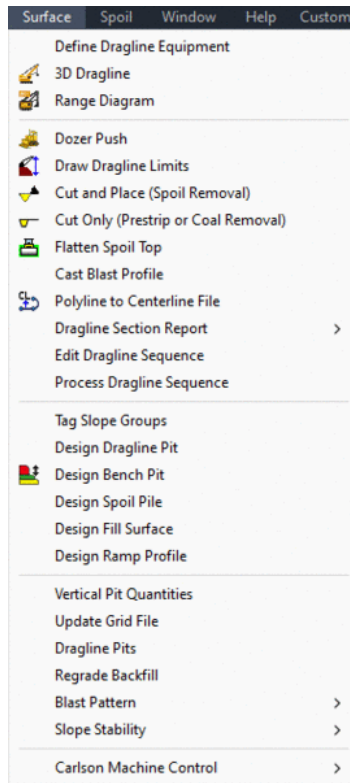


On the **Map Options** dialog, use the settings shown below and click **Finish**.



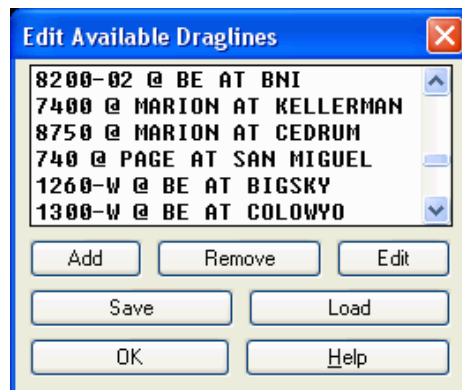
Surface Menu

The Surface menu has commands for cross section and surface modeling.



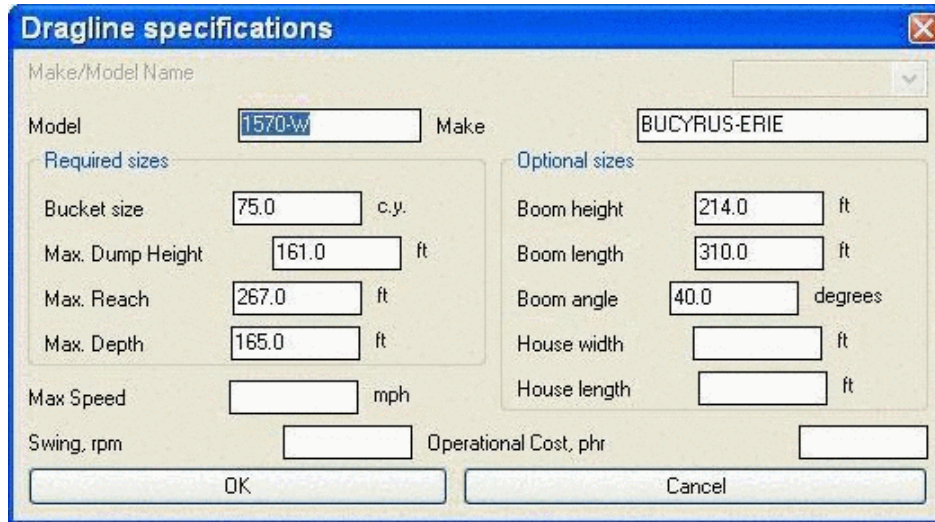
Define Dragline Equipment

The description of available draglines is created and edited using this function. While the most important parameters are those which define dragline operational range for the Cut and Place commands, there is a number of others which are used in the 3D dragline modeling. Bucket size is used to calculate a number of cycles needed to transport a given amount of material, which is used to get the time spent on material handling, together with swing speed and calculated average swing angle. Dragline house sizes are used to estimate how the dragline travels within a given area, while maximum speed gives the time required to complete these moves. Finally, if operational cost is specified, the total cost will appear in the final report. The first screen shows the draglines that are defined within the program.



Choosing Add or Edit will bring up the next screen. Notice the Required Sizes section. These must be all filled in. The remaining windows are all optional. Defaults will be used if they are not defined in the 3D Dragline command.

Depending on the Drawing Setup, either CY and FT, or CM and M are displayed as the units to enter. Carlson saves this information in the "dragline.dta" file in the USER folder of Carlson. The Save and Load buttons of the main dialog allow you to save and recall the dragline settings to a .DLN file.



The Save and Load buttons allow you to save and recall the dragline settings to a .DLN file.

Prompts

Edit Available Draglines and Dragline Specifications dialogs.

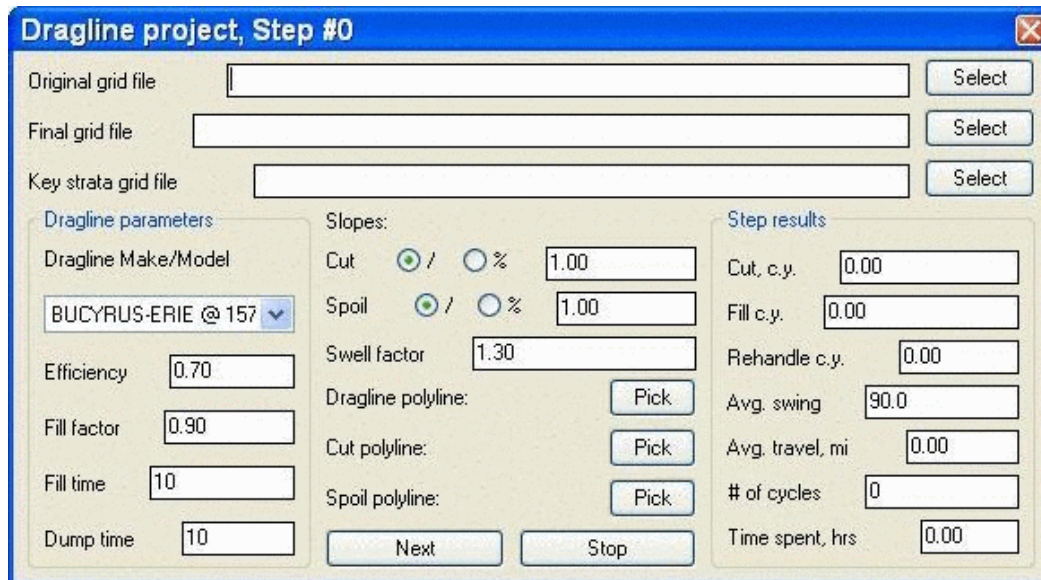
Pulldown Menu Location: Surface

Keyboard Command: edit_draglines

Prerequisite: None

3D Dragline

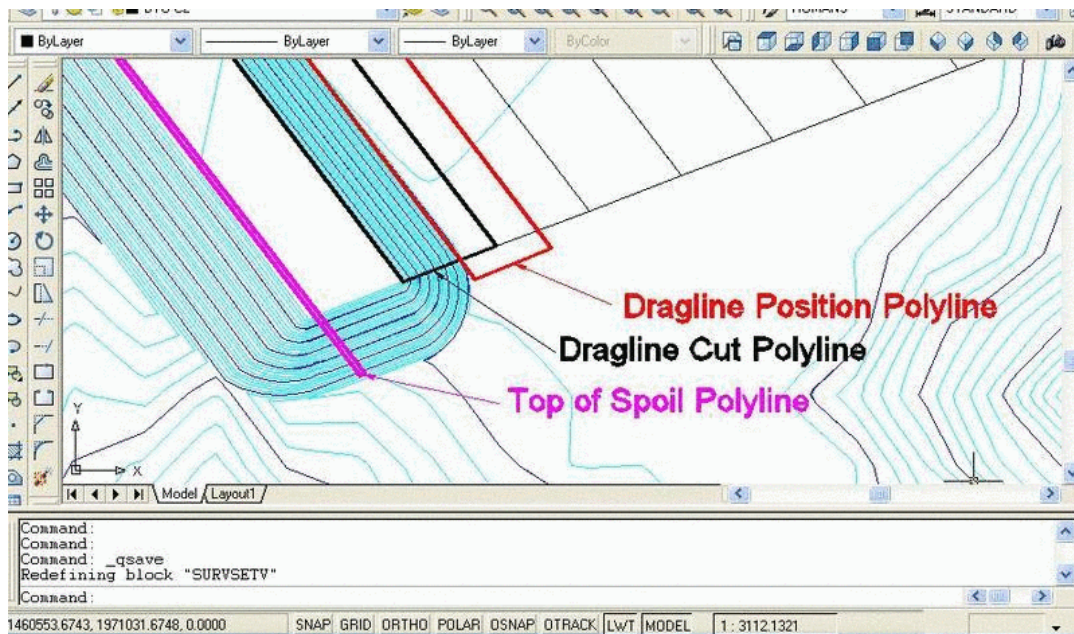
This command gives a 3D approach to the design and placement of dragline pits. Unless all the seams are perfectly uniform and horizontal, and pits are straight, the usual cross-section dragline modeling with range diagrams will not provide the appropriate level of precision and visualization. This function allows you to specify the cut and spoil areas and watch the ground surface change in 3D. Surface updates are recorded on every step and may be played back later.



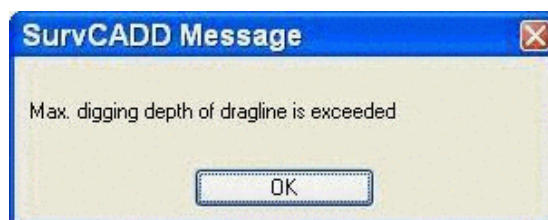
The first step is to define the three grid files at the top of the Dragline project window. They are the original topogrid file, the final grid file (create a new grid if one doesn't exist) and the top of the Key strata grid file.

The Dragline parameters section is for choosing the dragline. Then set its Efficiency %, Fill Factor %, Fill time (seconds) and Dump time (seconds). The Slope and Swell factor are set next, in the center of the screen. Slope is either set as ratio or percentage. A swell factor of 1.3 gives 30% swell.

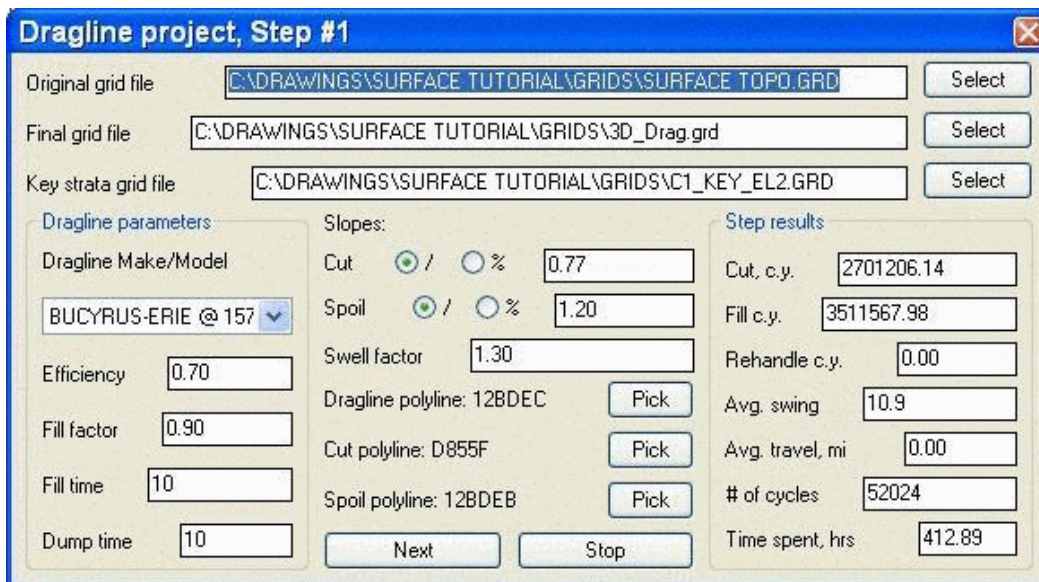
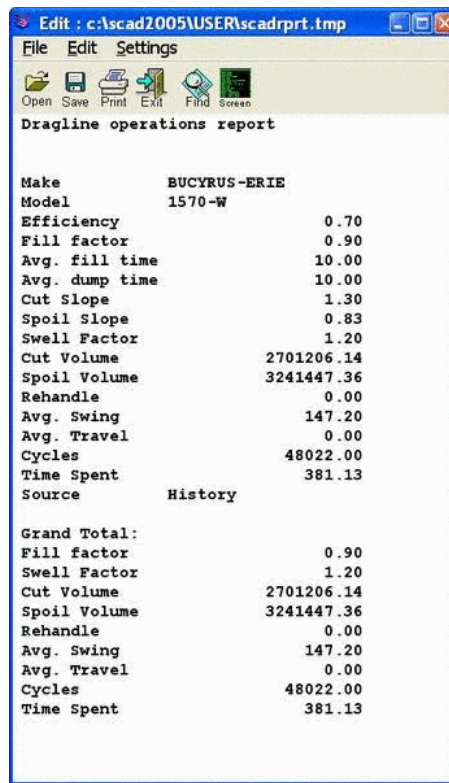
Finally, on each step of the sequence, three pre-drawn closed polylines are chosen: a polyline enclosing the area within which dragline is allowed to move, a bottom of cut area polyline (usually the next pit line), and a top of spoil area polyline (usually the previous pit line). The program then offsets the dragline polyline by the maximum reach distance and crops out the portions of cut and spoil polylines within the dragline reach. These modified polylines will be used in calculations. This allows the program to use larger cut and spoil polylines and move only the dragline on each step.

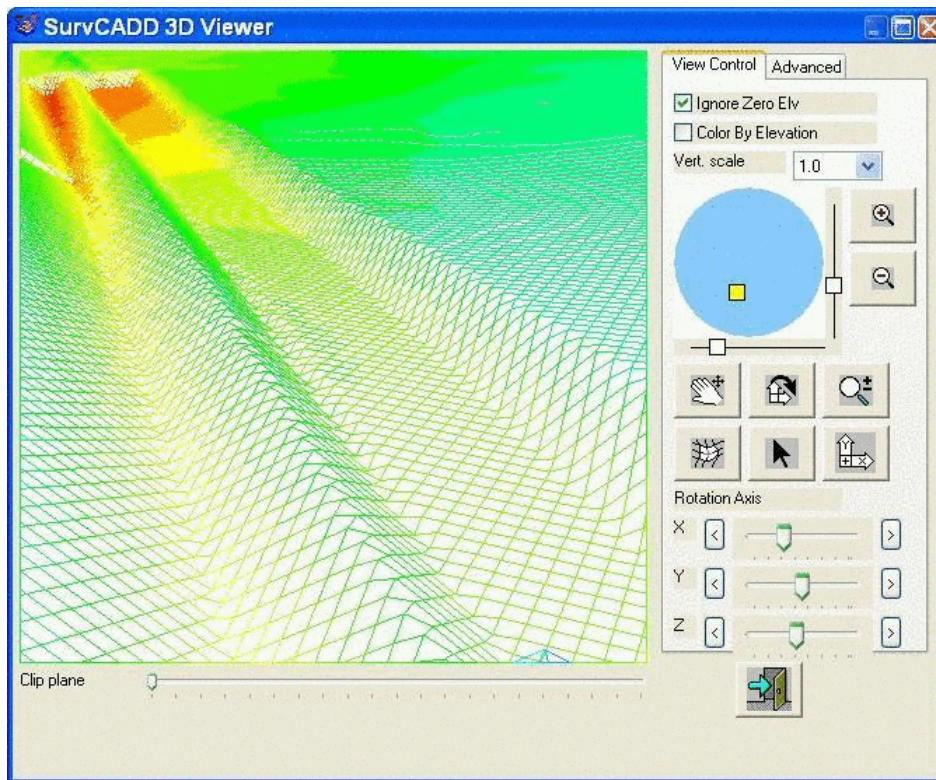


The program then calculates the overburden volume by using surface grid on top, coal top grid on bottom, and the highwall calculated using the cut polyline and given cut slope. The cut polyline is projected onto the bottom grid prior to calculations. The swell factor is then applied to the freshly cut material, while the rehandled material amount is taken as is. If the spoil polyline is a 3D polyline, its profile will be the elevations of the polyline. Otherwise the spoil ridge is considered to be at the same elevation. The spoil polyline is then moved up and down until the amount of the material between the spoil pile and the existing surface matches the given amount. If any dragline parameters are too small to physically mine here, then error messages, are displayed accordingly.



The calculated cut, spoil, rehandle and timing results are then displayed in the dialog. You may then modify the selection of cut, spoil or dragline polylines and proceed to the next step, or use the Stop button to exit. With use of the history file, the scheduling may be restarted from the point it was finished. Surfaces are updated with each step. To get to the next step, simply select then next 3 polylines, change the first two grid names (the top of coal usually remains the same) and choose next. After each step, the report formatter appears, allowing for customized reports of the steps.





3D View of the first step
The grid has been colored by elevation to show the ranges better.

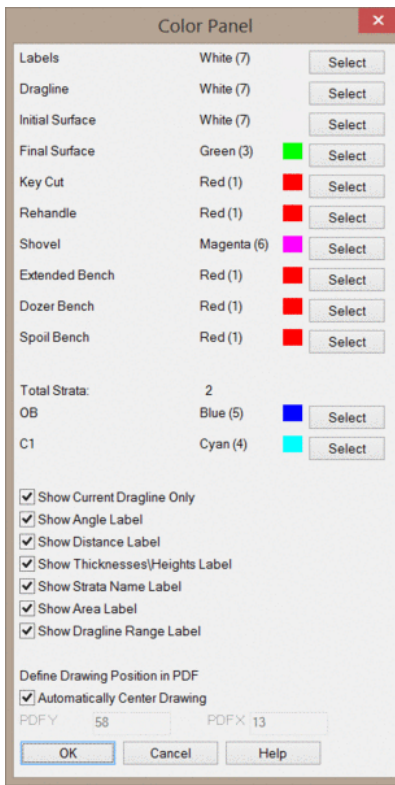
Prompts

Create or choose the DHT file. (Dragline history file.)
Dragline project dialog

Pulldown Menu Location: Surface
Keyboard Command: cut_n_drop

Range Diagram

The Range Diagram command is an interactive tool for analyzing dragline cut scenarios. This is a quick and easy way to find parameters such as pit width, or to size a dragline to fit the geology. Methods of mining include Side Cast, Extended Bench, Dozer Bench, and Spoil Bench. Output includes both drawing the sections in CAD and printing out reports as PDF files, where each step is a separate page. The main dialog is shown below.



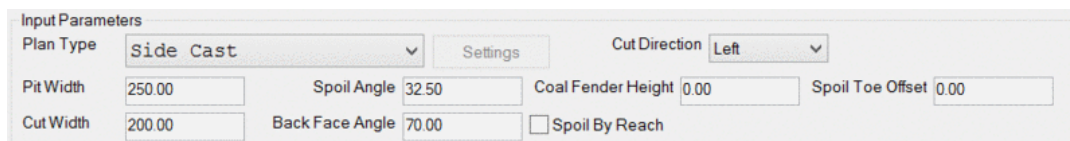
Show Current Dragline Only: This option will only show the current dragline, rather than showing all dragline passes at once.

Show ___ Label: These options will label the respective items in the graphics window.

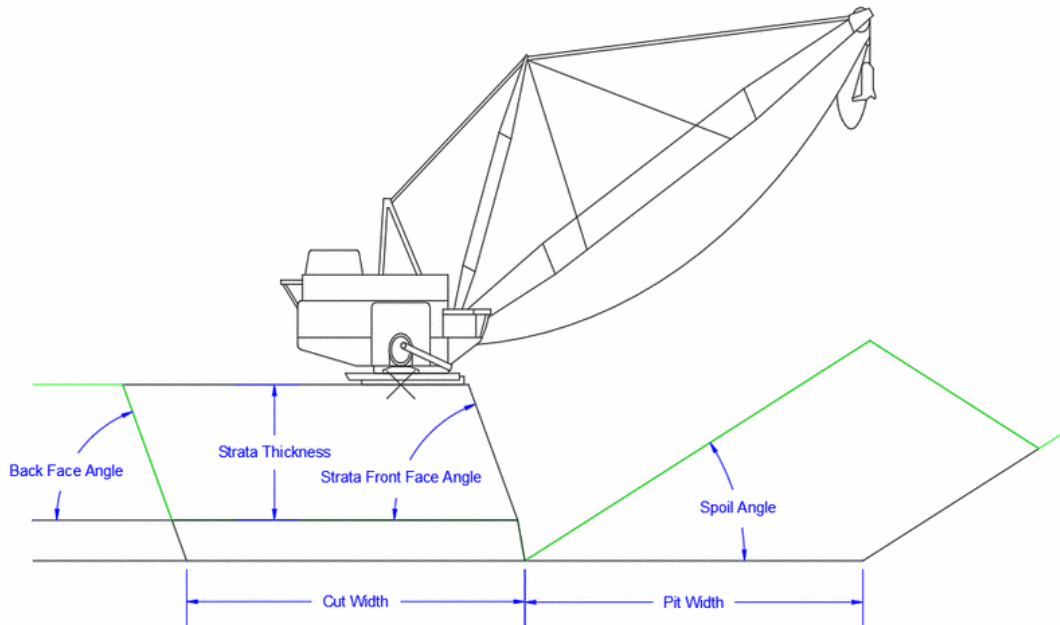
Automatically Center Drawing: This option will automatically center the range diagram on the PDF report.

PDF X/Y: These settings control the position of the range diagram in the PDF when it is not automatically centered.

This section of the help document discusses the Input Parameters, shown below.



Many of these options are visually explained below.



Plan Type: This option sets the mining method. Four options are available:

The *Side Cast* method places the dragline on the waste material with a previously cut pit on one side. The dragline strips the waste material and places it into the open pit.

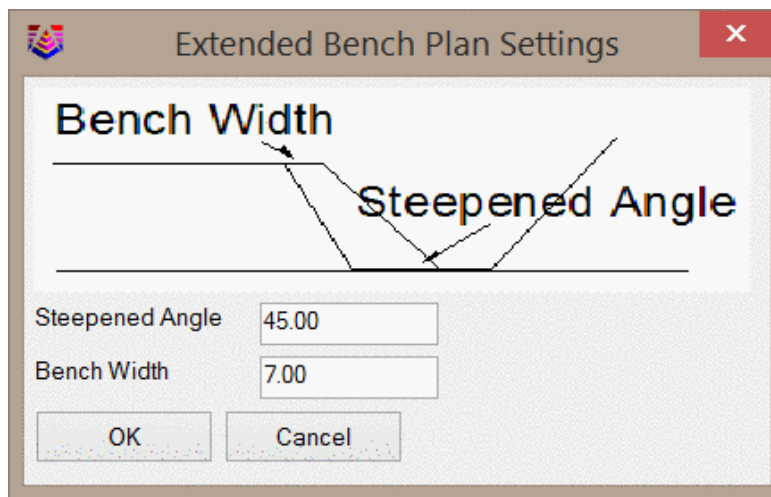
The *Extended Bench* method is similar to the Side Cast method, but the dragline will first place waste material at the edge of the existing highwall, thus extending the current bench further into the open pit. This allows the dragline to position itself on the extended bench, where it can then resume side casting material further into the open pit. This allows the dragline to spoil the material further into the open pit, but requires rehandle of the material used to create the extended bench.

The *Dozer Bench* method is similar to the Side Cast method, but is preceded by a dozer pushing a portion of the existing bench into the open pit. This allows draglines with a shallower digging reach to effectively dig out greater depths of overburden.

The *Spoil Bench* method is similar to the Side Cast method, but the dragline is positioned on the spoiled material rather than on the active bench.

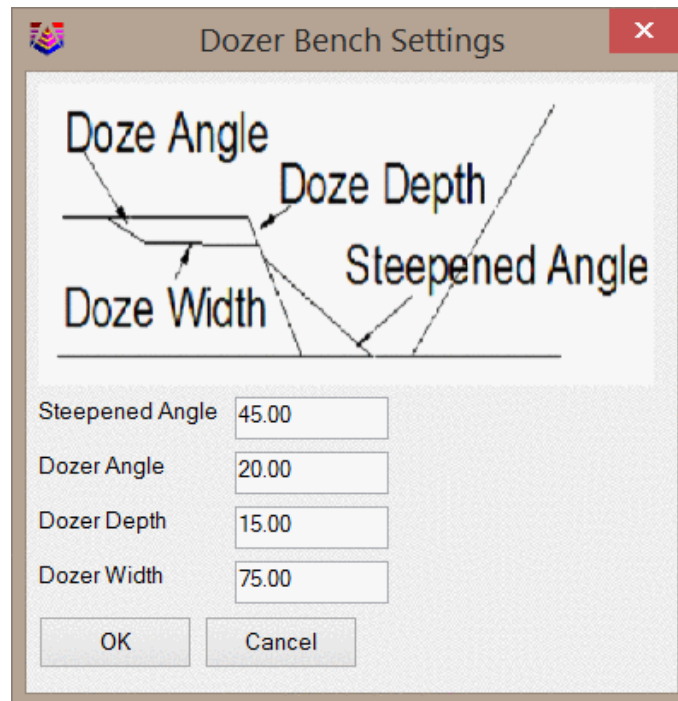
Settings: This button controls settings specific to the Extended Bench, Dozer Bench, and Spoil Bench Plan Types, depending on which Plan Type is currently selected. Each setting is visually represented in the settings dialog.

The Extended Bench Settings are shown below.



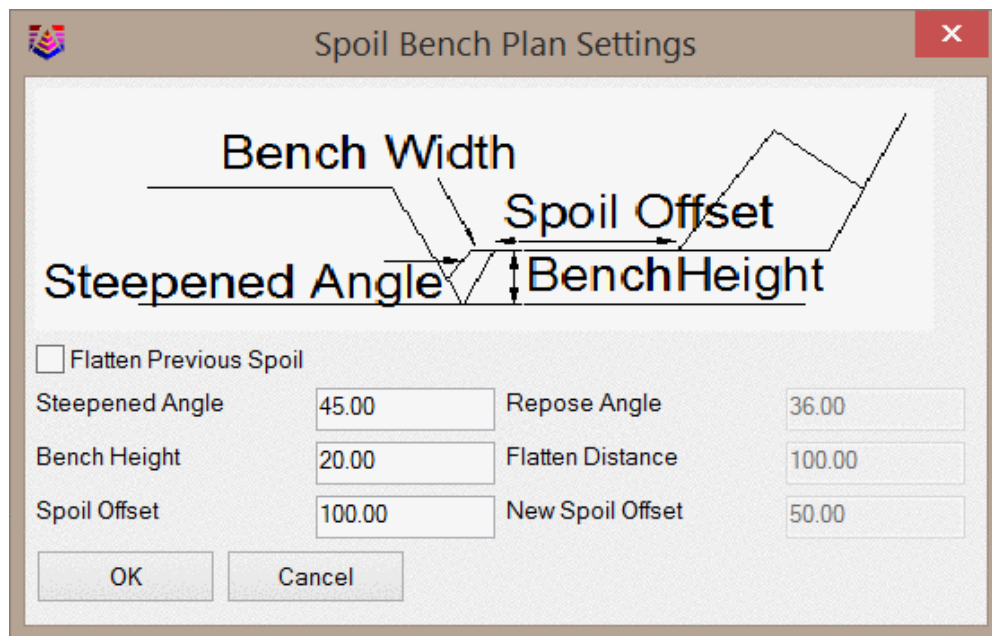
Steepened Angle: This value controls the angle, in degrees, of the new slope of the extended bench.
Bench Width: This value controls the width of the extended bench.

The Dozer Bench Settings are shown below.



Steepened Angle: This value controls the angle, in degrees, of the spoil pushed by the dozer into the pit.
Dozer Angle: This value controls the push angle of the dozer until it reaches the Dozer Depth.
Dozer Depth: This value controls the digging depth of the dozer.
Dozer Width: This value controls the width from the existing highwall to the start of the flat portion of the dozer bench. Note that the full dozer bench may be longer than this value depending on pit geometry.

The Spoil Bench Settings are shown below.



Flatten Previous Spoil: This option is only applied when two dragline passes are designed. When selected, the left three settings of this dialog will be made unavailable and the right three setting will become available. When a previous spoil pile is flattened for the dragline bench, it is always pushed further into the pit, away from the unmined ore.

Steepened Angle: This value controls the spoil angle from the pit floor up to the dragline spoil bench.

Bench Height: This value controls the height of the dragline spoil bench.

Spoil Offset: This value controls the width of the dragline spoil bench.

Repose Angle: When flattening a previous spoil pile for the new dragline bench, this value controls the repose angle, in degrees, of the pushed material.

Flatten Distance: When flattening a previous spoil pile for the new dragline bench, this value controls the width of the spoil that is flattened in preparation for the dragline.

New Spoil Offset: When flattening a previous spoil pile for the new dragline bench, this value controls the spoil offset from the previous spoil pile.

Cut Direction: This options determines whether the cut is advancing to the right or the left in the graphics window.

Pit Width: This value sets the width of the existing, open pit. This can represent a previous pass, or a box-cut.

Spoil Angle: This value sets the angle of repose, in degrees, for the spoiled material.

Coal Fender Height: This value sets the height of the remaining coal that may be covered with spoil. This wedge of coal that is covered by the spoil is known as a fender.

Spoil Toe Offset: This value places the toe of the spoil the specified distance from the bottom of the coal seam.

Cut Width: This value sets the width of the new cut. This is the field that is frequently changed in order to find an adequate pit size.

Back Face Angle: This value sets the highwall angle, in degrees, for the new cut for all strata.

Spoil by Reach: This option places the top of the pile at a distance specified in the **Spoil by Reach** column of the Dragline and Seam Data section of the dialog. When selected, the Coal Fender Height and Spoil Toe Offset fields will be unavailable.

This section of the help document discusses the Dragline and Seam Data inputs, shown below.

DRAGLINE AND SEAM DATA											
Dragline	Reach	Max Height	Max Depth	SW Dig	SW Dump	Strata	Offset				
650 @ BE AT KELLERMAN	175.0	105.0	90.0			OB	25.0				
Required	188.5	9.2	50.0	-90.0	90.0						
Strata	Key	Thickness	Front Face Angle	Density	Toe Offset	Swell %	Sh/Truck Thickness	Key Cut	Spoil By Reach	Coal Fender Height	Spoil Toe Offset
OB	NO	50.00	70.0	120.00	.00	20.0%	.00	Edit	300.00	.00	.00
C1	YES	30.00	80.0	80.00	.00						

Note that this section of the dialog uses colored text to highlight design flaws. Black text is used for simple input values, blue text is used for maximum available values, green text is used for values that are attainable with the current design, and red text is used for values that are not attainable with the current design.

Dragline: This dropdown menu sets the dragline to use for the current pass. When selected, a list of predefined draglines will be shown. Draglines may be added/edited via the Define Dragline Equipment command (Surface Mining Module > Surface Pulldown Menu).

Reach: This value displays the horizontal reach of the selected dragline.

Max Height: This value displays the vertical dumping height of the selected dragline, as measured from the bottom of the dragline.

Max Depth: This value displays the vertical digging depth of the selected dragline, as measured from the bottom of the dragline.

SW Dig: This value displays the required dragline Swing Angle, in degrees, on the digging side. If this value is shown in red, then the dragline cannot reach the entire cut, even with a -90 degree swing angle.

SW Dump: This value displays the required dragline Swing Angle, in degrees, on the dump side. If this value is shown in red, then the dragline cannot reach far enough into the pit to adequately dump the waste material, even with a 90 degree swing angle.

Strata (Top Section): This option sets the strata on which the dragline is positioned.

Offset: This value sets the distance between the highwall and the center of the dragline tub.

Strata (Bottom Section): This column lists all strata to be handled in the current dragline pass.

Key: This column marks each strata as key or non-key (waste) material. Only non-key material will be spoiled.

Thickness: This column controls the thickness of each strata.

Front Face Angle: This column controls the highwall angle, in degrees, of each strata. Note that a value of 90 represents a vertical highwall.

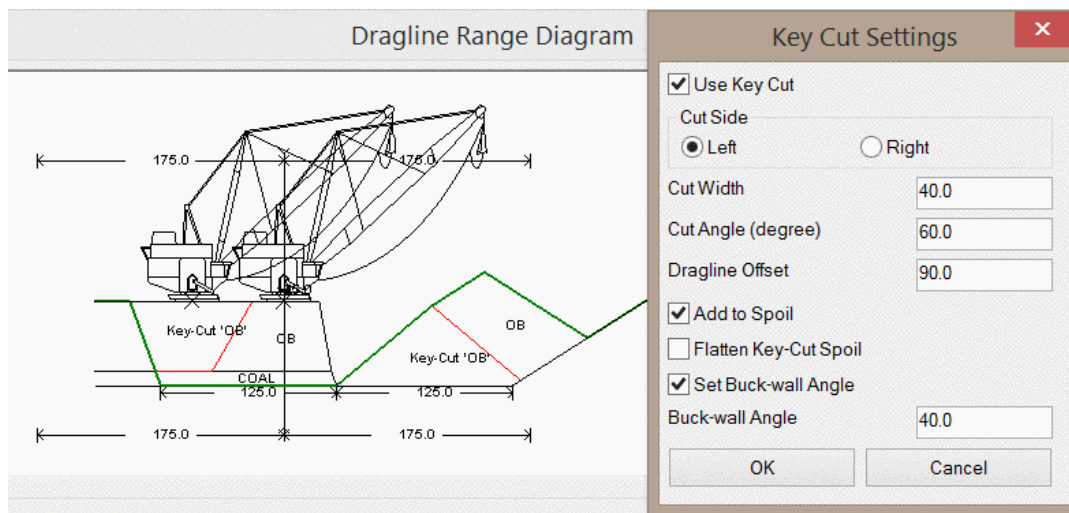
Density: This column sets the density for each strata. English units are set to lbs/ft³ and metric units are set to kg/m³.

Toe Offset: This column controls the bench width between strata.

Swell %: This column lists the swell percent of the spoiled material. A material with a swell of 10% will occupy 1.1 times it's original volume after it has been spoiled.

Sh/Truck Thickness: This column controls the thickness of each non-key strata that is to be excavated by truck and shovel (material that won't be spoiled into the open pit).

Key Cut: This column controls if each non-key material is to be excavated with a key cut. Clicking the **Edit** button will open the below dialog, which is shown next to an example of the results it will produce.



Use Key Cut: This option toggles the use of the key cut.

Cut Side: This option controls the location of the key cut - either the right or the left side of the cut area.

Cut Width: This value controls the width of the key cut.

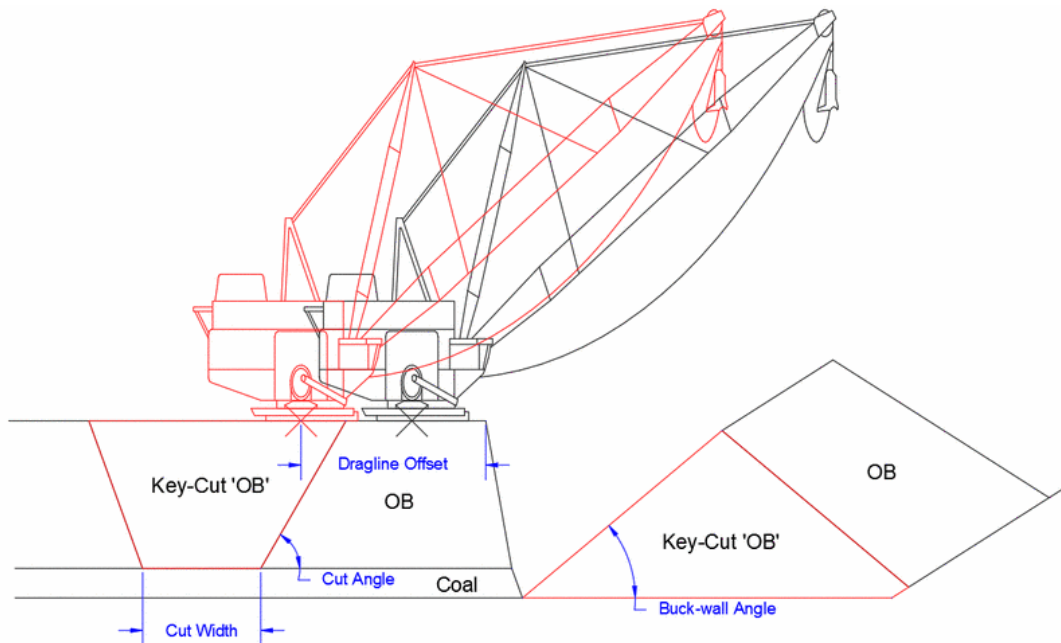
Cut Angle (degree): This value controls the cut angle of the key cut.

Dragline Offset: This value controls the distance between the crest of the highwall and the center of the dragline, during the key cut.

Add to Spoil: This option will actually spoil the material excavated in the key cut. If not selected, the material from the key cut is assumed to be haul away to another dump location.

Flatten Key-Cut Spoil: This option is only available when the Add to Spoil option is selected. This option will flatten the top of the spoil pile created from the key cut. When selected, the Spoil Bench Settings dialog will be used to set the parameters for the flattened bench.

Set Buck-wall Angle: This option will set the angle of repose for material spoiled from the key cut. This allows for two angles of repose to be displayed in the overall spoil pile. Several of the above parameters are represented below.

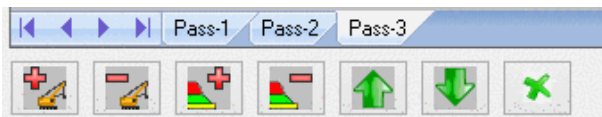


Coal Fender Height: This column displays the required coal fender height for the given Spoil by Reach value. This value is calculated automatically.

Spoil Toe Offset: This column displays the required spoil toe offset for the given Spoil by Reach value. This value is calculated automatically as the distance from the toe of the spoil to the toe of the strata.

Spoil by Reach: Yes or No. This column sets the spoil distance for each strata when the Spoil by Reach toggle above is selected.

Bottom-row Icons: At the bottom of the Dragline and Seam Data section of the dialog are six icons, shown below.



Listed in order from left to right:

Add Pass: This icon will add an additional dragline pass to the cut. This can be a different dragline or an additional pass of the same dragline. Each dragline pass is listed on a separate tab. Note that any strata layers listed for one dragline pass will be completely excavated in that pass. If a strata is to be excavated with dragline pass #2, that strata should not be listed in dragline pass #1. An example of the tabs shown for three dragline passes are shown above.

Remove Pass: This icon will remove the currently selected dragline pass.

Add Strata: This icon will add a new strata to the current dragline pass.

Remove Strata: This icon will remove the currently selected strata.

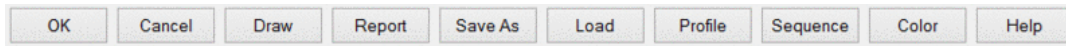
Move Up: This icon will move the currently selected strata up in the list.

Move Down: This icon will move the currently selected strata down in the list.

The output window will output the results of each step of the dragline sequence. Steps will be added and removed automatically.

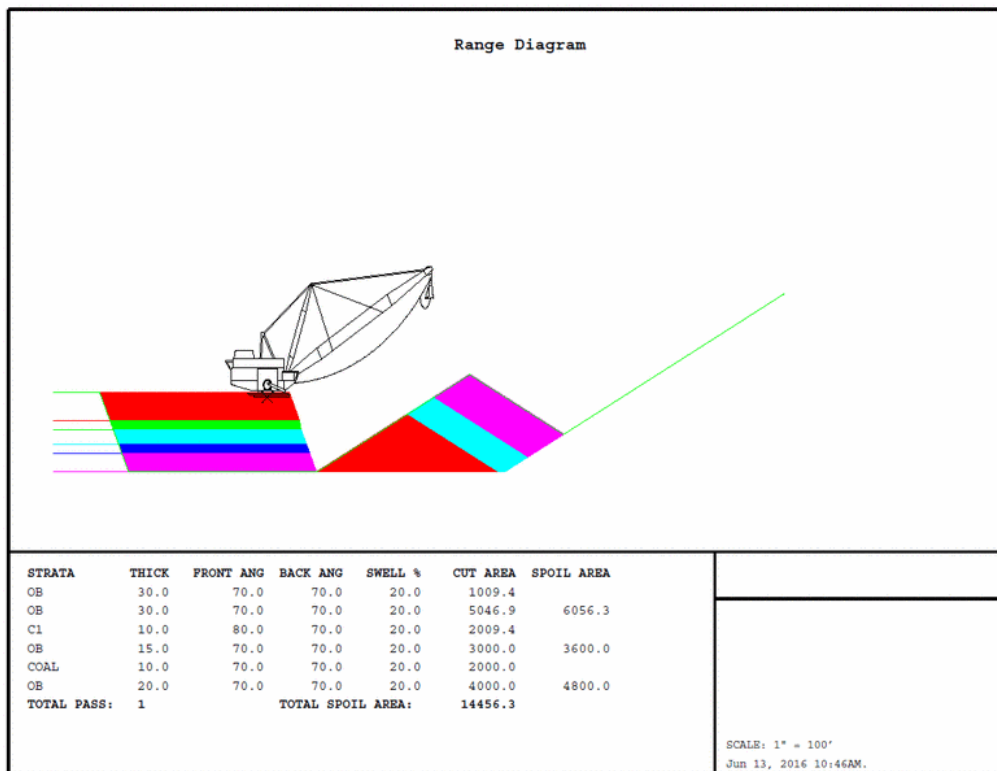
OUTPUT	
Report Title	
Generic Cut	
1. Ext Bench Rehandle	
Cut Area	5050.9
Spoil Area	5050.9
Toe Offset	50.8
Rehandle Rate%	24.9
2. Remove 'OB'	
650 @ BE AT KELLERMAN	
Cut Area	10281.5
Spoil Area	12337.8
Toe Offset	85.7
3. Remove 'C1'	
Cut Area	6084.4
4. Remove 'OB'	
650 @ BE AT KELLERMAN	
Cut Area	10000.0
Spoil Area	12000.0
Toe Offset	157.3
5. Remove 'COAL'	
Cut Area	2000.0
Total Spoil Area	29388.7

This section of the help document discusses the additional commands on the dialog, shown below.



Draw: This button will draw the dragline sequence in CAD. No dimensions will be labeled; only strata. Each step of the sequence will be drawn separately, with the first step shown at the bottom.

Report: This button will create a multi-page PDF document where each step in the section will be on a separate page. The last page of the report will show all steps, as shown below.



Profile: This button will save a profile (.pro file type) of the final step of the dragline sequence.

Sequence: This button will save the dragline sequence (.seq file type). This file can be used to extrude the dragline steps across a centerline via the Process Dragline Sequence command (Surface Mining Module > Surface Pulldown Menu).

Pulldown Menu Location: Surface

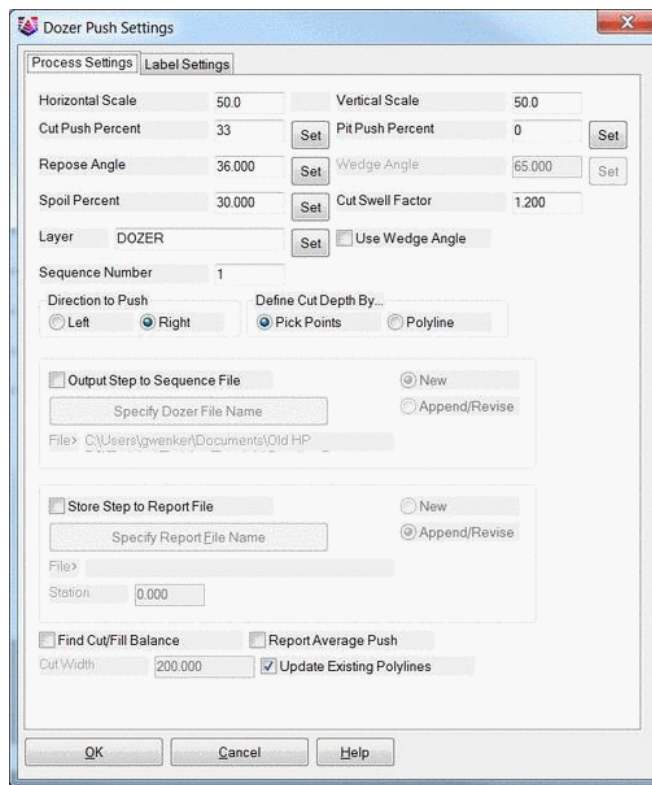
Keyboard Command: ranged

Prerequisite: dragline equipment defined

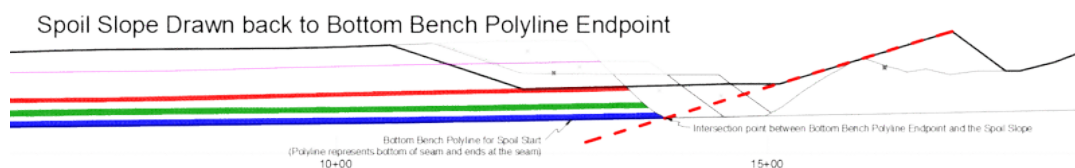
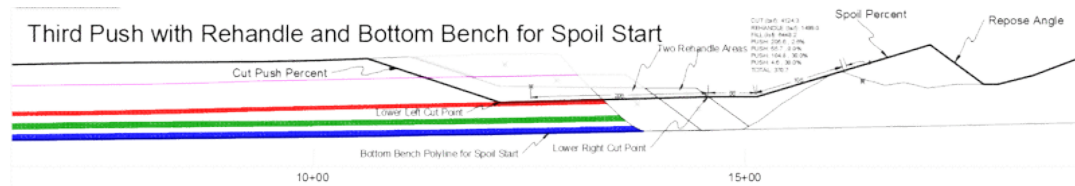
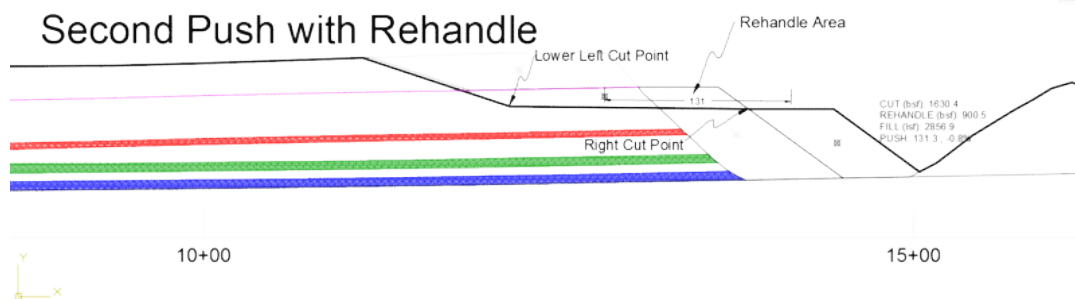
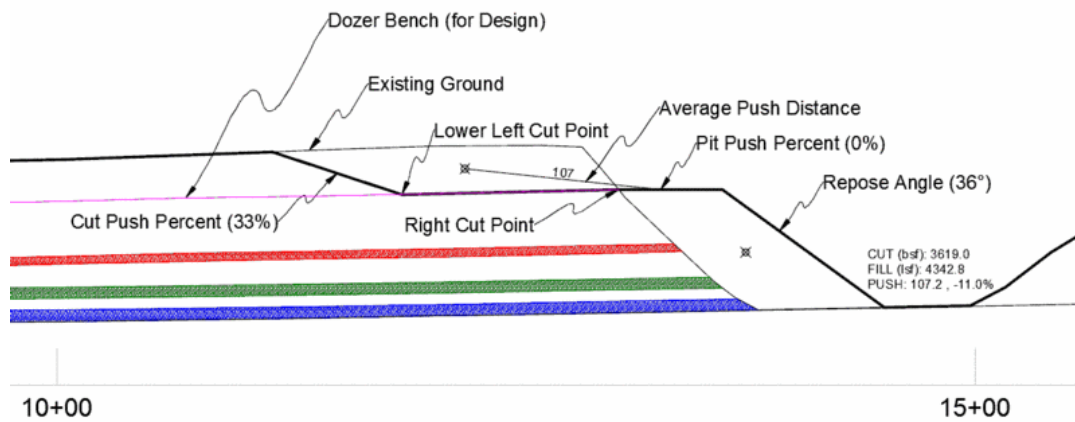
Dozer Push

This command simulates a dozer push design from a highwall into a pit on a cross section. The program draws closed polylines that represent the cut and fill areas and the path of the dozer. Dozer Push works with a cross-section view of the pit made from either the Fence Diagram command or Sections and Profiles from Civil. A prerequisite for this command is a polyline that defines the existing ground including the top surface, highwall and pit bottom. A second polyline optional polyline can represent the depth to the Cut Bench to push along, and a third polyline option could be one for the bench or pit bottom for spoil start. The Cut Bench polyline could be the top of of a seam to dig down to, for example. This command does one push at a time. To perform multiple cuts, run Dozer Push for each one.

The Dozer Push command starts with the dialog box shown here:

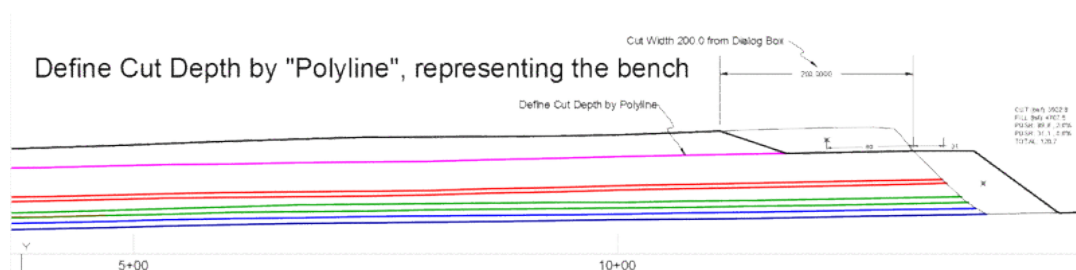


- **Horizontal Scale** and **Vertical Scale:** These are the scale factors for the drawing to generate text, symbols, and vertical exaggeration, among other things.
- **Cut Push Percent:** The Cut Push Percent is the percent grade the dozer cuts down from the surface to the bench polyline or to the selected cut point. There are separate angles for the left and right sides.
- **Pit Push Percent:** The Pit Push Percent is the percent grade that it will push out into the pit before spilling over to the repose angle. Enter as a percent, positive or negative, where positive will push down into the pit, and negative will push up, into a spoil fill pile.
- **Repose Angle:** Repose Angle is the angle that gravity pulls down the fill slope, as it spills over the edge. Enter as an angle.
- **Spoil Percent:** Spoil Percent is the percent grade of the spoil pile as the dozer pushes uphill. Enter as a percent (up). When there is not enough room in the pit to fit the cut at the pit push percent, the remainder will get placed by building up a spoil pile at this specified Spoil Percent. The program will prompt for a base point to start building the spoil slope with the prompt to Select bottom bench polyline for spoil start. This point is typically at the top or bottom of a seam at the pit bottom so that the spoil pile will not overlap the ore and create extra rehandle. The end point of the Bottom Bench Polyline is where the spoil slope will begin. To verify this, take an imaginary line and draw up from the endpoint of the bottom bench polyline, at the spoil slope, and it will pass through the fill, right on top of the spoil slope position.

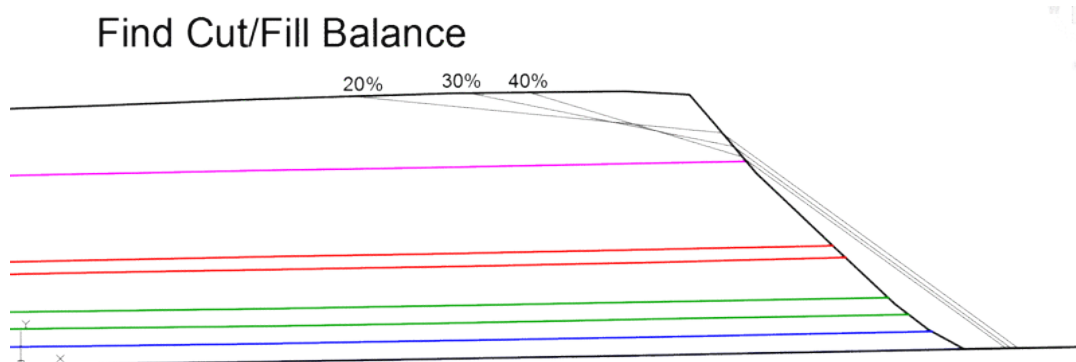


- **Cut Swell Factor:** The Cut Swell Factor is multiplied by the cut area to determine the amount of fill. Enter as 1.2 for 20 percent swell.
- **Layer:** This layer is used to draw the new entities created by this routine such as the cut and fill polylines and labels.

- **Use Wedge Angle:** With the Wedge Angle option, the cut will begin by starting at the top of pit and moving down at the Wedge Angle for the specified Cut Width. It will continue down at the wedge angle for the specified horizontal width. The Cut Push Percent is not used if the Wedge Angle is used.
- **Sequence Number:** This is the sequence step number that is added to the SEQ sequence file for Process Dragline Sequence.
- **Direction to Push:** The dozer must be specified to push to the right or to the left.
- **Define Cut Depth By:** The Polyline Method will prompt to select the Cut Bench Polyline. The polyline method can use either a closed or open polyline. For a closed polyline, the cut area is defined by the polyline. For the open polyline, the polyline represents the surface that the dozer will cut down to. The dozer cut will be located by finding the top of the pit and then moving back along the top surface for the Cut Width entered below. The top of the pit is identified by finding the first grade that is greater than or equal to the Push Percent grade. Then the dozer will cut down at the Push Percent until it reaches the bench polyline. The Pick Points method will prompt for the lower left point and the lower right point. The line drawn between them is then set to the cut surface. This controls the cut width, so the value entered below for Cut Width will not be used.

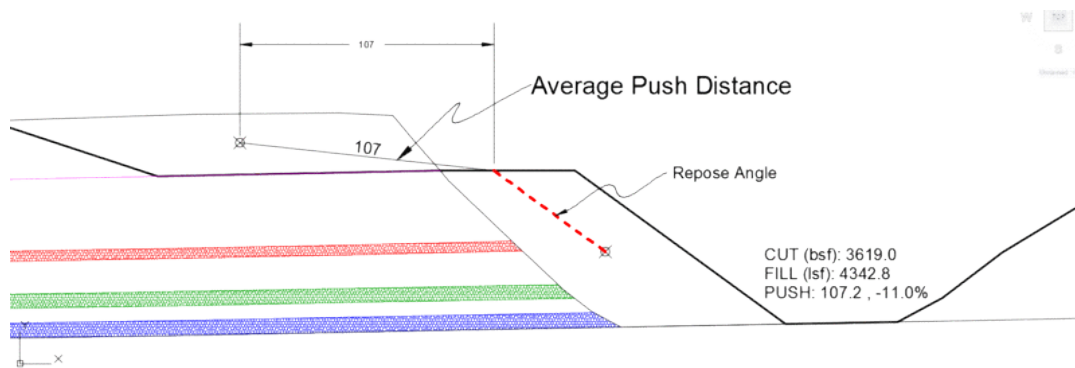


- **Output Step to Sequence File:** Turn this option on to add the step to a new or existing SEQ file. This file is processed with Process Dragline Sequence.
- **Store Step to Report File:** This option will create or append a CUT file that is processed with the command Dragline Section Report. This option requires the Dozer Push design on each cross section station the length of the pit.
- **Find Cut/Fill Balance:** The Find Cut/Fill Balance is an option that automatically calculates the cut area, where the cut and fill balance (plus swell factor) and so it does not prompt for picking the cut points and the bench polyline is not used. The command line prompt states "Select existing grade polyline at dozer push start:". The selected point along the existing ground polyline sets the Cut Push percent point. The program will cut down at this slope to define the cut, and then the fill is automatically placed for a Cut/Fill balance.



- **Report Average Push:** This option labels the average horizontal push distance from the cut centroid, to a

point that is up from the fill centroid, at the angle of repose.



- **Cut Width:** When using the Polyline method to Define the Cut Depth, this is the total width of the push, from the Cut Push distance down to the Cut Polyline, then across the Cut Polyline the remaining distance to get to the entered width. See the image above for Define Cut Depth By for the Cut Width labeled.
- **Update Existing Polylines:** This option will replace the existing ground polyline with the updated ground polyline. Otherwise the existing ground polyline will remain unchanged, and the cut and fill areas are drawn as separate polylines.
- **Label Style:** Sets the style for the text entities.
- **Text Size Scaler:** This scaler is multiplied by the Horizontal Scale to set the size of the text entities.
- **Area Decimals:** Sets the precision for the end areas.
- **Cut/Fill/Rehandle Labels:** Sets the names for these values.
- **Label Push Distances:** This option will label the distance from the centroid of the cut block to the edge of the highwall to a point above the fill centroid, where the repose angle intersects the fill surface. See the image above for Report Average Push
- **Draw Centroids:** Draws the specified symbol at the centroids for the cut and fill areas.

Prompts

This command handles many types of the dozer push. Which values to use from the dialog and the prompting depend on the design type of the dozer push.

Command: DOZER

To Define Cut Depth by Picking Points:

Select existing grade polyline: *pick the ground polyline*

Pick lower Left cut point:

Pick lower Right cut point:

Select fill areas to rehandle (Enter for None).

Select objects:

CUT (bsf): 9182.6 REHANDLE (bsf): 0.0 FILL (lsf): 9182.6

PUSH: 141.4,3.6%

Pick Point for Label (Enter for none):

Using Define Cut Depth by Polyline:

Select existing grade polyline: *pick the ground polyline*

Select cut bench polyline: *pick the bench polyline*

Select fill areas to rehandle (Enter for none)

Select objects: *press Enter*

Cut: 308.7 **Fill** 308.7

Pick Point for Label (Enter for none): *pick a point*

Using the Find Cut/Fill Balance option:

Select existing grade polyline at dozer push start: *pick the ground polyline where the cut is to begin Calculating...*

Pulldown Menu Location: Surface

Keyboard Command: dozer

Prerequisite: Surface and strata polylines

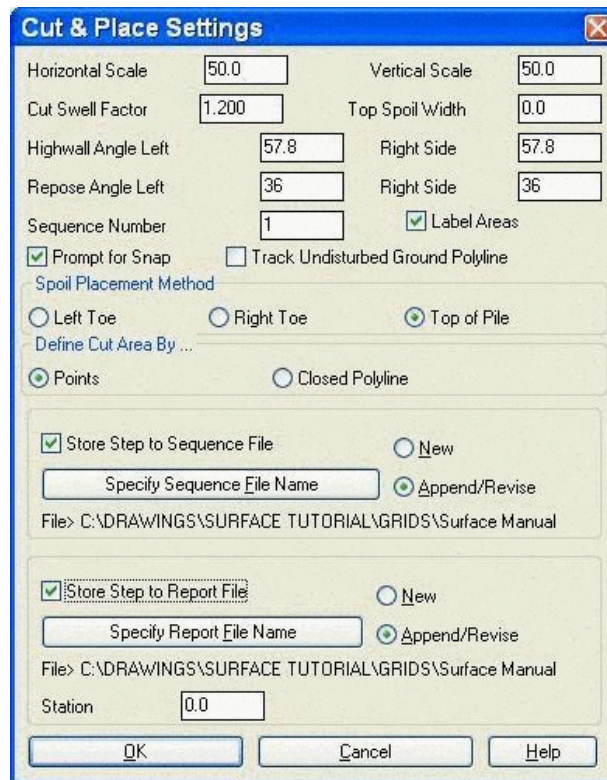
Draw Dragline Limits

This command draws an outline in profile view of the dragline reach, depth and height limits. The program draws the limits of the dragline as polylines in the current layer. These limit polylines serve as guides in other dragline commands like Cut & Place. Also these dragline commands will check the dragline limits and report if the dragline limits are exceeded when processing the sequence file. The dragline parameters are set in the Define Dragline Equipment command. Before starting this command, the dragline should be defined in Define Dragline Equipment and the surface where the dragline sits should be drawn as a polyline in profile view. This polyline can be drawn with the Draw Fence Diagram command or the profile commands in the Civil Design module.

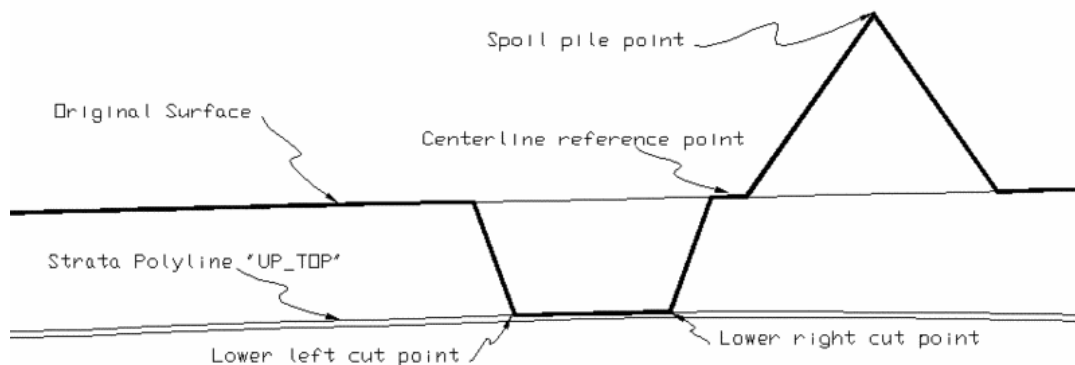


The window prompts for the horizontal and vertical scale. This is to determine the vertical exaggeration if any. There is an option to draw an icon of the dragline. Otherwise the limits are only drawn as polylines. Choose a direction for the boom to be facing, left or right. The draw swing angles option will label the angles along the top of the limit polyline. The Group Dragline Entities will keep it together as a block instead of individual polylines. If the program detects that dragline limits are already in the drawing, then there is an option to erase the existing limits. This option is useful for automatically erasing old limits before drawing the new limits, as the program tracks where the dragline is located to measure the reach. Then pick the surface polyline and where along the polyline the dragline is located. There is an option to specify the dragline location by picking a reference point such as the top of the highwall and then entering an offset distance from the reference point to the center pin of the dragline (left offset is a negative value).

Station, then set to Append/Revise for additional stations. These are processed with the Dragline Section Report commands.

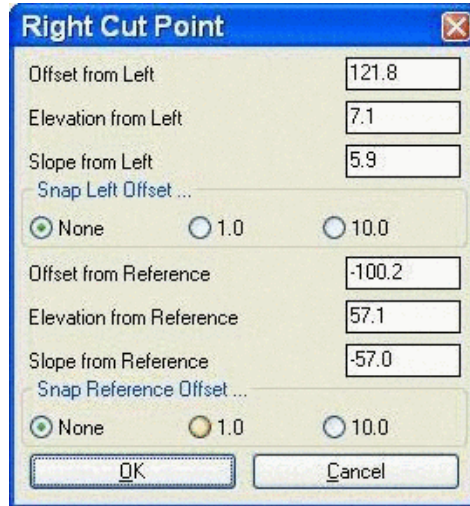


When picking points, there is an option to select a strata polyline which will be used as the bottom cutoff for the cut area. Also while picking the points, there is a real-time window that shows the point offset, slope and distance as you move the cursor. Once the point is picked a snap dialog allows you to adjust the picked point. For example, you could use the real-time window to visually pick the right cut point. The actual picked right position could be 201.32 from the left point and then you can use the snap dialog to round the position to 200. This adjust point dialog only appears if the Prompt for Snap option is on. You can also specify a reference point by typing 'R' for the Reference option at the command prompt. This option will prompt you to pick a reference point in the section. For example you could choose the toe of the existing highwall as the reference point and make the right cut point offset by 10.0.



After choosing the cut area, the spoil pile is placed by picking a point to place the spoil. This point can be either the left toe of slope, right toe of slope or top of pile. After defining the cut and spoil, there is an option to go back and adjust the cut or spoil points. Select L, R or S for left point, right point or spoil location. Adjusting the cut points will automatically resize the spoil pile-watch it dynamically change. Once everything is set, press Enter to end the routine. The surface profile polyline is redrawn with the cut area removed and the spoil added. The changed segments of the original surface are drawn a different color, usually gray.

The user input for this command can be saved to a sequence file (.SEQ) for the Process Dragline command or a section report file (.CUT) for the Dragline Section Report command. With the sequence file active, the program will prompt for a centerline reference point which is the point in the cross-section view that the centerline for Process Dragline passes through. It needs this point to locate the cross-section horizontally in the plan view relative to the centerline drawn in plan view. If you choose a strata polyline, the program will ask for the strata name of this polyline. The Process Dragline command will then prompt for a grid file that models this strata name. The data that is saved includes the depth and offset of the cut points, the spoil pile offset, and the slopes. Besides specifying the sequence file name to write to, the sequence number is also stored. This number orders the steps in Process Dragline. For the Dragline Section Report, the program will store the station, sequence number, and cut and fill areas.



Right Cut Point

Offset from Left: 121.8

Elevation from Left: 7.1

Slope from Left: 5.9

Snap Left Offset ...

None 1.0 10.0

Offset from Reference: -100.2

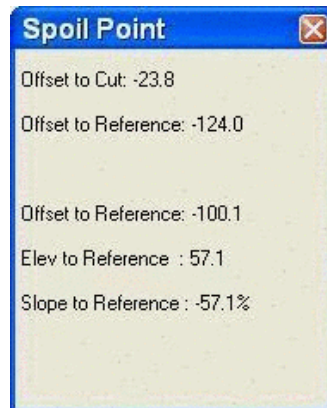
Elevation from Reference: 57.1

Slope from Reference: -57.0

Snap Reference Offset ...

None 1.0 10.0

OK Cancel



Spoil Point

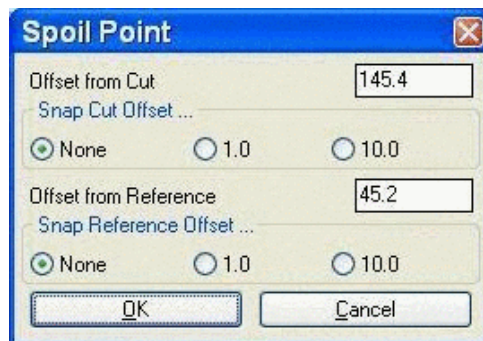
Offset to Cut: -23.8

Offset to Reference: -124.0

Offset to Reference: -100.1

Elev to Reference : 57.1

Slope to Reference : -57.1%



Spoil Point

Offset from Cut: 145.4

Snap Cut Offset ...

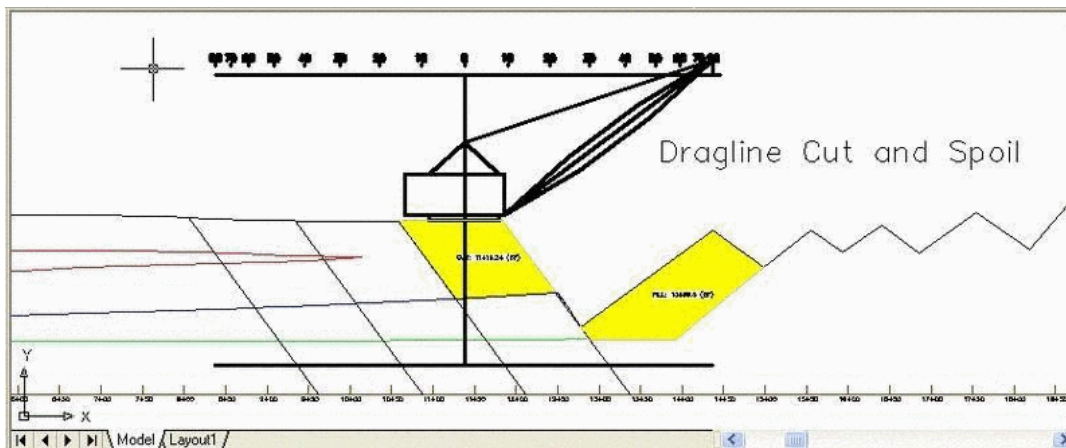
None 1.0 10.0

Offset from Reference: 45.2

Snap Reference Offset ...

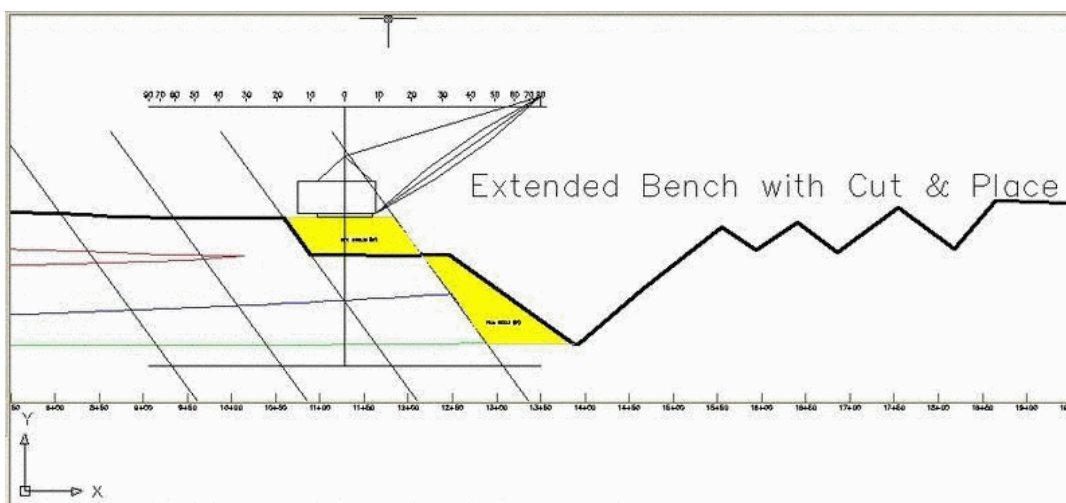
None 1.0 10.0

OK Cancel



The Track Undisturbed Ground Polyline option is for calculating rehandle. With this option on, the program will prompt for the existing ground polyline as usual plus an additional undisturbed ground polyline. The undisturbed polyline is the same as the existing ground polyline except that the undisturbed polyline does not have the spoil piles. One way to start this process is just copy the Surface line onto itself, then move one away from the other on one of the ends for ease of selecting for the first cut. After the first cut, it should be easy to select one or the other. Cut & Place will update the undisturbed polyline by removing the cut area and will not add the spoil pile. The program will also report any rehandle using the undisturbed polyline. The swell factor is not applied to the rehandle.

This command will also fill in for some of the Dozer Push short comings. If there is a simple extended bench needed, the set one of the spoil slopes to 0 and it will create a flat spoil for that step. Be sure to hit enter when prompted for the top of coal line. The next cut could be a pre-drawn, closed polyline or a series of cuts to remove the material. The lower right cut slope in the case could be the same as the spoil slope to create the left slope of the spoil.



Prompts

Cut & Place Settings Dialogs

Select existing ground polyline: *pick a polyline*

Pick centerline reference point on ground polyline: *pick a point on ground polyline*

Select strata polyline (Enter for none): *pick the strata polyline*

Enter strata name for strata polyline: *UB_TOP*

Reference/<Pick lower Left cut point>: *pick a point*

Reference/<Pick lower Right cut point>: *pick a point*

Right Cut Point dialog

Reference/<Pick spoil pile location>: *pick a point*

Control point to adjust (Left cut/Right cut/Spoil/<None>)? *press Enter*

CUT: 2327.00 (sf) FILL: 2327.00 (sf)

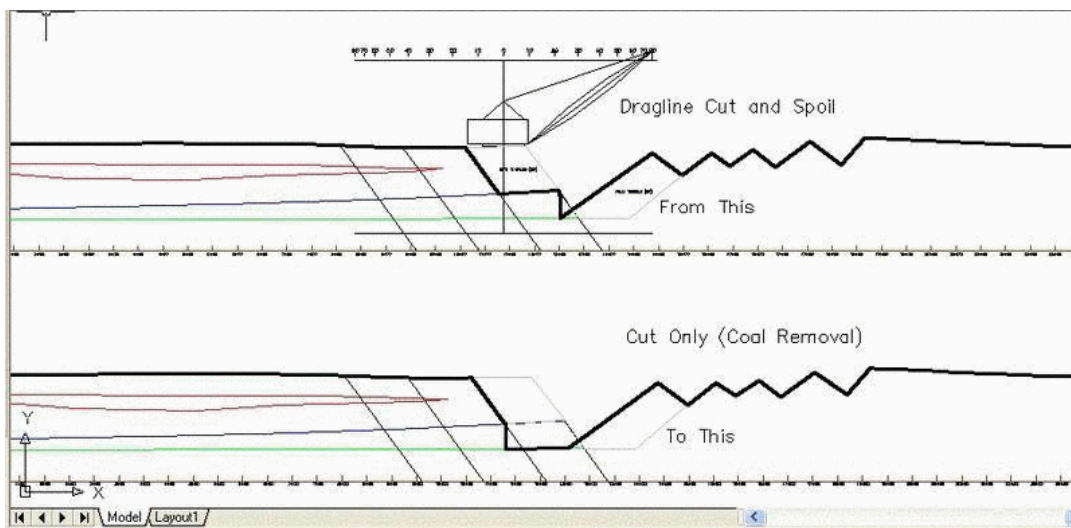
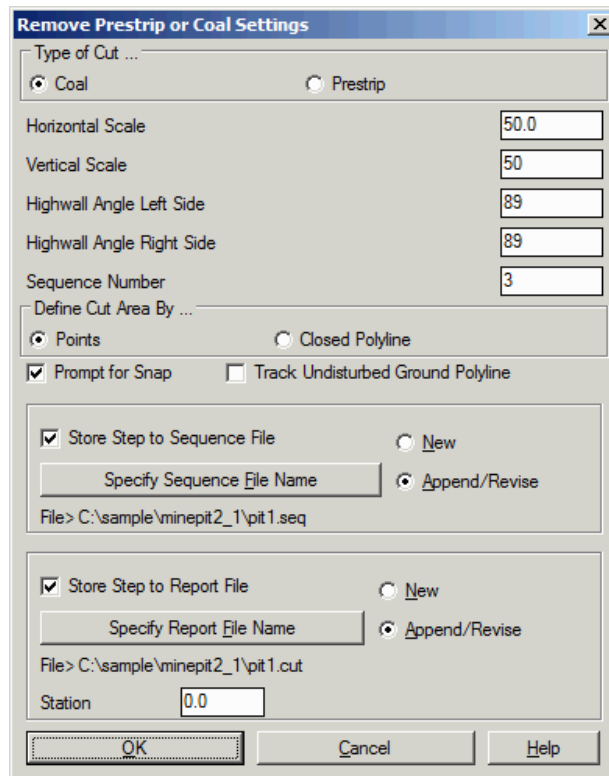
Pulldown Menu Location: Surface

Keyboard Command: range1

Prerequisite: Surface polyline in profile view from a Profile or Fence Diagram. Grids of the seams if writing sequence file.

Cut Only (Coal Removal)

This command is similar to Cut & Place. Cut Only removes a cut area in cross-section view from a surface polyline. The difference with Cut & Place is that this command does not place the cut area as a spoil pile. The cut area is just removed from the surface polyline for operations like remove coal. Try to stay away from a 90 degree face on the coal, AutoCAD doesn't like vertical faces (89 is entered below). The Type of Cut chooses between Coal (Key Only) and Prestrip (Non-Key Only). This setting is used in reporting and doesn't affect the rest of the routine. The same prompting and snap dialogs appear here as in the Cut and Place command.



Prompts

Select existing grade polyline: *pick the surface polyline*

Pick centerline reference point on ground polyline: *pick a point on the surface polyline*

Select strata polyline (Enter for none): *pick a strata polyline*

Enter strata name for strata polyline: *UB_BOT*

Pick lower Left cut point: *pick a point*

Pick lower Right cut point: *pick a point*

Control point to adjust (Left cut/Right cut/<None>)? *press Enter*

CUT: 91.42 (sf)

Pulldown Menu Location: Surface

Keyboard Command: rmcoal

Prerequisite: Surface polyline in profile view

Flatten Spoil Top

This command takes a surface polyline drawn with a spoil pile and flattens the top of the spoil pile. There are two methods to set the flatten width. The **By Distance** method sets the horizontal distance across the top to equal the user-specified distance. The **By Height** method sets the elevation of the flatten to the specified height above a reference point like the pit floor. With this method, the program will prompt to pick the reference point.

The flatten direction will either push to the left, right or both directions. This step can be stored to a sequence file (.seq) for the Process Dragline Sequence command.

Spoil Top Settings

Horizontal Scale: 50
Vertical Scale: 50
Repose Angle: 36.000
Flatten Distance: 20.000
Height: 100.0

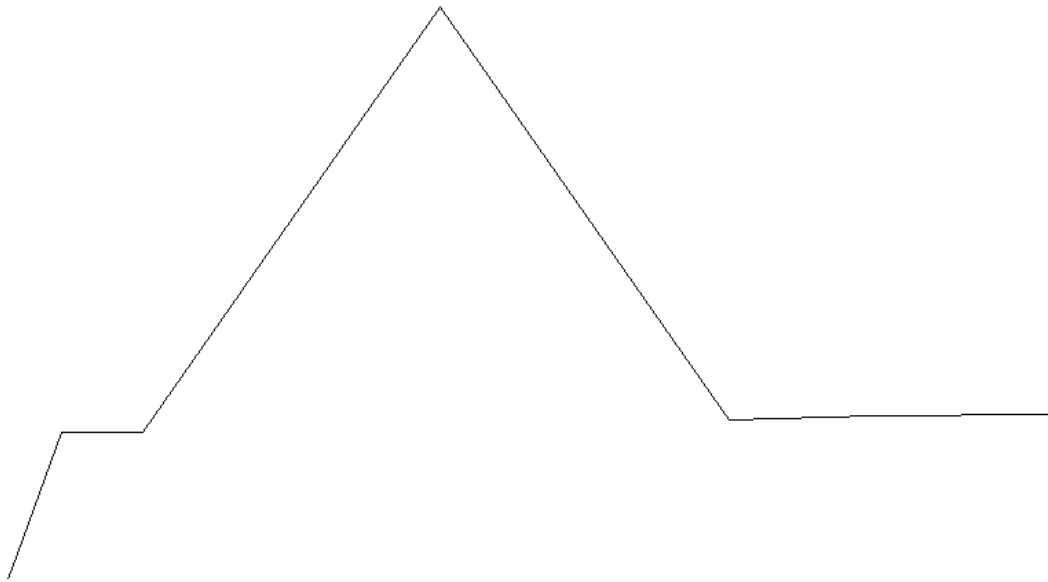
Flatten Reference:
 By Distance By Height

Direction to Flatten:
 Left Right Both

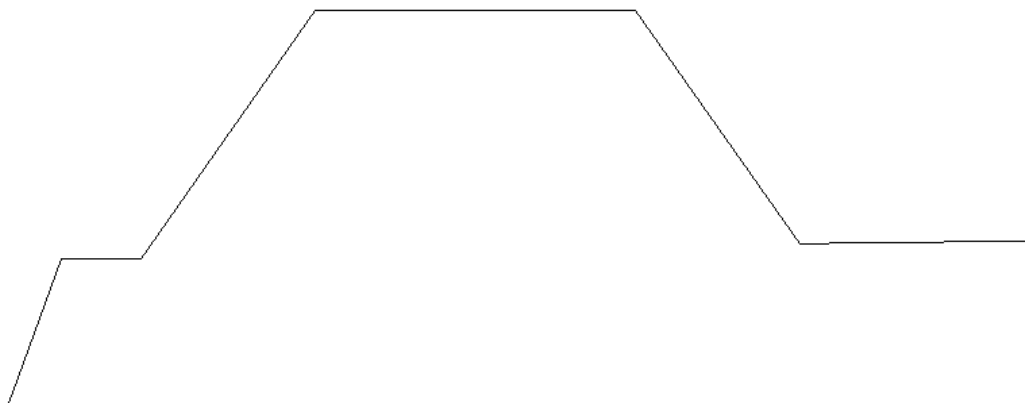
Store Step to File
 New Append/Revise

Specify Sequence File Name
File: C:\sample\tunnel\tunnel_orig.seq
Sequence Number: 2

OK Cancel Help



Before Flatten Spoil Top



After Flatten Spoil Top

Prompts

Select pile to flatten: *pick the surface polyline near the top of the spoil pile*

Pulldown Menu Location: Surface in the Mining Module

Keyboard Command: spoil

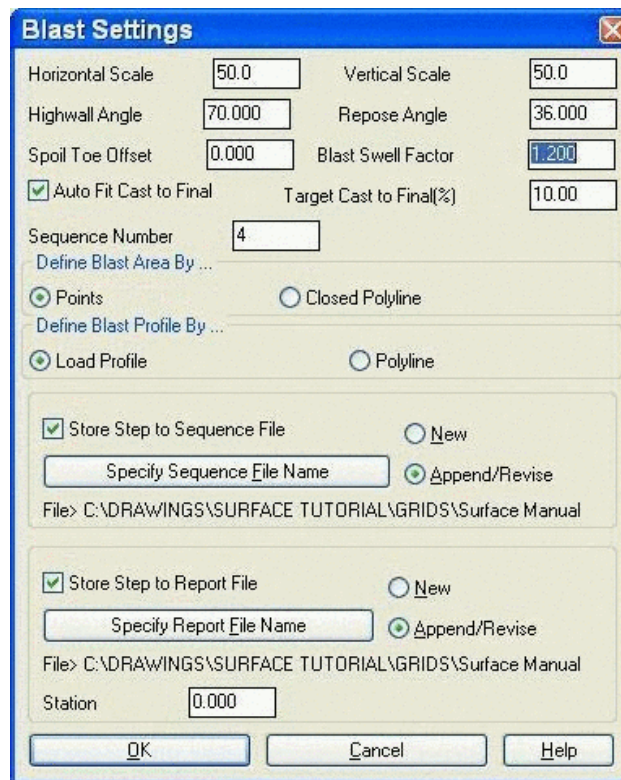
Prerequisite: Surface polyline in profile view with spoils

Cast Blast Profile

This command applies a blast profile to a cut area in cross-section view. Before starting this command, the surface should be drawn as a polyline from left to right.

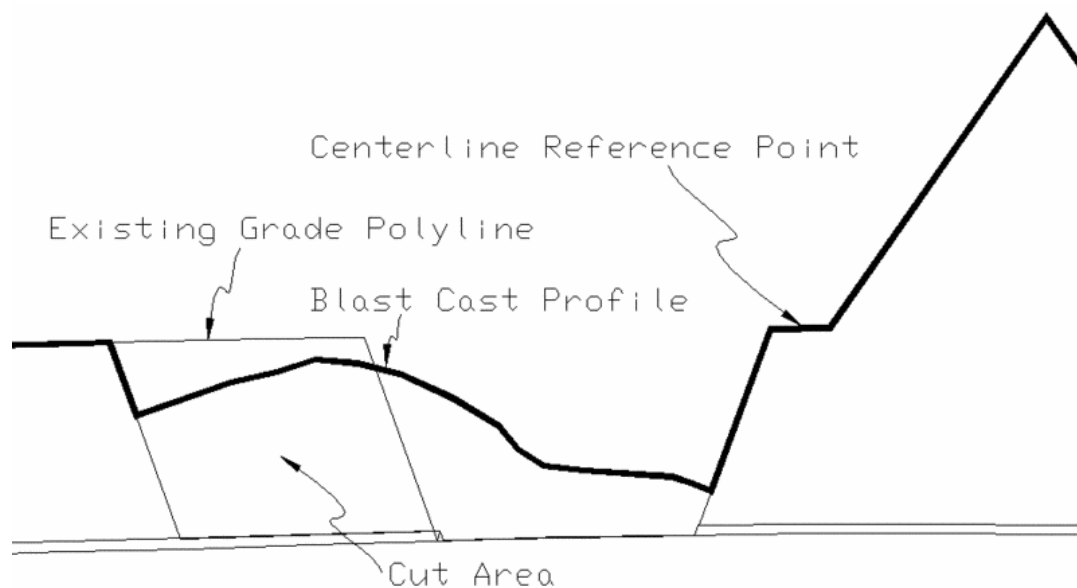
Cast Blast first brings up a dialog where you can set the scale, highwall angle and swell factor. The Horizontal Scale and Vertical Scale are used to determine the vertical exaggeration of the cross-section view. The Highwall Angle is used with the Define Blast Area By Points option as the angle from the picked points at the bottom of the blast area up to the surface. The Cut Volume Swell Factor is multiplied by the cut area to determine the fill area of the spoil pile.

The area to cut can be either a closed polyline or defined by picking the lower left and lower right points. When picking points, there is an option to select a strata polyline which will be used as the bottom cutoff for the blast area. The blast profile can be defined by either a polyline drawn from left to right on the screen or by a Carlson profile (.pro) file. Refer to the Section & Profile module for how to create a profile file.



After placing the blast profile, there is an option to adjust the blast profile points. To adjust a point, pick on the blast profile near the point to modify. This will pick up that point and you can pick a new position. When a point is moved, the program will automatically fit the updated blast profile. Once everything is set, press Enter to end the routine. The surface profile polyline is redrawn with the blast area showing the blast profile. The changed segments of the original surface are drawn in a different color.

The program reports the cut and fill end areas and the cast to final percent which is the percent of the fill area that is in the final spoil area. The cast to final area is defined by the area past the line starting at the toe of cut and going up at the Repose Angle. This starting point can be offset from the toe of cut by using the Spoil Toe Offset field. A negative offset moves to the left and a positive to the right. The Auto Fit Cast To Final option will adjust the blast profile by moving the profile horizontally such that the placed profile balances the cut and fill areas and meets the specified Target Cast to Final percent.



There is an option to save the user input for this command to a sequence file (.seq) for the Process Dragline command and to a section report file (.cut) from the Dragline Section Report command. With the sequence file active, the program will prompt for a centerline reference point which is the point in the cross-section view that the centerline for Process Dragline passes through. If you choose a strata polyline, the program will ask for the strata name of this polyline. The Process Dragline command then prompts for a grid file that models this strata name. Besides specifying the sequence file name to write to, the sequence number is also stored. This number orders the steps in Process Dragline.

Prompts

Select existing grade polyline: *pick a polyline*

Pick centerline reference point on ground polyline: *pick a point on the ground polyline*

Select strata polyline (Enter for none): *pick a polyline*

Enter strata name for strata polyline: **UB_TOP**

Pick lower Left cut point: *pick a point*

Pick lower Right cut point: *pick a point*

Select blast profile polyline: *pick the polyline representing the typical profile*

Cast to Final: **10.20%**

Pick profile point to modify (Enter to end): *pick a point on the blast polyline*

Pick new position: *pick a point for the new position*

Pick profile point to modify (Enter to end): *press Enter to end*

CUT: **1775.60 (sf)** **FILL:** **2308.28 (sf)**

Pulldown Menu Location: Surface

Keyboard Command: spoil

Prerequisite: Surface polyline in profile view

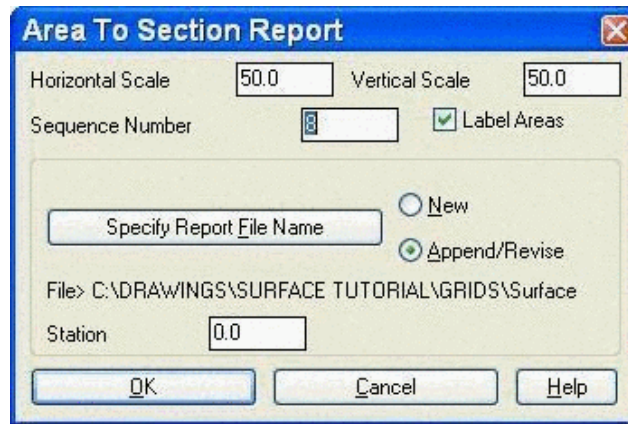
Polyline to Centerline File

This command is described in the Centerlines section of the manual. Please refer to it there.

Area To Section Report

This command stores the area of closed polylines into a section report file (.cut) that is used by the Dragline Section Report command. Along with the area of cut and fill, this command associates a station and sequence number. The station is the distance along the dragline centerline and the sequence number orders the area in the steps at that

station. There can be several steps at each station. This command prompts to select a cut area polyline which should be a closed polyline in section view. There is also an option to select a fill polyline. The areas can be assigned a description that is used in the Dragline Section Report.



Prompts

Select cut area polyline: *pick a closed polyline*

Cut Area: 17929.86

Select fill area polyline (Enter for None): *pick a closed polyline*

Fill Area: 22184.69

Description <Cut Area>: *Shovel Pass1*

Pulldown Menu Location: Surface

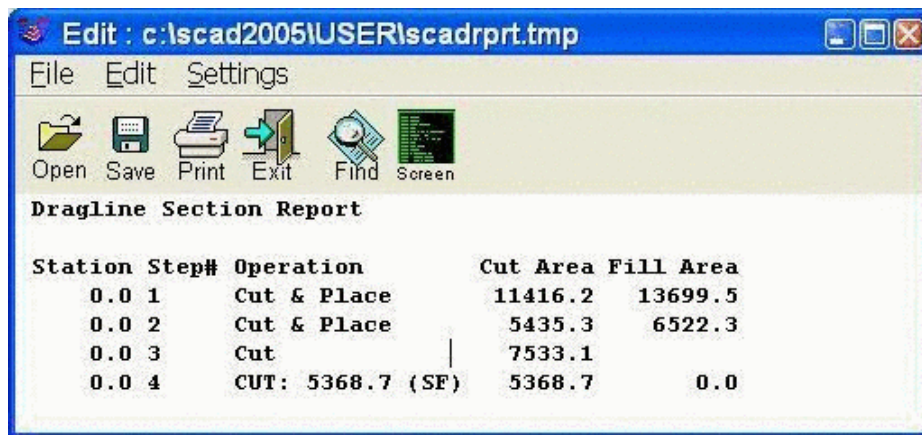
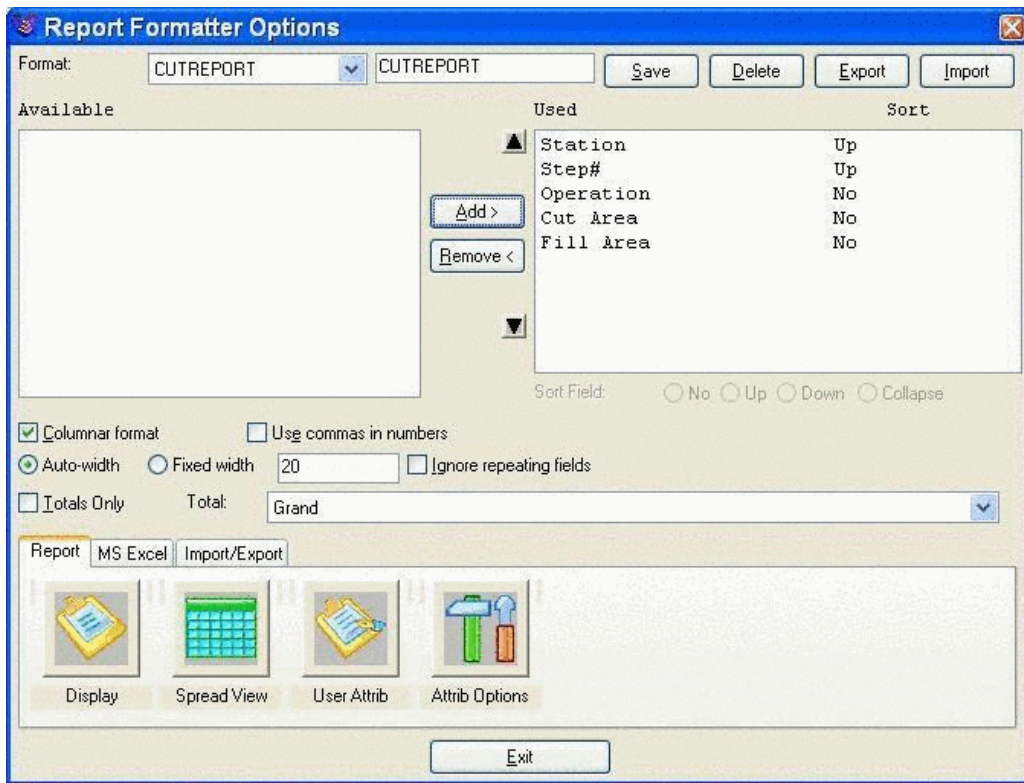
Keyboard Command: cutarearprt

Prerequisite: A closed polyline in section view

Dragline Section Report

This command reports the end areas at different stations based on the accumulated stored results of various dragline routines. The report data is read from a section report file (.CUT). The commands that can add to the section report include Cut & Place, Cut Only, Dozer Push, Cast Blast Profile and Area to Section Report.

The report formatter is used to create the section report. This formatter allows you to choose the fields to report and the report layout. The report can also be output to Excel or Access.



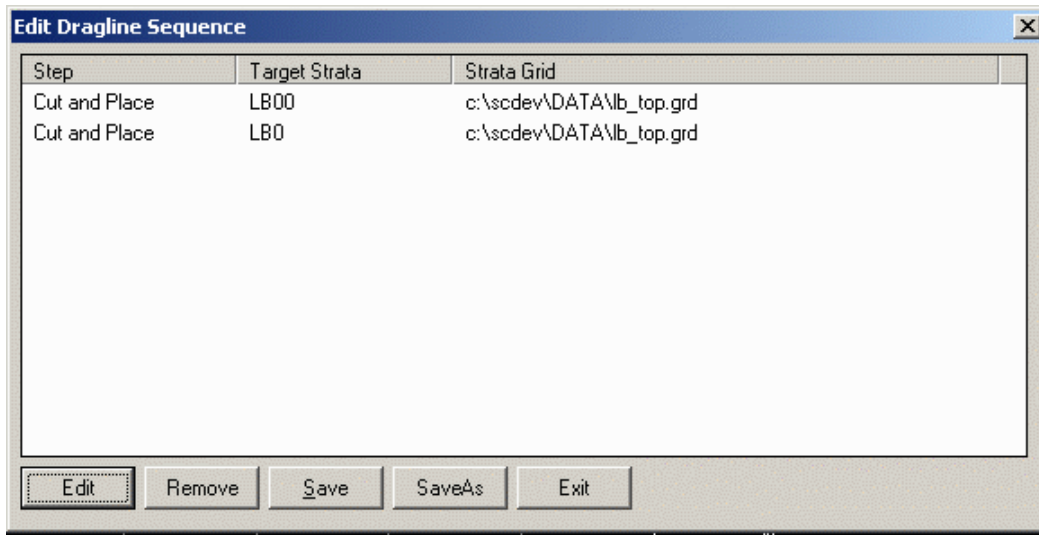
Pulldown Menu Location: Surface

Keyboard Command: cutreport

Prerequisite: A section report file (.cut)

Edit Dragline Sequence

This command edits the steps in a sequence file (.seq). The sequence file can be created with other commands such as Dozer Push, Cut and Place, and Cast Blast Profile. This command allows you to remove a step or edit a step. For editing, the target strata and surface file for the step can be changed.



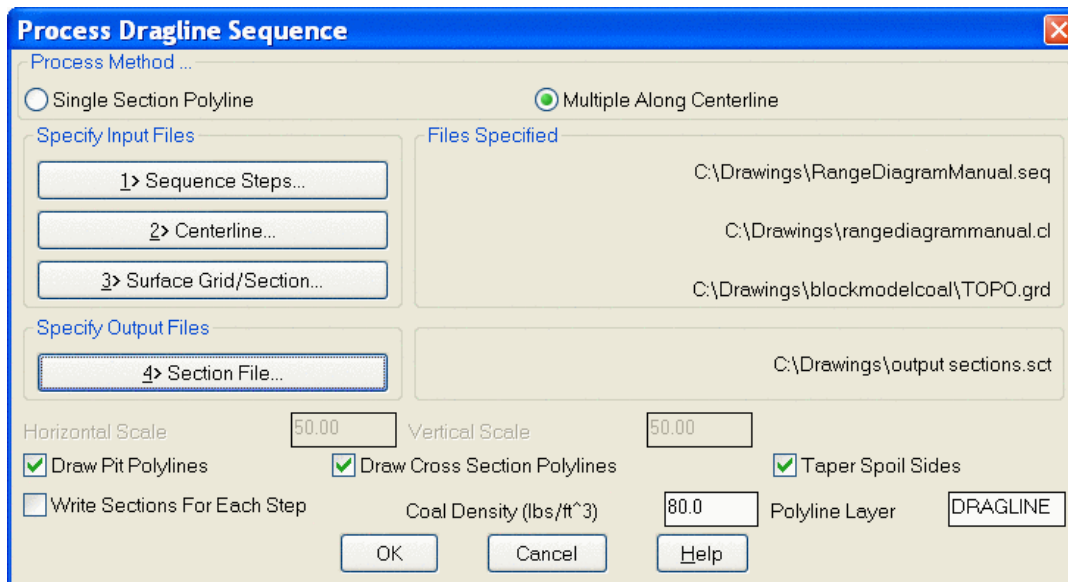
Pulldown Menu Location: Surface

Keyboard Command: edit_seq

Prerequisite: .seq file

Process Dragline Sequence

This command applies 2D cross-section dragline steps along a centerline to create a 3D model. The dragline steps are stored in a sequence file (.SEQ file) that can be created by the Cut & Place, Cut Only, Dozer Push, Flatten Spoil Top and Blast Cast commands.



There are two different methods for Process Dragline Sequence. The Multiple Along Centerline method applies the dragline sequence along a centerline for a range of stations. The centerline defines the dragline alignment and the mining surfaces are defined by grid files and/or section files. This method outputs section files, 3D polylines and a volume report. The Single Section Polyline method applies the dragline sequence to one station in cross section view. The dragline position is defined by picking the centerline position in section view and entering the elevation at this point. The mining surfaces are defined by picking polylines for the ground surface and strata in section view. The output is drawn as polylines in section view for this one station.

Each of the steps has a centerline reference point and the control points for the step are relative to this reference point. For example, the top of the spoil pile for Cut & Place could be 75 feet right of the centerline. Process

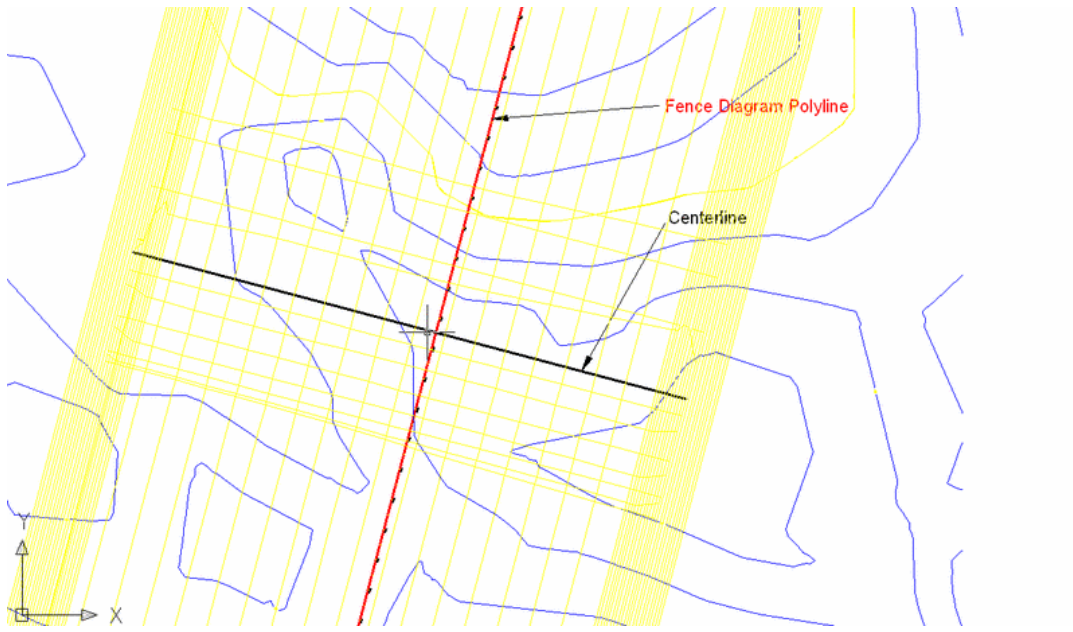
Dragline Sequence runs the steps along the centerline. A centerline file (.CL file) can be created from a polyline with the Polyline to Centerline command.

The existing ground can be modeled by either a grid file (.GRD file), triangulation file (.TIN file) or section file (.SCT file). Process Dragline Sequence works by repeating the 2D cross-section steps for each section along the centerline. When using a grid file surface, the program will prompt for a station interval and will create ground cross-sections at the specified offset left and right of the centerline at this interval before applying the dragline steps. When using a section file surface, the program will apply the dragline steps at the interval of the section file. There is an option to output the results of dragline steps to a section file. Besides outputting the final section results, sections at the end of each step can be saved to section files. The section files of each step are named automatically by adding the step sequence number to the original final section name.

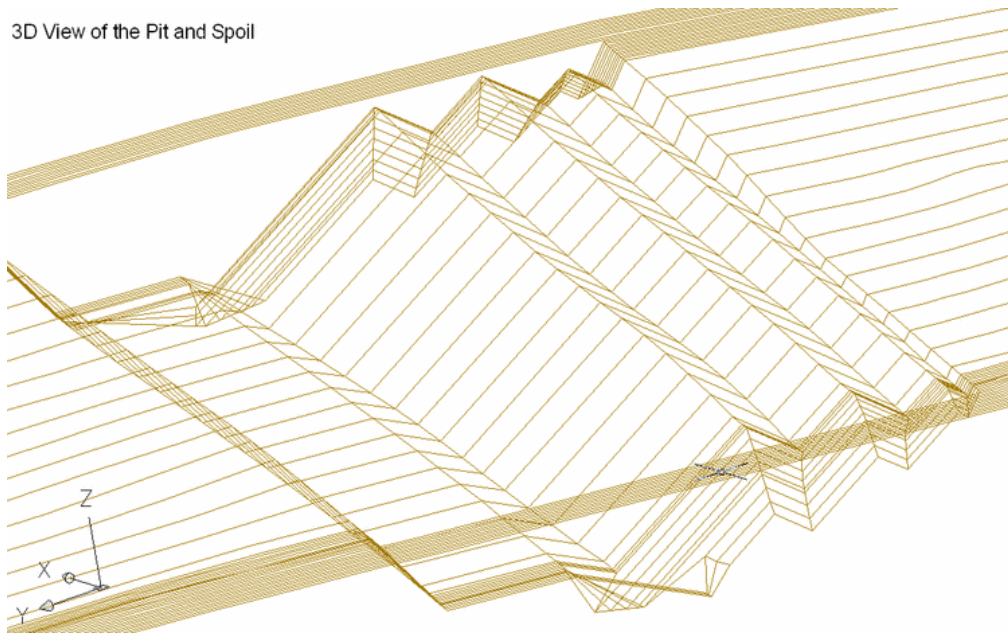
The dragline steps have an option to use a strata polyline which makes the control points relative to the strata. If strata polylines are used, then the Process Dragline will prompt for a grid file that models the strata. For example, you could use a use a top of coal polyline in the Cut & Place command to define the bottom of the cut area. With the store sequence file (.SEQ) option on, the program will prompt for a name for the polyline and you could enter "coal". This name is stored in the sequence file and when Process Dragline Sequence in run, the program will prompt for you to select the "coal" grid file.

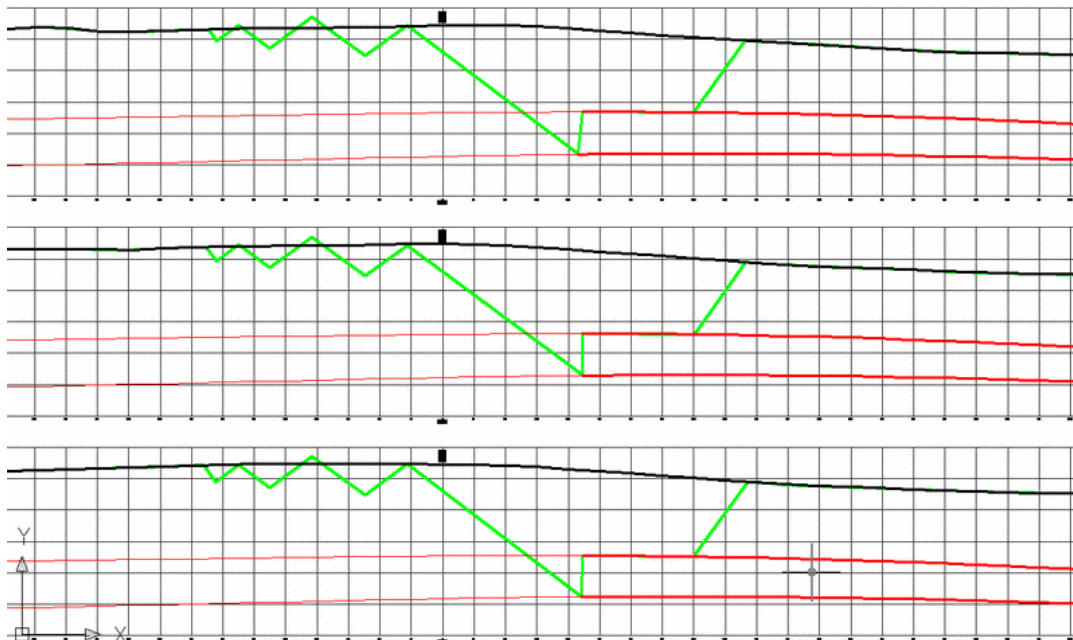
The program creates a report of the volumes for each step in the sequence. The Coal Density field in the dialog is used for calculating the coal tons for the report. The Taper Spoil Sides option adds stations before the first cut and after the last cut to show the repose angle of the ends of the fill spoil piles. There are options to draw 3D polylines of the pits. Draw Pit Polylines creates 3D polylines parallel to the centerline along the dragline control points such as the top of the spoil pile. Draw Cross Section Polylines creates 3D polylines perpendicular to the centerline for each cross section.





3D View of the Pit and Spoil





SurvCADD Edit : c:\scad2006\USER\scadrprt.tmp

File Edit Settings

Open Save Print Exit Find Screen Hide

Dragline Processing 04/11/200!

Sequence File> C:\Drawings\RangeDiagramManual.seq

From Station: 0.00 To Station: 1833.21

Step	Cut (C.Y.)	Fill (C.Y.)	Coal (Tons)
Cut	2437033.05		2631995.70
Cut	723039.60		780882.77
Cut & Place	852305.64	852305.64	
Cut	801051.35		865135.45
Cut & Place	1478998.33	1478998.33	
Cut	814463.22		879620.27
Cut & Place	1398399.47	1398399.47	
Total:	8505290.66	3729703.45	5157634.19

Prompts

Process Dragline Sequence dialog

Station interval <100.0>: *press Enter*

Processing step> 1

Processing step> 2

Drawing offset 3D polylines: STEP1-SPOIL3

Drawing cross section 3D polylines

Writing section file> D:\SC14\DATA\final.set

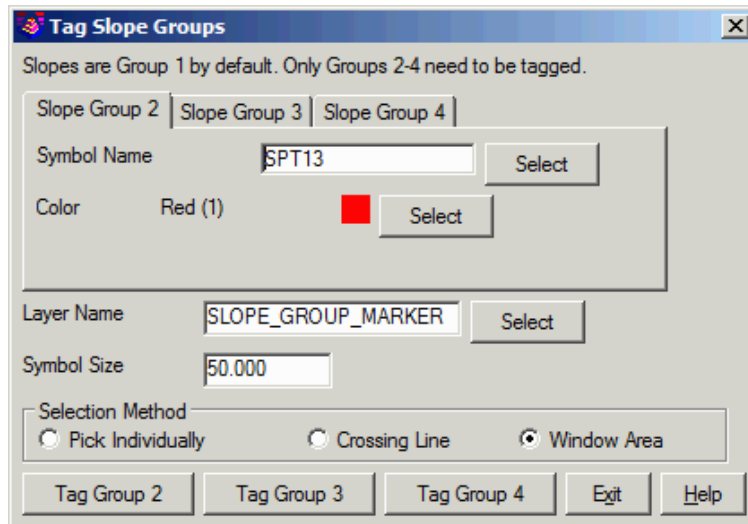
Pulldown Menu Location: Surface in Advance Mining

Keyboard Command: steps

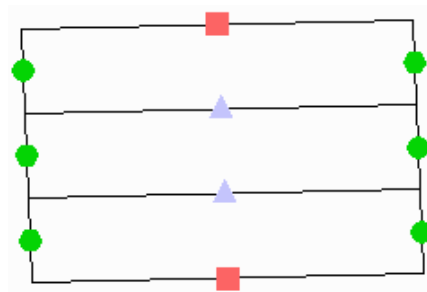
Prerequisite: Centerline file, Sequence file and surface grid file

Tag Slope Groups

This command places special symbols on pit polylines to indicate different slope groups to use in Design Bench Pit. The Design Bench Pit command supports up to four separate sets of cut slope definitions to be used on different sides of the pit polyline. Using the symbols from Tag Slope Groups is a way to batch process many pit polylines with separate slope groups. When Design Bench Pit finds symbols from Tag Slope Groups, the program skips the prompting to specify and slope group sides and uses the assignments from the symbols. For each edge in the pit polyline, the program looks for a slope group symbol. If a symbol is not found on a segment, then slope group #1 is used. So only symbols only need to be placed for slope groups 2-4.



The options dialog defines the symbol from the Symbol Library to use for each slope group as well as the symbol size, layer and color. There are a few methods for placing the symbols. The Pick Individually method prompts to a pick point on the pit segment and the symbols are added one pick at a time. The Crossing Line method prompts for two points and places a symbol at each pit polyline intersection along the line between the two points. The Window Area method prompts for a series of points to define the perimeter of an area and a symbol is drawn on the midpoint of all pit segments within this area.



Pulldown Menu Location: Surface

Keyboard Command: tag_slope_groups

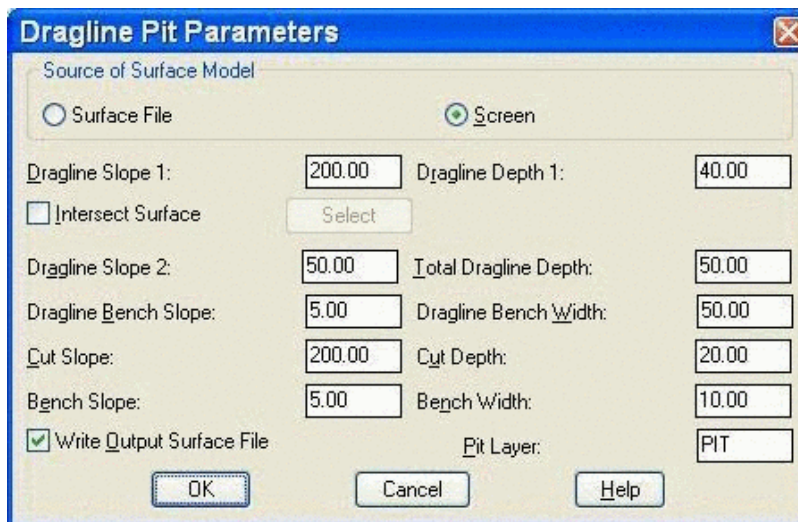
Prerequisite: Pit polylines

Design Dragline Pit

This command designs a pit into the existing ground surface using a set of dragline and bench slopes. The dragline slopes are for the portion of the pit excavated by the dragline and the bench slopes represent the truck-shovel excavation. The pit is drawn with 3D polylines and the pit volumes for the dragline and truck-shovel are calculated.

Before running this command, the base of the pit should be drawn as a closed 3D polyline. This pit polyline should also be assigned the elevations for the bottom of the pit. One way to set the elevation is to create a grid file for the bottom of the pit. For example, use Make Strata Grid Files to make a grid of the top of coal elevation. Then use the command 2D to 3D Polyline by Surface Model to convert the pit polyline into a 3D pit polyline with the grid elevations. This 2D to 3D command usually creates more vertices than necessary for Design Dragline Pit. So it is good to run Reduce Polyline Vertices.

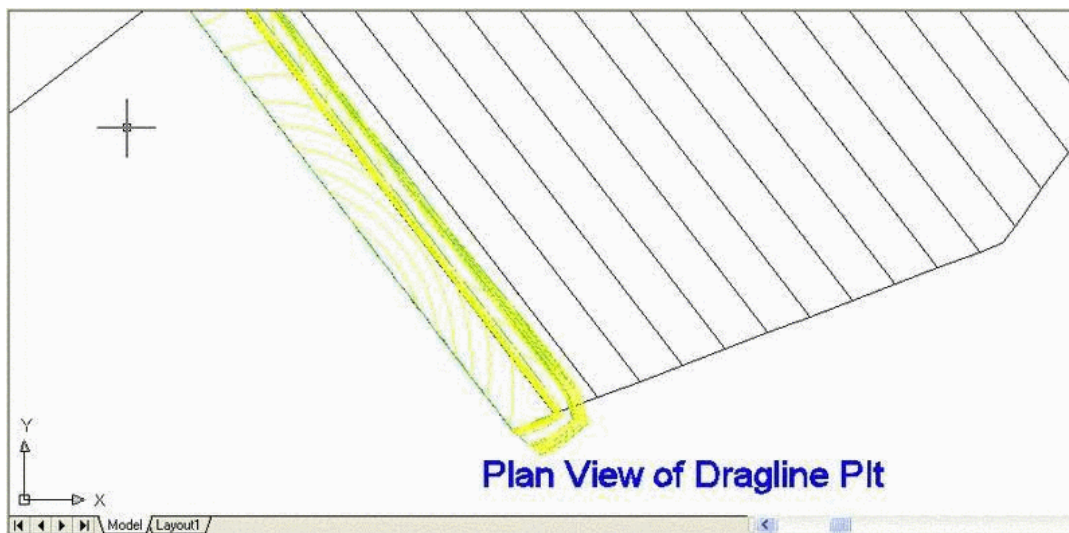
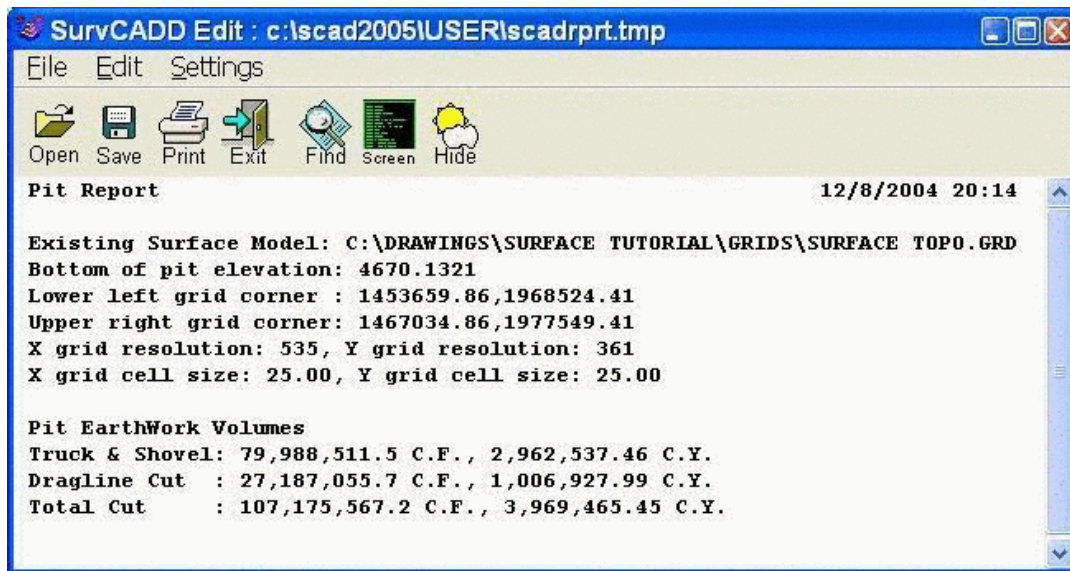
Design Dragline Pit starts with the dialog shown below. All slopes are entered as percent. There is an option for two dragline slopes to allow for different slopes in different material, though it will be reported as one volume. The break point between these slopes can be set as either depth or at a grid surface. The grid surface could be a grid file for the transition strata. Next there is a dragline bench width and slope followed by cut and bench slopes. The cut-bench slopes repeat until intersecting the surface. The resulting modified surface can be saved as a grid file (.grd) with the Write Output Grid File option.



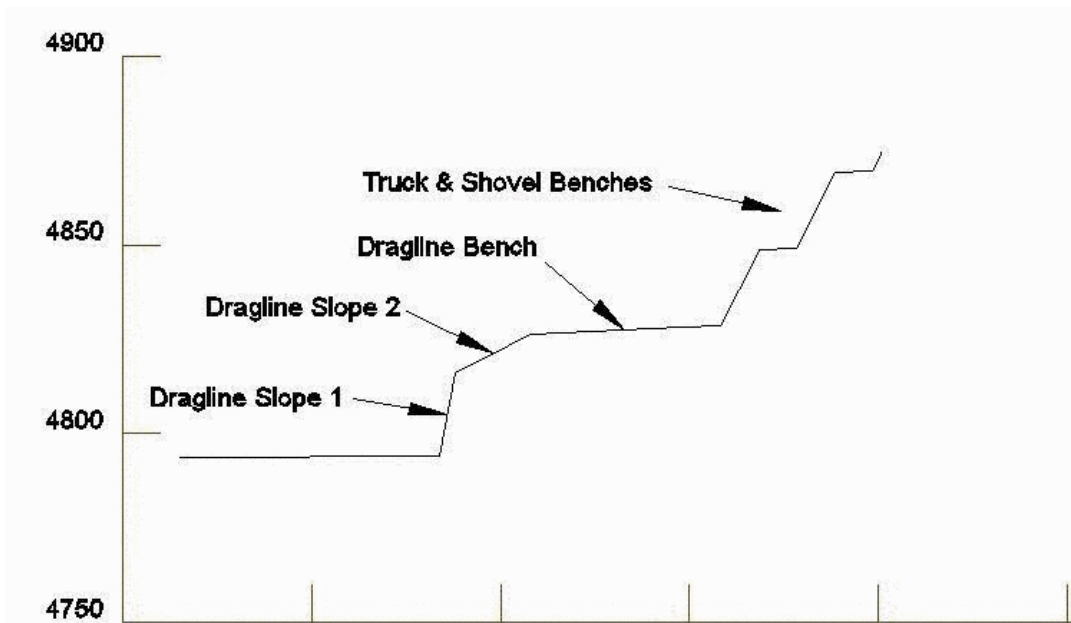
The program works by starting from the base pit polyline and applying the slopes until the surface is reached. The surface is modeled either by a grid file or from the entities on the screen. With the screen option, all 3D entities in the pit area are used to model the existing surface.

If a pit already exists to one side of the current pit, then this existing pit should be part of the surface model. Then the program will intersect the surface right away on this side without creating all the dragline and bench slopes. The Write Output Grid File allows you to save a grid file at the end of the command that represents the surface updated with the new pit. Then you can run Design Dragline Pit for the next pit and use the grid file output from the previous time as the new surface model.

The volume report shows the dragline cut and the truck-shovel cut. The dragline cut includes the volume from the pit base up to the top of Dragline Slope 2. The truck-shovel cut is the remaining volume up to the surface.



Plan view of dragline pit polylines



Profile view of dragline pit slopes

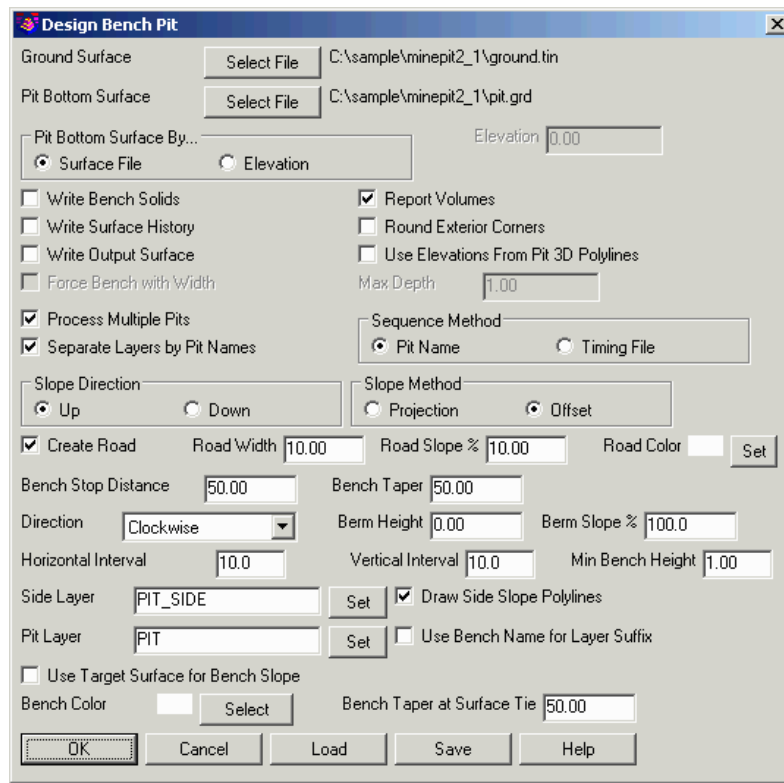
Pulldown Menu Location: Surface

Keyboard Command: minepit

Design Bench Pit

This command creates pits where the sides are a series of slopes with benches. The side slopes start from a closed polyline. This pit polyline can be either the bottom of the pit and the slopes run up and out to intersect the ground surface, or the pit polyline can represent the top of the pit and the slopes run down and in, to intersect the pit bottom grid. Different slopes can be used for different sides of the pit. The pit is drawn as 3D polylines and the pit volumes are reported.

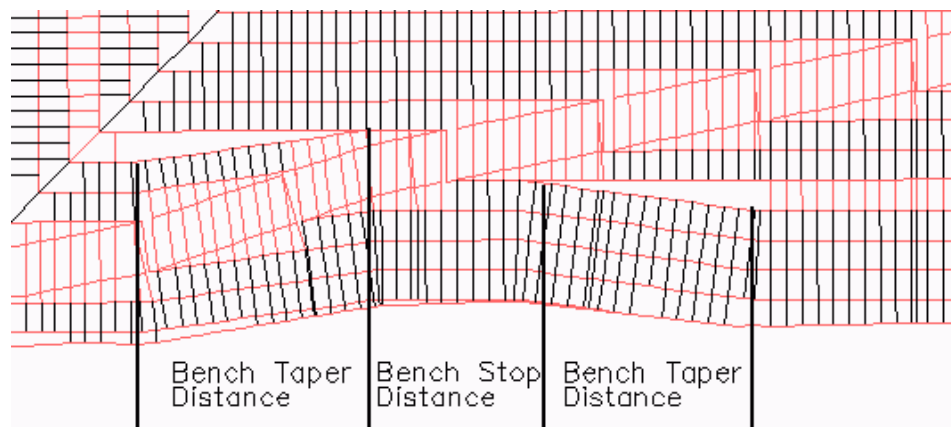
Before starting this command, the pit perimeter should be drawn as a closed polyline, 2D at zero elevation, or at 3D elevations on a surface. The program also needs triangulation or grid files for the existing ground surface and the bottom of pit surface. Design Bench Pit starts with the dialog shown below to specify these grid file names.



- **Ground Surface:** This is the surface for the top of pit perimeter.
- **Pit Bottom Surface:** This is the surface that will represent the bottom of the pit.
- **Pit Bottom Surface By:** This option chooses between using a surface model or fixed Elevation for the target for the bottom of the pit.
- **Write Bench Solids:** This option creates solid model files (.MDL) for the volume of each bench in the design.
- **Report Volumes:** You can turn off this option to save processing time if you don't need this routine to report the pit earthworks.
- **Write Surface History File:** This output file is a GSQ file that is used for volumes in Surface Mine Reserves which leads up to surface mine timing and scheduling on these volumes, or for 3D Viewing as a movie with View Surface History File command.
- **Round Exterior Corners:** This option will create rounded corners on the outside edges of the pit for a more realistic design. When this is off, the corners are sharp and angular.
- **Write Output Grid File:** This option will create a grid file (GRD) of the design. It includes the original ground surface grid file, with the pits built onto it.
- **Merge Design With Ground:** This option combines the bench pit surface with the existing ground for the output surface.
- **Use Elevations From Pit 3D Polyline:** This will use the elevations on the polyline as the start of the design. It will start the pits at the elevations the polylines are drawn at instead of starting at the Pit Bottom Grid for going up, or the topography when going down.
- **Force Bench with Width and Max Depth:** This option applies when the Slope Method is in Projection mode. This option creates the bench up to the specified Max Depth beyond the target surface.
- **Process Multiple Polyline by Pit Names:** This option is to process multiple pit perimeter polylines all at

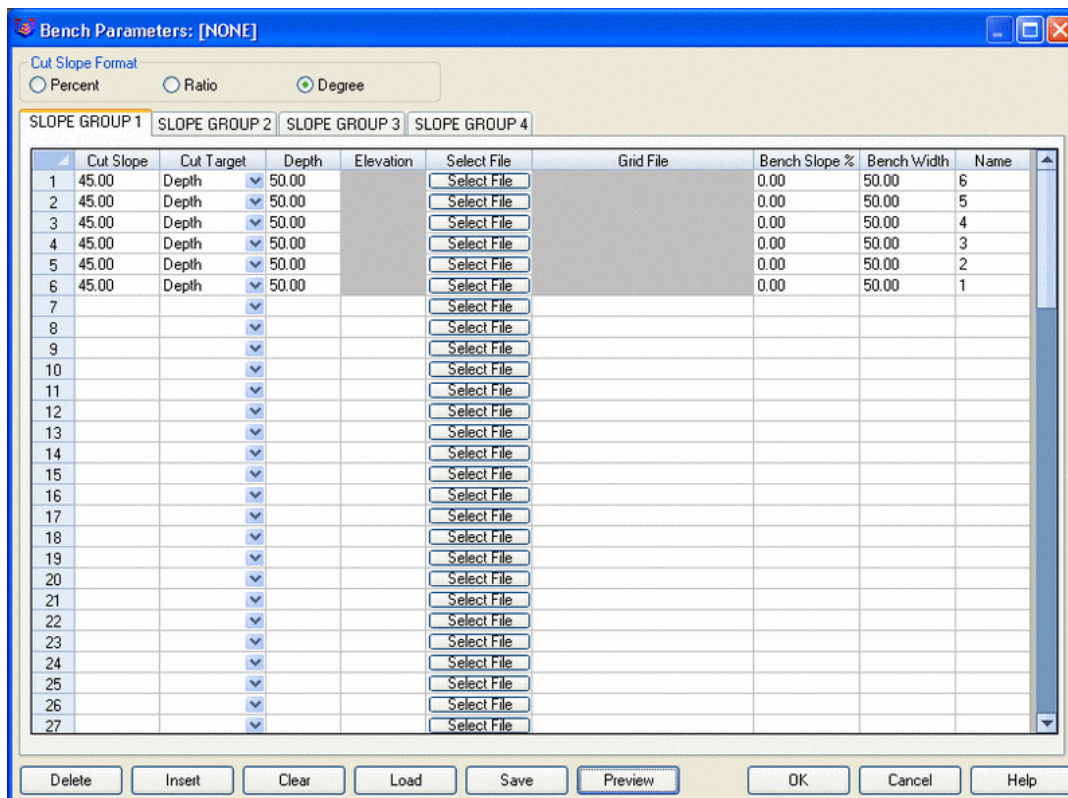
once. The pit polylines must have pit/site names assigned and the program will process them in the order of their pit names. As each pit polyline is processed, the ground surface is updated with the pit volume removed. Then the next pit will use this updated ground surface. When Process Multiple is active, the **Sequence Method** choose the way to order the pits for processing. The **Pit Name** method processes the pits in alphabetical order. The **Timing File** method uses the pit assignment order from Surface Equipment Timing from the .TIM file.

- **Separate Layers by Pit Names:** This option puts the 3D polyline break lines of each pit on their own separate layer. It takes the Pit Layer, and increments them by -2, -3 etc.
- **Create Road:** This option carves a rough road into the pit design. In the dialog, the road **Direction**, **Road Width** and **Road Slope %** are specified along with a **Road Color** for the road polylines to create. When this option is active, the program will prompt for a road starting point along the pit perimeter. The cut slopes are shifted to make room for the road. The **Bench Stop Distance** is for suspending the bench as it crosses the road to avoid combining the road and bench widths at the crossing which tends to shift the cut slopes too much. Within the Bench Stop Distance, the bench has a zero width. The Bench Stop is centered at the road crossing. The **Bench Taper** to the distance that the bench transitions from full width to zero at the Bench Stop. The **Berm Height** adds a berm and makes the road wider to fit the berm. The slope of the berm is set by the **Berm Slope**.

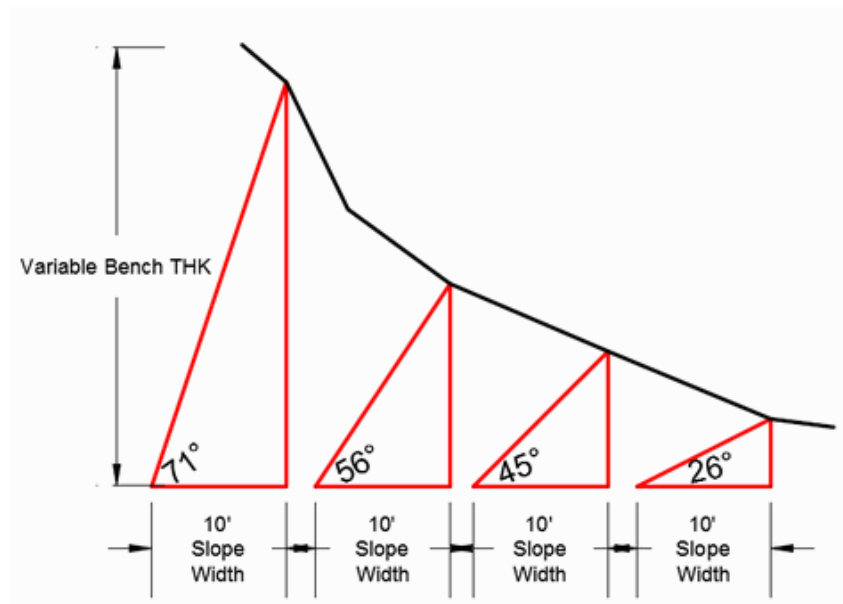


- **Slope Direction:** Up will start at the pit bottom and bench up and out to the surface topography. Down will start at the top surface, and bench down and in to the pit bottom.
- **Slope Method:** Projection is a method that projects the 3D polylines down the slope, across the benches. Offset is a method that offsets the horizontal "toe and crest" lines to create the pit shells this way. Both methods will have horizontal and vertical breaklines, but the method they are generated is different. Each method has its benefits and might work better than the other for each unique scenario. If there is a problem with one method, try the other to see if it handles it better.
- **Horizontal & Vertical Interval:** These settings control the distance to draw the 3D breakline polylines. The horizontal interval will run down the slope, and across the bench. The vertical interval will run parallel with the benches. It should be a factor of the bench height if possible, but not required.
- **Min Bench Height:** The bench is not created when the side slope depth is less than the specified amount.
- **Side Layer:** This is the layer of the 3D polylines drawn perpendicular to the benches, and running down slope.
- **Pit Layer:** This layer is applied to the 3D polyline break lines running parallel to the benches.

- **Use Bench Name for Layer Suffix:** This option adds the bench name to the Pit Layer so that each bench can have a unique name which can be useful for having different colors for each bench for visualization or for isolating benches by layer.
- **Draw Side Slope Polylines:** This option chooses whether to draw the 3D polylines running perpendicular to the benches, and down slope.
- **Use Target Surface for Bench Slope:** When the benches use a surface file for the target, this option makes the bench slope follow the target surface model. Otherwise, the bench slope is set in the next dialog.
- **Bench Color:** This setting puts the bench 3D polylines in the Pit Layer on a different, specified color so they stand out against the slope breaklines.
- **Bench Taper At Surface Tie:** When a bench meets the Ground Surface, this option tapers the bench width to zero over this specified distance.



The next dialog defines the cut and bench slopes. **Cut Slopes** are entered as ratios. The **Cut Target** can be either a fixed depth number, a fixed elevation, a fixed width or to a grid file. For a grid file, the program will find the intersection of the cut slope with the grid surface and will end the cut slope at this intersection. For example, you could make a grid file for a second coal seam and have the bench occur at this coal seam. For the width method, the program creates a cut slope to the target surface with a variable slope to reach the surface at a fixed cut slope width. This width method can only be used on the final bench.



Cut Slope by Width method

After the cut slope, the bench slope and width are used. The cut and bench slopes are applied in order until there is an intersection with the surface. Once you have all the cut slopes defined, you can use the Save button to save these slope settings to a .PIT file. Then these settings can be recalled later with the Load button.

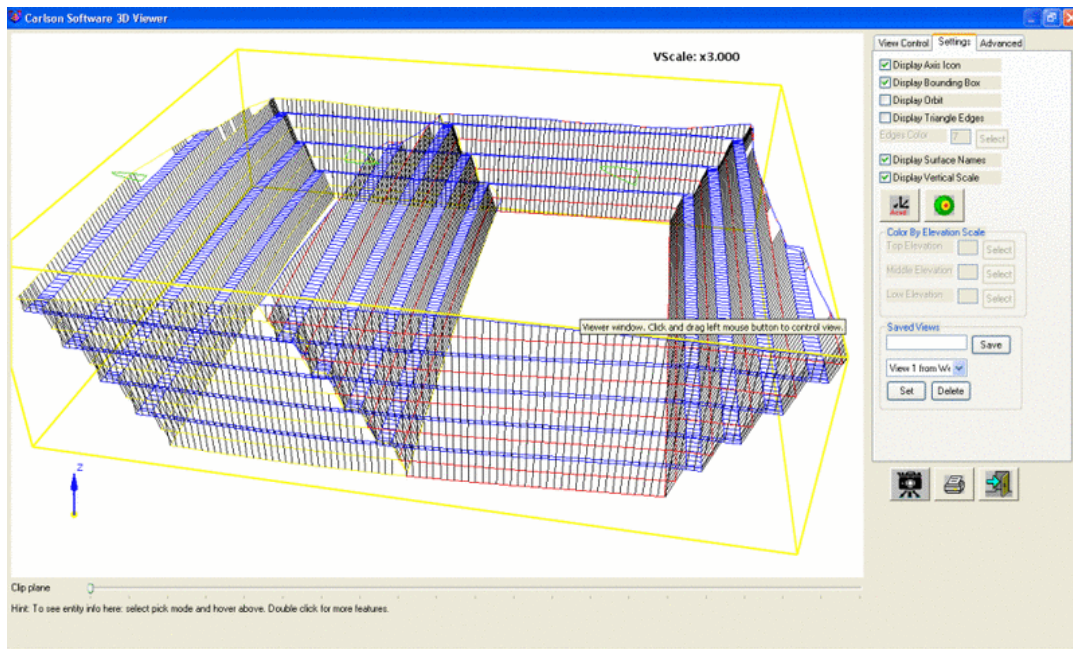
The **Pick Perimeter For Z Range** function prompts to select a perimeter polyline from the drawing. This function reports the elevation range of the Ground Surface along this perimeter which is useful as a reference for setting the bench elevations.

Four different sets of slope schemes can be defined. To define the another slope scheme, select other Slope Group tab. The set of slopes that you are currently editing is indicated by the selected tab. If you define different sets of slopes, then the program will prompt you to pick which sides to apply each set of slopes to. All sides are assumed to be slope type one. So you only have to identify types two, three and/or four. If only one slope group is defined, then you will not be prompted to select any additional sides.

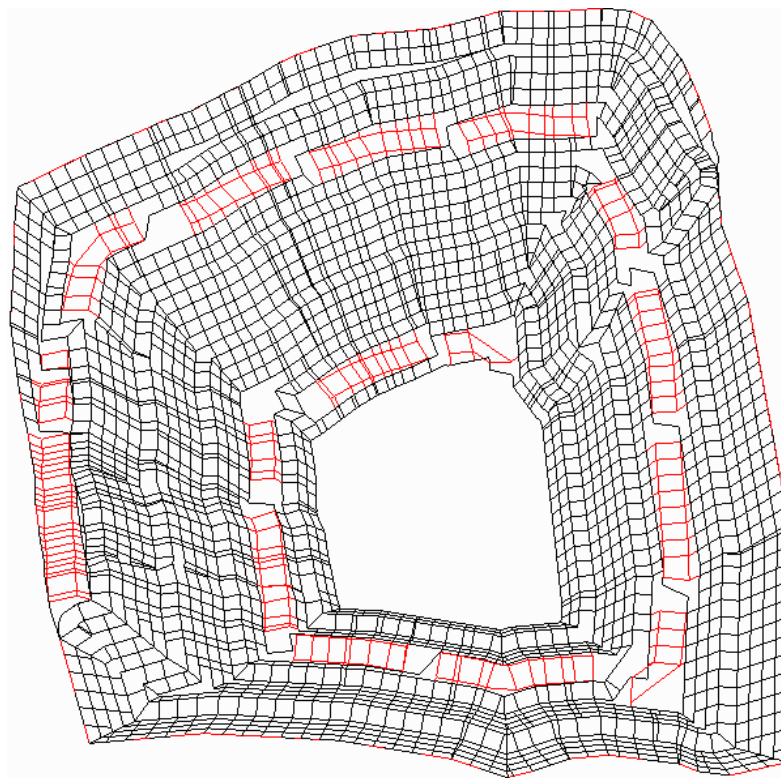
If a pit already exists to one side of the current pit, then this existing pit should be part of the Ground Surface grid file. Then the program will intersect the surface right away on this side without creating all the bench-cut slopes.

Here are a couple of examples of the various settings in Design Bench Pit.

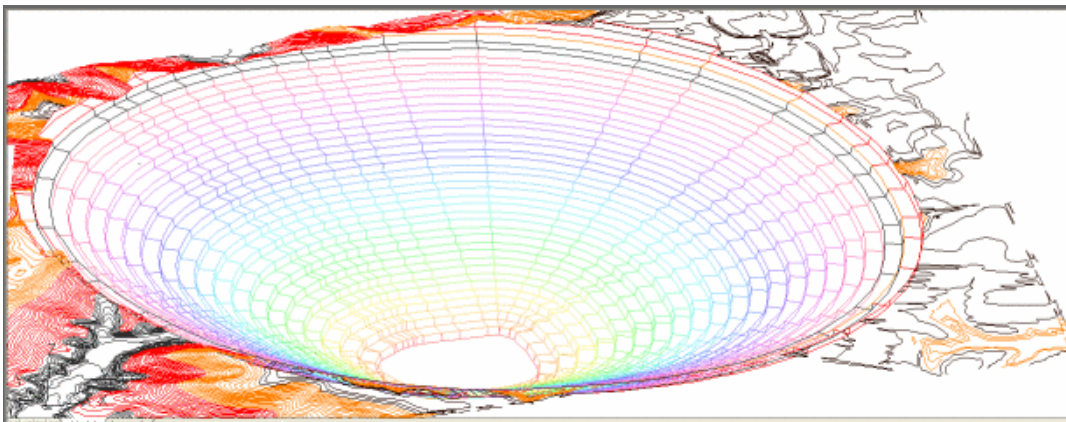
- Offset Method:



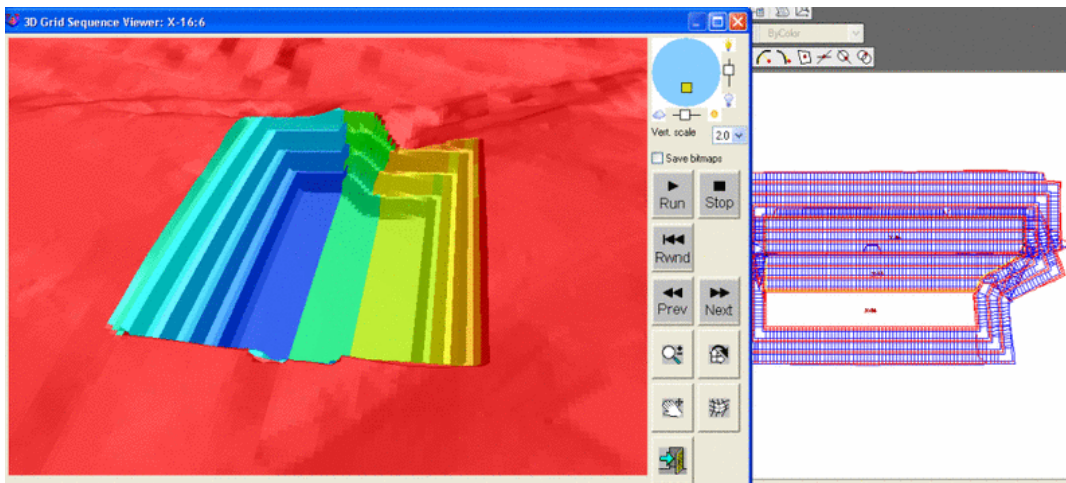
- Offset Method with Road in red:



Projection Method:



- **Multiple Pits with Offset Method and Viewing the Grid History File in 3D.**



Prompts

Pick the pit polyline: *pick the closed polyline*

Pick pit polyline segment for side 2 slopes (Enter to continue): *pick the segment near midway, it will highlight*

Pick pit polyline segment for side 2 slopes (Enter to continue): *pick the segment near midway, it will highlight*

Pick pit polyline segment for side 2 slopes (Enter to continue): *press Enter to continue*

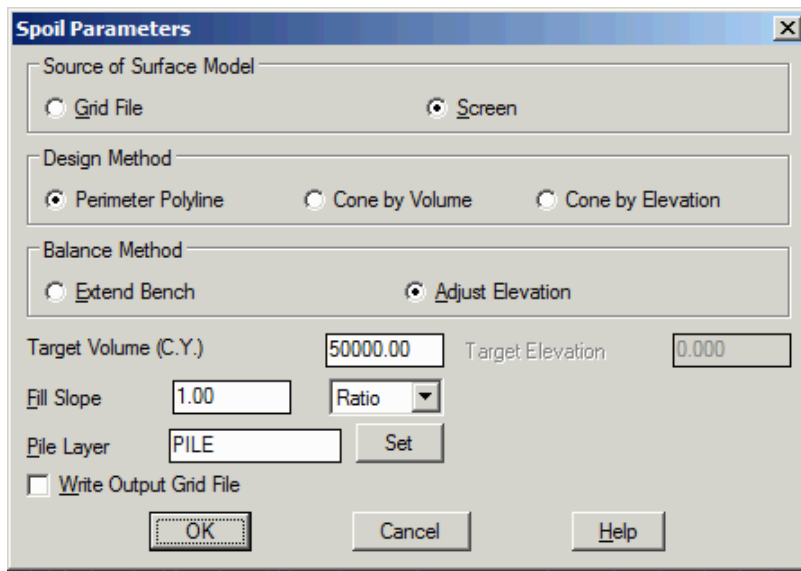
Pulldown Menu Location: Surface in Surface Mining Module

Keyboard Command: minepit2

Design Spoil Pile

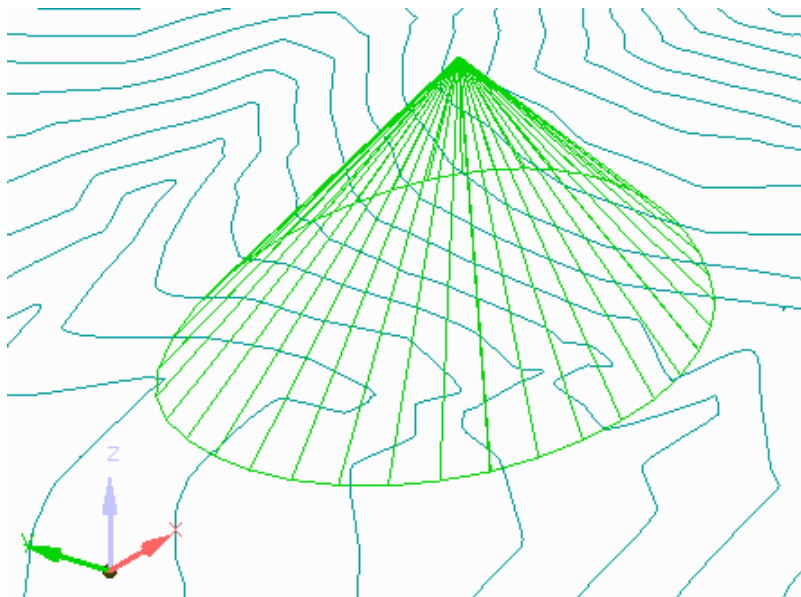
This command creates a spoil pile at a target volume or elevation. There are a few design methods. The Perimeter Polyline method takes a 3D polyline outline of the top of a spoil pile and adjusts this polyline by either changing the elevation or extending a side to reach a specified volume for the pile. The resulting pile is drawn as 3D polylines showing the side slopes and where the pile ties into the existing surface. The Cone methods take a single point and slope to build a spoil cone. The cone can be sized by either target volume or by height.

For the Perimeter Polyline method, you can choose between Extend Bench and Adjust Elevation. The extend bench option will prompt for a side of the pile to offset. Holding all the other sides with their elevations, the program will offset the selected side until the target volume is reached. For the adjust elevation, the entire pile polyline is moved up or down to obtain the target volume. Before starting this command in Perimeter Polyline mode, you need to draw a closed 3D polyline that represents the top of the pile. The fill slopes will start at this polyline and will run out and down until the slopes intersect the surface model.

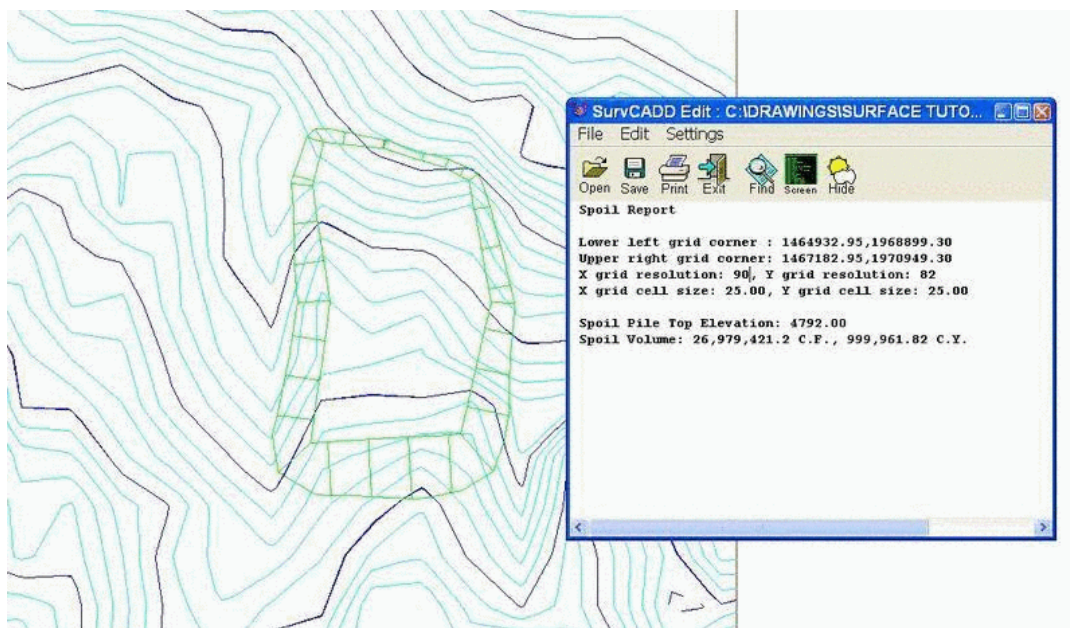
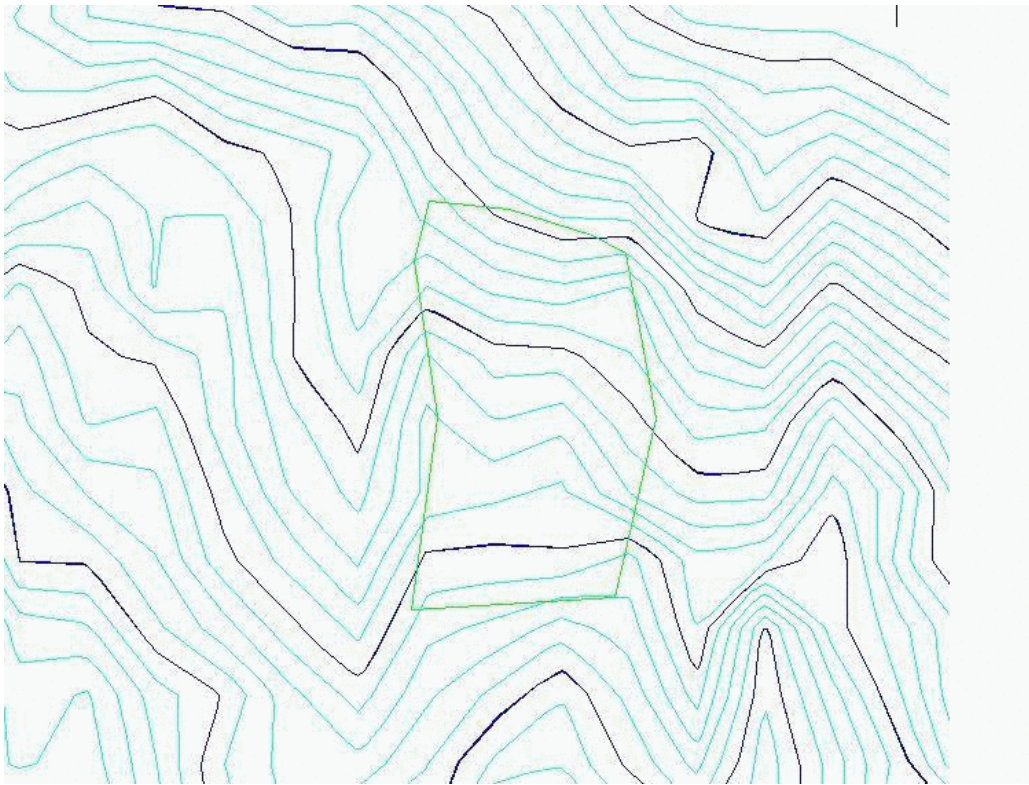


The Fill Slope can be specified in ratio, percent or degree format. The Pile Layer is for the 3D polylines that this command creates for the spoil. The Write Output Grid File option creates a .grid file of the spoil surface.

As the program runs, it will seem to draw several piles as it tries to find the target volume. The resulting volume will not match exactly the target volume but will be very close. The smaller the grid cell size, the closer it will be. This difference is just to save processing time.



Spoil Pile by Cone method



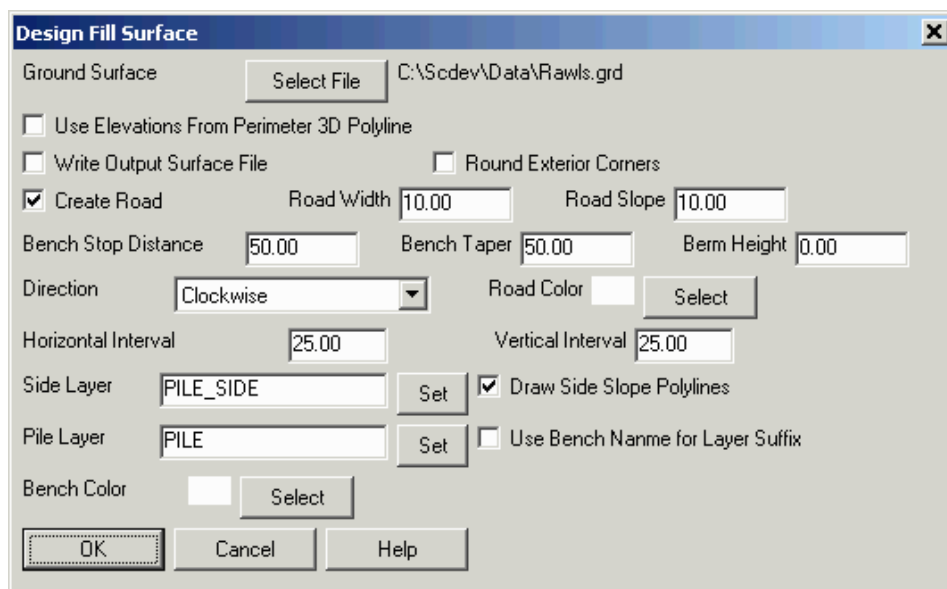
Pulldown Menu Location: Surface
Keyboard Command: spoil2

Design Fill Surface

This command creates fill pile where the sides are a series of slopes with benches. The side slopes start from a closed polyline. The perimeter polyline is the toe of the pile, and the slopes will be up and in from the footprint of the pile. Different slopes can be used for different sides of the fill. The fill is drawn as 3D polylines and the fill volumes are reported.

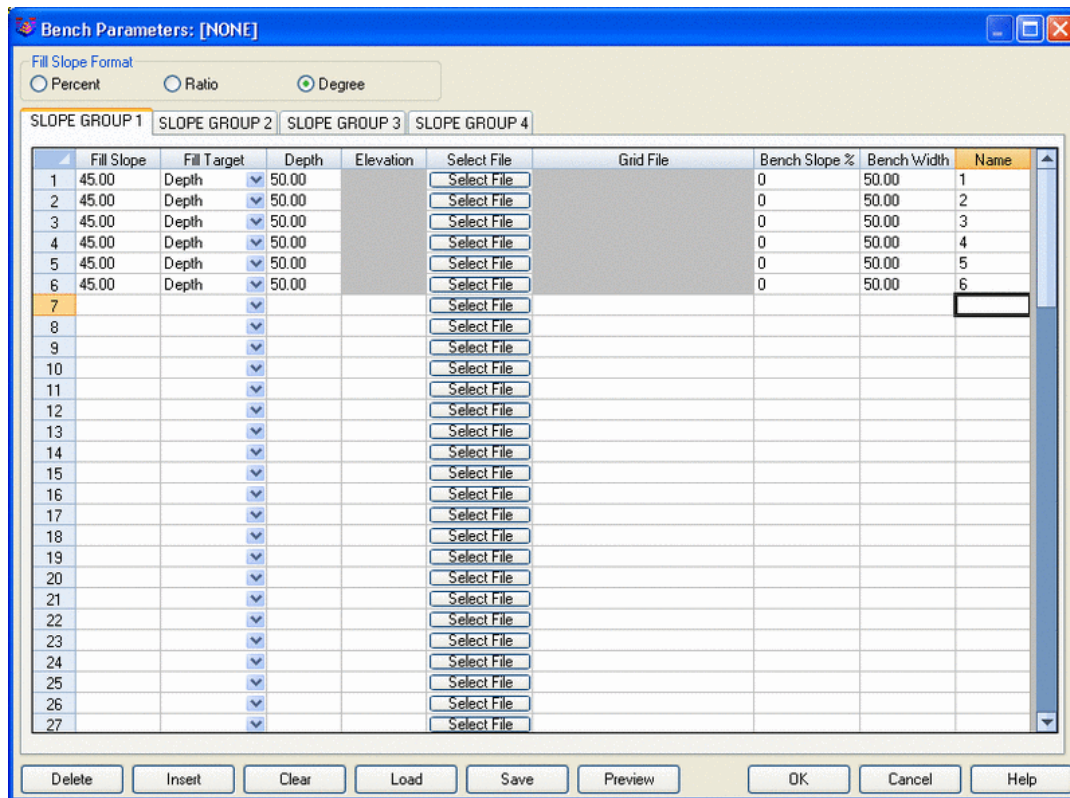
Before starting this command, the perimeter should be drawn as a closed polyline, 2D at zero elevation, or at 3D elevations draped onto a surface. You also need a grid file for the existing ground surface. Design Fill Surface starts

with the dialog shown below where you specify the grid file name.



- **Ground Surface:** This is the grid or triangulation surface to place the fill pile on.
- **Use Elevations From Perimeter 3D Polyline:** This will use the elevations on the polyline as the toe of the fill pile to start the design.
- **Write Output Grid File:** This option will create a grid file (GRD) of the fill. It includes the original ground surface grid file, with the fill pile built onto it.
- **Round Exterior Corners:** This option will create rounded corners on the outside edges of the pile for a more realistic design. When this is off, the corners are sharp and angular.
- **Create Road:** This option carves a rough road into the pile design. In the dialog, the road **Direction**, **Road Width** and **Road Slope %** are specified along with a **Road Color** for the road polylines to create. When this option is active, the program will prompt for a road starting point along the pile perimeter. The cut slopes are shifted to make room for the road. The **Bench Stop Distance** is for suspending the bench as it crosses the road to avoid combining the road and bench widths at the crossing which tends to shift the cut slopes too much. Within the Bench Stop Distance, the bench has a zero width. The Bench Stop is centered at the road crossing. The **Bench Taper** to the distance that the bench transitions from full width to zero at the Bench Stop. The **Berm Height** adds a berm and makes the road wider to fit the berm. The slope of the berm matches the cut slope.
- **Horizontal & Vertical Interval:** These settings control the distance to draw the 3D breakline polylines. The horizontal interval will run down the slope, and across the bench. The vertical interval will run parallel with the benches. It should be a factor of the bench height if possible, but not required.
- **Min Bench Height:** The bench is not created when the side slope depth is less than the specified amount.
- **Side Layer:** This is the layer of the 3D polylines drawn perpendicular to the benches, and running down slope.
- **Pile Layer:** This layer is applied to the 3D polyline break lines running parallel to the benches.
- **Draw Side Slope Polylines:** This option chooses whether to draw the 3D polylines running perpendicular to the benches, and down slope.
- **Use Bench Name for Layer Suffix:** This option adds the bench name to the Pile Layer so that each bench can have a unique name which can be useful for having different colors for each bench for visualization or for isolating benches by layer.

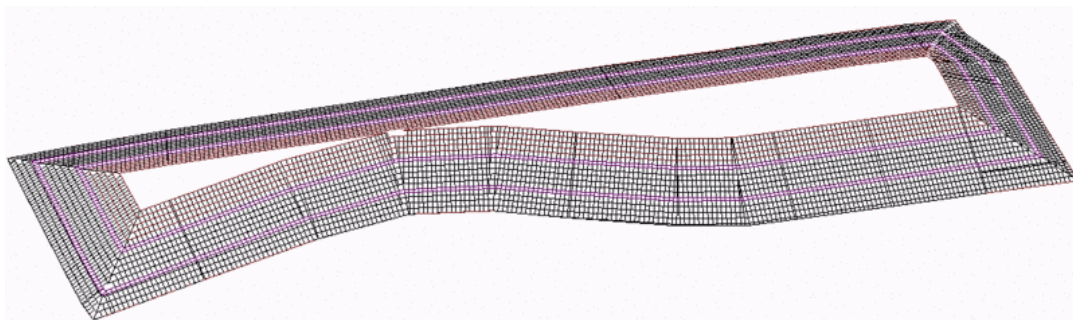
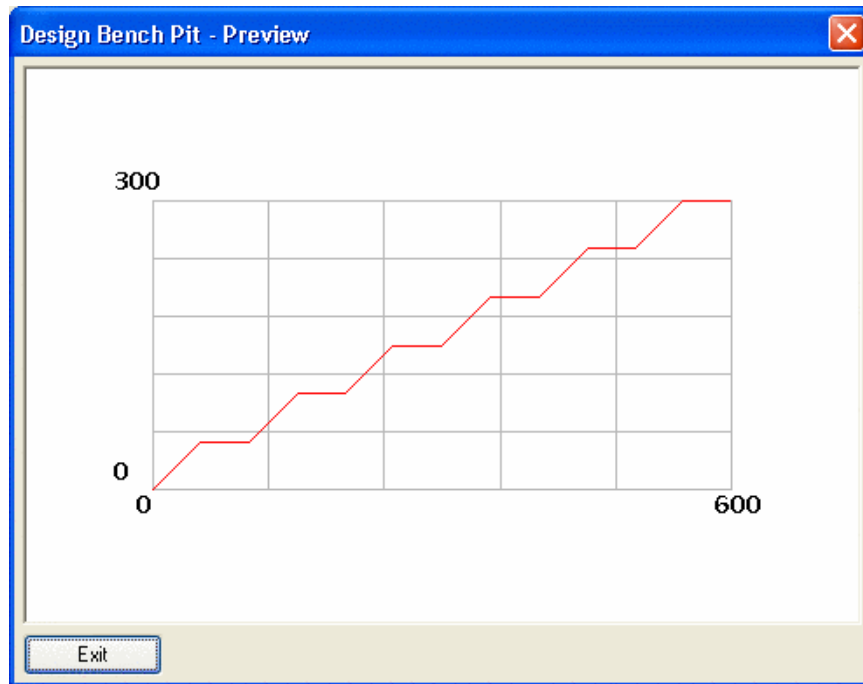
- **Bench Color:** This puts the bench 3D polylines in the Pile Layer on a different, specified color so they stand out against the slope breaklines.
- **Bench Taper At Surface Tie:** When a bench meets the Ground Surface, this option tapers the bench width to zero over this specified distance.



Fill Slope Format: The slope angle can be defined in percent, ratio or degrees. This is what is entered in the Fill Slope.

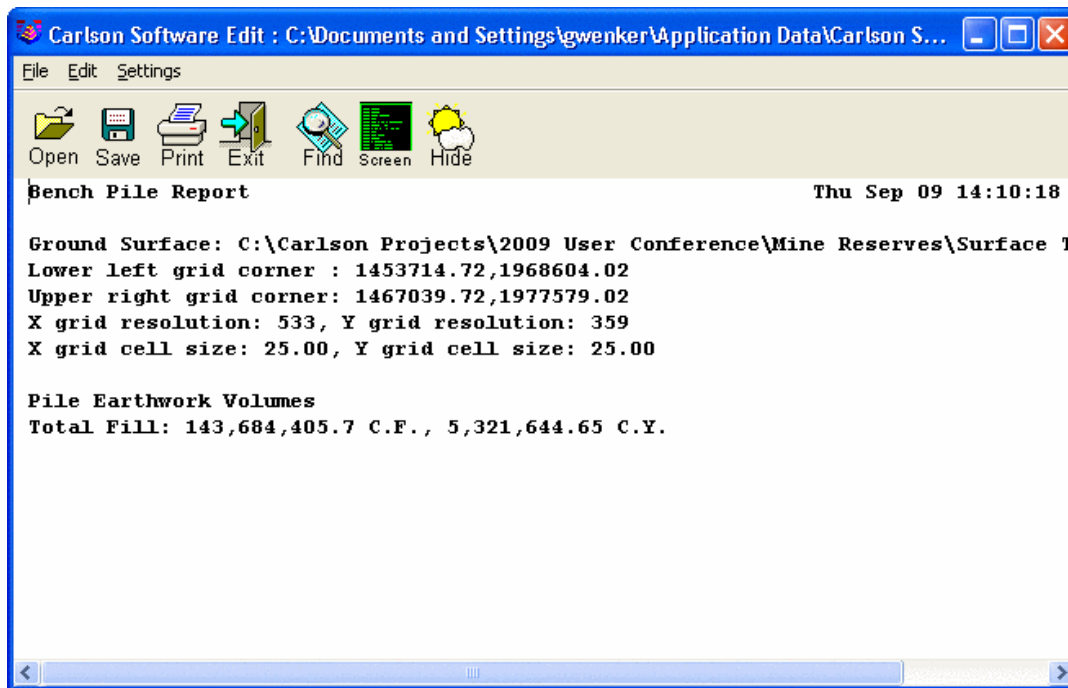
- **Slope Group:** Four different sets of slope schemes can be defined. To define the another slope scheme, select other Slope Group tab. The set of slopes are currently being edited is indicated by the selected tab. If different sets of slopes are defined, then the program will prompt to pick which sides to apply each set of slopes to. All sides are assumed to be slope type one. it is only necessary to select types two, three and/or four. If only one slope group is defined, then it will not prompt to select any additional sides.
- **Fill Slope:** The slope angle can be defined in percent, ratio or degrees. This is what is entered in the Fill Slope.
- **Fill Target:** The target to fill to for each bench can be a flat elevation, a depth above the previous surface or bench, or up to a grid file surface.
- **Depth:** The Depth must be filled in when the target is set to depth.
- **Elevation:** The Elevation must be filled in when the target is set to elevation.
- **Select File:** The file must be selected when the target is set to file.
- **Bench Slope %:** This setting will slope the benches down for drainage if desired. Normal setting for flat bench is 0.
- **Bench Width:** This is the horizontal width for the bench in feet or meters.
- **Bench Name:** This is used for pile design to name the benches for volumes. It does not have much application in the Fill version of this command.

- **Delete:** This deletes one line of data.
- **Insert:** This inserts a blank row to be filled in.
- **Clear:** This clears the entire slope group page, only the current group.
- **Load:** This loads a previously saved PIT file with the template saved.
- **Save:** This saves the template as shown in the window.
- **Preview:** This displays a preview of the current group to show how the cross section of the fill will look.



Design Fill Surface with 2 benches

The report window appears at the end of the command to display the volume of the fill in CY or CM.



Prompts

Design Fill Surface dialogs

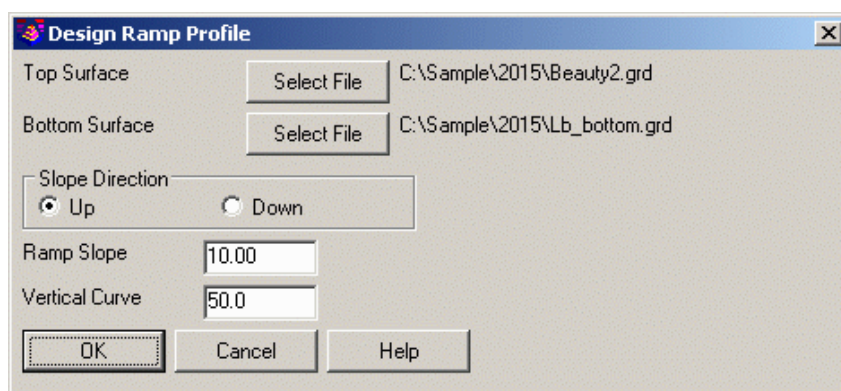
Pick pile perimeter polyline: Pick the perimeter

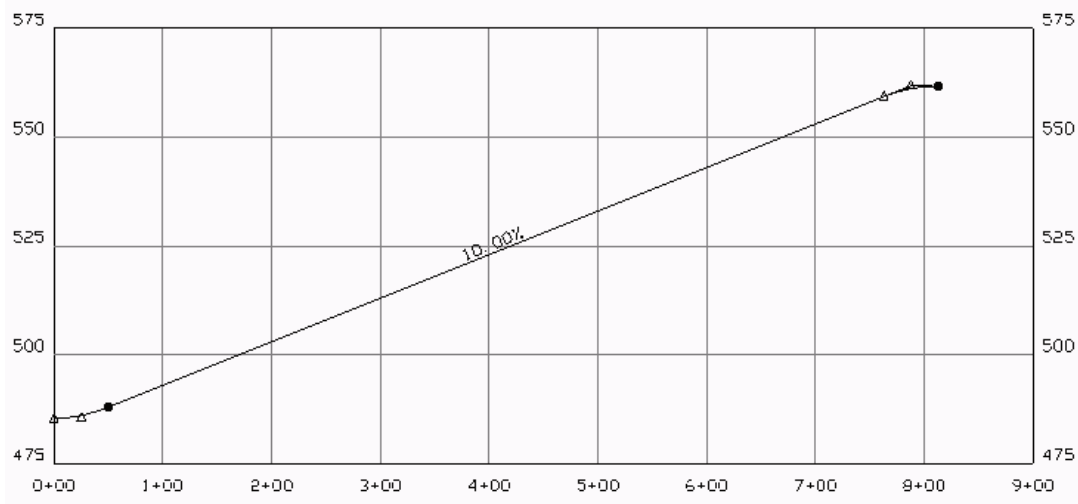
Pulldown Menu Location: Surface in Surface Mining Module

Keyboard Command: spoil3

Design Ramp Profile

This command creates a profile as well as the centerline for a ramp between two benches. The program uses two triangulation or grid surfaces for the top and bottom of the profile. The program prompts for the starting point for the ramp which can be either on the top or bottom surfaces depending on the Slope Direction setting in the dialog. The dialog also has settings for the Ramp Slope and the Vertical Curve for the ramp profile to use at the top and bottom of the profile. Besides the starting point for the ramp, the program also prompts for a point to set the ramp direction. A centerline file is created from the starting point and going straight in the direction for the length needed for the profile.





Prompts

Design Ramp Profile dialog

Pick start point: *pick a point*

Pick end point: *pick a point*

Profile To Write file selection

Centerline To Write file selection

Pulldown Menu Location: Surface

Keyboard Command: ramppro

Prerequisite: Top and bottom surfaces

Vertical Pit Quantities

This command calculates overburden and strata quantities. The strata thicknesses are defined in coal sections or drillholes. This routine requires a 3D closed polyline with elevations of the top of the seam. This polyline is also used as the perimeter of the pit. The routine calculates a surface model for the top of seam using this 3D polyline. Then a surface model for the ground surface is calculated from the selected surface entities such as points, lines or polylines with elevation (contour polylines). The overburden volume is calculated as the volume between these two surfaces within the 3D polyline. Each strata is modeled and the volume calculated for the area within the 3D polyline. It is important to note that the strip ratio reported in this command will use units of nonkey volume to key tonnage (yd^3/ton or m^3/tonne)

Prompts

Coal Section Configuration File

Select the file that defines the coal section sample points.

Pick top of coal polyline: *pick the closed polyline*

Choose the .grd file to update.

Select surface entities and at least 3 coal sections or drillholes.

Select objects: *pick the entities*

Make Grid File dialog Choose a grid resolution

Beginning date of takeup [format mm-dd-yy] : 6-1-92

Ending date of takeup [format mm-dd-yy] : 6-30-92

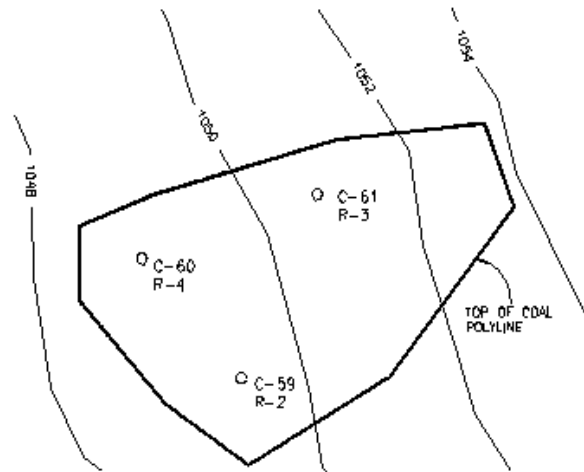
These dates used by Report Tons & Acres.

Ownership/description: Jones

This is the name of the data file to be updated for Report Tons & Acres.

Coal recovery percent <100.0>: 95

Write column report file (Yes/<No>)? *press Enter* This option creates a file minearea.dat in the data directory with the calculated quantities and other information in a column format.



Input for Vertical Pit Quantities: coal sections, 3D top of coal perimeter polyline, surface entities (contour polylines)

VERTICAL PIT WITH GRID METHOD

Individual Stratas Configuration

MINE: Watertown **MINED FROM TO**

AREA NO. 1 DESCRIPTION: .

AREA MINED (S.F.): 29686.84 **DEPLETED ACRES:** 0.682

VOLUME Coal: 147959.43 (S.F.), 5479.98 (C.Y.)

AVERAGE Coal THICKNESS (INCHES): 59.81 **(FEET):** 4.98

VOLUME Rock: 6785.11 (S.F.), 251.30 (C.Y.)

AVERAGE Rock THICKNESS (INCHES): 2.74 **(FEET):** 0.23

TOTAL MINING HEIGHT (INCHES): 62.55 **(FEET):** 5.21

AVERAGE Coal WT. (LBS/CU. FT.): 80.00

AVERAGE Rock WT. (LBS/CU. FT.): 120.00

Coal (TONS): 5918.38

Rock (TONS): 407.11

NON-RECOVERABLE COAL (TONS): 0.00 **COAL RECOVERY PERCENT:** 100.00%

TOTAL TONS : 6325.48 **PERCENT COAL BY WGT.:** 93.56%

VOLUME OVERBURDEN: 541529.65 (S.F.), 20056.65 (C.Y.)

STRIP RATIO: 3.39

Pulldown Menu Location: Surface

Keyboard Command: vertvol

Prerequisite: Coal sections

Update Grid File

This command recalculates the surface model in a .grd file by using additional point data along with the original point data. Only .grd files created by the Make Drillhole Grid File command, in StrataCalc, can be used by Update Grid File. Make Drillhole Grid file stores a point file of the original data used to calculate the grid in a file with the

same name as the grid file plus the extension .pnt. For example Make Drillhole Grid File will create strata.grd and strata.pnt.

The new point data for recalculation can be specified by picking points, entering point numbers that reference stored points in the current coordinate file, or selecting coal sections (from Place Coal Sections). When updating from coal sections, the program will prompt for a grid file that models the top of the coal sections. Since coal sections have thickness values but no Z values, this top of section grid is used to set the coal sections in the real Z axis. The program then updates the grid files that model each of the bottom elevation surfaces of the strata in the coal sections.

The updated grid can be saved in a new .grid file name or overwrite the original grid file by specifying the same file name. The location and resolution of the grid cannot be updated. So the original location and resolution should be made large enough to include any new area of point data to be added by Update Grid File.

Prompts

Update grid from points or coal sections (Points<Sections>)? *Points*

Grid File to Update file selection dialog

Choose the .grid file to update.

Pick point or point number: *35*

Pick point or point number: *press Enter*

Choose modeling method (<Triangulation>/Inverse distance/Kriging)? *press Enter*

Specify Grid File to Make

The updated grid will be stored in this .grid file.

Pulldown Menu Location: *Surface*

Keyboard Command: *grdupdate*

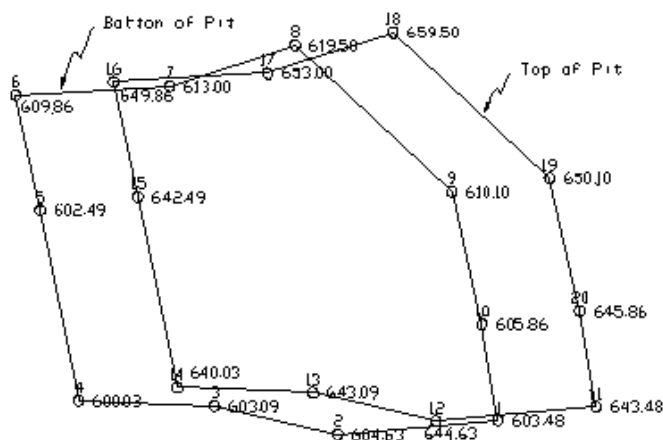
Prerequisite: Grid file created by Make Drillhole Grid File

Dragline Pits

This command calculates the total volume cut from a pit and the volume of a strata cut from the pit. The program requires two closed 3D polylines that represent the top and bottom of the pit. The top surface for volume calculations is modeled by the top polyline and any points in the bottom polyline that are outside the top polyline. Likewise the bottom surface is modeled by the bottom polyline and any points in the top polyline that are outside the bottom polyline. The top and bottom polylines are used as inclusion area polylines to limit volumes to within these polylines.

The program also processes two grid files that define the top and bottom of a strata. This volume is calculated between these two grids. This volume is the amount of strata cut from the pit. The inclusion area for the strata volume is the bottom pit polyline. The program assumes that the strata is entirely above the bottom of the pit and below the top of the pit.

This example shows a pit that is advancing from left to right. The left side of this pit was the edge of the previous pit.



Prompts

Select closed polyline of pit Top: *pick a closed polyline*

Select closed polyline of pit Bottom: *pick a closed polyline*

Make Grid File Dialog Choose a grid resolution.

Select Top of Strata Grid Choose .grd file that models the top of the strata.

Select Bottom of Strata Grid Choose .grd file that models the bottom of the strata.

Enter density of strata (lbs/cf): 75

Top of pit area: 31390.12 S.F.

Bottom of pit area: 31390.12 S.F.

Total cut: 1199184.98 C.F., 44414.26 C.Y.

Other cut: 810572.37 C.F., 30021.20 C.Y.

Strata cut: 388612.60 C.F., 14393.06 C.Y.

Strata density: 75 (lbs/cf)

Tons of strata: 14572.97

Strip ratio (C.Y. other/Ton strata): 2.06

Pulldown Menu Location: Surface

Keyboard Command: dragpit

Prerequisite: Two 3D polylines for top and bottom of pit and two .grd files for the top and bottom surfaces of a strata.

Regrade Backfill

This command applies to narrow valleys such as in Appalachia where the goal is to regrade an existing valley fill with benches and specific slopes. Before starting this command you must create grid files (.grd) that represent the existing surface without any fill and the surface with the existing backfill. These grid files can be created with the Make 3D Grid File routine of the Civil Design module.

Regrade Backfill also requires a pre-drawn alignment polyline which defines the shape of the face of the regrade. This polyline must be drawn below the existing backfill and it must be at least long enough to reach across the widest part of the regrade.

To run the routine, specify the two grid files, the alignment polyline, the direction up the valley, the slopes, and the bench dimensions. The program then draws in the regrade as 3D polylines and calculates the cut and fill. To balance the cut and fill, choose the *Adjust parameters and redesign* option.

Prompts

Select Existing Surface Grid File

Select Backfill Grid File

Pick existing backfill perimeter polyline: *pick a closed polyline that represents the limits of the backfill disturbed area*

Pick the regrade alignment polyline: *pick a polyline*

Pick direction up regrade: *pick a point towards the back end of the valley*

Enter the vertical lift between benches <50.0>: *press Enter*

Enter the slope ratio (x:1) <2.0>: *press Enter*

Enter the bench width <20.0>: *press Enter*

Enter the bench percent slope <2.0>: *press Enter*

Enter the top percent slope <2.0>: *press Enter* This slope is used after the regrade reaches the top of the existing backfill until the end of the regrade.

Example Regrade Backfill Report:

Regrade Backfill Report

Vertical lift between benches: 50.00

Slope percent: 50.00%, slope ratio: 2.00:1

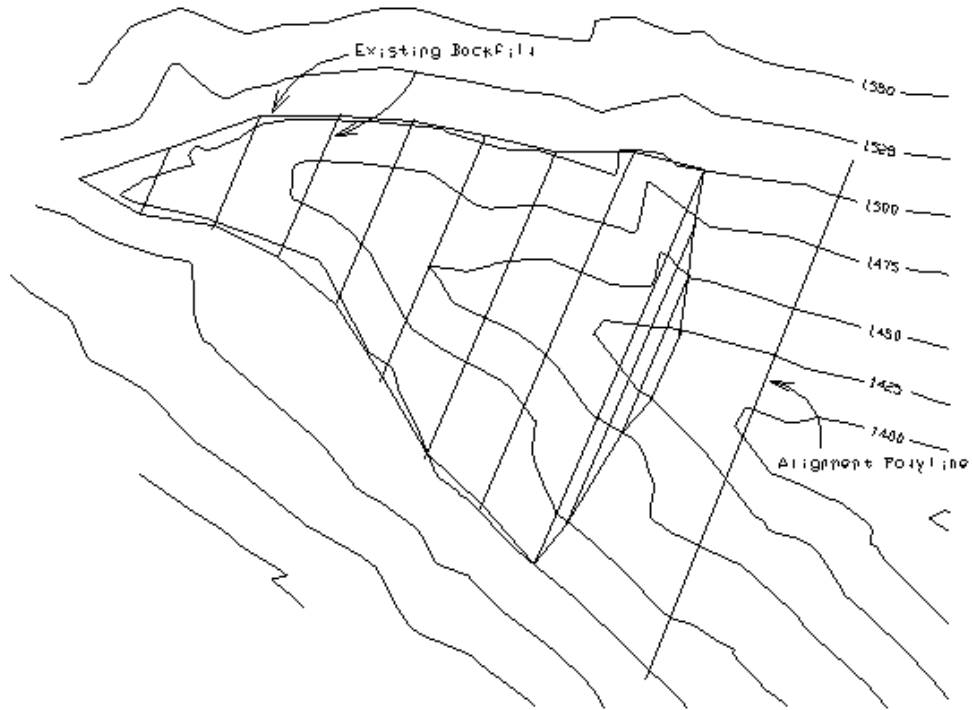
Bench width: 15.00

Bench slope percent: 2.00 slope ratio: 50.00:1

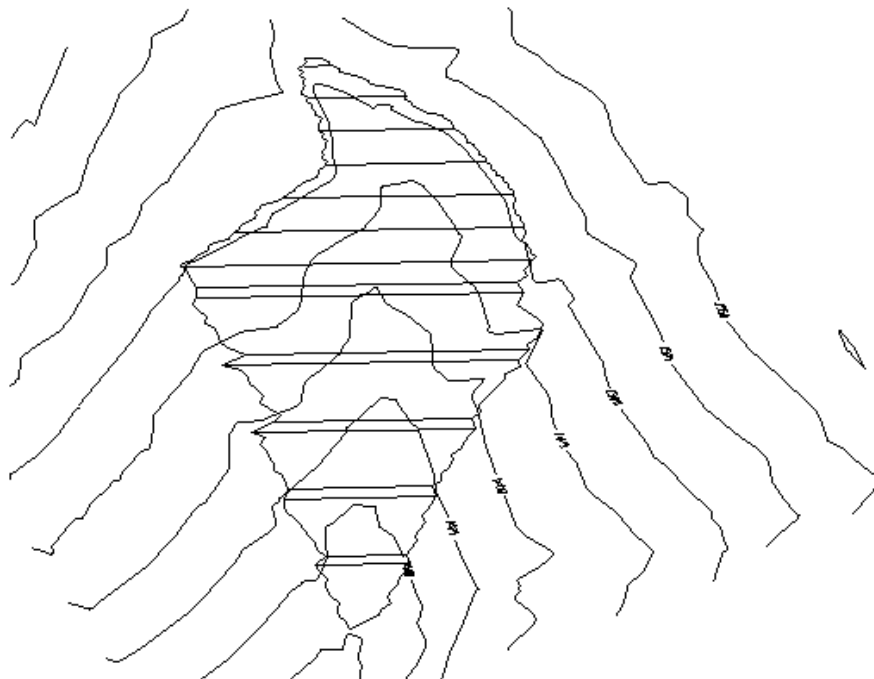
Top slope percent: 2.00 slope ratio: 50.00:1

Total fill: 13105.1 C.Y., 353840.0 C.F.

Total cut: 13288.2 C.Y., 358782.9 C.F.



Existing backfill and existing contours



3D view of regrade backfill. Also showing existing contours.

Pulldown Menu Location: Surface

Keyboard Command: backfill

Prerequisite: Grid files of the existing surface and existing backfill and an alignment polyline.

Blast Pattern Layout

This command creates a pattern of points in rows and columns using surface and shot bottom grids and dip direction. The options for this command are set in the dialog. Grid or triangulation files for the ground surface and drill bottom are required. The Number By Row/Column option will number the holes by the hole row and column. Otherwise the holes are simply numbered sequentially. Up to four descriptions can be assigned to the holes such as pattern name, driller name, etc. The row and column distances can be specified separately. The drilling angle can be set in the Vertical Dip field which is entered as an angle where 0 degrees is straight down and 90 degrees is horizontal. The Dip Direction defines the horizontal orientation of the vertical dip. The row and column directions are entered as bearing angles. You need to figure these angles before running Blast Pattern Layout by using routines such as Inverse or Bearing & 3D Distance. The format is in dd.mmss where the degrees.minutes.seconds are separated by a decimal.

This command can generate a report using the report formatter that lets you choose the fields for the report and the fields layout. The report formatter can also output to Excel and Access. The points can be also be stored to the current coordinate file (.crd). The Preview button shows a preview of the pattern in the drawing so that if something isn't right, it can be adjusted before choosing OK. If the Create Points is turned on, then the points will be drawn in the blast perimeter. Use the Create Lines option to draw 3D lines for each blast hole on the specified layer.

Blast Pattern Layout

Ground Surface C:\Carlson Projects\2008 User Conference\4

Blast Bottom C:\Carlson Projects\2008 User Conference\4

Number by Row/Column Starting Row# Starting Column#

Point Description Row/Column: Requires Alphanumeric CRD

Row Distance Interval Extra Description #1

Column Distance Interval Extra Description #2

Vertical Dip (dd.mmss) Extra Description #3

Dip Direction(dd.mmss) NE SE SW NW

Row Direction(dd.mmss) NE SE SW NW

Column Direction(dd.mmss) NE SE SW NW

Pattern Layout

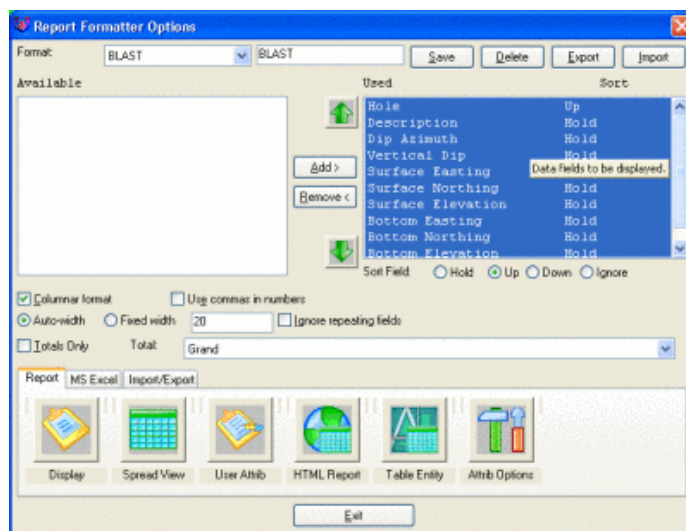
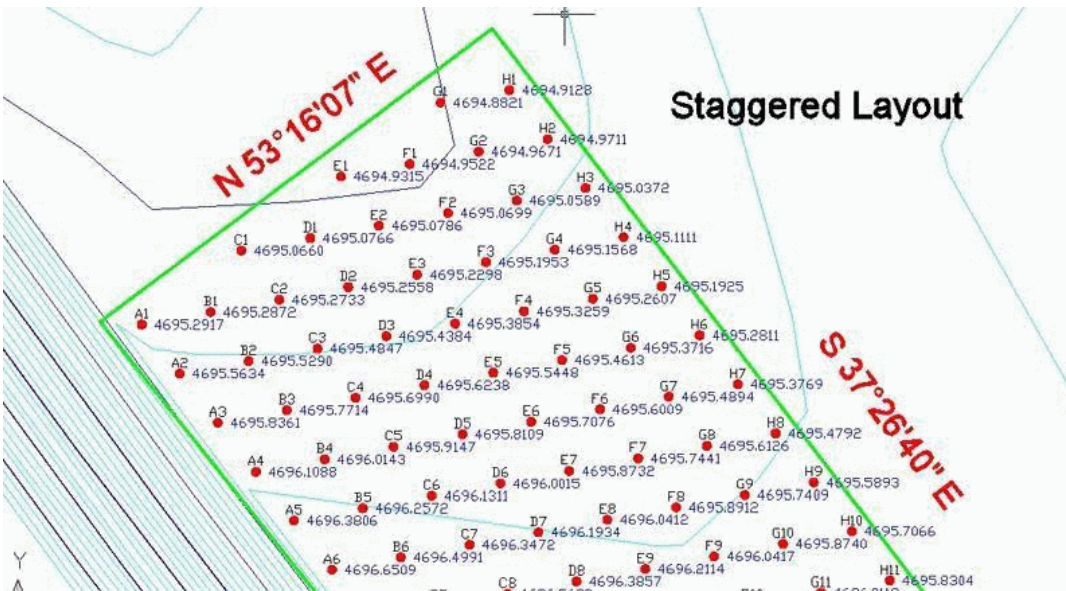
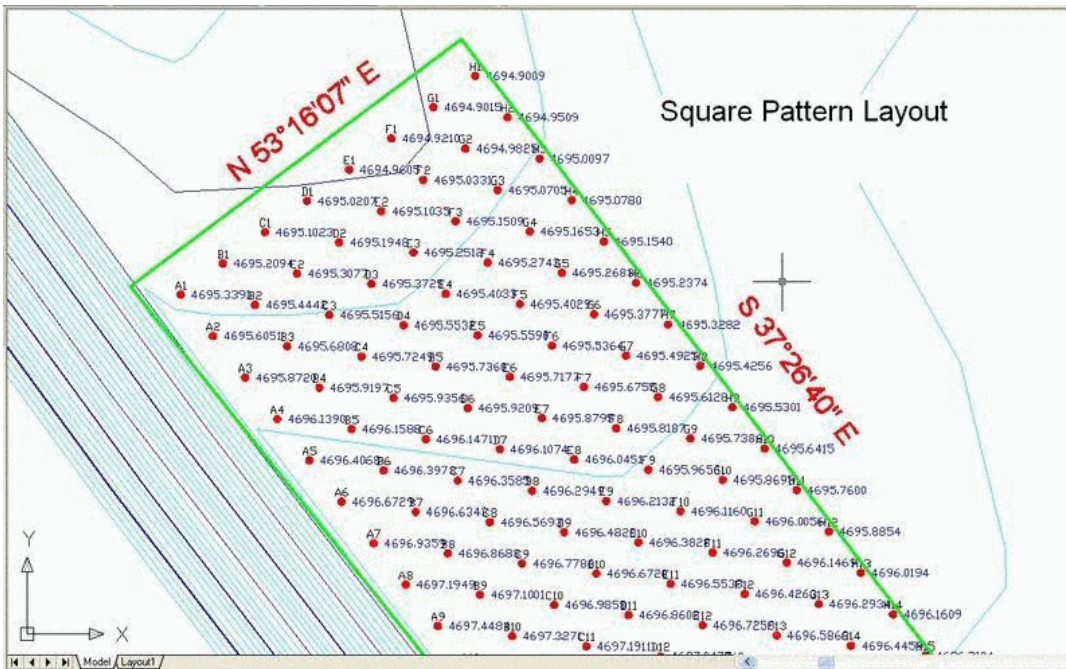
Square Staggered

Create Points Create Report

Points to Create

Blast Bottom Ground Surface

After filling out the dialog, the program prompts for the blast boundary polyline which should be a closed polyline of the blast area. Finally you specify the starting point within this boundary polyline and the program calculates the points.



Hole	Description	Dip	Azimuth	Vertical Dip	Surface Easting	Surface Northing	Surface Elevation	Bottom Easting	Bottom Northing	Bottom Elevation
A1	Shot2	0.0000	0.0000	0.0000	145537.51	1975673.00	4745.67	145537.51	1975673.00	4650.06
A2	Shot2	0.0000	0.0000	0.0000	145558.79	1975645.21	4746.86	145558.79	1975645.21	4650.43
A3	Shot2	0.0000	0.0000	0.0000	145580.07	1975617.42	4748.02	145580.07	1975617.42	4650.81
A4	Shot2	0.0000	0.0000	0.0000	145560.35	1975589.63	4749.16	145560.35	1975589.63	4651.19
A5	Shot2	0.0000	0.0000	0.0000	1455622.63	1975561.85	4750.47	1455622.63	1975561.85	4651.56
A6	Shot2	0.0000	0.0000	0.0000	1455643.91	1975534.06	4752.22	1455643.91	1975534.06	4651.93
A7	Shot2	0.0000	0.0000	0.0000	1455665.19	1975506.27	4753.77	1455665.19	1975506.27	4652.30
A8	Shot2	0.0000	0.0000	0.0000	1455686.47	1975478.48	4755.11	1455686.47	1975478.48	4652.67
A9	Shot2	0.0000	0.0000	0.0000	1455707.75	1975450.69	4756.29	1455707.75	1975450.69	4653.05
A10	Shot2	0.0000	0.0000	0.0000	1455729.03	1975422.91	4757.22	1455729.03	1975422.91	4653.42
A11	Shot2	0.0000	0.0000	0.0000	1455750.31	1975395.12	4758.14	1455750.31	1975395.12	4653.78
A12	Shot2	0.0000	0.0000	0.0000	1455771.59	1975367.33	4759.53	1455771.59	1975367.33	4654.15
A13	Shot2	0.0000	0.0000	0.0000	1455792.87	1975339.54	4761.00	1455792.87	1975339.54	4654.52
A14	Shot2	0.0000	0.0000	0.0000	1455814.15	1975311.75	4762.02	1455814.15	1975311.75	4654.89
A15	Shot2	0.0000	0.0000	0.0000	1455835.43	1975283.97	4762.85	1455835.43	1975283.97	4655.26
A16	Shot2	0.0000	0.0000	0.0000	1455856.71	1975256.18	4763.53	1455856.71	1975256.18	4655.64
A17	Shot2	0.0000	0.0000	0.0000	1455877.99	1975228.39	4763.95	1455877.99	1975228.39	4656.01
A18	Shot2	0.0000	0.0000	0.0000	1455899.27	1975200.60	4764.21	1455899.27	1975200.60	4656.39
A19	Shot2	0.0000	0.0000	0.0000	1455920.54	1975172.81	4764.44	1455920.54	1975172.81	4656.76
A20	Shot2	0.0000	0.0000	0.0000	1455941.82	1975145.03	4764.64	1455941.82	1975145.03	4657.14
A21	Shot2	0.0000	0.0000	0.0000	1455963.10	1975117.24	4764.86	1455963.10	1975117.24	4657.53

Prompts

Blast Pattern dialog

Select blast boundary polyline: *pick a polyline*

Pick starting point: *pick a point*

Pulldown Menu Location: Surface

Keyboard Command: blast2

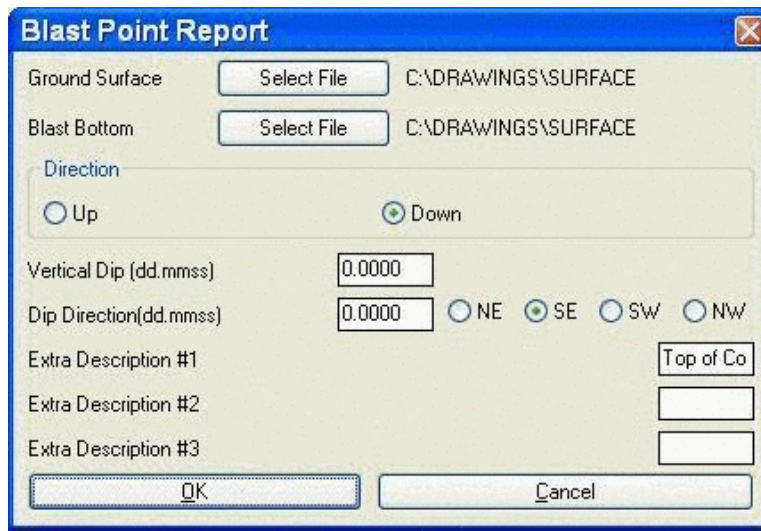
Prerequisite: Surface and pit grids

Interval Along Entity

This command is described in the Survey section of the manual. Please refer to it there.

Blast Point Report

This command projects points either from the blast bottom up to the surface at a dip direction or from the surface down to the blast bottom. These points are then reported using the report formatter that lets you customize the fields to report and the layout. Before running this command, you need the ground surface grid file, the blast bottom grid file and points drawn on the screen. While the Blast Pattern Layout routine also can generate a report, this routine is more flexible because it uses all selected points from the screen and is not limited to only pattern points. For example, you could use Blast Pattern Layout to create a grid of points. Then use Interval Along Entity to create points along a polyline that represents the setback line for the first row of holes. You could also use any of the other Carlson point routines to add or remove points. Then once all the points are placed, you can run Blast Point Report.



Hole Description	Extra #1	Dip Azimuth	Vertical Dip	Bottom Easting	Bottom Northing	Bottom Elevation	Surface Easting	Surface Northing	Surface Elevation
A1	Top of Coal	180.0000	360.0000	1458499.84	1972761.44	4762.39	1458499.84	1972761.44	4851.98
A1	Top of Coal	180.0000	360.0000	1458499.84	1972761.44	4762.39	1458499.84	1972761.44	4851.98
A1	Top of Coal	180.0000	360.0000	1458499.84	1972761.44	4762.39	1458499.84	1972761.44	4851.98
A1	Top of Coal	180.0000	360.0000	1458499.84	1972761.44	4762.39	1458499.84	1972761.44	4851.98
A2	Top of Coal	180.0000	360.0000	1458521.30	1972733.80	4763.48	1458521.30	1972733.80	4853.39
A2	Top of Coal	180.0000	360.0000	1458521.30	1972733.80	4763.48	1458521.30	1972733.80	4853.39
A2	Top of Coal	180.0000	360.0000	1458521.30	1972733.80	4763.48	1458521.30	1972733.80	4853.39
A2	Top of Coal	180.0000	360.0000	1458521.30	1972733.80	4763.48	1458521.30	1972733.80	4853.39
A3	Top of Coal	180.0000	360.0000	1458542.77	1972706.16	4764.56	1458542.77	1972706.16	4852.49
A3	Top of Coal	180.0000	360.0000	1458542.77	1972706.16	4764.56	1458542.77	1972706.16	4852.49
A3	Top of Coal	180.0000	360.0000	1458542.77	1972706.16	4764.56	1458542.77	1972706.16	4852.49
A3	Top of Coal	180.0000	360.0000	1458542.77	1972706.16	4764.56	1458542.77	1972706.16	4852.49
A4	Top of Coal	180.0000	360.0000	1458564.24	1972678.52	4765.57	1458564.24	1972678.52	4858.66
A4	Top of Coal	180.0000	360.0000	1458564.24	1972678.52	4765.57	1458564.24	1972678.52	4858.66
A4	Top of Coal	180.0000	360.0000	1458564.24	1972678.52	4765.57	1458564.24	1972678.52	4858.66
A4	Top of Coal	180.0000	360.0000	1458564.24	1972678.52	4765.57	1458564.24	1972678.52	4858.66
A5	Top of Coal	180.0000	360.0000	1458585.71	1972650.87	4766.49	1458585.71	1972650.87	4860.42
A5	Top of Coal	180.0000	360.0000	1458585.71	1972650.87	4766.49	1458585.71	1972650.87	4860.42
A5	Top of Coal	180.0000	360.0000	1458585.71	1972650.87	4766.49	1458585.71	1972650.87	4860.42
A5	Top of Coal	180.0000	360.0000	1458585.71	1972650.87	4766.49	1458585.71	1972650.87	4860.42
B1	Top of Coal	180.0000	360.0000	1458538.67	1972768.48	4761.28	1458538.67	1972768.48	4853.57
B1	Top of Coal	180.0000	360.0000	1458538.67	1972768.48	4761.28	1458538.67	1972768.48	4853.57
B1	Top of Coal	180.0000	360.0000	1458538.67	1972768.48	4761.28	1458538.67	1972768.48	4853.57
B1	Top of Coal	180.0000	360.0000	1458538.67	1972768.48	4761.28	1458538.67	1972768.48	4853.57

Prompts

Blast Point Report dialog

Select blast bottom points.

Select objects: *pick the points*

Report Formatter dialog

Pulldown Menu Location: Surface

Keyboard Command: blast3

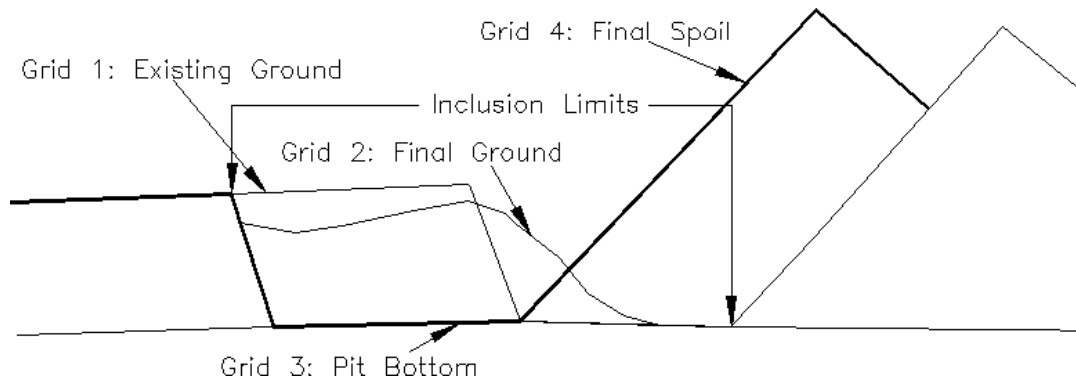
Prerequisite: Ground and bottom grids; points in drawing

Cast Blast Report

This command uses before and after grid files to report blast volumes, swell and cast to final. Before running this command, you need four grid files and an inclusion perimeter. The inclusion perimeter is a closed polyline that follows the back of the cut area and the toe of the previous spoil pile. The first grid file is the original ground surface before the blast. The second grid file is the ground surface after the blast. The third grid file represents the bottom of the pit (design, or bottom of ore). For example, if you are blasting down to the top of ore, this grid should be the top of ore surface. The fourth grid defines the final pit surface including the final cut and spoil pile. This final grid can be created with the Make 3D Grid File command using 3D polylines for the toe of the highwall, toe of the spoil pile and top of the spoil pile.

The program calculates the volume between the original ground grid and the pit bottom grid within the inclusion

perimeter. This volume is reported as the Original Volume. The After Blast Volume is the volume between the final ground grid and the pit bottom grid. The difference between the After Blast Volume and Original Volume is reported as the Swell Factor. The After Blast volume that is below the final pit surface is the Cast to Final volume.



Cast Blast Report:

```
Original Surface: C:\scad\DATA\ex.gnd.grd
Final Surface   : C:\scadv\DATA\final.grd
Pit Bottom      : C:\scad\DATA\coal.grd
Final Spoil     : C:\scad\DATA\spoil.grd
Lower left grid corner : 509141.84,595450.20
Upper right grid corner: 511087.39,596727.09
X grid resolution: 100, Y grid resolution: 100
X grid cell size: 19.46, Y grid cell size: 12.77
```

```
Original Volume: 1602319.50 C.F., 59345.17 C.Y.
After Blast Volume: 2066992.20 C.F., 76555.27 C.Y.
Blast Swell Factor: 1.29
Cast to Final Volume: 144689.22 C.F., 5358.86 C.Y.
Cast to Final: 9.03%
```

Prompts

Original Surface Grid File

Final Surface Grid File

Pit Bottom Grid File

Final Spoil Grid File

Select the Inclusion perimeter polylines.

Select objects: *pick a closed polyline*

Pulldown Menu Location: Surface, Blast Pattern

Keyboard Command: blastinfo

Prerequisite: Original, final, pit and spoil grid files. Inclusion perimeter polyline.

Output To Reame

Output to Reame creates a file with a .dat extension that is readable by the REAME2004 stability analysis program for cylindrical failure. The REAME program was developed by Dr. Huang of the University of Kentucky and is widely used in coal mine permitting in the state of Kentucky. The linkage has been tested and verified with the REAME2004.

The program is typically used after a final surface profile is laid graphically over an existing ground profile. You then graphically select the profiles to create the REAME input file. The routine not only creates the input file but provides options to draw the failure study region and key coordinates on the actual profile drawing. The only significant condition to be aware of is that the profiles should rise upwards from left to right in the direction of increasing station.

To run the Reame program after Output to Reame, it is best to copy the .dat file just made into the directory containing REAME.EXE. Then run REAMEINP to revise the .dat file (to change aspect like cohesion, friction angle and unit weight) or REAME to run the stability analysis. It will ask: FILE NAME -? Enter the name with the .dat extension included. The enter 1 to input from file. The analysis is then completed and stored in a file REAME.TXT.

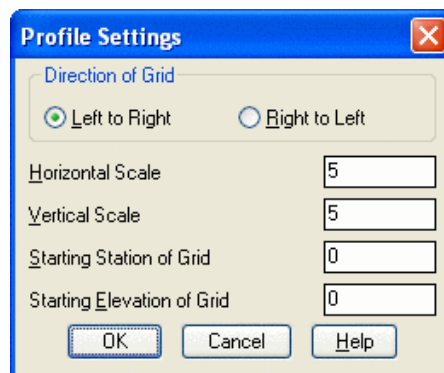
NOTE: The REAME program has been renamed to LEAME (Limit Equilibrium Analysis of Multilayered Earthworks). This software is distributed as an instructional tool by ASCE Press (<https://ascelibrary.org/doi/book/10.1061/9780784477991>). A companion slope stability analysis book is also available (<https://ascelibrary.org/doi/book/10.1061/9780784412886>).

Prompts

Title of Stability Report: *Valley Fill No. 1 (Reame expects this title line, so enter something!)*

Select Files or Pick from screen (F or <P>): *P, press Enter for the Pick option here. The select file option would require that you enter the name of an existing and final .PRO file.*

Next a Profile Settings menu comes up, in which it is important verify scales and the correct starting station and elevation.



[end on] Pick Lower Left Grid Corner <0.00,1445.00>: *pick where the starting station and starting elevation grid lines intersect Switch to int snap if that point is not an end point.*

Select Final Grade Polyline: *pick final grade*

Select Lowest Surface Polyline: *pick existing grade Phantom lines at right angles representing the proposed study area for stability analysis now appear.*

Pick another Surface (y/<n>): *n, Yes for adding more surface polylines to reame input*

Minimum Depth (DMIN) for Failure Slice:

Lowest Surface:

Value for C <0>:

Value for Phid <40>:

Value for G <125>:

Note: Lines will not Appear at Right Angles if Vert. Scale is Exaggerated.

Translated Coords: (-10.0 68.9902) (515.0 173.99)

Is Stability Analysis Zone Acceptable (<y>/n):

Plot Stability Analysis Zone Lines (<y>/n):

Label the Zone Line Endpoints (<y>/n):

[end on] Pick Points for Coord. Label (enter to continue): *pick any ground line or final grade line vertices or endpoints for coordinate labeling Each time you will also be asked:*

[none] Pick Starting Point for Text: *pick a point*

[end on] Pick Points for Coord. Label (enter to continue):

[none] Pick Starting Point for Text:

[end on] Pick Points for Coord. Label (enter to continue):

[none] Pick Starting Point for Text:

[end on] Pick Points for Coord. Label (enter to continue):

Number of divisions along face of slope <4>:

Number of divisions perpendicular to slope <8>:

This would lead to 32 radius points from which to test cylindrical failure.

Pulldown Menu Location: Surface > Slope Stability

Keyboard Command: reame

Prerequisite: Surface profiles or polylines

Draw Reame File

The function allows the user to import their existing Reame file and draw it in the drawing. It prompts to select the RME file created in Reame.

Pulldown Menu Location: Surface > Slope Stability

Keyboard Command: reameplt

Prerequisite: Stability Reame file

Output to SB-Slope

The widely accepted and computationally powerful stability analysis package, SB-Slope has very limited functions for data and geometry modification. The procedure outputs the data files in the SB-Slope data format, thus providing powerful interface for geometry editing. The meaning of all data parameters remains the same as in conventional SB-Slope.

The user is prompted with a single dialog defining the majority of necessary parameters, followed by prompts for soil and water polylines, soil properties and finally the parameters of applied loads if any. For each soil type, the program prompts first to pick polylines which denote upper boundary of the soil (multiple polylines may be selected), and then to pick polylines which represent boundary of the soil and are phreatic lines.

Polylines from the second set may appear in the first set as well.

After all soil polylines, the free-water surface is entered if any and then user is prompted for the soil mechanical properties. At last, distributed loads are entered by picking a horizontal line, representing the load support area, and input of the load value.

Pulldown Menu Location: Surface > Slope Stability

Keyboard Command: sbslope

Draw SB-Slope File

The function allows user to import his existing SB-Slope file into the drawing. Segments of the neighbor lines corresponding to the same soil are joined to simplify creation of new SB-Slope data file from the drawing. The following color scheme is used to mark elements of the layout:

blue = free water surface

green = phreatic surface

red = places of load application

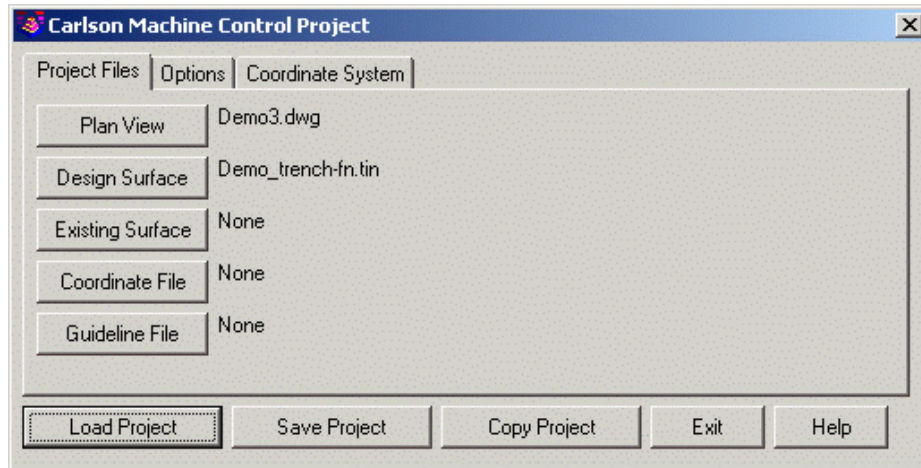
Pulldown Menu Location: Surface > Slope Stability

Keyboard Command: rsbslope

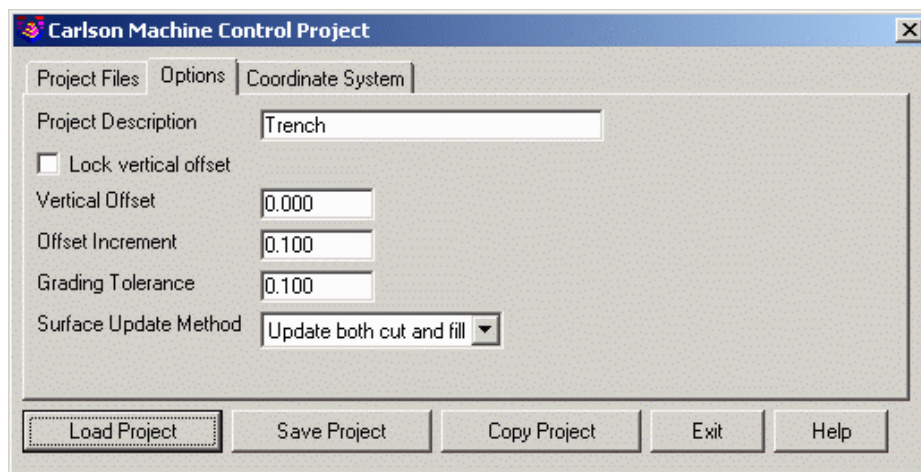
Carlson Grade Project

This command creates a project file (.PRJ) to use for Carlson Machine Control programs including Carlson Grade. The project file is a text file that contains the names of the data files for the project (ie. the Design Surface) and the project options (ie. Units). The Load Project and Save Project functions read and write the .PRJ file. The Copy Project function both saves the .PRJ file and copies the data files into the DESIGN and ACTUAL sub-folders. So after setting up the project in the dialog, the Copy Project button is a handy way to send the data to Carlson Grade by copying the .PRJ and related data files onto a USB drive to take to the field or to a folder for Fleet Manager Office to use.

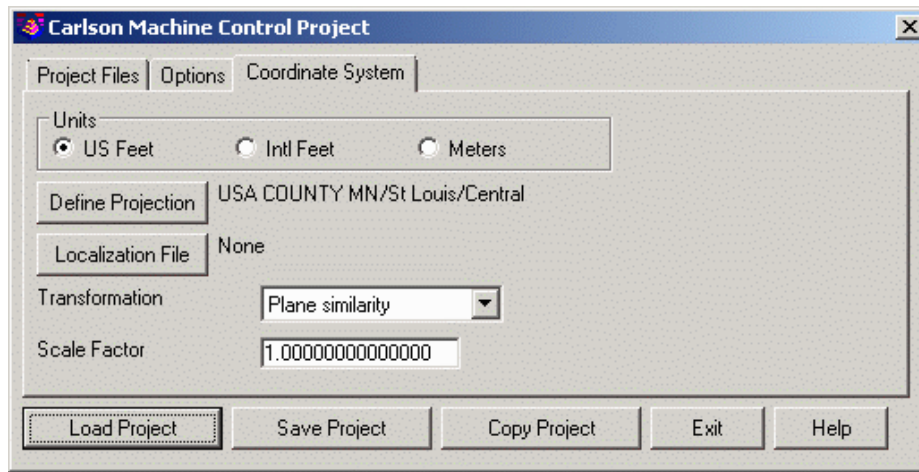
On the Project Files tab, the Plan View for the map view can be a DWG, DXF or PLN file. The PLN file is created by the File > Polyline File > Write Polyline File command. The Design Surface is the target grading surface and can be a triangulation or grid file (TIN or GRD). The other files are optional and are explained in the Carlson Grade manual.



The settings on the Options tab are specific to Carlson Grade and described in the Carlson Grade manual.



On the Coordinate System tab, define the units and projection for the project. In Define Projection, you can use a CSL file from SurvCE or the projection from the Settings > Drawing Setup command. The Localization File takes either .DAT or .LOC file formats. The Drawing Setup command can create a localization file. Or you can use a localization created by SurvCE or Carlson Grade.



Pulldown Menu Location: Surface > Carlson Machine Control in Surface Mining and Construction modules

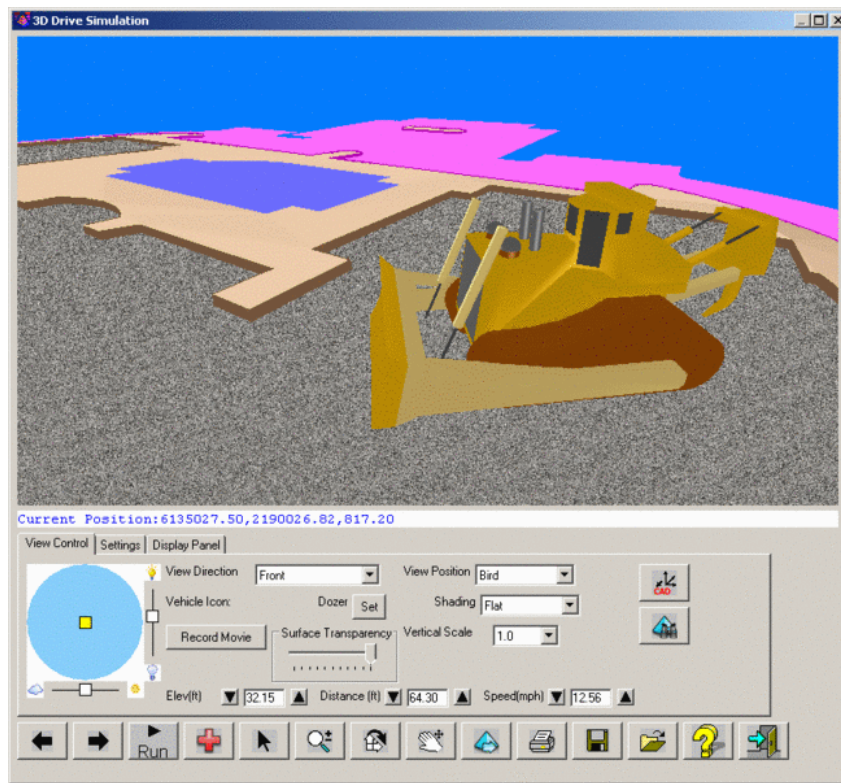
Keyboard Command: grade_project

Prerequisite: Surface file

3D Grade Simulation

This command selects a project file (.PRJ) and loads the associated design surface into the 3D simulation. This simulation is a way to verify the surface. You can choose a Vehicle Icon, set the Speed and pick Run to drive over the surface. See the Surface 3D Flyover topic in the manual for descriptions on the controls in this viewer.

The program looks for the design surface in the DESIGN sub-folder from the PRJ file location. The Copy Project function in the Carlson Grade Project command is a way to prepare the PRJ file with the surface in the DESIGN folder.



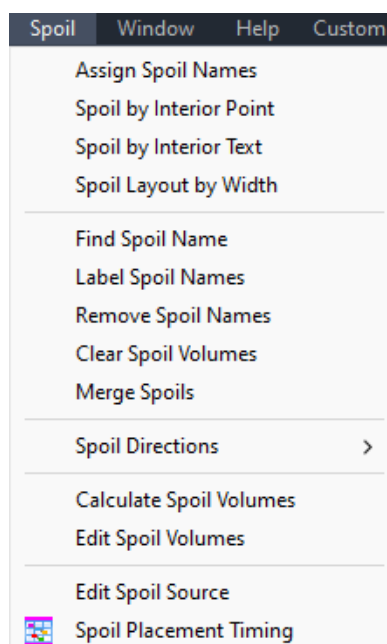
Pulldown Menu Location: Surface > Carlson Machine Control in Surface Mining and Construction modules

Keyboard Command: grade.flyby

Prerequisite: PRJ file

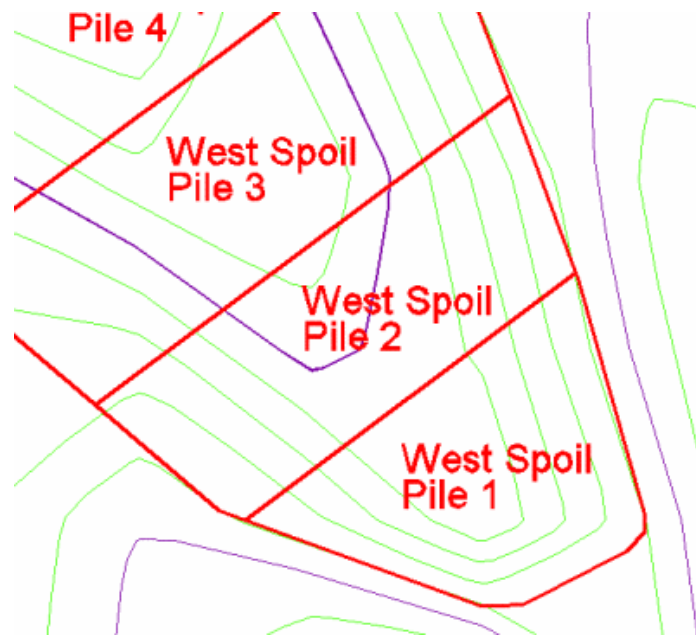
Spoil Menu

The Spoil menu commands are for spoil placement.



Assign Spoil Names

This command attaches a spoil name and site name to closed polylines that are used as inclusion perimeters in the spoil routines like Calculate Spoil Volume and Spoil Placement Timing. This will create them one at a time, and is recommended if there are just a few to name. Other commands will create more advanced layouts of spoil perimeters for designs that require more perimeters.



Prompts

Label area names [**<Yes>/No**]? Choose whether or not to label.

Text height **<2.78>**: *press Enter to accept 2.78 drawing units (ft or m tall) or enter a new text height*

Auto place labels in center [**<Yes>/No**]? *press Enter to accept to manually pick the label position, answer No to this prompt.*

Select Inclusion perimeter polylines.

Select objects: *pick closed polylines for areas to include in calculations*

Site name **<Site 1>**: *press Enter to accept Site 1, or type a new site name*

Pit name **<Pile 1>**: *press Enter to accept Pile 1, or type a new pit name*

Specify another area [**<Yes>/No**]? *press Enter to accept Yes or type N for no* If you do specify another area, then the Site Name remains the same, but the Pile Name is automatically incremented by one for efficient naming.

Keyboard Command: spoilname

Prerequisite: Closed polyline

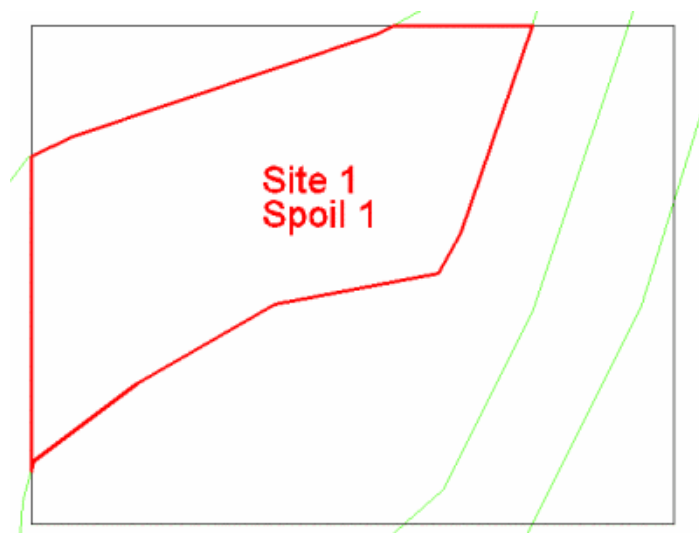
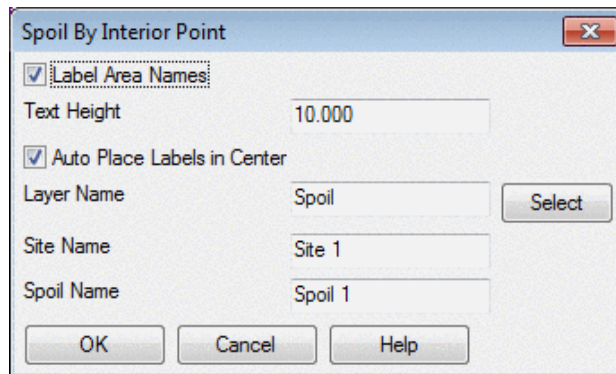
Pulldown Menu Location: Spoil in Surface Mine Module

Spoil by Interior Point

This routine is used to create spoil polylines from the existing linework by making closed polylines around points picked. The site and spoil names are user-specified and the spoil name is incremented after each new pile perimeter

is created. This command uses the Boundary Polyline logic, using any linework (open or closed) and searches out from the picked point for the closed perimeter. It draws a new polyline in the specified layer.

- **Label Area Names:** Choose whether to label the pile perimeters or not
- **Text Height:** Set the text height in feet or meters based on the drawing setup
- **AutoPlace Labels in Center:** Yes to put the text insertion point on the centroid of the perimeter
- **Layer Name:** Select or enter the layer name to draw the perimeter and text
- **Site Name:** Specify a site name to give to all the new pile perimeters
- **Spoil Name:** Specify a name to give the first pile perimeter. Subsequent piles will increment the name by one automatically.



Prompts

Pick point inside pile perimeter: *pick inside a loop or linework representing the pile for the spoil polyline*

Specify another area (<Yes>/No)? If yes is selected, then the Pile Name is automatically incremented by one for efficient entry, such as Spoil 2. **Site name <Site 1>:**

Pit name <Spoil 2>:

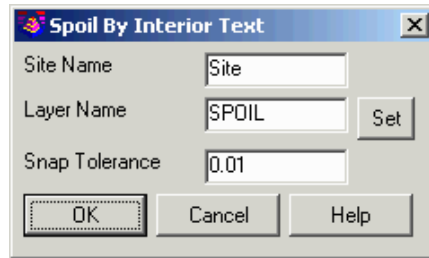
Pulldown Menu Location: Spoil in Surface Mine Module

Keyboard Command: pickspoil

Spoil By Interior Text

This command creates closed polylines assigned spoil names using selected interior text and surrounding linework. The text is used as the spoil name. For each text, the program finds the bounding linework to create the closed polyline for the spoil area.

In the options dialog, set the Site Name to assign the spoil polylines and set the Layer Name for the polylines. The Snap Tolerance is the amount of gap in the linework to close automatically.



Prompts

Spoil dialog

Select spoil lines, polylines and text.

Select objects: *pick linework and labels*

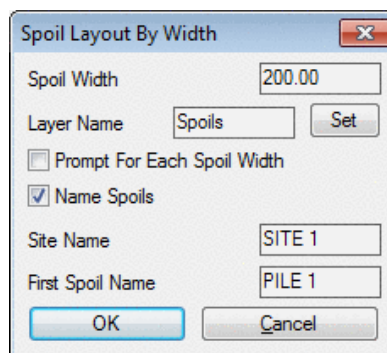
Pulldown Menu Location: Spoil

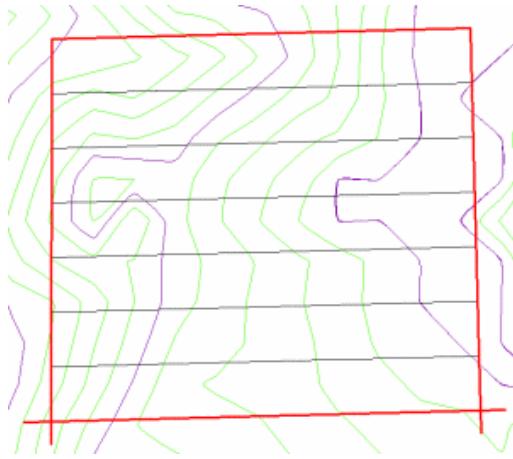
Keyboard Command: txtspoil

Prerequisite: linework and spoil labels

Spoil Layout by Width

This command creates closed polylines that represent spoil placement inclusion areas for the spoil timing routines. The spoil polylines are created by advancing a baseline polyline by a fixed pile width. The sides and end of the pit polylines are defined by the disturbance boundary polyline. This polyline should be open at the end of the baseline polyline. Be sure to create the baseline polyline longer than the overall boundary so that the program can always find the intersection of the advancing baseline with the boundary. There is an option to automatically assign pile and site names to the polylines. The name of the first pile is set in the dialog shown below. The next spoils will be named by incrementing the spoil name by one. There is an option to prompt for each spoil perimeter width or use the same width for all the piles.





Prompts

Spoil Layout By Width dialog Set the spoil width, layer and names

Select baseline polyline: *pick the polyline*

Select spoil boundary polyline: *pick the polyline*

Calculating spoils ...

Created 7 spoils.

Pulldown Menu Location: Spoil

Keyboard Command: spoilsz

Prerequisite: One open polyline boundary and one baseline polyline extending across the boundaries open end.

Find Spoil Name

This command will find a certain named spoil polyline on screen. It will zoom to the pile extents and center it. The spoil line is also dashed or highlighted to distinguish it from surrounding polylines.

Prompts

Spoil name to find: **pile 4**

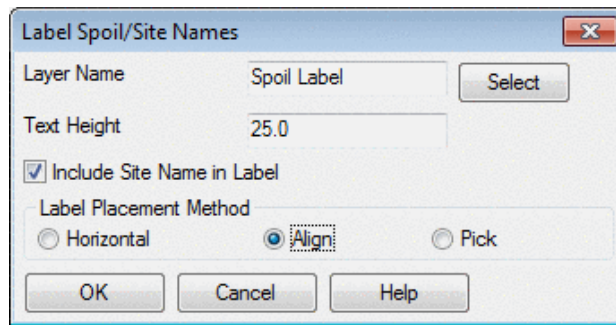
Site name to find <*>:

Pulldown Menu Location: Boundary

Keyboard Command: find_spoil

Label Spoil Names

This command labels the spoil name and site name that are attached to the selected polyline. Options for layer and text height are provided. The Horizontal and Align placement methods will draw the labels in the center of the polyline. The difference is that the horizontal option will draw the label horizontal to the current twist and the Align method will rotate the label to follow the main direction of the spoil line. The Pick method prompts you for a center point to pick and an alignment for the text.



Prompt

Label Spoil/Site Names Dialog

Pulldown Menu Location: Spoil

Keyboard Command: spoillabel

Remove Spoil Names

This command removes the extended entity data of spoil/site names from the selected polylines. For example if you are done calculating for an area, or you just want to clear and rename them, you can use this command to clear the spoil names from the polylines in the old area so that these polylines are not picked up as spoil perimeters anymore by routines for timing the spoil.

Prompts

Select polylines to remove spoil names from.

Select objects: *pick the polylines*

Removed pit names from XX polylines.

Pulldown Menu Location: Spoil
Keyboard Command: clearspoilnm

Clear Spoil Volumes

This command removes all quantity values and grid paths from the selected spoil polylines. There is the option to clear all benches at once, or if Specific is selected, then it will prompt to enter a bench number to clear.

Prompts

Clear all bench quantities or a specific bench [All/<Specific>]? A

Select spoil polylines to clear.

Select objects: Specify opposite corner: 5 found, 5 total

Cleared 10 bench quantities.

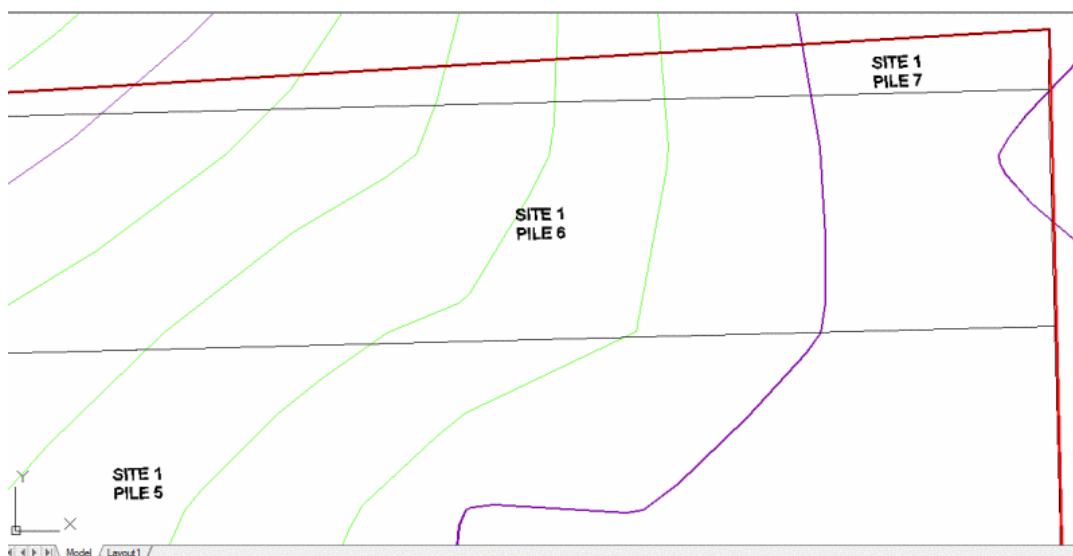
Pulldown Menu Location: Spoil

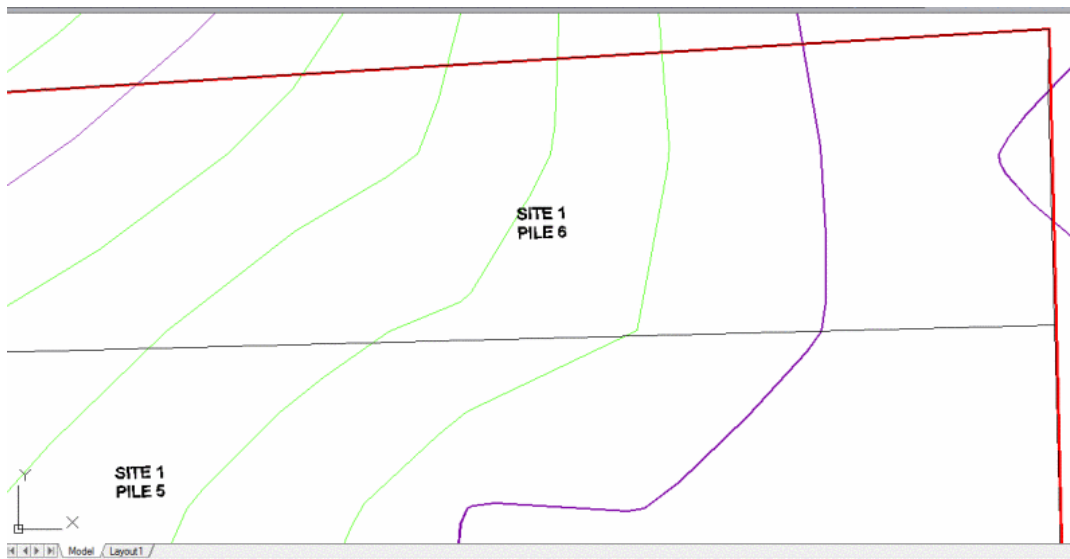
Keyboard Command: clearspoilvol

Merge Spoils

This routine takes two adjacent spoil perimeters and merges them to create just one. Routines such as Spoil Layout by Width can sometimes leave small irregular shaped pits along complex boundaries. It is common to combine or add a small sliver of a spoil perimeter with an adjacent one so that the volume is added to that one. Pick first in the larger spoil that is to be kept, then in the smaller spoil that is to be removed. A new polyline is drawn around both, representing the new spoil with the same name as the first, larger spoil picked inside of. The last step is to simply erase the text of the smaller, deleted spoil (if they were labeled). If a spoil is selected on or near a common line, then the spoil is highlighted it prompts to hit Enter to accept the spoil or press N to highlight the other nearby spoil. The volumes are combined if they are already stored in the spoils as values.

The images below show the before and after of the Merge Spoils command.





Prompts

Pick inside 1st spoil polyline to merge: pick inside perimeter to keep

Pick inside 2nd spoil polyline to merge: pick inside perimeter to remove, and merge with previous

Created a shrink-wrap polyline successfully.

Done.

If the pick is near a common spoil line, then the following prompts appear:

Pick inside 1st spoil polyline to merge (Enter to end):

Press N for next selection or Enter to accept current:

Pick inside 2nd spoil polyline to merge:

Press N for next selection or Enter to accept current:

Created a shrink-wrap polyline successfully.

Done.

Pulldown Menu Location: Spoil

Keyboard Command: mergespoil

Prerequisite: Two adjacent Carlson named spoil perimeters to be combined into one larger spoil perimeter.

Assign Directions

In order to schedule the spoil polylines (which are closed polylines), they must have directions assigned. The Assign Directions command places the direction of spoil placement into the spoil polyline itself, where it is permanently stored along with other aspects of the drawing. Directions can vary by bench. The command prompts to assign the same direction to the Whole spoil polyline, or by bench. If doing by bench, the routine must be run separately for each bench.

Assign Directions is found within the Spoil menu in the Carlson Surface Mining. There are six methods employed to assign direction: Automatic, Text, Sequence, Polyline, Bearing and Azimuth. The "automatic" method will move "longways" across the pile, following the longest axis detected, but may not choose the preferred direction along that axis. The text method finds the side closest to the insertion point of the pile name and will mine from that side perpendicular across the pile away from the text. The sequence method will mine left to right and/or right to left across a series of piles as specified by the user, the polyline method will follow a "direction polyline" across a pile or series of piles and the Bearing or Azimuth method will mine at defined bearing or azimuth angle. Below is the prompting and results obtained with each method:

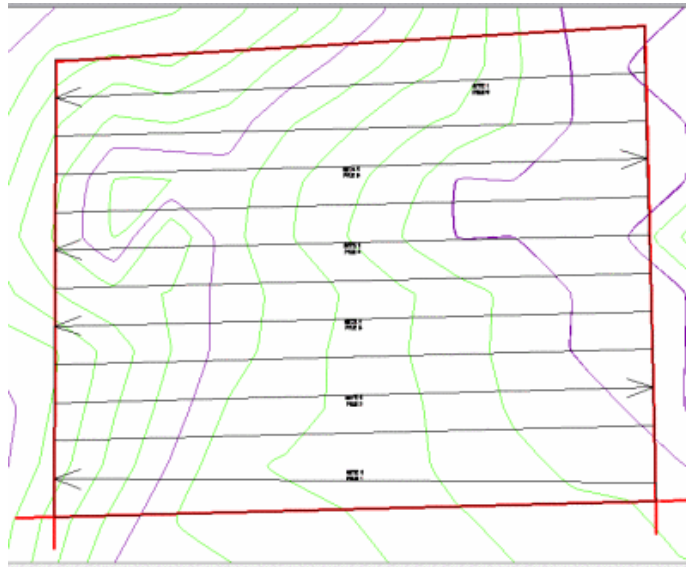
Prompts

Assign direction using which method [`<Auto>/Text/Sequence/Polyline/Bearing/AZimuth`]: *A or press Enter*

Select pile polylines to have direction assigned to:

Select objects: *Pick the piles and the direction is assigned as shown below*

In this example, it uses the long axis of the pile, and alternates back and forth for direction. It is a quick way to get directions assigned. A pile direction can be easily reversed using the *Reverse Directions* command.



Next are the prompts for the Text option.

Assign direction using which method [`<Auto>/Text/Sequence/Polyline/Bearing/AZimuth`]: *T*

Select pile polylines to have direction assigned to:

Select objects: *pick the piles and the required pile text as shown below*

The highlighted pile has the direction information assigned already.

Would you like to overwrite this (`<No>`, Yes, None, All): *A for All.* This prompt only occurs when piles are chosen that already have pile direction. Note that the command "Clear Directions" could be used to remove directions prior to using Assign Directions.

The arrows show the resulting directions. Note that this routine looks for the pile name for the direction, not for the "site" name. (All piles have a two-tiered naming convention: site and pile, which can be re-worked as pile and block or any other two-level form, to adapt to company practices.)

Next is the Sequence option.

Assign direction using which method [`<Auto>/Text/Sequence/Polyline/Bearing/AZimuth`]: *S*

Select pile polylines to have direction assigned to:

Select objects: *select the piles (Don't worry if you also select other polylines. It only finds piles.)*

Select a direction polyline.

Select objects: *select a polyline that crosses all of the piles*

Assign direction in which sequence (`<LL>`, LR, RL, RR): *LR*

In our example, we selected LR, which causes the piles to be filled left to right on the first pile (with respect to the south-to-north direction polyline—imagine yourself standing at the beginning of the polyline looking down it. LL would fill all piles coming from your left. LR would fill from the Left first, then the Right in the next one and so forth. RR and RL are just the opposite.), then right to left on the second pile, then left to right on the third, etc. An entry of LL would cause the piles to be filled from the left side to the right side. An entry of RR, for example, would fill all the piles from right to left. The entry of RL would fill first right to left, then left to right. The sequence method is ideal for assigning direction to a series of long and narrow piles that have not been broken up into small blocks.



If these same piles were each subdivided into 10 or more blocks, or there are many piles that would not be intersected by the sequence polyline, for example, then the following method, direction by polyline, is most appropriate.

Next is the Polyline option.

Assign direction using which method [\langle Auto \rangle /Text/Sequence/Polyline/Bearing/AZimuth]: P

Select pile polylines to have direction assigned to:

Select objects: *select the piles*

Select all direction polylines.

Select objects: *In the example above, select the single direction polyline.*

The polyline-based selection is ideal for piles that follow sinuous property lines. Direction polylines cannot have arcs, so if you've used an arc to draw the polyline, use the command *Remove Polyline Arcs* on the Edit menu to remove them.

Lastly, are the Bearing and Azimuth option

Bearing Option:

Assign direction to whole pile or to a specific bench [\langle Whole \rangle /Bench]: W or Enter to assign direction to whole pile

Assign direction using which method [\langle Auto \rangle /Text/Sequence/Polyline/Bearing/AZimuth]: B for Bearing

Select pile polylines to have direction assigned to:

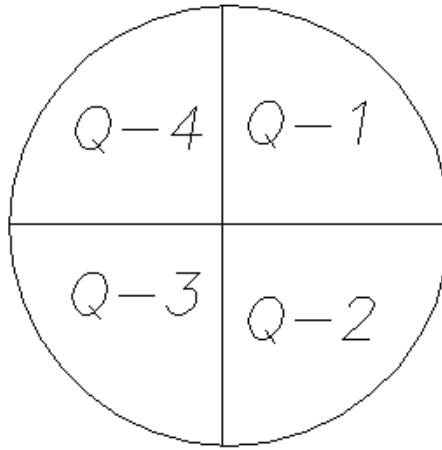
Select objects: *select the piles*

Enter Bearing (Qdd.mmss): 145.0000

The highlighted pile has the direction information assigned already.

Would you like to overwrite this [\langle No \rangle /Yes/None/All]: A for all

The Bearing is entered in the format (Qdd.mmss), the following figure shows the quarter numbers and the angle is calculated clockwise.



Azimuth Option:

Assign direction to whole pile or to a specific bench [\langle Whole \rangle / \langle Bench \rangle]: W or Enter to assign direction to whole pile

Assign direction using which method [\langle Auto \rangle / \langle Text \rangle / \langle Sequence \rangle / \langle Polyline \rangle / \langle Bearing \rangle / \langle AZimuth \rangle]: B for Bearing

Select pile polylines to have direction assigned to:

Select objects: *select the piles*

Enter Azimuth (ddd.mmss): 135.0000

The highlighted pile has the direction information assigned already.

Would you like to overwrite this [\langle No \rangle / \langle Yes \rangle / \langle None \rangle / \langle All \rangle]: A

The Azimuth angle is calculated from the true north as 0, rotating clockwise.

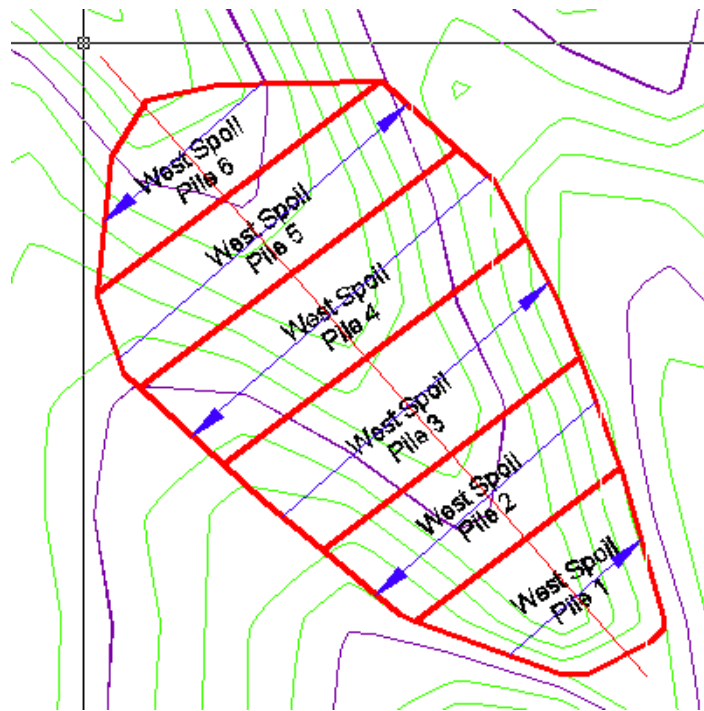
Pulldown Menu Location: Spoil

Keyboard Command: spoil_assign_dir

Related Commands: Display Directions, Reverse Directions, Clear Directions, Create pile Plines from Mineplan

Display Directions

If directions have been assigned to spoil polylines, this command will display the directions. It therefore serves two purposes: (1) to verify that directions have, in fact, been assigned previously, and (2) to review the direction of filling spoils. When pile directions are detected, arrows are displayed as shown below. These direction arrows will disappear with any "Zoom" command such as Pan or Window, and will also disappear if a Redraw or Regen is executed. The direction leaders can be used to draw direction arrow entities in the drawing, with the leader arrow size a prompt option.



Prompts

Display direction to whole pile or to a specific bench [<Whole>/Bench]: *W*

Draw directions as leaders or temporary arrows [Leaders/<Arrows>]?: *L for Leaders or A for Arrows*

Select pile polylines to have direction displayed:

Select objects: *Pick the polylines to display*

Pulldown Menu Location: Spoil

Keyboard Command: `spoil_display_dir`

Related Commands: Assign Directions, Reverse Directions, Clear Directions

Reverse Directions

This command reverses the direction of advance within a spoil polyline. It is particularly useful in conjunction with the command Assign Directions, <Auto>, since the automatic mode may assign direction to many of the piles opposite from the desired direction. The result redraws the arrows in the new direction. If Leader arrows were drawn, then they are erased and redrawn in the updated direction.

Prompts

Reverse direction to whole pile or to a specific bench [<Whole>/Bench]:

Select pile polylines to have direction reversed:

Select objects:

Pulldown Menu Location: Spoil

Keyboard Command: `spoil_reverse_dir`

Related Commands: Assign Directions, Display Directions, Clear Directions

Clear Directions

This command removes the directions from piles. The Extended Entity Data (EED) designating pile direction is removed from the pile polyline entity. This command has no effect if direction has not been previously assigned to a spoil polyline. Keep in mind that it is not necessary to first remove "old" directions before assigning "new" directions. The Assign Direction command will recognize that directions exist and will prompt the user to overwrite the direction for individual piles or for all piles.

Prompts

Clear direction to whole pile or to a specific bench [<Whole>/Bench]:

Select pile polylines to have direction removed:

Select objects: *pick the pile polylines* This can be done individually or with a window.

Pulldown Menu Location: Spoil

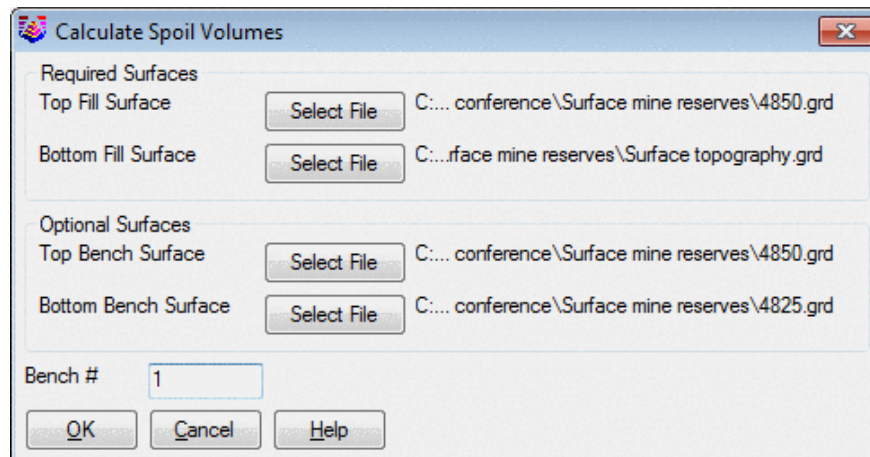
Keyboard Command: spoil_clear_dir

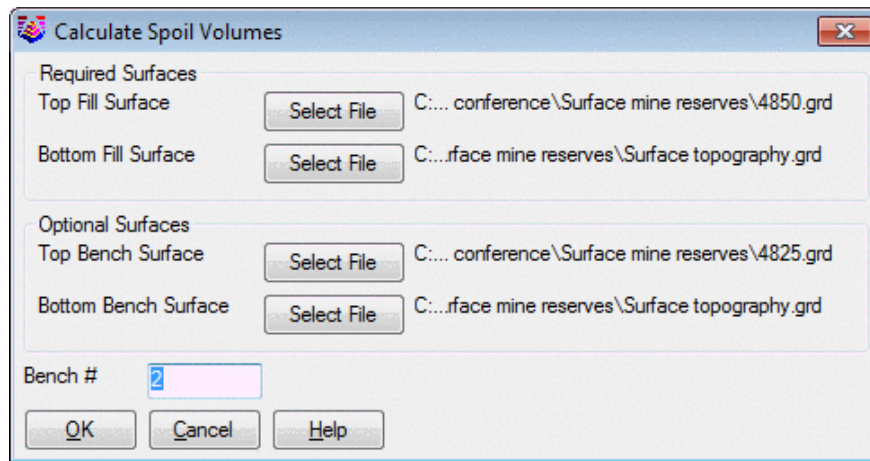
Related Commands: Assign Directions, Display Directions

Calculate Spoil Volumes

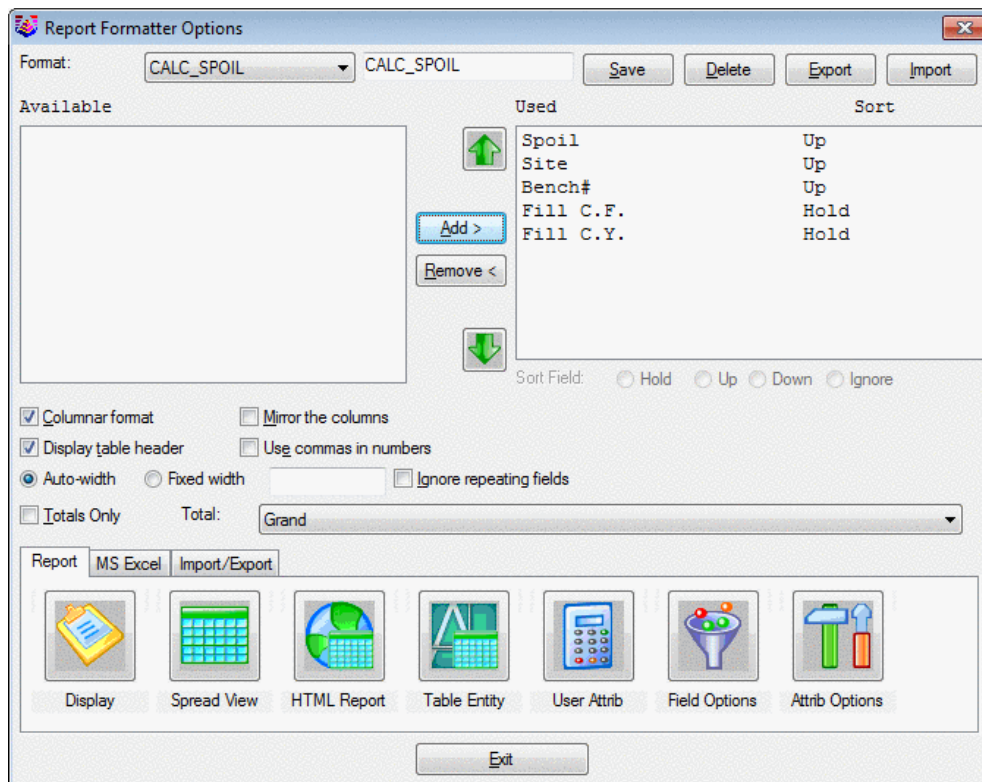
This command calculates the volume of the spoil based on the selected grid surfaces. There are options to select the Top Fill Surface, Bottom Fill Surface and two optional top and bottom of bench grids. This command needs to be run separately for each bench to calculate the volumes for each one. The volumes are automatically stored in the spoil polylines, ready for the timing process. They can be verified and edited with the command Edit Spoil Volumes.

The following example shows how to store volumes for two benches. The required surfaces are set by picking the top of the pile elevation of 4850, and the bottom of the pile, stacked onto the original Surface Topography grid. Bench 1 will be from the top of pile, also 4850, down to 4825 elevation grid. Bench 2 will then be from 4825 elevation to the bottom surface, the Surface Topography grid.

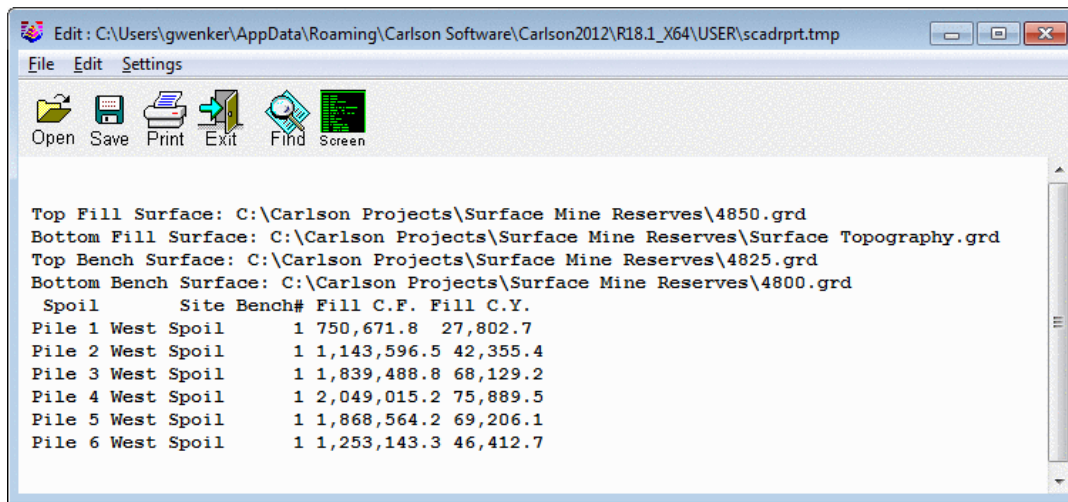




The Report Formatter window displays the available attributes to report, such as Spoil, Site Name, Bench#, Fill in CF, CY or CM if metric.



The report can be viewed with the Display button, or taken into Excel with the export to Excel option.



Prompts

Select spoil polylines to process.

Select objects: Specify opposite corner: 6 found

Select objects:

Pre-processing grid cells ...

Keyboard Command: calc_spoil

Prerequisite: Closed polylines named as spoil polylines

Pulldown Menu Location: Spoil in Surface Mine Module

Edit Spoil Volumes

This command displays the spoil destination results in a spreadsheet editor. The Site Name, Pile Name, Bench, Volume and Centroid are in individual columns. This is an editor where the data can be reviewed to see what the volumes are. Any changes made here can be used for the spoil timing and reporting.

	Site	Pit	Bench	Full Name	Volume	Centroid Location		
					(C.Y.)	X	Y	Z
1	West Spoil	Pile 1	1	West Spoil-Pile 1-B1	49231.44	1456651.45	1971993.14	0.00
2			2	West Spoil-Pile 1-B2	27802.66	1456651.45	1971993.14	0.00
3		Pile 2	1	West Spoil-Pile 2-B1	60187.45	1456536.40	1972122.53	0.00
4			2	West Spoil-Pile 2-B2	43379.76	1456536.40	1972122.53	0.00
5		Pile 3	1	West Spoil-Pile 3-B1	82467.54	1456438.03	1972259.85	0.00
6			2	West Spoil-Pile 3-B2	76883.51	1456438.03	1972259.85	0.00
7		Pile 4	1	West Spoil-Pile 4-B1	80422.86	1456333.39	1972390.34	0.00
8			2	West Spoil-Pile 4-B2	97880.03	1456333.39	1972390.34	0.00
9		Pile 5	1	West Spoil-Pile 5-B1	69594.77	1456234.33	1972499.26	0.00
10			2	West Spoil-Pile 5-B2	105778.22	1456234.33	1972499.26	0.00
11		Pile 6	1	West Spoil-Pile 6-B1	46412.72	1456144.20	1972601.84	0.00
12			2	West Spoil-Pile 6-B2	90432.37	1456144.20	1972601.84	0.00

Prompts

Select spoil polylines to process.

Select objects: Specify opposite corner: 6 found

Select objects:

Keyboard Command: edit_spoil

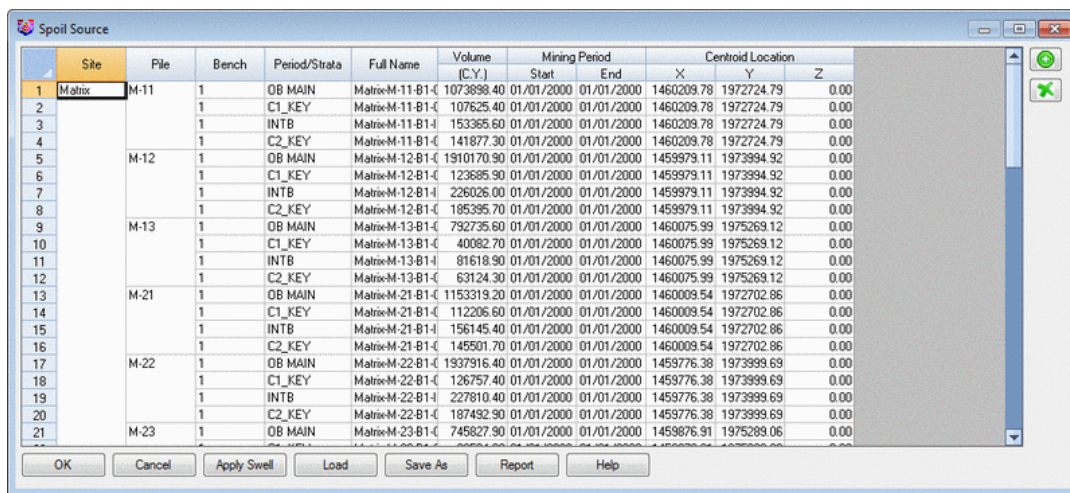
Prerequisite: Closed polylines named as spoil polylines assigned volumes.

Pulldown Menu Location: Spoil in Surface Mine Module

Edit Spoil Source

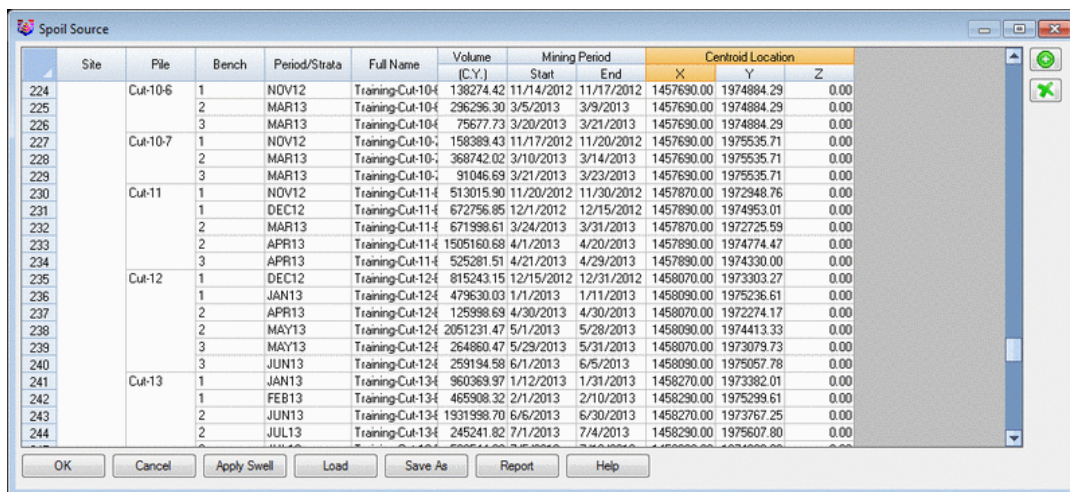
This command brings up a table of the Spoil Source that is required for the Spoil Timing commands. This displays where the material to be placed in the spoil polylines will come from. It shows the Site and Pit names to be mined. It also shows the Bench number, period or strata for each bench, full name and volume. If the SPO was created with the Surface Equipment Timing commands, then the Period will be used instead of the Strata Names, and the Mining Period Start and End dates will be correctly assigned. Otherwise, from Surface Mine Reserves, just the Strata Names and their volumes will apply.

This image shows the source of the spoil material calculated with Surface Mine Reserves.



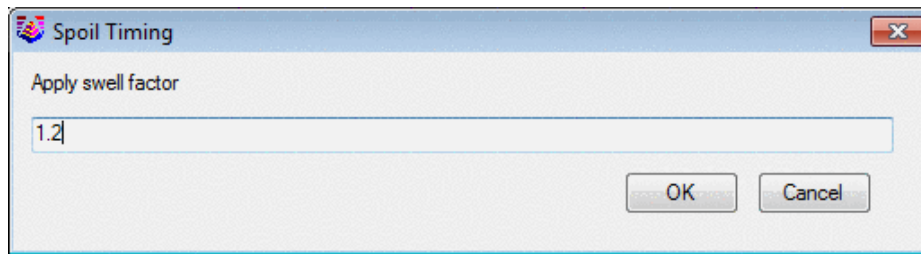
Site	File	Bench	Period/Strata	Full Name	Volume (C.Y.)	Mining Period Start	Mining Period End	Centroid Location X	Centroid Location Y	Centroid Location Z
1	M-11	1	OB MAIN	Matrise-M-11-B1-C	1073898.40	01/01/2000	01/01/2000	1460209.78	1972724.79	0.00
2		1	C1_KEY	Matrise-M-11-B1-C	107625.40	01/01/2000	01/01/2000	1460209.78	1972724.79	0.00
3		1	INTB	Matrise-M-11-B1-C	153365.60	01/01/2000	01/01/2000	1460209.78	1972724.79	0.00
4		1	C2_KEY	Matrise-M-11-B1-C	141877.30	01/01/2000	01/01/2000	1460209.78	1972724.79	0.00
5	M-12	1	OB MAIN	Matrise-M-12-B1-C	1910170.90	01/01/2000	01/01/2000	1459979.11	1973994.92	0.00
6		1	C1_KEY	Matrise-M-12-B1-C	123685.90	01/01/2000	01/01/2000	1459979.11	1973994.92	0.00
7		1	INTB	Matrise-M-12-B1-C	226026.00	01/01/2000	01/01/2000	1459979.11	1973994.92	0.00
8		1	C2_KEY	Matrise-M-12-B1-C	185395.70	01/01/2000	01/01/2000	1459979.11	1973994.92	0.00
9	M-13	1	OB MAIN	Matrise-M-13-B1-C	792735.60	01/01/2000	01/01/2000	1460075.99	1975269.12	0.00
10		1	C1_KEY	Matrise-M-13-B1-C	40082.70	01/01/2000	01/01/2000	1460075.99	1975269.12	0.00
11		1	INTB	Matrise-M-13-B1-C	81618.90	01/01/2000	01/01/2000	1460075.99	1975269.12	0.00
12		1	C2_KEY	Matrise-M-13-B1-C	63124.30	01/01/2000	01/01/2000	1460075.99	1975269.12	0.00
13	M-21	1	OB MAIN	Matrise-M-21-B1-C	1153319.20	01/01/2000	01/01/2000	1460009.54	1972702.86	0.00
14		1	C1_KEY	Matrise-M-21-B1-C	112206.60	01/01/2000	01/01/2000	1460009.54	1972702.86	0.00
15		1	INTB	Matrise-M-21-B1-C	156145.40	01/01/2000	01/01/2000	1460009.54	1972702.86	0.00
16		1	C2_KEY	Matrise-M-21-B1-C	145501.70	01/01/2000	01/01/2000	1460009.54	1972702.86	0.00
17	M-22	1	OB MAIN	Matrise-M-22-B1-C	1937916.40	01/01/2000	01/01/2000	1459776.38	1973999.69	0.00
18		1	C1_KEY	Matrise-M-22-B1-C	126757.40	01/01/2000	01/01/2000	1459776.38	1973999.69	0.00
19		1	INTB	Matrise-M-22-B1-C	227810.40	01/01/2000	01/01/2000	1459776.38	1973999.69	0.00
20		1	C2_KEY	Matrise-M-22-B1-C	187492.90	01/01/2000	01/01/2000	1459776.38	1973999.69	0.00
21	M-23	1	OB MAIN	Matrise-M-23-B1-C	745827.90	01/01/2000	01/01/2000	1459876.91	1975289.06	0.00

This image shows the Spoil Source calculated on the same pits, but with Surface Mine Reserves, over three benches instead. It shows the Period instead of strata names, and the Mining Period Start and End dates are straight from the Surface Timing command, into the SPO file.



Site	File	Bench	Period/Strata	Full Name	Volume (C.Y.)	Mining Period Start	Mining Period End	Centroid Location X	Centroid Location Y	Centroid Location Z
224	Cut-10-6	1	NOV12	Training-Cut-10-6	138274.42	11/14/2012	11/17/2012	1457690.00	1974884.29	0.00
225		2	MAR13	Training-Cut-10-6	296296.30	3/5/2013	3/9/2013	1457690.00	1974884.29	0.00
226		3	MAR13	Training-Cut-10-6	75677.73	3/20/2013	3/21/2013	1457690.00	1974884.29	0.00
227	Cut-10-7	1	NOV12	Training-Cut-10-7	158389.43	11/17/2012	11/20/2012	1457690.00	1975535.71	0.00
228		2	MAR13	Training-Cut-10-7	368742.02	3/10/2013	3/14/2013	1457690.00	1975535.71	0.00
229		3	MAR13	Training-Cut-10-7	91046.69	3/21/2013	3/23/2013	1457690.00	1975535.71	0.00
230	Cut-11	1	NOV12	Training-Cut-11-4	513015.90	11/20/2012	11/30/2012	1457870.00	1972948.76	0.00
231		1	DEC12	Training-Cut-11-4	672756.85	12/1/2012	12/15/2012	1457870.00	1974953.01	0.00
232		2	MAR13	Training-Cut-11-4	671998.61	3/24/2013	3/31/2013	1457870.00	1972725.59	0.00
233		2	APR13	Training-Cut-11-4	1505160.68	4/1/2013	4/20/2013	1457870.00	1974774.47	0.00
234		3	APR13	Training-Cut-11-4	525281.51	4/21/2013	4/29/2013	1457870.00	1974330.00	0.00
235	Cut-12	1	DEC12	Training-Cut-12-4	815243.15	12/15/2012	12/31/2012	1458070.00	1973303.27	0.00
236		1	JAN13	Training-Cut-12-4	479630.03	1/1/2013	1/11/2013	1458070.00	1975236.61	0.00
237		2	APR13	Training-Cut-12-4	125998.69	4/30/2013	4/30/2013	1458070.00	1972274.17	0.00
238		2	MAY13	Training-Cut-12-4	2051231.47	5/1/2013	5/28/2013	1458070.00	1974413.33	0.00
239		3	MAY13	Training-Cut-12-4	264860.47	5/29/2013	5/31/2013	1458070.00	1973079.73	0.00
240		3	JUN13	Training-Cut-12-4	259194.58	6/1/2013	6/5/2013	1458070.00	1975067.78	0.00
241	Cut-13	1	JAN13	Training-Cut-13-4	960369.97	1/12/2013	1/31/2013	1458270.00	1973382.01	0.00
242		1	FEB13	Training-Cut-13-4	465908.32	2/1/2013	2/10/2013	1458290.00	1975299.61	0.00
243		2	JUN13	Training-Cut-13-4	1931998.70	6/6/2013	6/30/2013	1458270.00	1973767.25	0.00
244		2	JUL13	Training-Cut-13-4	245241.82	7/1/2013	7/4/2013	1458290.00	1975607.80	0.00

The Apply Swell will swell the calculated volume stored in the SPO file. If it isn't swelled, then it will use the in-place, bank volume. The value is a percentage greater than 1.0. For example, 1.2 is a 20% swell of the spoiled material. Once OK is selected, the volumes should change on screen to a larger, swelled volume.



The Report button will bring up the Report Formatter where the requested fields will be reported in the text window, or exported to Excel. Shown here is an example text report.

Site	File	Bench	Period		Spoil Volume (C.Y.)	Centroid X	Centroid Y	Centroid Z
Pits X-10		1	TOPSOIL	Pits-X-10-B1-TOPSOIL	23280.40	1459556.57	1972145.09	0.00
Pits X-10		1	PREBENCH	Pits-X-10-B1-PREBENCH	655701.30	1459556.57	1972145.09	0.00
Pits X-10		1	OB MAIN	Pits-X-10-B1-OB MAIN	1259852.20	1459556.57	1972145.09	0.00
Pits X-10		1	INTB	Pits-X-10-B1-INTB	265296.60	1459556.57	1972145.09	0.00
Pits X-11		1	TOPSOIL	Pits-X-11-B1-TOPSOIL	26029.20	1459556.57	1972395.09	0.00
Pits X-11		1	PREBENCH	Pits-X-11-B1-PREBENCH	629760.40	1459556.57	1972395.09	0.00
Pits X-11		1	OB MAIN	Pits-X-11-B1-OB MAIN	1336106.70	1459556.57	1972395.09	0.00
Pits X-11		1	INTB	Pits-X-11-B1-INTB	279654.00	1459556.57	1972395.09	0.00
Pits X-12		1	TOPSOIL	Pits-X-12-B1-TOPSOIL	20612.90	1459503.81	1972641.37	0.00
Pits X-12		1	PREBENCH	Pits-X-12-B1-PREBENCH	571127.40	1459503.81	1972641.37	0.00
Pits X-12		1	OB MAIN	Pits-X-12-B1-OB MAIN	1239599.70	1459503.81	1972641.37	0.00
Pits X-12		1	INTB	Pits-X-12-B1-INTB	257164.90	1459503.81	1972641.37	0.00
Pits X-13		1	TOPSOIL	Pits-X-13-B1-TOPSOIL	17798.50	1459398.77	1972894.44	0.00
Pits X-13		1	PREBENCH	Pits-X-13-B1-PREBENCH	517758.60	1459398.77	1972894.44	0.00
Pits X-13		1	OB MAIN	Pits-X-13-B1-OB MAIN	1057885.70	1459398.77	1972894.44	0.00
Pits X-13		1	INTB	Pits-X-13-B1-INTB	218376.20	1459398.77	1972894.44	0.00
Pits X-14		1	TOPSOIL	Pits-X-14-B1-TOPSOIL	17096.10	1459387.48	1973144.87	0.00
Pits X-14		1	PREBENCH	Pits-X-14-B1-PREBENCH	640626.00	1459387.48	1973144.87	0.00
Pits X-14		1	OB MAIN	Pits-X-14-B1-OB MAIN	1228960.40	1459387.48	1973144.87	0.00
Pits X-14		1	INTB	Pits-X-14-B1-INTB	251686.00	1459387.48	1973144.87	0.00
Pits X-15		1	TOPSOIL	Pits-X-15-B1-TOPSOIL	19032.50	1459451.77	1973399.68	0.00

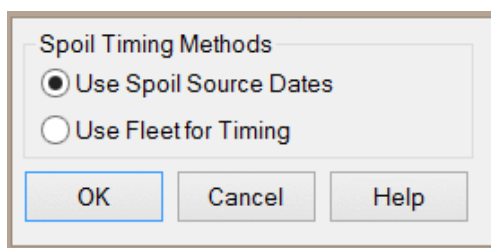
Keyboard Command: edit_spo

Prerequisite: SPO file from Surface Mine Reserves or Surface Equipment Timing.

Pulldown Menu Location: Spoil in Surface Mine Module

Spoil Placement Timing

This command brings all the spoil commands together as a final step and performs the timing and scheduling of the spoil placement. The first screen will allow you to choose two options for timing; either 'Use Spoil Source Dates' or 'Use Fleet for Timing' (shown below). These two options will produce very different results, and should be selected depending on the needs of spoil removal and placement.

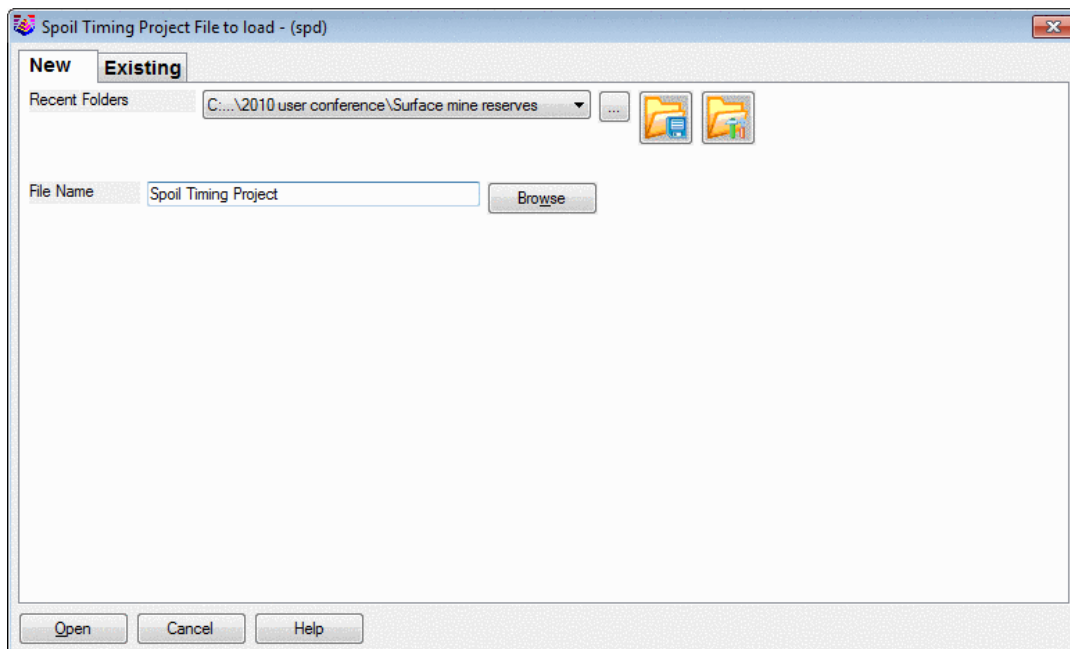


The 'Spoil Source Dates' option will move the material to the spoil area as it is excavated from the pit based on the production schedule. This option requires a spoil file (.spo) generated from Surface Equipment Timing. A .spo file

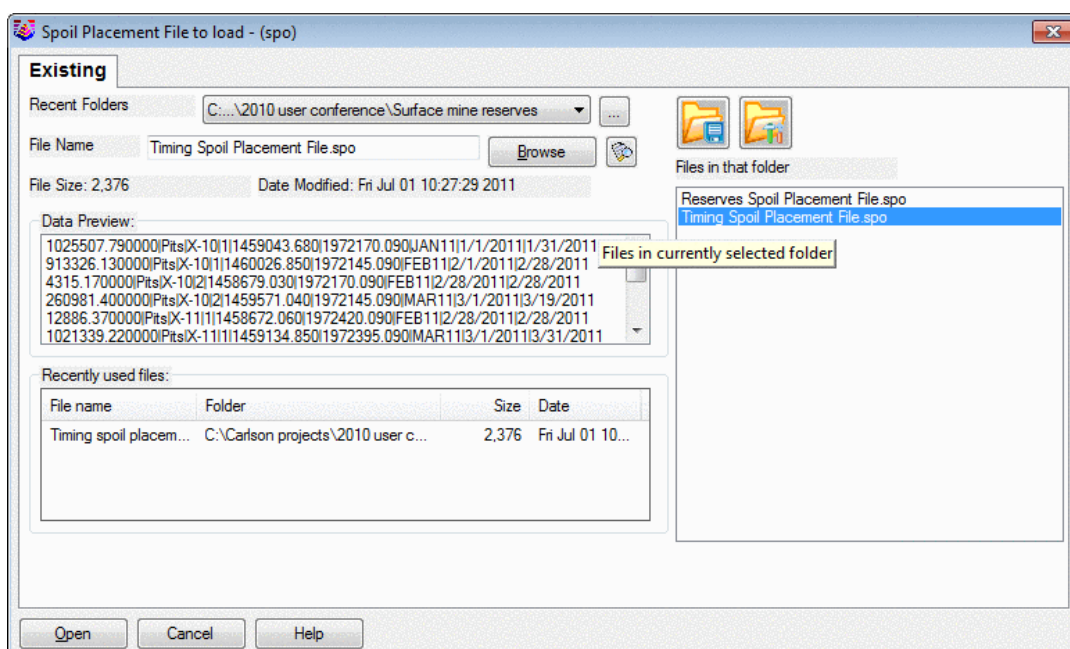
generated from Surface Mine Reserves does not include start and end dates for the pit excavation, and therefore there are no Spoil Source Dates to be used.

The 'Use Fleet for Timing' option can be used with a .spo file generated from either Surface Mine Reserves or Surface Equipment Timing. This option does not consider the production schedule of the mine, and will instead move the spoil material as quickly as the haulage fleet will allow. As this option does not consider the production schedule, in general it will be sensible to only use a .spo file generated with Surface Mine Reserves with this option.

After selecting the timing option, you will be prompted for a Spoil Timing Project file (.spd). This file stores information concerning the spoil sources, spoil volumes, start and end dates of excavation, spoil/pit centroid locations, haul fleet production capacities, spoil attributes, etc.



After specifying the .spd file, you will be prompted to select the .spo file.



Once the .spo file has been selected, you will see a preview of the file which can be edited before it is used for the spoil timing. The preview of the spoil source is shown below.

	Site	Pile	Bench	Period/Strata	Full Name	Volume	Mining Period		Centroid Location		
						[C.Y.]	Start	End	X	Y	Z
1	Pits	X-10	1	JAN11	Pits-X-10-B1-JAN11	1025507.79	1/1/2011	1/31/2011	1459043.68	1972170.09	0
2			1	FEB11	Pits-X-10-B1-FEB11	913326.13	2/1/2011	2/28/2011	1460026.85	1972145.09	0
3			2	FEB11	Pits-X-10-B2-FEB11	4315.17	2/28/2011	2/28/2011	1458679.03	1972170.09	0
4			2	MAR11	Pits-X-10-B2-MAR11	260981.40	3/1/2011	3/19/2011	1459571.04	1972145.09	0
5		X-11	1	FEB11	Pits-X-11-B1-FEB11	12886.37	2/28/2011	2/28/2011	1458672.06	1972420.09	0
6			1	MAR11	Pits-X-11-B1-MAR11	1021339.22	3/1/2011	3/31/2011	1459134.85	1972395.09	0
7			1	APR11	Pits-X-11-B1-APR11	957670.68	4/1/2011	4/30/2011	1460018.22	1972395.09	0
8			2	APR11	Pits-X-11-B2-APR11	13406.79	4/30/2011	4/30/2011	1459701.56	1972420.09	0
9			2	MAY11	Pits-X-11-B2-MAY11	266247.23	5/1/2011	5/21/2011	1459599.20	1972395.09	0
10		X-12	1	APR11	Pits-X-12-B1-APR11	31096.14	4/30/2011	4/30/2011	1458678.79	1972670.09	0
11			1	MAY11	Pits-X-12-B1-MAY11	1033773.71	5/1/2011	5/31/2011	1459166.86	1972645.09	0
12			1	JUN11	Pits-X-12-B1-JUN11	766470.15	6/1/2011	6/23/2011	1460098.14	1972645.22	0
13			2	JUN11	Pits-X-12-B2-JUN11	94194.52	6/24/2011	6/30/2011	1459911.98	1972670.09	0
14			2	JUL11	Pits-X-12-B2-JUL11	162970.36	7/1/2011	7/13/2011	1459978.43	1972645.22	0
15		X-13	1	JUN11	Pits-X-13-B1-JUN11	236581.40	6/24/2011	6/30/2011	1458754.36	1972920.09	0
16			1	JUL11	Pits-X-13-B1-JUL11	1045374.37	7/1/2011	7/31/2011	1459364.70	1972895.09	0
17			1	AUG11	Pits-X-13-B1-AUG11	311486.94	8/1/2011	8/10/2011	1460024.51	1972880.37	0
18			2	AUG11	Pits-X-13-B2-AUG11	218376.23	8/10/2011	8/25/2011	1459405.93	1972903.66	0
19		X-14	1	AUG11	Pits-X-14-B1-AUG11	721985.54	8/10/2011	8/31/2011	1458888.01	1973170.09	0
20			1	SEP11	Pits-X-14-B1-SEP11	995246.04	9/1/2011	9/30/2011	1459598.85	1973145.09	0
21			1	OCT11	Pits-X-14-B1-OCT11	169450.84	10/1/2011	10/6/2011	1460056.99	1973165.76	0
22			2	OCT11	Pits-X-14-B2-OCT11	251696.00	10/6/2011	10/24/2011	1459388.26	1973183.14	0
23		X-15	1	OCT11	Pits-X-15-B1-OCT11	866490.64	10/6/2011	10/31/2011	1458958.07	1973420.09	0
24			1	NOV11	Pits-X-15-B1-NOV11	997494.34	11/1/2011	11/30/2011	1458897.32	1973389.72	0
25			2	DEC11	Pits-X-15-B2-DEC11	240976.76	12/1/2011	12/17/2011	1459450.17	1973411.45	0
26		X-16	1	NOV11	Pits-X-16-B1-NOV11	6492.37	11/30/2011	11/30/2011	1458669.25	1973670.09	0
27			1	DEC11	Pits-X-16-B1-DEC11	1041749.98	12/1/2011	12/31/2011	1459031.73	1973645.09	0
28			1	JAN12	Pits-X-16-B1-JAN12	1041749.98	1/1/2012	1/31/2012	1459751.32	1973645.09	0
29			1	FEB12	Pits-X-16-B1-FEB12	484217.53	2/1/2012	2/15/2012	1460309.82	1973621.18	0
30			2	FEB12	Pits-X-16-B2-FEB12	203985.24	2/15/2012	2/29/2012	1459123.00	1973670.09	0

The next dialog shown will be the Spoil Timing Project Manager. This shows a tree structure of the project, which includes the haul fleet (if used), the spoil sources, the spoil destinations, and the spoil attributes. Certain aspects of this dialog can be edited using the 'Edit' button when a field is highlighted. This dialog is shown below.

The assigned and available lists shows the spoil destinations that can be used for timing. Spoil destinations can be moved between the available and assigned lists by using the left/right arrows between the two lists. The up/down arrows between the lists controls the order in which the spoil areas will be filled. The arrow-and-road icon will automatically assign spoil destinations based on the shortest haul distance between the spoil source and the spoil destination. This command requires you to have selected the haul roads using the Haul Road Manager command.

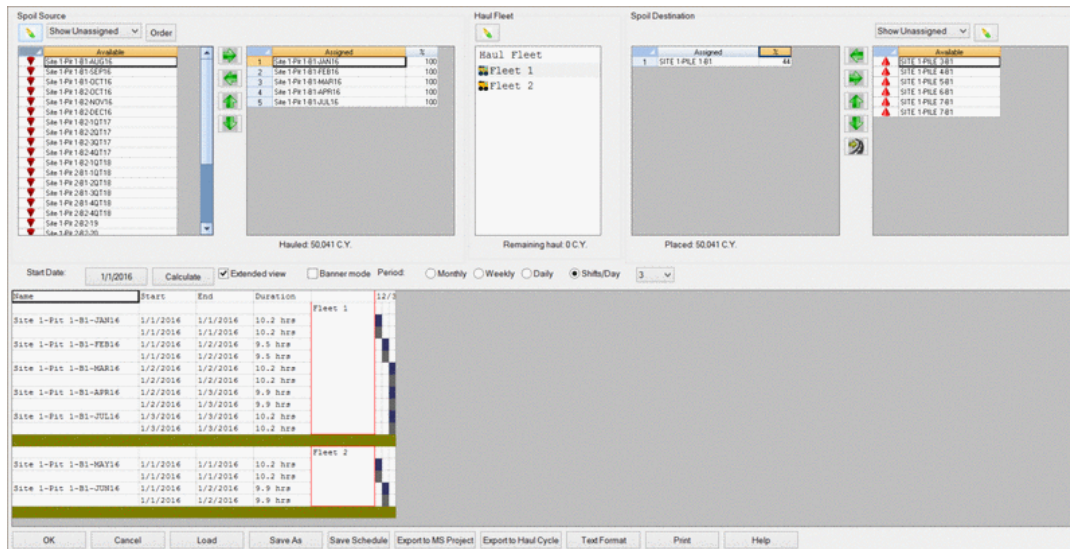
The assigned list also shows the fill percentage of each spoil pile. Note that each spoil pile will be filled to capacity before spoil is placed in the next pile. Both the spoil source and spoil destination information can be directly edited by clicking the green pen icons above each list. The total amount of spoil to be moved is listed under the spoil source list. The total amount of spoil that has been assigned to a spoil destination is listed under the assigned list.

The bottom section of the dialog shows a Gantt chart of the spoil placement timing. The use of Gantt charts are discussed in the Surface Equipment Timing section of the help manual. Clicking 'Calculate' will determine the final timing of the spoil placement. As the dialogs shown after clicking 'Calculate' are the same for both the 'Use Spoil Source Dates for Timing' and the 'Use Fleet for Timing' options, discussion of the output is discussed in the Spoil Calculation Output section of this page. One item that is specific to Gantt charts in the Spoil Timing dialog is the Export to Haul Cycle option. This option exports a .sph file which can be used with the Haul Cycle Analysis command to calculate a more detailed report of the timing associated with spoil placement.

You may also save and load the spoil timing assignments in a Spoil Assignment file (.stm)

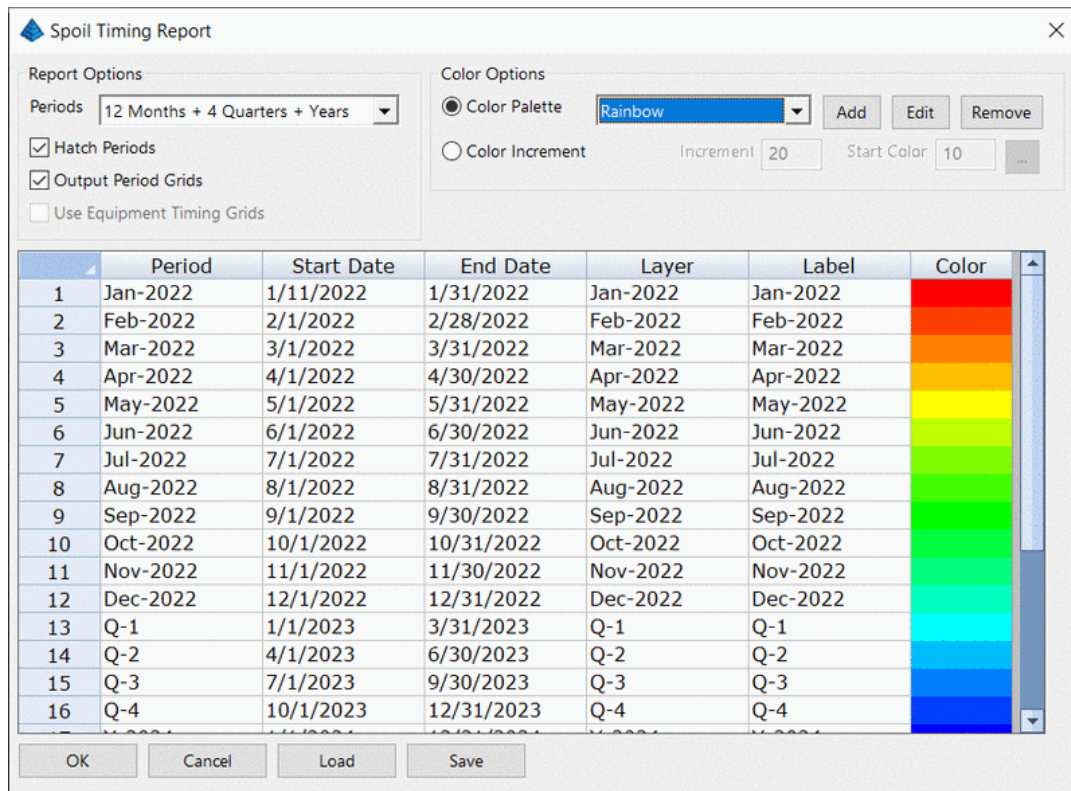
Note that this option will require a haulage fleet for timing. With this option, you may create multiple haulage fleets, which can be assigned to various spoil sources and destinations for timing.

The Spoil Placement Timing dialog for this option will appear as shown below.

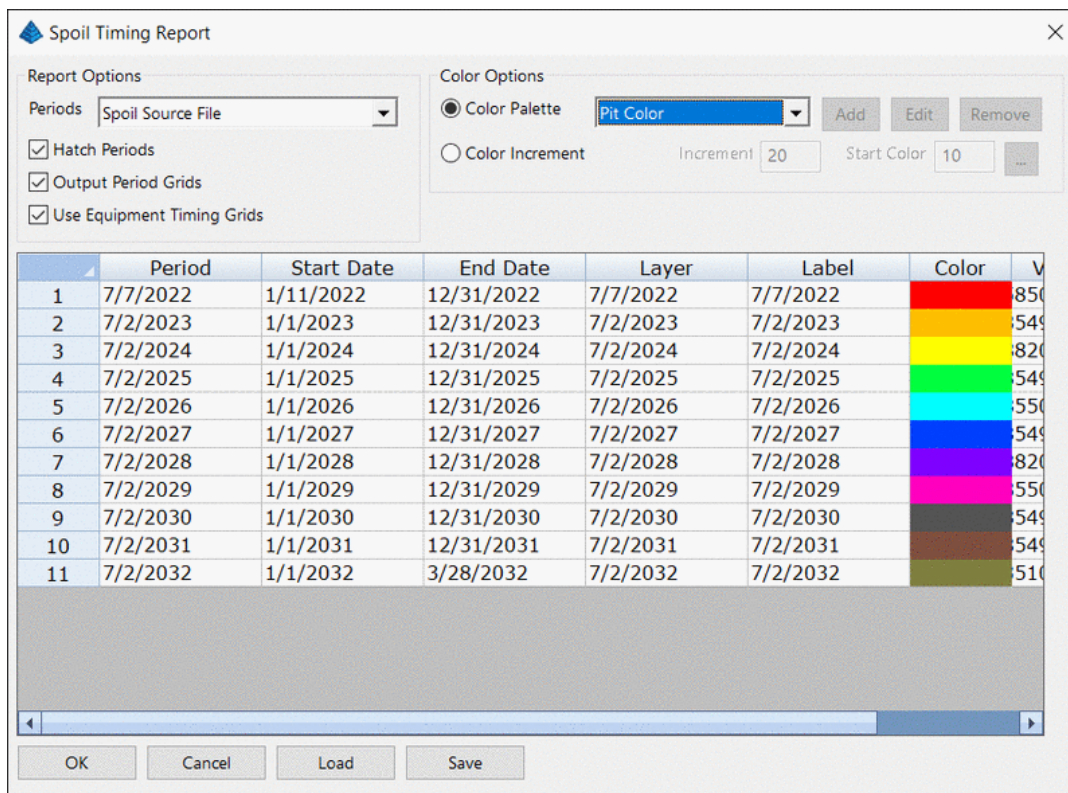


Unlike the dialog shown for the 'Use Spoil Source Dates for Timing' option, this dialog allows you to use only specified spoil sources for timing. Aside from this difference, the two dialogs are similar in functionality. Spoil sources and destinations can be moved between the assigned and available lists and reordered using the green arrows between the lists. This dialog also shows a new list of the haul fleets that can be used. Selecting a haul fleet will list the spoil sources and destinations that are assigned to that fleet. Another additional feature to this dialog is that you can specify a start date for the spoil timing.

After clicking the 'Calculate' button, you will see the dialog shown below. This allows you to set the type and appearance of the hatch, text and timing report output.

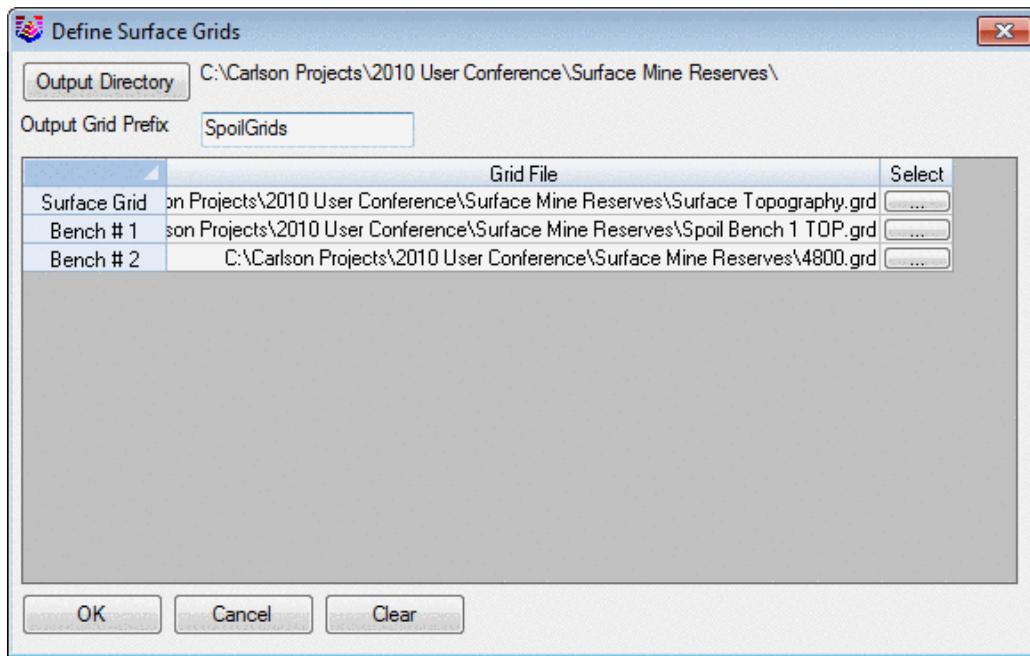


The Report Options will set the time periods for the timing report. The spreadsheet will be automatically updated based on the selected report type. The Layer, Label, and Color columns may be manually updated, but the Period, Start Date, and End Date columns are fixed. If the Spoil Source File report type is selected, the periods will be set by the .spo file created in Surface Equipment Timing. This option will also enable the Use Equipment Timing Grids option, which will merge the spoil timing output grids or TIN's with a .gsq file created from the Surface Equipment Timing command (shown below) The Pit Color option in the Color Palette will match up the period colors of the spoil timing map to the period colors of the pit advance periods.



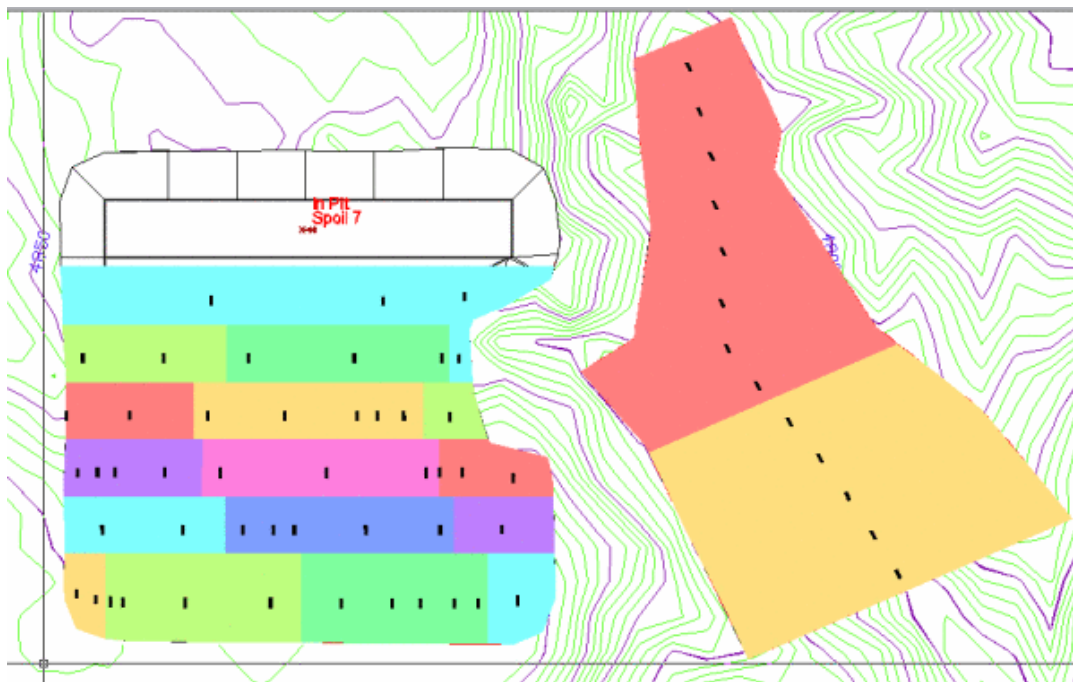
- **Hatch Periods:** This option will allow you to hatch the time periods with a solid fill color. If this option is turned off, the timing map will only show outlines and labels of the timing periods.
- **Output Period Grids:** This option will output grid or TIN files representing the surface at various time periods. These surfaces can be used to review the development of the mine area.
- **Use Equipment Timing Grids:** This option will merge the grid/TIN files created with the Output Period Grids option with a .gsq file created from Surface Equipment Timing. This can be used for detailed review of the surface development, showing the timing and locations of material movement.

If you use the Output Period Grids option, you will see the Define Surface Grids dialog after clicking OK (shown below). This dialog allows you to set the surfaces which define the existing ground and bench surfaces of the spoil piles.

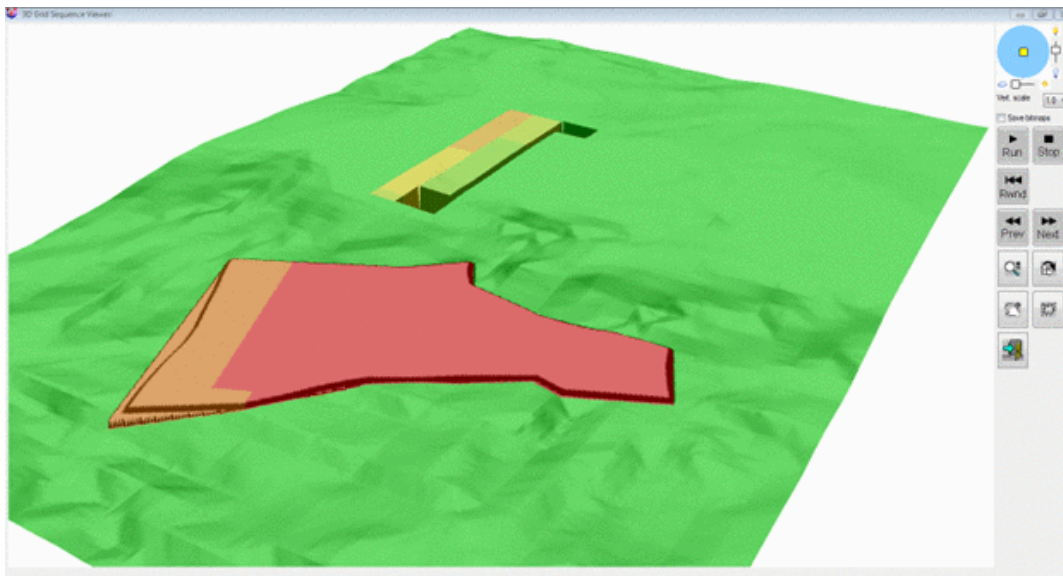


If you use the Use Equipment Timing Grids option, you will be prompted to select a .gsq file to update. Note that you must have an existing .gsq file for this option. This command will merge the output surfaces with the .gsq file at the respective time periods, which can be saved as a new .gsq file. This will represent the full mining progression, showing the advance of the pits, and the following of the spoil and dumps.

The below plan view map is hatched with the color period blocks to illustrate where each spoil is placed in each period.



Viewing the new .gsq file with the View 3D Surface History command will show the mining progression with the spoil fills.



Using the Report Formatter can export the report directly into Excel. An example Excel dump is shown here.

1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24	25	26	27	28	29	30	31	32						
Period Dates	Fleet	Source	Source Period	Spoil	Site	Pile	Bench	Start Date	Finish Date	Volume (C.Y.)	Shifts	Centroid X	Centroid Y	Centroid Z	Area (S.F.)	Source X	Source Y	Source Z	Spoil X	Spoil Y																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 1- Out of Pile 1	1	1		01/01/2012	01/01/2012	39946.34	0.80	1461218.00	1974373.49	0.00	99182.64	1458952.67	1972170.09	0.00	1461218.00	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 2- Out of Pile 2	1	1		01/01/2012	01/02/2012	34979.05	1.10	1461266.01	1974176.81	0.00	119555.74	1458952.67	1972170.09	0.00	1461266.01	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 3- Out of Pile 3	1	1		01/02/2012	01/03/2012	79779.29	1.60	1461316.34	1973982.01	0.00	125085.78	1458952.67	1972170.09	0.00	1461316.34	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 4- Out of Pile 4	1	1		01/03/2012	01/04/2012	76172.52	1.52	1461332.54	1973769.99	0.00	123282.19	1458952.67	1972170.09	0.00	1461332.54	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 5- Out of Pile 5	1	1		01/04/2012	01/05/2012	96459.87	1.93	1461367.01	1973564.86	0.00	150635.55	1458952.67	1972170.09	0.00	1461367.01	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 6- Out of Pile 6	1	1		01/05/2012	01/05/2012	36.99	0.00	1461316.34	1973982.01	0.00	125085.78	1458952.67	1972170.09	0.00	1461316.34	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 6- Out of Pile 6	1	2		01/05/2012	01/07/2012	129847.59	2.60	1461364.60	1973341.35	0.00	197164.29	1458952.67	1972170.09	0.00	1461364.60	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 4- Out of Pile 4	2	1		01/07/2012	01/07/2012	3050.66	0.00	1461332.54	1973769.99	0.00	123282.19	1458952.67	1972170.09	0.00	1461332.54	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 7- Out of Pile 7	1	1		01/07/2012	01/20/2012	380469.66	3.81	1461389.41	1973142.79	0.00	238184.59	1458952.67	1972170.09	0.00	1461389.41	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 5- Out of Pile 5	2	1		01/10/2012	01/20/2012	7374.60	0.15	1461367.01	1973564.86	0.00	150635.55	1458952.67	1972170.09	0.00	1461367.01	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 7- Out of Pile 8	1	1		01/10/2012	01/13/2012	171108.84	3.42	1461318.46	1972980.03	0.00	232757.26	1458952.67	1972170.09	0.00	1461318.46	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 6- Out of Pile 6	2	1		01/13/2012	01/13/2012	24695.75	0.49	1461364.60	1973341.35	0.00	197164.29	1458952.67	1972170.09	0.00	1461364.60	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 9- Out of Pile 9	1	1		01/13/2012	01/16/2012	180860.64	3.62	1461657.32	1972823.15	0.00	247622.52	1458952.67	1972170.09	0.00	1461657.32	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 7- Out of Pile 7	2	1		01/16/2012	01/17/2012	61229.38	1.22	1461389.41	1973142.79	0.00	238184.59	1458952.67	1972170.09	0.00	1461389.41	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-JAN12	JAN12	Out of Pit-Pile 10 Out of Pile 10	1	1		01/17/2012	01/21/2012	47711.37	0.95	1461322.67	1972459.70	0.00	146862.83	1458952.67	1972170.09	0.00	1461390.42	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-FEB12	FEB12	Out of Pit-Pile 10 Out of Pile 10	1	1		01/17/2012	01/20/2012	151538.79	3.03	1461387.61	1972204.42	0.00	203471.84	1459713.99	1972145.09	0.00	1461790.42	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-FEB12	FEB12	Out of Pit-Pile 8- Out of Pile 8	2	1		01/20/2012	01/21/2012	58704.64	1.17	1461318.46	1972980.03	0.00	232757.26	1459713.99	1972145.09	0.00	1461518.46	0.00																	
1/1/2012-1/31/2012	In Pit Haulage	Pits-X-10-B1-FEB12	FEB12	Out of Pit-Pile 11 Out of Pile 11	1	1		01/21/2012	01/24/2012	186031.73	3.72	1461841.13	1972406.47	0.00	251863.10	1459713.99	1972145.09	0.00	1461912.96	0.00																	
										1549997.52		31.00																									
26	27	28	29	30	31	32																															
Period Dates	Fleet	Source	Source Period	Spoil	Site	Pile	Bench	Start Date	Finish Date	Volume (C.Y.)	Shifts	Centroid X	Centroid Y	Centroid Z	Area (S.F.)	Source X	Source Y	Source Z	Spoil X	Spoil Y																	
2/1/2012-2/29/2012	In Pit Haulage	Pits-X-10-B1-FEB12	FEB12	Out of Pit-Pile 11 Out of Pile 11	1	1		02/01/2012	02/01/2012	23022.78	0.46	1462493.41	1972743.06	0.00	31170.07	1459713.99	1972145.09	0.00	1461912.96	0.00																	
2/1/2012-2/29/2012	In Pit Haulage	Pits-X-10-B1-FEB12	FEB12	Out of Pit-Pile 9- Out of Pile 9	2	1		02/01/2012	02/02/2012	66646.56	1.33	1461657.32	1972823.15	0.00	247622.52	1459713.99	1972145.09	0.00	1461657.32	0.00																	
2/1/2012-2/29/2012	In Pit Haulage	Pits-X-10-B1-FEB12	FEB12	Out of Pit-Pile 12 Out of Pile 12	1	1		02/02/2012	02/06/2012	230625.23	4.61	1462022.72	1972326.59	0.00	292739.45	1459713.99	1972145.09	0.00	1462022.72	0.00																	
2/1/2012-2/29/2012	In Pit Haulage	Pits-X-10-B1-FEB12	FEB12	Out of Pit-Pile 10 Out of Pile 10	2	1		02/06/2012	02/08/2012	116830.43	2.34	1461790.42	1972601.03	0.00	207534.66	1459713.99	1972145.09	0.00	1461790.42	0.00																	
2/1/2012-2/29/2012	In Pit Haulage	Pits-X-10-B1-FEB12	FEB12	Out of Pit-Pile 13 Out of Pile 13	1	1		02/06/2012	02/12/2012	205894.48	4.92	1462134.32	1972149.03	0.00	246572.83	1459713.99	1972145.09	0.00	1462132.41	0.00																	
2/1/2012-2/29/2012	In Pit Haulage	Pits-X-10-B1-MAR12	MAR12	Out of Pit-Pile 10 Out of Pile 10	1	1		02/12/2012	02/12/2012	6465.39	0.13	1462026.73	1972429.18	0.00	7246.99	1460246.59	1972145.09	0.00	1462132.41	0.00																	
2/1/2012-2/29/2012	In Pit Haulage	Pits-X-10-B1-MAR12	MAR12	Out of Pit-Pile 11 Out of Pile 11	2	1		02/12/2012	02/14/2012	127739.89	2.55	1461912.96	1972496.93	0.00	280393.17	1460246.59	1972145.09	0.00	1461912.96	0.00																	
2/1/2012-2/29/2012	In Pit Haulage	Pits-X-10-B1-MAR12	MAR12	Out of Pit-Pile 12 Out of Pile 12	2	1		02/14/2012	02/17/2012	156636.96	3.13	1462022.72	1972326.59	0.00	282739.45	1460246.59	1972145.09	0.00	1462022.72	0.00																	

Keyboard Command: timespoil

Prerequisite: SPO file from Surface Mine Reserves or Surface Equipment Timing, spoil polylines with direction, and haulage fleet.

Pulldown Menu Location: Spoil in Surface Mine Module

Case Studies

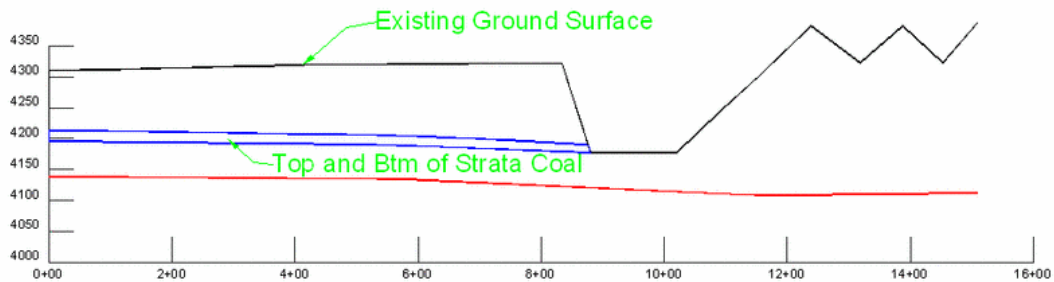
Case Study #7: Dragline Range Diagrams

This tutorial shows how to design steps of Cast Blast Profile, Dozer Push and Cut & Place in cross section view for dragline extended bench and spoil side methods. It then takes this design "template" and applies it to plan view in 3D, extruding along the pit using a centerline file (CL). Other routines we will use include Define Dragline Equipment, Cut Only (coal removal), Draw Dragline limits, and Flatten Spoil Top.

Creating Surface and Strata Cross-Sections

(Geology module > Surface pull-down menu> Cast Blast Command)

The first step is to generate a cross section of the ground surface and desired strata. This can be accomplished with the fence diagram routine, or drawing multiple profiles or sections that represent these surfaces on a grid. If the fence diagram method is used, be sure to turn off hatching, and turn on draw strata polylines as single polylines. Otherwise, the strata will be closed polylines and the program won't use them correctly. If the profile crosses an existing, open pit and seams are cropping, be sure to turn on Draw Surface Polyline in Fence Diagram. This will draw a continuous polyline across the entire length of the section, needed for Cut & Place. Shown below is a cross-section of an existing pit.



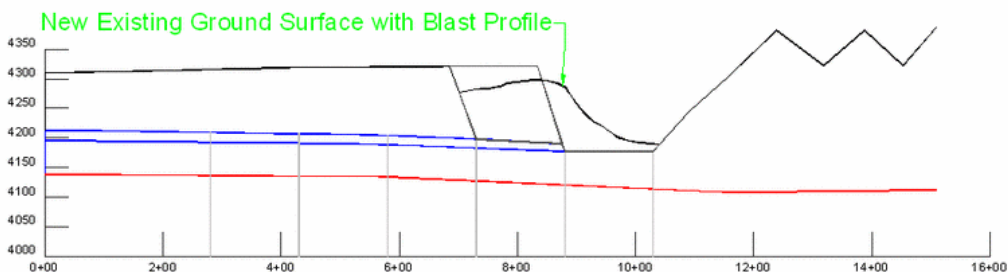
EXTENDED BENCH EXAMPLE

Cast Blast

(Surface Mining module > Surface pull-down menu> Cast Blast Command)

The first operation is to simulate the cast blast for creating the extended bench. The generic profile of the blast can be defined as either a Carlson profile (*.pro) or as a polyline drawn on the screen, above the fence diagram. The profile file or polyline must be created before running Cast Blast. For example, you can draw the typical blast profile on the cross-section view. The program will move the polyline up or down to balance the cut and fill areas. The program can also save your blast profile polyline into a *.pro file via the Create Profile from Polyline on Profile Grid command. First, the program will display the below dialog box for options.

In this case, the swell factor was set to 1.2, or 20% and the profile by polyline option was used. Then select the ground and top-of-strata polylines. Next pick the blast or cut area, this example used the lower left and lower right cut points. The left point in this case should be at the cut width, and the right point is snapped to the intersection of the top of coal outcropping at the existing ground surface.



After the program places the blast, it prompts the user if there is any vertex on the blast that needs to be moved or changed. Simply pick the position and the line will move to that location. The existing ground surface line has now been changed, and is ready for the next step.

Steps needed to complete a Cast Blast (required blast profile file)

1. Select cast blast profile command.
2. Adjust settings in the dialogue to the image shown above accordingly.
3. Command: C:BLAST
 - Select existing grade polyline: *pick existing surface polyline*
 - Select strata polyline (Enter for none): *none*
 - Pick lower Left cut point: *any point along the top of the coal seam (upperblue polyline)*
 - Pick lower Right cut point: *intersection of the top of coal outcropping at the existing ground surface*
 - Cast to Final: 20.15%
 - Pick profile point to modify (Enter to end):

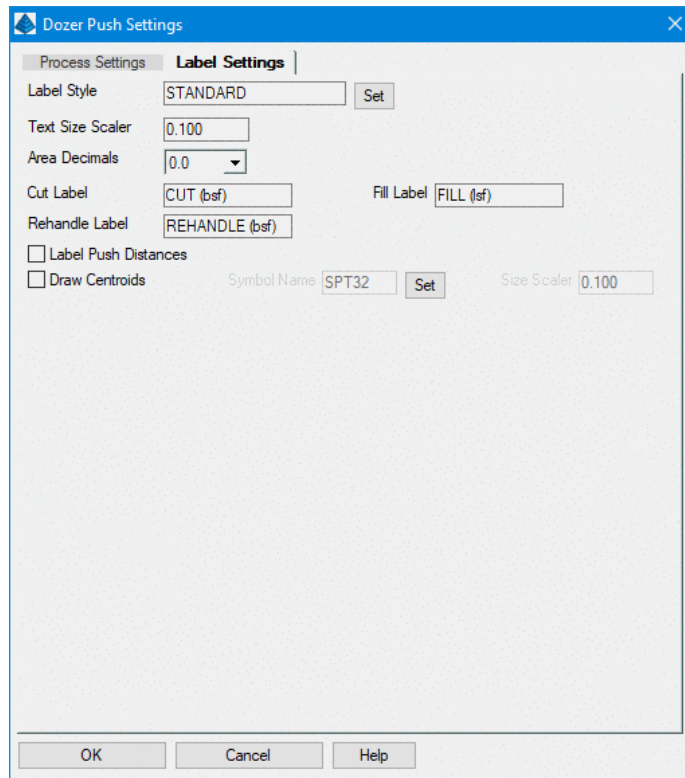
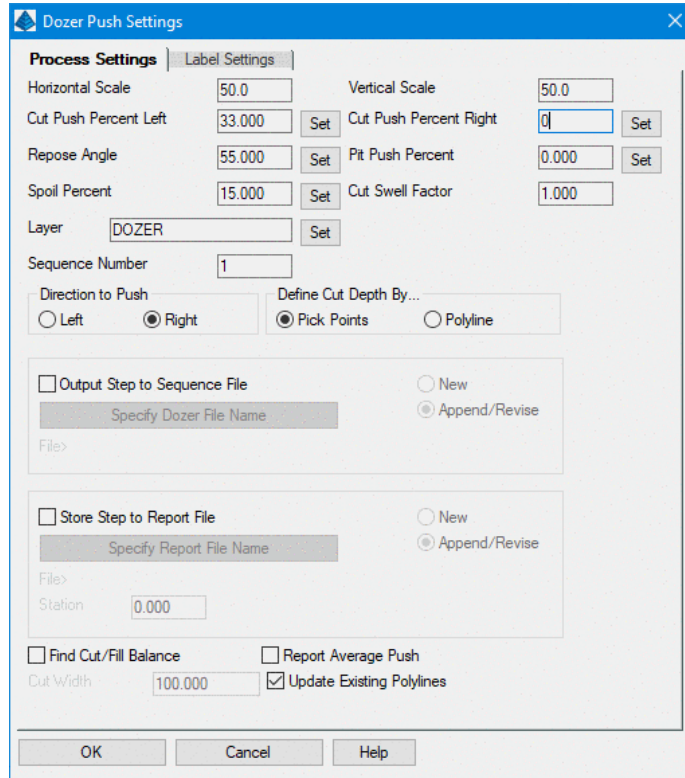
CUT: 17220.87 (sf) FILL: 20665.04 (sf)

4. Command is finished, check fence diagram for polyline updates.

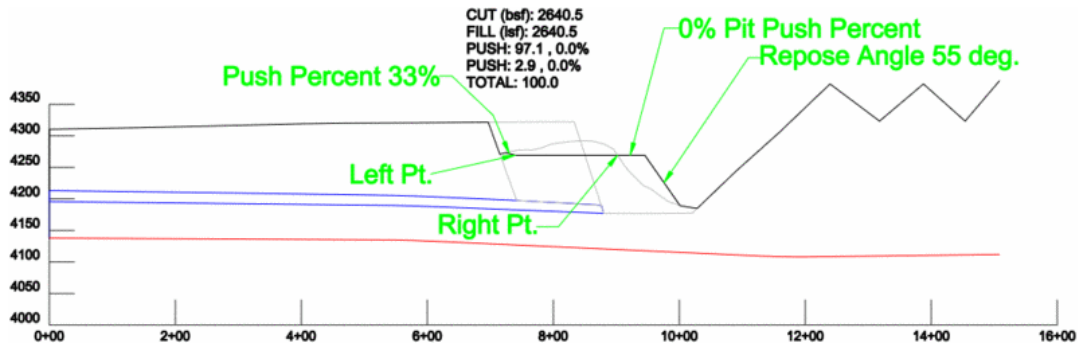
Dozer Push

(Surface Mining module > Surface pull-down menu > Dozer Push Command)

Now the Dozer Push command is used to create the extended bench.



In the dialog box, set the Pit Push Percent to 0%. This will push the material horizontally out into the pit. Turn off Label Push Distances, otherwise it will label the complete surface line. Also set the swell to 1.0 since this material has already had the swell factor applied from the Cast Blast. Choose OK and pick the left and right cut points as shown below. Use the Nearest Snap to pick the right cut point so it connects directly on the existing surface line. The left and right cut points should be selected at the same elevation if the push is to be flat before the pit also. The existing surface polyline has now been updated and is ready for the next operation.



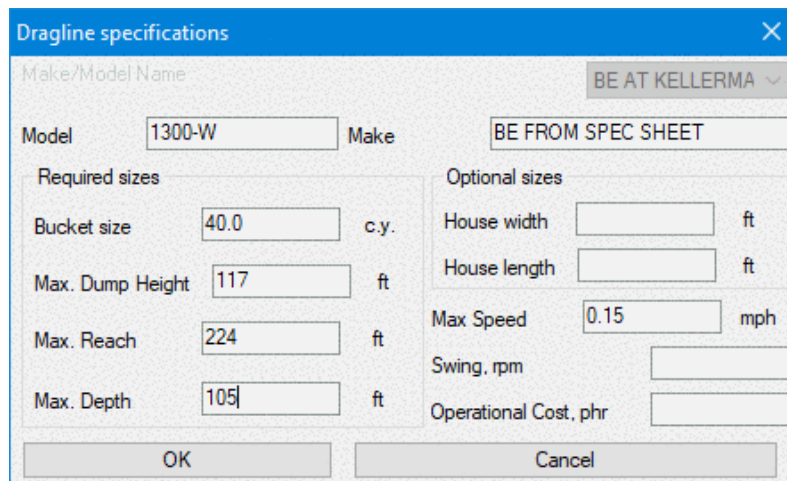
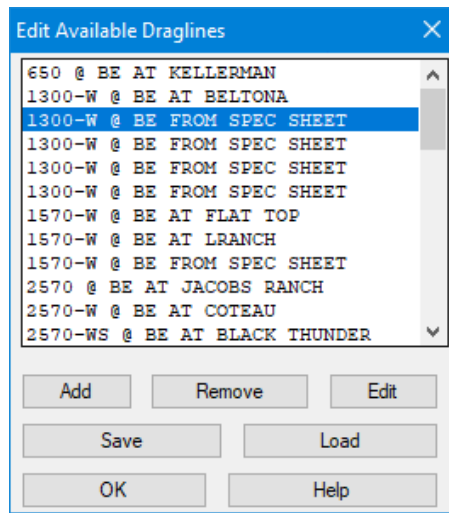
Steps needed to complete a Dozer Push

1. Select cast dozer push command.
2. Adjust settings in the dialogue to the image shown above accordingly.
3. Command: C:DOZER
 Select existing grade polyline: *pick new existing surface polyline*
 Pick lower Left cut point: *shown above*
 Pick lower Right cut point: *shown above*
 Select fill areas to rehandle (Enter for None). *none*
 CUT (bsf): 2640.5
 REHANDLE (bsf): 0.0
 FILL (lsf): 2640.5
 PUSH: 97.1, 0.0% to 2.9, 0.0%
 Total: 100
 Pick Point for Label (Enter for none): *anywhere on the fence diagram to show the summary*
4. Command is finished, check fence diagram for polyline updates.

Define Dragline Equipment

(Surface Mining module > Surface pull-down menu> Define Dragline Equipment Command)

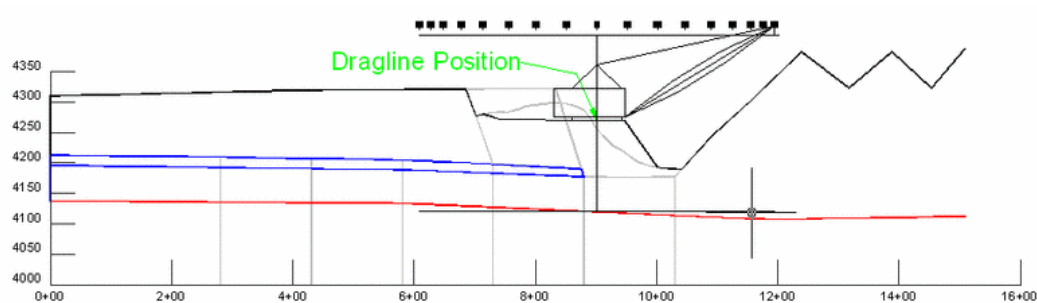
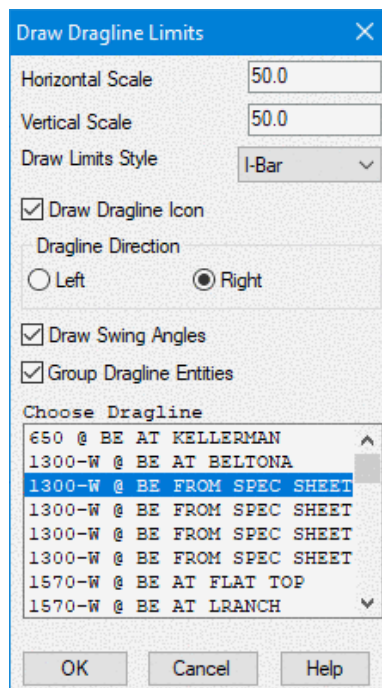
This routine simply defines the dragline parameters and dimensions for drawing its reach, depth and height on the range diagram. Fill in at least the required size parameters on the left side of the box. Example dragline dimensions are provided for your convenience.



Draw Dragline Limits

(Surface Mining module > Surface pull-down menu> Draw Dragline Limits Command)

This draws the workable limits of the dragline on the range diagram cross-section. Pick the existing ground polyline and the location that the center of the machine is sitting. A block of three lines is drawn representing the height, digging depth and horizontal reach of the dragline. There is a vertex placed at the intersection of the ground surface, this is the point to pick for moving the block to the next cut. The vertical line above the vertex is the height, and the line below the vertex is the digging depth. These are the limits to stay within when cutting and placing spoil. There is an option to draw a dragline icon, and also to label the swing angles across the top. This allows you to keep the spoil placement within a certain swing angle. There is a prompt to use a reference point. This could be the crest of the highwall. If one is picked, then it prompts for a distance from that reference point to the center pin of the dragline. Negative values will offset it to the left.



Steps needed to complete Draw Dragline Limits

1. Select draw dragline limits command.
2. Adjust settings in the dialogue to the image shown above accordingly.
3. Command: C:DLIMITS
 Select ground polyline: *pick new existing surface polyline*
 Pick reference point (Enter for None): *Press Enter*
 Dragline distance from reference <0.0>: *-35.84 (distance from center of dragline base to rightmost edge of the dragline main body/machinery house, but any distance closer to the right most edge is acceptable)*
4. Command is finished, check fence diagram for polyline updates.

Cut and Place (Spoil Removal)

(Surface Mining module > Surface pull-down menu> Cut and Place Command)

The next step is the Cut & Place command for the dragline work. In the dialog, set the right highwall angle to the repose angle of the spoil. This will ensure the cut and spoil are the same angle. Also set the spoil placement method to Left Toe.

Cut and Place Settings [X]

Horizontal Scale: 50.0 Vertical Scale: 50.0

Cut Swell Factor: 1.000 Top Spoil Width: 0

Highwall Angle Left: 70.000 Right Side: 36

Repose Angle Left: 36.000 Right Side: 36.000

Sequence Number: 1 Label Areas

Prompt for Snap Track Undisturbed Ground Polyline

Spoil Placement Method

Left Toe Right Toe Top of Pile

Define Cut Area By ...

Points Closed Polyline

Store Step to Sequence File New

Specify Sequence File Name Append/Revise

File: D:\...case studies\Case study 7\Rangediagrammanual.seq

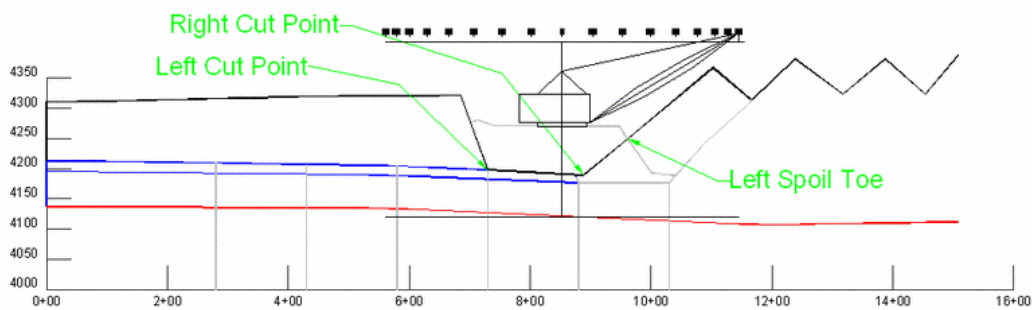
Store Step to Report File New

Specify Report File Name Append/Revise

File: D:\...case studies\Case study 7\Rangediagrammanual.cut

Station: 0.0

OK Cancel Help



After selecting OK, pick the ground and strata (top) polylines. Next, pick the left cut, right cut and spoil location points as shown. It is very helpful to use the Osnap endpoint and intersection to get the exact points. Notice how the right highwall angle and the spoil match for an accurate representation of the slope. Again, there is no need to swell the material as it has been already swollen from previous operations.

Steps needed to complete Cut and Place (Spoil Removal)

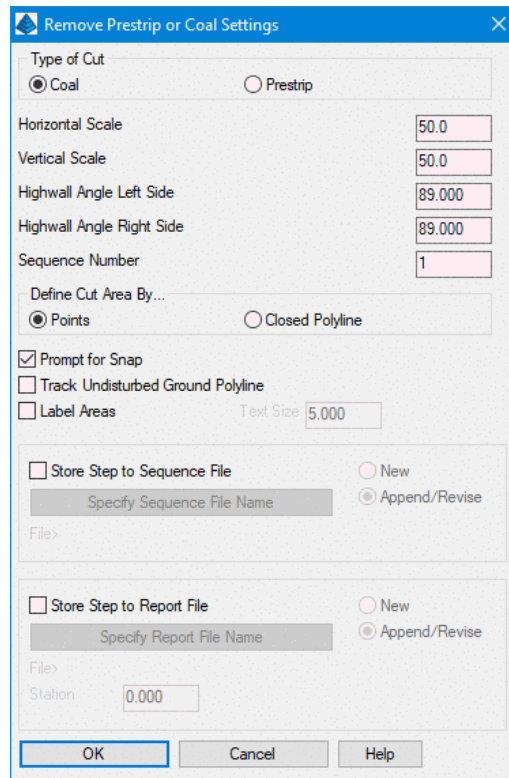
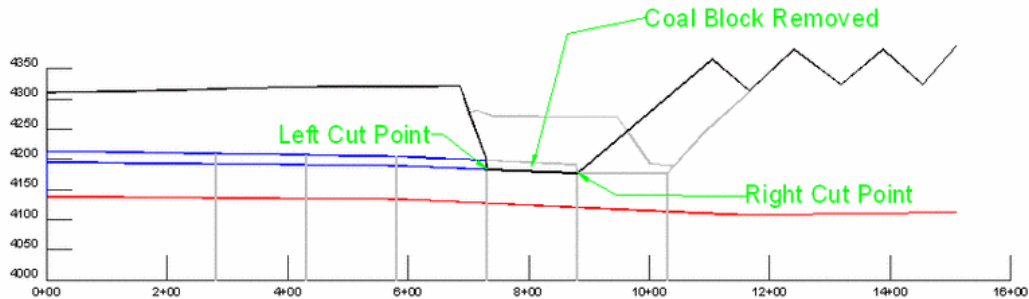
1. Select cut and place command.
2. Adjust settings in the dialogue to the image shown above accordingly.
3. Command: C:RANGE1
 Select current ground polyline:
 Select bottom strata polyline (Enter for none): *pick new existing surface polyline*
 Reference/Limits/<Pick lower Left cut point>: *shown above*
 Reference/Limits/<Pick lower Right cut point>: *shown above*
 Reference/Limits/<Pick spoil position>: *(snap spoil peak to dragline right side 60 degree)*
 Control point to adjust [Left/Right/Spoil/<None>]? *None*
 CUT: 15159.16 (sf)
 FILL: 15159.16 (sf)

4. Command is finished, check fence diagram for polyline updates.

Cut Only (Coal Removal)

(Surface Mining module > Surface pull-down menu > Cut Only (Coal Removal))

The final step in this example sequence is the removal of the coal to prepare the new existing ground surface for the next cut/spoil placement. Like Cut and Place, this routine also allows for differing left and right cut slopes. The strata polyline in this case will be the bottom of coal polyline, which will bring the existing ground polyline down to it.



Steps needed to complete Cut and Place (Coal Removal)

1. Select cut and place coal removal command.
2. Adjust settings in the dialogue to the image shown above accordingly.
3. Command: C:RMCOAL
Select existing grade polyline:
Select strata polyline (Enter for none): *pick new existing surface polyline*
Reference/<Pick lower Left cut point>: *shown above*
Reference/<Pick lower Right cut point>: *shown above*
Control point to adjust [Left/Right/<None>]? *None*

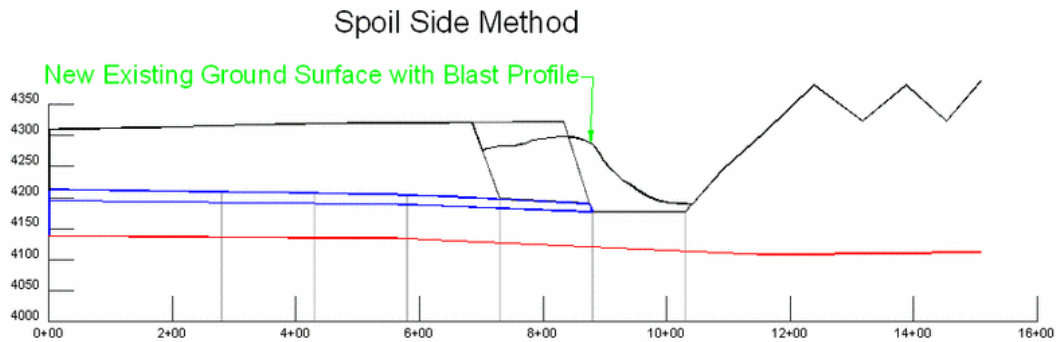
CUT: 1886.68 (sf)

4. Command is finished, check fence diagram for polyline updates.

SPOIL SIDE BENCH EXAMPLE

Cast Blast

The initial cast blast is the same as the extended bench example in the previous pages. Shown is the cross-section after the cast.

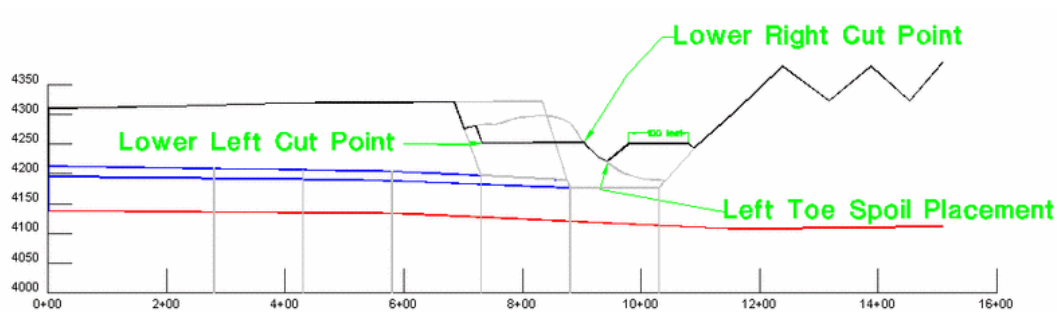


Spoil Side Bench

The Cut & Place routine will be used to create the spoil side bench in this example. In the dialog box, set the Top Spoil Width to the bench width (in this case, 100 feet). Then choose OK and select the ground polyline and cut points as shown. Again, the snap nearest is useful for selecting the right cut point. Do not select a strata polyline, as we are not going down that deep in the cut. To place the spoil, be sure to select either left or right toe for location method, as there is no spoil peak with which to place it. After placing the spoil pile, there is an option to adjust the left and right cut points. Any adjustment of the cut area is automatically reflected in the spoil bench, which allows you to adjust the spoil bench height.

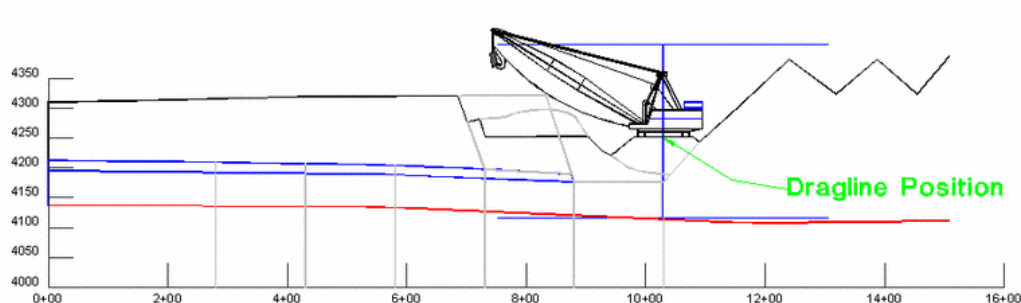
Cut and Place Settings

Horizontal Scale	50.0	Vertical Scale	50.0
Cut Swell Factor	1.000	Top Spoil Width	100
Highwall Angle Left	70.000	Right Side	36
Repose Angle Left	36.000	Right Side	36.000
Sequence Number	1	<input type="checkbox"/> Label Areas	
<input checked="" type="checkbox"/> Prompt for Snap	<input type="checkbox"/> Track Undisturbed Ground Polyline		
Spoil Placement Method			
<input type="radio"/> Left Toe <input type="radio"/> Right Toe <input checked="" type="radio"/> Top of Pile			
Define Cut Area By ...			
<input checked="" type="radio"/> Points <input type="radio"/> Closed Polyline			
<input type="checkbox"/> Store Step to Sequence File <input type="radio"/> New			
Specify Sequence File Name			
<input checked="" type="radio"/> Append/Revise			
File: D:\...case studies\Case study 7\RangeDiagrammanual.seq			
<input type="checkbox"/> Store Step to Report File <input type="radio"/> New			
Specify Report File Name			
<input checked="" type="radio"/> Append/Revise			
File: D:\...case studies\Case study 7\RangeDiagrammanual.cut			
Station: 0.0			
<input type="button" value="OK"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/>			



Draw Dragline Limits

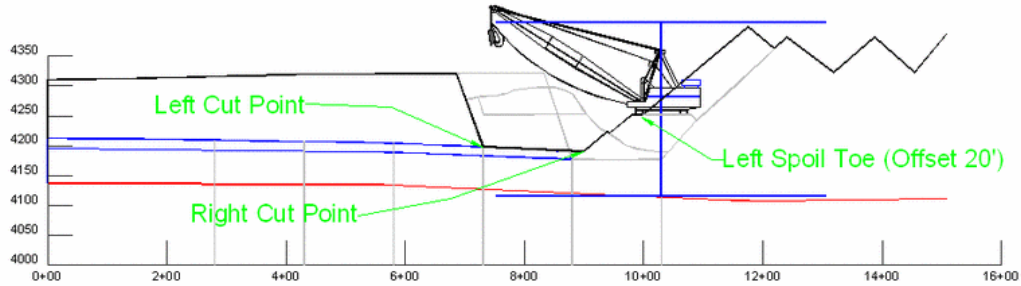
The next step is to show the dragline limits when placed on the spoil side bench. Use the same procedure as in the extended bench example. Another option is to create a block drawing of a dragline and insert it into the drawing to represent the correct dimensions. Shown below is a block of a dragline to scale, inserted into the above range diagram.



Cut & Place

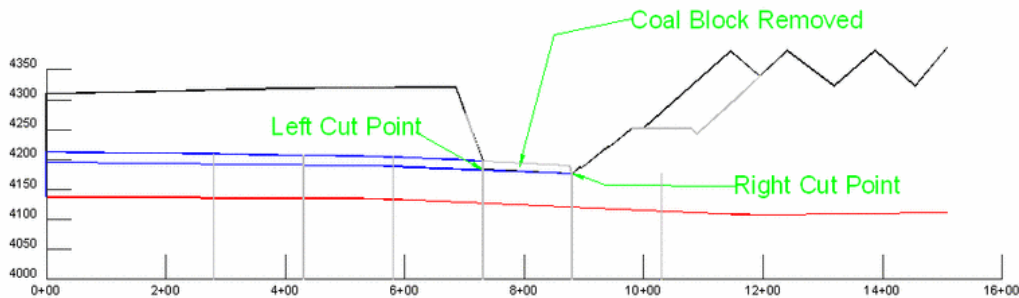
The next step is the Cut & Place routine for the dragline pass. In the dialog, set the right highwall angle to the repose angle and set the spoil placement method to Left Toe. Also set the Top Spoil Width back to zero. Then click OK and select the ground and strata polylines. Next, pick the left cut, right cut and spoil pile points as shown. Notice the right cut point is to the right of the coal block to ensure it is uncovered and minimize rehandling. You can use

the Osnap-endpoints to get the exact points. To start the spoil pile 20 feet from the spoil side bench, use the distance option and pick the edge of the spoil side bench as the reference. Type in a positive value to offset to the right and negative to offset to the left.



Cut Only (Coal Removal)

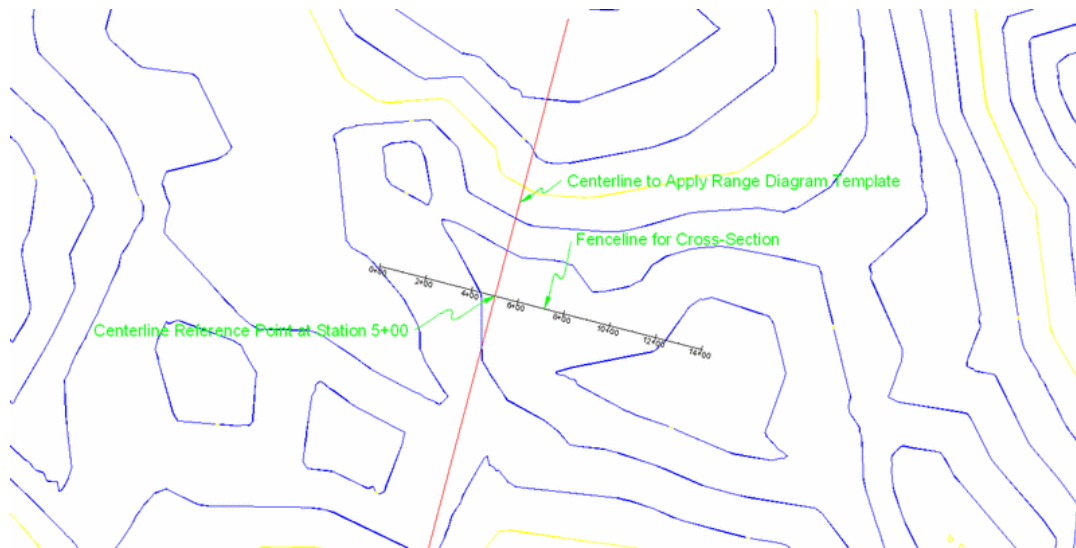
As in the previous example, the coal removal is the last step to prepare the existing ground surface for the next cut. The right side cut angle is again set to the same angle as the spoil repose to simulate the spoil down to the bottom of the removed coal block.



Process Dragline Sequence

(Surface Mining module > Surface pull-down menu > Process Dragline Sequence Command)

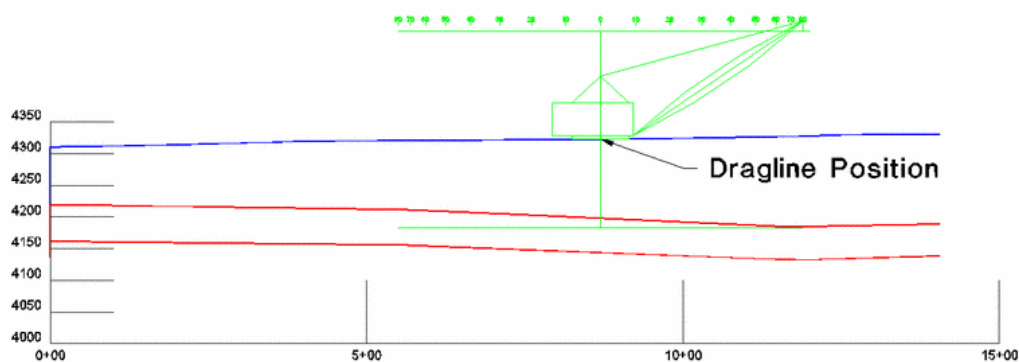
One thing not mentioned in the previous examples is the saving of each step to a dragline sequence file. The option to store each step in a sequence file is set at the bottom of the dialog boxes. Typically the first step is saved to a New file as sequence step #1, and each subsequent step is appended to that file as #2 and up. The next example will show a simplified two-cut Cut and Place on a fence diagram and then processed with a centerline to generate 3D polylines drawn in plan view and sections. The first step is drawing the fence line and intersecting it with the centerline for laying out the cuts/pits as shown in the diagram below. This intersection will be known in the commands as the centerline reference point. Then draw the fence diagram with the hatching turned off and draw as single polylines turned on. This will provide an existing ground surface with the strata lines below it. It is usually better if there is no exaggeration in the cross-section. Since this fence diagram is already available in the drawing file, use the first fence diagram under Processing Dragline Sequence on the left of the drawing.



New dragline dimensions

Dragline specifications			
Make/Model Name			BE AT KELLERMA
Model	1300-W	Make	BE FROM SPEC SHEET
Required sizes		Optional sizes	
Bucket size	40.0 c.y.	House width	ft
Max. Dump Height	170.0 ft	House length	ft
Max. Reach	300.0 ft	Max Speed	0.15 mph
Max. Depth	130.0 ft	Swing, rpm	
		Operational Cost, phr	
OK		Cancel	

Processing Dragline Sequence



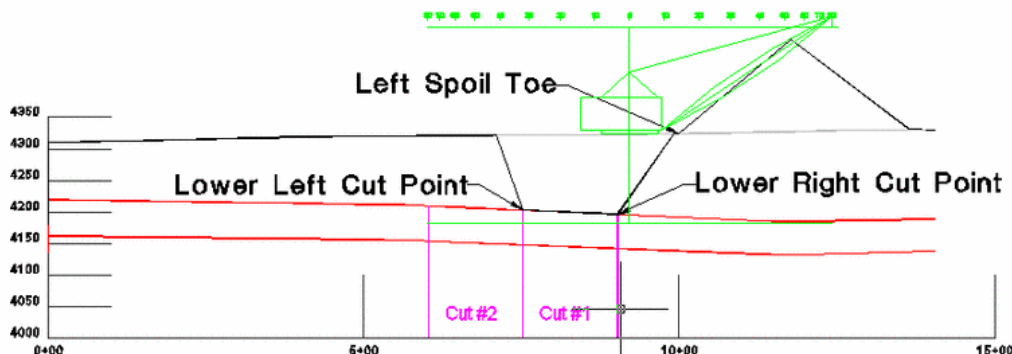
Cut & Place

After drawing the dragline limits, the Cut & Place settings need to be set as shown to the right. A 20% swell has been set, and it will be stored to a new step file as shown with sequence number one. This will be a simple box cut for the first sequence. Step two will be the coal removal, and step three will be the next cut and spoil placement. After step three, the sequence file will be processed and turned into 3D polylines and a section file (*.sct). The first step is shown below:

Cut and Place Settings

Horizontal Scale	50.0	Vertical Scale	50.0
Cut Swell Factor	1.2	Top Spoil Width	0.0
Highwall Angle Left	70.000	Right Side	55
Repose Angle Left	36.000	Right Side	36.000
Sequence Number	1	<input type="checkbox"/> Label Areas	
<input checked="" type="checkbox"/> Prompt for Snap	<input type="checkbox"/> Track Undisturbed Ground Polyline		
Spoil Placement Method			
<input checked="" type="radio"/> Left Toe <input type="radio"/> Right Toe <input type="radio"/> Top of Pile			
Define Cut Area By ...			
<input checked="" type="radio"/> Points <input type="radio"/> Closed Polyline			
<input checked="" type="checkbox"/> Store Step to Sequence File <input checked="" type="radio"/> New			
Specify Sequence File Name <input type="radio"/> Append/Revise			
File: D:\...case studies\Case study 7\Rangediagrammanual.seq			
<input type="checkbox"/> Store Step to Report File <input type="radio"/> New			
Specify Report File Name <input checked="" type="radio"/> Append/Revise			
File: D:\...case studies\Case study 7\Rangediagrammanual.cut			
Station: 0.0			
OK		Cancel	
Help			

The first prompt after selecting the existing ground surface is to pick the centerline reference point. In this example, it is station 5+00 and it should be "snapped" to at the intersection of the horizontal grid line and the 5+00 tick mark. After selecting the top of strata polyline, you are prompted to enter the name of the strata. This name will be used later in prompting to select that surface grid file. Select the left and right cut points and the spoil location. Notice the two cuts were drawn in by hand to guide the user in the correct location of the cuts. This is sequence step number one.



Cut Only (Coal Removal)

Step number two will be the removal of the coal block. The cut angle will be set to 89 degrees on each side to simulate a near vertical face of coal after it is removed. Try to stay away from pure 90 degree slopes, as this can create difficulties for the program. The strata line will be the bottom of coal this time, and you will be prompted to specify a name for that surface. (BOC for bottom of coal works) You will also be asked to snap to the centerline reference point (5+00).

Remove Prestrip or Coal Settings

Type of Cut
 Coal Prestrip

Horizontal Scale: 50.0
 Vertical Scale: 50.0
 Highwall Angle Left Side: 89
 Highwall Angle Right Side: 89
 Sequence Number: 2

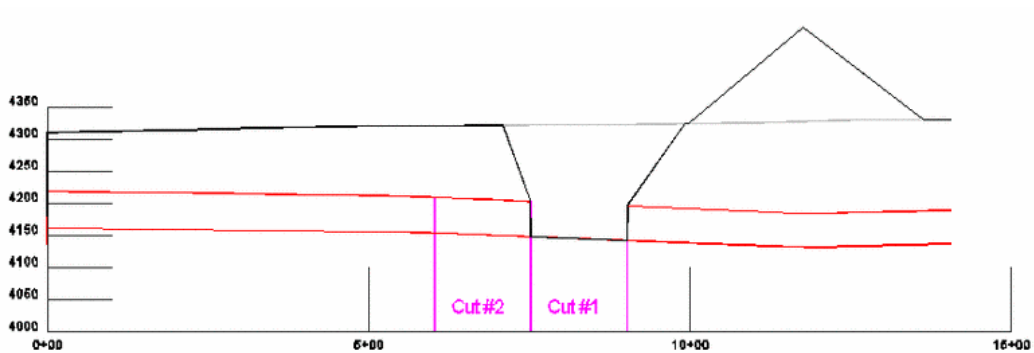
Define Cut Area By...
 Points Closed Polyline

Prompt for Snap
 Track Undisturbed Ground Polyline
 Label Areas Text Size: 5,000

Store Step to Sequence File New
 Append/Revise
 File>

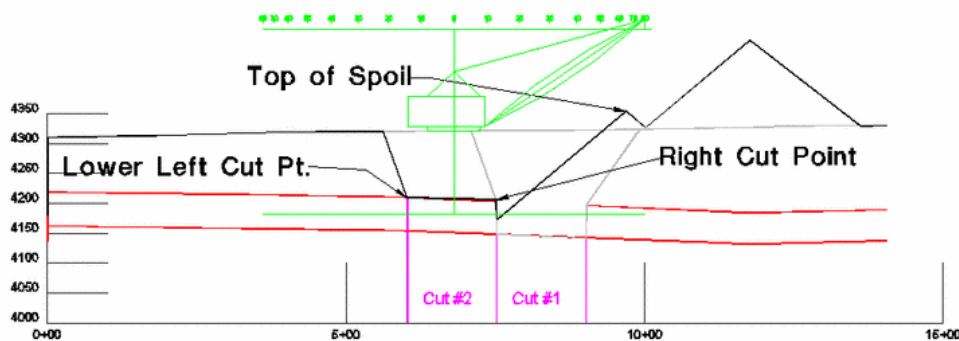
Store Step to Report File New
 Append/Revise
 File>
 Station: 0.000

OK Cancel Help



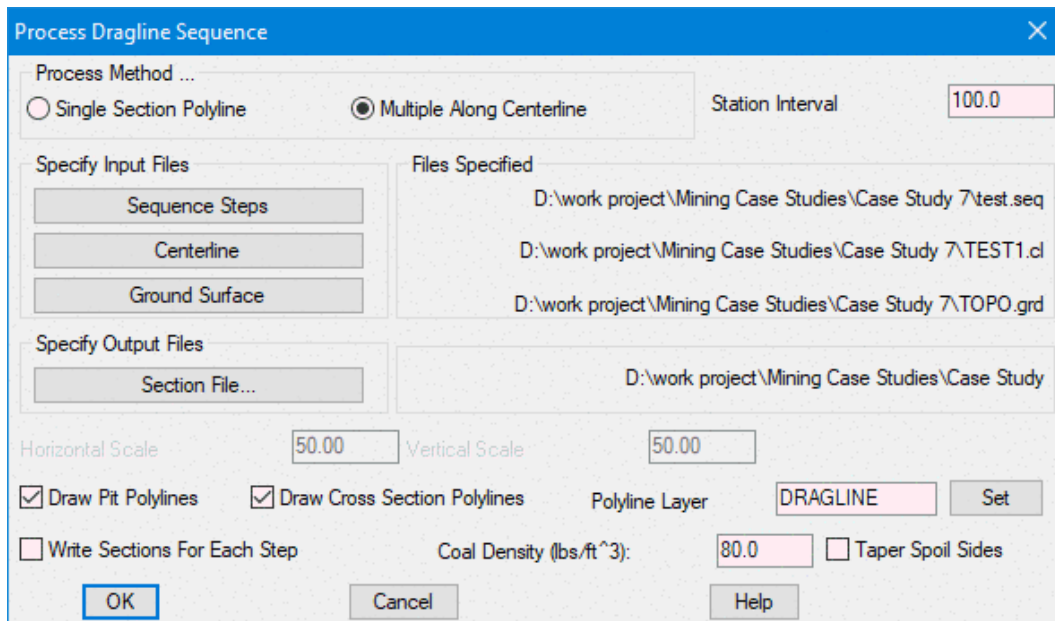
Cut & Place

The third and final operation in the sequence will be the second cut with spoil placement into the first cut. Everything is about the same except for the sequence number needs to be set to three, and it should change automatically. Shown below is the final result, sort of a template that will be applied now to our centerline (*.CL file) utilizing the surface grids and create a new surface for possible PMT application.



Process Dragline Sequence

If it has not already been done, use the routine Polyline to Centerline File to create the *.CL necessary for the processing of the sequences. The following window will appear, and needs to have at least the first three files set. It will also create an output section file for later manipulation. The Taper Spoil Sides will taper the ends of the spoil gradually down to the existing ground. The draw pit polylines will create new pit/cut polylines to run for scheduling. After selecting OK, you will be prompted for the grids representing the strata named in the previous routines.



Steps needed to create a Centerline file

1. Select polyline to centerline command. (Surface Mining module > Surface pull-down menu> Polyline to Centerline Command)

2. Command: C:CLPLINE

Centerline File to Write (*creating a new file, enter file name*)

Polyline should have been drawn in direction of increasing stations.

Select polyline that represents centerline:

Centerline station [Reverse/Ending/<Beginning: 0+00>]: *Reverse (centerline arrow direction is from north to south, if the direction is not correct press Reverse. otherwise press enter to skip this step)*

Centerline station [Reverse/Ending/<Beginning: 0+00>]: *(Press enter to finish this step)*

Station North(y) East(x) Description (*centerline summary*)

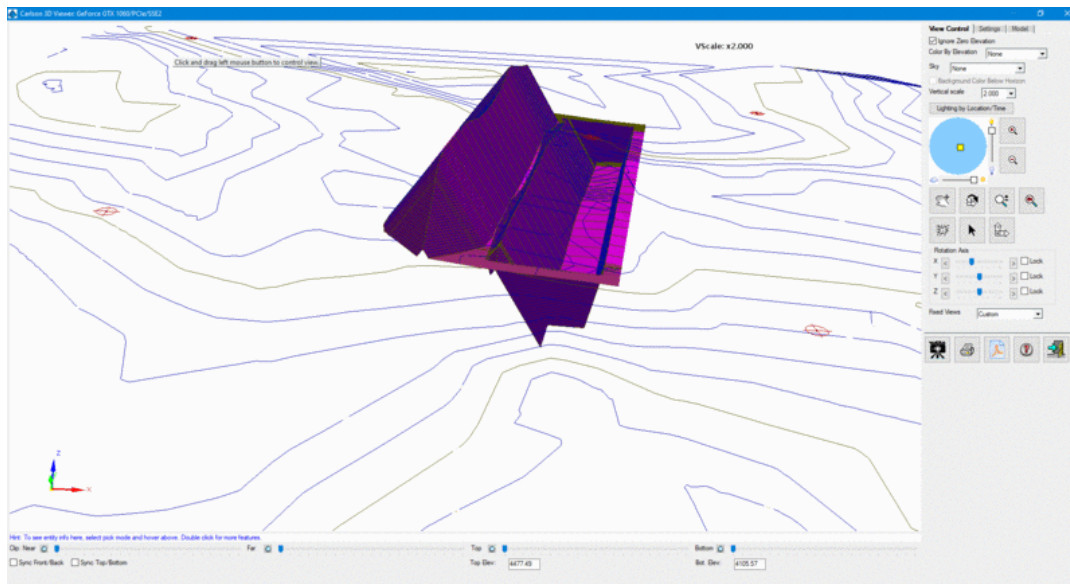
0.000 935218.592 1003501.740 LI

2364.899 932932.065 1002897.967 LI (2364.899 is the total distance of the centerline)

Press ENTER to continue.

3. Centerline file created

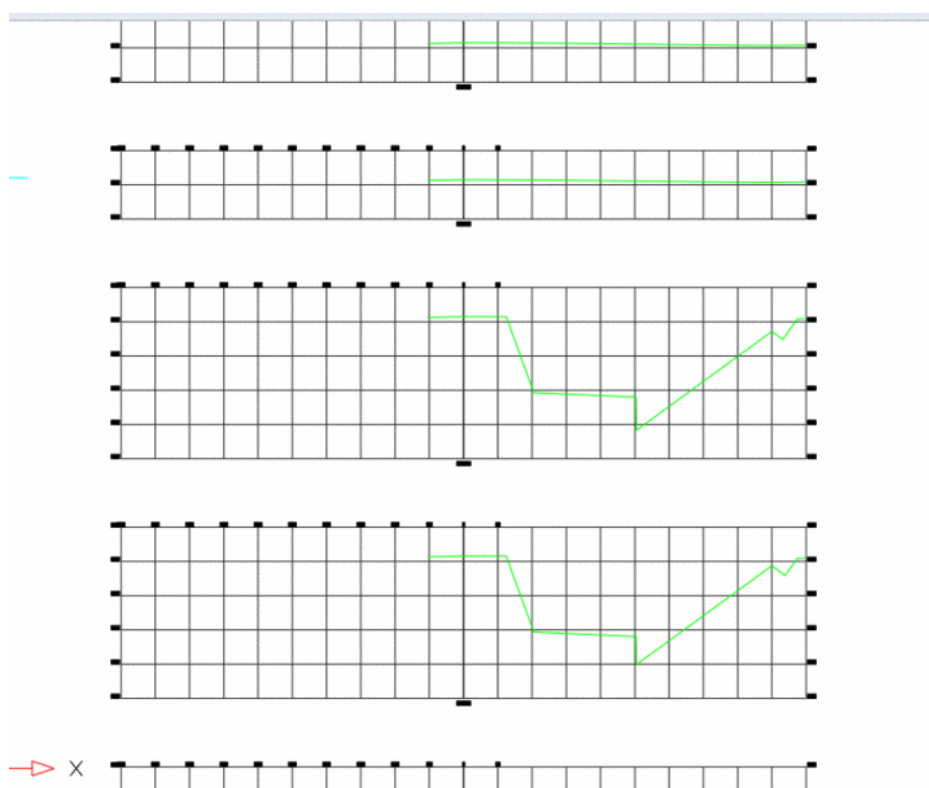
Shown below is 3D view of the cross section. For visual clarification, the 3D polylines were also contoured to begin building the new surface. Notice the pit is only as long as my centerline was drawn. The top image is a rendered image of Triangle Faces that were drawn.



Draw Section File

(Civil module > Section pull-down menu > Draw Section file Command)

Another step is to draw the section file created in the processing of the sequence. Shown is a partial series of sections created with the Carlson Range Diagram routines.



Case Study #9: Surface Timing With Benches

This tutorial shows how to run Surface Equipment Timing through a series of benches in pits. This example starts with a pit layout mineplan and structure grids. It is assumed that you already know those operations. Then Design Bench Pit is run on the pits, creating final pit grids and a Grid Sequence File. Next the quantities for each bench will be calculated and stored in the pit polylines. Then the sequence of mining the benches and pits is assigned with Surface Equipment Timing.

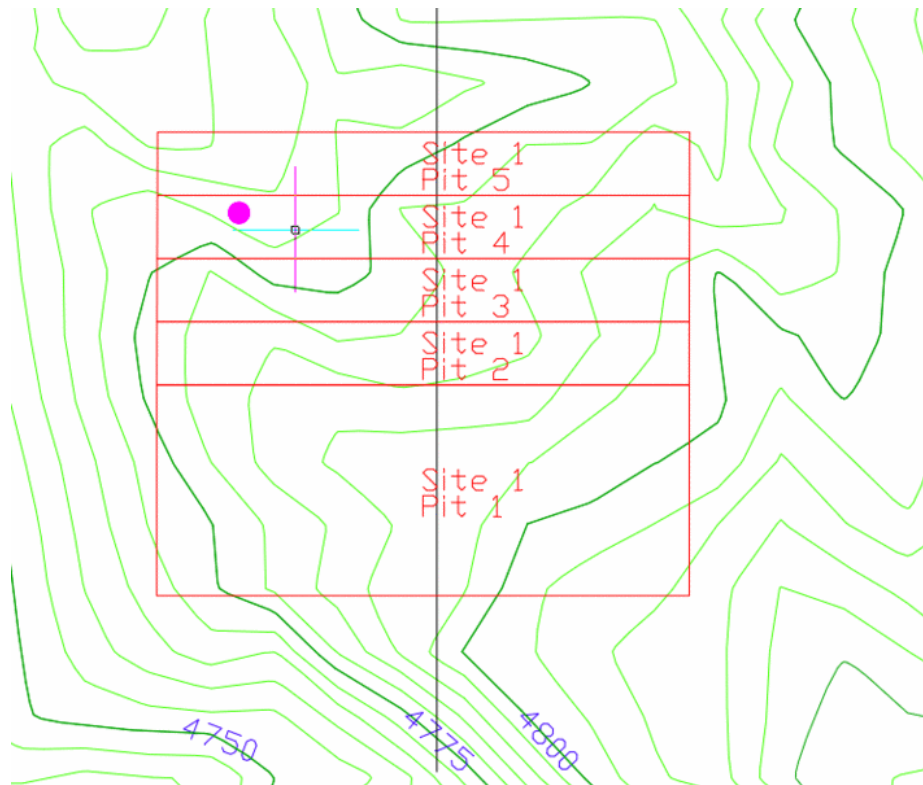
Creating Pit Polylines

(Surface Mining module > Boundary pull-down menu > Pit Layout By Advance Command)

The first step is to sub-divide the overall pit perimeter into smaller pits. In this example, the larger rectangular mine perimeter is subdivided into 5 smaller pits, named Pit-1 through Pit-5. These will be draped onto the bottom of the pit and benches will go up and out from them. The command Pit Layout By Advance was used to lay them out.

Steps needed to create pit polylines

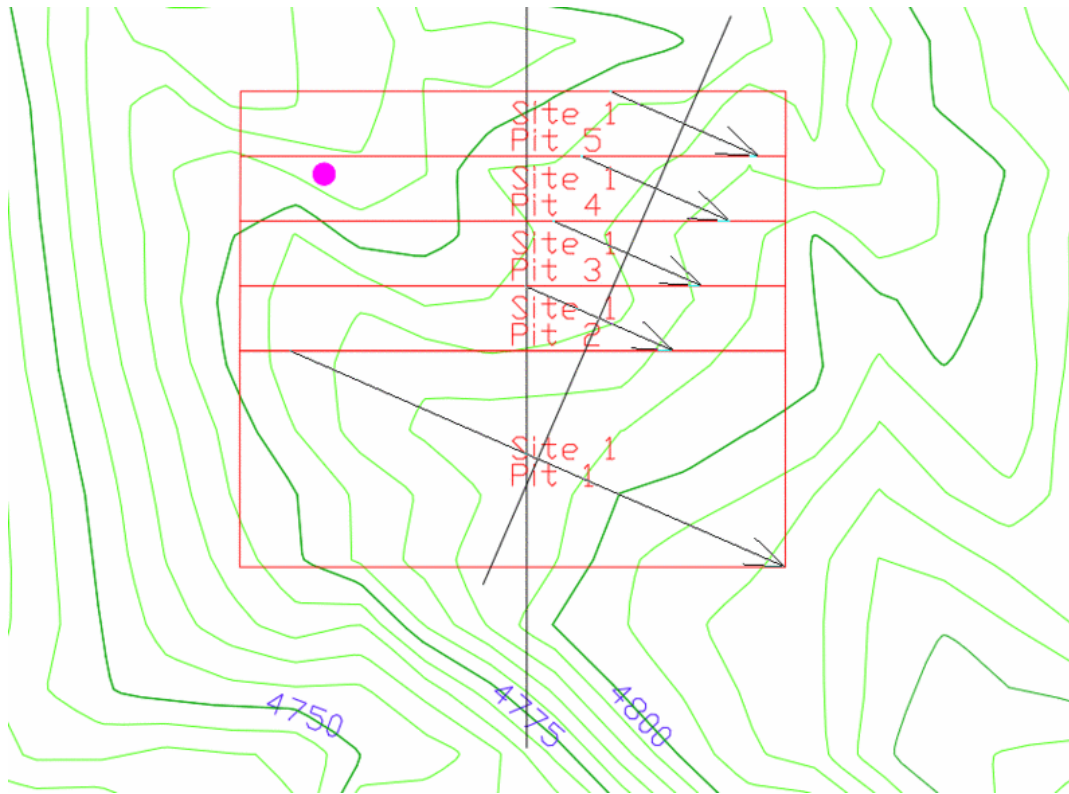
1. Select Pit Layout by Advance command.
2. Command: C:CUTPITADV
Select all pit polylines. *select one large rectangle that will be including pit 2 to pit 5.*
Entities in set: 1
Select all direction polylines. *(In this command, direction polyline indicates direction of the pit advance in an area. So choose black vertical line as directional polyline)*
Entities in set: 1
3. Command is finished, New pit polyline has been created.



Assigning Pit Directions

(Surface Mining module > Boundary pull-down menu > Pit direction Tab > Assign Direction Command)

In order to schedule the equipment through the pits, the direction of mining needs to be assigned to each pit. In this case, the direction of mining for each pit will be from left to right. First draw a direction polyline through the pits from bottom to top. Then run the Assign Directions command and use the Sequence method with the LL option.



Steps needed to assign pit direction

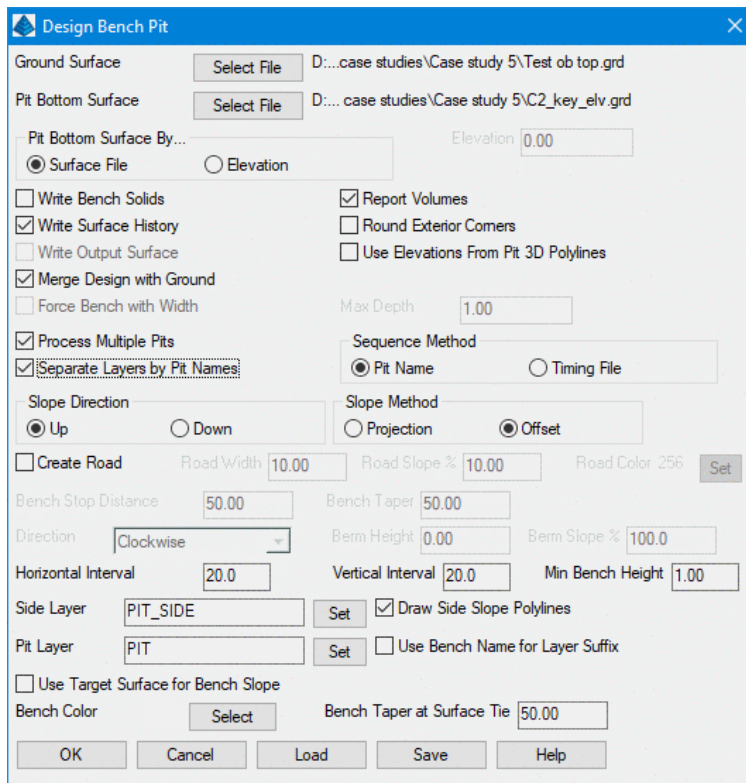
1. Select Assign Direction Command.
2. Command: C:ASSIGN_DIR
 Assign direction to whole pit or to a specific bench [*<Whole>/Bench*]? *Whole*
 Assign direction method [*<Auto>/Text/Sequence/Polyline/Bearing/AZimuth/Gon*]? *Sequence*
 Select pit polylines to have direction assigned to. *Select all the pits*
 Entities in set: 5
 Select a direction polyline:
 Assign direction in which sequence [*<LL>/LR/RL/RR*]? *LL*
3. Command is finished, directional polyline has assigned.

The pits now have direction assigned from west to east.

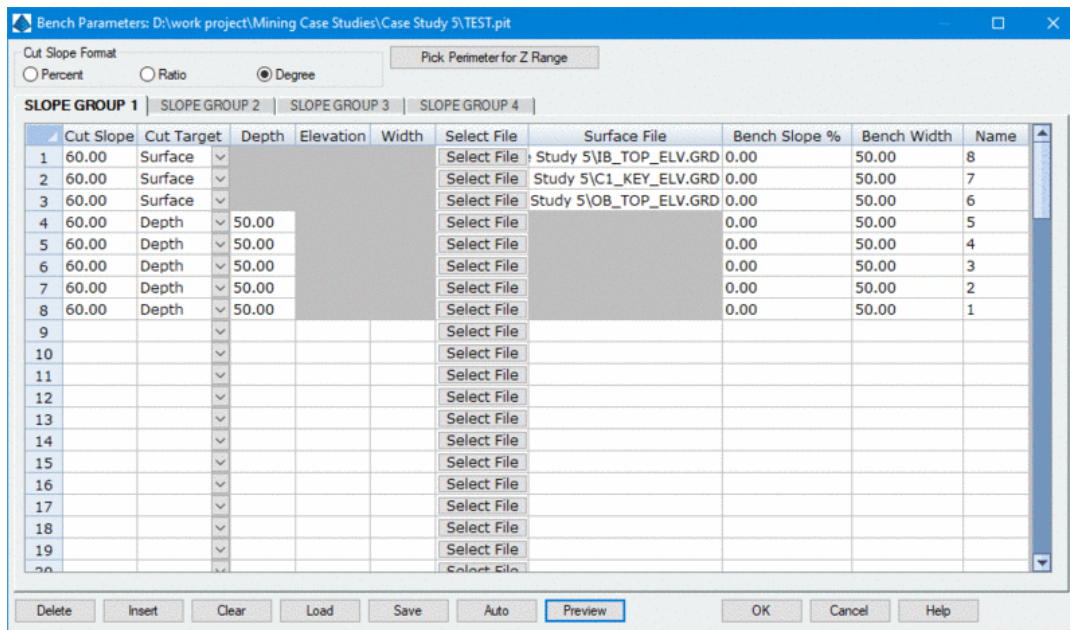
Design Bench Pit

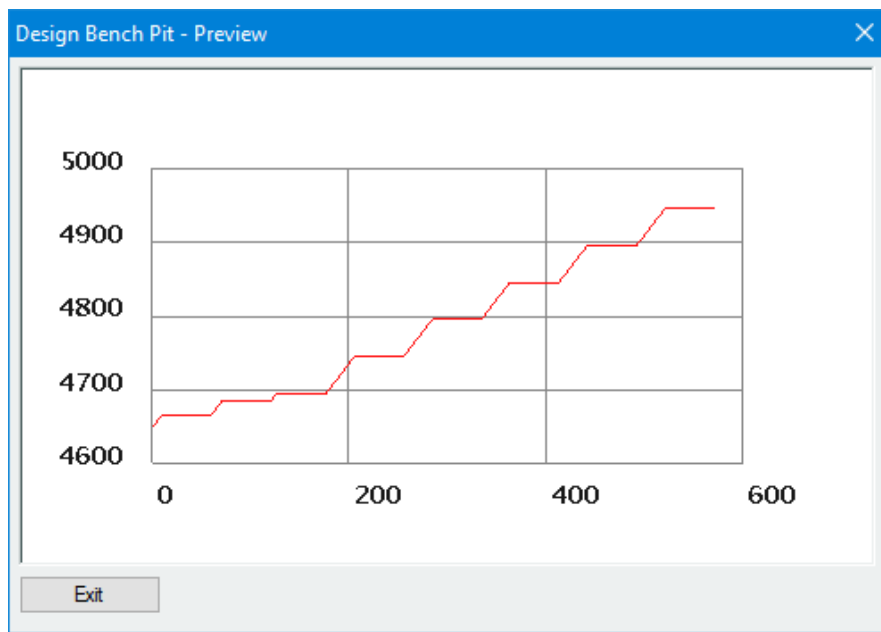
(Surface Mining module > Surface pull-down menu > Design bench pit Command)

Using the pit perimeters as the inclusion areas, the program will draw the 3D pit shells, output grids, and calculate volumes for each pit and bench. Before running Design Bench Pit, you need the pit bottom perimeter polylines and grid files for the ground surface and benches. In the first dialog, specify the ground surface grid and the bottom elevation grid for the last bench. Also be sure to enable the options for Process Multiple Pits, Separate Layers by Pit Names, and Write Surface History. A grid file for each pit and each bench will be created and automatically organized in the Grid History File (GSQ).

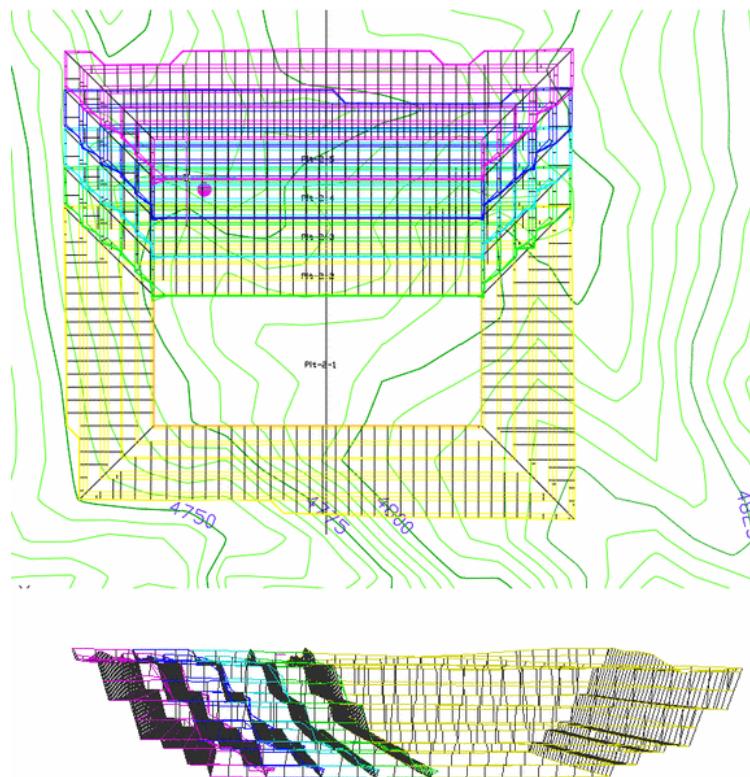


In the next dialog, you specify the side and bench slopes. The height of the side slopes can be specified by either a depth number or a grid file. In this case we will use grid files as well as by depth. Four different sets of slopes can be applied to different sides of the pit. For this example, the same slopes will be used on all sides. You can use up to 100 benches.

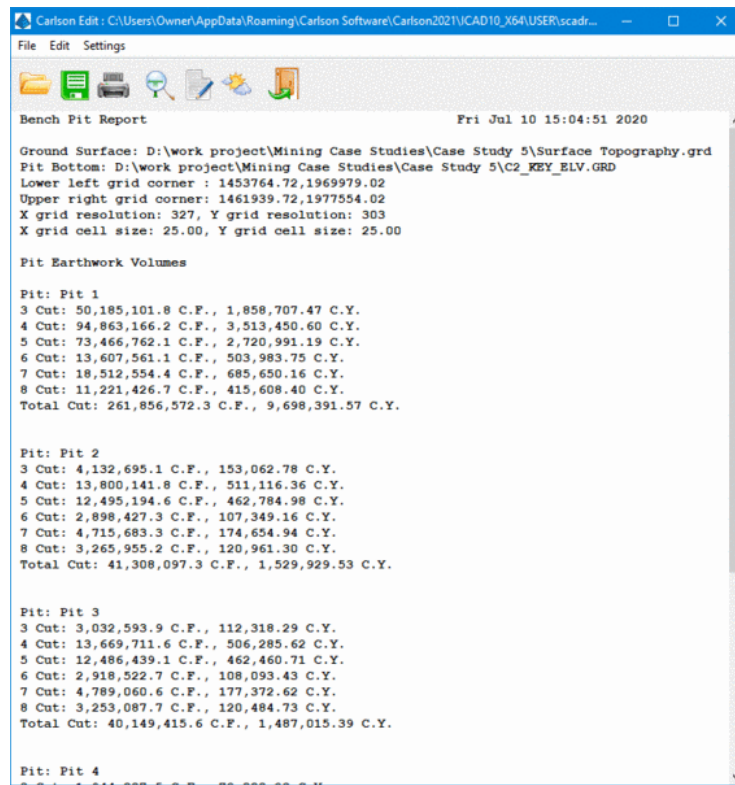




The output will look as such. Each pit/pushpack is in its own layer and colored differently. In the above view, the pits are laid down on the side, to show the slope to each grid.



Each pit and bench quantity is reported in the basic text report.



Steps needed to design bench pit

1. Select design bench pit command.
2. Follow design bench pit dialogue and bench parameters dialogue that are shown above. Click "OK" to finish each command
3. Save Surface History file
4. Command: minepit2
Select the site/pit perimeter polylines. (select all 5 pit perimeters)
Entities in set: 5
Preparing bench grids ...
Calculating bench pit ...
Processing edge 9059, intersections found 3622
Calculating grid elevations
Writing grid file: D:\work project\Mining Case Studies\Case Study 5\pit.grd
... program is writing grid files for each pit
...
...
5. Report dialogue will show up and give summary in volume for all the cuts.
6. Command is finished, New pit polyline has been created.

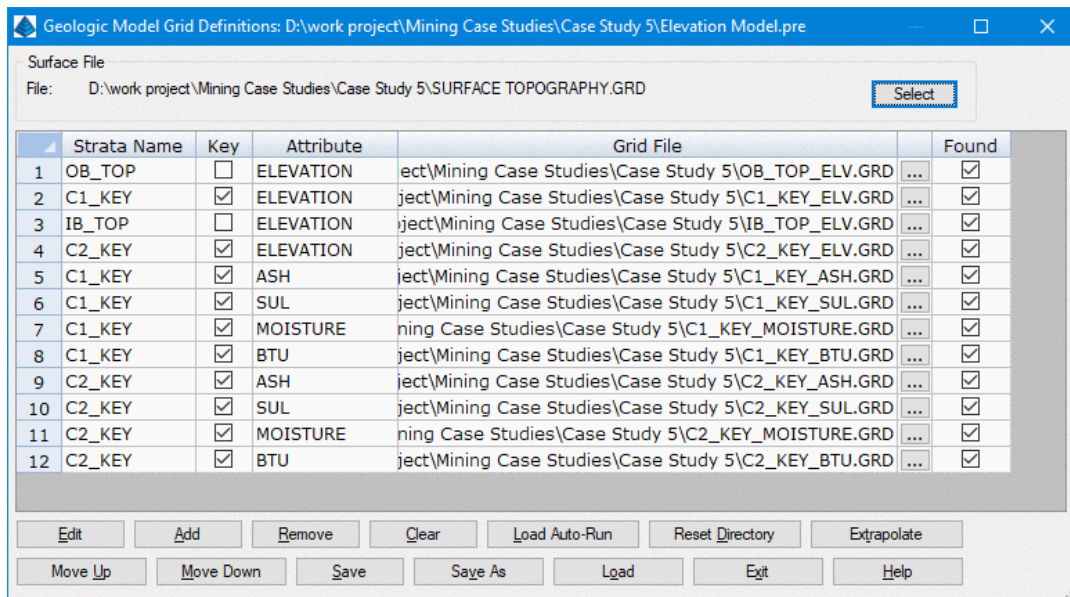
Assigning Bench Quantities

(Geology module > StrataCalc pull-down menu > Define Geologic model Command)

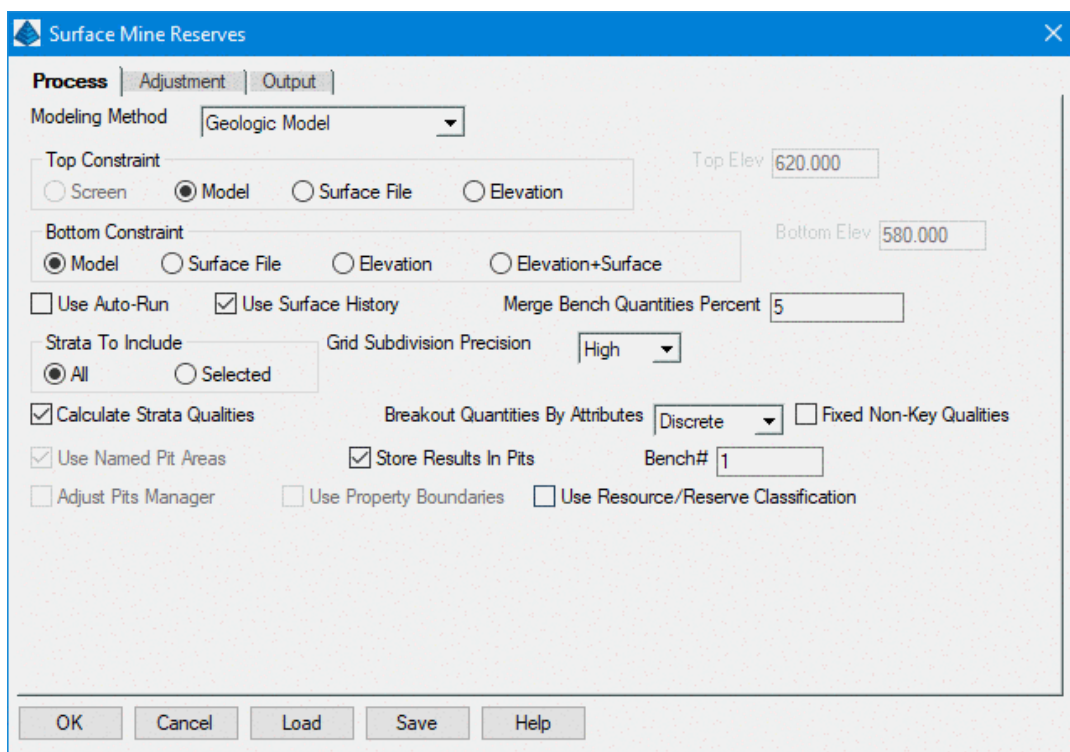
(Geology module > StrataCalc pull-down menu > Surface Mine Reserves command Command)

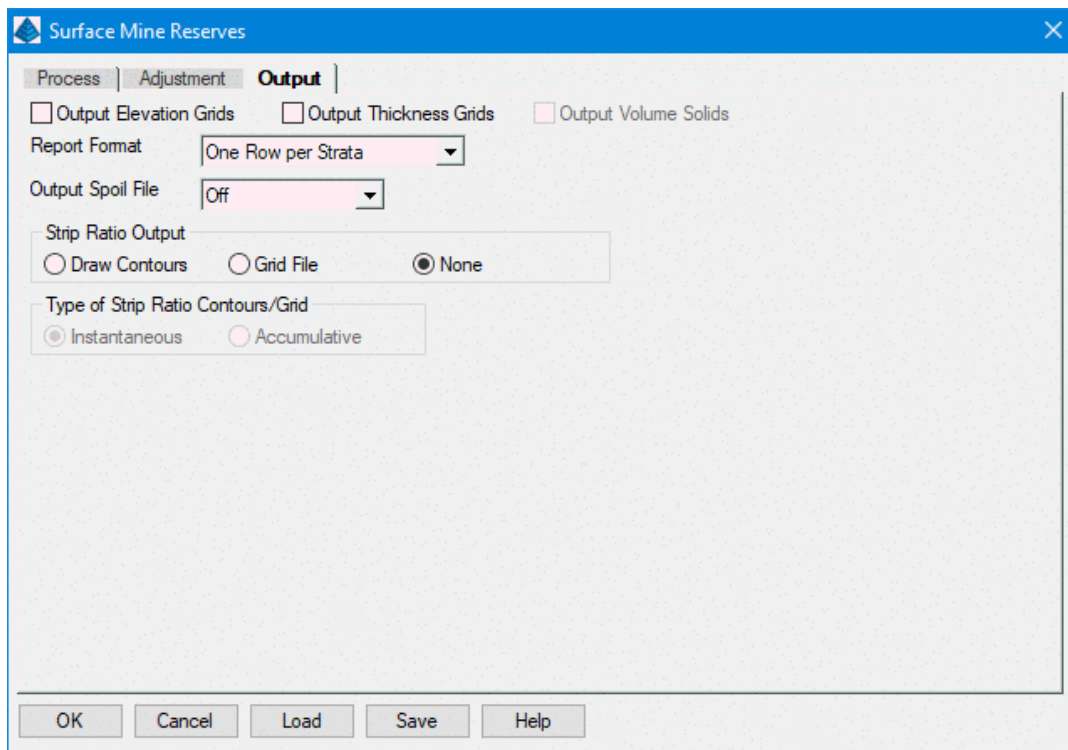
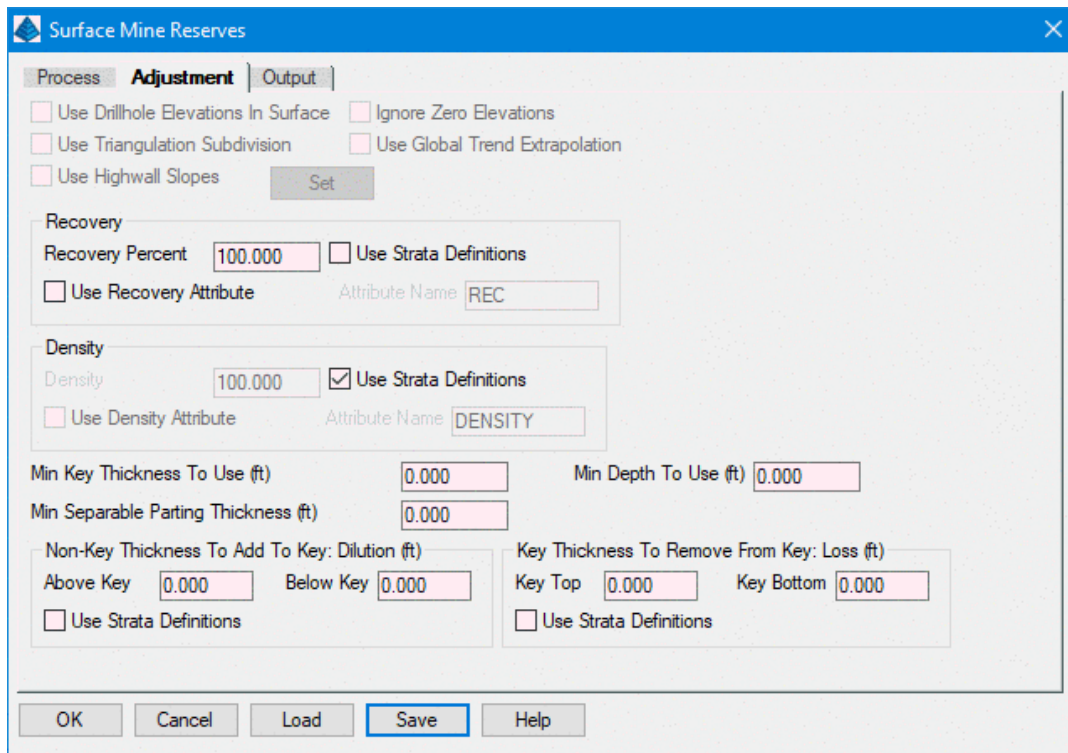
The next step is to store quantities in each pit for each bench using the Surface Mine Reserves command. Before running this command, you need to have a Pre-Calc Grid file (.PRE) in addition to the GSQ history file. This is

what determines the bench quantities and automatically assigns them to the benches.. Use the Define Geologic model command to specify the grids as shown below.



Then run Surface Mine Reserves. The dialog shown below appears. Set the Modeling Method to Geologic Model. Since we are using the Surface History File, all five benches will be assigned automatically. Turn on all the options you see here.





After clicking OK on the dialog, select the Geologic Model to use for the calculation. Next the program prompts for the Grid History File created earlier by the Design Bench Pit routine. The Merge Bench Quantities Percent applies here. If there is some grid "bleeding" where one seam bleeds into another bench due to the grid cell size, then this option will put that material back into the correct bench, as long as the amount is not less than this percentage.

Now the program will calculate the quantities of all the benches for all the pits. A report formatter allows you to optionally print or display the results. The quantities are stored in the pit polylines for scheduling. If you are not

using the GSQ file, then you will have to run this for each bench number, with all the same options except set the Bench # to 2 and choose only Bench 2 from the strata list (Selected Strata turned on). Then repeat this command a third time using bench 3, etc. That is what is nice about the GSQ file, it does them all at once, even calculating and assigning the layback slope volume accordingly. Shown next is the pit by pit and bench by bench volume report.

Surface Mine Reserves Mon Aug 3 13:33:40 2020

Geologic Model: D:\work project\Mining Case Studies\Case Study 5\Elevation Model.pre
 Surface History: D:\work project\Mining Case Studies\Case Study 5\test.gsq
 Key strata recovery: 100.00%
 Minimum depth to use: 0.00 ft
 Minimum key thickness to use: 0.00 ft
 Minimum separable parting thickness to use: 0.00 ft
 Non-Key thickness Above key to add to key: 0.00 ft
 Non-Key thickness Below key to add to key: 0.00 ft
 Key top thickness to remove from key: 0.00 ft
 Key bottom thickness to remove from key: 0.00 ft

Pit: Pit 1

Strata Bench#	KEY Tons	Non-key C.Y.	Thick	Area acres
OB_TOP 1		1,853,465.7	122.12	9.408
OB_TOP 2		3,512,296.4	100.48	21.667
OB_TOP 3		2,721,361.3	59.48	28.359
C1_KEY 4	544,600.0		11.90	26.260
IB_TOP 5		685,998.3	20.75	20.493
C2_KEY 6	447,783.7		17.06	15.066
Subtotal for Pit: Pit 1				
	992,383.7	8,773,121.7	331.78	

Pit: Pit 2

Strata Bench#	KEY Tons	Non-key C.Y.	Thick	Area acres
OB_TOP 1		152,097.9	84.96	1.110
OB_TOP 2		511,067.2	78.62	4.029
OB_TOP 3		462,520.3	57.32	5.002
C1_KEY 4	115,925.0		11.84	5.619
IB_TOP 5		174,483.8	20.94	5.165
C2_KEY 6	130,185.2		16.80	4.448
Subtotal for Pit: Pit 2				
	246,110.3	1,300,169.2	270.47	

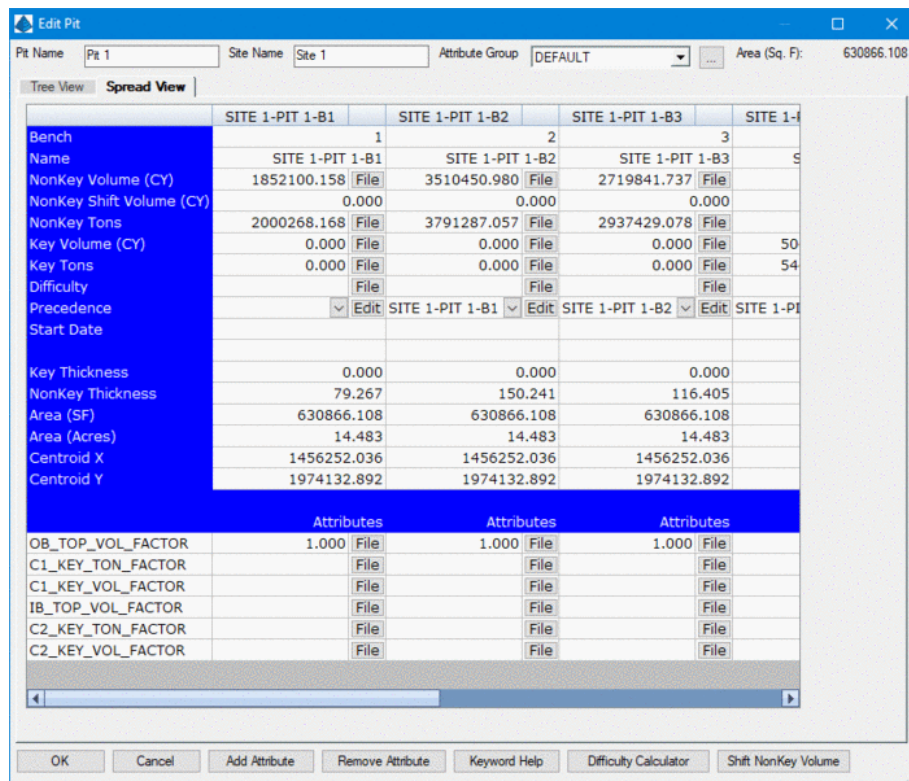
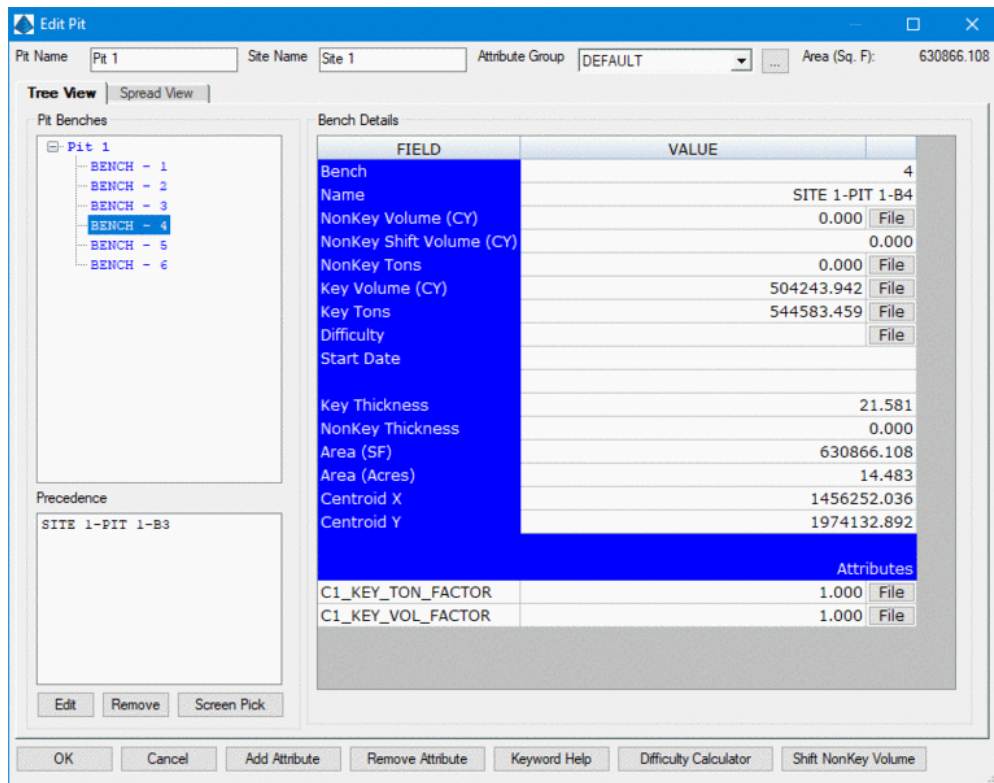
Pit: Pit 3

Strata Bench#	KEY Tons	Non-key C.Y.	Thick	Area acres
OB_TOP 1		111,381.6	87.04	0.793
OB_TOP 2		505,909.8	82.07	3.821

These values are now stored in the pits and can be verified with Edit Pit under Boundary. Click inside any one of the pits to see the values. All benches can be accessed by selecting the bench number in the tree view on the left of the dialog or by selecting the Spread View tab to view data for all benched at a time.

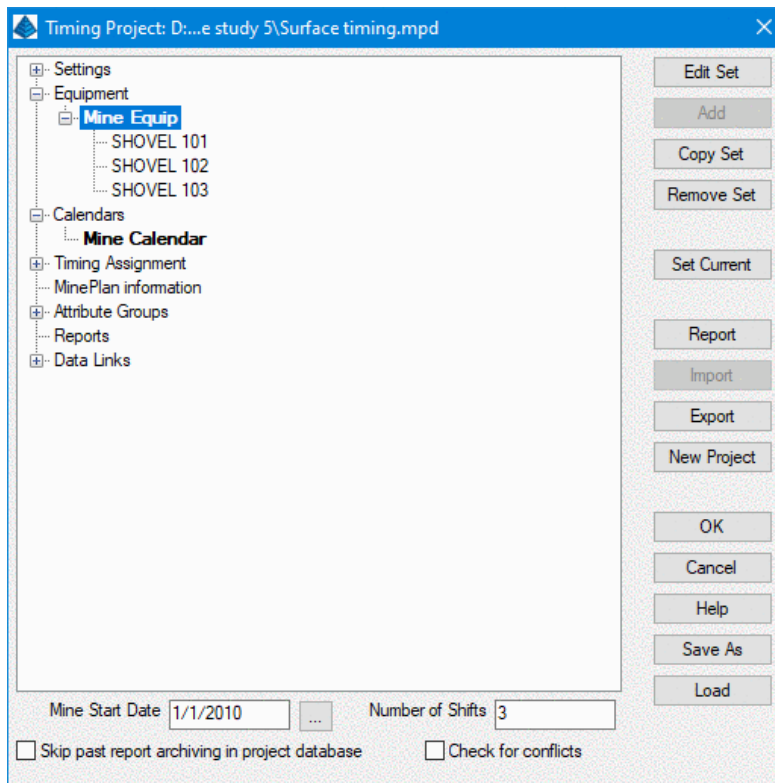
Note: Surface mine reserve command will not completely overwrite previous run. In case of recreating new bench pit, use the Clear Bench Qualities Command to clear all the information that is stored in the pit. (Surface Mining module > Boundary pull-down menu > Pit Timing Qualities Tab > Clear Bench Qualities Command)

(Surface Mining module > Boundary pull-down menu > Edit pit Command)



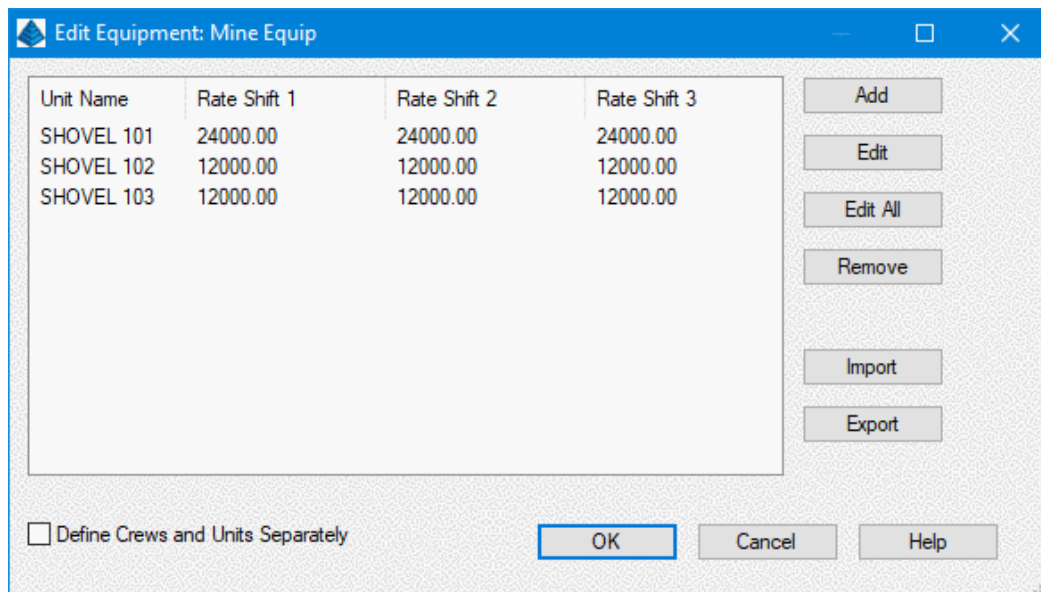
Surface Timing Project Manager (Surface Mining module > Reserve/Timing pull-down menu > Timing Project Manager Command)

The Timing Project Manager can be accessed under the Reserves/Timing pull-down menu. It allows user to define Equipment, Timing Calendar, and Pit Attributes as well as other essential parameters to be used in the timing.



Define Equipment

Select the Edit Set in Project Manager to create an equipment definition to mine the pits. This example will use three shovels. This command shows a list of existing equipment definitions. Choose the Add button and fill out the dialog as shown below for a three shift operation for all shovels. Shovel1 mines 24000 CY/shift and Shovel 2 and Shovel3 mines 12000 CY/Shift.



Equipment Production Rate

Name:

Unit of Production
 Tons Distance Cubic yards Linear ft of Advance

	Shift 1	Shift 2	Shift 3	Shift 4
Advance Rate/shift	<input type="text" value="12000.00"/>	<input type="text" value="12000.00"/>	<input type="text" value="12000.00"/>	<input type="text" value="600.00"/>
Retreat Rate/shift	<input type="text" value="600.00"/>	<input type="text" value="600.00"/>	<input type="text" value="600.00"/>	<input type="text" value="600.00"/>
Hours/shift	<input type="text" value="8.00"/>	<input type="text" value="8.00"/>	<input type="text" value="8.00"/>	<input type="text" value="8.00"/>
Advance Rate/hour	<input type="text" value="1500.00"/>	<input type="text" value="1500.00"/>	<input type="text" value="1500.00"/>	<input type="text" value="75.00"/>
Retreat Rate/hour	<input type="text" value="75.00"/>	<input type="text" value="75.00"/>	<input type="text" value="75.00"/>	<input type="text" value="75.00"/>
Availability	<input type="text" value="1.000"/>	<input type="text" value="1.000"/>	<input type="text" value="1.000"/>	<input type="text" value="1.000"/>
Advance Min. Height	<input type="text"/>		Advance Max. Height <input type="text"/>	
Retreat Min. Height	<input type="text"/>		Retreat Max. Height <input type="text"/>	
Main Maintenance Freq.	<input type="text"/>		Delay <input type="text"/> shifts	
Routine Maintenance	<input type="text"/>		Delay <input type="text"/> shifts	
Extraction factor	<input type="text" value="1.0000"/>		Cost per hour <input type="text" value="0.00"/>	

Define Calendar

The next step is to define the calendar. By default, both units are working every day, every shift. It is your job to take off holidays, weekends, and maintenance down time. Select a date and press apply to schedule downtime.

Editing Calendar: Mine Calendar

Date Selection: Current Date 12/25/2020

Jump to Year:

Jan Feb Mar Apr May Jun
 Jul Aug Sep Oct Nov **Dec**

Su	Mo	Tu	We	Th	Fr	Sa
		1	2	3	4	5
6	7	8	9	10	11	12
13	14	15	16	17	18	19
20	21	22	23	24	25	26
27	28	29	30	31		

Event Rule Definition

Event Entity Filter:
 All Entities
 All Equipment
 All Crews
 All Pits/Panels
 Selected
 Matching

Equipment: SHOVEL 101
 Equipment: SHOVEL 102
 Equipment: SHOVEL 103

Shifts 1st 2nd 3rd 4th

Due to:

Repeat:

For:

Label:

Events for date selected

Event Name
 Christmas Holiday

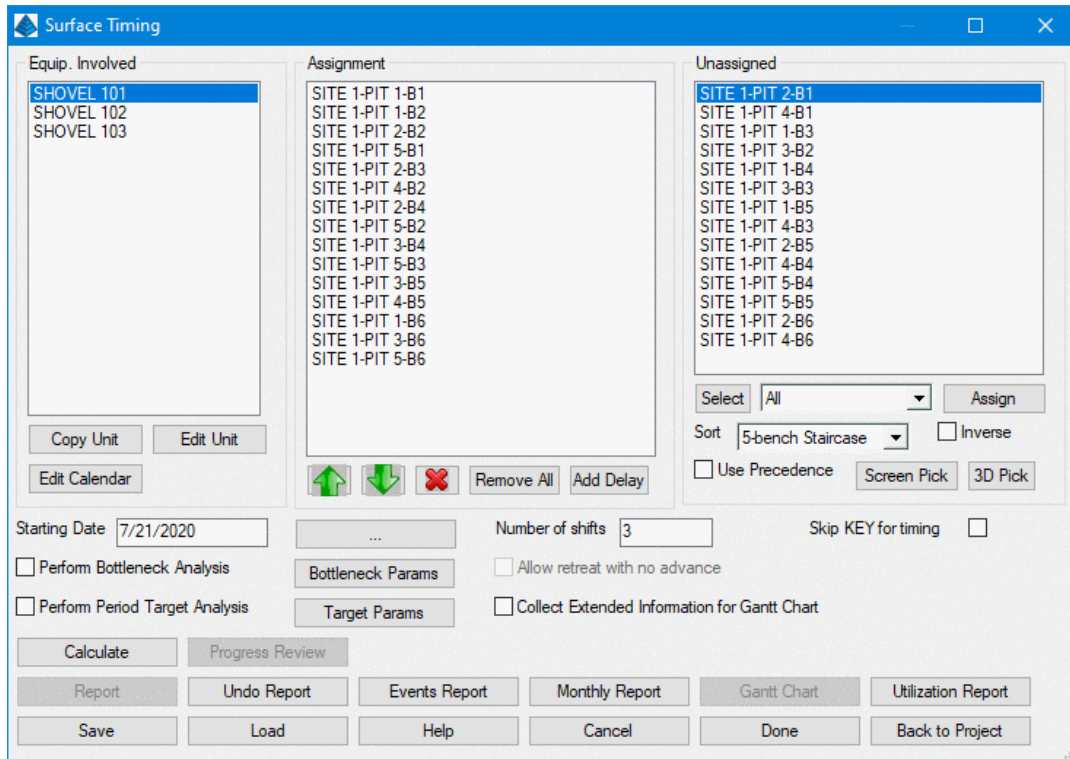
List All Rules

Surface Equipment Timing

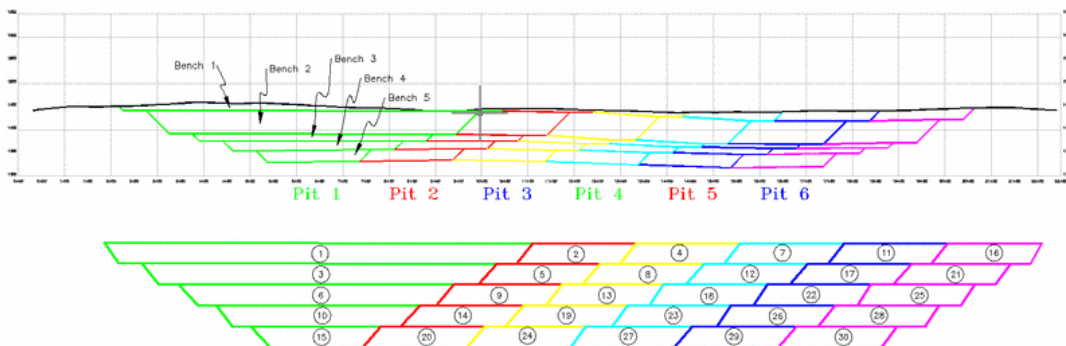
(Surface Mining module > Reserve/Timing pull-down menu > Surface Equipment Timing Command)

The Surface Equipment Timing command schedules the equipment through the pits. The first dialog shows three list boxes for the equipment, the pits assigned to the equipment and the unassigned pits. The pit benches are listed

as site name - pit name - bench number. For example, "Site 1-Pit 1-2" is for bench 2 in pit 1. To set the equipment to use, choose the Add Equip button and select the equipment name defined in the last step.

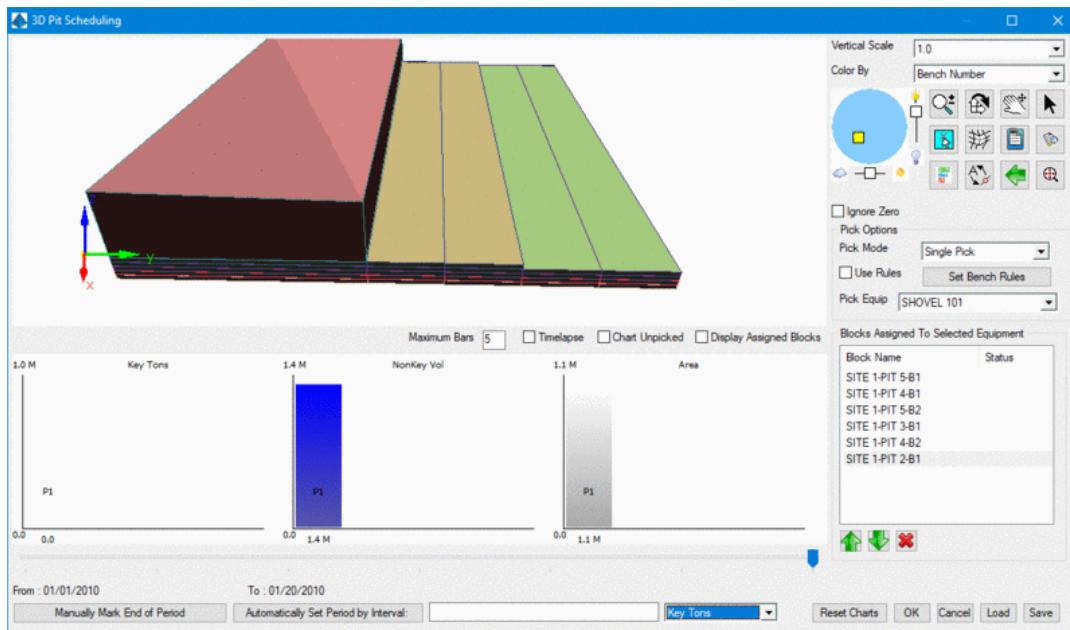


The sequence of mining the pit benches must be assigned. In this case, we want a 5-step staircase that advances through the pits. By default the pit benches are sorted in the unassigned list by pit number then bench number. To arrange the pit benches in the staircase order, choose the Sort dropdown list and select 5-Bench Staircase. Then pick the Select button followed by the Assign button. This will assign the pit benches to the equipment in staircase order. In this example, just manually select the first pit/bench, and holding the CTRL button down, select every other one. Then hit Assign, assigning them to Shovel1. Then highlight Shovel2, hit Select All, and assign the remaining pits/benches to Shovel2.

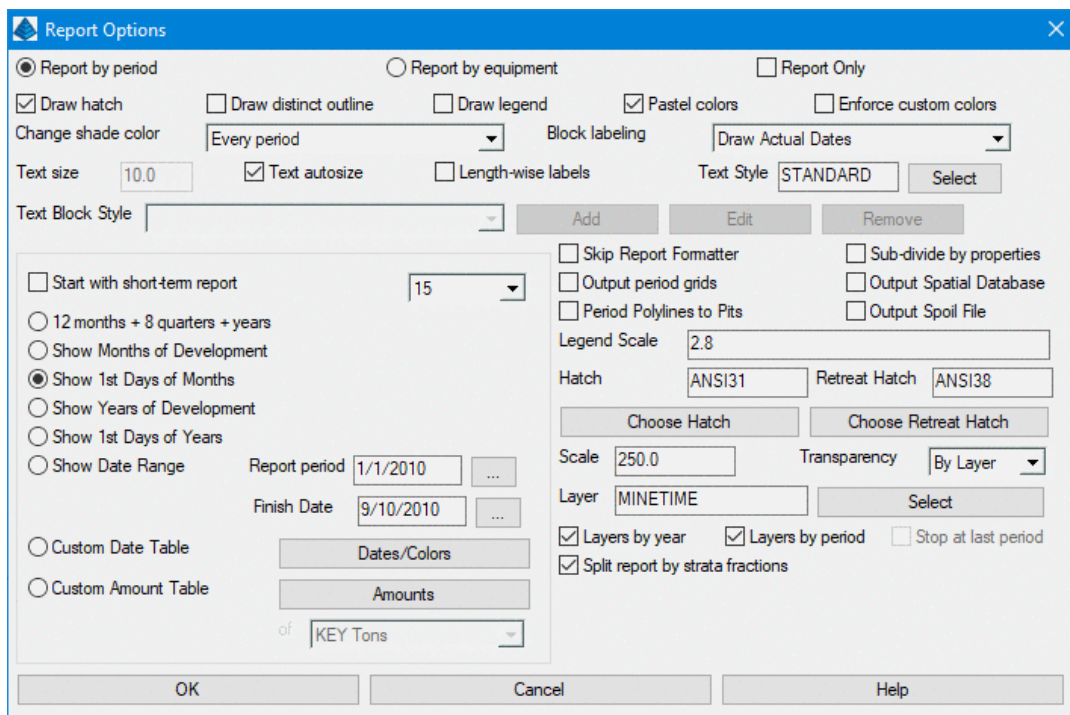


Shown above are 6 profiles of the bench pits. They have been manually connected with polylines and the pits and benches are labeled to show the "blocks". This is to illustrate the real bench width in cross section view. The lower graphic shows this profile exaggerated and numbered exactly as each block is going to be sequenced in the 5 bench staircase method. The sequencer automatically sorts them in this order.

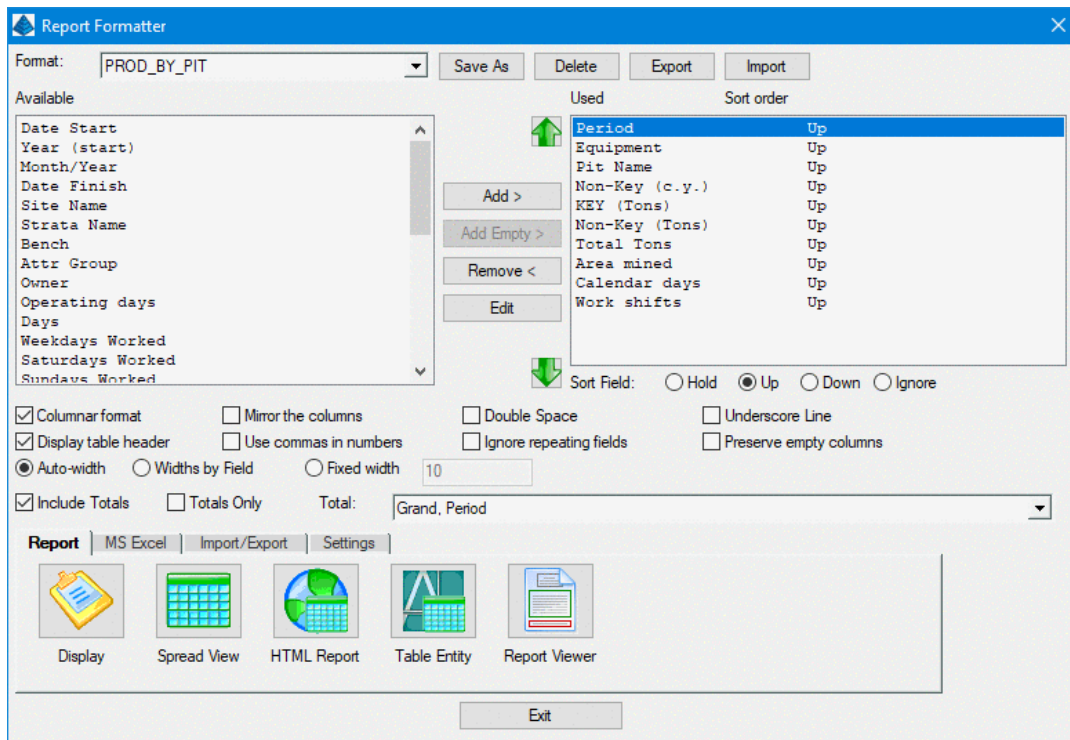
The 3D Pick option can be used to graphically select each bench and pit in the order to be mined. It is very useful for short-term scheduling. Just select 3D Pick, rotate it to an useful 3D view and then double click on each block. It will be removed and assigned to the highlighted unit.



When all the pit benches are assigned as shown in the dialog, click the Calculate button. The program will then run the equipment through the pits and report the completion date. Then click the Detailed Report button. This brings up the Report Options button.



In this case, the report period is set to Show 1st Days of Months. This will give a monthly schedule from the first of each month. Following is a report by monthly period. This can be formatted in many ways and exported to Excel.



KEY production report Mon Aug 3 14:21:34 2020

Period: 1/1/2010 - 2/1/2010

Equipment	Pit Name	Non-Key (c.y.)	KEY (Tons)	Non-Key (Tons)	Total Tons	Area mined	Calendar days	Work shifts
SHOVEL 101	Pit 1	378534.3		408817.1	408817.1	67991.0	5.26	15.77
SHOVEL 101	Pit 1	1853465.7		2001742.9	2001742.9	630866.1	25.74	77.23
SHOVEL 102	Pit 2	152097.9		164265.8	164265.8	189169.7	4.22	12.67
SHOVEL 102	Pit 3	111381.6		120292.1	120292.1	189169.7	3.09	9.28
Subtotal for Period: 1/1/2010 - 2/1/2010		2495479.5	0.0	2695117.9	2695117.9	1077196.4	38.32	114.96

Period: 2/1/2010 - 3/1/2010

Equipment	Pit Name	Non-Key (c.y.)	KEY (Tons)	Non-Key (Tons)	Total Tons	Area mined	Calendar days	Work shifts
SHOVEL 101	Pit 1	2016000.0		2177280.0	2177280.0	362106.7	28.00	84.00

Period: 3/1/2010 - 4/1/2010

Equipment	Pit Name	Non-Key (c.y.)	KEY (Tons)	Non-Key (Tons)	Total Tons	Area mined	Calendar days	Work shifts
SHOVEL 101	Pit 1	1117762.1		1207183.0	1207183.0	200768.4	15.52	46.57
SHOVEL 101	Pit 2	462520.3		499522.0	499522.0	189169.7	6.42	19.27
SHOVEL 101	Pit 2	511067.2		551952.5	551952.5	189169.7	7.10	21.29
SHOVEL 101	Pit 4	71159.7		76852.5	76852.5	189169.7	0.99	2.96
SHOVEL 101	Pit 5	54810.6		59195.4	59195.4	189169.7	0.76	2.28
SHOVEL 102	Pit 1	552000.0		596160.0	596160.0	127964.7	15.33	46.00
Subtotal for Period: 3/1/2010 - 4/1/2010		2769319.8	0.0	2990865.4	2990865.4	1085411.8	46.13	138.39

Period: 4/1/2010 - 5/1/2010

Equipment	Pit Name	Non-Key (c.y.)	KEY (Tons)	Non-Key (Tons)	Total Tons	Area mined	Calendar days	Work shifts
SHOVEL 102	Pit 1	1080000.0		1166400.0	1166400.0	250365.6	29.86	89.57

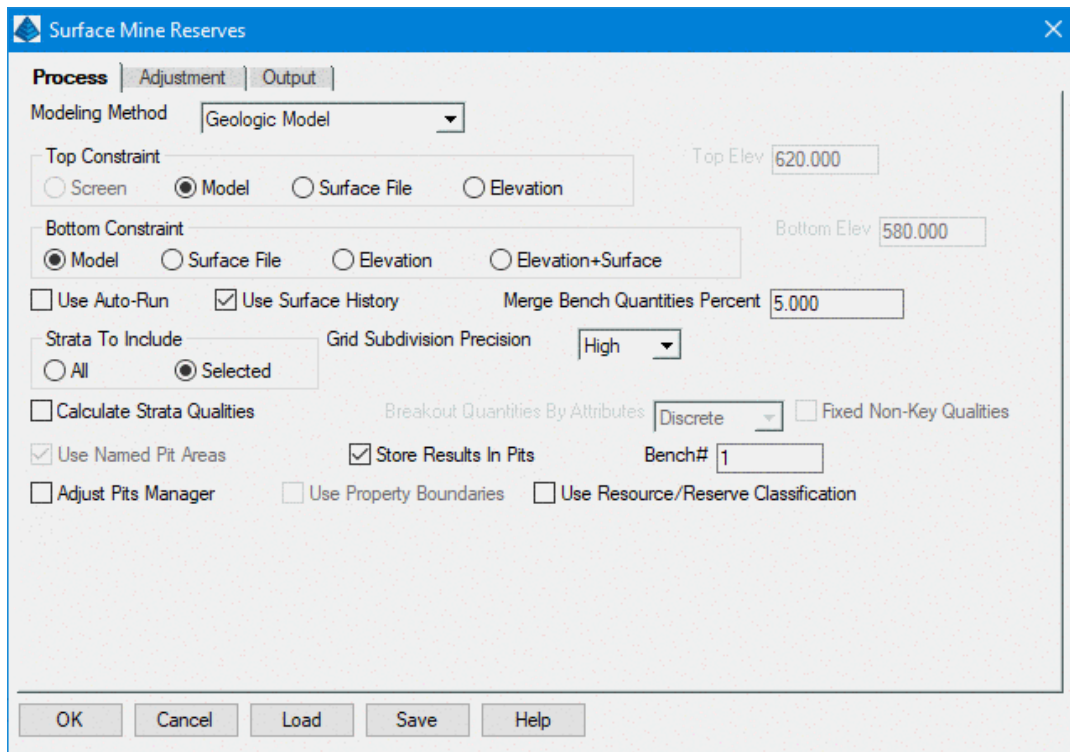
Period: 5/1/2010 - 6/1/2010

Equipment	Pit Name	Non-Key (c.y.)	KEY (Tons)	Non-Key (Tons)	Total Tons	Area mined	Calendar days	Work shifts
SHOVEL 102	Pit 1	1080000.0		1166400.0	1166400.0	250365.6	29.86	89.57

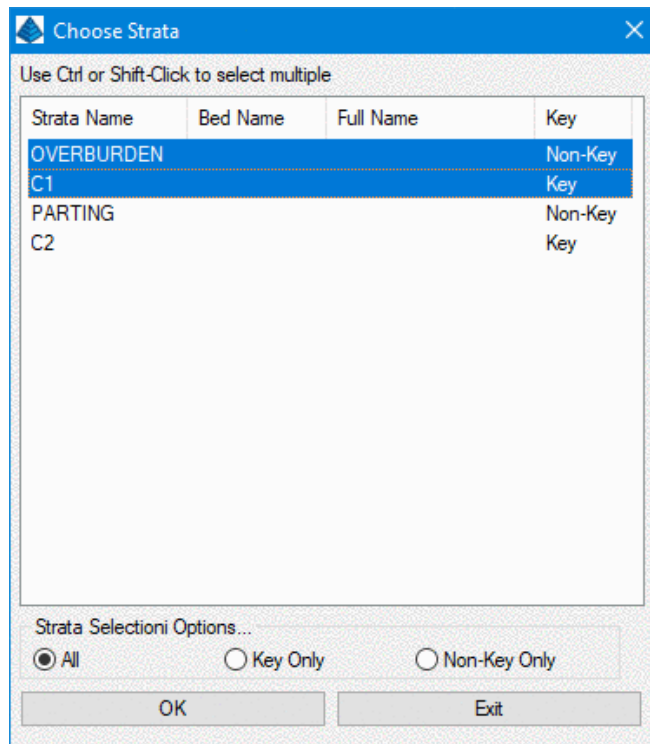
Multi-Bench Mining Example Setup:

The defining of equipment, crews and equipment calendar are (potentially) one-time operations. Similarly, the making of the Geologic Model needs be done only once in advance of numerous timing runs. Surface Equipment Timing, also distinct from Surface Production Timing, allows multi-bench mining by use of different equipment or the same equipment mining different benches, with different production rates. Since single-bench mining is a subset of multi-bench mining, this will illustrate the command with a multi-bench example. Surface Mine Reserves

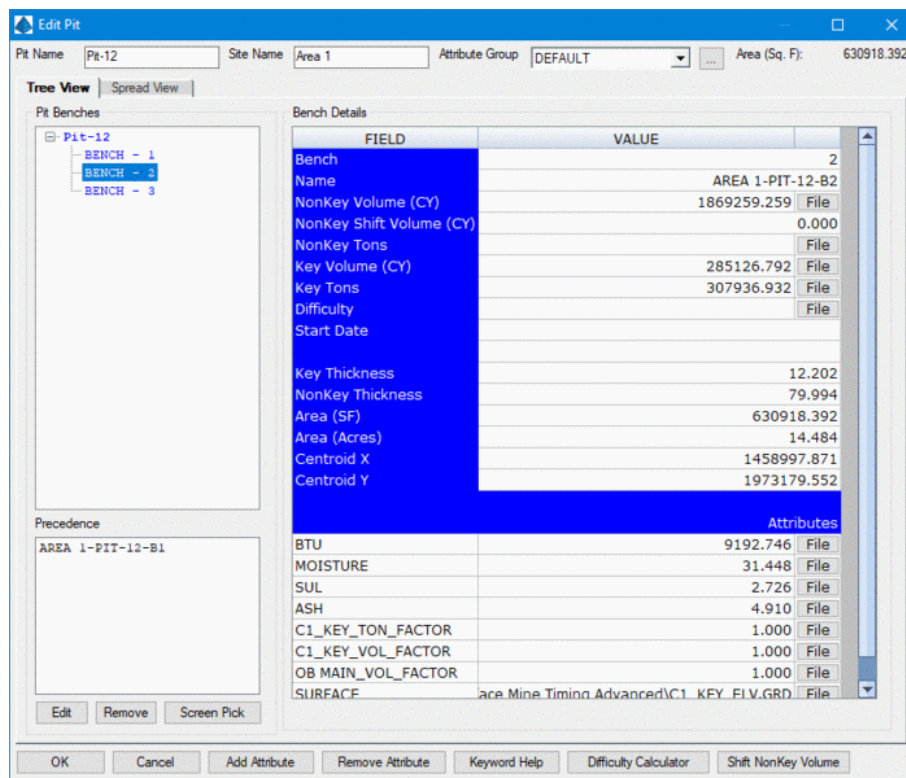
is one of three methods to place quantity and quality information into the pits, by bench. This example will use Surface Mine Reserves. If there are two benches, Surface Mine Reserves must be run twice, one time for each bench. The dialog within Surface Mine Reserves should be completed as follows for Bench 1:



The key is to choose "Selected" strata (for benching) as well as "Store Results in Pits". You should also select "Calculate Strata Qualities" to assign quality information to the pits. If single-bench mining is conducted, you may omit the "Selected" option. Other items in the dialog are set according to user preference. Note the option "Output Thickness Grids". If this is not selected, total quantity and composited quality information is placed in the pits, and mining across a single pit is proportional. However, if the "Make Thickness Grids" option is selected, mining across the pits picks up the varying of the overburden thickness, coal thickness and quality information. For small pits or "blocks" as they are sometimes called, storage of total information will still lead to reliable results. Thickness grids are recommended for large and long pits, where the thickness will vary. The choice of "Selected" strata requires that the user select which strata will be mined in Bench 1. The following Choose Strata dialog at will appear:



Holding the CTRL or SHIFT keys, select Overburden and C1 to mine down to the bottom of the first coal (C1). Bench 2 would be comprised of Parting and C2. Following two runs of Surface Mine Reserves, new information has been added to the pits. Pits contain up to three categories of information: pit name (verified by using "Identify Pit Polylines"), pit direction, and pit quantities/qualities organized by bench. Pit quantities and qualities are needed only for Surface Equipment Timing, and can be verified using the command "Edit Pit". For example, the information in Edit Pit is shown below. Select the "Attribute" option to reveal the quality attributes.



You will note that Bench 2 has Bench 1 as a "precedence" bench. In other words, you cannot mine Bench 2 unless

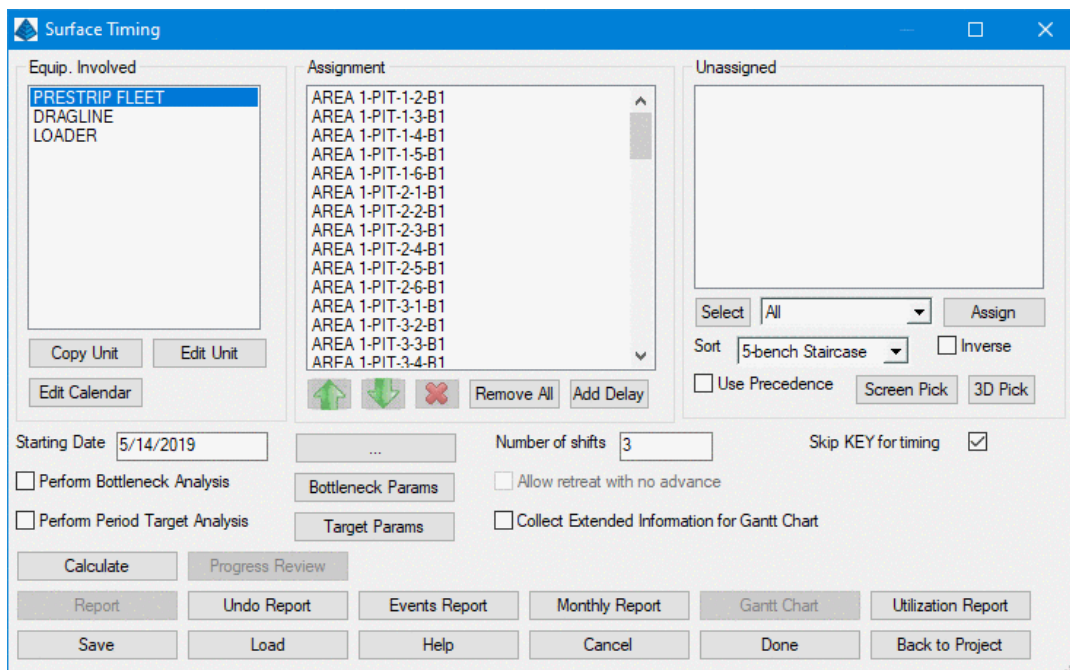
you first mine Bench 1. Lower benches are automatically set to require prior mining of upper benches (The ordering of the mining comes later!). The Pit Attributes show any attributes set by Define Pit Attributes. Some may be seen above.

Assuming that a calendar has been established (multiple calendars can be saved and recalled), and assuming that equipment has been defined, we are now ready for Surface Equipment Timing. In our three-bench mine, we will use one piece of equipment for the prestrip and different piece of equipment for the lower bench (parting).

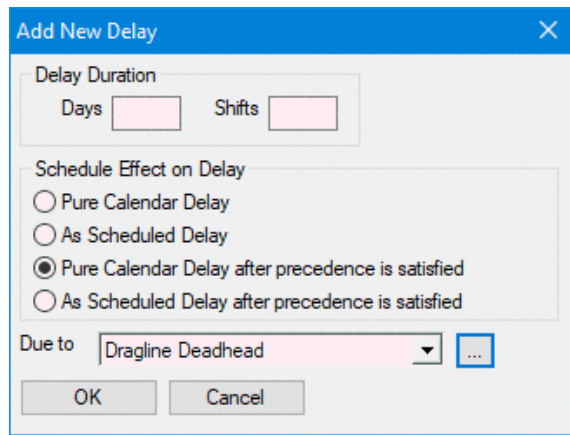
Technically, separate equipment would probably be used to remove the coal, but in this example, the coal is not included in the schedule quantities, just uncovered reporting. In fact, a 4-bench mine could be set up that considers each of the two coal seams as separate benches. In reality, however, the overburden and parting removal rate will typically govern the overall progress of the mining. When "Add Unit" is selected, the following dialog appears.

Select the equipment and add them one at a time. The draglines used here are defined based on three 8 hour shifts. The "Availability" option can be used to de-rate the mining speed in the Edit Equipment. Any of the above items can be revised from within Surface Equipment Timing. The next step is to assign "Dragline" to Bench 2. It is best to Sort by "Bench, Pit" and then Select "Bench 2", using the options at middle right in the Surface Timing Dialog.

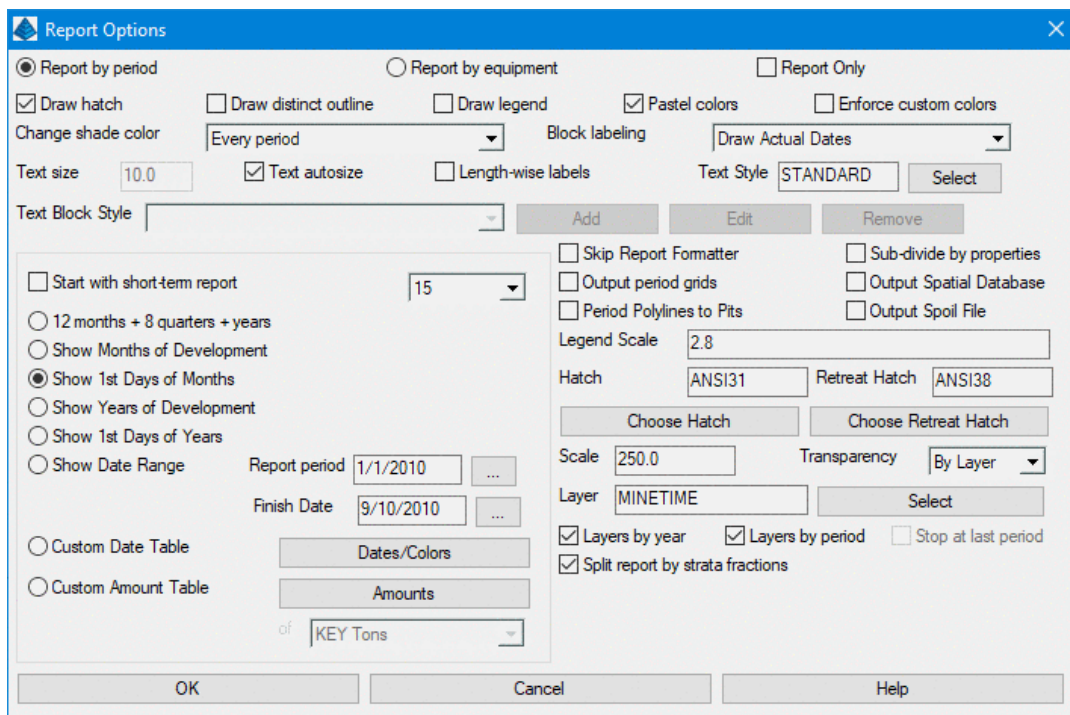
Also, assign all the Bench 3 to loader. These leads to the following appearance:



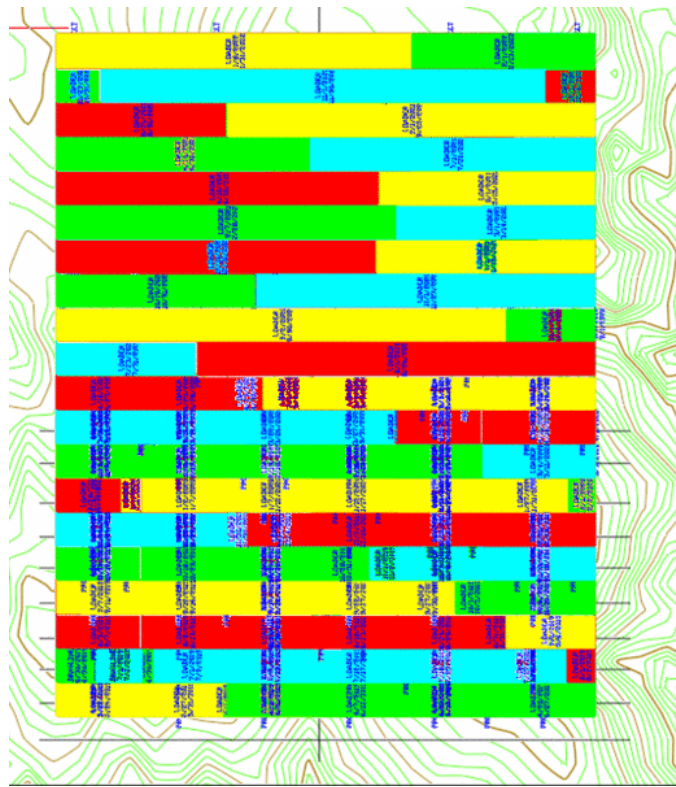
It is also important to note the option "Skip Coal for timing" at the lower right of the dialog box. Here you can base all progress on the NonKey overburden and parting only, on the assumption that the loaders or other equipment digging the coal will "keep pace" with the equipment removing the overburden and interburden. If "Skip Coal for timing" is not selected, the coal volume will be included in the total quantity mined by the equipment. We can add a second piece of equipment, then highlight it within the Surface Timing dialog and assign all the remaining pits to it (Select All or Bench 2, then Assign). Note also that we can add a delay between completing the Bench 1 pit and beginning the Bench 2 pit. Delays are automatically inserted below the highlighted pit, so if a pit is highlighted, a delay will be added below it that can be moved up or down also. There is a dialogue to add this as a calendar or schedule delay (described below). Shown below is the assignment for Bench 2.



To complete the calculation, fill in the Starting Date (if the default is not correct), then click on Calculate. This leads to a completion date and the option of a unit report or going directly to the Report Options screen. The Unit Report is very instructive because it highlights when equipment has been idled. Our Bench 2 dragline is waiting each time for the "slower" Bench 1 dragline to complete bench 1 before it can launch into bench 2. The planner could try lower rated equipment, or reverse the assignments of the equipment, or simply re-shuffle the assignments in any desired manner to maximize efficiency. The last step is to choose "Detailed Report" when returned to the Surface Timing dialog which brings up the Report Options dialog.



This example will show the first days of months. This leads to the final hatching shown below as bench 1 (the bench 2 layers were frozen for better appearance), as well as the quantity report which can be formatted many different ways and exported to Excel or Access:



Period	Equipment	Pit Name	Site Name	Bench	Operating Days	Weekdays	Saturdays	Sundays	Work shifts	KEY (Tons)	Non-Key (Tons)	Total Tons	KEY (c.y.)	Non-Key (c.y.)	
5/14/2019 - 6/1/2019	DRAGLINE	Pit-1-1	Area 1	1	2.59	2.59	2.59	0.00	0.00	5.18	0.0	352527.0	352527.0	0.0	174087.4
5/14/2019 - 6/1/2019	PRESTRIP FLEET	Pit-1-2	Area 1	1	4.44	4.44	4.00	0.44	0.00	8.88	0.0	269716.5	269716.5	0.0	133193.3
5/14/2019 - 6/1/2019	DRAGLINE	Pit-1-1	Area 1	2	5.02	5.02	3.02	1.00	1.00	10.04	44212.4	600000.0	644212.4	40937.4	296296.3
5/14/2019 - 6/1/2019	PRESTRIP FLEET	Pit-1-3	Area 1	1	2.40	2.40	0.84	0.56	1.00	4.80	0.0	145713.6	145713.6	0.0	71957.3
5/14/2019 - 6/1/2019	PRESTRIP FLEET	Pit-1-4	Area 1	1	3.15	3.15	3.15	0.00	0.00	6.29	0.0	191095.1	191095.1	0.0	94368.0
5/14/2019 - 6/1/2019	DRAGLINE	Pit-1-2	Area 1	2	5.05	5.05	3.39	1.00	0.66	10.10	46654.0	600000.0	646654.0	43198.2	296296.3
5/14/2019 - 6/1/2019	LOADER	Pit-1-1	Area 1	3	5.34	5.34	3.39	1.00	0.95	10.68	64927.8	137738.0	202665.8	60118.3	68018.8
5/14/2019 - 6/1/2019	PRESTRIP FLEET	Pit-1-5	Area 1	1	6.25	6.25	4.25	1.00	1.00	12.50	0.0	379618.2	379618.2	0.0	187465.8
5/14/2019 - 6/1/2019	DRAGLINE	Pit-1-3	Area 1	2	5.08	5.08	4.74	0.00	0.34	10.16	48792.1	600000.0	648792.1	45177.9	296296.3
5/14/2019 - 6/1/2019	LOADER	Pit-1-2	Area 1	3	5.05	5.05	5.00	0.00	0.05	10.10	61958.4	129355.7	191314.1	57368.9	63879.4
5/14/2019 - 6/1/2019	PRESTRIP FLEET	Pit-1-6	Area 1	1	1.77	1.77	1.77	0.00	0.00	3.53	0.0	107356.5	107356.5	0.0	53015.6
5/14/2019 - 6/1/2019	DRAGLINE	Pit-1-4	Area 1	2	0.26	0.26	0.26	0.00	0.00	0.52	2617.2	30146.1	32763.3	2423.3	14887.0
5/14/2019 - 6/1/2019		varies	Area 1		46.39	46.39	36.39	5.00	5.00	92.78	269161.8	3543266.9	3812428.7	249223.9	1749761.4
6/1/2019 - 7/1/2019	LOADER	Pit-1-2	Area 1	3	0.23	0.23	0.00	0.23	0.00	0.47	2871.5	5995.1	8866.6	2658.8	2960.5
6/1/2019 - 7/1/2019	DRAGLINE	Pit-1-4	Area 1	2	4.87	4.87	2.87	1.00	1.00	9.74	49472.8	569853.9	619326.7	45808.1	281409.3
6/1/2019 - 7/1/2019	LOADER	Pit-1-3	Area 1	3	5.35	5.35	3.58	0.77	1.00	10.69	67187.3	133896.1	201083.3	62210.4	66121.5
6/1/2019 - 7/1/2019	PRESTRIP FLEET	Pit-1-6	Area 1	1	7.90	7.90	5.00	1.90	1.00	15.80	0.0	480064.2	480064.2	0.0	237068.7
6/1/2019 - 7/1/2019	DRAGLINE	Pit-1-5	Area 1	2	5.18	5.18	3.18	1.00	1.00	10.36	55792.5	600000.0	655792.5	51659.7	296296.3
6/1/2019 - 7/1/2019	LOADER	Pit-1-4	Area 1	3	5.42	5.42	3.42	1.00	1.00	10.85	70590.5	131182.1	201772.6	65361.6	64781.3
6/1/2019 - 7/1/2019	PRESTRIP FLEET	Pit-2-1	Area 1	1	6.95	6.95	5.00	0.95	1.00	13.91	0.0	422409.0	422409.0	0.0	208597.1
6/1/2019 - 7/1/2019	DRAGLINE	Pit-1-6	Area 1	2	8.95	8.95	6.95	1.00	1.00	17.91	76262.6	785250.0	861512.6	70613.5	387777.8
6/1/2019 - 7/1/2019	LOADER	Pit-1-5	Area 1	3	5.47	5.47	3.47	1.00	1.00	10.93	72638.1	129425.0	202063.1	67257.5	63913.6
6/1/2019 - 7/1/2019	PRESTRIP FLEET	Pit-2-2	Area 1	1	5.28	5.28	4.13	0.14	1.00	10.55	0.0	320476.6	320476.6	0.0	158260.0
6/1/2019 - 7/1/2019	DRAGLINE	Pit-2-1	Area 1	2	5.05	5.05	3.05	1.00	1.00	10.10	46406.0	600000.0	646406.0	42968.5	296296.3

Summarized below are numerous Carlson tutorials and the (*module[s]*) they cover. If you are new to CAD (*e.g.* IntelliCAD or AutoCAD) or are starting Carlson for the first time, it is suggested you explore:

- CAD Basics (*General/All*)

As you get familiar with the basics of CAD and Carlson, numerous tutorials designed to assist you in learning the Carlson Desktop CAD product line. They are:

- Lesson 1 - Entering a Deed (*Survey*)
- Lesson 2 - Making a Plat (*Survey*)
- Lesson 3 - Field to Finish - Faster Survey Work (*Survey*)
- Lesson 4 - Intersections and Subdivisions (*Survey*)
- Lesson 5 - Contouring DTM and Design (*Civil*)
- Lesson 6 - Contour Break Lines Stockpiles (*Survey*)
- Lesson 7 - Basic Road Design with Volumes (*Civil*)
- Lesson 8 - Hydrology & Watershed Analysis (*Hydrology*)
- Lesson 9 - Dozen Tools of Surface Design (*Civil*)
- Lesson 10 - Takeoff Basics (*Construction*)
- Lesson 11 - Takeoff Drillholes and Strata (*GeoTech*)
- Lesson 12 - Trenches (*Construction, Trench*)
- Lesson 13 - Digitizing (*CADnet, Construction*)

Other tutorials of interest include:

- Cut Sheets (*Survey*)
- ESRI Office Field (*GIS, SurvCE, Survey*)
- Geodetic Reports (*Survey*)
- SurvNET Least Squares Adjustment (*Survey*)
- Calculate Volumes By 5 Methods (*Civil*)
- Road Rehab (*Civil*)
- Carlson Hydrology/Hydrocad Tutorial (*Hydro*)
- Stormwater Network Design (*Hydro*)
- Pressure Pipe Network (*Hydro*)
- PDF Section Import (*CADnet, Construction*)
- GeoTech Reports (*GeoTech*)

Pulldown Menu Location(s): Help

Keyboard Command: -na-

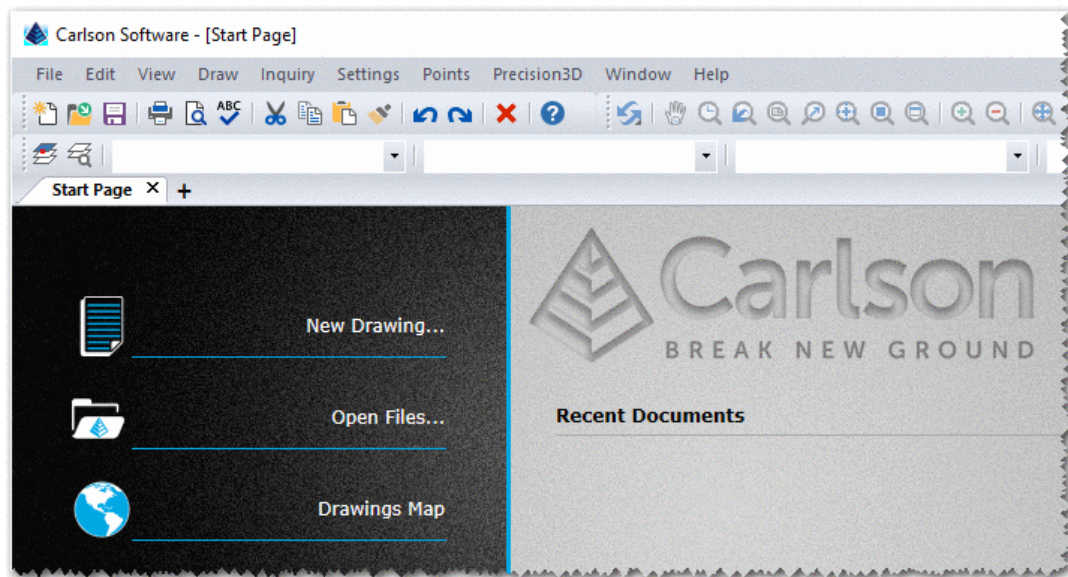
Prerequisite: An available license of the specified module/product

CAD Basics

Carlson Software's office products utilize either an AutoCAD or IntelliCAD-based CAD engine. This section will review some basics of CAD software that will be useful in running Carlson Software. Running Carlson on IntelliCAD is largely the same as running on AutoCAD.

Once installed to your computer, one or more Carlson start-up icons (based on the product[s] contained within the product serial number) will typically be placed on your computer desktop. Carlson recommends launching the software through the desktop icon so proper software initialization parameters can be set. Once the Carlson software environment has been initialized, you can choose to open an existing document type (*e.g.* an existing *.DWG file) or create a new document type (*e.g.* a new *.DWG file). Based on your desired (or established) interface settings, one of three "start-up" environments may be presented:

1. **Start Page** (IntelliCAD users) - When this environment is active (*i.e.* variable STARTMODE = On), a generic "Start Page" will be displayed with common first-time commands and the software will wait for you to generally create a New Drawing or Open Files that the CAD application supports:



- Clicking **New Drawing** will launch the Startup Wizard which provides a step-by-step process for the creation of a new, supported document type.
- Clicking **Open Files** will launch a File Selector which permits navigation to a desired folder location on your computer (and/or shared network drive, if available) which provides the ability to directly open an existing, supported document type.

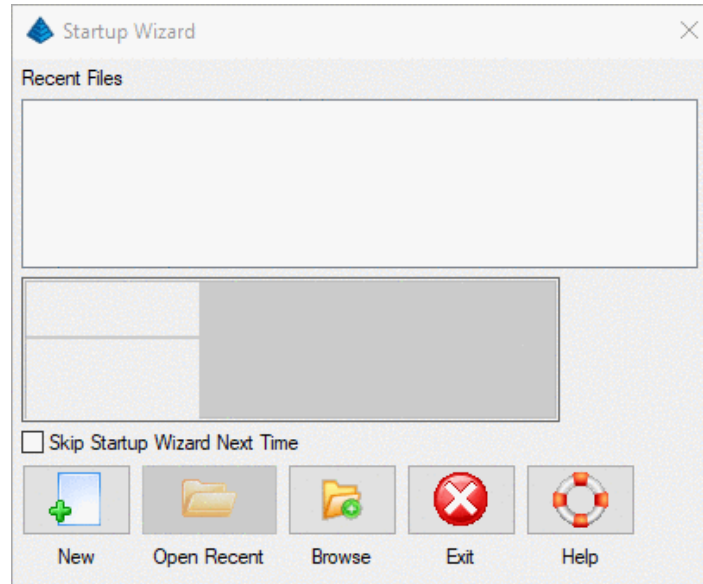
A primary benefit of the **Start Page** environment is that only the drawing(s)/documents that you choose to create or open will be in use by the application. In this fashion, when exiting from the software, you'll have the benefit of only having to save changes to documents that are open and that have changed.

NOTE: The complement of commands offered by the Carlson module(s) will not be fully loaded until at least one supported drawing document type is either created or opened.

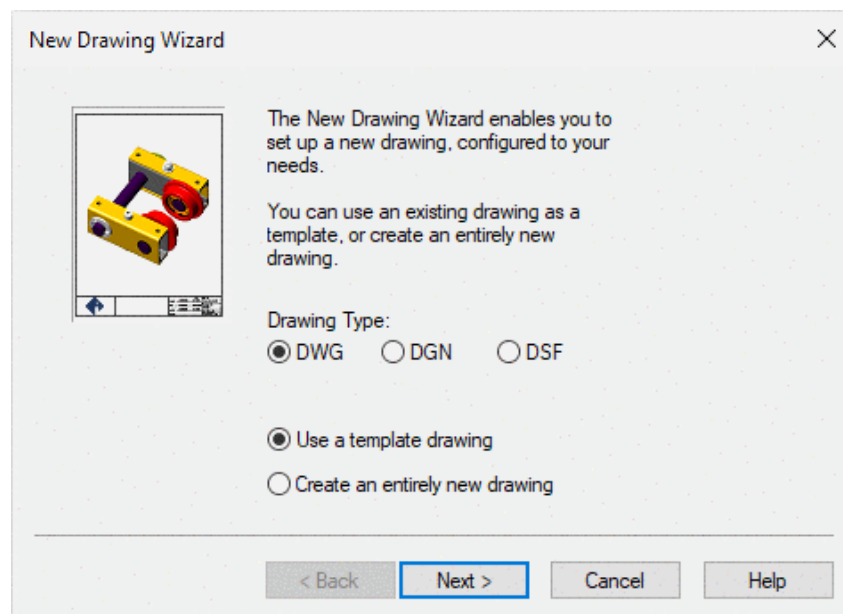
NOTE: The "Start Page" environment can be suppressed by enabling the **Skip Start Page Next Time** toggle in the lower left corner of the "Start Page." Enabling this feature will (by default) revert to the Startup Wizard interface the next time the application is restarted. The "Start Page" environment can be re-enabled by:

- (a) Opening an existing drawing document, and,
 - (b) Keying-in **STARTMODE** into the Command: prompt, and,
 - (c) Setting the value of the variable to **On**.
2. **Startup Wizard** - When this environment is active, CAD will launch with a default drawing document (*e.g.* Drawing1.dwg) pre-loaded along with an interface with a step-by-step (*aka wizard*) approach for the creation

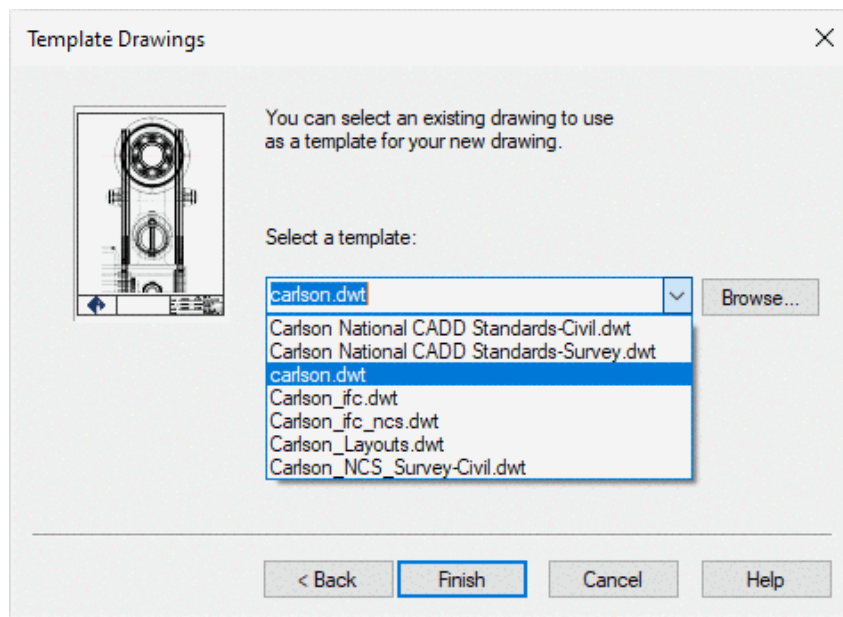
of a new drawing document type:



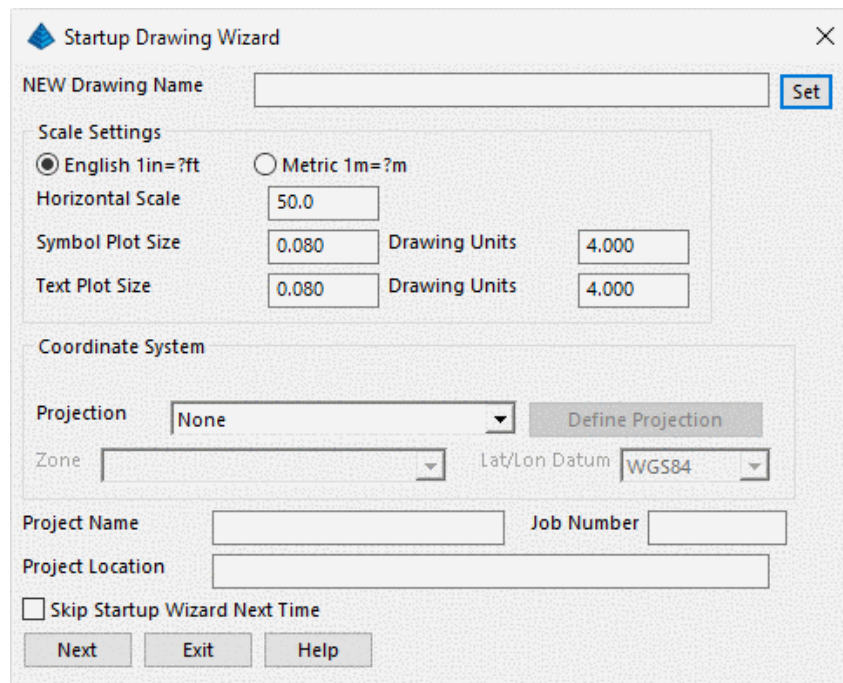
- Clicking **Browse** will launch a File Selector which permits navigation to a desired folder location on your computer (and/or shared network drive, if available) which provides the ability to directly open an existing, supported document type.
- Clicking **Open Recent** will open the currently selected/highlighted drawing document which has been previously opened by the application.
- Clicking **Exit** will dismiss the Startup Wizard dialog box and leave the default drawing document active and available for data entry.
- Clicking **New** will start the formal Startup Wizard process which generally entails the following sequence:
 - (a) Establish the desired document type (and/or source) and click **Next** >:



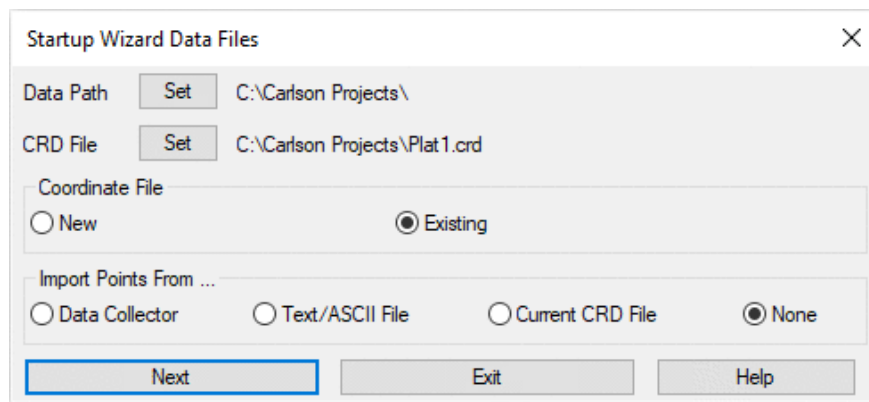
- (b) Establish the source document type (*aka* "template") and click **Finish** to launch into the drawing-specific settings:



- (c) Establish the document name (e.g. use the **Set** button to establish the desired Folder and Document name) along with expected plot scale and desired Coordinate System information as desired and click **Next** to continue with additional information:



- (d) Validate the Destination Folder and establish the related "Coordinate File" name to house designated point data and whether an existing Coordinate File (e.g. such as that created by Carlson SurvPC or SurvCE) or if a new Coordinate File should be created along with the source for any inbound Point Data and click **Next** to continue with further processing instructions (if applicable):

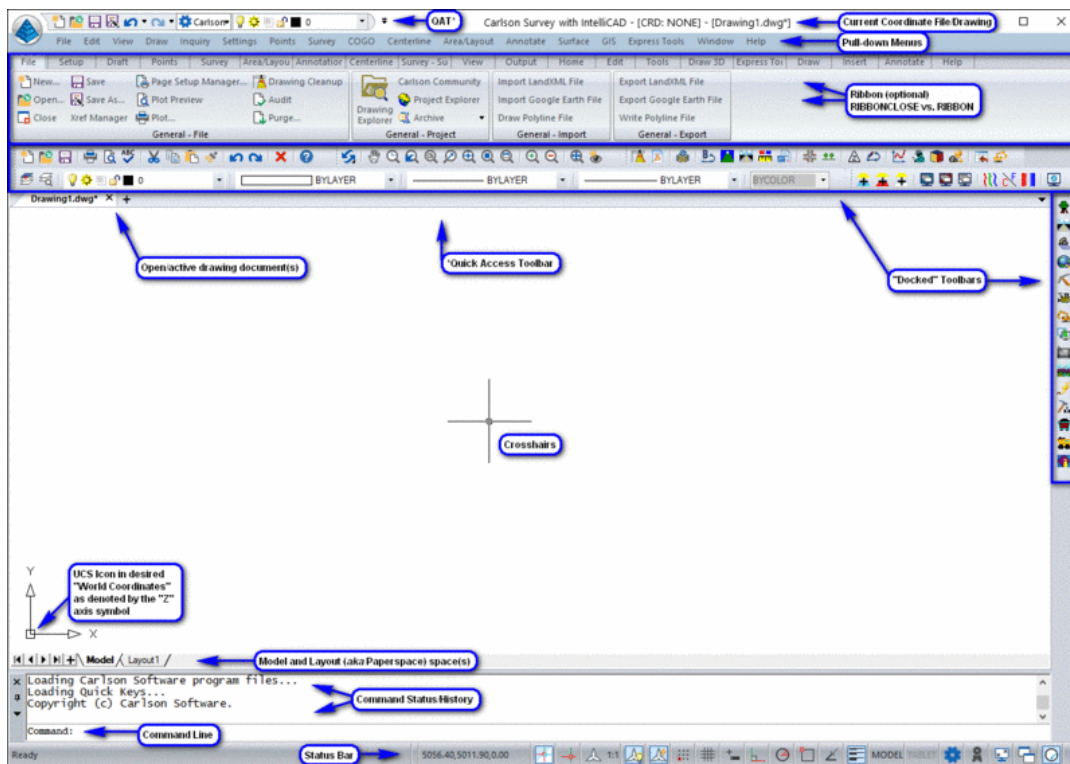


NOTE: The precise sequence of steps within the Startup Wizard will vary based on the option(s) selected within each step of the Wizard. It is recommended that the Wizard be followed to its logical conclusion and the click of a resulting **Finish** button.

NOTE: The "Startup Wizard" environment can be suppressed by enabling the **Skip Startup Wizard Next Time** toggle in the lower left corner of the opening "Startup Wizard" dialog box or the tertiary Document Setup Name dialog box. Enabling this feature will (by default) revert to the standard CAD interface the next time the application is restarted. The "Startup Wizard" environment can be re-enabled by:

- (a) Opening an existing drawing document, and,
 - (b) From the Settings menu, click Carlson Configure, and,
 - (c) Clicking the General Settings and enabling the **Use Startup Wizard** option.
3. **Standard CAD** - When this environment is active, the CAD application starts with a default document name (e.g. Drawing1.dwg) pre-loaded and ready for data entry. It is up to the user to properly save and rename the drawing document to a desired folder and file name.

When configuring the "GUI" (goeey) interface and/or effectively communicating with Carlson Technical Support, it is helpful to understand the various components, terminology and their locations within the CAD application:



Virtually all commands in CAD software have three or more ways they can be initiated. The two most common are

through the Pull-down Menus and the Toolbars, but the persons who are comfortable with the "Ribbon" interface or Command line interface (for quick keystroke entry) have various options available to them. Using the Menus and Toolbars should be second nature to Windows users, so this section will mainly focus on the Command line usage in CAD.

Command Line Prompt (Command:)

CAD softwares have a Command line prompt where commands are issued and the status of a command is reported. When you select a command from a menu or toolbar, that key-in command name is sent to the Command line and is executed. When a command is issued, the Command line acts as a status bar that will show the available options and prompt for input from you (either in the form of Command line options and/or a dialog box).

Before most commands can begin, all other commands must be terminated. The exceptions are referred to as Transparent commands. The easiest way to see if no commands are running is to look at the Command line. If it displays "Command:" no commands are currently in progress.

Enter

When you are required to press the Enter key in CAD, you can use the Enter key on the keyboard or you can press the spacebar or click the right mouse button.

Pressing the Enter key will perform different operations depending on your location within a command:

- If you are in the process of running a command which has no sub-options, the Enter key will end the command.
- If you are in the process of running a command which has sub-options and a desired sub-option other than the default option has not been specified, the Enter key will initiate the default sub-option (the option between the < > characters).
- If you are in the process of running a command which has sub-options and you wish to exercise a sub-option other than the default, left-click the desired sub-option or type the Capitalized character(s) (not case-sensitive) and press Enter.
- If you are in the process of selecting entities for a desired action, the Enter action will complete the current selection method and a second Enter will initiate the command on the selected entity(ies).

To avoid any problems with using the above methods to end a command, you can press the Escape key (**Esc**) as described below.

Repeating Commands

When you press the Enter key (or its equivalents) at the Command: prompt without a command running, you will repeat the last command.

Right Mouse Button

As mentioned above, the right mouse button can be used as Enter. However, you can set the right mouse button to perform different functions as well. When so configured, a right mouse may provide a context-sensitive list of commands and can select the desired one (*e.g.* Move the cursor the desired option and click on it with the left mouse button).

Canceling a Command (Esc)

The Escape key (**ESC**) key can be used to cancel any command. Some commands (such as Transparent commands) may require pressing the Escape key more than once.

For example, if the command line displays something other than Command: or if a command you want to run does not start because you are using another command, you will want to end the command and return to the command prompt. Using the Escape (Esc) key will accomplish this every time.

NOTE: If you accidentally left-click a location on the screen and start a selection window, the command prompt will prompt for another corner. Either select another point on the screen to finish the selection (not advised) or press the **Esc** key to return to the Command: prompt.

Undo

Undo reverses the effect of the previous command. Type **U** or use the shortcut **Ctrl+Z** to run Undo.

Redo

Redo reverses an UNDO command. You must use the UNDO command before you can use the REDO command. Type **REDO** or use the shortcut **Ctrl+Y** to run Redo.

NOTE: Panning and Zooming is made easy if your mouse has a center wheel. You can use it to zoom in and out by rolling the mouse wheel up or down. By holding the center wheel down on the mouse, you can pan in any direction.

Pan

Press and hold the middle (wheel) button of a 3-button mouse and drag the view to the desired location. Alternatively, type **P** for a transparent Pan then click and hold the left-button of the mouse on the point from which you want to move the display. Drag the mouse to the desired area, then release the mouse.

Zoom

Typing **Z** for Zoom will give you several options: To specify the scale, enter the magnification value. To zoom in, choose In. To zoom out, choose Out. To display the entire drawing, choose All. To view a window defined by a center point and the magnification value or height, choose Center. To display the drawing extents, choose Extents. To view a window by selecting the desired area in the drawing, choose Dynamic. To view a window defined by the left point and the magnification value or height, choose Left. To return to the previous display, choose Previous. To view a window defined by the right point and the magnification value or height, choose Right. To view a window defined by specifying two (opposite) corners, choose Window.

Transparent Commands

Several commands in CAD software can be run transparently. This means that they can be performed while another command is running. For example, if you are in a command and are trying to select something in the drawing but it is too small to see, you can use the zoom command transparently. Zoom to the area where the object is, then select the object without ending the initial command. The most commonly used commands are the View commands of Zoom and Pan, and the Properties commands including the Layer dialog box.

To issue a transparent command, type an ' (apostrophe) before the command name. For example, **'Z** would be Transparent Zoom. Note that many commands will automatically be assumed to be transparent if they are issued from the toolbar while another command is running.

When you are in a command that is running transparently, this will be indicated with a >> at the far left of the command line preceding any options or other text. When a transparent command is complete, you will return to the command you were previously running. If you are in a transparent command and want to end the command to get back to the command prompt, you will need to press the **Esc** key twice.

NOTE: If you select a View command while running another command, the other command will not end. You will be running the View command transparently. This is one of the exceptions to the rules for ending a command. If you do not want to run the View command transparently, you need to complete the other command or end it by pressing the **Esc** key.

Command Sub-options

The Command line changes as a command is running. When there are options available for the command you are running, they will display at the Command line with capital letters in the option name. To use one of the options, type the capital letter(s) at the prompt (but know that these "accelerators" are case-insensitive).

For example, if you issue the Zoom command, the Command prompt will typically show sub-options of *[In/Out/All/Center/Dynamic/Extents/Left/Previous/Right/Window/ENtity/Scale]<Scale (nX/nXP)>*:

To select the Dynamic option, type **D** (or **d**) at the Command line, then press the Enter key. If you do not input an option and press Enter, you will be selecting the option that appears between the < > brackets. For the example shown above, that is the Zoom – Scale option.

Most commands in CAD software require the selection of objects. When you need to select objects, the command line will prompt **Select objects:** (or **[FILter]/<Select entities ...>**: in IntelliCAD) . When you are at this prompt, your next step will be to create a selection set. While creating the selection set, the prompt **Select Objects:** repeats and you can continue to select objects until you press the Enter key, at which time the command you are using will continue and use the objects selected.

Selection Sets

There are several ways to create a selection set from the **Select objects:** prompt. With all selection methods, the number of objects selected will be displayed in the Command line along with any objects that were duplicated. Following are the most commonly used methods for creating a selection set:

Single

A single selection is made when you move the object selection target to an object on the screen and click on it. The selected object will highlight and the **Select objects:** prompt will return. The cursor changes to a small square when the Command line displays **Select objects:**.

Window

A Window will select all objects completely inside of the rectangle drawn. Create a window by selecting a point on the view screen and then moving the cursor right. The window will display as a solid rectangle. You can also create a window by typing **W** at the **Select objects:** prompt. In this case, the direction of the window creation does not matter.

Crossing

A crossing will select all objects within the rectangle as well as those touched by the rectangle. If you select a point on the view screen and move the cursor to the left, you are creating a crossing. The crossing will display as a dashed rectangle. You can force a crossing by typing **C** at the **Select objects:** prompt, allowing you to move the cursor to the right and create the crossing.

Previous

After you select several objects, they will be temporarily stored as a selection set. Should you want to re-use the same objects that were selected by the last command for a different command, you can generally type **P** at the **Select objects:** prompt.

Remove

If you select incorrect objects, you can type **R** to remove objects from the selection set. When you are in Remove mode, the prompt will be **Remove objects:**. Click on the objects that you do not want to include in your selection. To return to select (or "add" mode), type **A** at the Command line.

Snaps

When creating geometry, it is often desirable to locate aspects of the new geometry piece to the precise location of key-points of an existing piece of geometry (*i.e.* setting the endpoint of one line to the midpoint of another line). This can be accomplished through the use of *entity snaps* (object snaps in AutoCAD). Desired snaps can be turned on to be available all the time (through the status bar area of the GUI or via the **F3** function key) or issued within a command through a key-in of the first three letters of the snap (*e.g.* **end**point, **mid**point, **near**est, **int**ersection, **per**pendicular, *etc*).

Grips

Objects may also be selected before issuing the command and receiving the **Select objects:** prompt. This selection will turn on CAD grips. Grips appear as small blue squares (*aka* "cool grips") in the drawing. All objects with grips will be used when a command is issued. Grips are turned on similar to selecting objects as described in the previous section, but with the command line at the **Command:** prompt, not the **Select Objects:** prompt. You can select single objects by clicking on an object with the standard cursor or you can select multiple objects by clicking in the view where there are no objects, then creating the window (right) or crossing (left). To remove grips, press the **Esc** key twice.




Using Grips

Grips can also be used to edit or change the location of objects in the drawing. Move your cursor to a grip and click. You should notice the cursor snap to the grip. The grip will turn from blue to red (*aka* a "hot grip"). The object grip is now attached to the cursor so you can move the object to the desired location and release it by clicking again.

Layers (and pertinent layer properties) define how an object in CAD is organized and/or displayed. One of the most common properties is the Layer. Layers can be turned off or frozen so the objects on that layer are hidden from the view of the drawing. Layers that are turned off can still be selected while frozen layers are essentially removed from the working set of the drawing. After layers are turned off or on, a redraw (which is done automatically) will update the view. A redraw of a file is a rather quick process. After a frozen layer is thawed, a regeneration may be required to update the view. Regenerations on large files may take a considerable amount of time depending on your hardware.



If this toolbar (or any toolbar) is not displayed, you can open it using the Toolbars dialog box. Type **toolbar** at the Command line or right-click on an existing toolbar for toolbar options. Common usage of this toolbar includes:

Common Name	Key-in	Toolbar	Description
Set Current	SETLAYER		<p>The "MOLC" (Make Object Layer Current) button permits you to select an existing entity in the drawing which makes the layer of the selected entity current. The current layer will be the one shown in the Layer drop list box. You can change the current layer by selecting the desired current layer from the drop list. Finally, you can highlight a layer in the Layer dialog box and click on the Current button to make the highlighted layer current.</p>
Layer	LA		<p>The Layer dialog box provides control of the drawing layers. You can turn layers on, off, freeze or thaw them, change the layer color and linetype, set the current layer, add new layers, delete layers, <i>etc.</i> To perform any of these functions, click on the Layers button. In the Layer dialog box, you can highlight several individual layers to perform actions on at once by holding down the Control key and clicking on the desired layers. You can also highlight a continuous range of layers by highlighting one layer, holding down the Shift key, and clicking on another layer. All layers between the two will be selected. To select all of the layers, hold down the CTRL key and press A on the keyboard. You cannot freeze the current layer, but you can turn that layer off (not recommended).</p>
Current Layer	-na-		<p>The Current Layer is the layer that you are on and will be working with. The Current Layer is the one shown in the Layer drop list. For example, in the above toolbar illustration, the Current Layer is PNTS. When you click on the</p>

For additional assistance on general CAD software basics, please consult the Introduction section of the Carlson documentation or visit our **Knowledge Base** at www.carlsonsw.com/support/knowledge-base/.

Entering a Deed

<<<<<<<< .mine

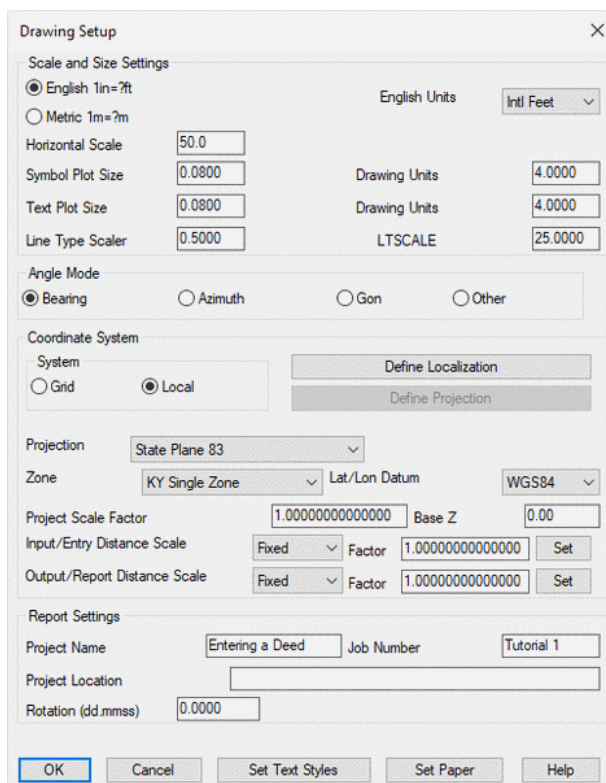
In this short lesson you will create a simple drawing. You will enter a 6-sided deed, add a title block, bar scale, and north arrow, add a title and certification text, and plot the deed area.

Note that the Esc key will cancel most commands, so if you choose the wrong command or enter something incorrectly and want to start over, just press Esc.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **New Drawing**.
 - If you get the Startup Wizard dialog box, click the **Exit** button.
 - If you are taken directly into CAD, continue as is.

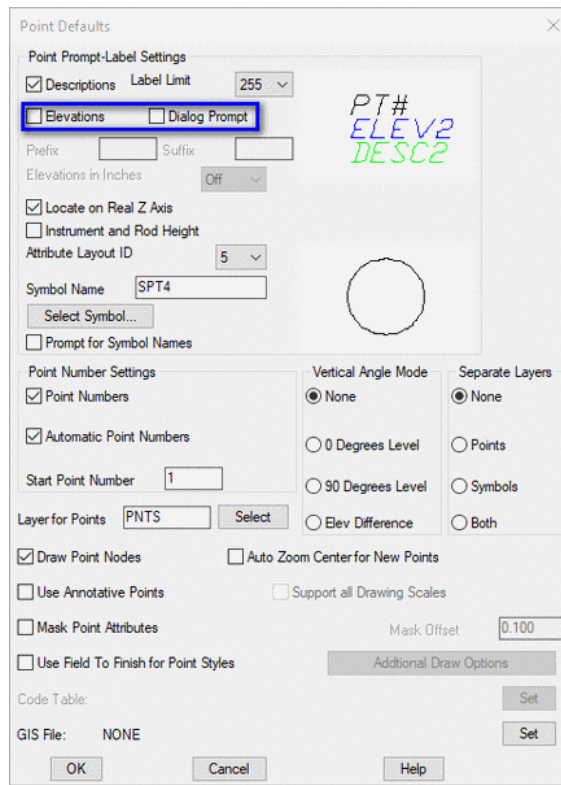
Verify the Survey module is loaded by clicking Settings – Carlson Menus – Survey Menu.

2. Issue the Settings – Drawing Setup to display a dialog box similar to that shown below:

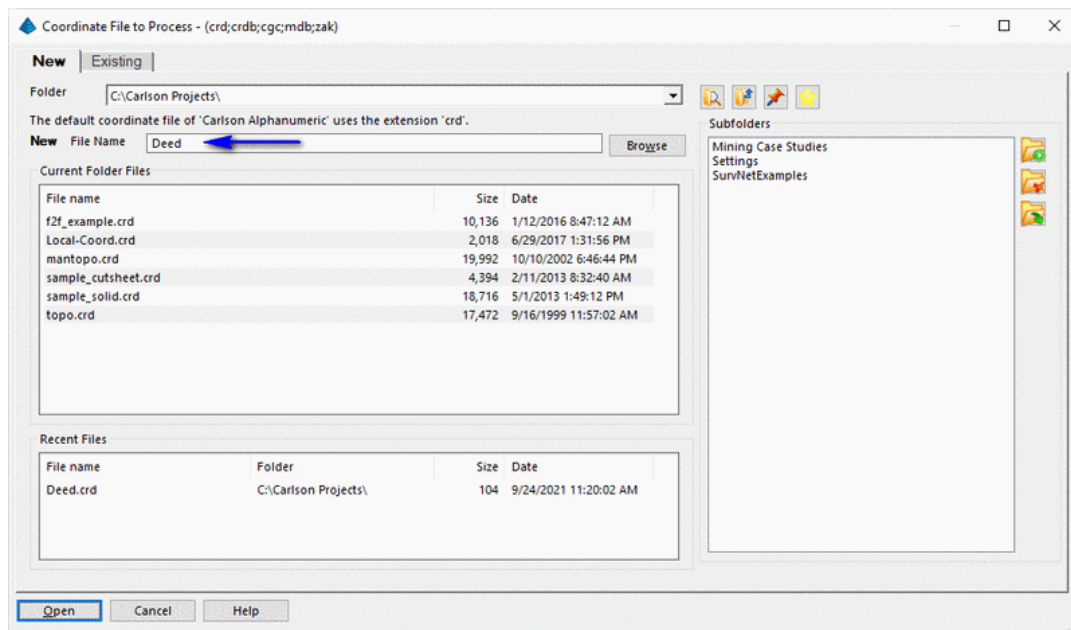


Set the unit setting to **English** and the **Horizontal Scale** to **50**. Match other settings as desired and as shown below and click **OK**.

3. Issue the Points – Point Defaults command and perform the following:
 - Clear the Elevations toggle to eliminate the Elevation prompt, and,
 - Enable the Descriptions toggle, and,
 - Use the Select Symbol button to set the Point Symbol name to symbol 4 (SPT4), which is the open circle, and,
 - Enable the Automatic Point Numbering toggle, and,
 - Match other settings as desired and as shown below and click **OK**.

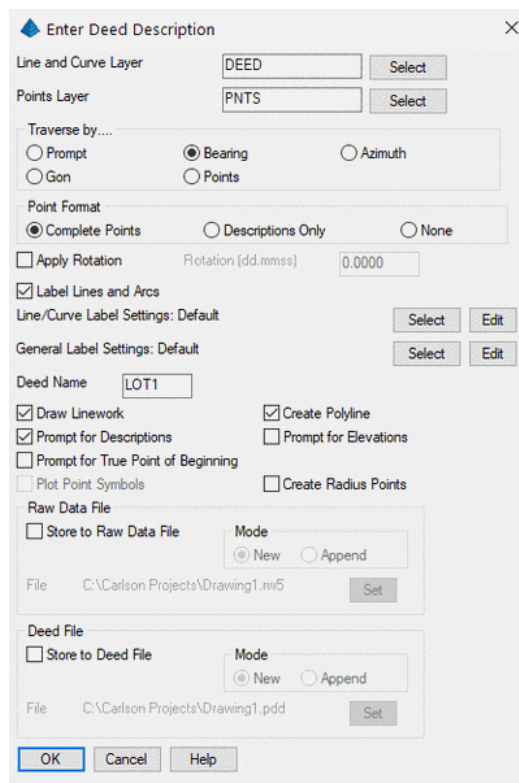


- Issue the Survey – Enter Deed Description. The following dialog box will appear so you can specify where to store the coordinates:



Select the **New** tab and set the file name as shown above which will externally house any collected or computed point locations.

- Utilize the default settings as shown below in the dialog box image:



Review and click **OK** when ready. When prompted at the Command line:

Pick point or point number: Left mouse click a point in the lower left quadrant of your screen to start the deed plotting.

Enter Point Description <>: respond with **Fence Post** and press Enter.

With our Point of Beginning established, supply the additional course information cited below:

125.3500

The quadrant (Q) is 1 for Northeast (2 is Southeast, 3 is Southwest and 4 is Northwest). The bearing is 25 degrees, 35 minutes, and 00 seconds.

If all digits for the minutes and seconds are entered as shown above, then the deed call will be fully plotted, including the seconds. If only the degrees and minutes were entered (as in 125.35), then the plot would appear as "N 25° 35' E".

Undo/Select/Varas/Meters/Rods/Chains/<Distance(ft)>:

200.51

Note that you can enter old deeds in the forms of Poles and Links, Chains and Links and even Varas (a unit of measurement formerly used in the southwestern states of the U.S.)

Enter Point Description <Fence Post>:

Iron Pin
Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<125.3500>:

189.4321

NOTE: In the data entry prompts you'll see an Undo option (*e.g.* Undo/Exit/Curve.... Pressing U for Undo would allow you to reenter the previous deed call.

Undo/Select/Varas/Meters/Rods/Chains/<Distance(ft)>:

225.00

Enter Point Description <Iron Pin>:

press Enter

Pressing Enter selects the default, which in this case is **Iron Pin**.

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<189.4321>:

press C (or c)

This initiates the Curve sub-command option.

Curvecalc/Tangent-out/<Radius>:

75

Curve direction [Left(-)/<Right(+)>]?

press Enter for right

Non-tangent/Reverse-tangent/Bearing/Chord/DeltaAng/Tangent/<Arc Len>:

118.17

If you don't know the arc length, but you know the tangent, you would choose "T" for tangent.

Enter Point Description <Iron Pin>:

press Enter

Undo/Exit/Curve/Select/Tangent/Bearing (Qdd.mmss)<c>:

200.0000

This is due South. If you were to enter just 2 (no degrees, minutes, or seconds), then the deed call would be plotted "S 000 E".

Undo/Select/Varas/Meters/Rods/Chains/<Distance(ft)>:

178.00

Enter Point Description <Iron Pin>:

Concrete Monument

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<200.0000>:

488.2300

This entry specifies Northwest 88 degrees, 23 minutes.

Undo/Select/Varas/Meters/Rods/Chains/<Distance(ft)>:

300.34

Enter Point Description <Concrete Monument>:

Fence Post

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<488.2300>:

454.1109

Undo/Select/Varas/Meters/Rods/Chains/<Distance(ft)>:

Enter Point Description <Fence Post>:

press Spacebar, then press Enter

Simply pressing Enter uses the default text (Fence Post) again. To avoid drawing the text "Fence Post" twice on the end point, press the spacebar, skip a blank character, and press Enter.

You have now completed the 6-sided figure (including one curve).

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<454.1109>:

press E

This entry specifies Exits the Deed Entry input algorithm.

Select entity for area calculation:

Graphically select the polyline representation of the deed.

Deed Description Courses

The following results are reported:

Area = 83921.758997 sq. units, Perimeter = 1128.95

SQ. FEET: 83922 SQ. YARDS: 9324.6 SQ. MILES: 0.003

ACRES: 1.9266

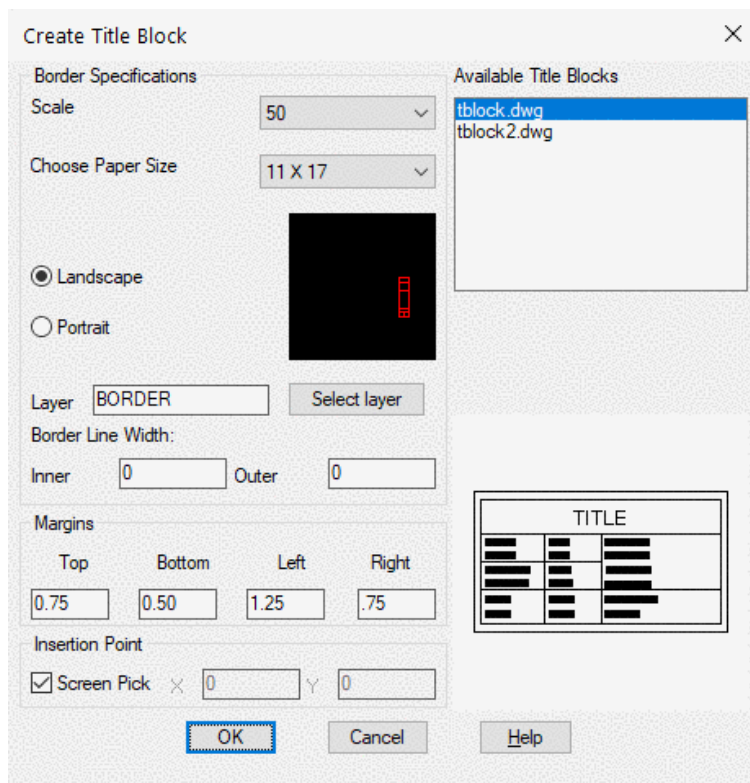
Closure error distance> 0.0171 Error Bearing> S 52°5'26" E

Closure Precision> 1 in 66076.9 Total Distance Traversed> 1128.950

The resulting deed has a closure of 1:66076.9.

Use the View – **Extents** command (key-in short-cut **ze**) to see the entire area. Then choose Zoom Out (key-in short-cut **zout**) under the View menu giving you adequate room for the next step.

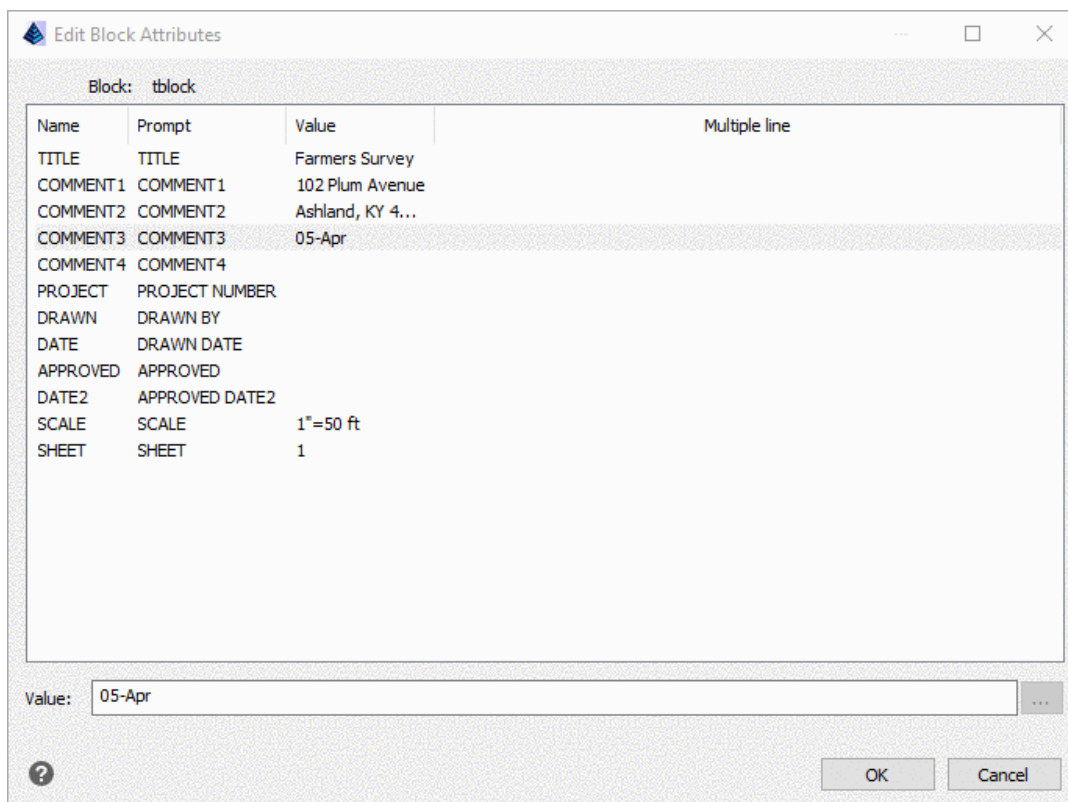
Issue the Settings – Title Block – Standard command to display a dialog box similar to that shown below:



Complete the title block entries, as shown above. Select Paper Size B2 (11 x 17), and enter the layer name of BORDER, then choose **OK** and when prompted:

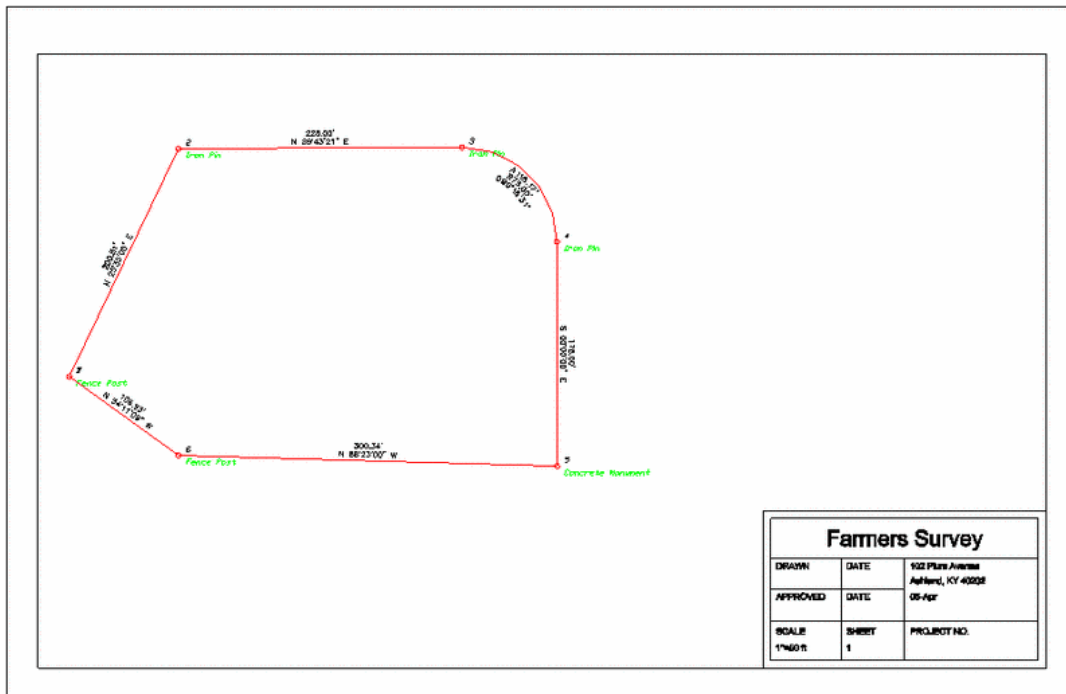
Pick location: pick a point in the lower left of the survey

The following dialog appears, allowing you to enter the attributes for the *Title Block*:



Set the values as suggested above and click **OK** when ready.

NOTE: The title line is plotted in large text on the title block. Its length, therefore, should not exceed 15 characters. Your drawing should look similar to the example below at this point:

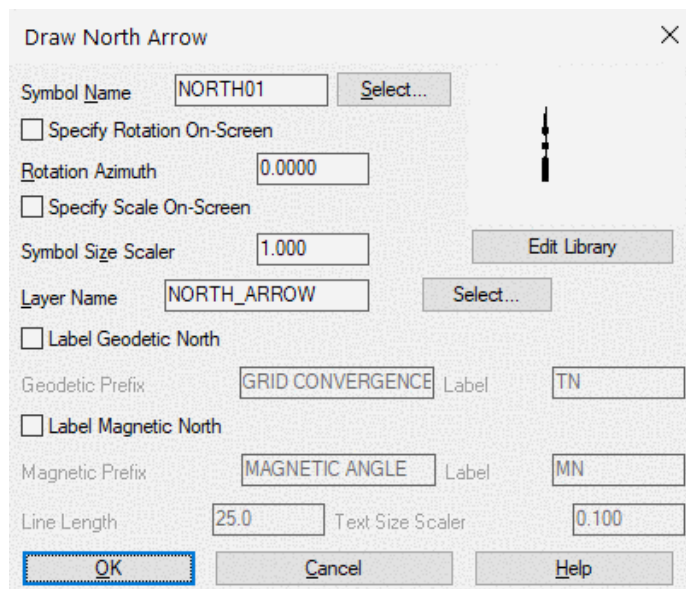


Use the View – **Extents** command to see the entire working area. If you want to move the border, use the Edit – **Move** command (keyin shortcut **M**). Pick the border lines and the title block objects (up to 3 picks total), press Enter (to end object selection), then pick two points representing the vector of the move.

If you want to see a margin around the working area after you use the Extents command, use the View – **Zoom Out** command. Then use the View – **Window** command to capture the view and margin you prefer.

If you make a mistake, enter U for Undo, or select the back arrow toolbar icon that appears at the top of the screen.

Issue the Annotate – Draw North Arrow command to display a dialog box similar to that shown below:

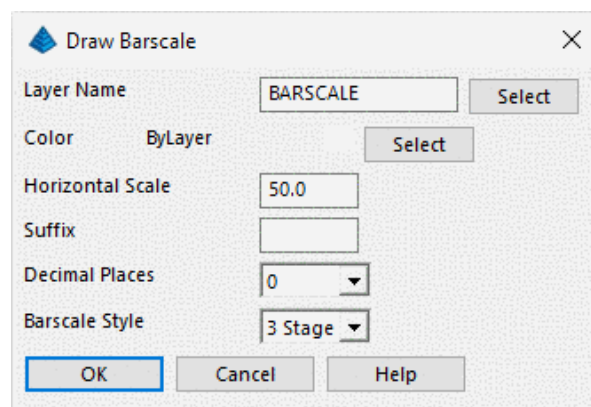


Accept the default north arrow that is shown at the right side of the dialog, click **OK**, and when prompted:

Pick insertion point: place it in the above the *Title Block* (leave enough space for the *Barscale* to follow)

If you want to reposition the North Arrow, issue the Edit – **Move** command.

Issue the Annotate – Draw Barscale to display a dialog box similar to that shown below:



Accept the defaults and click **OK** and when prompted:

Pick location for barscale: pick an insertion point below the *North Arrow* and directly above the "a" in *Farmer*, and approximately the same distance from both.

You can use the Edit – **Move** command if you want/need to re-position the barscale.

Issue the Draw – Text – **Standard** command and when prompted:

Specify start point of text or [Justify/Style]: J

Enter an option [Left/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR]: C

Center point of text: pick a point near the top-center of the drawing

Height of text <4.00>: 10 (Entering 10 makes the title text bigger than the default).

Rotation angle of text <E>: press Enter

Text: Farmer Survey

Text: Ashland, KY

Text: press Enter

To enter a certification in the lower-right of the drawing, again select Draw – Text – **Standard** (or type "dtext" at the Command line).

NOTE: If you haven't done anything else, such as Zoom or Pan, you can simply press Enter to repeat the last command. If pressing Enter does not repeat the Text command, restart the command as suggested above and respond to the resulting prompts as shown below:

Specify start point of text or [Justify/Style]: Pick a point above and to the left of the *Title Block* for the certification. You don't have to enter L for left-justification. The DText command defaults to left-justification every time.

Height <10.00>: 4

Rotation angle <E>: press Enter

Text: Surveyor's Certification

Text: *press spacebar, then press Enter*

Text: I do hereby certify that the survey shown hereon

Text: is a true and correct representation.

Text: *press spacebar, then press Enter*

Text: -----

Text: John Doe, PLS #1234

Text: *press Enter twice to end*

Enlarge the two title lines (*i.e.* **Farmer Survey** and **Surveyor's Certification**) by a factor of 2.0 using the Edit – Text – Text Enlarge/Reduce command. When prompted:

Scaling Multiplier: type 2 and press Enter

Select Text for Scaling.

[FILter/?]/<Select entities>: pick the **Farmer Survey** text and press Enter

[FILter/?]/<Select entities>: pick the **Surveyor's Certification** text and press Enter

[FILter/?]/<Select entities>: press Enter

NOTE: When you are selecting objects, if you select something you don't want, you have the option to remove undesired entities with the (R) option.

The following is a closeup of the *North arrow*, *Bar Scale* and *Certification label* that we just entered:

Surveyor's Certification

I do hereby certify that the survey shown hereon
is a true and correct representation.

John Doe, PLS #1234

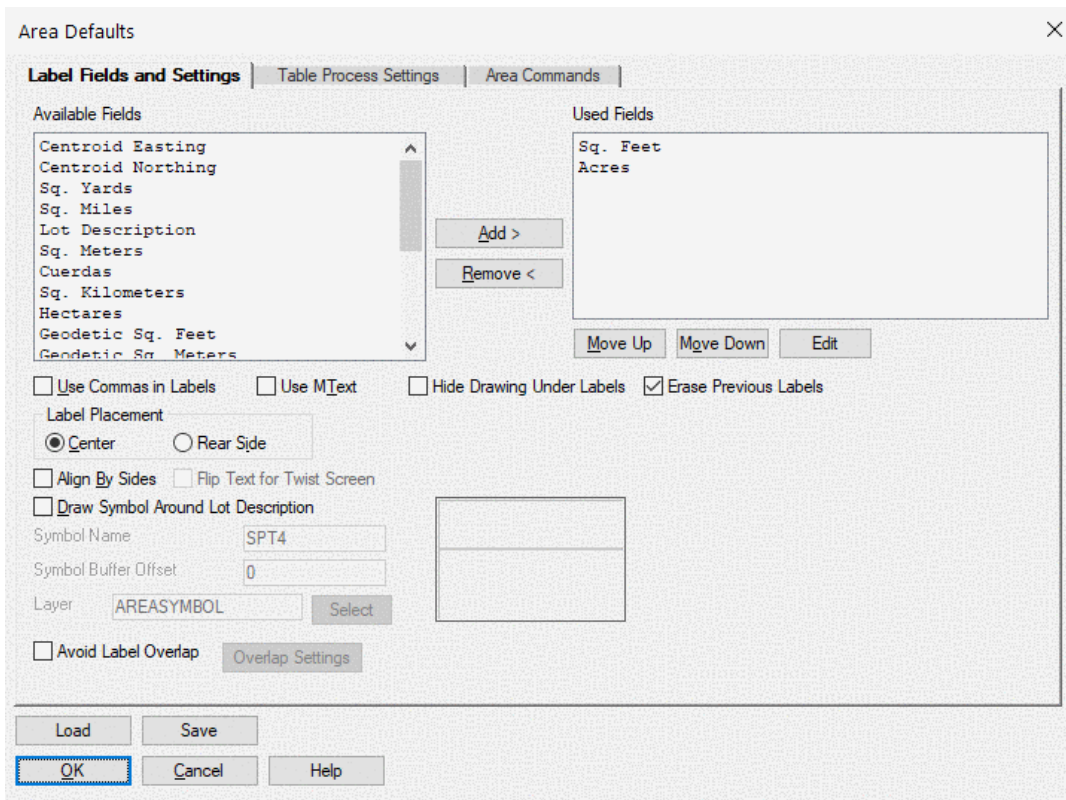
Farmers Survey		
DRAWN	DATE	102 Plum Avenue Ashland, KY 40202
APPROVED	DATE	05-Apr
SCALE 1"=50 ft	SHEET 1	PROJECT NO.

Our next task will be to refine a bearing label; click the Edit – Text – **Edit Text** command. When you are prompted:

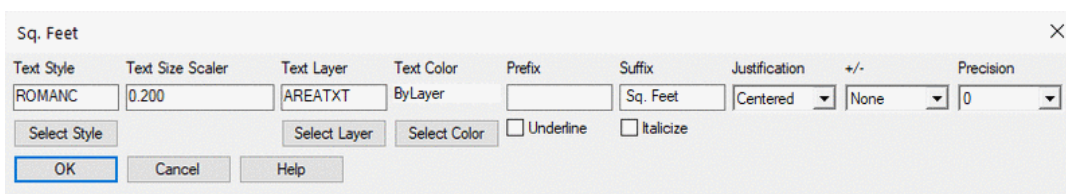
Select Text to Edit: pick the due South bearing text of S 00°00'00" E and change it to S 00° E.

NOTE: The degree symbol in CAD *SHX* fonts is represented as `%%d`. For example, if you had typed N 15%%d25'35" E in the *DText* command, Carlson would draw that entry as N15°25'35"E. Click in the text to the immediate right of the quotation mark and press the Backspace key until the text reads "S 00%%d E and press Enter (or click **OK** if in a dialog box) to exit the command.

Prepare for area labeling by selecting the Area/Layout – Area Defaults command. The dialog box shown below appears:



Select the **Sq. Feet** item and click on the **Edit** button and make the **Area Text Size Scaler** 0.2 (doubled from the default of 0.1) as shown below. Click **OK** when complete.



Next select the **Acres** item and click on the **Edit** button. Also, make the **Area Text Size Scaler** 0.2 and click **OK** to dismiss the *Acres* dialog box and click **OK** to dismiss the *Area Defaults* dialog box.

You are going to compute the area by point number by inverting from 1 through 7 and back to 1. Select the Area/Layout – Inverse with Area and respond to the prompts as shown below:

¹
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

2-3

Supplying a point range (separated by a dash) is very helpful when dealing with many consecutive straight-leg courses.

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

r

Point 3 is the PC of the curve so we need to find the radius point of the curve.

Radius point number or pick point:

cen

We wish to use a CENter snap.

Snap to CEN of:

pick the arc-portion of the polyline
Curve direction [Left/<Right>]?

r

Pressing Enter would have also
executed the default <Right> option.

Pick End of Arc or point number (U-Undo,Enter to end):

4

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

5-6

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

1

We are establishing the desire to
Inverse back to the starting point.

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

press Enter to end.

Inverse with Area

A Standard Report Viewer dialog box showing the *Inverse with Area* results will appear. Click the **Exit** (Doorway) button at the top of the dialog box and also note the results to the Command window:

Area = 83921.725598 sq. units, Perimeter = 1128.97
SQ. FEET: 83922 SQ. YARDS: 9324.6 SQ. MILES: 0.003
ACRES: 1.9266 PERIMETER: 1129.0
You should be prompted:

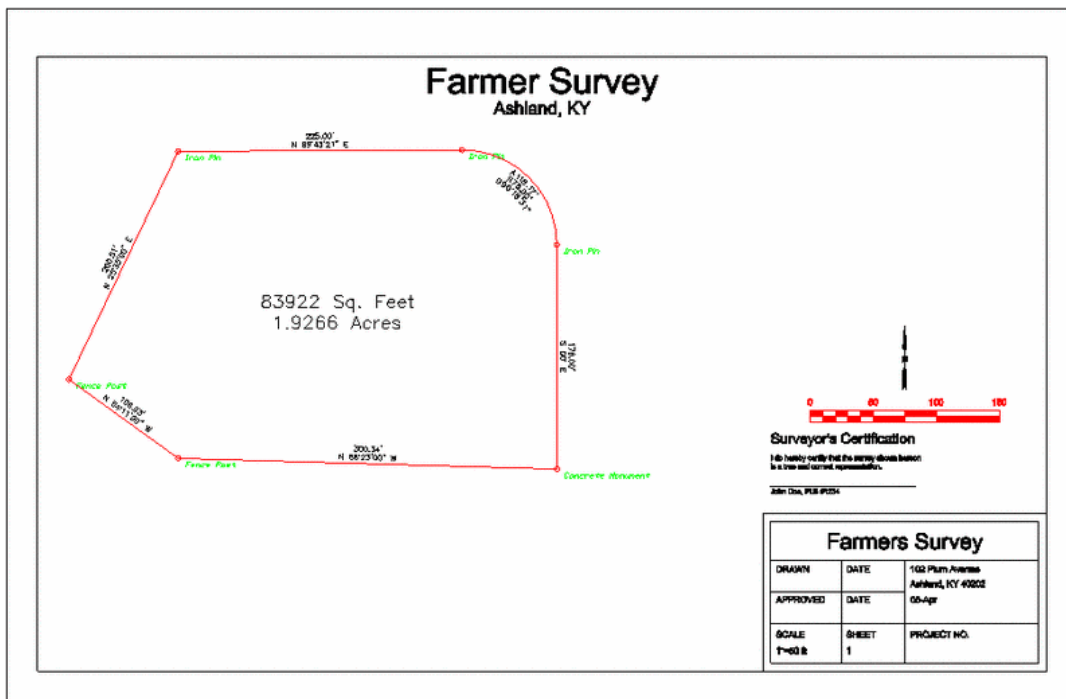
Pick area label centering point (Enter for none): Pick a point near the center of the figure, in its interior. The area units you chose in Area Defaults are labeled on the screen.

You snapped to the radius point using the manually keyed "cen" snap. Additional object snaps appear under Settings – Object Snap. Since all plotted points have a node, you could have inverted around this figure by using the "nod" snap for points 1 through 7, and the "cen" snap to capture the radius point. Snaps are typically entered at the keyboard as 3 characters (for example, "int" for intersect and "end" for endpoint).

Also, you could have chosen the Area by Lines & Arcs command. In that command, you would pick the lines and arcs that make up the figure. But since the closure was 0.017 off (the distance from point 7 to point 1), you would exceed the default Max gap tolerance. Unless you change that tolerance in the command to something larger than 0.017, you would not get a result using this method.

Freeze the point numbers to finish the drawing by choosing View – Layer Control. In the **PNTNO** row, click the Sun icon to change it to a Snowflake icon, which freezes the PNTNO layer. Click **OK**. The point numbers remain in the drawing, waiting to be "thawed", but they are not displayed.

The final drawing is shown here:



This completes the Lesson 1 tutorial: Entering a Deed.

_____ .r4440

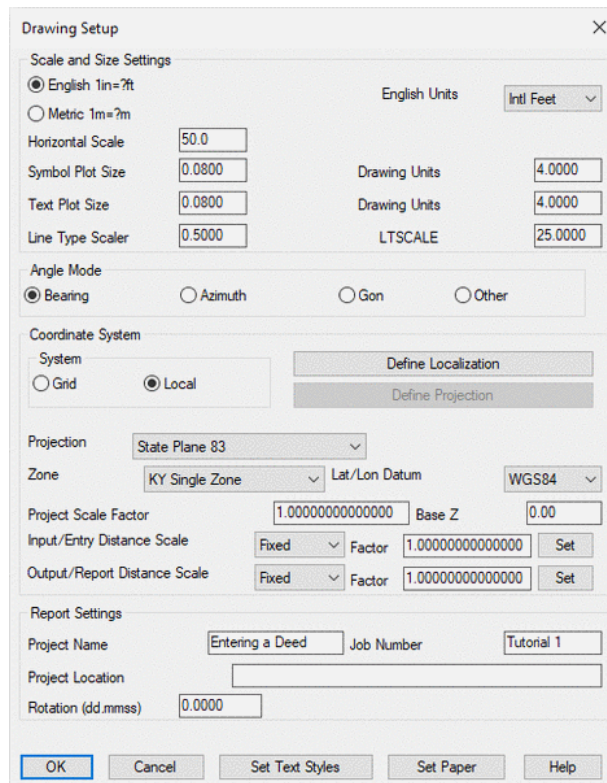
In this short lesson you will create a simple drawing. You will enter a 6-sided deed, add a title block, bar scale, and north arrow, add a title and certification text, and plot the deed area.

Note that the Esc key will cancel most commands, so if you choose the wrong command or enter something incorrectly and want to start over, just press Esc.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **New Drawing**.
 - If you get the Startup Wizard dialog box, click the **Exit** button.
 - If you are taken directly into CAD, continue as is.

Verify the Survey module is loaded by clicking Settings – Carlson Menus – Survey Menu.

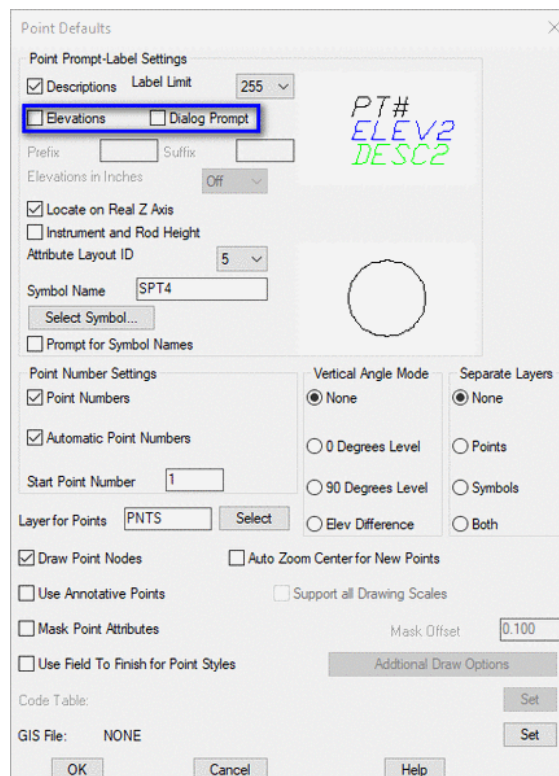
2. Issue the Settings – Drawing Setup to display a dialog box similar to that shown below:



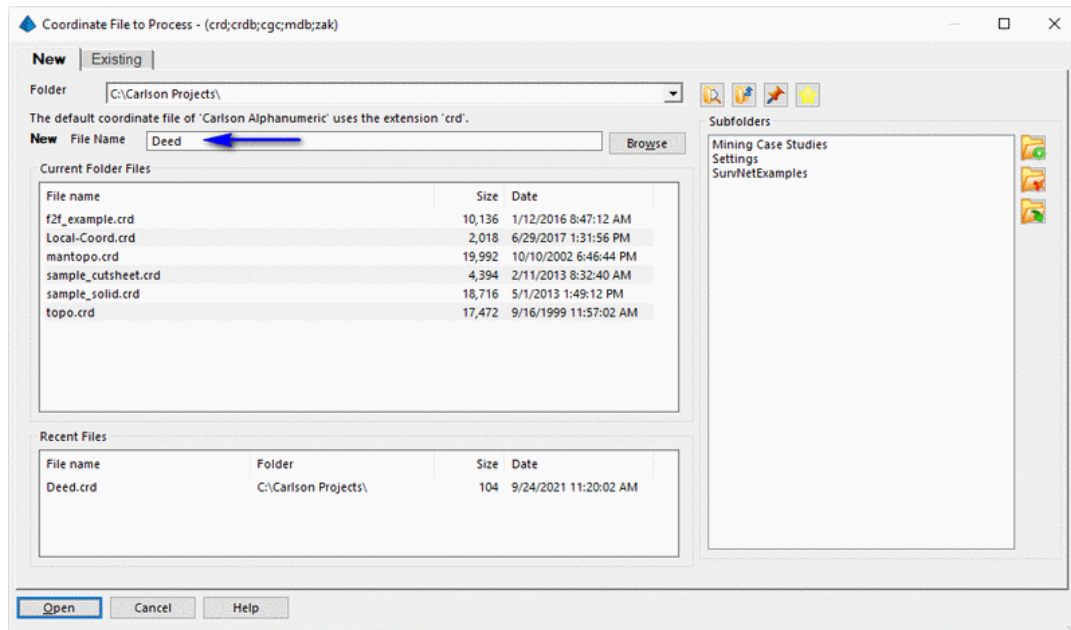
Set the unit setting to **English** and the **Horizontal Scale** to **50**. Match other settings as desired and as shown below and click **OK**.

3. Issue the Points – Point Defaults command and perform the following:

- Clear the Elevations toggle to eliminate the Elevation prompt, and,
- Enable the Descriptions toggle, and,
- Use the Select Symbol button to set the Point Symbol name to symbol 4 (SPT4), which is the open circle, and,
- Enable the Automatic Point Numbering toggle, and,
- Match other settings as desired and as shown below and click **OK**.

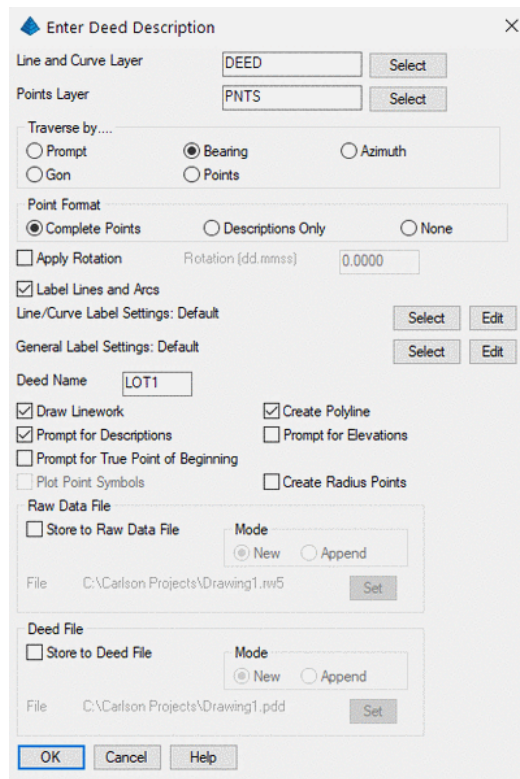


- Issue the Survey – Enter Deed Description. The following dialog box will appear so you can specify where to store the coordinates:



Select the **New** tab and set the file name as shown above which will externally house any collected or computed point locations.

- Utilize the default settings as shown below in the dialog box image:



Review and click **OK** when ready. When prompted at the Command line:

Pick point or point number: Left mouse click a point in the lower left quadrant of your screen to start the deed plotting.

Enter Point Description <>: respond with **Fence Post** and press Enter.

With our Point of Beginning established, supply the additional course information cited below:

125.3500

The quadrant (Q) is 1 for Northeast (2 is Southeast, 3 is Southwest and 4 is Northwest). The bearing is 25 degrees, 35 minutes, and 00 seconds.

If all digits for the minutes and seconds are entered as shown above, then the deed call will be fully plotted, including the seconds. If only the degrees and minutes were entered (as in 125.35), then the plot would appear as "N 25° 35' E".

Undo/Select/Varas/Meters/Rods/Chains/Distance(ft):

200.51

Note that you can enter old deeds in the forms of Poles and Links, Chains and Links and even Varas (a unit of measurement formerly used in the southwestern states of the U.S.)

Enter Point Description <Fence Post>:

Iron Pin

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<125.3500>:

189.4321

NOTE: In the data entry prompts you'll see an Undo option (*e.g.* Undo/Exit/Curve.... Pressing U for Undo would allow you to reenter the previous deed call.

Undo/Select/Varas/Meters/Rods/Chains/Distance(ft):

225.00

Enter Point Description <Iron Pin>:

press Enter

Pressing Enter selects the default, which in this case is **Iron Pin**.

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<189.4321>:

press C (or c)

This initiates the Curve sub-command option.

Curvecalc/Tangent-out/Radius>:

Curve direction [Left(-)/Right(+)]?

press Enter for right

Non-tangent/Reverse-tangent/Bearing/Chord/DeltaAng/Tangent/Arc Len>:

118.17

If you don't know the arc length, but you know the tangent, you would choose "T" for tangent.

Enter Point Description <Iron Pin>:

press Enter

Undo/Exit/Curve/Select/Tangent/Bearing (Qdd.mmss)<c>:

200.0000

This is due South. If you were to enter just 2 (no degrees, minutes, or seconds), then the deed call would be plotted "S 000 E".

Undo/Select/Varas/Meters/Rods/Chains/<Distance(ft)>:

178.00

Enter Point Description <Iron Pin>:

Concrete Monument

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<200.0000>:

488.2300

This entry specifies Northwest 88 degrees, 23 minutes.

Undo/Select/Varas/Meters/Rods/Chains/<Distance(ft)>:

300.34

Enter Point Description <Concrete Monument>:

Fence Post

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<488.2300>:

454.1109

Undo/Select/Varas/Meters/Rods/Chains/<Distance(ft)>:

106.93

Enter Point Description <Fence Post>:

press Spacebar, then press Enter

Simply pressing Enter uses the default text (Fence Post) again. To avoid drawing the text "Fence Post" twice on the end point, press the spacebar, skip a blank character, and press Enter.

You have now completed the 6-sided figure (including one curve).

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<454.1109>:

press E

This entry specifies Exits the Deed Entry input algorithm.

Select entity for area calculation:

Graphically select the polyline representation of the deed.

Deed Description Courses

The following results are reported:

Area = 83921.758997 sq. units, Perimeter = 1128.95

SQ. FEET: 83922 SQ. YARDS: 9324.6 SQ. MILES: 0.003

ACRES: 1.9266

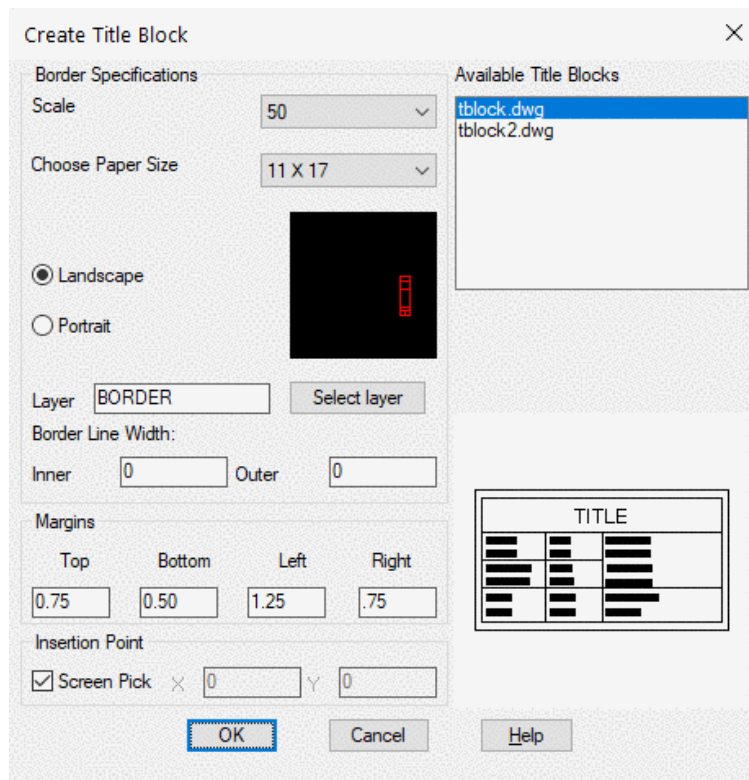
Closure error distance> 0.0171 Error Bearing> S 52Â°5'26" E

Closure Precision> 1 in 66076.9 Total Distance Traversed> 1128.950

The resulting deed has a closure of 1:66076.9.

Use the View – **Extents** command (key-in short-cut **ze**) to see the entire area. Then choose Zoom Out (key-in short-cut **zout**) under the View menu giving you adequate room for the next step.

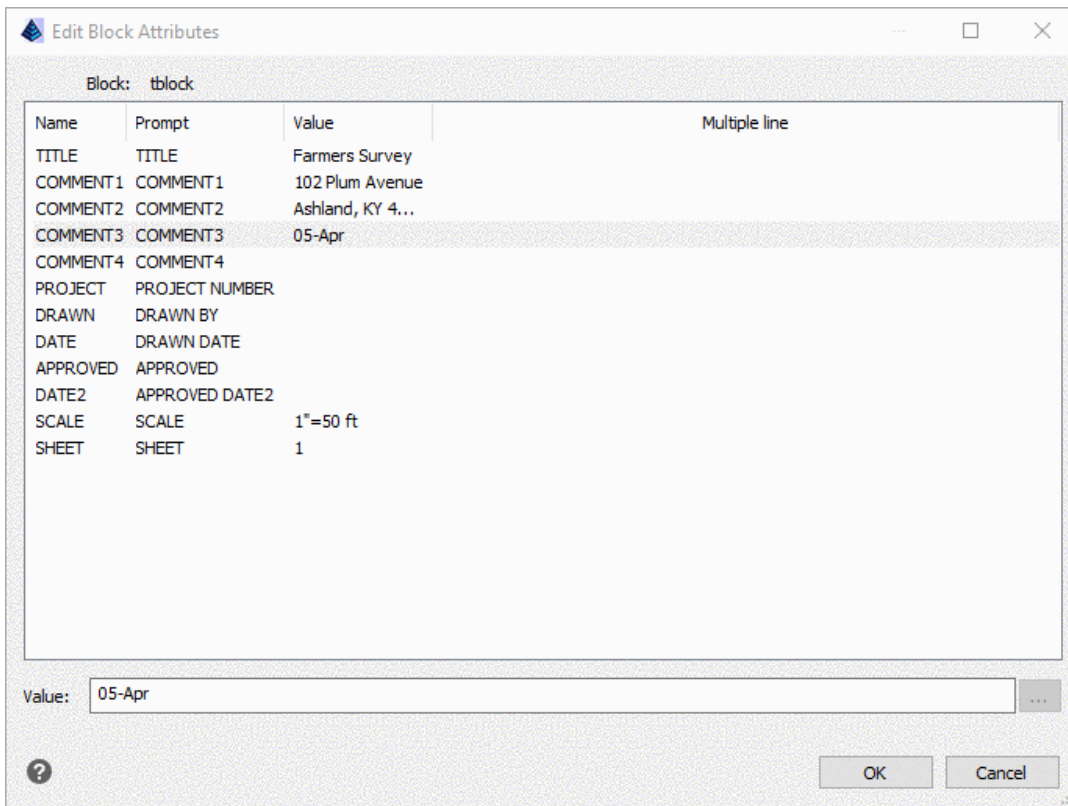
Issue the Settings – Title Block – Standard command to display a dialog box similar to that shown below:



Complete the title block entries, as shown above. Select Paper Size B2 (11 x 17), and enter the layer name of BORDER, then choose **OK** and when prompted:

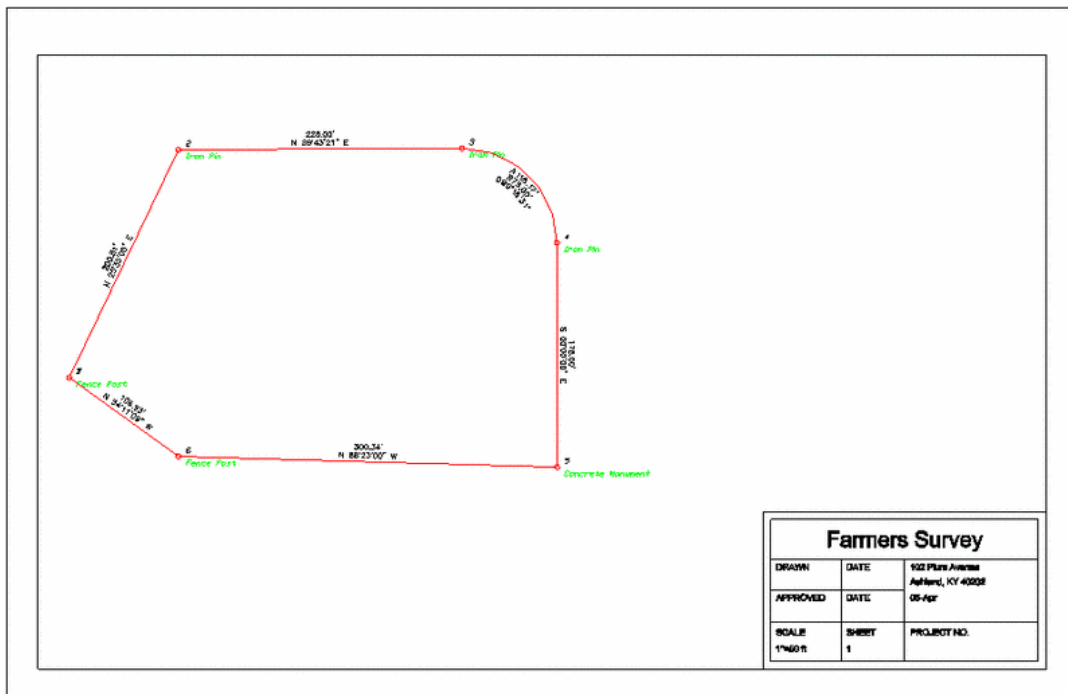
Pick location: pick a point in the lower left of the survey

The following dialog appears, allowing you to enter the attributes for the *Title Block*:



Set the values as suggested above and click **OK** when ready.

NOTE: The title line is plotted in large text on the title block. Its length, therefore, should not exceed 15 characters. Your drawing should look similar to the example below at this point:

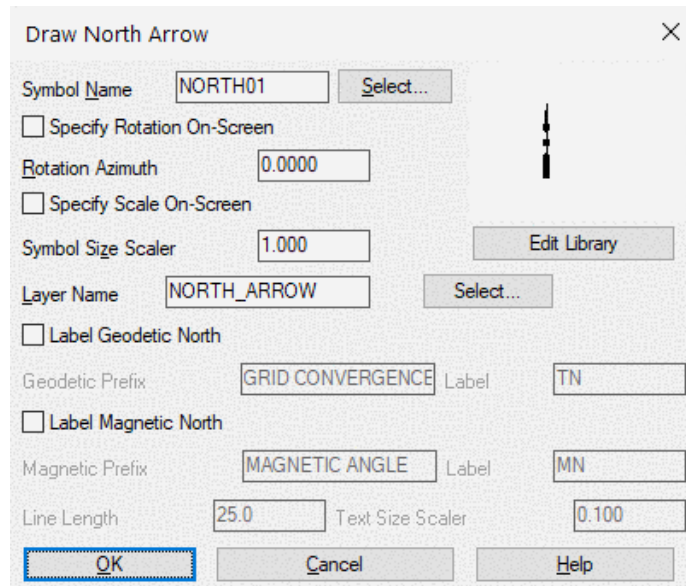


Use the View – **Extents** command to see the entire working area. If you want to move the border, use the Edit – **Move** command (keyin shortcut **M**). Pick the border lines and the title block objects (up to 3 picks total), press Enter (to end object selection), then pick two points representing the vector of the move.

If you want to see a margin around the working area after you use the Extents command, use the View – **Zoom Out** command. Then use the View – **Window** command to capture the view and margin you prefer.

If you make a mistake, enter U for Undo, or select the back arrow toolbar icon that appears at the top of the screen.

Issue the Annotate – Draw North Arrow command to display a dialog box similar to that shown below:

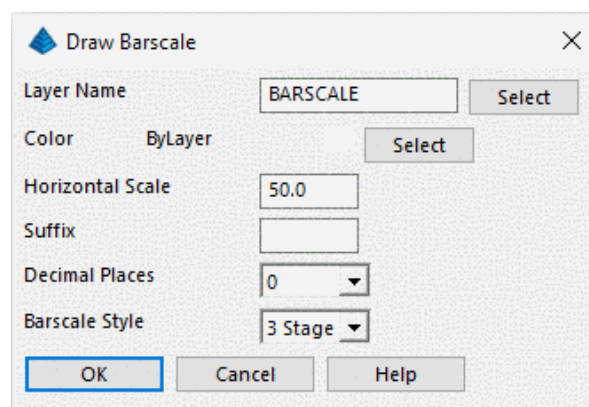


Accept the default north arrow that is shown at the right side of the dialog, click **OK**, and when prompted:

Pick insertion point: place it in the above the *Title Block* (leave enough space for the *Barscale* to follow)

If you want to reposition the North Arrow, issue the Edit – **Move** command.

Issue the Annotate – Draw Barscale to display a dialog box similar to that shown below:



Accept the defaults and click **OK** and when prompted:

Pick location for barscale: pick an insertion point below the *North Arrow* and directly above the "a" in *Farmer*, and approximately the same distance from both.

You can use the Edit – **Move** command if you want/need to re-position the barscale.

Issue the Draw – Text – **Standard** command and when prompted:

Specify start point of text or [Justify/Style]: J

Enter an option [Left/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR]: C

Center point of text: pick a point near the top-center of the drawing

Height of text <4.00>: 10 (Entering 10 makes the title text bigger than the default).

Rotation angle of text <E>: press Enter

Text: Farmer Survey

Text: Ashland, KY

Text: press Enter

To enter a certification in the lower-right of the drawing, again select Draw – Text – **Standard** (or type "dtext" at the Command line).

NOTE: If you haven't done anything else, such as Zoom or Pan, you can simply press Enter to repeat the last command. If pressing Enter does not repeat the Text command, restart the command as suggested above and respond to the resulting prompts as shown below:

Specify start point of text or [Justify/Style]: Pick a point above and to the left of the *Title Block* for the certification. You don't have to enter L for left-justification. The DText command defaults to left-justification every time.

Height <10.00>: 4

Rotation angle <E>: press Enter

Text: Surveyor's Certification

Text: *press spacebar, then press Enter*

Text: I do hereby certify that the survey shown hereon

Text: is a true and correct representation.

Text: *press spacebar, then press Enter*

Text: -----

Text: John Doe, PLS #1234

Text: *press Enter twice to end*

Enlarge the two title lines (*i.e.* **Farmer Survey** and **Surveyor's Certification**) by a factor of 2.0 using the Edit – Text – Text Enlarge/Reduce command. When prompted:

Scaling Multiplier: type 2 and press Enter

Select Text for Scaling.

[FILter/?]/<Select entities>: pick the **Farmer Survey** text and press Enter

[FILter/?]/<Select entities>: pick the **Surveyor's Certification** text and press Enter

[FILter/?]/<Select entities>: press Enter

NOTE: When you are selecting objects, if you select something you don't want, you have the option to remove undesired entities with the (R) option.

The following is a closeup of the *North arrow*, *Bar Scale* and *Certification label* that we just entered:

Surveyor's Certification

I do hereby certify that the survey shown hereon
is a true and correct representation.

John Doe, PLS #1234

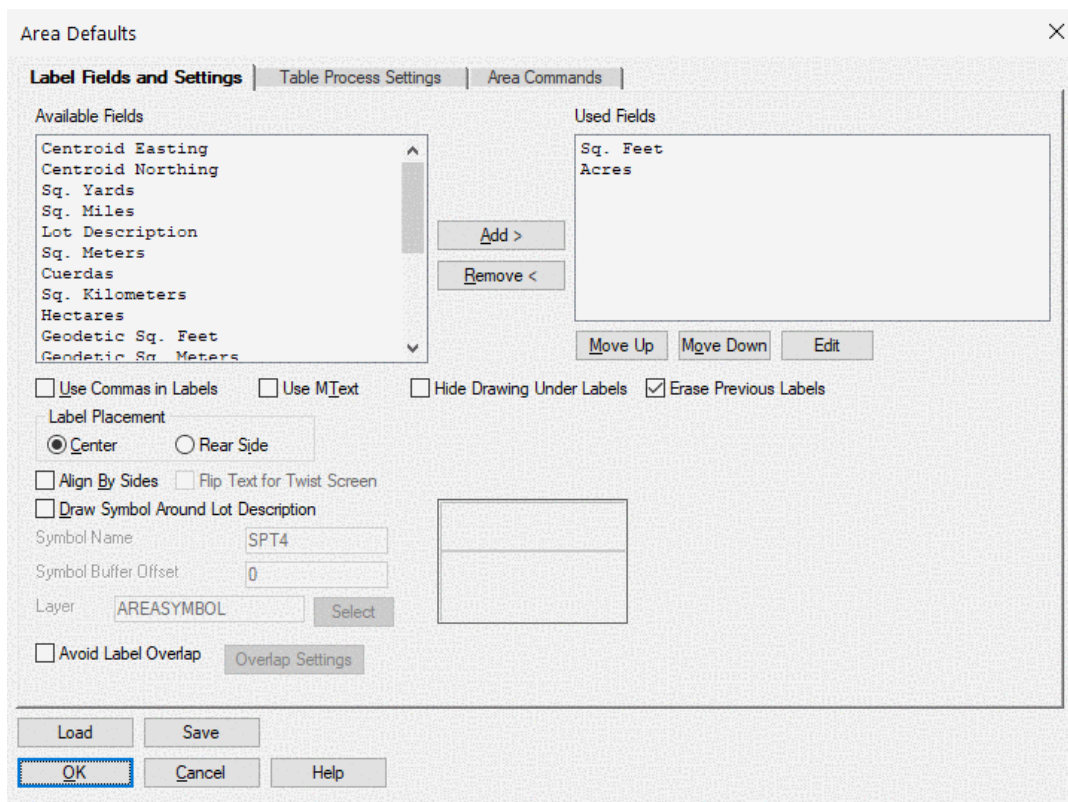
Farmers Survey		
DRAWN	DATE	102 Plum Avenue Ashland, KY 40202
APPROVED	DATE	05-Apr
SCALE 1"=50 ft	SHEET 1	PROJECT NO.

Our next task will be to refine a bearing label; click the Edit – Text – **Edit Text** command. When you are prompted:

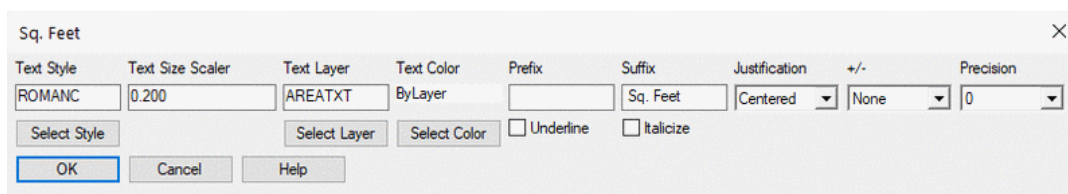
Select Text to Edit: pick the due South bearing text of S 00°00'00" E and change it to S 00° E.

NOTE: The degree symbol in CAD *SHX* fonts is represented as `%%d`. For example, if you had typed N 15%%d25'35" E in the *DText* command, Carlson would draw that entry as N15°25'35"E. Click in the text to the immediate right of the quotation mark and press the Backspace key until the text reads "S 00%%d E and press Enter (or click **OK** if in a dialog box) to exit the command.

Prepare for area labeling by selecting the Area/Layout – Area Defaults command. The dialog box shown below appears:



Select the **Sq. Feet** item and click on the **Edit** button and make the **Area Text Size Scaler** 0.2 (doubled from the default of 0.1) as shown below. Click **OK** when complete.



Next select the **Acres** item and click on the **Edit** button. Also, make the **Area Text Size Scaler** 0.2 and click **OK** to dismiss the *Acres* dialog box and click **OK** to dismiss the *Area Defaults* dialog box.

You are going to compute the area by point number by inverting from 1 through 7 and back to 1. Select the Area/Layout – Inverse with Area and respond to the prompts as shown below:

¹
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

2-3

Supplying a point range (separated by a dash) is very helpful when dealing with many consecutive straight-leg courses.

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

r

Point 3 is the PC of the curve so we need to find the radius point of the curve.

Radius point number or pick point:

cen

We wish to use a CENter snap.

Snap to CEN of:

pick the arc-portion of the polyline
Curve direction [Left/<Right>]?

r

Pressing Enter would have also
executed the default <Right> option.

Pick End of Arc or point number (U-Undo,Enter to end):

4

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

5-6

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

1

We are establishing the desire to
Inverse back to the starting point.

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

press Enter to end.

Inverse with Area

A Standard Report Viewer dialog box showing the *Inverse with Area* results will appear. Click the **Exit** (Doorway) button at the top of the dialog box and also note the results to the Command window:

Area = 83921.725598 sq. units, Perimeter = 1128.97
SQ. FEET: 83922 SQ. YARDS: 9324.6 SQ. MILES: 0.003
ACRES: 1.9266 PERIMETER: 1129.0
You should be prompted:

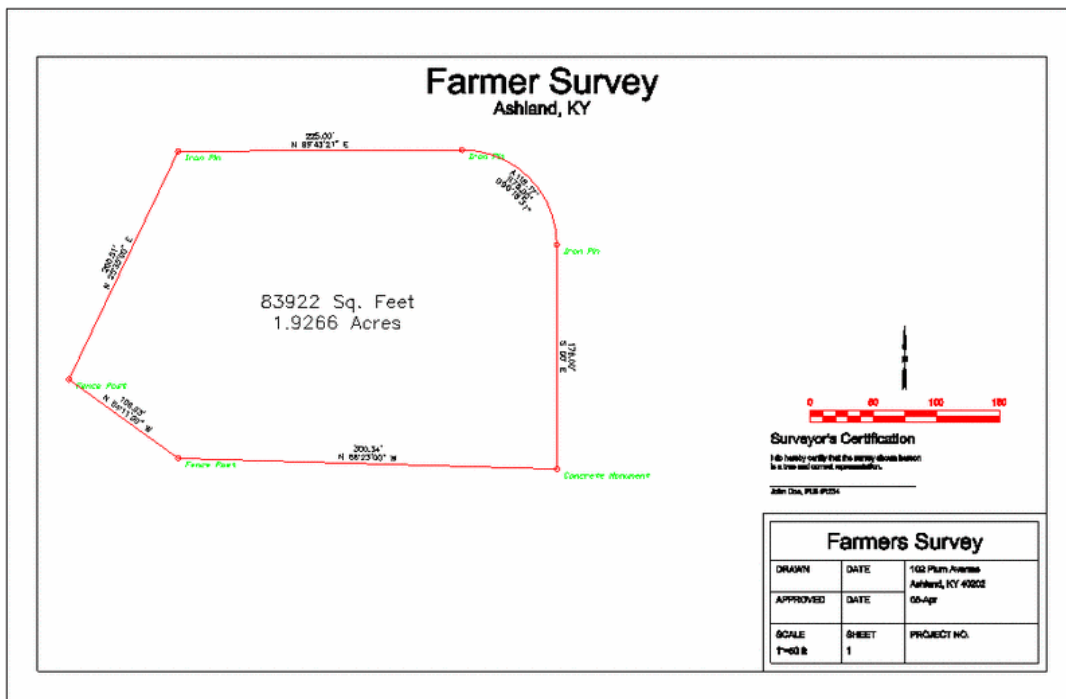
Pick area label centering point (Enter for none): Pick a point near the center of the figure, in its interior. The area units you chose in Area Defaults are labeled on the screen.

You snapped to the radius point using the manually keyed "cen" snap. Additional object snaps appear under Settings – Object Snap. Since all plotted points have a node, you could have inversed around this figure by using the "nod" snap for points 1 through 7, and the "cen" snap to capture the radius point. Snaps are typically entered at the keyboard as 3 characters (for example, "int" for intersect and "end" for endpoint).

Also, you could have chosen the Area by Lines & Arcs command. In that command, you would pick the lines and arcs that make up the figure. But since the closure was 0.017 off (the distance from point 7 to point 1), you would exceed the default Max gap tolerance. Unless you change that tolerance in the command to something larger than 0.017, you would not get a result using this method.

Freeze the point numbers to finish the drawing by choosing View – Layer Control. In the **PNTNO** row, click the Sun icon to change it to a Snowflake icon, which freezes the PNTNO layer. Click **OK**. The point numbers remain in the drawing, waiting to be "thawed", but they are not displayed.

The final drawing is shown here:



This completes the Lesson 1 tutorial: Entering a Deed.

=====

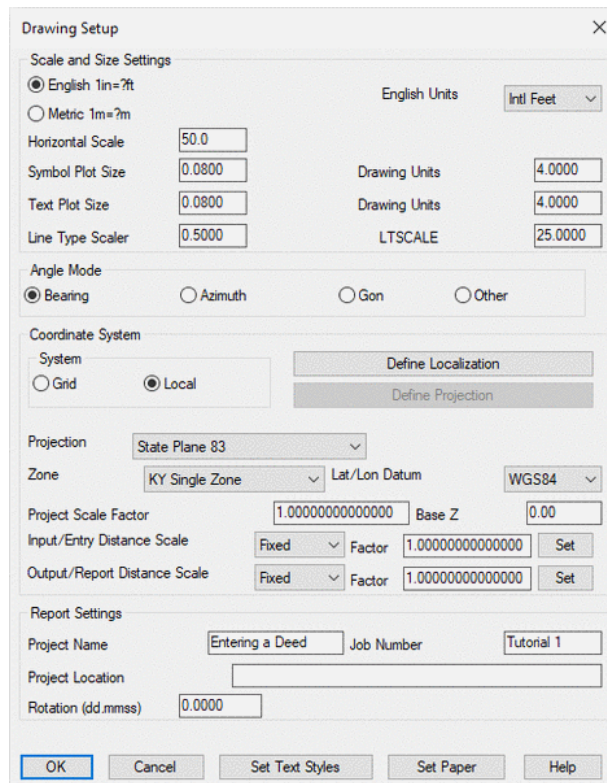
In this short lesson you will create a simple drawing. You will enter a 6-sided deed, add a title block, bar scale, and north arrow, add a title and certification text, and plot the deed area.

Note that the Esc key will cancel most commands, so if you choose the wrong command or enter something incorrectly and want to start over, just press Esc.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **New Drawing**.
 - If you get the Startup Wizard dialog box, click the **Exit** button.
 - If you are taken directly into CAD, continue as is.

Verify the Survey module is loaded by clicking Settings – Carlson Menus – Survey Menu.

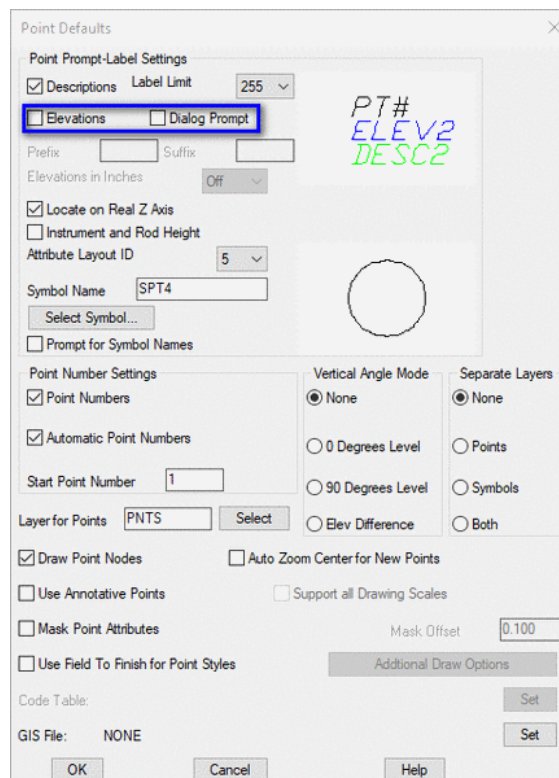
2. Issue the Settings – Drawing Setup to display a dialog box similar to that shown below:



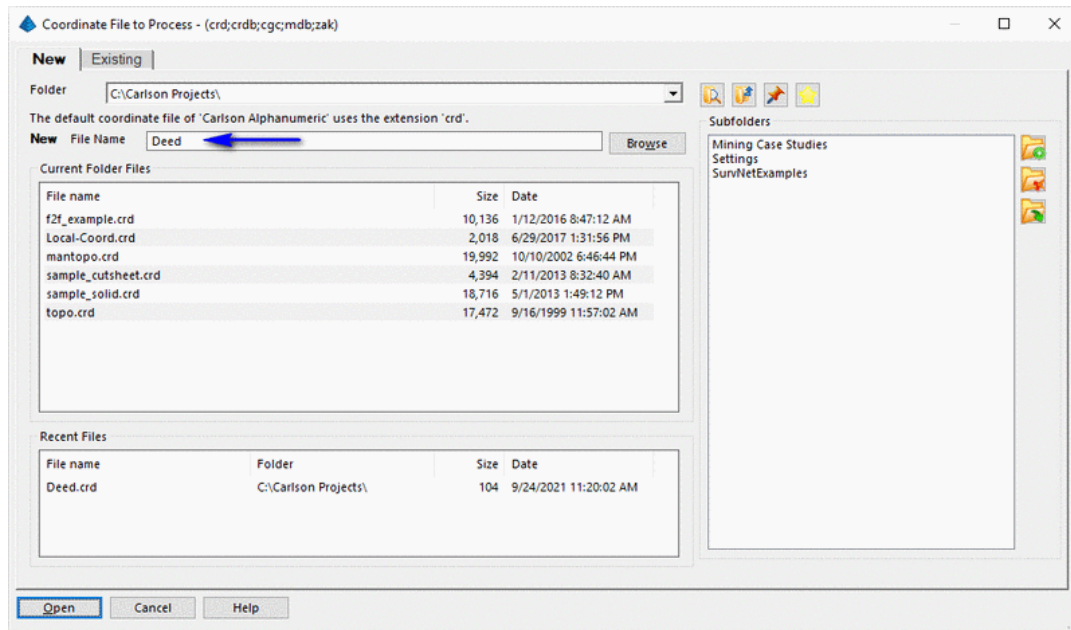
Set the unit setting to **English** and the **Horizontal Scale** to **50**. Match other settings as desired and as shown below and click **OK**.

3. Issue the Points – Point Defaults command and perform the following:

- Clear the Elevations toggle to eliminate the Elevation prompt, and,
- Enable the Descriptions toggle, and,
- Use the Select Symbol button to set the Point Symbol name to symbol 4 (SPT4), which is the open circle, and,
- Enable the Automatic Point Numbering toggle, and,
- Match other settings as desired and as shown below and click **OK**.

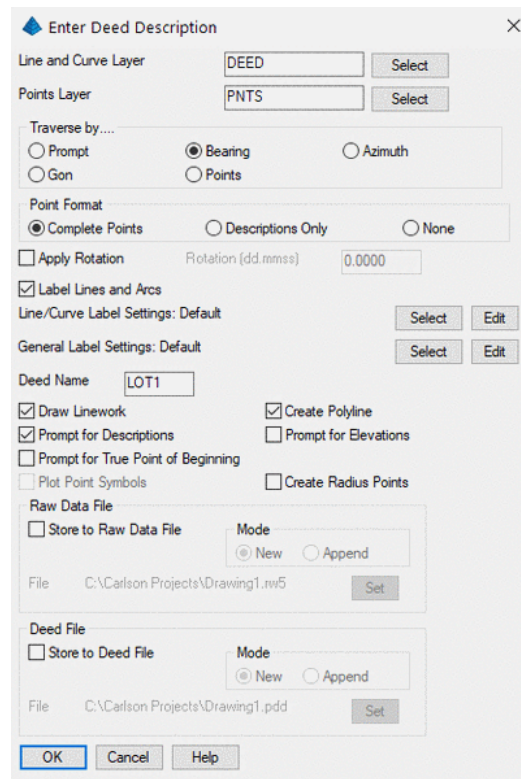


- Issue the Survey – Enter Deed Description. The following dialog box will appear so you can specify where to store the coordinates:



Select the **New** tab and set the file name as shown above which will externally house any collected or computed point locations.

- Utilize the default settings as shown below in the dialog box image:



Review and click **OK** when ready. When prompted at the Command line:

Pick point or point number: Left mouse click a point in the lower left quadrant of your screen to start the deed plotting.

Enter Point Description <>: respond with **Fence Post** and press Enter.

With our Point of Beginning established, supply the additional course information cited below:

125.3500

The quadrant (Q) is 1 for Northeast (2 is Southeast, 3 is Southwest and 4 is Northwest). The bearing is 25 degrees, 35 minutes, and 00 seconds.

If all digits for the minutes and seconds are entered as shown above, then the deed call will be fully plotted, including the seconds. If only the degrees and minutes were entered (as in 125.35), then the plot would appear as "N 25° 35' E".

Undo/Select/Varas/Meters/Rods/Chains/Distance(ft):

200.51

Note that you can enter old deeds in the forms of Poles and Links, Chains and Links and even Varas (a unit of measurement formerly used in the southwestern states of the U.S.)

Enter Point Description <Fence Post>:

Iron Pin

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<125.3500>:

189.4321

NOTE: In the data entry prompts you'll see an Undo option (*e.g.* Undo/Exit/Curve.... Pressing U for Undo would allow you to reenter the previous deed call.

Undo/Select/Varas/Meters/Rods/Chains/Distance(ft):

225.00

Enter Point Description <Iron Pin>:

press Enter

Pressing Enter selects the default, which in this case is **Iron Pin**.

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<189.4321>:

press C (or c)

This initiates the Curve sub-command option.

Curvecalc/Tangent-out/Radius>:

Curve direction [Left(-)/Right(+)]?

press Enter for right

Non-tangent/Reverse-tangent/Bearing/Chord/DeltaAng/Tangent/Arc Len>:

118.17

If you don't know the arc length, but you know the tangent, you would choose "T" for tangent.

Enter Point Description <Iron Pin>:

press Enter

Undo/Exit/Curve/Select/Tangent/Bearing (Qdd.mmss)<c>:

200.0000 This is due South. If you were to enter just 2 (no degrees, minutes, or seconds), then the deed call would be plotted "S 000 E".

Undo/Select/Varas/Meters/Rods/Chains/<Distance(ft)>:

178.00
Enter Point Description <Iron Pin>:

Concrete Monument
Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<200.0000>:

488.2300 This entry specifies Northwest 88 degrees, 23 minutes.

Undo/Select/Varas/Meters/Rods/Chains/<Distance(ft)>:

300.34
Enter Point Description <Concrete Monument>:

Fence Post
Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<488.2300>:

454.1109
Undo/Select/Varas/Meters/Rods/Chains/<Distance(ft)>:

106.93
Enter Point Description <Fence Post>:

press Spacebar, then press Enter Simply pressing Enter uses the default text (Fence Post) again. To avoid drawing the text "Fence Post" twice on the end point, press the spacebar, skip a blank character, and press Enter.

You have now completed the 6-sided figure (including one curve).

Undo/Exit/Curve/Select/Bearing (Qdd.mmss)<454.1109>:

press E This entry specifies Exits the Deed Entry input algorithm.

Select entity for area calculation:

Graphically select the polyline representation of the deed.

Deed Description Courses

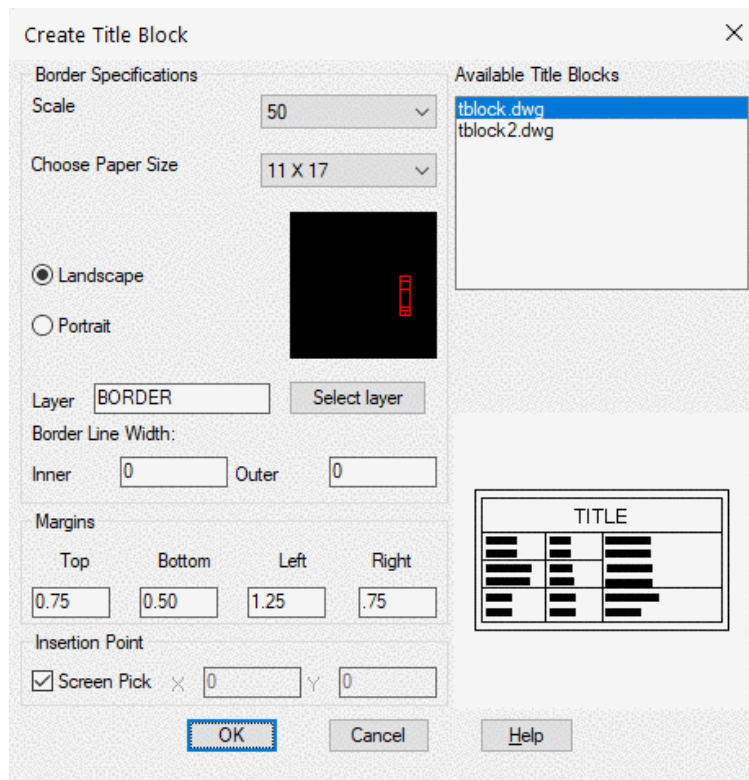
The following results are reported:

Area = 83921.758997 sq. units, Perimeter = 1128.95
SQ. FEET: 83922 SQ. YARDS: 9324.6 SQ. MILES: 0.003
ACRES: 1.9266
Closure error distance> 0.0171 Error Bearing> S 52°5'26" E
Closure Precision> 1 in 66076.9 Total Distance Traversed> 1128.950

The resulting deed has a closure of 1:66076.9.

Use the View – **Extents** command (key-in short-cut **ze**) to see the entire area. Then choose Zoom Out (key-in short-cut **zout**) under the View menu giving you adequate room for the next step.

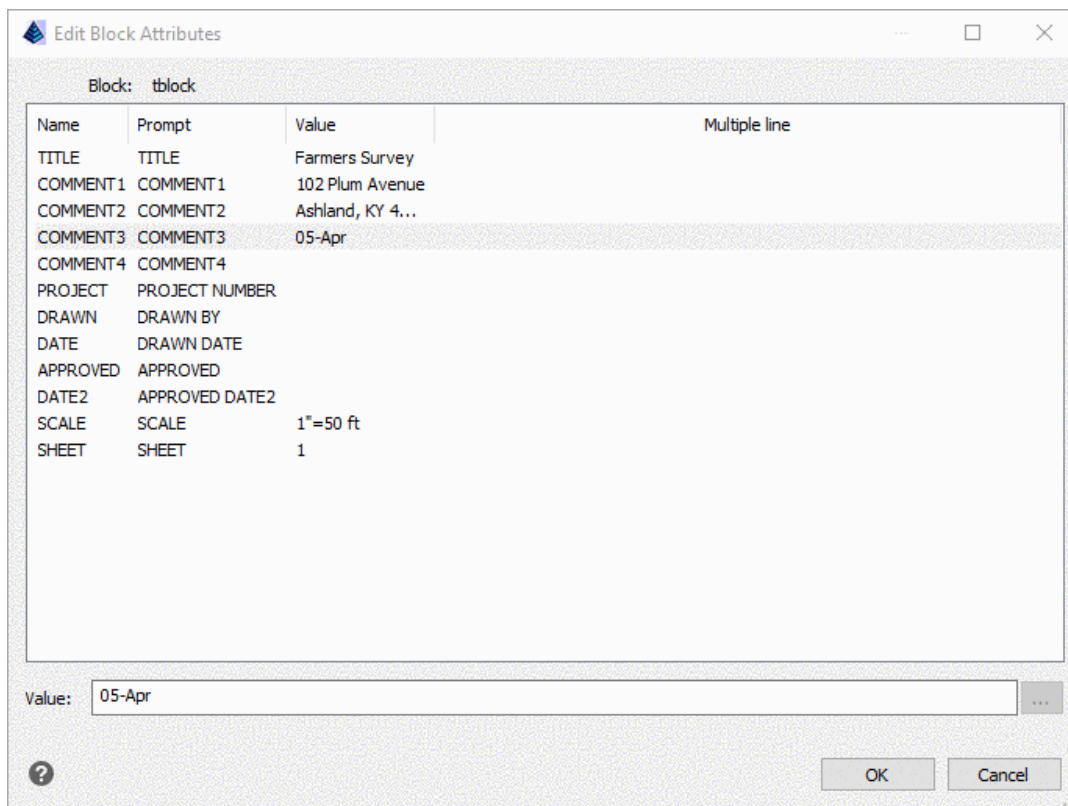
Issue the Settings – Title Block – Standard command to display a dialog box similar to that shown below:



Complete the title block entries, as shown above. Select Paper Size B2 (11 x 17), and enter the layer name of BORDER, then choose **OK** and when prompted:

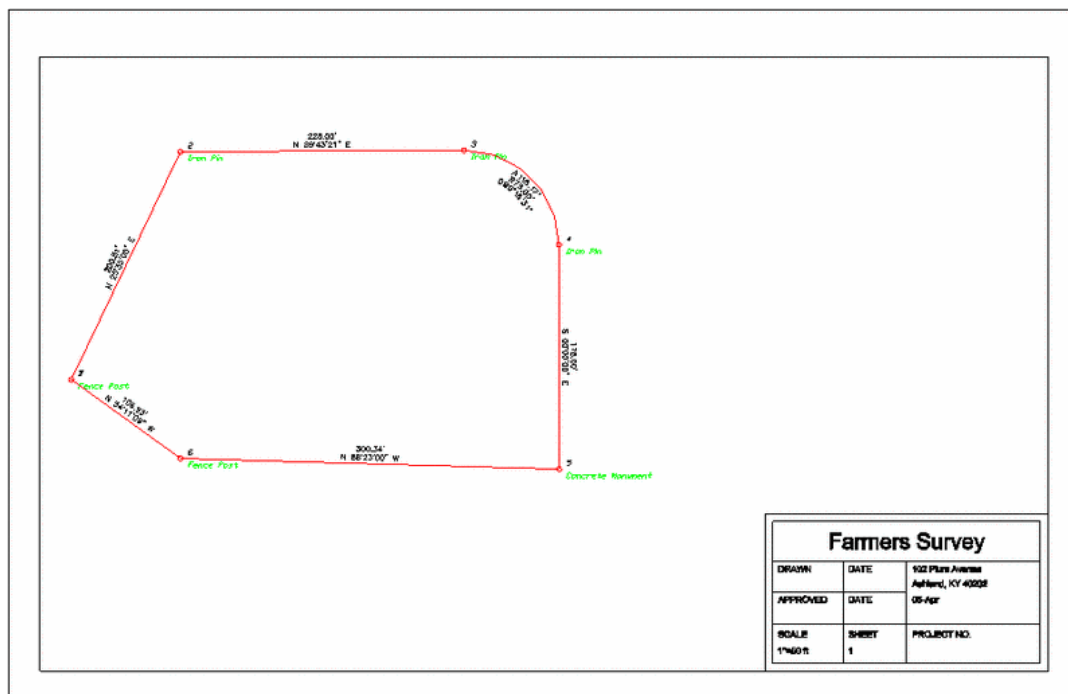
Pick location: pick a point in the lower left of the survey

The following dialog appears, allowing you to enter the attributes for the *Title Block*:



Set the values as suggested above and click **OK** when ready.

NOTE: The title line is plotted in large text on the title block. Its length, therefore, should not exceed 15 characters. Your drawing should look similar to the example below at this point:

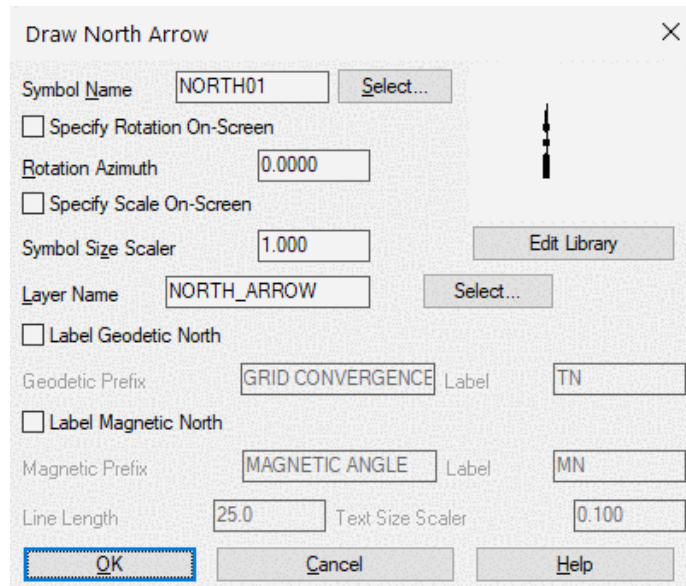


Use the View – **Extents** command to see the entire working area. If you want to move the border, use the Edit – **Move** command (keyin shortcut **M**). Pick the border lines and the title block objects (up to 3 picks total), press Enter (to end object selection), then pick two points representing the vector of the move.

If you want to see a margin around the working area after you use the Extents command, use the View – **Zoom Out** command. Then use the View – **Window** command to capture the view and margin you prefer.

If you make a mistake, enter U for Undo, or select the back arrow toolbar icon that appears at the top of the screen.

Issue the Annotate – Draw North Arrow command to display a dialog box similar to that shown below:

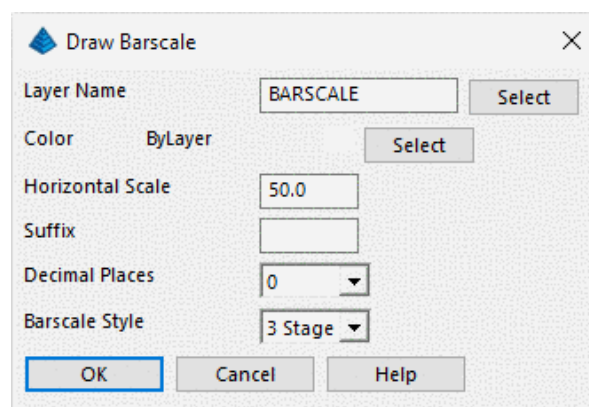


Accept the default north arrow that is shown at the right side of the dialog, click **OK**, and when prompted:

Pick insertion point: place it in the above the *Title Block* (leave enough space for the *Barscale* to follow)

If you want to reposition the North Arrow, issue the Edit – **Move** command.

Issue the Annotate – Draw Barscale to display a dialog box similar to that shown below:



Accept the defaults and click **OK** and when prompted:

Pick location for barscale: pick an insertion point below the *North Arrow* and directly above the "a" in *Farmer*, and approximately the same distance from both.

You can use the Edit – **Move** command if you want/need to re-position the barscale.

Issue the Draw – Text – **Standard** command and when prompted:

Specify start point of text or [Justify/Style]: J

Enter an option [Left/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR]: C

Center point of text: pick a point near the top-center of the drawing

Height of text <4.00>: 10 (Entering 10 makes the title text bigger than the default).

Rotation angle of text <E>: press Enter

Text: Farmer Survey

Text: Ashland, KY

Text: press Enter

To enter a certification in the lower-right of the drawing, again select Draw – Text – **Standard** (or type "dtext" at the Command line).

NOTE: If you haven't done anything else, such as Zoom or Pan, you can simply press Enter to repeat the last command. If pressing Enter does not repeat the Text command, restart the command as suggested above and respond to the resulting prompts as shown below:

Specify start point of text or [Justify/Style]: Pick a point above and to the left of the *Title Block* for the certification. You don't have to enter L for left-justification. The DText command defaults to left-justification every time.

Height <10.00>: 4

Rotation angle <E>: press Enter

Text: Surveyor's Certification

Text: *press spacebar, then press Enter*

Text: I do hereby certify that the survey shown hereon

Text: is a true and correct representation.

Text: *press spacebar, then press Enter*

Text: -----

Text: John Doe, PLS #1234

Text: *press Enter twice to end*

Enlarge the two title lines (*i.e.* **Farmer Survey** and **Surveyor's Certification**) by a factor of 2.0 using the Edit – Text – Text Enlarge/Reduce command. When prompted:

Scaling Multiplier: type 2 and press Enter

Select Text for Scaling.

[FILter?]/<Select entities>: pick the **Farmer Survey** text and press Enter

[FILter?]/<Select entities>: pick the **Surveyor's Certification** text and press Enter

[FILter?]/<Select entities>: press Enter

NOTE: When you are selecting objects, if you select something you don't want, you have the option to remove undesired entities with the (R) option.

The following is a closeup of the *North arrow*, *Bar Scale* and *Certification label* that we just entered:

Surveyor's Certification

I do hereby certify that the survey shown hereon
is a true and correct representation.

John Doe, PLS #1234

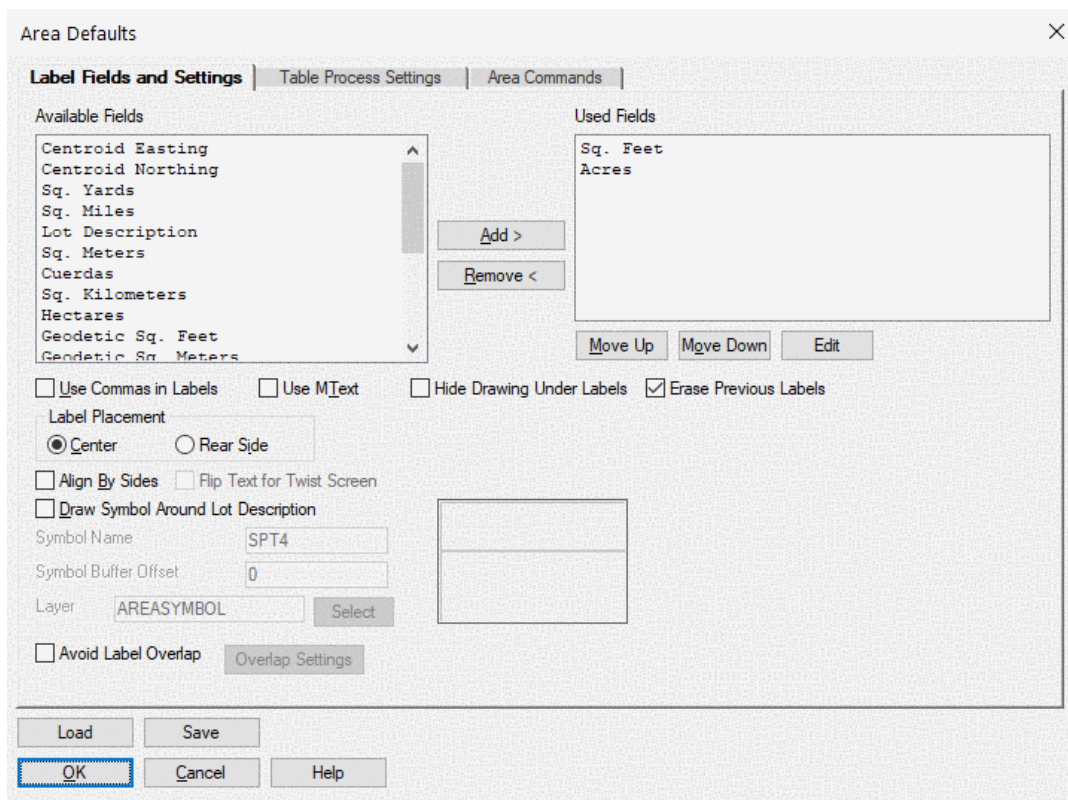
Farmers Survey		
DRAWN	DATE	102 Plum Avenue Ashland, KY 40202
APPROVED	DATE	05-Apr
SCALE 1"=50 ft	SHEET 1	PROJECT NO.

Our next task will be to refine a bearing label; click the Edit – Text – **Edit Text** command. When you are prompted:

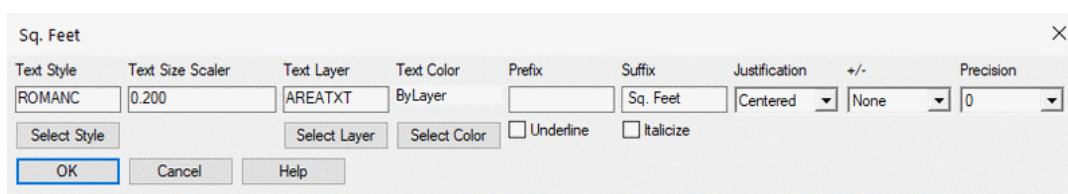
Select Text to Edit: pick the due South bearing text of S 00°00'00" E and change it to S 00° E.

NOTE: The degree symbol in CAD *SHX* fonts is represented as `%%d`. For example, if you had typed N 15%%d25'35" E in the *DText* command, Carlson would draw that entry as N15°25'35"E. Click in the text to the immediate right of the quotation mark and press the Backspace key until the text reads "S 00%%d E and press Enter (or click **OK** if in a dialog box) to exit the command.

Prepare for area labeling by selecting the Area/Layout – Area Defaults command. The dialog box shown below appears:



Select the **Sq. Feet** item and click on the **Edit** button and make the **Area Text Size Scaler** 0.2 (doubled from the default of 0.1) as shown below. Click **OK** when complete.



Next select the **Acres** item and click on the **Edit** button. Also, make the **Area Text Size Scaler** 0.2 and click **OK** to dismiss the *Acres* dialog box and click **OK** to dismiss the *Area Defaults* dialog box.

You are going to compute the area by point number by inverting from 1 through 7 and back to 1. Select the Area/Layout – Inverse with Area and respond to the prompts as shown below:

¹
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

2-3 Supplying a point range (separated by a dash) is very helpful when dealing with many consecutive straight-leg courses.

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

r Point 3 is the PC of the curve so we need to find the radius point of the curve.

Radius point number or pick point:

cen

We wish to use a CENter snap.

Snap to CEN of:

pick the arc-portion of the polyline
Curve direction [Left/<Right>]?

r

Pressing Enter would have also
executed the default <Right> option.

Pick End of Arc or point number (U-Undo,Enter to end):

4

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

5-6

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

1

We are establishing the desire to
Inverse back to the starting point.

Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):

press Enter to end.

Inverse with Area

A Standard Report Viewer dialog box showing the *Inverse with Area* results will appear. Click the **Exit** (Doorway) button at the top of the dialog box and also note the results to the Command window:

Area = 83921.725598 sq. units, Perimeter = 1128.97
SQ. FEET: 83922 SQ. YARDS: 9324.6 SQ. MILES: 0.003
ACRES: 1.9266 PERIMETER: 1129.0
You should be prompted:

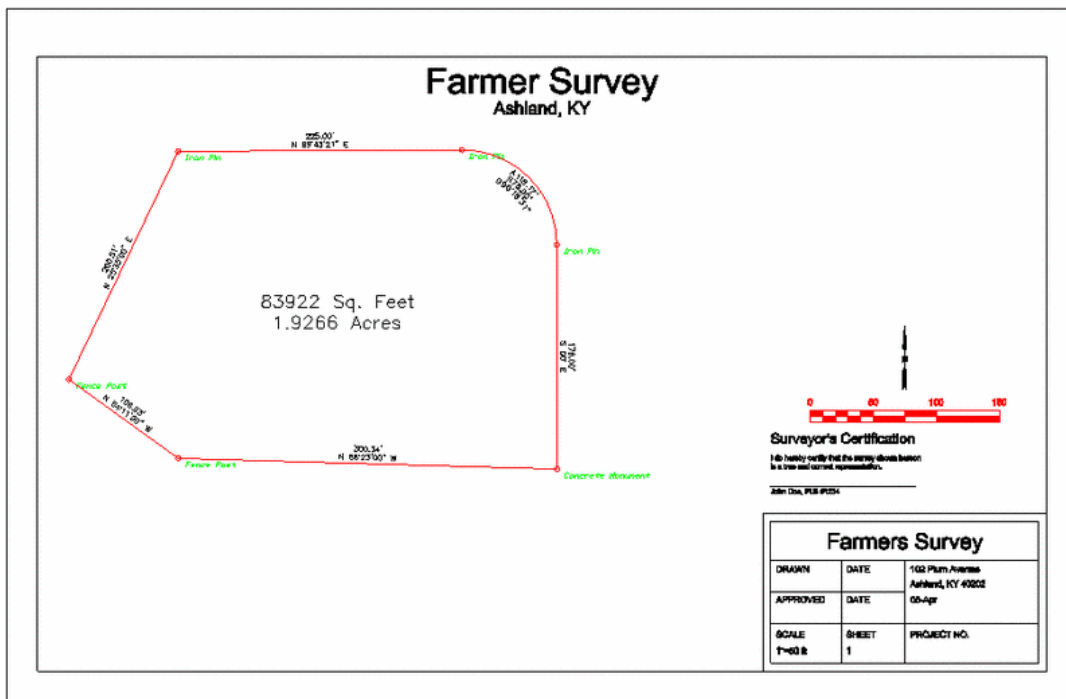
Pick area label centering point (Enter for none): Pick a point near the center of the figure, in its interior. The area units you chose in Area Defaults are labeled on the screen.

You snapped to the radius point using the manually keyed "cen" snap. Additional object snaps appear under Settings – Object Snap. Since all plotted points have a node, you could have inverted around this figure by using the "nod" snap for points 1 through 7, and the "cen" snap to capture the radius point. Snaps are typically entered at the keyboard as 3 characters (for example, "int" for intersect and "end" for endpoint).

Also, you could have chosen the Area by Lines & Arcs command. In that command, you would pick the lines and arcs that make up the figure. But since the closure was 0.017 off (the distance from point 7 to point 1), you would exceed the default Max gap tolerance. Unless you change that tolerance in the command to something larger than 0.017, you would not get a result using this method.

Freeze the point numbers to finish the drawing by choosing View – Layer Control. In the **PNTNO** row, click the Sun icon to change it to a Snowflake icon, which freezes the PNTNO layer. Click **OK**. The point numbers remain in the drawing, waiting to be "thawed", but they are not displayed.

The final drawing is shown here:



This completes the Lesson 1 tutorial: Entering a Deed.

>>>>>>> .r4753

Making a Plat

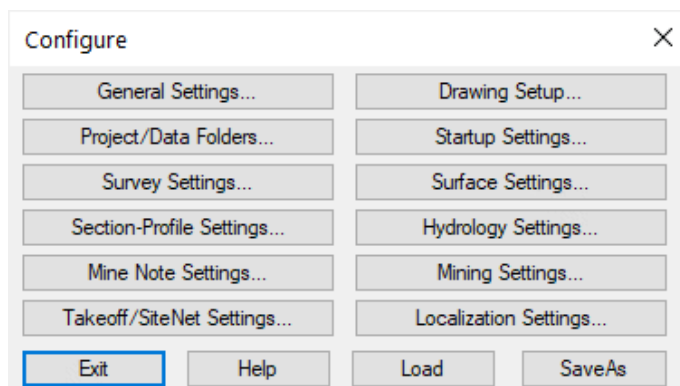
In this lesson you will draw out a plat of a single lot, using Carlson drafting techniques. You will make the plat from an ASCII file of points named **Plat.txt**.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **New Drawing**.
 - If you get the Startup Wizard dialog box, click the **Exit** button.
 - If you are taken directly into CAD, continue as is.

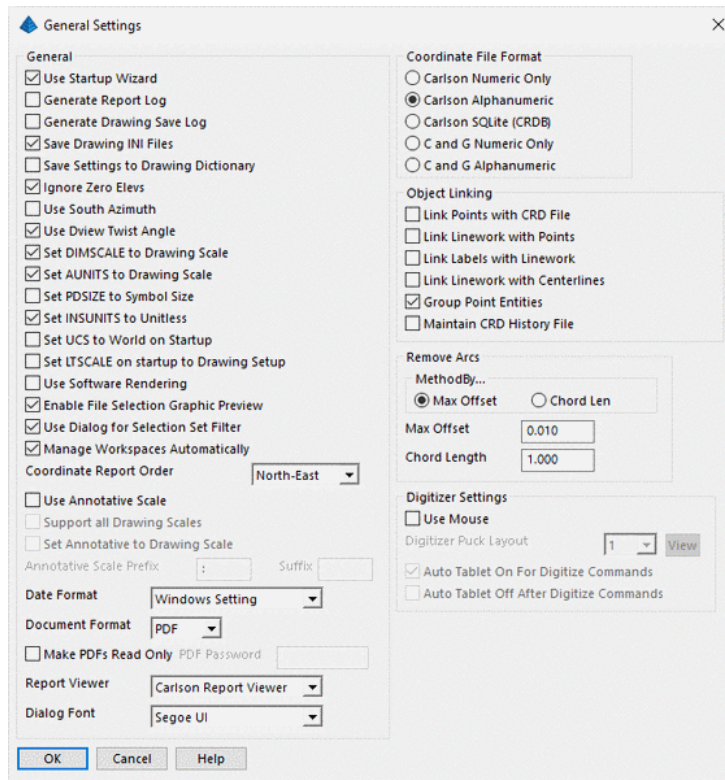
Verify the Survey module is loaded by clicking Settings – Carlson Menus – Survey Menu.

NOTE: You will use the Startup Wizard in Lesson 3 to perform a series of commands to begin a drawing.

2. We'd like to explore some basic configuration settings. Click Settings – Carlson Configure to display the following menu:



Click General Settings to display the dialog box shown here:

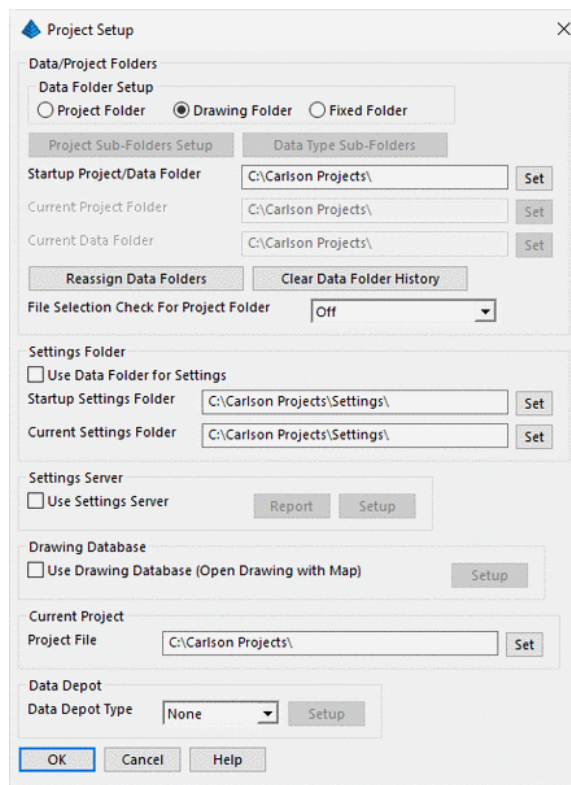


The options in this dialog box, along with the options in other Configure sub-options, determine default working conditions for Carlson. Make/set the following values:

- In anticipation of Lesson 3, make sure the **Use Startup Wizard** is enabled.
- Choose **Carlson AlphaNumeric** to provide the ability to store points in alphanumeric form (*e.g.* 1A, 1B, 1C, HUB5, CTRL, SS10, *etc.*). Generally, there is a slight speed advantage to working with purely numeric point numbers. The highest numeric-only point number allowed is 99999. However, there is no limit to the number of points in an alphanumeric coordinate file.
- Enable the **Group Point Entities** option which groups point elevations, numbers, and descriptions (all aspects of the points) into a single entity for moving, erasing and other commands.

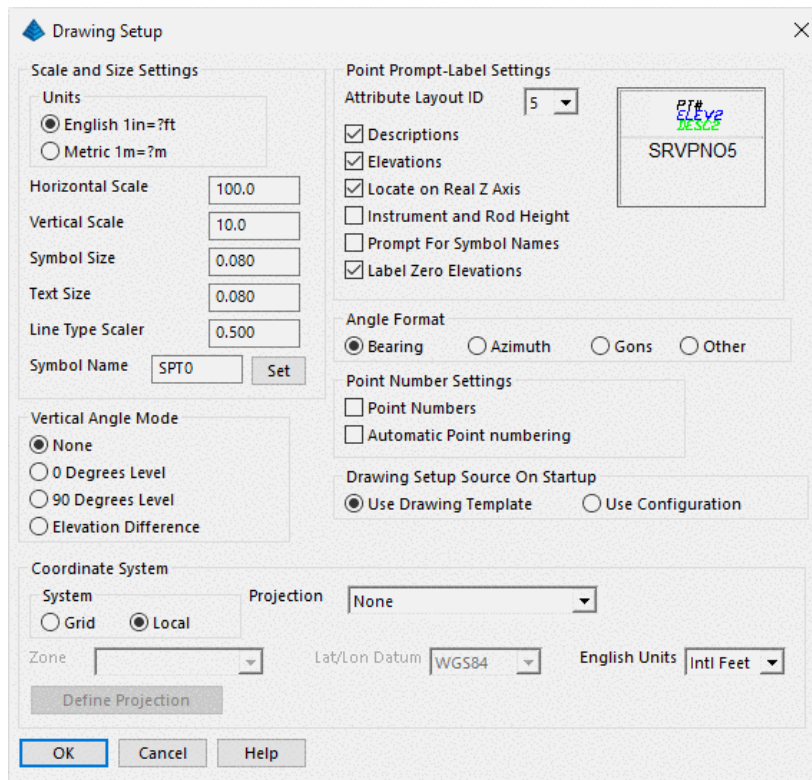
Click **OK** at the bottom of this dialog box.

3. Now we want to set the data path. Another of the Configure sub-options is Project/Data Folders. Click this option and you will see this dialog box:



For this lesson, you will keep it simple. Enable the **Drawing Folder** at the top. This option will, by default, send any data files to the same folder location where the active drawing is located. Notice the **Current Data Folder** section at the bottom. This specifies where data files, such as .crd files in this case, are to be stored. Set the folder to **C:\Carlson Projects** and click **OK** to return to the main *Configure* dialog box.

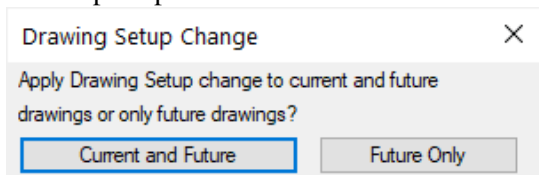
4. Select Drawing Setup from the *Configure* dialog box to display the dialog box below:



The **Horizontal Scale** acts as a multiplier on all text annotation. For example, $100 * \text{Text Size} (0.08) = 8$ (text

height of 8 CAD units). The **Text Size** setting is the effective height, in inches, that the text will appear when plotted at the specified *Horizontal Scale* (example here, 100).

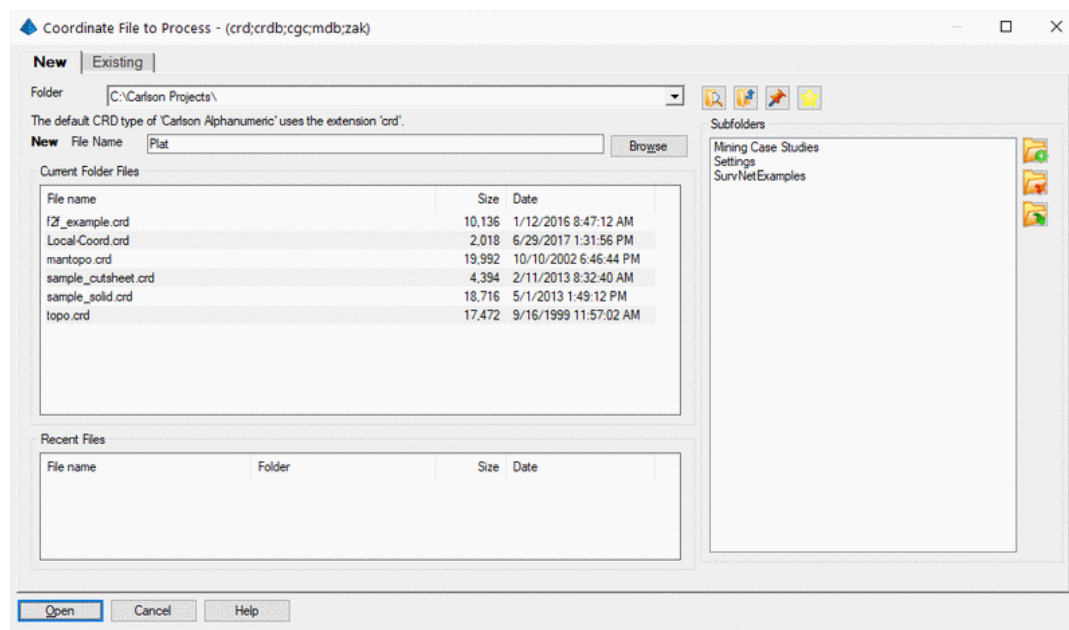
Bearing and Distance labels, Legends, Title Blocks, and Point Symbols will size up or down on the basis of the **Horizontal Scale** set within *Drawing Setup*. Set the Horizontal Scale to **100** and match other settings as desired. Then click **OK** to dismiss the dialog box then click **Exit** to close the Configure dialog box. If prompted:



Click on the **Current and Future** button.

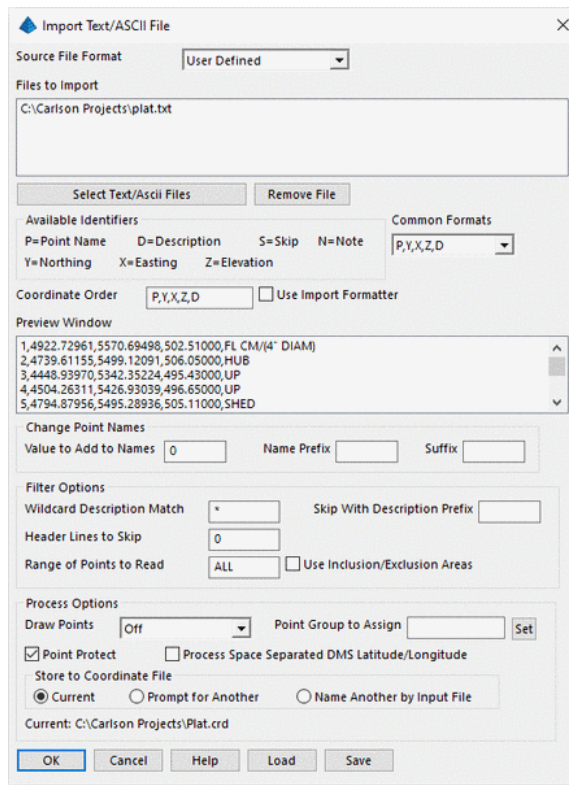
5. If you have not already saved your drawing, now is a good time to do it. Use the File – **Save** command and call the file Lesson2.dwg (store in the C:\Carlson Projects\ folder).
6. Next, you will import the ASCII file called **Plat.txt** and store the points in a Coordinate file called **Plat.crd**. However, since you are in a new drawing, you have not yet set a coordinate file to store the points in. You must have a Carlson coordinate file (.crd) open and established as the container for your points.

Click Points – Set CoORDinate File to display a dialog box similar to that shown below:



Click the **New** tab, as shown here. To the right of File name, type **Plat** and click **Open**. You have now created the required coordinate file.

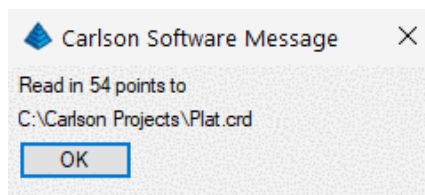
7. You are now ready to import the points. Click the Points – Import Text/ASCII File command to display the dialog box:



Click the **Select Text/ASCII Files** button and then choose **Plat.txt** listed in the file list. It is found in the default data folder (C:\Carlson Projects) and click **Open** when ready.

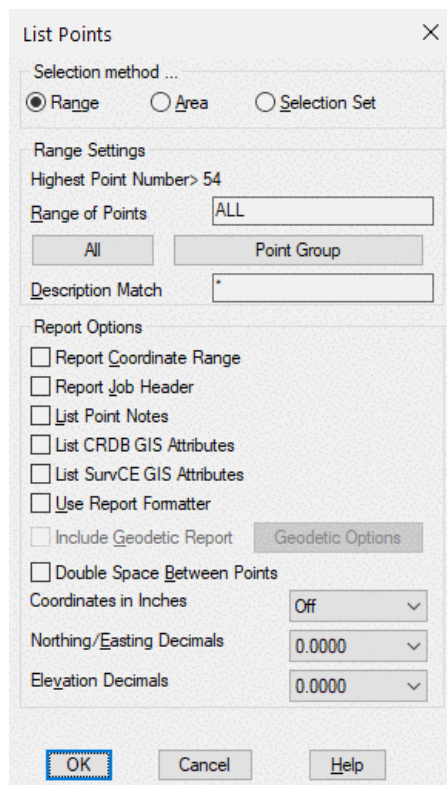
Plat.txt is an ASCII file containing 54 points in the form of *Point Number, Northing, Easting, Elevation* and *Description*. The format of the points appears in the **Preview Window**. The format is: Point (P), Northing (Y), Easting (X), Elevation (Z), Description (D), or, in short, **P,Y,X,Z,D**. You must match this format in the **Coordinate Order**. If you don't see P,Y,X,Z,D in the **Coordinate Order** box, then select that format from the **Common Formats** option. Or, you can type the list directly into the **Coordinate Order** box. Make sure that **Draw Points** is set to Off.

Click **OK** to save/store the data in **Plat.crd**. A confirming dialog appears as follows:



Click **OK** to dismiss the alert.

8. Choose the **Points – List Points** command to display the dialog box below:



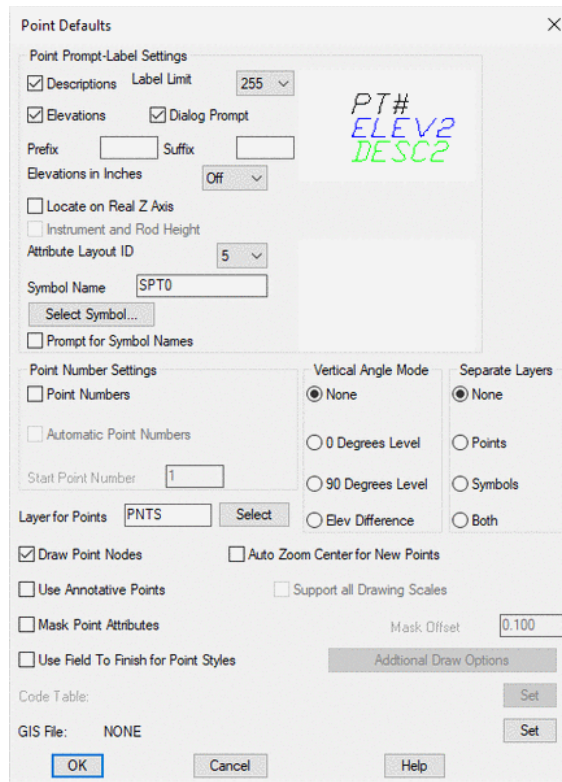
The List Points dialog box will typically default to the full range of points (ALL), which is 1 through 54 in this exercise. You can control the decimal places for the Northing/Easting and the Elevation of the points in the lower portion of the dialog box. Click **OK** and the settings shown above result in the report exhibited below in the Standard Report Viewer:

PointNo.	Northing (Y)	Easting (X)	Elev (Z)	Description
1	4922.7296	5570.6950	502.5100	FL CM/ (4" DIAM)
2	4739.6115	5499.1209	506.0500	HUB
3	4448.9397	5342.3522	495.4300	UP
4	4504.2631	5426.9304	496.6500	UP
5	4794.8796	5495.2894	505.1100	SHED
6	4771.8550	5486.6609	505.5300	SHED
7	4782.6478	5457.8611	505.8200	SHED
8	4854.7655	5357.2213	501.9700	IP FL
9	4467.7479	5476.2376	496.3700	IP
10	4571.0030	5617.4467	499.1300	IP
11	4889.9902	5551.4909	503.0100	18D/OAK
12	4811.6154	5579.1710	504.1200	24D/OAK
13	4675.9325	5564.3498	504.9700	12P
14	4714.7994	5608.3889	502.7500	8P
15	4684.3432	5652.9519	501.4700	IP
16	4573.4034	5443.7462	497.9400	PL
17	4792.0208	5435.9996	505.9000	UP
18	4651.5225	5516.5361	504.0300	BLD
19	4656.3297	5535.9497	504.7200	BLD
20	4854.6447	5565.3310	503.5700	18D/OAK
21	4528.3876	5369.1237	496.9300	HUB
22	4667.4819	5449.3567	501.2700	LP
23	4760.9888	5397.1521	502.1700	X

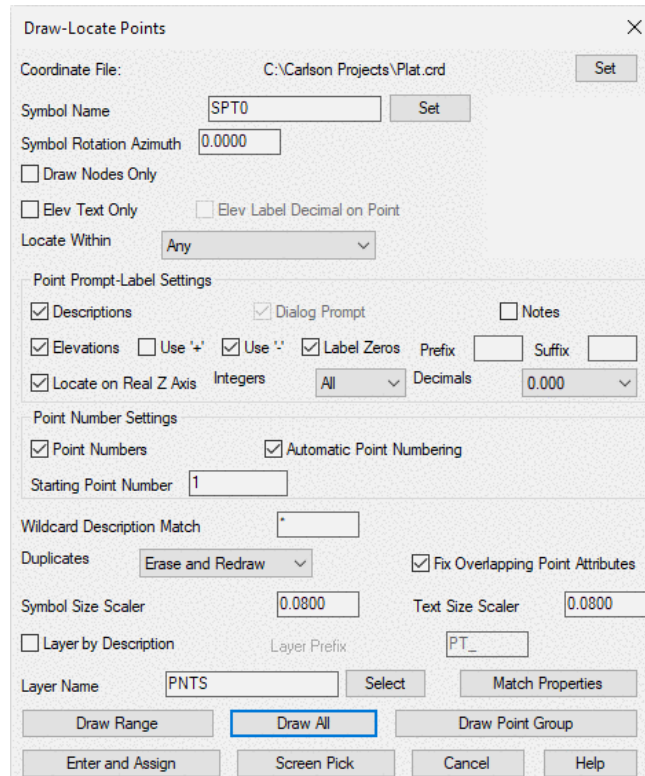
Exit the report by selecting the **Exit** (doorway) icon at the top of this report viewer box, or by clicking the X in the upper right of the window.

9. Issue the Points – Point Defaults command and perform the following:

- Enable the Elevations toggle, and,
- Enable the Descriptions toggle, and,
- Use the Select Symbol button to set the Point Symbol name to symbol 0 (SPT0), which is a standard point, and,
- Enable the Automatic Point Numbering toggle, and,
- Match other settings as desired and as shown below and click **OK**.



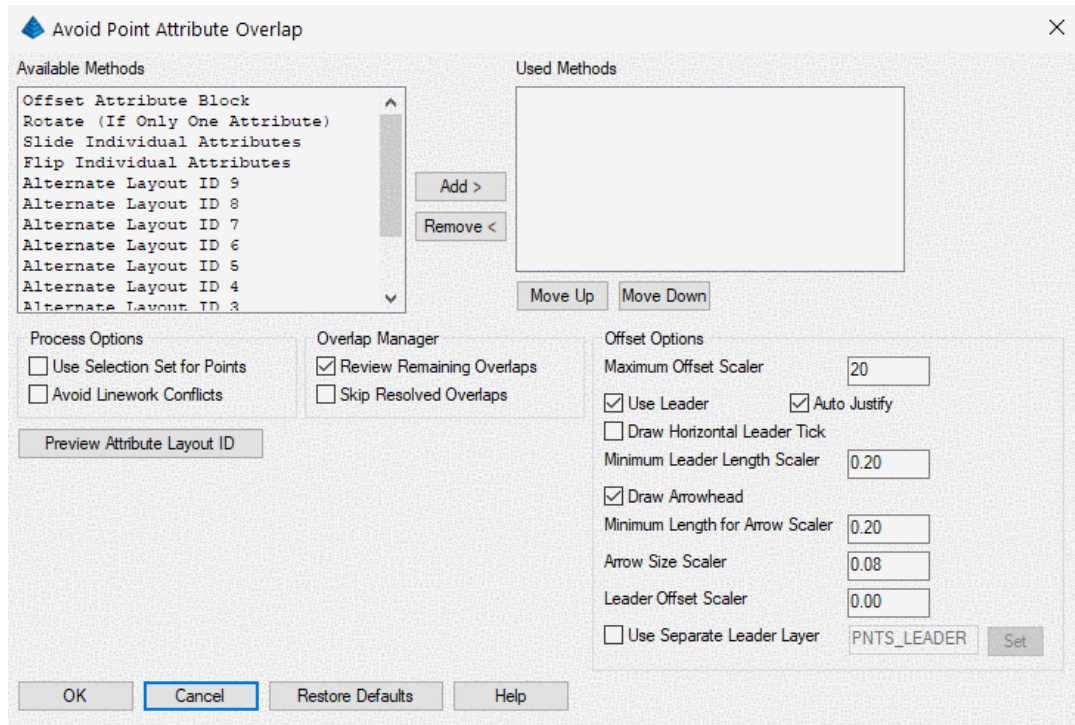
10. Select the Points – Draw-Locate Points command to draw the points on the screen:



As shown in the figure shown above, the current **Symbol Name** is showing as SPT0, which is a standard CAD node symbol (*i.e.* a point). You can select a different default symbol using the Points – Point Defaults command.

In this exercise you, let's continue to use SPT0. Later, you will add official property corner and utility symbols. Although you are working without a default symbol, there will always be a "dot" or a node at the correct insertion point of each point number.

Enable the Fix Overlapping Point Attributes toggle if it is not already checked. Click **Draw All** and the following dialog will display:



The Avoid Point Attribute Overlap dialog uses different adjustment methods, such as moving attributes and creating leader lines, to fix conflicts with the point labels. Click **Cancel** for now so we can view these conflicts:

Select Carlson Software points.

[FILter]/<Select entities>: wp

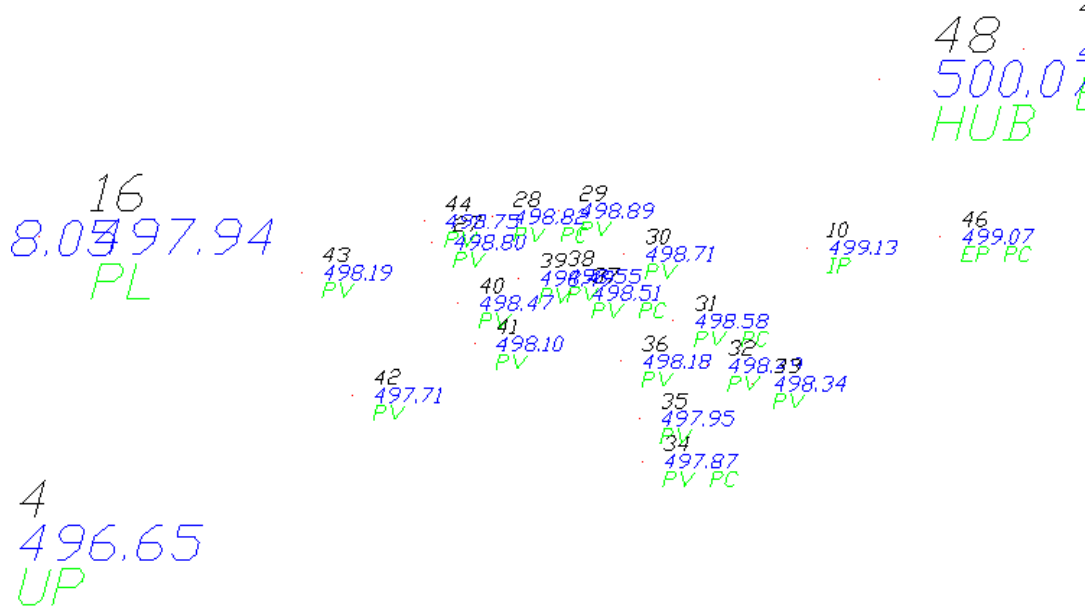
First vertex of inclusive polygon: click a starting point for the polygon

Next vertex: continue picking desired locations

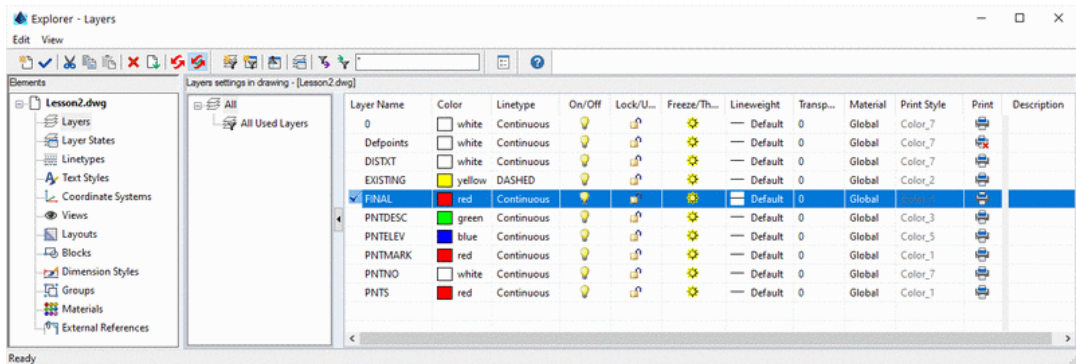
Next vertex: when satisfied, press Enter to complete the Window Polygon

[FILter]/<Select entities>: press Enter to complete the selection set

The following shows the scaled points:



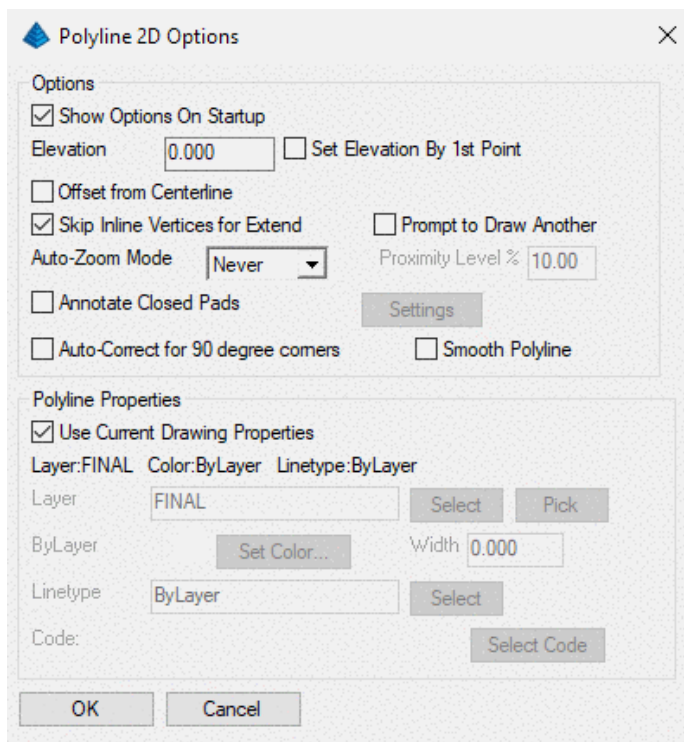
- Next, you will prepare for drawing linework by setting the current layer. You should draft linework and symbol work in designated layers. In this example, you will put linework and symbol work in a layer named **Final**. A "best-practice" would be to place property linework on a **PropLines** (or similar) and utility linework on a **Utilities** layer, *etc*, but for now, you will put all linework and symbols on the layer Final. To pick the current working layer, select the View – Layer Control command (IntelliCAD shown):



Click **Final**. Right click, select **Current** and close the Layer control.

NOTE: If the "Final" layer does not exist, create it and assign it a suggested color of red.

- Issue the Draw – 2D Polyline command which allows you to easily draw a polyline between point numbers. A dialog box might appear as illustrated below. If it does, accept the defaults and click **OK**.



First, connect portions of the property line via the steps outlined below:

1 This instructs the 2D polyline to commence at Point #1.

[Arc/Close/Distance/Follow/Undo/⟨Pick point or point numbers⟩]:

8 This instructs the 2D polyline to be drawn to Point 8

[Arc/Close/Distance/Follow/Undo/⟨Pick point or point numbers⟩]:

press Enter This creates a polyline. Keep this as a separate polyline because later you will turn this back lot line into a fence line. Press Enter to repeat the **2D Polyline** command.

[Arc/Close/Distance/Follow/Undo/⟨Pick point or point numbers⟩]:

8-10 This instructs the 2D polyline to be drawn through Points 8, 9 and 10.

⟨Pick point or point numbers⟩

a This indicates a tangential arc to be drawn starting from Point 10.

[Radius pt/radius Length/Arc length/Chord/Second pt/Undo/⟨Endpoint or point number⟩]:

15 This indicates the arc shall terminate at Point 15.

[Arc/Close/Distance/Extend/Follow/Line/Undo/⟨Pick point or point numbers⟩]:

1 This instructs the polyline to be drawn back to the starting point. The segment from 15 to 1 is not guaranteed to be tangent to the previous arc.

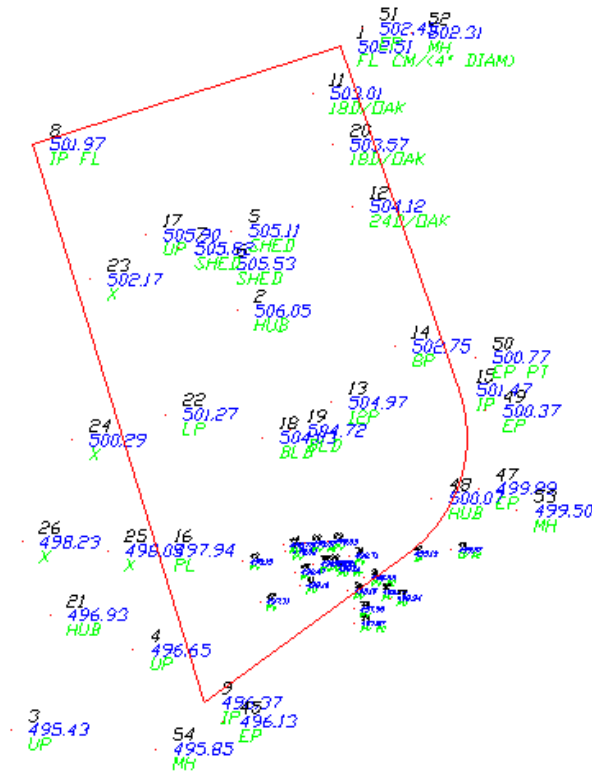
[Arc/Close/Distance/Follow/Undo/⟨Pick point or point numbers⟩]:

press Enter

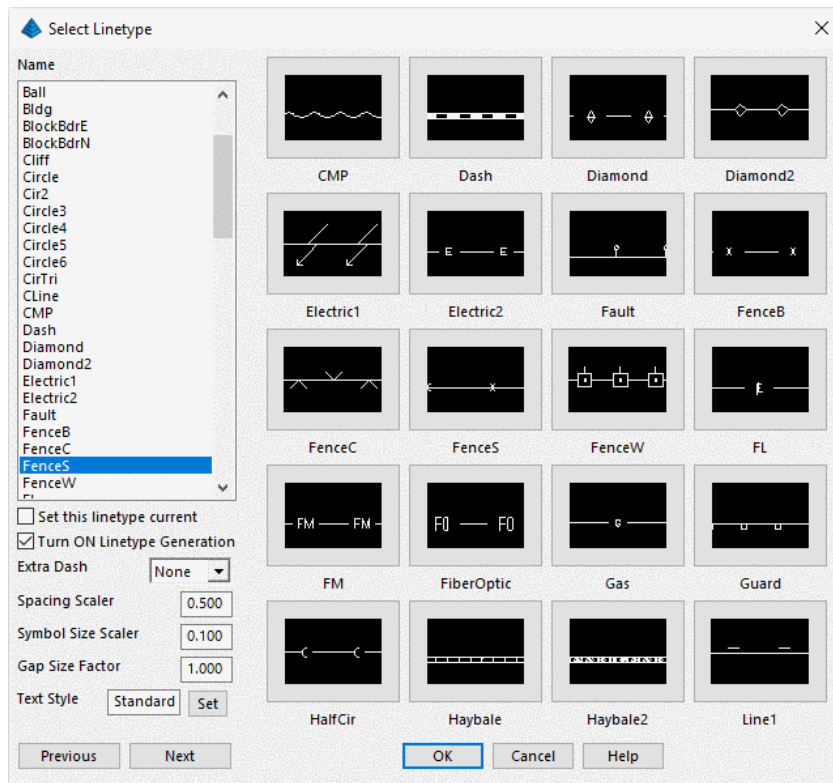
This creates the full lot.

2D Polyline Courses

You should have the following linework at this point:



You will now create a fence line on the polyline you drew from points 1 to 8. Click the Annotate – Line Types – Change Polyline Linetype command. This command creates polylines that respond as one entity when selected.



Notice in the dialog box above that the current Line Type Scaler, governing spacing, should be 0.5 (inches) and the Text (height) Scaler is 0.1. If your settings are different, you may want to set these items to match the example.

Choose the **FenceS** option (the solid fence line) and click **OK** and when prompted:

Select linework to change.

[FILter/?]/<Select entities>: pick the polyline you created from points 1 to 8

[FILter/?]/<Select entities>: press Enter

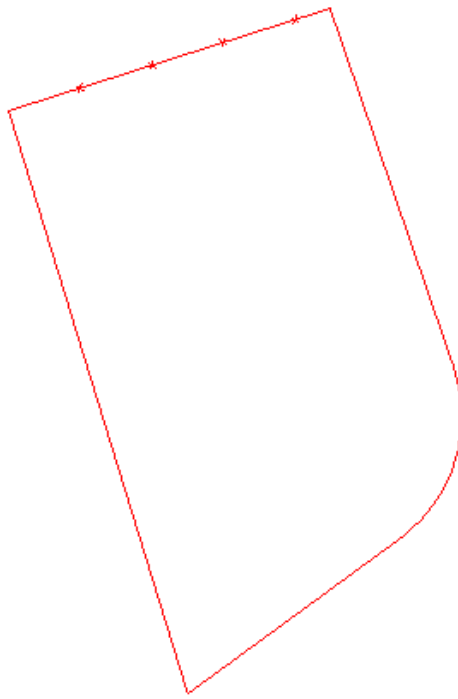
Inspect the result by clicking View – Isolate Layers command and when prompted:

Select objects on layers to isolate.

[FILter/?]/<Select entities>: pick the property line

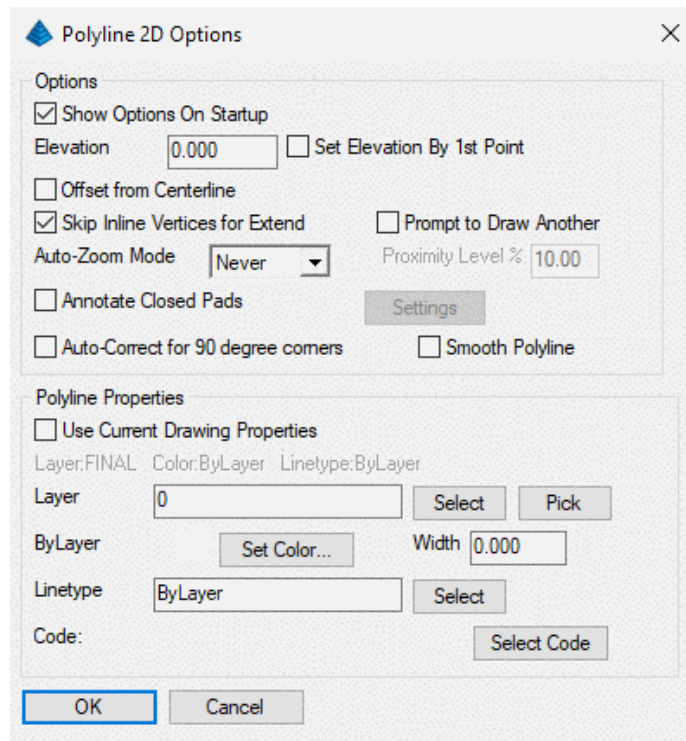
[FILter/?]/<Select entities>: press Enter

Here is the result:



Select the View – Restore Layers command to recover your points. Experiment with the "cadence" of *Isolate Layers* and *Restore Layers*. Select **Isolate Layers**, pick the layer(s) to isolate and then press Enter twice. Then select **Restore Layers**.

Next, you will connect up the edge of pavement. Select the Draw – **2D Polyline** command. Again, a dialog box might appear as shown below. If it does, make sure that the options selected are the same. In the future you can choose not to see this box:

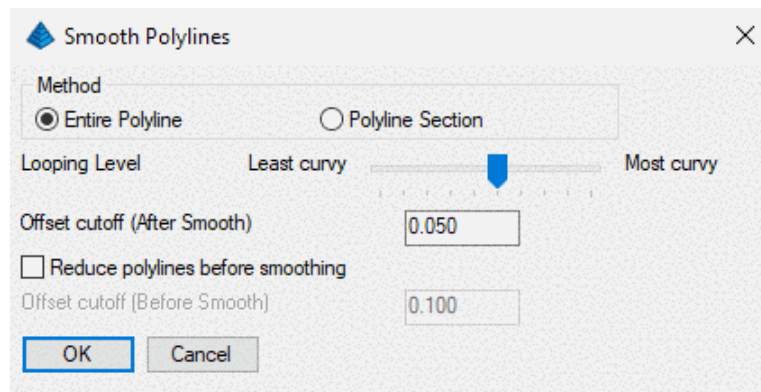


Click **OK**. Proceed as follows to connect up the edge of pavement:

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: 45-47,49-51

Press Enter at the next prompt to exit the command and create the road. Note how you can separate range entries using a comma.

To smooth the edge of the road, select the Edit – Polyline Utilities – Smooth Polylines command and set the values as shown below and click **OK**:



When prompted:

Select polylines to smooth.

[Filter/?]/<Select entities>: pick the edge of road polyline

[Filter/?]/<Select entities>: press Enter

To offset the smoothed edge-of-road polyline by 24 feet to make the opposite edge of the road, select the Edit – Offset – **Standard Offset** command and provide the following responses:

Offset: ENTER for Through point/<Distance>: 24

[Undo]/<Select entity>: Pick the edge-of-road polyline

[Both sides]/<Side for parallel copy>: Pick to the right of the polyline

[Undo]/<Select entity>: press Enter

Now, select the View – **Isolate Layers** command again. Pick on any of your linework and only the entities on the picked layers are displayed.

Next, you will draw the shed. Select the Draw – **2D Polyline** command. To draw a two-sided shed, connect points 5 through 7 as follows:

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: 5-7

Press Enter at the next prompt to exit the command and create two-sides of a shed as shown here:

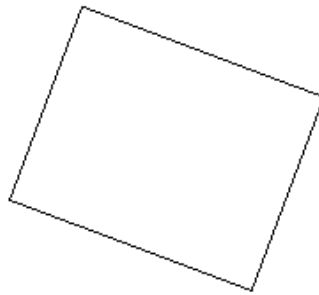


Turn the 2-sided shed into a 4-sided shed by issuing the Survey – 4 Sided Building command. Complete the command as follows:

Options/Points/<Pick a line or polyline>: Pick the shed

Options/Points/<Pick a line or polyline> (Enter to end): press Enter

Now the 2-sided building looks like this:



Focus your attention on the area of tightly spaced points with point numbers ranging from 27 to 44. This is the driveway and paving area. In the case of the driveway, assume that the surveyor who collected the points shot in 3-point arcs. They came up to a PC, shot a point on the arc, and finished up at the PT.

Click on the View – **Window** and pick a lower left and upper right point that windows the driveway area. If you wish to use the View – **Previous** command to zoom out, then use View – **Window** to zoom in again.

Select the Draw – **2D Polyline** command and walk the polyline through the two arcs as follows:

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: 27

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: 28

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: a

[Radius pt/radius Length/Arc length/Chord/Second pt/Undo/<Endpoint or point number>]: s (Use S for 3-pt arcs)

Second point or point number: 29

Endpoint or point number: 30

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: 31

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: a

[Radius pt/radius Length/Arc length/Chord/Second pt/Undo/<Endpoint or point number>]: s

Second point or point number: 32

Endpoint or point number: 33

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: press Enter

In the above exercise you started at point 27, went to the PC at point 28 and inserted a 3-point arc through points 29 and 30. You proceeded on a tangent direction to point 31, which was another PC, then completed a 3-point arc through points 32 and 33, and ended. Now, connect up the basketball court area. Run the Draw – **2D Polyline** (or press Enter to repeat the previous command).

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: 27

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: 44

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: 43-39 (you can enter "backwards" ranges)

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: a

[Radius pt/radius Length/Arc length/Chord/Second pt/Undo/<Endpoint or point number>]: s

Second point or point number: 38

Endpoint or point number: 37

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: 36

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: a

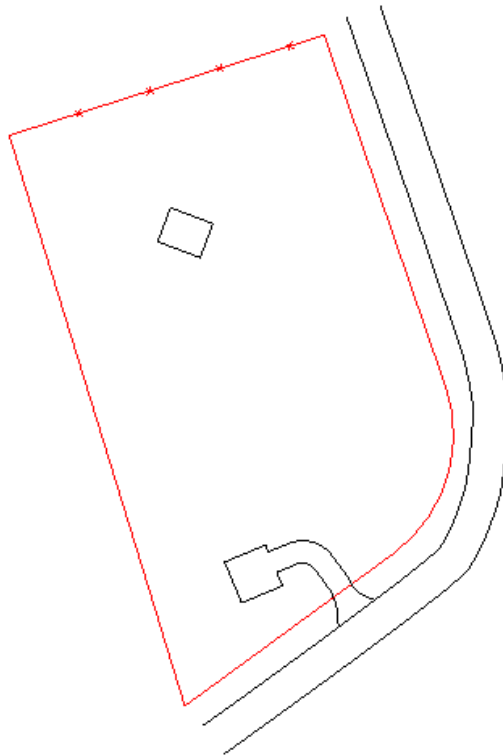
[Radius pt/radius Length/Arc length/Chord/Second pt/Undo/<Endpoint or point number>]: s

Second point or point number: 35

Endpoint or point number: 34

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: press Enter

Shown below is your drawing to this point.



Next you will make a building footprint. Points 18 and 19 are two shot corners of a building and assume that the surveyors taped the main house, going clockwise from point 18, as follows:

10'L, 20'R, 40'L, 20'R, 20'L, 83'L, 60'L, 23'L, 10'R.

Let's begin by using the **2D Polyline** (keyin **2dp**) command to draw the first wall:

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: 18
[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: 19
[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: press Enter

You can easily create the building "jogs" through the Edit – Extend – by Distance command. Complete the prompts as follows:

Pick arc, line or polyline to extend: pick the building line closer to point 18 (This makes the arrow point toward 18 rather than 19. Now you can go clockwise:)

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): L10 (lower case "l" and "r" also work)

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): R20

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): L40

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): R20

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): L20

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): L83

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): L60

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): L23

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): C

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): press Enter

Next, you will complete the linework for the sewer line and the electric utility line. Use the View – **Extents** (keyin ze) command so you can see all your points.

The sewer line runs from points 52 to 53 to 54. Use the **2D Polyline** command and create the sewer line as follows:

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: 52-54

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: press Enter

To annotate the sewer line with an S, initiate the Annotate – Line Types – **Change Polyline Linetype** command as used earlier. Within the dialog box, set the **Spacing Scaler** to **1.5**. This will label "S" on the sewer line every 1.5" when plotted at the current scale (1"=100'). Then select the **Sewer** linetype from the list, and then select the sewer polyline that runs next to the road. The polyline will be annotated.

Next, create the electric utility line with the **2D Polyline** which runs from point 3 to point 4 to point 17:

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: 3

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: 4

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: 17

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: press Enter

No points were taken beyond point 17, due to obstructions from the various setups in the field. So you must extend the polyline from point 17 to beyond the property. Click the Edit – Extend – **By Distance** command as used earlier. Pick on the electric utility polyline near point 17. Then pick beyond the property line and press Enter to end.

Before you annotate the electric utility line, you must offset it 25' on both sides, for a 50' total right-of-way. You will do this using Edit – Offset – **Standard Offset** command. Enter the offset distance of 25. Pick the electric utility polyline and then pick to one side for the first offset. Repeat for the other side, by first picking the electric utility polyline, then picking the other side for the offset. Press Enter to end.

The next task will be to annotate the central electric line using the Annotate – Line Types – Polyline to Special Line command. Choose the **Electric** linetype, which appears on the third page of linetypes and then click **OK**. Then select the electric utility polyline to annotate it, and press Enter.

Next, make the Property lines bold. Use the Edit – Polyline Utilities – Change Polyline Width command and provide the responses as indicated:

New width <0.00>: 1.5

Select polylines to change width of.

[FILter]/<Select entities>: Pick once for the fence line portion

[FILter]/<Select entities>: Pick once for the remaining property lines

[FILter]/<Select entities>: press Enter

To add color and improve layer management, make a layer for your road and driveway. Select the View – Layer Control command.

- Click the New Layer button, and enter the name **Road** for the new layer.
- Choose the color **Cyan** by clicking the color square to the right of the layer name.
- Close the dialog.

Initiate the View – Change Layer command. When prompted:

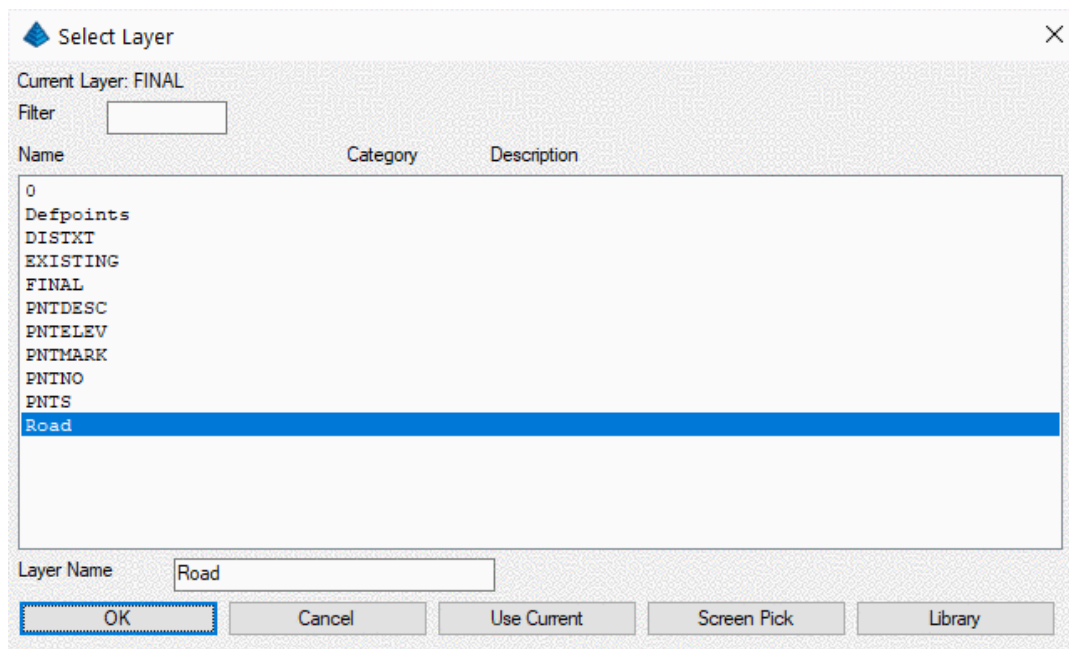
Select entities to be changed.

[FILter]/<Select entities>: Pick all driveway and road entities

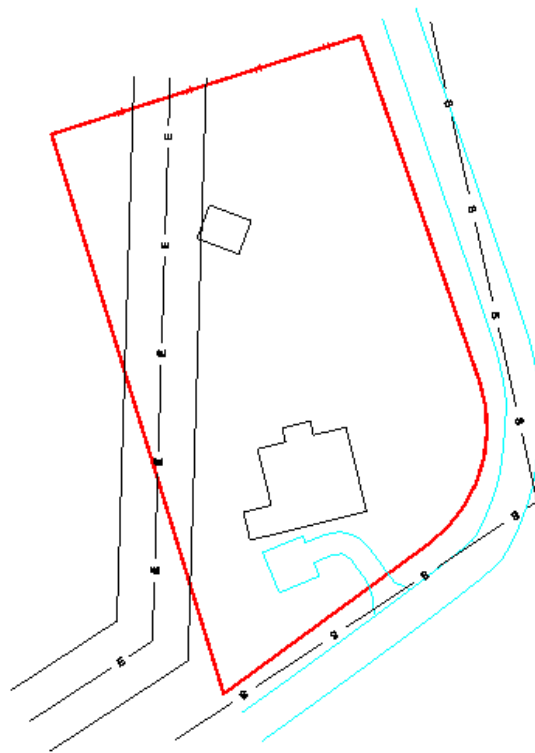
[FILter]/<Select entities>: press Enter

Specify new layer [Name/Current/<pick entity>]: n

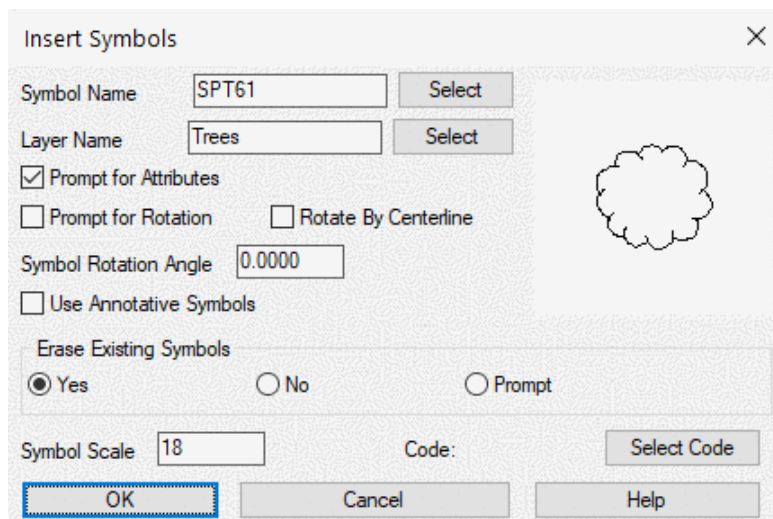
This brings up the dialog box shown below. Select **Road** and click **OK**.



Your linework is now complete and is shown below:



You will add symbols for trees, property corners, manholes and a light pole. Start with the trees. Points 11, 12, and 20 are oak trees of different sizes, and point 14 is a pine tree. Use symbol 61 for the deciduous oak trees and symbol 53 for the pine tree. We'll use the Draw – Symbols – Insert Symbols command. The following dialog box appears:



The various items to place are summarized below:

Small Deciduous Tree: Symbol: **SPT61**, Layer: **Trees**, Size: **18**
Options/Select entities/Enter coords/<Pick point or point numbers>: 11
Options/Select entities/Enter coords/<Pick point or point numbers>: 20
Options/Select entities/Enter coords/<Pick point or point numbers>: press Enter

Press Enter to re-run the command for:

Large Deciduous Tree: Symbol: **SPT61**, Layer: **Trees**, Size: **24**
Options/Select entities/Enter coords/<Pick point or point numbers>: 12

Options/Select entities/Enter coords/<Pick point or point numbers>: press Enter

Press Enter to re-run the command for:

Small Coniferous Tree: Symbol: **SPT53**, Layer: **Trees**, Size: **8**

Options/Select entities/Enter coords/<Pick point or point numbers>: 14

Options/Select entities/Enter coords/<Pick point or point numbers>: press Enter

Press Enter to re-run the command for:

Found Iron Pins: Symbol: **SPT5**, Layer: **FINAL**, Size: **8**

Options/Select entities/Enter coords/<Pick point or point numbers>: 8-10,15

Wildcard match of point description <*>: press Enter

Options/Select entities/Enter coords/<Pick point or point numbers>: press Enter

Press Enter to re-run the command for:

Concrete Monument: Symbol: **SPT13**, Layer: **FINAL**, Size: **8**

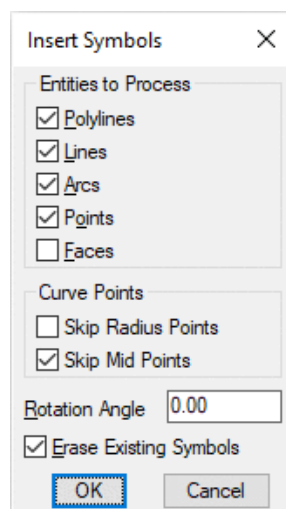
Options/Select entities/Enter coords/<Pick point or point numbers>: 13

Options/Select entities/Enter coords/<Pick point or point numbers>: press Enter

Press Enter to re-run the command for:

Sewer Manholes: Symbol: **SPT34**, Layer: **FINAL**, Size: **8**

Options/Select entities/Enter coords/<Pick point or point numbers>: s (the following dialog box appears, click **OK**:)



Select arcs, circles, faces, points, text, lines and polylines.

[FILter]/<Select entities>: pick the Sewer polyline

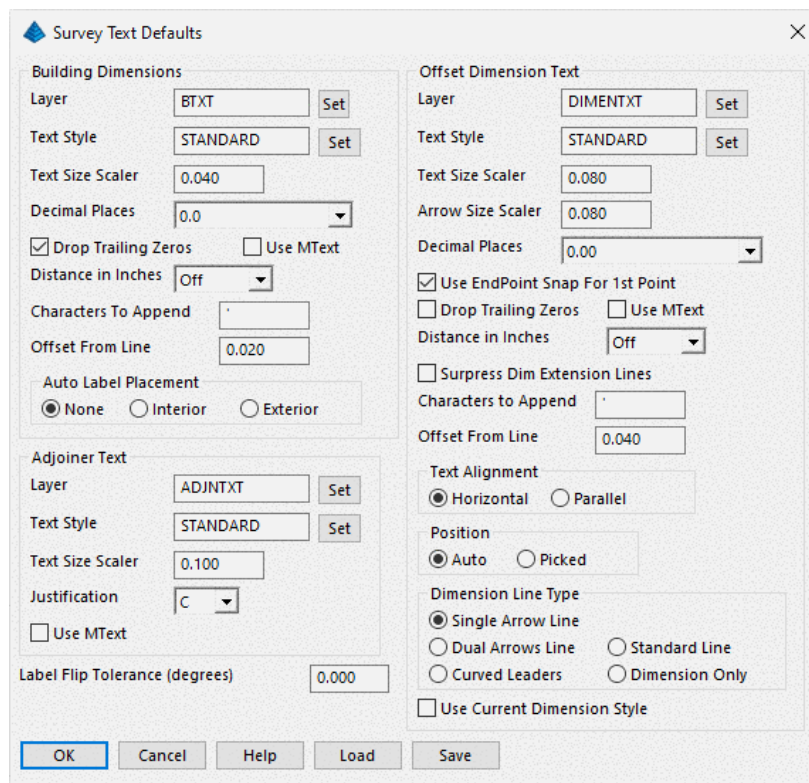
[FILter]/<Select entities>: press Enter to terminate the selection option

Options/Select entities/Enter coords/<Pick point or point numbers>: press Enter

The symbols are inserted at the three polyline endpoints.

Reduce clutter by selecting the View – Freeze Layer by Selection command and picking a point number. The points freeze, leaving only linework and symbols. To bring the points back, use the View – Thaw Layer command. The Freeze Layer/Thaw Layer commands go together, just like the **Isolate Layers/Restore Layers** commands.

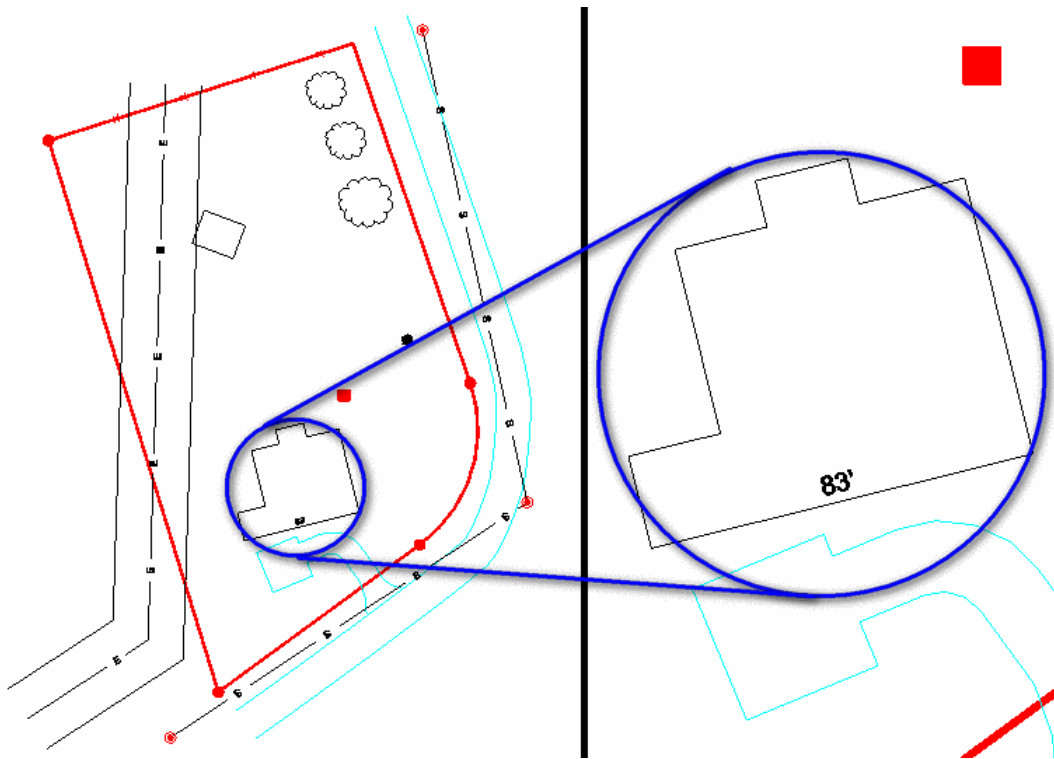
Next, you will create (in reduced size) your building dimensions. You can set the building dimension text size for the current work session using the Annotate – Survey Text – Survey Text Defaults command. The following dialog box appears:



The changes you will make are in the upper-left section "Building Dimensions." Change the following:

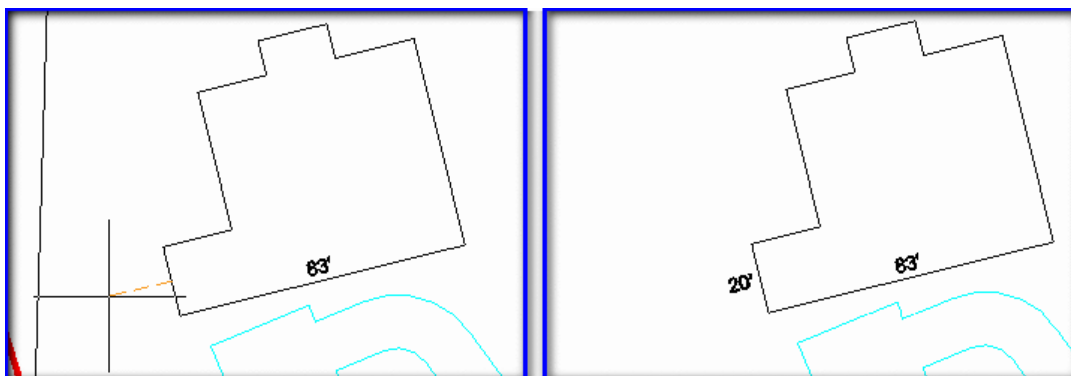
- set the **Layer** to BTXT, so that building dimensions can be frozen to reduce the clutter even more. It is generally a good strategy to use layers for selective freezing and thawing.
- set the **Text Size Scaler** to 0.04
- enable the **Drop Trailing Zeros** (this option will label 17.0' as 17').
- set the **Offset From Line** to 0.02.

Click **OK** on the above dialog box. Run the Annotate – Survey Text – Building Dimensions command. Click on the middle of the bottom segment of the building and then drag the alignment to the right, along the same bottom segment being dimensioned. The resulting label is shown below.

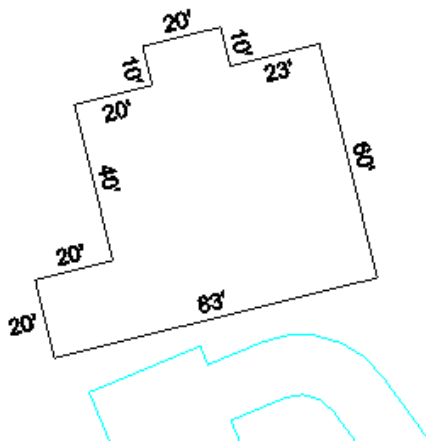


If you had dragged the cursor to the left rather than to the right, with the same near-parallel angle to the line, the 83' would be drawn below the building rather than above.

Another example is shown below. Select Annotate – Survey Text – **Building Dimensions** and click on the left-most segment of the building. Then click roughly perpendicular to the left. This creates a perpendicular, rather than parallel, label as shown below.



Label the rest of the building. Notice that the sides of the building that you are dimensioning are measured in even feet. Because you had selected the **Drop Trailing Zeros** option when you set the **Survey Text Defaults** (and even with the **Decimal Places** defaulting to 0.00), the ".00" is not reflected in the labels:



NOTE: If you choose the wrong direction while you are labeling, a suggestion would be to re-pick the segment and place it in the correct direction and complete the command. Then **Erase** the undesired label(s) at the conclusion of the command.

Next, you will label the offset dimension from property lines to two building corners, the SE corner as offset from the south property line, and the SW corner as offset from the west property line. Because of the options you set in the **Survey Text Defaults** dialog box above, **Offset Dimensions** will be created on layer DIMENTXT, and they will be horizontal, with arrowheads.

Re-issue the Annotate – Survey Text – **Survey Text Defaults** command. The dialog previously shown will reappear. Change the following in the **Offset Dimension Text** grouping:

- set the **Text Size Scaler** to 0.04
- set the **Arrow Size Scaler** to 0.04
- enable the **Dual Arrows Line** option

Click **OK** on the above dialog box. Run the Annotate – Survey Text – Offset Dimensions command and complete the prompts:

[end on] Pick Bldg/Object Corner: pick on the SE building corner

[perp] Pick Line To Offset From: pick on the South property line (before the arc, near the end of the driveway)

The setback is labeled 43.49'. Note the distance suffix is/was labeled as an apostrophe "" rather than "ft." If you desire an alternate label, re-examine the *Characters to Append* control of the **Survey Text Defaults** command.

Re-issue the Annotate – Survey Text – **Survey Text Defaults** command. The dialog previously shown will reappear. Change the following:

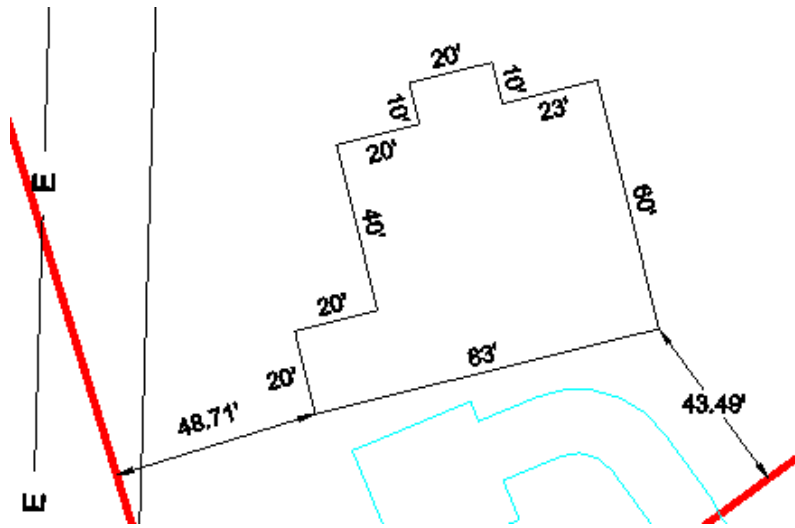
- set the **Text Alignment** to Parallel

Click **OK** on the above dialog box. Run the Annotate – Survey Text – **Offset Dimensions** command again and complete the prompts:

[end on] Pick Bldg/Object Corner: pick on the SW building corner

[perp] Pick Line To Offset From: pick on the West property line (avoid the electric right-of-way lin

The result is shown below:



Notice within the above prompts, the use of the [end on] and [perp] snaps. When Carlson sets a snap for temporary use, it displays the snap within the brackets as shown. A building corner is always an endpoint, so the **end** snap always applies to the first pick. The offset is the perpendicular distance to the property lines, so the **[perp]** snap always applies to the second pick. The **per** (perpendicular) snap applies to offsets from arcs as well. In the case of arcs, the **per** snap finds the shortest, radial distance to the arc.

You could use the **Offset Dimension** command to label the Electric utility right-of-way distance of 50' total by entering **nea** (for nearest snap) for the first pick, then entering the default **per** snap for the second pick on the other side of the right-of-way.

Next, you will add adjoiner ownership text to the property lines. Annotate – Survey Text – **Survey Text Defaults** command. The dialog previously shown will reappear. Change the following:

- set the **Adjoiner Text Justification** to C for centered
- set the **Text Size Scaler** to 0.06

Click **OK** on the above dialog box. Run the Annotate – Survey Text – Adjoiner Text command and complete the prompts:

Pick Line Or Polyline: pick the west property line

Starting point: pick a centering point west of the property for the adjoiner text

Enter text at the cursor, press Enter to finish the command... supply the text as follows:

Brian W. and Mary T. Jones (press Enter)

D.B. 101, P. 37 (press Enter twice) This produces parallel, center-justified text on the west side of the property.

Press Enter to repeat the command and complete the following prompts:

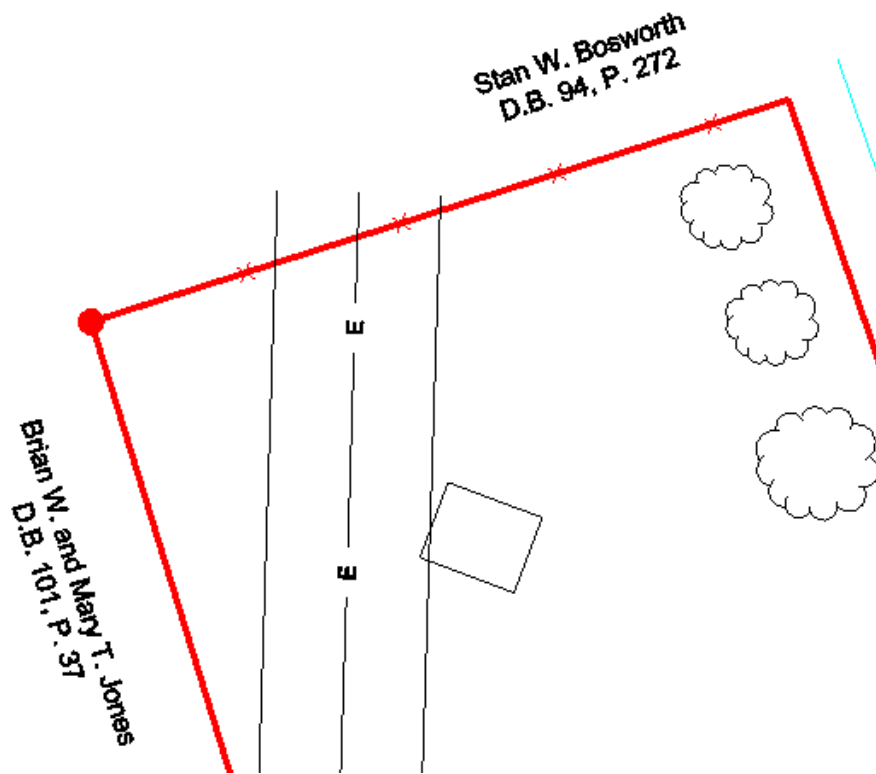
Pick Line Or Polyline: pick the north property line

Starting point: pick a centering point north of the north property line

Enter text at the cursor, press Enter to finish the command... supply the text as follows:

Stan W. Bosworth (press Enter)

D.B. 94, P. 272 (press Enter twice) The results are shown here:



Next, you will add bearing annotation. Select the Annotate – Angle/Distance – **BearingDistance** option to place Bearing and Distance above the line and follow the prompts below:

Select line or polyline [Size/Points]: pick the northern property line to the east, or right side. The bearing direction will be labeled towards the picked end, which is Northeast.

Select line or polyline [Size/Points] (Enter to end): pick the eastern property line closest to the southern endpoint of the line

Select line or polyline [Size/Points] (Enter to end): press Enter

To label the western property line on the lower (western) side of the line, select the the Annotate – Angle/Distance – **BearingDistance** option to place Bearing and Distance below the line and follow the prompts below:

Select line or polyline [Size/Points]: pick the western property line closest to the northern endpoint of the line

Select line or polyline [Size/Points] (Enter to end): press Enter

To label the southern line segment with a leader, run the Annotate – Annotate with Leader – Brg-Dist with Leader command and follow the prompts below:

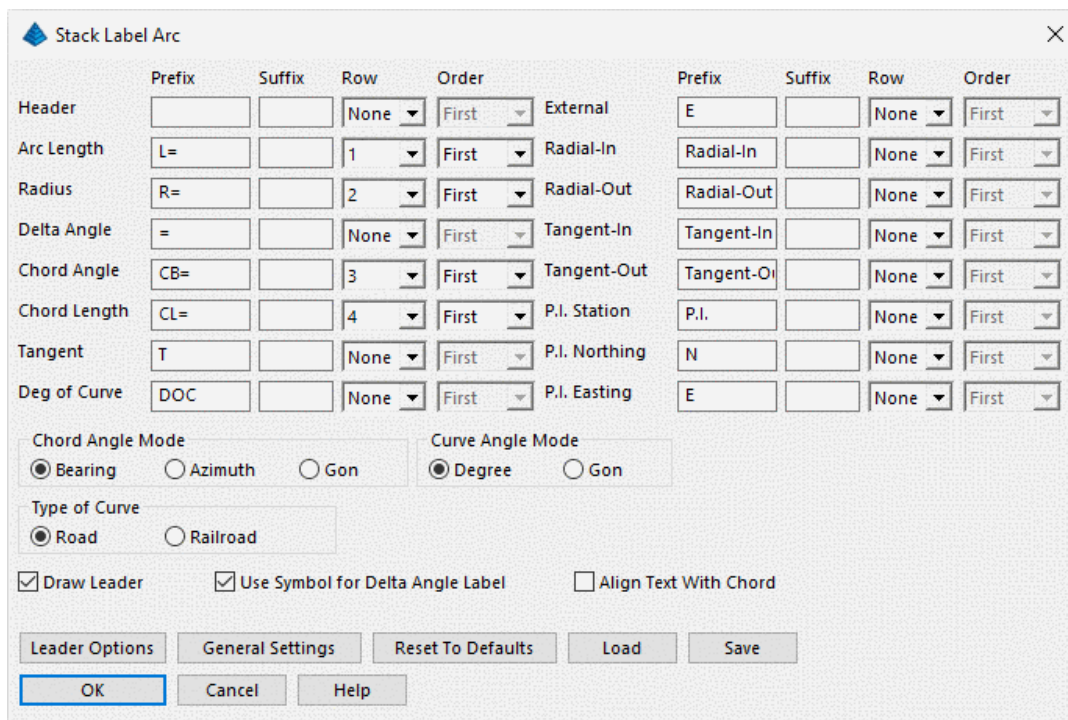
Select line or polyline [Options/Size/Points]: pick the southern property line segment on the southwest side

Pick end point for leader: pick a point for the label position

Select line or polyline [Options/Size/Points] (Enter to end): press Enter

Next, you will want to annotate the arc in the drawing. The label will consist of four entries: arc length, radius, chord bearing (direction) and chord distance. Run the Annotate – Annotate Arc – Stack Label Arc Pick Side command and follow the prompts below:

Options/Points/<Select arc>: type O for Options and the Stack Label Arc dialog box appears



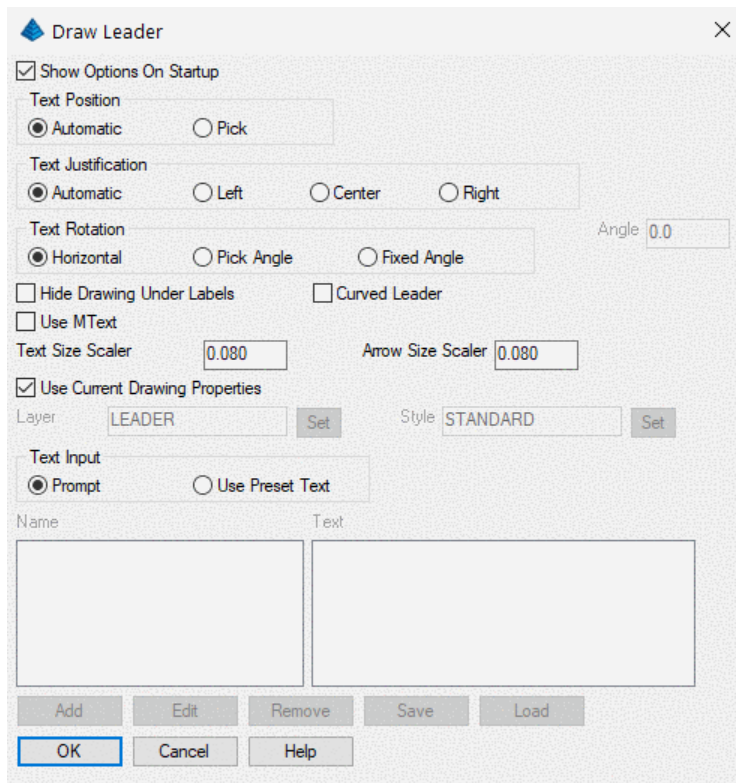
Set the sequence column to 1, 2, 3 and 4 as shown. When you are done with the dialog box, click **OK** to resume the command prompts:

Options/Points/⟨Select arc⟩ (Enter to end): pick the arc

Pick end point for leader: pick a point to the right to place the label. As the cursor moves, the text "ghosts" which allows you to make the best possible placement decision

Options/Points/⟨Select arc⟩ (Enter to end): press Enter

Next, you will label the trees, the shed, and the building using a special leader, for a hand-drafted appearance. Use the Draw – Leader – Special Leader command which displays the following dialog box:



Click **OK** and complete the prompts as follows:

Options/Size/Pick Arrow Location: pick near the southernmost corner of the shed

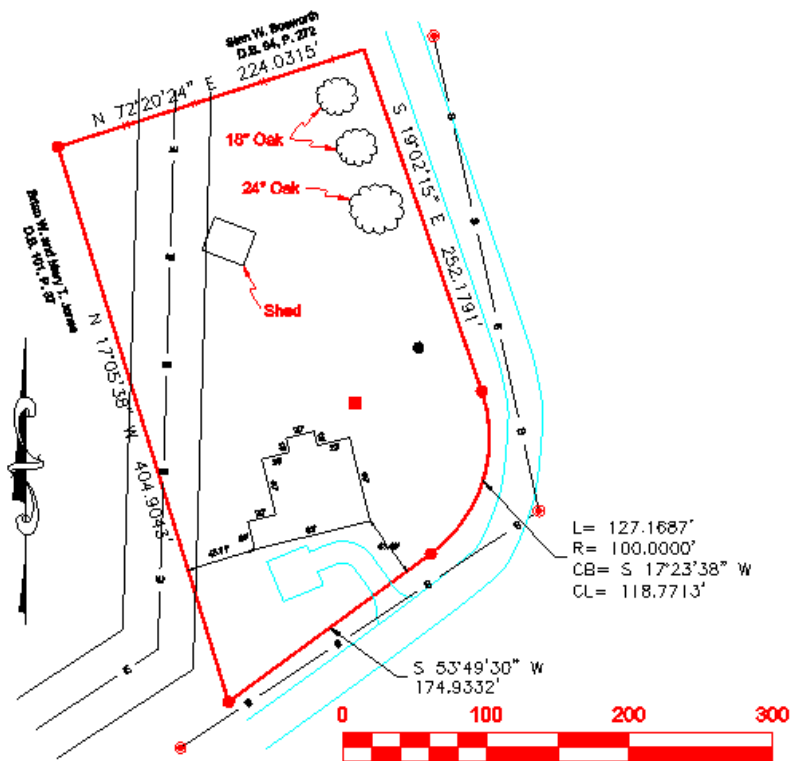
Text location: pick slightly down and to the right

Text: Shed

Text: press Enter

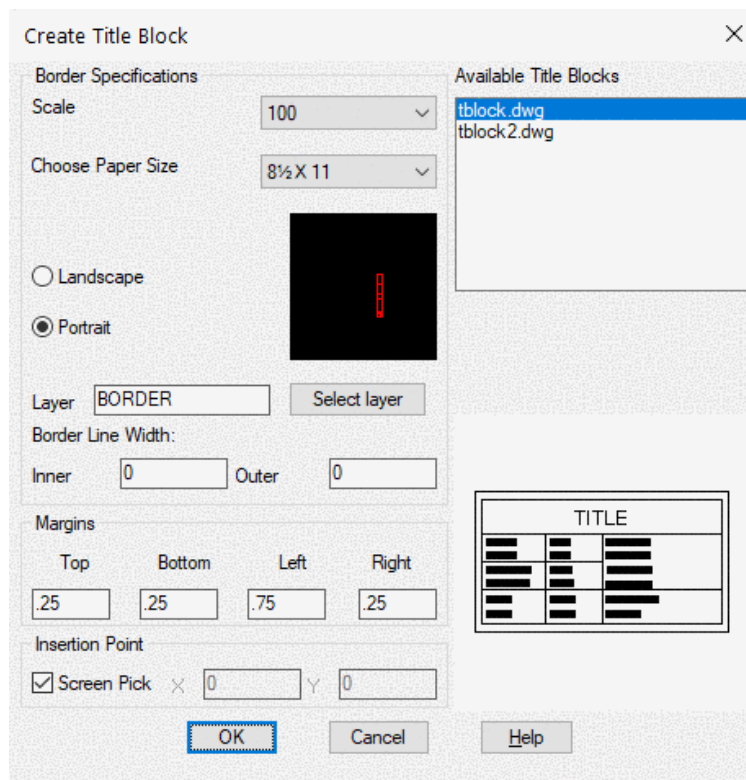
Repeat the process for all the special leader text items shown in the drawing below. In the case of the 18" Oak trees, create just one leader with text and on the second oak tree, create only the leader and then press Enter when asked for Text.

Your drawing should be similar to this one:



You can add a North Arrow and Bar Scale by selecting these options under the **Annotate** menu. When you place the North Arrow, pick your North Arrow symbol, maybe change the scale, and click **OK**. Then pick an insertion point. You place the Bar Scale by answering the prompts and picking a location. Both the North Arrow and the Bar Scale can be moved to desired locations with the Edit – **Move** command.

Next, you will insert a title block with a border. Select the Settings – Title Block – Standard command to display the dialog box below:

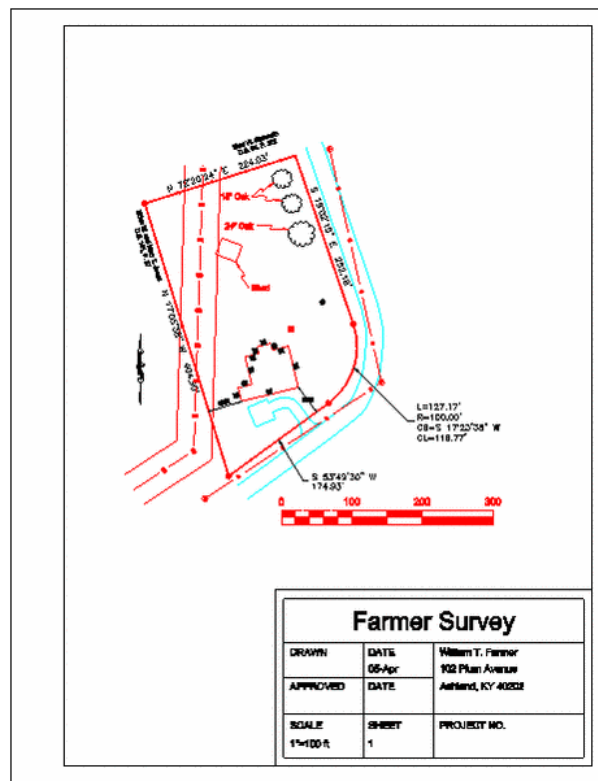


Choose paper size A1 (portrait view, 846mm by 1194mm) and click **OK**. Pick a point below and to the left of the survey in order to locate the lower-left corner of the border outer line. Remember that the title block will be at the bottom, so leave extra room at the bottom.

The following dialog appears, prompting you for the attributes of the title block. Be sure to also click Next in order to enter in more attributes.

Name	Prompt	Value	Multiple line
TITLE	TITLE	Farmer Survey	
COMMENT1	COMMENT1	William T. Farmer	
COMMENT2	COMMENT2	102 Plum Avenue	
COMMENT3	COMMENT3	Ashland, KY 40202	
COMMENT4	COMMENT4		
PROJECT	PROJECT NUMBER		
DRAWN	DRAWN BY		
DATE	DRAWN DATE	05-Apr	
APPROVED	APPROVED		
DATE2	APPROVED DATE2		
SCALE	SCALE	1"=100 ft	
SHEET	SHEET	1	

Your drawing should resemble the one shown below:

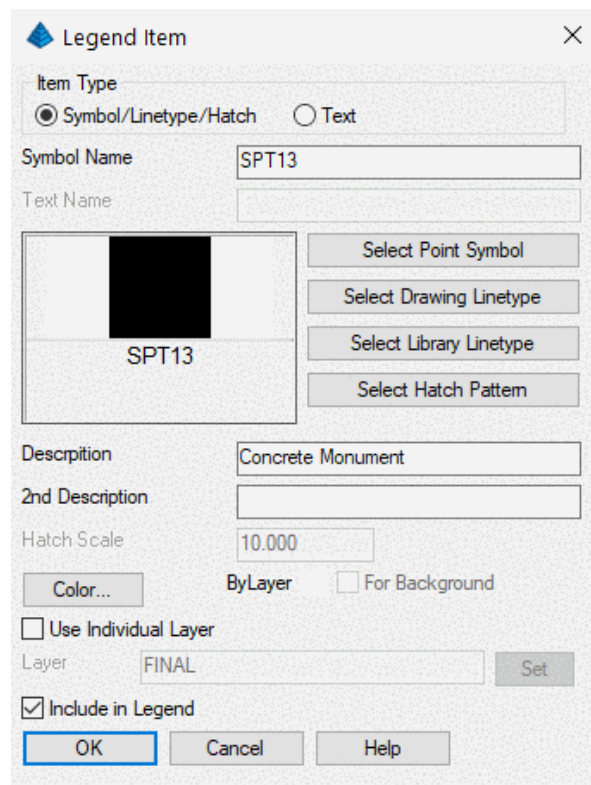


Our next task will add a legend. Pick the Annotate – Draw Legend command. Choose the **New** tab and then open a default legend name (*i.e.* Plat1). When the dialog box appears, select **Add from Drawing**. Window select each symbol/linetype you want to appear in the legend and press Enter. You will then see the symbols that you picked listed.

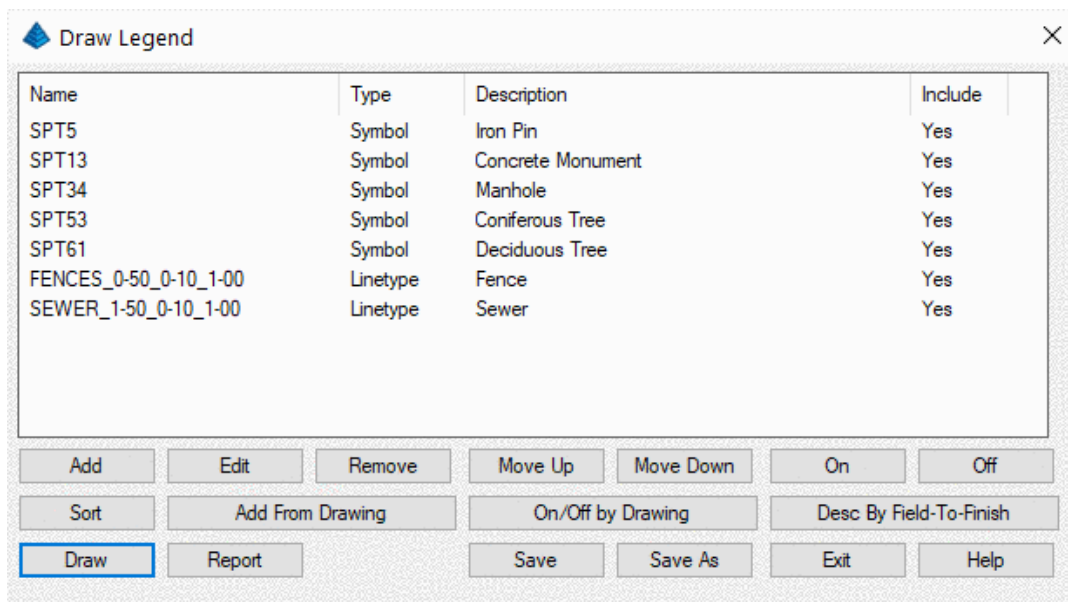
If you want to change the order of the items in the list, use the **Move Up** and **Move Down** buttons, after first selecting and highlighting the item to be moved. After the list is ordered correctly, highlight one item on the list and click the **Edit** button to edit the symbol definition. Edit each symbol definition individually and type the following descriptions in the description box:

- SPT5 = **Iron Pin**
- SPT13 = **Concrete Monument**
- SPT34 = **Manhole**
- SPT53 = **Pine Tree**
- SPT61 = **Oak Tree**

Below is the symbol definition, with Description, for SPT13:



Save the completed legend which is shown below:



Select the **Draw** option from the Legend Definitions dialog box. Set the defaults as shown below:

Draw Legend [X]

Text Size Scaler: 0.100

Symbol Size Scaler: 0.100

Hatch Size Scaler: 2.000

Line Size Scaler: 1.000

Gap Factor: 1.000

Layer Name: LEGEND [Select]

Style Name: STANDARD [Select]

Draw 2nd Description

Draw Legend Title Underline Title

Name: LEGEND Size Scaler: 0.200

Title Line 1: These standard symbols

Title Line 2: be found in the drawing.

Draw Header Symbol: Symbol

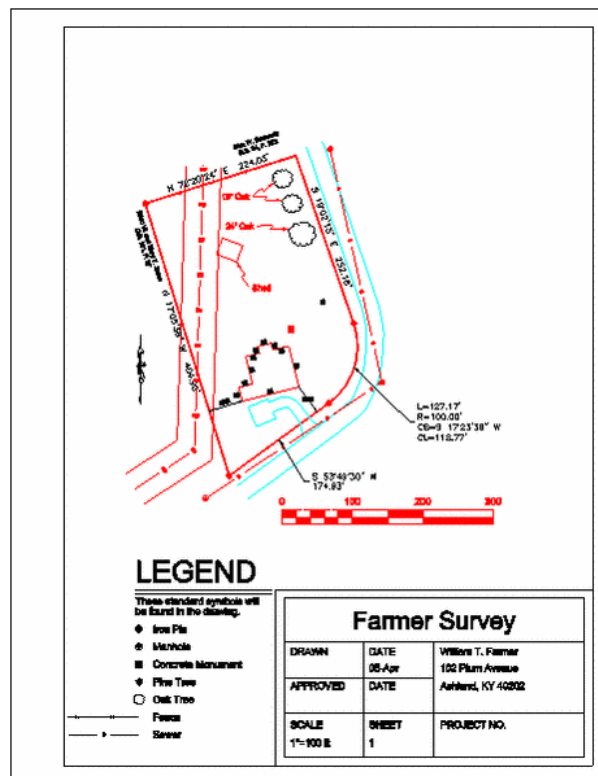
Desc: Description 2nd Desc: Code

Draw Grid Lines

Layout: Left Justified Right Justified Right Symbol

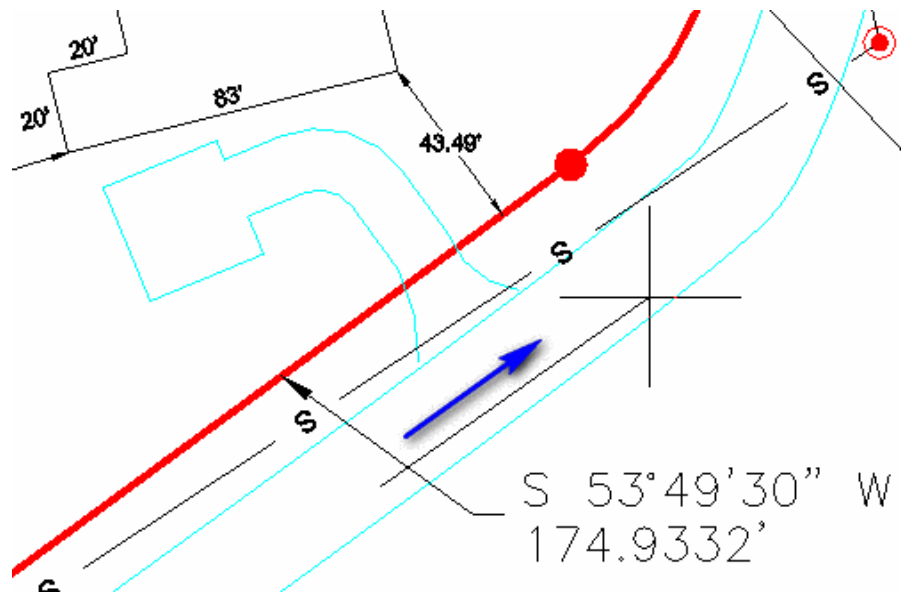
[OK] [Cancel] [Help]

Click **OK**. Pick a point for the legend at a coordinate location approximately at 5311.5,4275 (CAD coordinates) then click Exit. You may need to adjust the position and/or portions of the legend to fit in the tight space. You also may need to move the previously drawn bar scale. Use the Move command to do this. The following shows the drawing to this point:



NOTE: If you wish to reset the spacing of the sewer and electric utility annotation, set a desired value for the LTSCALE box in the Settings – Drawing Setup dialog box (the setting is 50 in this example).

Next, you will use DText (Dynamic Text) to label the road and MText (Multiline Text) to create a certification block. Zoom in on the area shown below:



At the command line, type **dtext** and complete the following prompts:

Specify start point of text or [Justify/Style]: j

Enter an option [Left/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR]: r

Right point of text: pick a point as shown below, just to the left of the leader annotation

Height of text <8.00>: 10

Rotation angle of text <E>: pick a point as shown below by the location of the crosshair

Text: Meadow Lane

Text: press Enter

This right-justifies the label **Meadow Lane**, ending it before it contacts the leader line.

Now you will enter a certification using MText. The MText command stretches an entire block of text. This command breaks up the lines in the block of text depending on how you edit and adjust the MText window. First, use the View – **Extents** command to view the entire drawing. Then type **mtext** at the command line and follow the prompts:

Multiline Text: First corner for block of text: 5660,4980 (or pick an approximate location)

[Justification/Rotation/Style/Height/Width]/<Opposite corner for block of text>: pick a point below and to the right of the first, but inside the inside border line

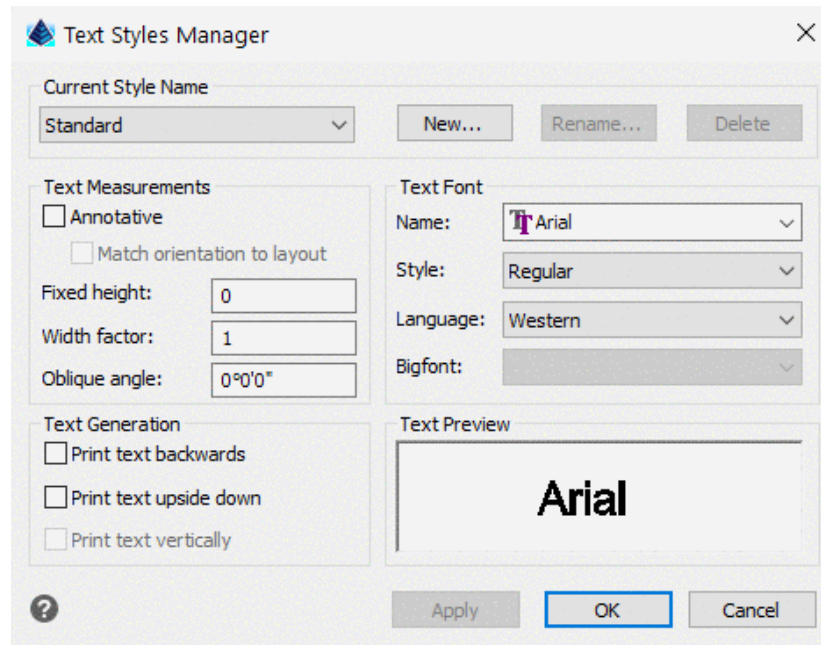
You now see the MText interface that displays word-processing style editing options. Indicate text height of 8 (press Enter to commit) then type the following into the dialog box: Surveyor's Certification

I do hereby certify that the survey shown heron was performed under my direction by method of random traverse and that the error of closure was 1:52544.

John Doe, PLS #1234 The command "wraps" the text when it runs out of space in the MText window. Click **OK** at the upper right to place this text into the drawing.

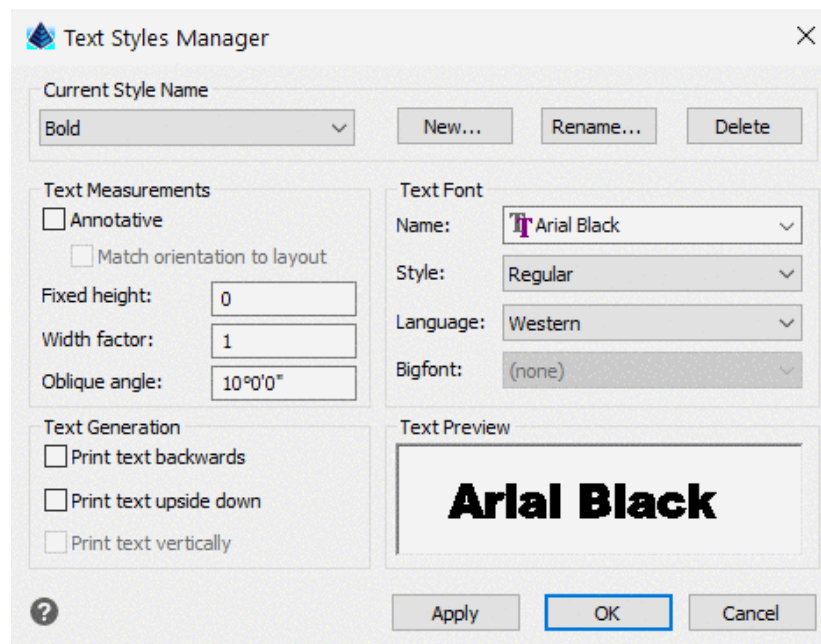
After the MText is placed, you can click on the text to activate the grips. All four corners highlight as grips. When you pick on a grip, you can expand or change the shape of the MText rectangle. When you do this the text adjusts automatically, wrapping into more or fewer lines of text based on the size of the MText window. You can also move the entire text block to a new location.

Next, you will define a text style and then add text using that style. Use the Draw – Text – **Set Style** command to display the dialog box below:



Click **New**, enter **Bold** in the New Text Style dialog, and click **OK**.

Define the **Bold** style consisting of the **Arial Black** font and an Oblique Angle of 10° by entering the settings as show below:



Then click **OK**. Now, run the DText command by typing **dtext** at the command line, and place the text at the top of the drawing as follows:

Specify start point of text or [Justify/Style]: pick a point near the northwest corner of the drawing

Height of text <8.00>: 20

Rotation angle of text <N53d56'51"E>: E (for due East)

Text: William T. Farmer

Text: press Enter

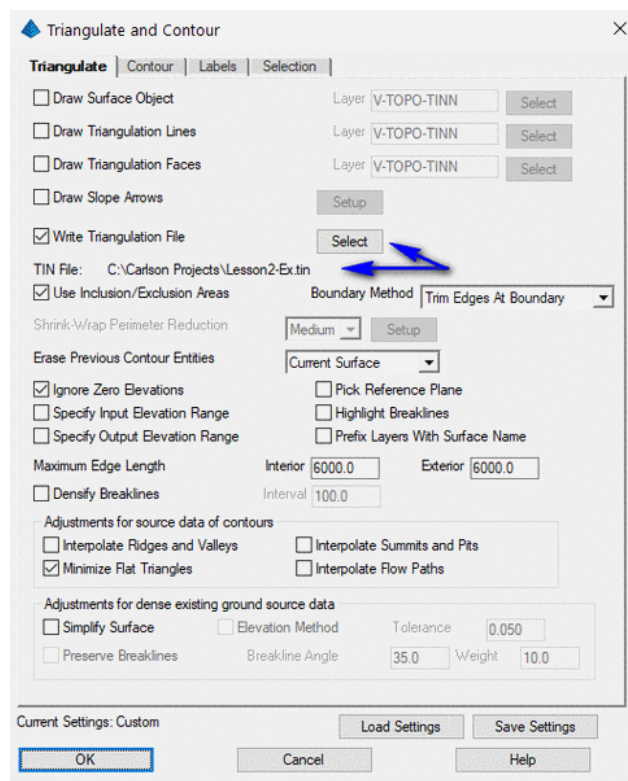
Next, you will create an area label for the drawing. Select the Area/Layout – Area Defaults command. Select the Acres entry and click **Edit** to change the Precision for the Acres labels to 2 decimal places. Click **OK** to dismiss the *Acres* settings dialog box and click **OK** to dismiss the *Area Defaults* dialog box.

Select the Area/Layout – Area by Lines & Arcs command. When prompted to **Select objects**, pick the two polylines that taken together, completely enclose the property.

Pick an area labeling centering point for the area label under the **William T. Farmer** title at the top of the drawing.

Next, bring the points back and draw a contour map. To re-see the points, use the View – Thaw/On All Layer command.

Initiate the Surface – Triangulate & Contour command. On the Triangulate tab, set the following values:



Also make sure to enable the **Use Inclusion/Exclusion Areas** option. Navigate to the **Contour tab** and set the **Contour Interval** to 1.0. Click **OK** and then answer as follows:

Select Inclusion perimeter polylines or ENTER for none.

[FILTER]/<Select entities>: press Enter for none.

Select Exclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: select the building and the shed, and then press Enter. Since the building and shed are closed polygons acting as exclusion perimeters, the contours will not pass through them when they are created.

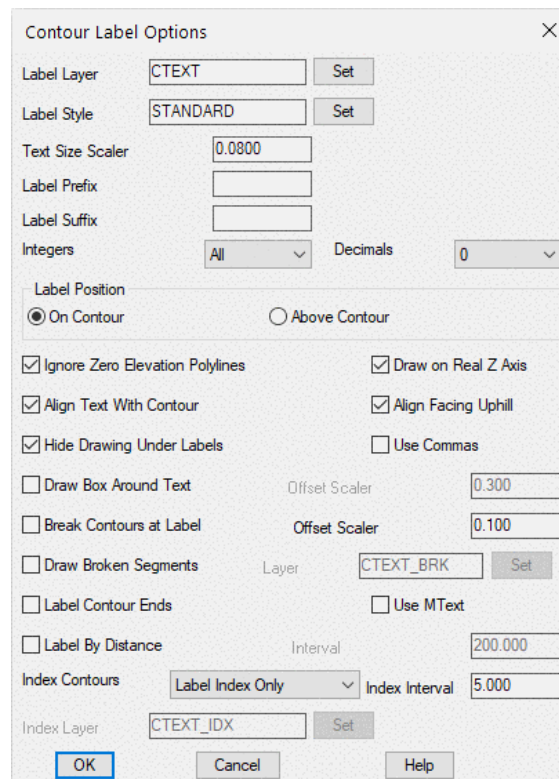
Select points and breaklines to Triangulate.

[FILter]/<Select entities>: all

[FILter]/<Select entities>: press Enter

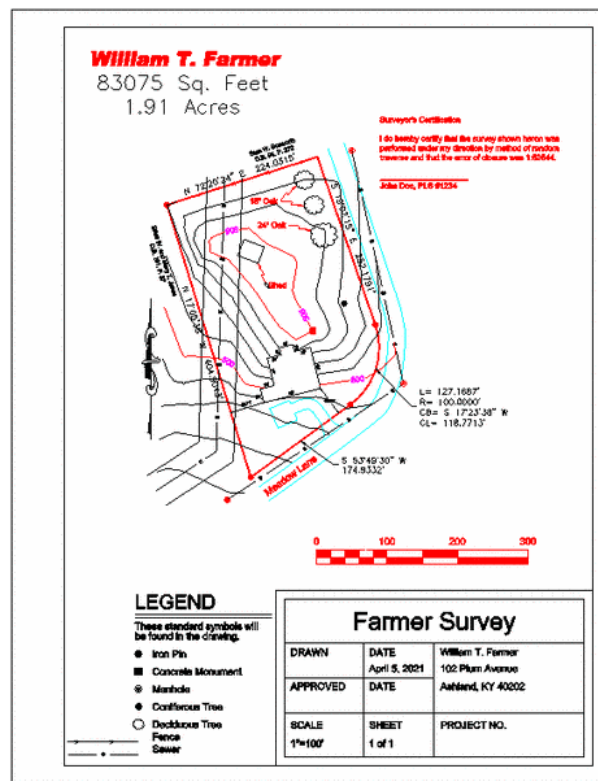
The contour map is created. Freeze the points again by using View – **Freeze Layer by Select** and picking one of the points.

Next, label the contours (this could have also been accomplished via the **Triangulate & Contour** interface but this approach gives us more control). Run the Surface – Contour Utilities – Contour Elevation Label command. Select **OK** after matching the settings in the dialog box shown here:



Specify the surface model (*.tin) created earlier and click the **Open** button on the Carlson File Selector dialog box. Now pick two points that cross through one or more Index contours. The contours are automatically labeled using the specified text style.

The Completed Plat is shown here:



If you have not saved your drawing for awhile, now is a good time to do it. Use the File – **Save** command.

This completes the Lesson 2 tutorial: **Making a Plat**.

Field to Finish for Faster Drafting

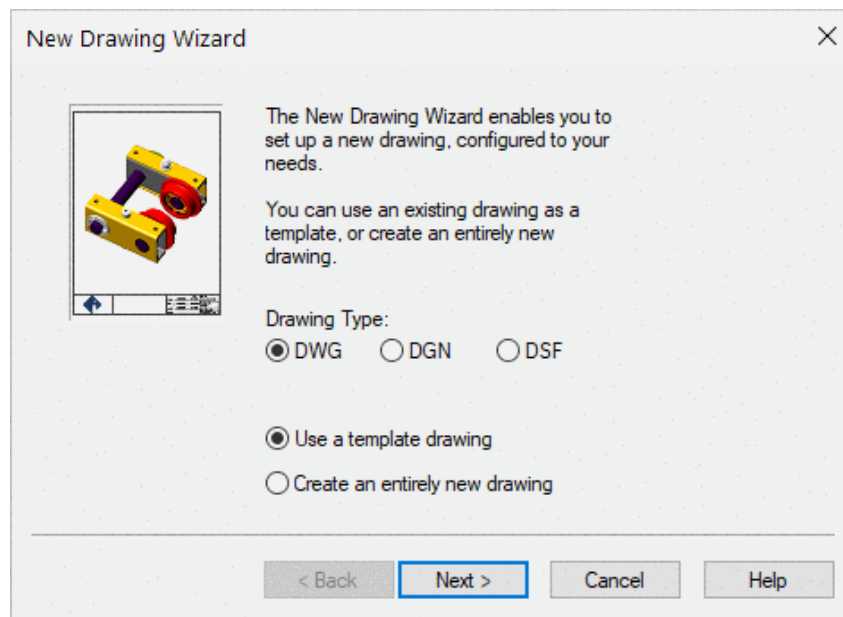
In this lesson, you will make a plat using "Field to Finish" techniques, with the help of the **Startup Wizard**.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **New Drawing**.
 - If you get the Startup Wizard dialog box, click **New**.
 - If you are taken directly into CAD, click the File – **New** command.

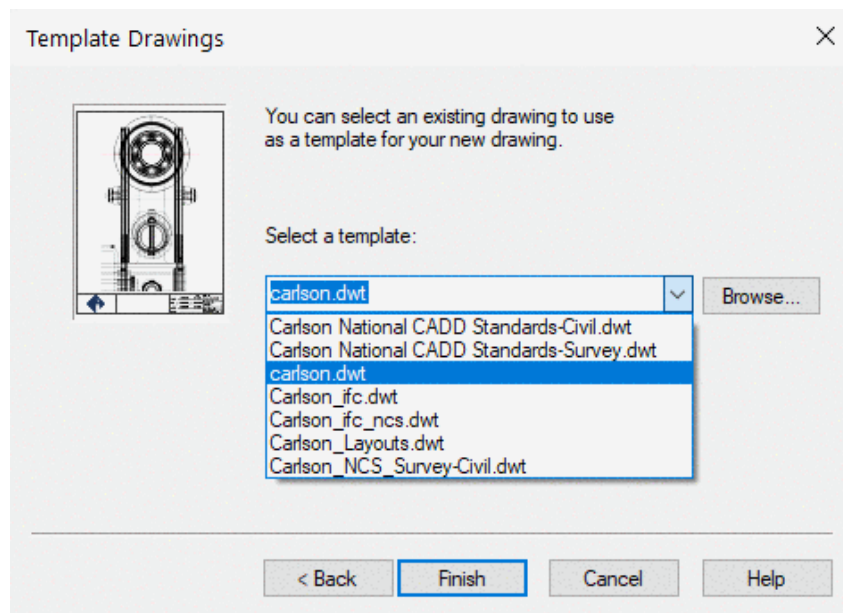
NOTE: If you want to consistently use the **Startup Wizard**, issue the Settings – Carlson Configure – General Settings and enable the **Use Startup Wizard** option. Click **OK** to dismiss the dialog boxes. The **Startup Wizard** will be used the next time the software is started.

The first of several Startup Wizard dialog boxes appears:

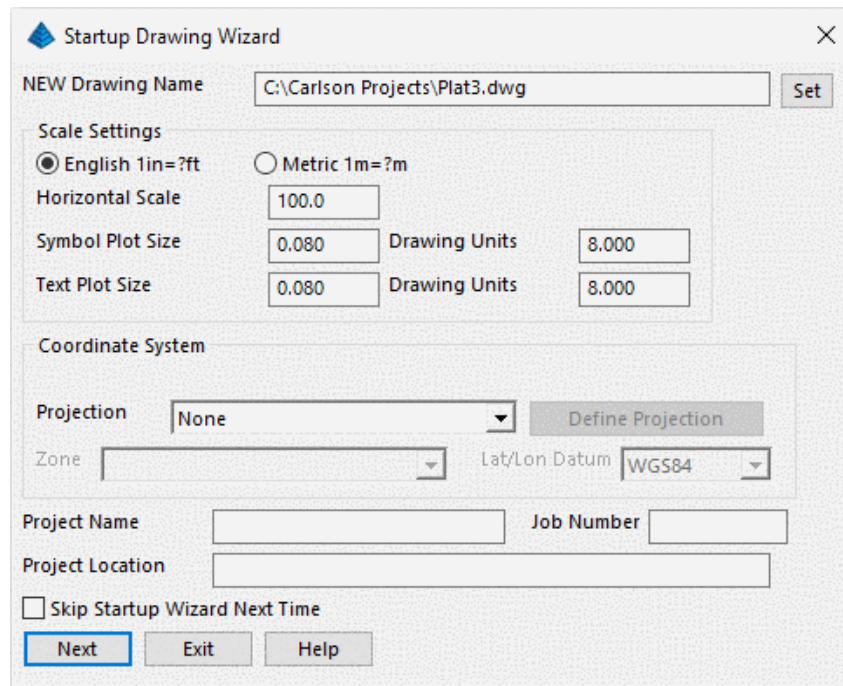
- (a) Choose the DWG document type and the desire to base the document on a **Drawing Template** as illustrated below and then click **Next >**:



(b) Choose the **carlson.dwt** as illustrated below (or *surv.dwt* if *carlson.dwt* is not available) and click **Finish**:

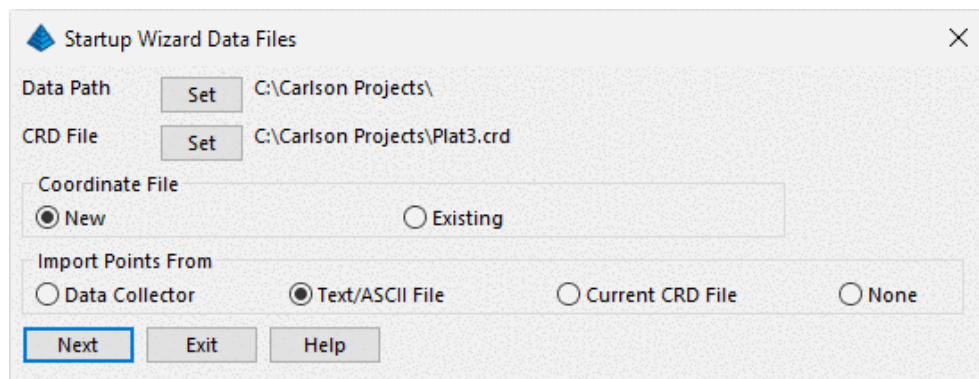


(c) We can now begin the more pertinent settings for the project to come based on some preliminary settings that should be similar to the default scenario shown below:

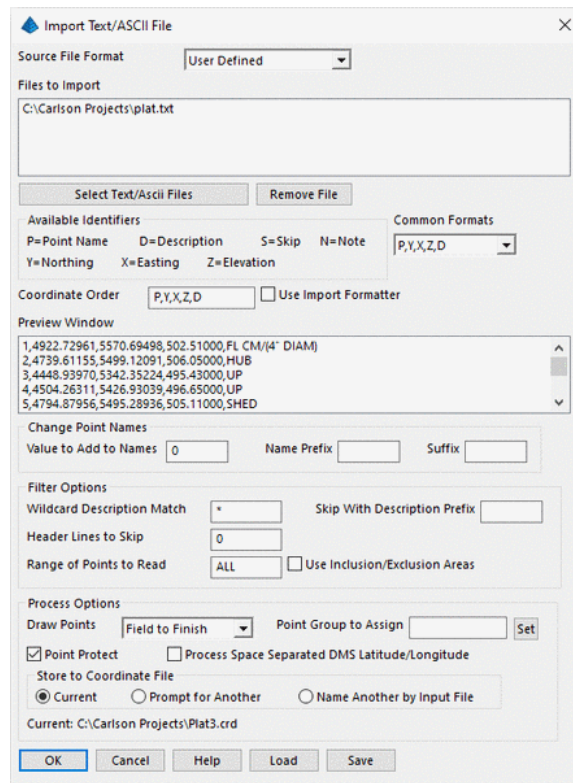


Click **Set** at the top of the dialog box, and enter in a NEW Drawing Name called **Plat3**. Verify that the other settings match the settings shown below, and click **Next**.

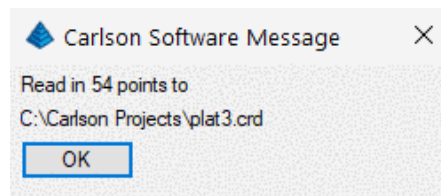
- (d) You will see the Startup Wizard Data Files dialog to set/confirm where to store data and indicate an information source for points/coordinates. Set/match the values as shown below and click **Next**:



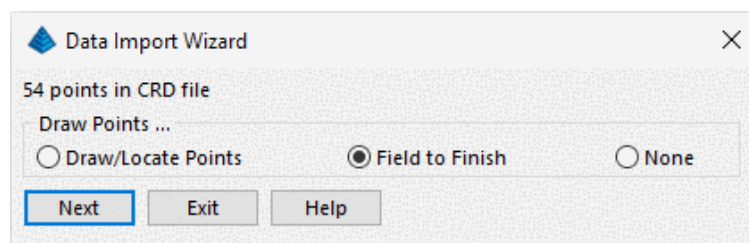
- (e) The Import Text/ASCII File dialog box appears. Our source is the same file as in Lesson 2 (e.g. the ASCII-based **Plat.txt** file). Click the **Select Text/ASCII Files** button and navigate to and choose the **C:\Carlson Projects\Plat.txt** file and click Open when ready. Set the other values as shown in the dialog box below paying particular attention to the **Draw Points** toward the bottom of the dialog box and click **OK** when ready:



(f) When you complete the import, the following dialog box appears (click **OK** to dismiss):

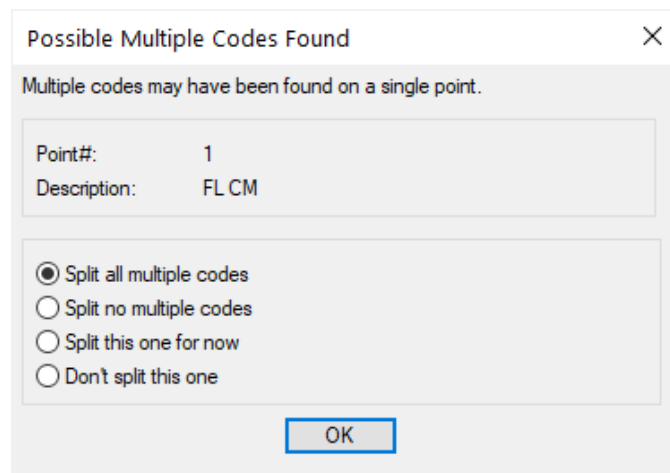


(g) A **Draw Points** confirmation dialog box displays. Set/confirm the Field to Finish option, and click Next:



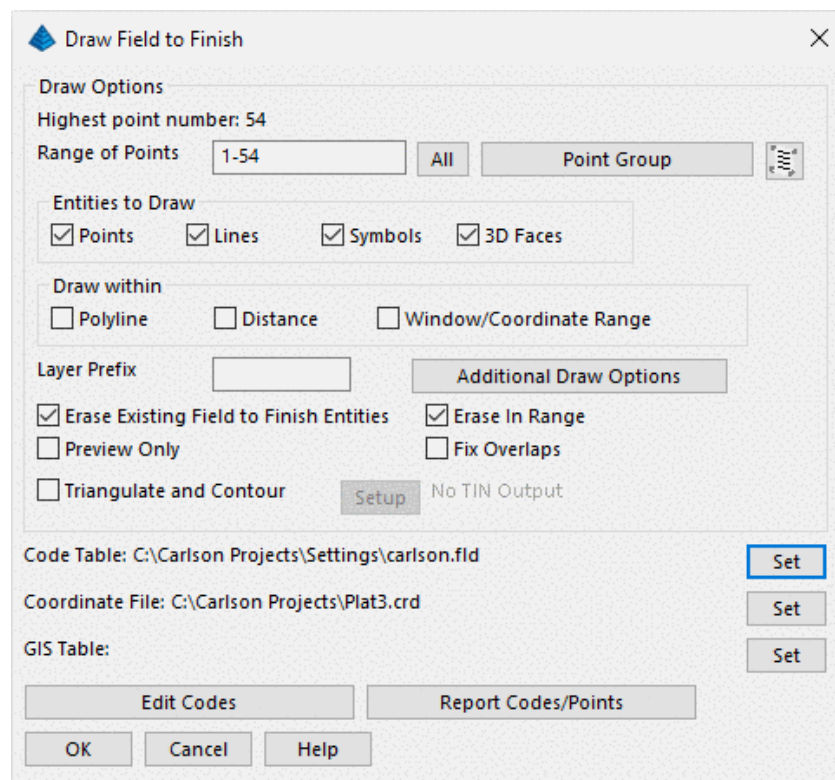
(h) If this is your first time running Field-To-Finish, then the program will prompt to select a **Field Code Definition** file. In this case, select the **C:\Carlson Projects\Settings\carlson.fld** file and click Open when ready.

(i) A dialog box now appears with a warning that some codes have two descriptions:



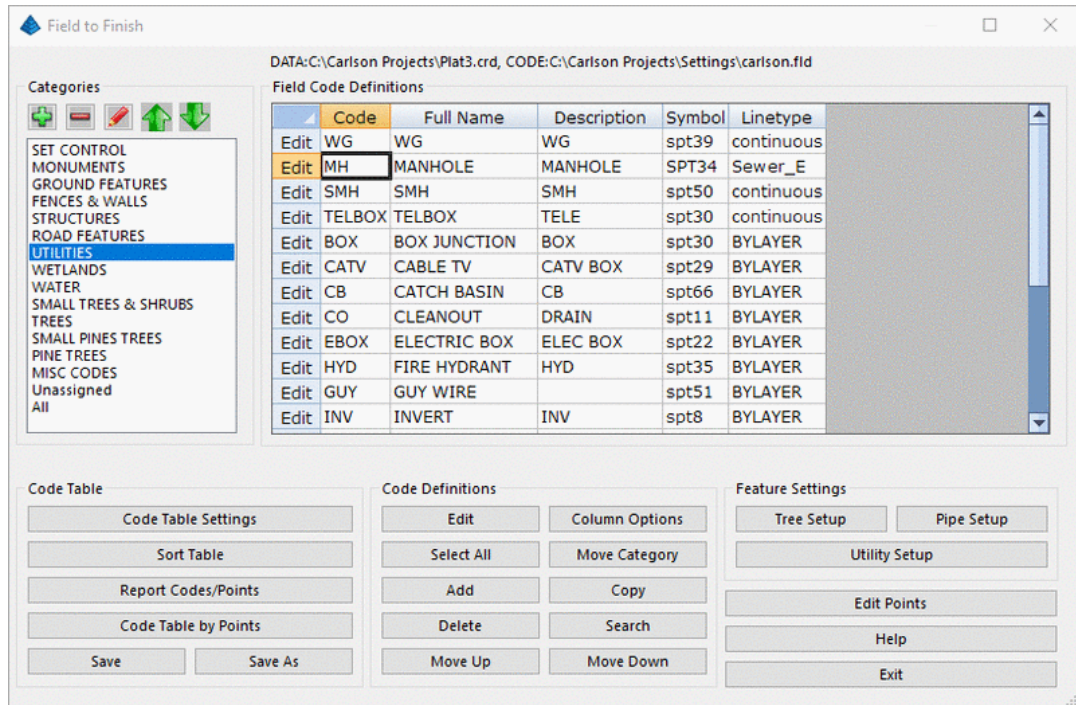
The command is asking whether these codes are to be treated as two separate descriptions, or as one description that has a space in it. Choose the **Split all multiple codes** to tell the command that codes with spaces are really two separate descriptions and click **OK**.

- (j) We are about ready to place the Field to Finish data into the drawing. Take a moment to review (and optionally set) the values shown below:

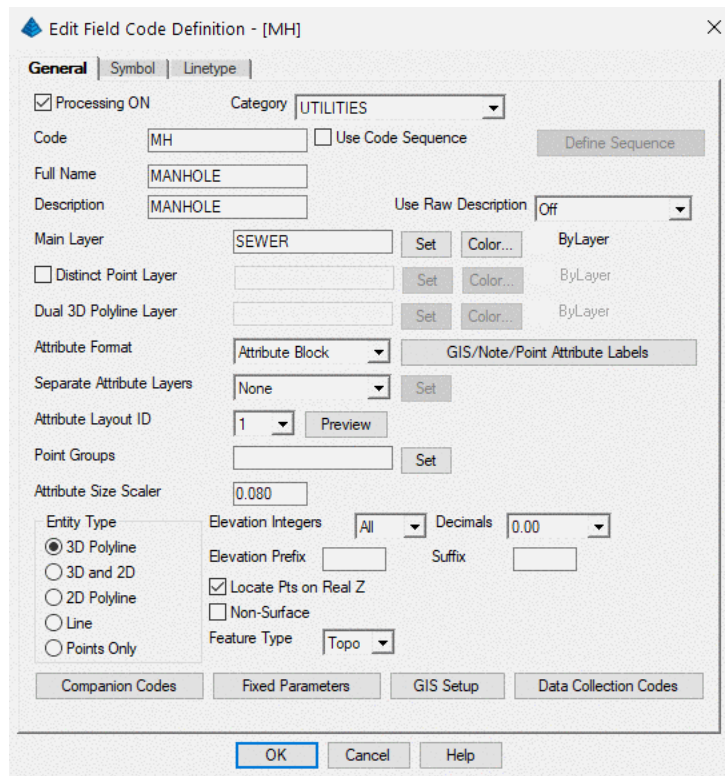


- (k) Let's review/set some additional options. Click **Additional Draw Options** to display the dialog box below:

prompted about **Possible Multiple Codes Found**, accept the default to **Split all multiple codes** option and click **OK**. On the Draw Field to Finish dialog box, select the **Edit Codes** button. This takes you to the **Field to Finish** dialog box as shown below:



The display window shows a list of point codes, such as IP for iron pin and FL for fence line, that are converted to special symbols and linetypes by **Draw Field to Finish**. For an example of how the codes are used, look at the sewer line running from point 52 to 53 to 54 (the southernmost point), which is based on a field code of MH. Select the **Utilities** category and select the **MH** (for Manhole) code as shown above, and then click **Edit**. The following dialog box is displayed:



The MH code has several attributes that are used by **Draw Field to Finish**. For this code, **Draw Field to Finish**:

- places the manhole on layer SEWER
- labels a text description of MANHOLE underneath the symbol (Descriptions can be upper or lower case)
- Click on the **Symbol** tab and notice that it draws a manhole using the symbol SPT34
- Click on the **Linetype** and notice that it draws a sewer line with the letter S (for Sewer)

When you are done looking at the MH field code definition dialog, click **OK**.

Other codes have different attributes:

- LP is set only to draw a symbol and text (Light Pole), but not to draw linework
- FL (for fence line) is set to draw linework but not corner symbols or points descriptions

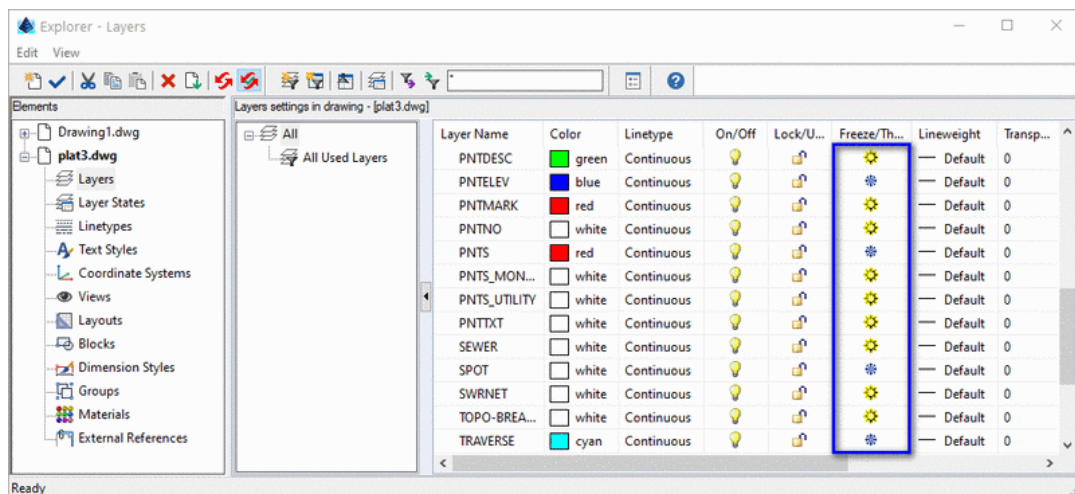
The CAD symbology attributes for a code depends on the entries in the Layer, Description, Entity Type, Set Symbol and Set Linetype options (among others).

The "Carlson.fld" Field to Finish code table is provided with Carlson Software. This table shows one possible system, but with likely far too many codes for a field crew to remember. You can make your own code table by:

- choosing the **Code Table Settings** option from the Field to Finish dialog box
- choosing the **Set** button at the top right, and,
- selecting the **New** or the **Existing** tab from the top of the **Specify the Code Definition File** dialog box.

Click **Exit** to dismiss the Field to Finish dialog box and then click **OK** to re-draw the data again.

- Use the **Inquiry – Layer ID** command to verify the layers of the various entities that have been placed into the drawing. Pick on the fence line, the road and the utility line, and notice the different layers (FENCE, EOP, UTILITY). You can also use the **Inquiry – Drawing Inspector** command to hover over entities to see their layer. You should study the layers in a drawing before deciding what to freeze and thaw. To reduce clutter on the screen, select the **View – Layer Control** command (the appearance of this dialog box might vary from that shown below). Freeze the following layers by turning the sun into a snowflake:
 - PNTS
 - PNTELEV
 - SPOT



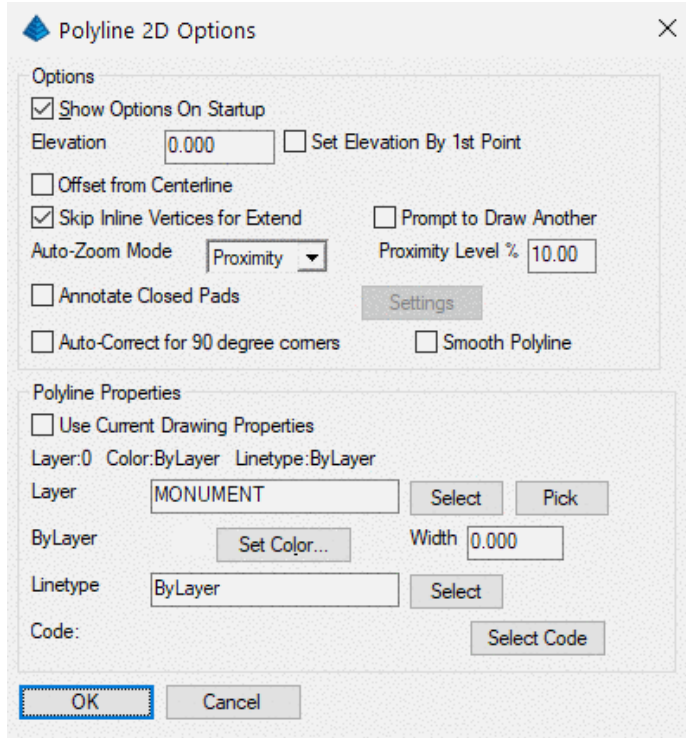
Exit the *Layer Manager* when done.

- Now you will do some drawing cleanup. Note that a single property line is drawn from point 8 to 9 to 10 and to 15. The chord from point 10 to 15 should be an arc. You will erase the segment from 9 to 10 and from 10 to 15, so that you can re-draw it, establish the tangent, then draw the arc and finish back at point 1. Click **Edit – Polyline Utilities – Remove Polyline – Remove Polyline Segment** command and follow the prompts below:

Mode: break polyline at removal

- Select polyline segment to remove [Options]:** Pick the segment from 9 to 10
- Select polyline segment to remove (Enter to end):** Pick the segment from 10 to 15
- Select polyline segment to remove (Enter to end):** press Enter

To draw the correct polyline, use the Draw – 2D Polyline command (keyin **2dp**). Presuming the **Polyline 2D Options** dialog box appears, use the **Pick** button to and select the western property line to get its layer as shown below (this keeps the new linework and the existing linework on the same layer) and click **OK** and follow the prompts below (these steps are also covered in Lesson 2):



9-10

This instructs the 2D polyline to be drawn through Points 9 and 10.

<Pick point or point numbers>

a

This indicates a tangential arc to be drawn starting from Point 10.

[Radius pt/radius Length/Arc length/Chord/Second pt/Undo/<Endpoint or point number>]:

15

This indicates the arc shall terminate at Point 15.

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]:

1

This instructs the polyline to be drawn back to the starting point. The segment from 15 to 1 is not guaranteed to be tangent to the previous arc.

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]:

press Enter

This completes the needed polyline segments.

Turn the 2-sided shed into a 4-sided shed by issuing the Survey – 4 Sided Building command. Complete the command as follows:

Options/Points/<Pick a line or polyline>: Pick the shed

Options/Points/<Pick a line or polyline> (Enter to end): press Enter

Next, we will create a building which will be less complex than the building you created in Lesson 2 but with an added twist. Thankfully, **Field to Finish** already started one building wall segment for us. Click the Edit – Extend – by Distance command. Complete the prompts as follows:

Pick arc, line or polyline to extend: pick the western side of the small line segment west of the 12" pine and north of the driveway (This makes the arrow point toward 18 rather than 19. Now you can go clockwise:)

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): T50 ("T" or "t" means "total" distance or "to" the distance - so extend "to" 50 feet total)

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): L62.5

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): L35

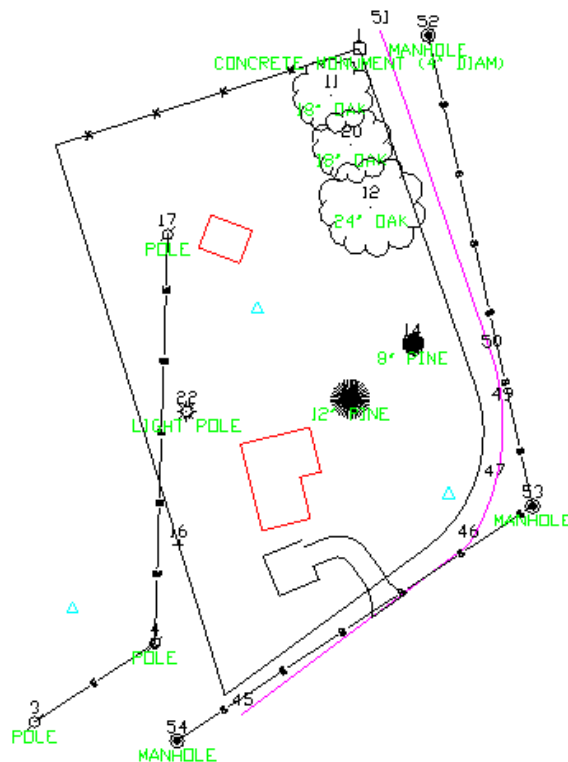
Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): L30

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): R15

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): C

Enter or pick distance to Draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,V,Z,?,Help): press Enter

In Extend by Distance, the "T" option (for total distance) solves the dilemma of making an existing line of unknown length extend to an exact known length. The end result is shown below:

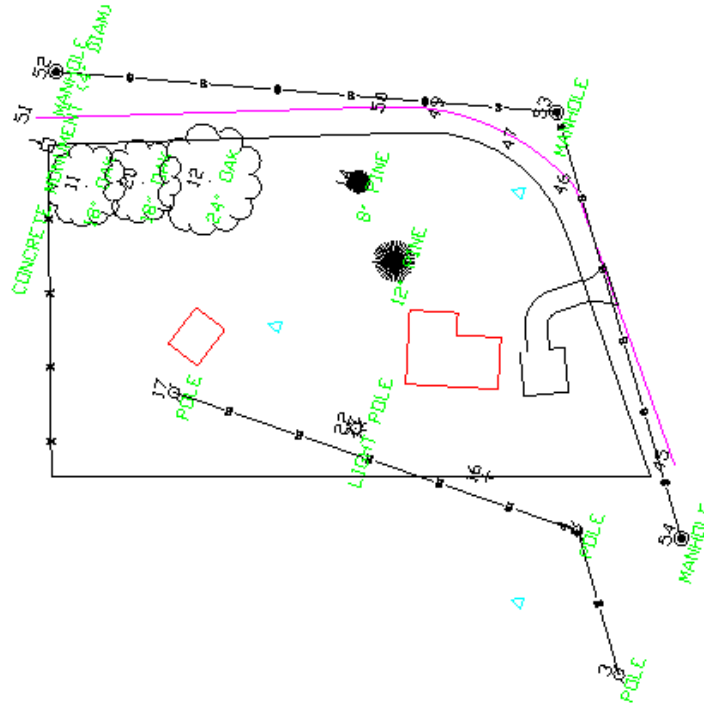


NOTE: Not every drawing can (or should) be plotted "due North." Sometimes the plat needs to be oriented so that property lines and important features run nearly left-to-right or top-to-bottom on the plotted page. In this drawing, you want the western line from point 8 to point 9 to run left-to-right on a sheet that will be plotted in landscape

style (longer left-to-right than top-to-bottom). Use the View – Twist Screen – Line, Polyline or Text command to accomplish this and follow the prompts below:

Pick a line, polyline or text: pick the western line from point 8 to point 9, closer to point 9
Twist azimuth for selected POLYLINE (Use 90 for Due East) <90.0>: press Enter

Now the drawing appears as shown below:



NOTE: Within CAD, the desired **World Coordinate System** (as indicated by the USCICON, if shown at the lower left) is still maintained and coordinate locations for all points and directions of lines have not changed. In other words, only the orientation of the data on the screen has changed!

Much of the text in the above drawing, such as tree sizes and types, the manhole text, and the light pole text can be used in the final drawing. But some of the text, such as the text placed for iron pins and poles, can be fully described in a **Legend** without the redundancy of plotting to the screen. However, if you use the Edit – **Erase** command to remove the iron pin and pole text, the entire point will be erased because the attributes are grouped with the point. Instead, use the Points – Erase Point Attributes command and follow the prompts below:

Select attribute(s) to erase (Enter to end): pick the 3 poles and the 4 iron pins and then press Enter when complete

Let's correct the orientation of the attributes. Use the Points – Twist Point Attributes command and follow the prompts below:

Twist by [<Twist screen>/Azimuth/Entity segment/Follow polyline]? press Enter for Twist Screen
Attributes to twist [<All>/Symbol/Name/Elevation/Description]? press Enter for All
Enter angle relative to current twist screen<45.0>:? 0 (press Enter)
Select points from screen, group, or by point number [<Screen>/Group/Number]? press Enter for Screen
Select Carlson Software points.

[FILter]/<Select entities>: all (press Enter)

[FILter]/<Select entities>: press Enter

The points then twist back orthogonal to the screen, reading once again from left-to-right.

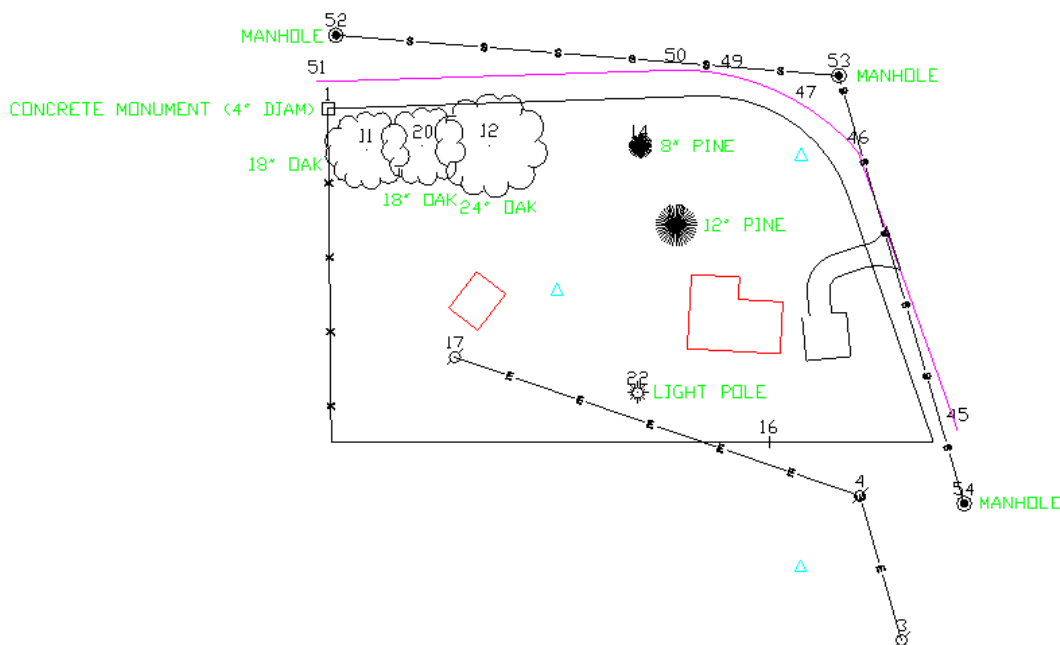
The remaining descriptions associated with the points can be used in the final drawing, but they should be moved slightly for a better appearance. For example, the tree descriptions would look better if they were not inside the tree canopies. Use the Points – Move Point Attributes. The steps of the command are:

Select attribute(s) to move: pick an attribute to move

Displacement: pick the new location for the attribute

Rotation: press Enter to keep the existing orientation or pick the new orientation for the attribute

Then the command repeats. Notice how the text "ghosts" as it moves, which helps you place it in the best position. Try to duplicate this result:

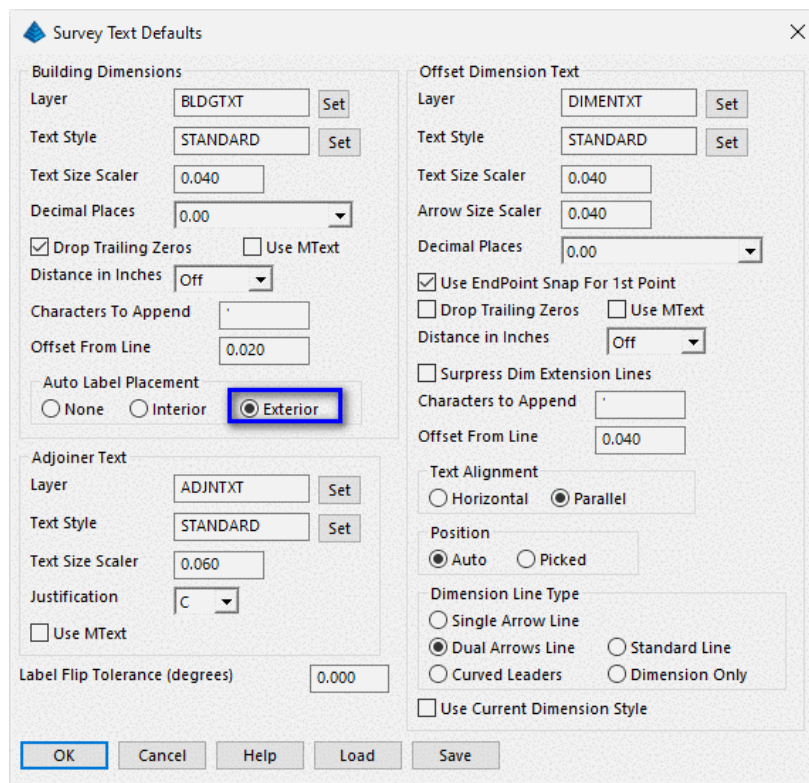


Because of the earlier **Twist Screen** command, the E's in the electric utility polyline are upside down. Use the Edit – Text – Flip Selected Text command to correct this through the prompt below:

Select the text to flip.

[FILter]/<Select entities>: pick the upside down E's individually and then press Enter (see above image)

We wish to apply building dimension labels to the exterior of the building created earlier. Click the Annotate – Survey Text – Survey Text Defaults command and set the **Exterior** option and set other values as shown below:



Click **OK** when complete. Then click the Annotate – Survey Text – Building Dimensions command and pick on the house.

In preparation for the vector annotation to follow, let's check (set) some desired annotation values. Click the Annotate – Annotate Defaults command. On the various tabs, check/set the following:

- **General** tab - **Previous Labels**: Erase
- **Angle** tab - **Bearing Annotation Precision**: Deg, Min, Sec
- **Distance** tab - **Decimals**: 0.00

Click **OK** when ready.

We now wish to annotate linear bearings and distance, as well as arcs. Pick Annotate – Auto Annotate command. When the dialog box appears, under the **Lines** tab, select the options you would like to use so that the bearings and distance labels appear as you would like and click **OK**. Then pick the three polylines that fully define the perimeter:

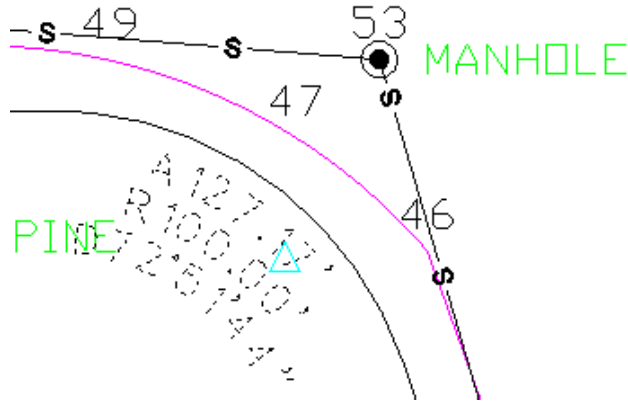
- the fence line
- the polyline containing the arc
- the lower polyline, which is still the western polyline although you have twisted the screen so that it runs along the lower portion of the drawing

Use the Move command to move the bearing and distance labels to avoid overwriting other features.

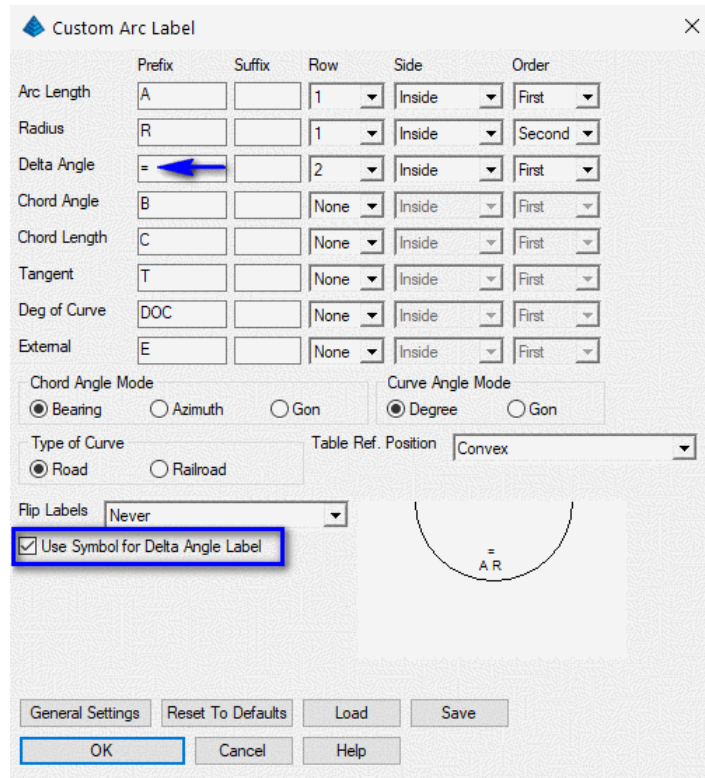
NOTE: When you move the lower bearing or distance label (the 404.90' or S 17°05'38" E bearing) to the left, you want to move perfectly level to the screen, since this was the line you used to twist the screen, and it runs perfectly left-to-right. To do this, press the F8 function key to activate Orthogonal (ortho) mode. Then pick the desired piece(s) of text and move it/them to the left, picking the final position. After you move the item(s), press F8 again to turn off Ortho. Sometimes you will load a drawing from another client or source, and the Ortho setting has been left on. This may initially confuse you during the Move commands. Press F8 to deactivate Ortho. Notice that F8 works even with **Twist Screen** active.

Auto Annotate may center the arc annotation above and/or below the arc, which may cause the arc data to overwrite the surveyed edge-of-pavement (EOP) polyline. Let's experiment with some more precise annotation.

If needed, erase selected portions of the arc annotation via Edit – Erase – **Select** (keyin e). When prompted to **Select objects**:, simply click the pieces of arc annotation and press Enter when complete:



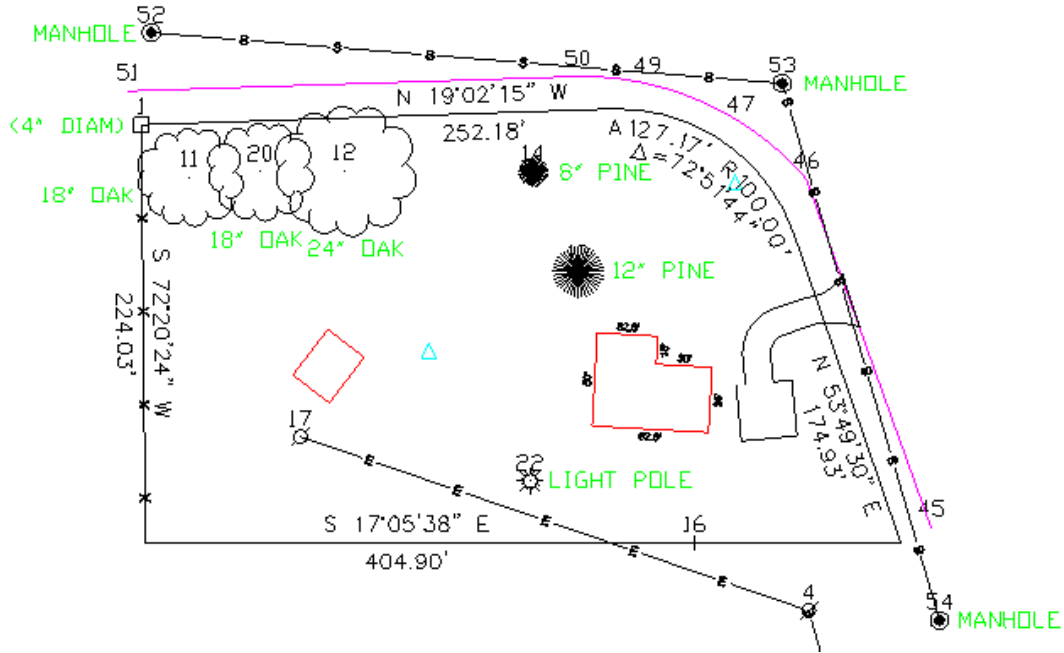
As needed, use the Annotate – Annotate Arc – Label Arc command to better control the placement of the arc annotation. When prompted, select the arc from the screen. The Label Arc Settings dialog box appears:



Experiment with:

- locate the Arc Length and Radius inside and closest to the arc and in the order cied, and,
- locating the Delta Angle inside the arc and a bit further from the arc, and,
- replacing the "D" letter with the triangular Greek "Delta" letter as is often commonly practiced.

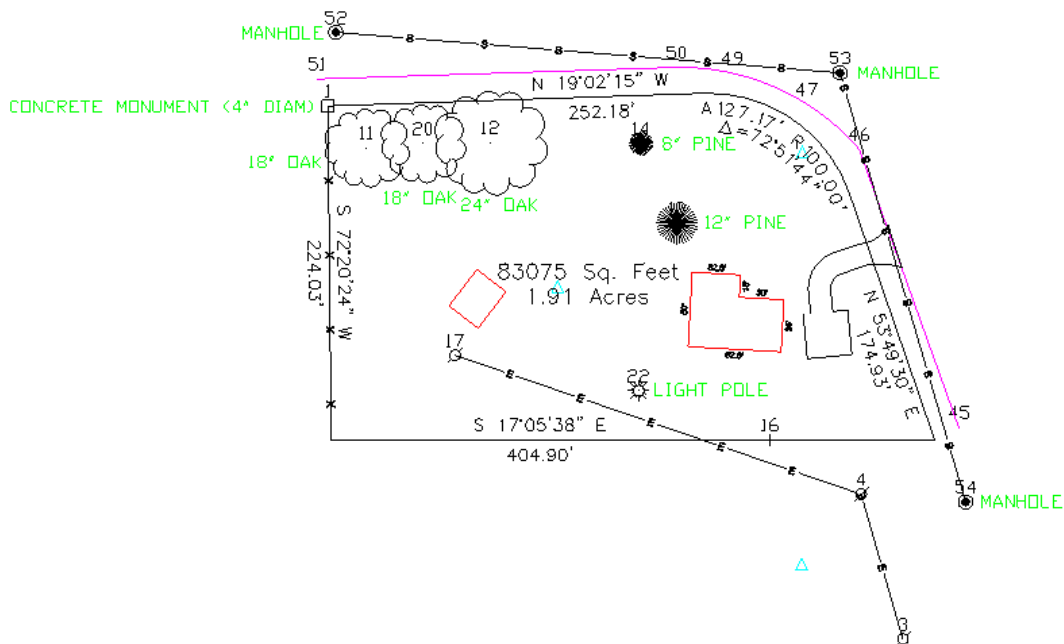
Click **OK** when ready. The new arc text might overwrite the **8" Pine** and if it does, use the previously discussed Points – **Move Point Attributes** command to relocate the **8" Pine** description. With the annotations placed in new positions, your drawing should be similar to the one shown below. Move your annotations to match this drawing:



Next, you will create an area label for the drawing. Select the Area/Layout – Area Defaults command. Select the **Sq. Feet** entry and click **Edit** to change to the nearest whole unit (no decimals) with a Text Size of 0.100 and Acres to 2 decimal places with a Text Size of 0.100. Click **OK** to dismiss the Acres settings dialog box and click **OK** to dismiss the Area Defaults dialog box.

Select the Area/Layout – Area by Lines & Arcs command. When prompted to **Select objects**, pick the three polylines that taken together, completely enclose the property.

Pick an area labeling centering point for the area label approximately centered within the parcel as shown below:



Before completing the final formatting of your drawing, you need to do some minor cleanup. Items we don't want

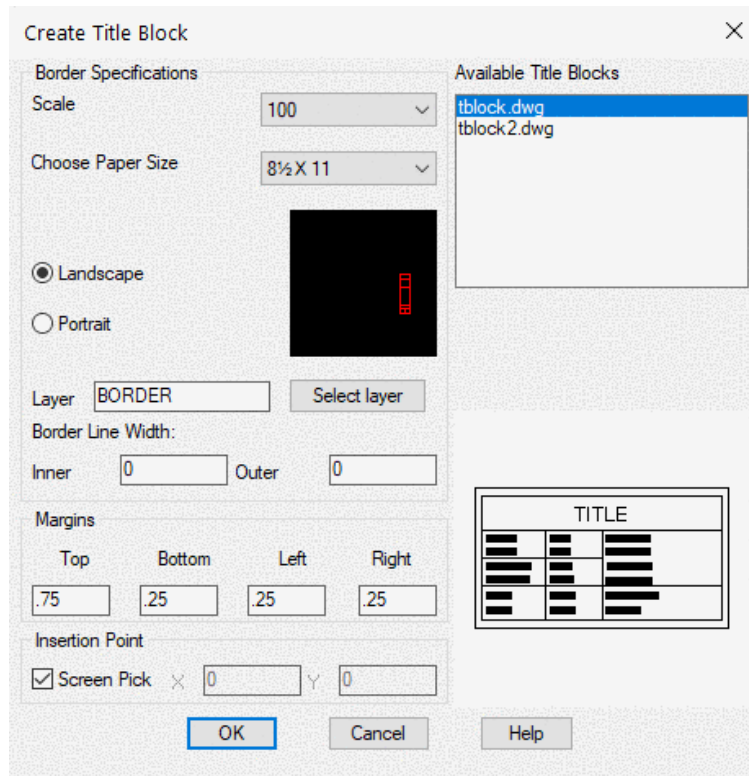
to see prior to plotting are:

- the symbol of Point 16 (the PL point)
- the "hub" symbols (triangles)

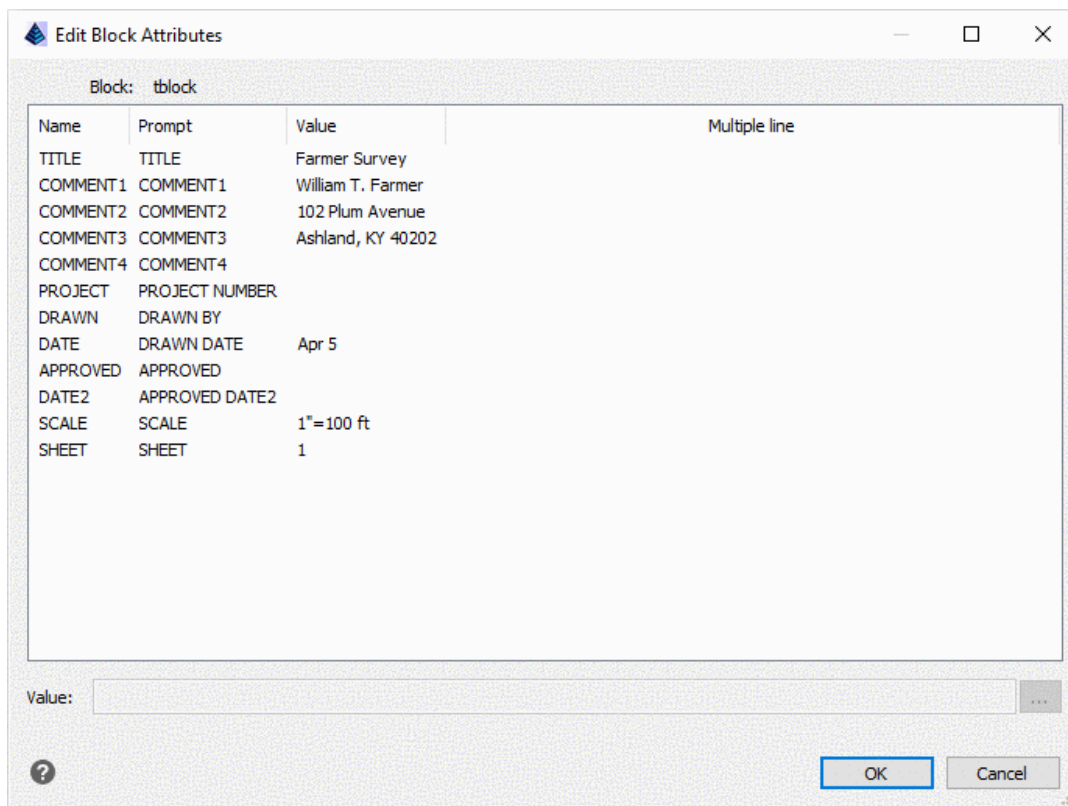
We'll freeze these items via the View – Freeze Layer by Pick command. Pick each of the items specified above and press Enter when done.

We'll complete the remainder of the cleanup process by manually freezing the PNTNO layer via the View – **Layer Control** command. Locate the PNTNO layer and turn the sun icon into a snowflake and close the Layer Control dialog box when complete.

To insert an A1 (8 1/2 x 11) border and title block with a landscape orientation (not portrait), select the Settings – Title Block – Standard command to display this dialog box.



Be sure the items above match your own. Click **OK**. For the insertion point, select a point at the very lower-left of the screen, so that your drawing plan entities fit inside the border and somewhat nearer to the top. Pick your screen location. You will then be prompted for the attributes of the title block. Fill them in and click **OK**.



If you prefer, you can use the Edit – **Move** (keyin **m**) command (pick the title block and two border perimeters) to a desired location. Refrain from moving the drawing because you will change the coordinates if you do.

Click the Annotate – Draw Legend command. Select the Existing tab and choose the **.lgd** file that you saved in Lesson 2 and click Open. Then select Draw and **OK** to close out the dialog boxes that follow, and then click Exit.

Pick an upper-left location point in the available space to the lower-left of the plat. If you did not save a legend in Lesson 2 (or you skipped Lesson 2), follow the steps in that lesson. As desired, you may use the Points – Resize Point Attributes command and scale up the oak tree symbol in the Legend by a factor of 1.5.

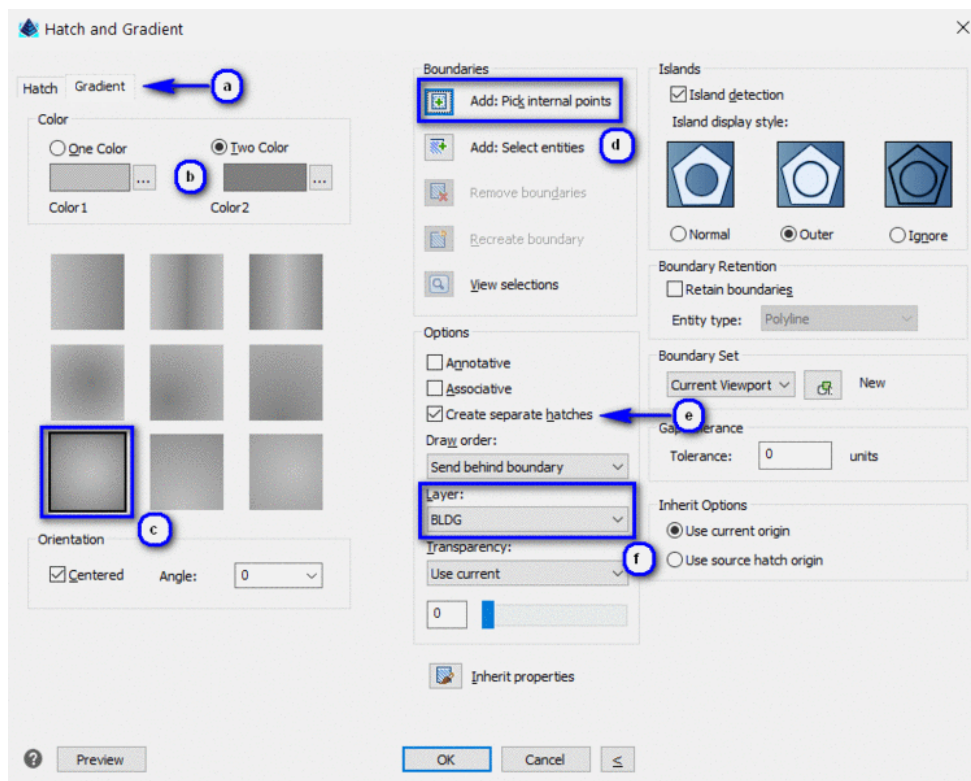
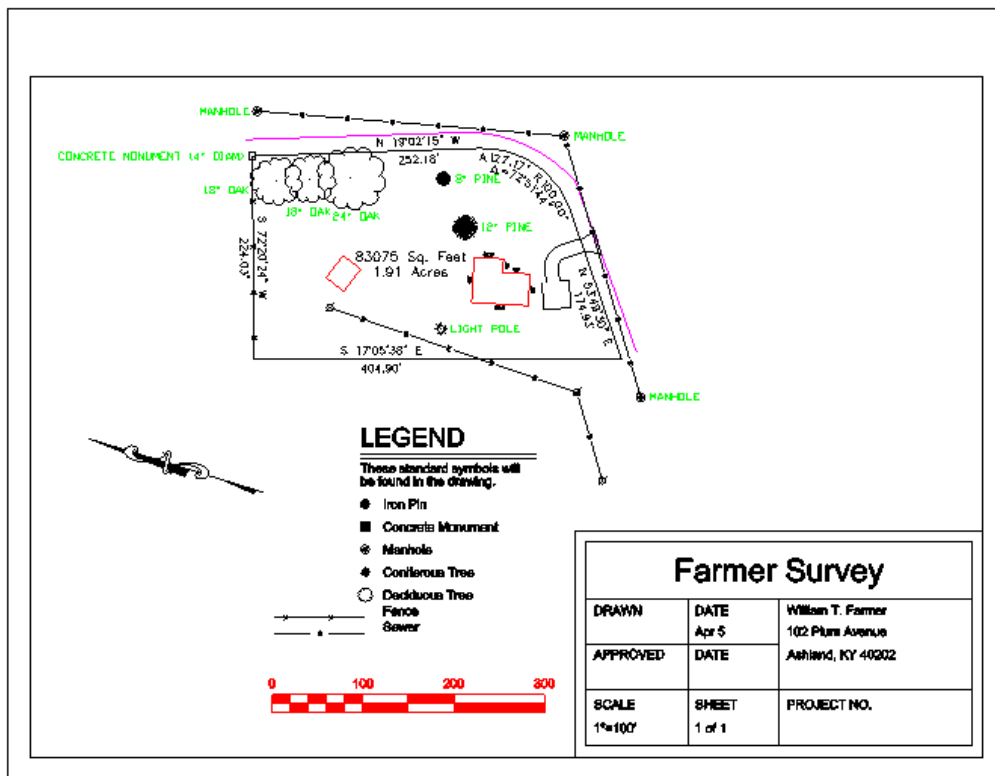
Use the Annotate – Survey Text – **Survey Text Defaults** command. Change the **Offset Dimension Text - Text Alignment** to Horizontal and click **OK**. Then, click the Annotate – Survey Text – Offset Dimensions command and pick the lower right corner of the building, and then the lowermost property line (in the current twist screen position). This labels the offset dimension horizontal to the current twist screen.

Click the Annotate – Draw North Arrow command and find and select the North Arrow symbol that is shown in the figure below. Change the Symbol Size Scaler, if necessary. Click **OK**. Then pick an appropriate location and press Enter. Note how the arrow draws due North, respecting the twist screen.

Click the Annotate – Bar Scale command and pick a location near the lower-left portion of the drawing.

Your drawing should now look similar to this:

Pick the Draw – Hatch – **Standard Hatch** command to display a dialog similar or as shown below:



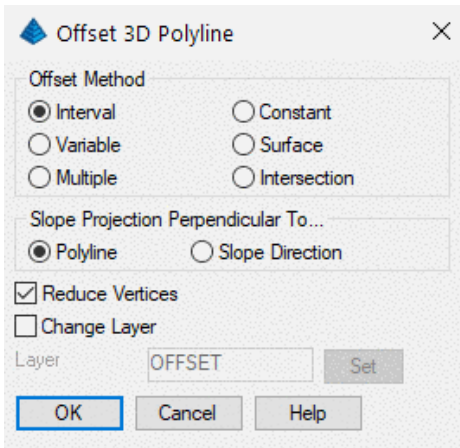
1. Select the Gradient tab
2. Use the "... " color-picker and choose Index color 9 for Color 1 and Index color 8 for Color 2
3. Click the "Radially centered" tile

4. Click the **Add: Pick internal points** and when prompted, pick an internal point in the house and the shed, and press Enter
5. Enable the **Create separate hatches** toggle
6. Set the Layer to be **BLDG** (the same as the house and shed)

Click **OK** when ready.

To offset the EOP polyline:

1. First try using the Edit – Offset – **Standard Offset** command and try offsetting the edge-of-pavement polyline that runs roughly parallel to the sewer line. You will see an error message because that object is a 3D Polyline, created by the **Draw Field to Finish** command.
2. To offset a 3D Polyline, you must use a command specifically designed to offset 3D Polylines. Use the Edit – 3D Polyline Utilities – Offset 3D Polyline command. Enter the offset method as Interval, press **OK**.



When prompted:

Vertical/<Horizontal offset amount>: 30

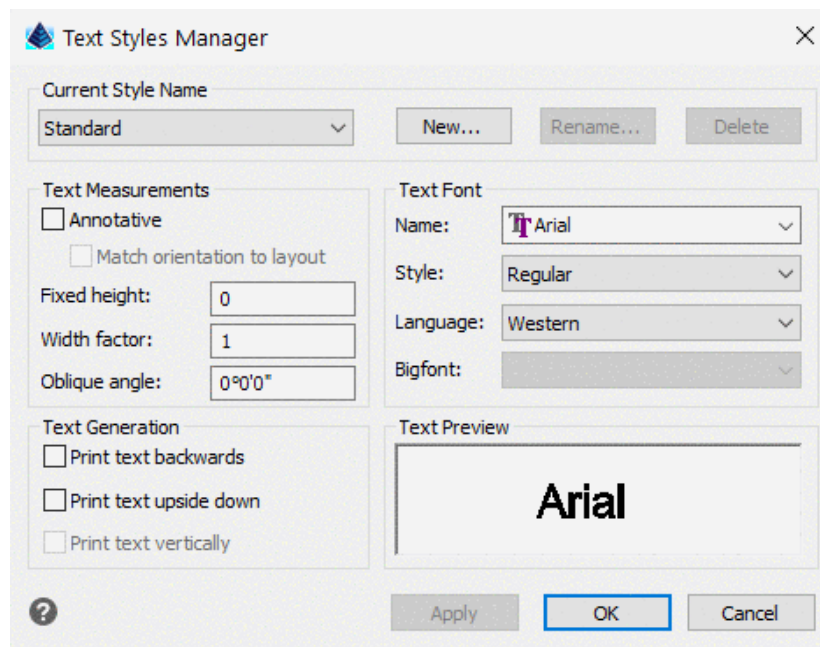
Percent/Degree/Ratio/Vertical offset amount <0>: press Enter

Select a polyline to offset: pick the EOP polyline

Select side to offset [Both]: pick out and away from parcel for the other side of the road

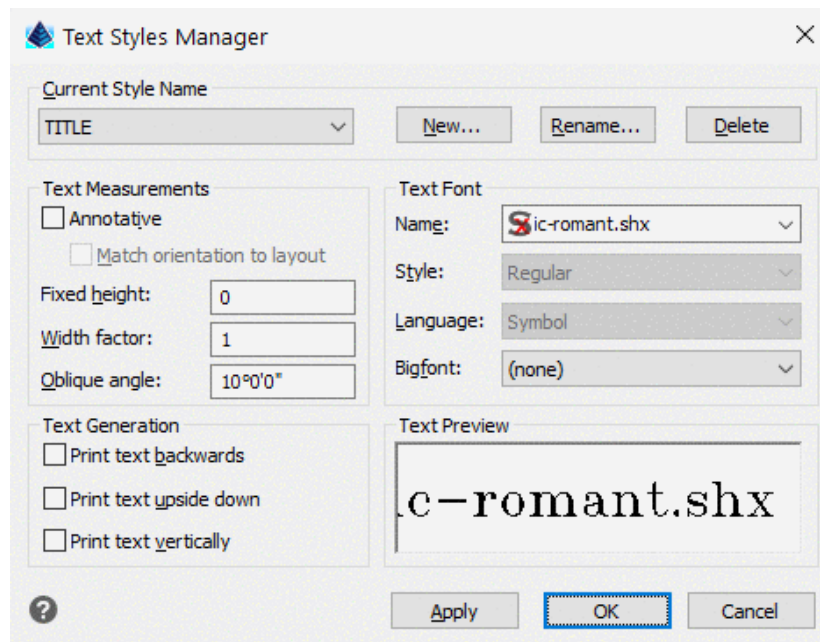
Select a polyline to offset (Enter to end): press Enter

Before you add a title to the drawing, create a text style for the title. Use the Draw – Text – **Set Style** command to display the dialog box below:



Click **New**, enter **TITLE** in the New Text Style dialog, and click **OK**.

Define the **TITLE** style consisting of the **ic-romant.shx** font (or **romant.shx** if using an AutoCAD-based platform) and an Oblique Angle of **10°194'** by entering the settings as show below:



Then click **OK**. Now, run the DText command by typing **dtext** at the command line, and place the text at the top of the drawing as follows:

Specify start point of text or [Justify/Style]: J

Enter an option [Left/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR]: c

Specify start point of text or [Justify/Style]: pick a point near the top-right of the drawing

Height of text <8.00>: 20

Rotation angle of text <E>: pick a point to right of first point with <Ortho on> (F8), dynamically stretch right

Text: Farmer Survey (press Enter)

Text: April 5, 2021 (press Enter) **Text:** press Enter

Use the Edit – Text – Text Enlarge/Reduce command. When prompted for Scaling Multiplier, enter 0.8 and pick the date you just entered.

Verify your Horizontal plot scale through the Settings – Drawing Setup command. Your drawing should have a Horizontal Scale of 100 with a Text Plot Size of 0.08. Change the Text Plot Size to 0.06 to shrink the building labels we are about to place and click **OK**.

Click Draw – Leader – Leader with Text command and follow the prompts below:

Options/Size/Pick Arrow Location: nea (we want to use a NEArrest snap)

Snap to NEA of: pick near or on the left side of the house

To point: pick off to the left

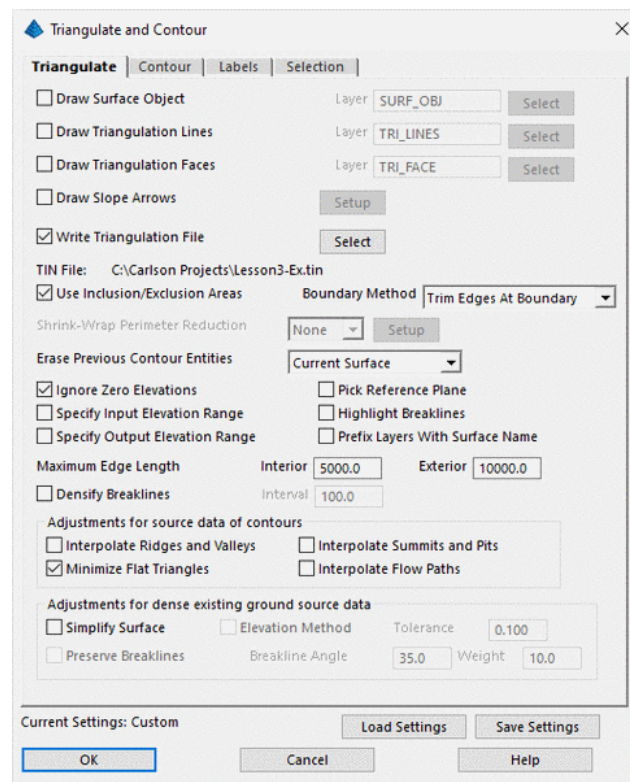
Next point (Enter to end): press Enter

Text: 2-Story

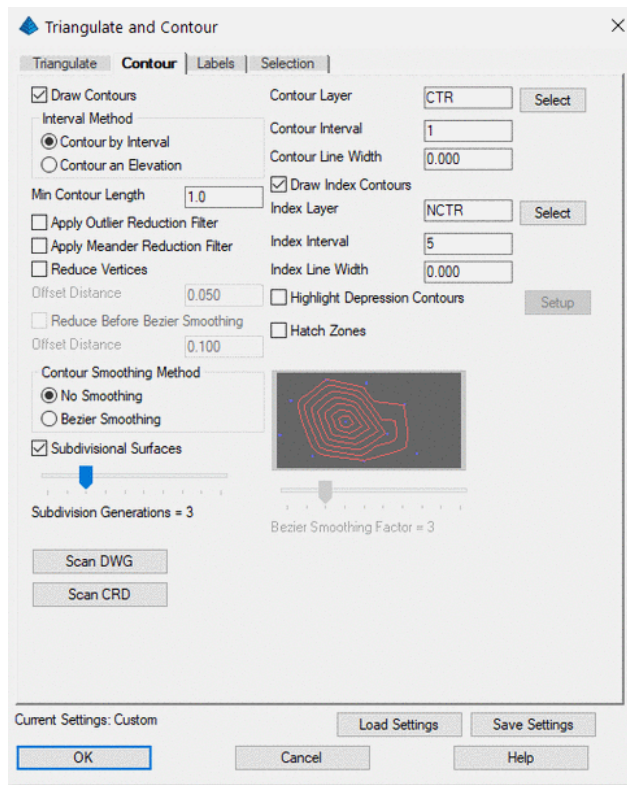
Text: Farm House

Text: press Enter

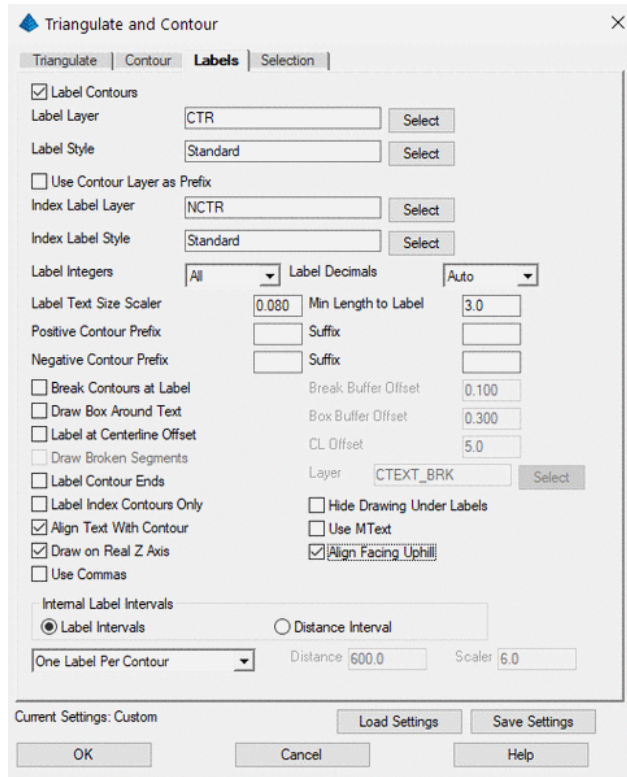
Initiate the Surface – Triangulate & Contour command. On the Triangulate tab, set the following values:



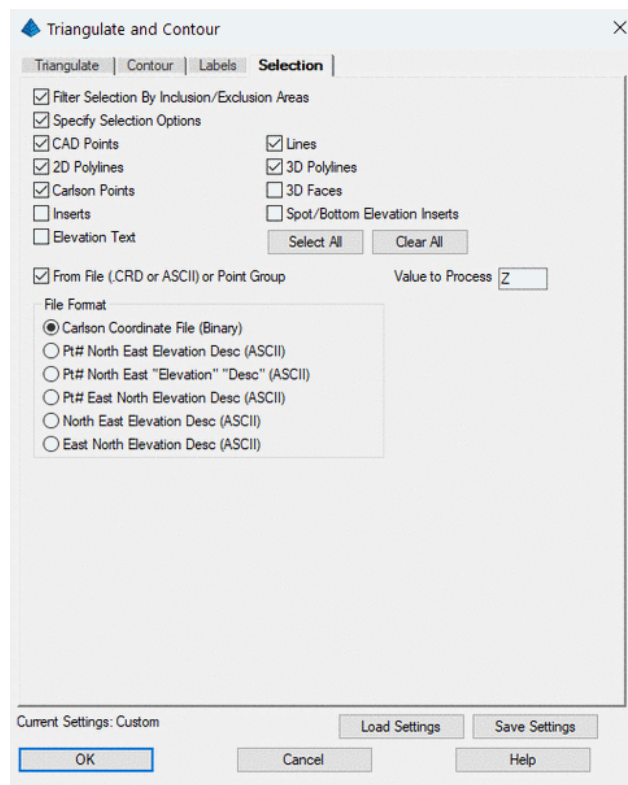
Also make sure to enable the **Use Inclusion/Exclusion Areas** option. Navigate to the **Contour** tab and set the values as shown below:



Click on the Labels tab and match the following dialog:



Click on the Selection tab and fill out to match the following:



Press **OK** and then answer as follows:

Select Inclusion perimeter polylines or ENTER for none.

[FILTER]/<Select entities>: press Enter for none.

Select Exclusion perimeter polylines or ENTER for none.

[FILTER]/<Select entities>: select the outlines of the building and the shed, and then press Enter

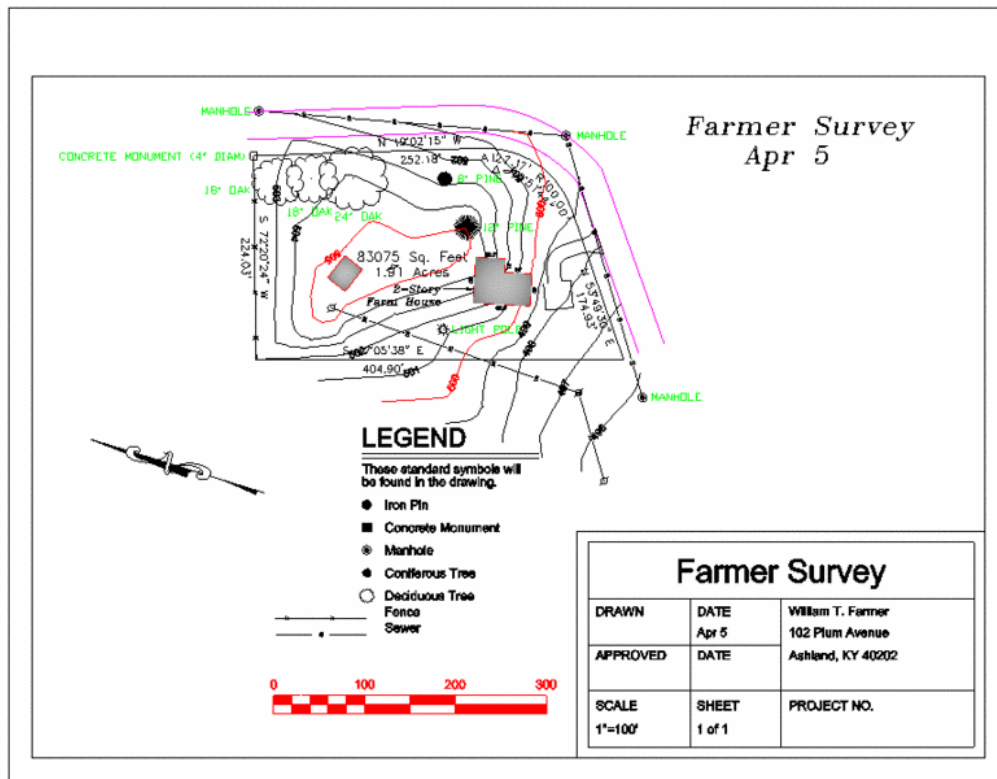
Select points and breaklines to Triangulate.

[FILTER]/<Select entities>: press Enter (we're going to fetch the coordinates out of a specified coordinate file)

Select Coordinate File to Read (dialog): plat3.crd (and click Open)

Points to Triangulate (dialog): click **OK**

The contour map is created. The final drawing will look similar to this:



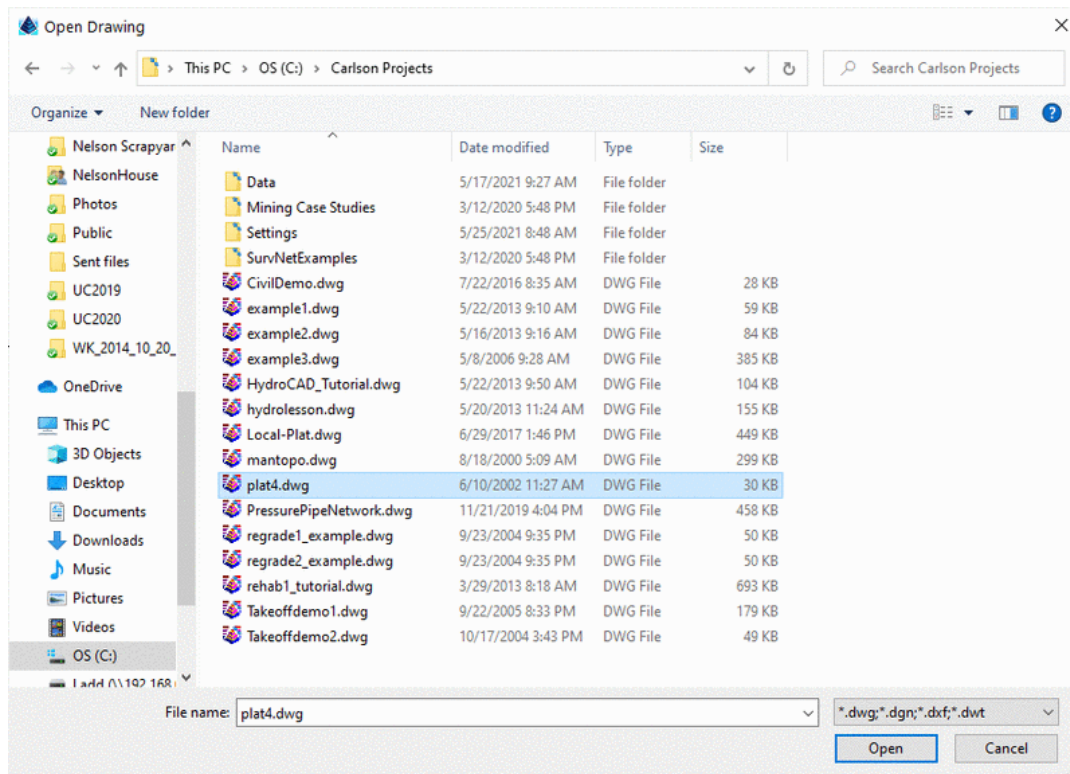
This completes the Lesson 3 tutorial: Field to Finish for Faster Drafting.

Intersections and Subdivisions

In this lesson you will create a basic subdivision including a street intersection. You will make the subdivision from a drawing file named **Plat4.dwg**.

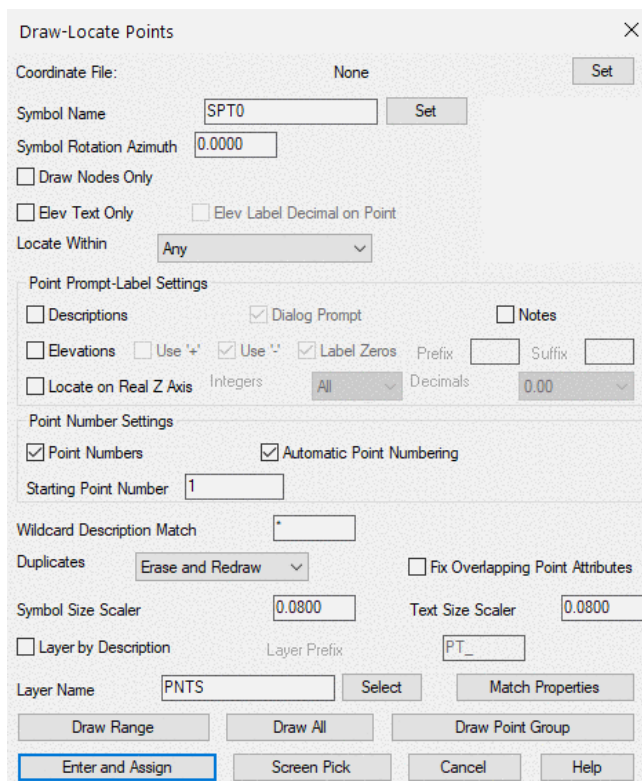
1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **Plat4.dwg** file.



Verify the Survey module is loaded by clicking Settings – Carlson Menus – Survey Menu.

2. We'd like to establish a starting coordinate for a street centerline to follow. Select the Points – Draw-Locate Points command and obtain the dialog shown below:



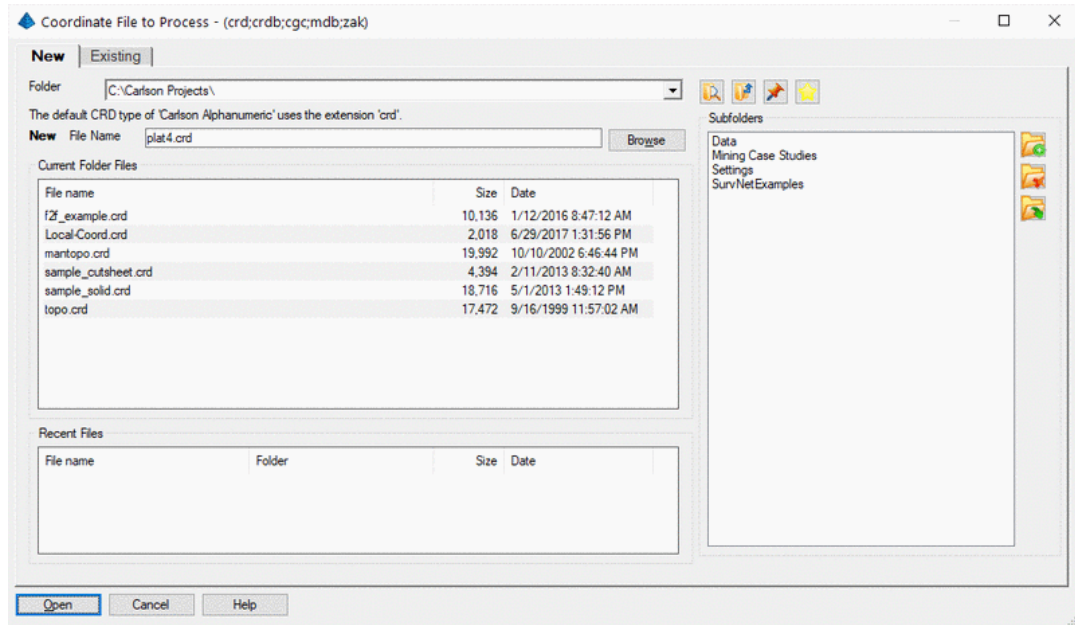
At the top, change the symbol to SPT0 by picking Select at the very top of the dialog, and choosing symbol SPT0 from the dialog of symbol choices (not shown here). Disable the prompting and labeling for **Descriptions, Elevations and Locate on Real Z Axis** (make them blank as shown). Match the other entries

as shown (which are mostly the default conditions) and click **Enter and Assign** at the lower left and respond to the prompts as follows:

Enter North(y): 4809.17

East(x): 4391.28

The program will recognize that you have not yet started a coordinate file, so click the **New** tab and enter the File Name as **Plat4.crd** (which should be the default). If you enter Plat4, you do not need to enter the extension .crd. The program will add extensions automatically. You will see this:



Click Open. You will be prompted again:

Enter North(y): press Enter

3. We'd like to start the creation of a roadway centerline. Select the COGO – Traverse command (or alternately just enter **T** at the command line. **T** is one of several "hotkeys." Other hotkeys are **I** for Inverse and **SS** for Sideshot). We're going to traverse from PI to PI (to the two endpoints of our centerline) so reply to the prompts as follows:

Traverse, Line OFF, RAW FILE OFF

Occupied Point ?

Pick point or point number: 1

Exit/Help/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <7>: 1

Enter Bearing Angle (dd.mmss) <90.0000>: 58.1848

Points/<Distance>: 736.73

Exit/Help/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <1>: e (to exit)

You could keep on traversing, but we will stop here to review. You have created point 2, traversing NE from point 1. To review, code:

- 1 is for NE
- 2 for SE
- 3 for SW
- 4 for NW

- 5 for Azimuth
- 6 for Angle Left
- 7 for Angle Right
- 8 for Deflection Left
- 9 for Deflection Right

This is the standard way that traverses and sideshots are entered in Carlson with a code entry (followed by Enter), then the angle or bearing entry (followed by Enter). Lesson 1: Entering a Deed, presented another method, where the angle and bearing are together in the form of *158.1848*. That is a rare form, designed to save keystrokes, and used primarily only in Enter Deed Description. Now you have been exposed to both!

- Let's explore the including linework with the traverse information. Click COGO – Line On/Off to turn on simultaneous linework with traversing. This command toggles on and off each time you click it, with the **On** status indicated by a check mark. Now repeat the **Traverse** command (try *T* at the command line and follow the prompts below:

Traverse, Line ON, RAW FILE OFF

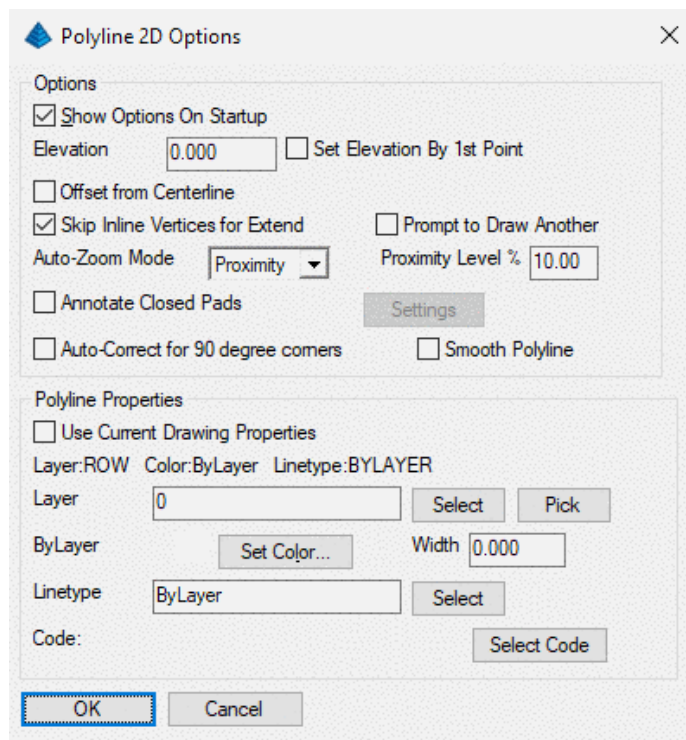
Exit/Help/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <1>: 2

Enter Bearing Angle (dd.mmss) <90.0000>: 75.0627

Points/<Distance>: 553.69

Exit/Help/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <2>: e (to exit)

- We'd like to draw a polyline from Point 1 to Point 2 and then connect the segments together (we could have turned linework with traverse on before we got started, but now we will do it after-the-fact for illustrative purposes). Pick the Draw – 2D Polyline command (or simply type-in **2DP** at the command line) which starts the Carlson enhanced Polyline command. If the Polyline 2D Options dialog box appears, set the values shown below and click the **OK** button:



[Continue/Extend/Follow/Options/<Pick point or point numbers>]: 1-2

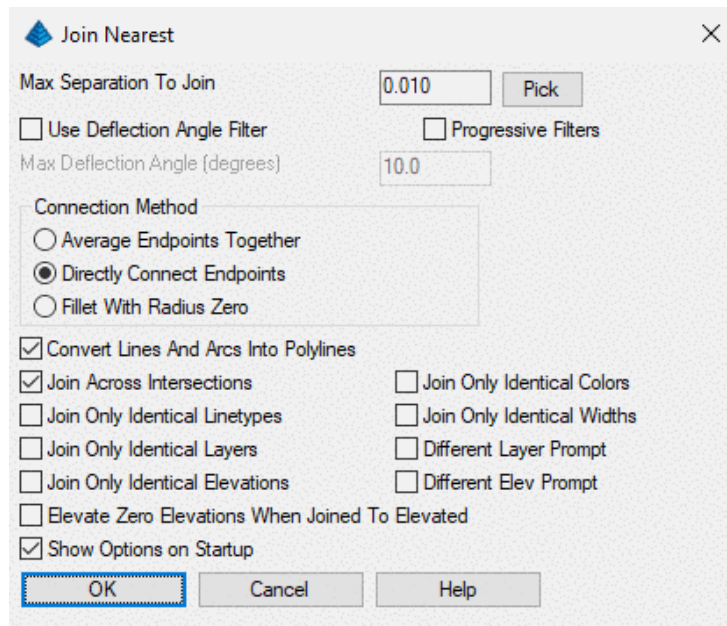
[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: press Enter to end

Now we have two vector entities. The first, from point 2 to point 3 is a **Line**. The second, from point

1 to point 2, is a **Polyline** (even though it is only one segment long). It will (by default) be a lightweight polyline (LWPOLYLINE). This can be verified by picking it using the Inquiry – **List**. Polylines are linked combinations of one or more line/arc segments that behave as one unit. Carlson encourages the use of polylines versus lines because they offset as a unit, will take on a thickness or width, are easier to select and have superior editing capabilities.

NOTE: Many entities (*e.g.* lines, arcs, *etc*) can be turned into a polyline by issuing the Edit – Polyline Utilities – Entities to Polyline command but our next command will also offer this functionality.

To join the polyline and line entities into a single polyline, pick the Edit – Join Nearest and set the prompts to those shown below and click **OK** when ready:



Select lines, arcs and unclosed polylines to join [Settings].

Select entities: pick the polyline from 1 to 2 and the line from 2 to 3, and then press Enter for no more

Now, see the grips on the new polyline by picking it with the cursor. See how the whole thing highlights? That is proof that it is joined up as a polyline.

- Let's design an arc with a 500' Radius by using the Draw – Arc – 2 Tangents,Radius command and following the prompts below:

Radius of Arc <0.000>: 500

[nea] Pick Point on 1st Tangent Line: pick on the 1st polyline segment closer to point 2

[nea] Pick Point on 2nd Tangent Line: pick on the 2nd polyline segment close to point 2

The arc draws in, and the centerline remains a polyline, now with 3 segments.

- The **Inverse** command is a handy way to reset a new "occupied point" to get on a point to begin another traverse so let's practice inversing. Key-in **I** at the Command prompt and inverse from point 1, then to point 2, then to point 3 then back to 1. Notice the results being echoed to the Command prompt window (pressing the **F2** function button on most keyboards toggles the visibility of the text window results).

You can also inverse (go to) a snapped position on a line or polyline, such as the midpoint of an arc. Let's do that, because we want to traverse south from the *midpoint* of the arc. Key-in **I** and follow the prompts below:

Calculate Bearing & Distance from starting point?

Traverse/SideShot/Options/Arc/Pick point or point number: mid (for midpoint snap of)

Snap to MID of: select the arc

Traverse/SideShot/Options/Arc/Multiple/Pick point or point number: T

Traverse, Line ON, RAW FILE OFF

Exit/Help/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <2>: press Enter

Enter Bearing Angle (dd.mmss) <75.0627>: 10.11

Points/<Distance>: 400

Exit/Help/Options/Arc/Points/Line/SideShot/Inverse/Angle-Bearing Code <2>: press E to exit

Notice that you can transition from Inverse, to Traverse, to Sideshot, *etc.*, with these COGO options. We were in inverse, but we did T for traverse, and could have done I for inverse to return to inverse. This cuts down on keystrokes, and adds to the sense of fluidity of the software.

Turn the resulting Line into a Polyline with the aforementioned Edit – Polyline Utilities – **Entities to Polyline** command as the following command requires Polyline entities.

8. We'd now like to create edge of pavement lines and form the intersection. Pick the Area/Layout – Layout Utilities – Offsets & Intersections command and follow the prompts below:

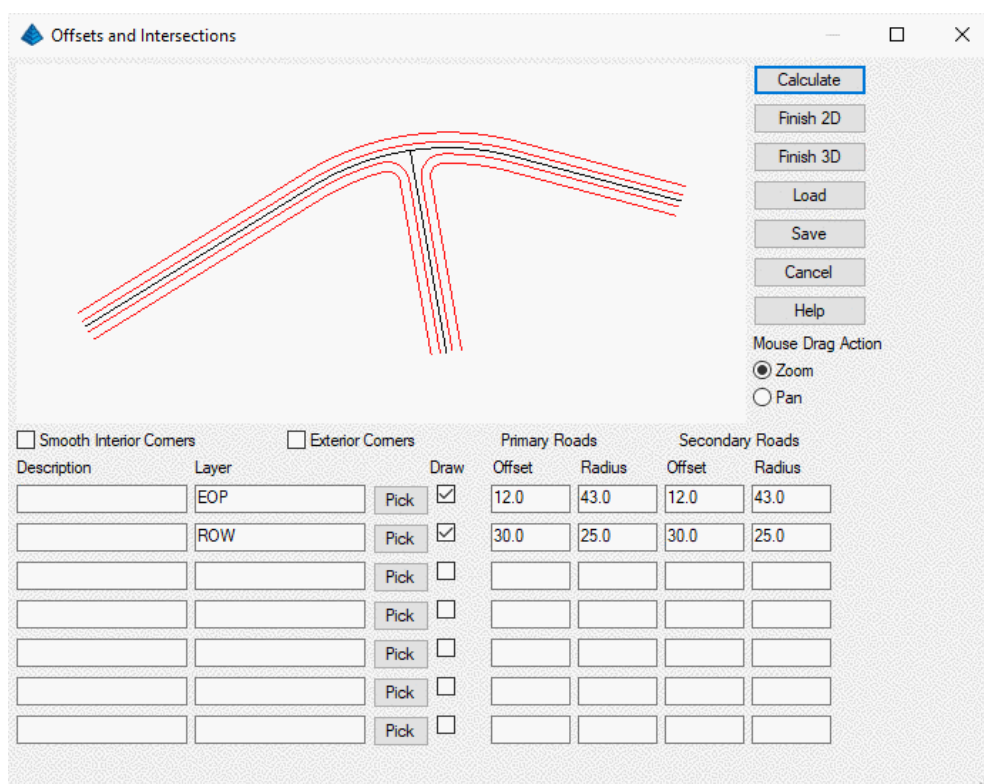
Select all PRIMARY road polylines.

[FILter]/<Select entities>: press Enter (we will consider both these subdivision streets secondary)

Select all SECONDARY road polylines.

[FILter]/<Select entities>: pick the main centerline and pick the side road

[FILter]/<Select entities>: press Enter



The street intersections are presented in a dynamic dialog as shown above. Try experimenting with different radii under the **Secondary Roads** column, then clicking Calculate. The streets will re-draw in the upper graphical area. But after experimenting, change the four values under Secondary Roads to those shown (ignore Primary Roads – those don't apply here), and click **Calculate**. Then click **Finish 2D**. Note the drawn-out street intersection.

Now select the Inquiry – Drawing Inspector command. Right click and make sure **Layer** is checked. Hover over the outside polyline (it is layer ROW). Hover over the next polyline inwards from the outer polyline (it is layer EOP). For example, if you had clicked off EOP under the **Draw** column in the above dialog, the edge-of-pavement polyline would not have drawn.

9. We'd now like to add some cul-de-sacs to the project. Click the Area/Layout – Layout Utilities – Cul-de-Sacs command. You may want to zoom into the area of the bottom center, near point 4. When finished with the procedure below, zoom back out:

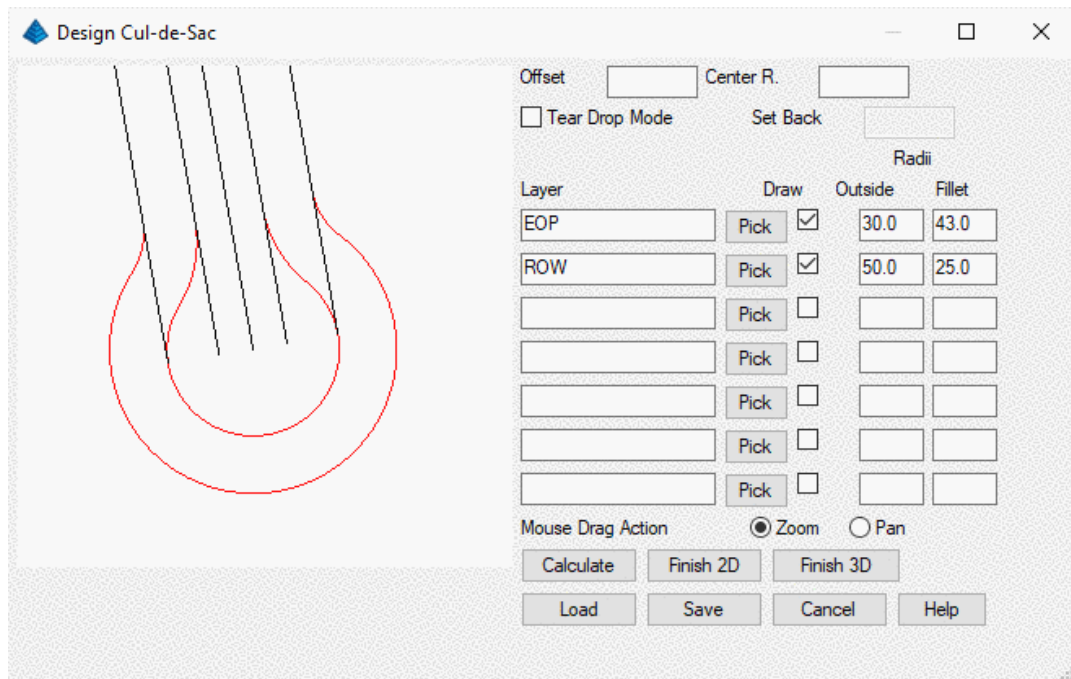
Select all offset polylines to end with cul-de-sac.

[FILter]/<Select entities>: form a crossing selection from right to left across the lower side road, selecting all 5 polylines (ROW-L, EOP-L, CL, EOP-R, ROW-R)

[FILter]/<Select entities>: press Enter

Pick cul-de-sac center projection onto centerline: pick near the endpoint of the centerline of the lower side road near point 4. Make sure the pick is on the centerline polyline, or the routine will say the centerline was not found.

This brings up the following dialog:



Again, you can change the **Fillet Radius** and the **Outside Radius** on the EOP or ROW, click **Calculate**, and check out its effect (don't make the Outside radii too small or it will fail to Calculate if there is no workable solution). Set values as shown above and then click on **Finish 2D**.

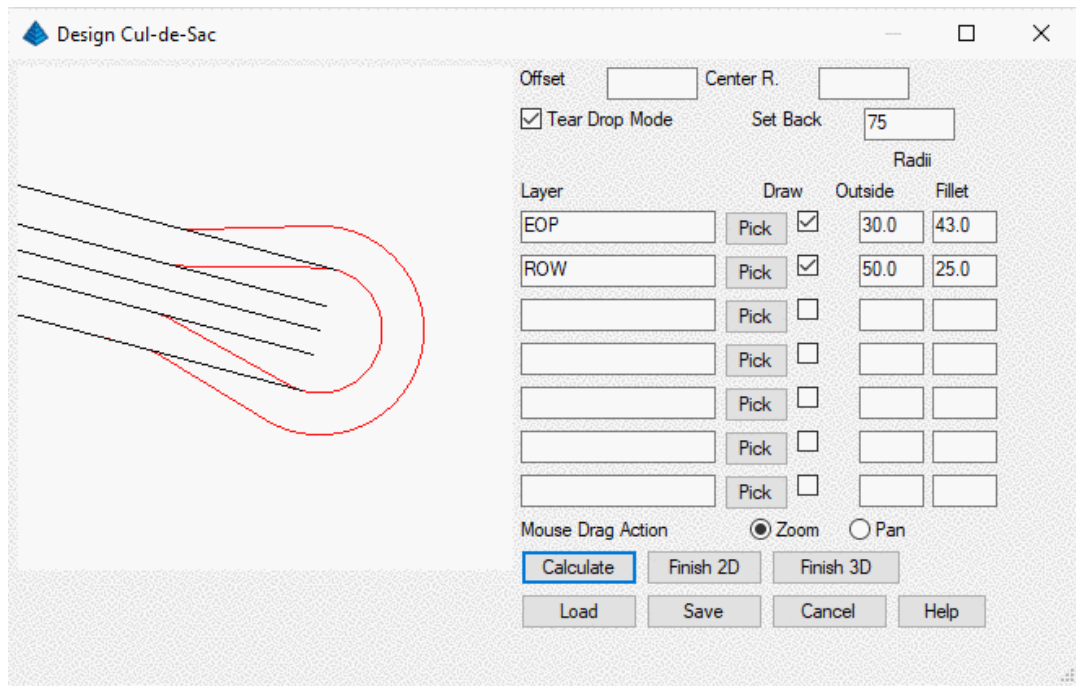
10. Repeat the Area/Layout – Layout Utilities – **Cul-de-Sacs** command to place a tear-dropped shape Cul-de-Sac:

Select all offset polylines to end with cul-de-sac.

[FILter]/<Select entities>: form a crossing selection pick from right to left across the right main road, selecting all 5 polylines (ROW-L, EOP-L, CL, EOP-R, ROW-R)

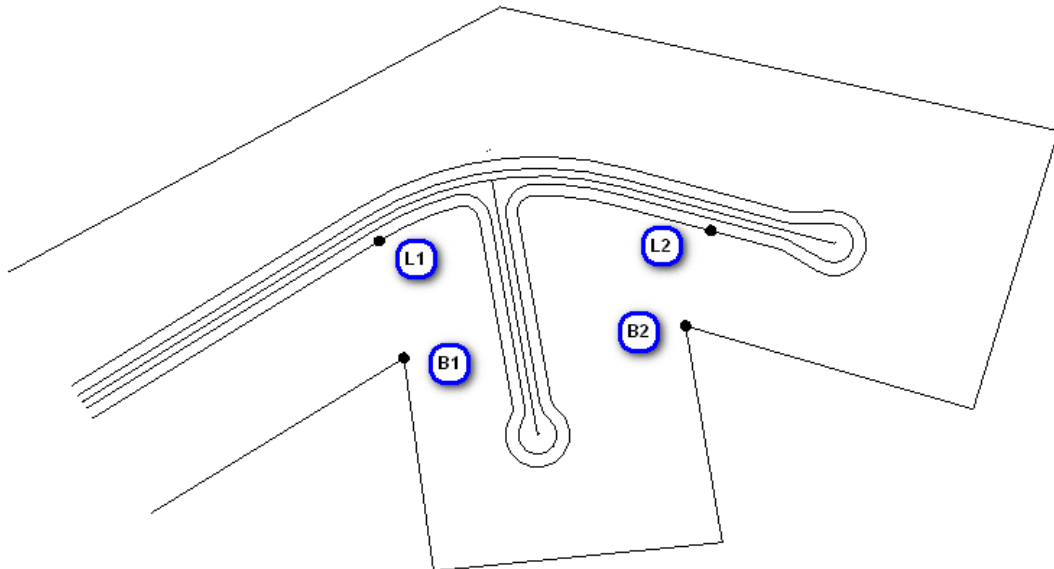
[FILter]/<Select entities>: press Enter

Pick cul-de-sac center projection onto centerline: pick the endpoint of the centerline of the lower side road near point 3

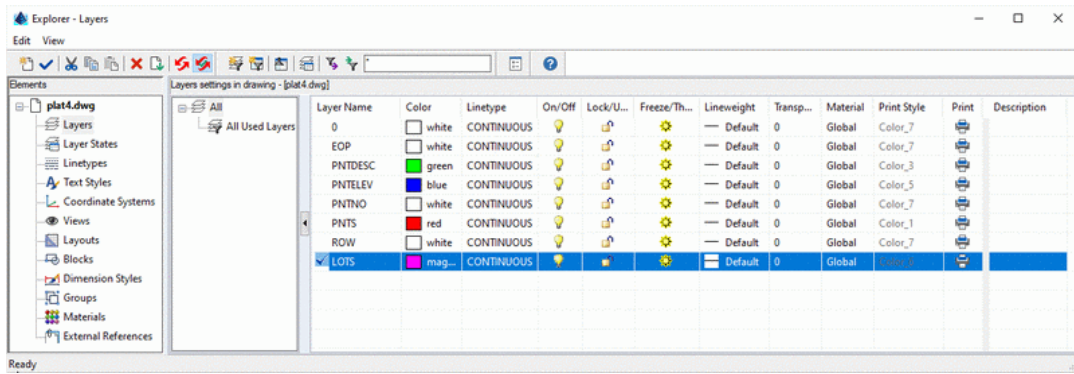


For a teardrop cul-de-sac, fill out the dialog as shown above then click on **Calculate** and **Finish 2D**.

NOTE: Teardrop cul-de-sacs allow moving vans and other large vehicles more turning room, and have been popular in many residential subdivision areas. Our drawing now appears as shown below, with the exception of the filled reference dots:



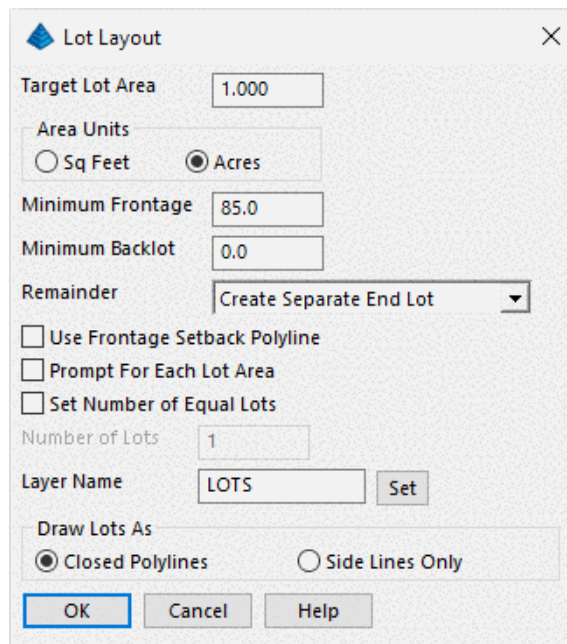
11. It's a good idea to create a layer and set it current before beginning the design process. Let's make a layer called LOTS using the View – **Layer Control** command and set the values as shown below:



- (a) Click on for New layer
- (b) When *Layer1* highlights, type over it with LOTS
- (c) Click under the Color column and change the color to Magenta
- (d) Click the (Set) Current button up top to make this layer current and then dismiss the dialog box

Our goal is to make 1-acre lots. The automated parcel creation tool prefers frontage lines that do not contain lots of zigs, zags, and jogs in the polylines as this may cause the perpendicular offset logic to fail to find a solution (Lot sidelines will radiate perpendicular from the frontage polyline). Not only should the front and back lines tend to run parallel with each other, but they should end at some point before the calculation runs into difficulty with impossible math.

12. Pick the Area/Layout – Layout Utilities – Lot Layout command. A dialog appears:



Fill out as shown. In particular, change the **Remainder** to *Create Separate End Lot* so that we force 1.000 acre lots and don't just get equal lots of some size such as 1.0017 (because the remainder lot that would not fit was added onto all lots). Using the **Closed Polylines** option means that our side lines will be doubled up, each lot sharing a side line. Click **OK**.

Select front polyline: pick north ROW

[Offset/<Select back polyline>]: Pick northernmost polyline (the back property line)

The 1.00 acres lots are laid out as far as possible. However, due to the geometry, it is likely that more Lots are able to be generated. As it turns out, the direction of the **front polyline** can have an impact on what is able to be created.

13. Let's reverse the direction of the **front polyline** to see what we get. Click the Edit – Polyline Utilities – Reverse Polyline command and when prompted:

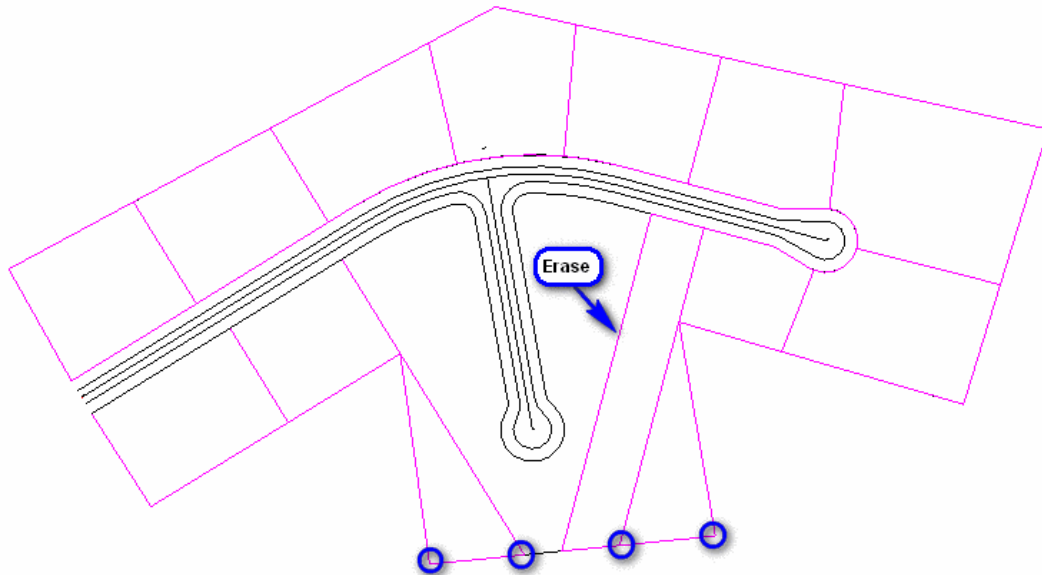
Select polyline or line to reverse: pick the ROW polyline and shows phantom direction lines of the new current direction (which are automatically removed when the command ends)

Re-run the Area/Layout – Layout Utilities – **Lot Layout** command. Use same dialog entries and when prompted:

Select front polyline: pick the ROW polyline in a convenient location

[Offset/<Select back polyline>]: pick the outer polyline at a convenient location

The result should look similar to that below, with the exception of the open reference dots:



Let's clean this up for new work to follow:

- Click the Edit – Erase – **Standard Erase** command and erase the Lot as shown above.
- Let's remove polyline **vertex** locations which permits to keep parts of the Lots intact. Click Edit – Polyline Utilities – Remove Polyline – Remove Polyline Vertex and when prompted:

Select polyline to remove from: pick one of the LOTS polyline (cue markers will display where vertex locations can be found)

Pick point to remove: pick one of the circled vertex locations shown above (the polyline changes shape)

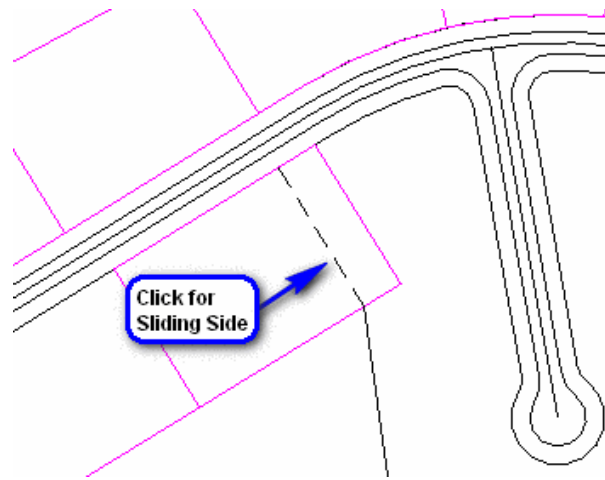
Pick point to remove (Enter to end): pick the other indicated vertex location

Pick point to remove (Enter to end): press Enter

- Re-run the **Remove Polyline Vertex** command by pressing Enter and repeat the process for the other Lot indicated

The Lots are now more regularly shaped but less than the 1 acre target area we desire.

14. Let's adjust the two smaller lots to the exact 1 acre target size we desire through the use of the Area/Layout – Adjust Areas – Sliding Side Area command as illustrated below:

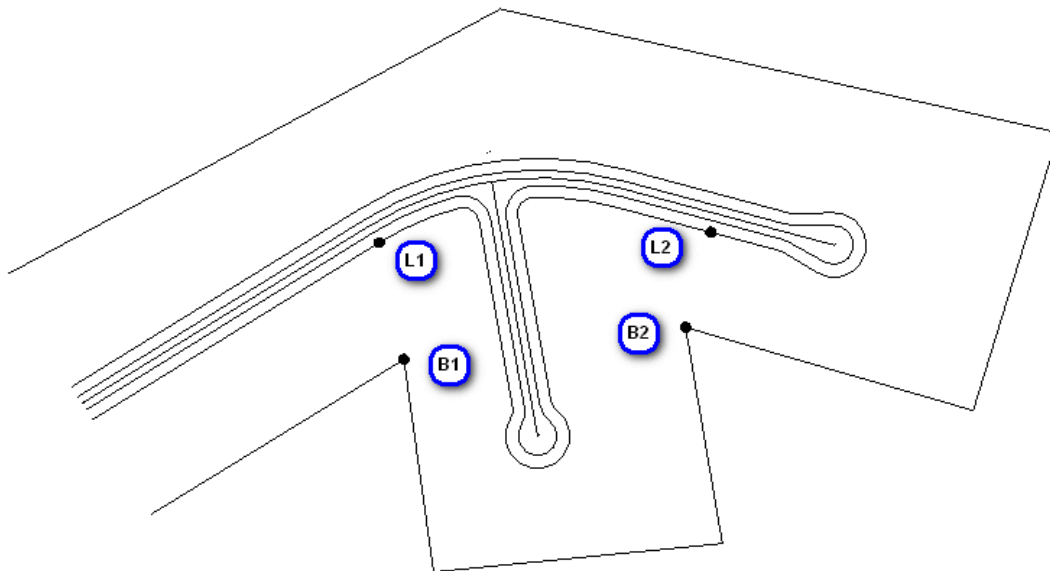


When prompted:

Define area by points or closed polyline [Points/<Linework>]? press Enter to accept the Linework option
Select polyline segment to adjust: pick the easterly-line of the Western lot as indicated by the dashed line
Keep existing polyline [Yes/<No>]? indicate the No response so the end result is a single lot polyline
Define new line by selected line, another line, angle or points [<Selected>/Line/Angle/Points]? press Enter to indicate the new lotline should be simply slid
(the current area is echoed for reference purposes)
Remainder/Yards/SF/<Enter target area (acres)>: type 1 to indicate the desired acreage size
Calculating ...
Done.

The new lot at the target size is drawn. Press the **Enter** key to repeat the command to adjust the westerly-line of the Eastern lot.

- The lower back property line is still continuous. We can work with it in small pieces rather than as one big polyline. Say we want to break it at the inside corner identified by the points **B1** and **B2** as cited in a previous image (reproduced below):



To do this, use the Edit – Break – At Intersection command and when prompted:

Select Line, Arc, or Polyline to Break: pick the lower back property line

[app on] Pick Intersection to break at: pick the **B1** intersection point

Press the Enter button to repeat the command to break the lower back property line at **B2**.

16. Our next task will be to draw a Polyline from the lower back property line to the Right-of-Way polyline. Use the Draw – **2D Polyline** command. If the Polyline 2D Options dialog box appears, click the **OK** button:

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: use a NEArest snap to start the polyline just south of **B1**

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: use a PERpendicular snap to end the polyline at the ROW polyline

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: press Enter to end

Press Enter to re-run the command and repeat the process starting just south of **B2**.

17. We have just created two new lots, but the lots are not defined by one, single, closed polyline and are likely not the target 1 acre lot size. To verify the areas, select the Area/Layout – Area by Interior Point command. When prompted:

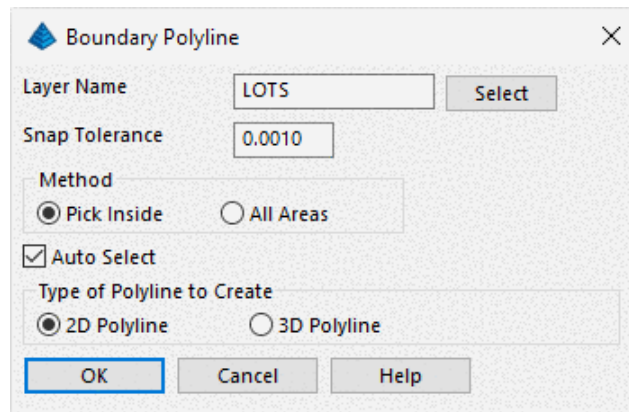
Pick point inside area perimeter: pick inside either (or both) of the new lots (the results are echoed to the Command line area)

Pick area label centering point (Enter for none): press Enter to end

Pick point inside area perimeter: press Enter to end

The lot(s) is/are probably not the 1 acre target. We wish to obtain the target size but would prefer to work with a closed polyline perimeter for the new lots.

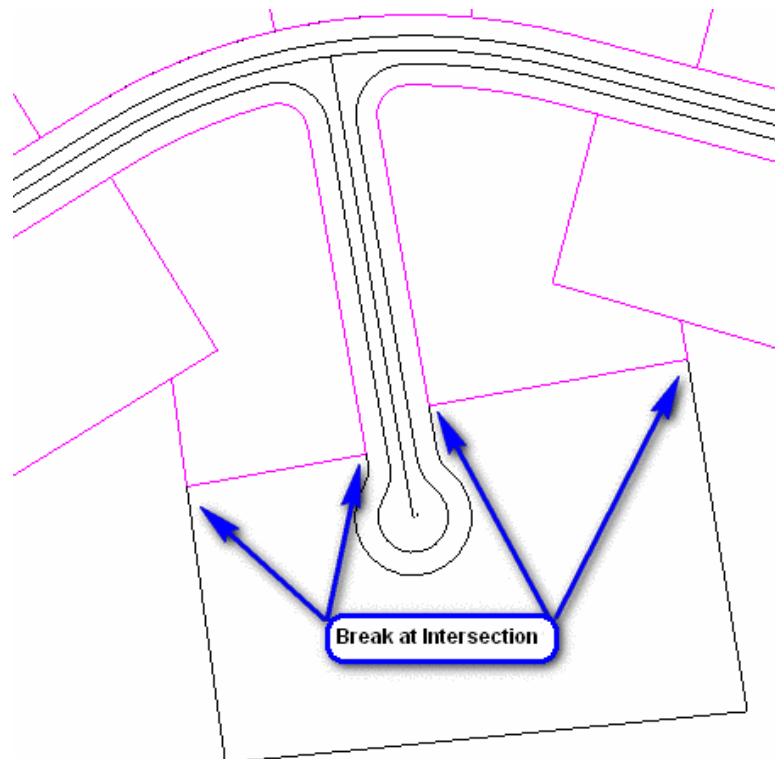
18. We'll use the Draw – Boundary Polyline command that will trace out the bounded perimeter (if available) for a point we pick. Set the values as shown below and click **OK** when ready:



When prompted:

Pick an internal point: pick inside the new lot(s) (the new boundary polyline(s) are created).

19. We'll continue to create the lots at the target size. Re-issue the Area/Layout – Adjust Areas – **Sliding Side Area** command and adjust the southern property line of the newly created lots. When completed, our layout should resemble that shown below:



20. Let's create the remaining lots. Re-issue the Edit – Break – **At Intersection** and when prompted:

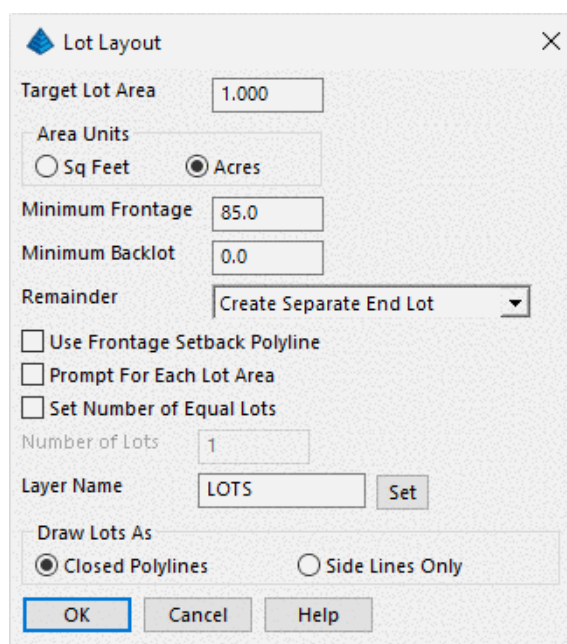
Select Line, Arc, or Polyline to Break: pick the lower back property line

[app on] Pick Intersection to break at: pick a lower back property line break location cited in the image above

Press the Enter button to repeat the command to break the lower back property line at its other designated break point.

Press the Enter button in succession to repeat the command to break the right-of-way property line at its designated break points.

21. Re-issue the Area/Layout – Layout Utilities – **Lot Layout** command. A dialog appears:

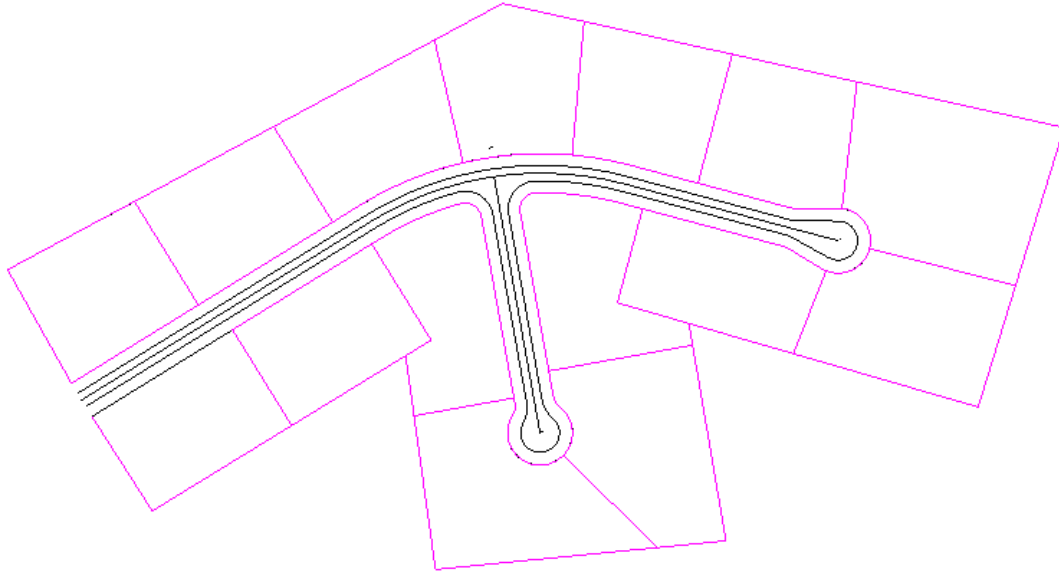


Fill out as shown, *except* change the **Remainder** to *Apply Equally to All Lots* so that we force a minimum of 1.000 acre lots and spread the remainder equally between the lots. Click **OK** and when prompted:

Select front polyline: pick the remaining ROW polyline

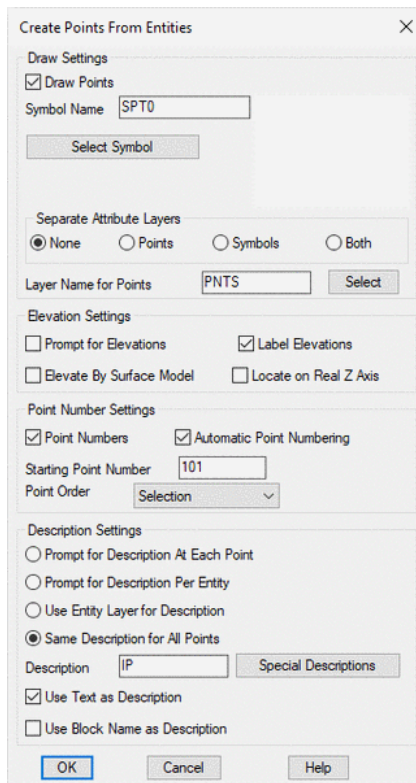
[Offset/<Select back polyline>]: pick the remaining lower back property line

The lots are laid out as evenly as possible and our completed site should resemble the following:

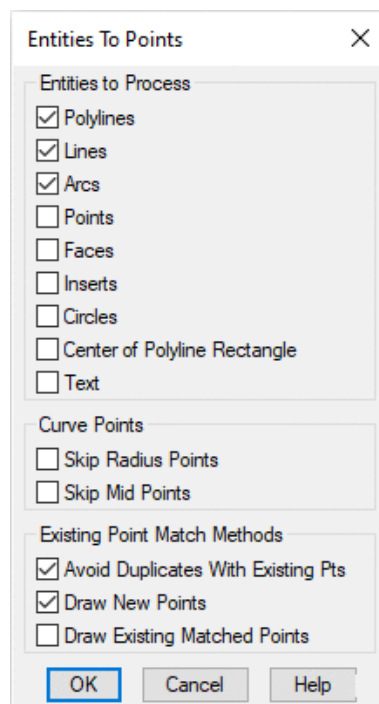


It may not be the most aesthetic subdivision, but we learned a lot of tools making it. But we're not done. There's some real automation ahead.

22. We have designed a subdivision, in effect, without point numbers. This is the beauty of CAD. But it is often desirable to make point numbers in order to stake-out the subdivision. To do this, run the COGO – Create Points from Entities which displays the following dialog box:



Set the **Starting Point Number** to 101 (helpful as our previously entered points [1-4] can be thought of as "control" points) and verify the dialog as shown and click **OK**. A second dialog, covering what entities to capture, appears next:

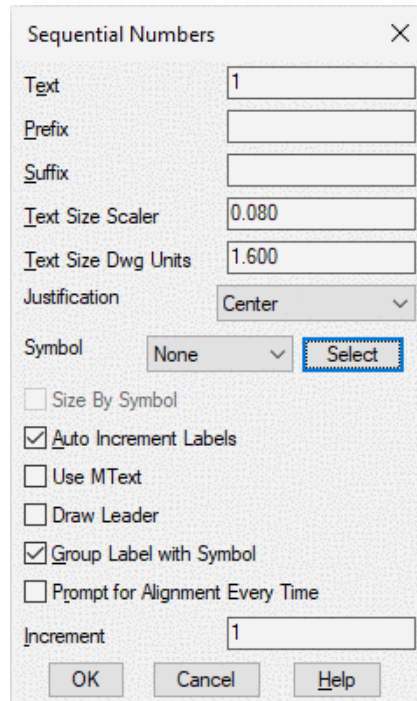


Accept the default settings and click **OK**. When prompted:

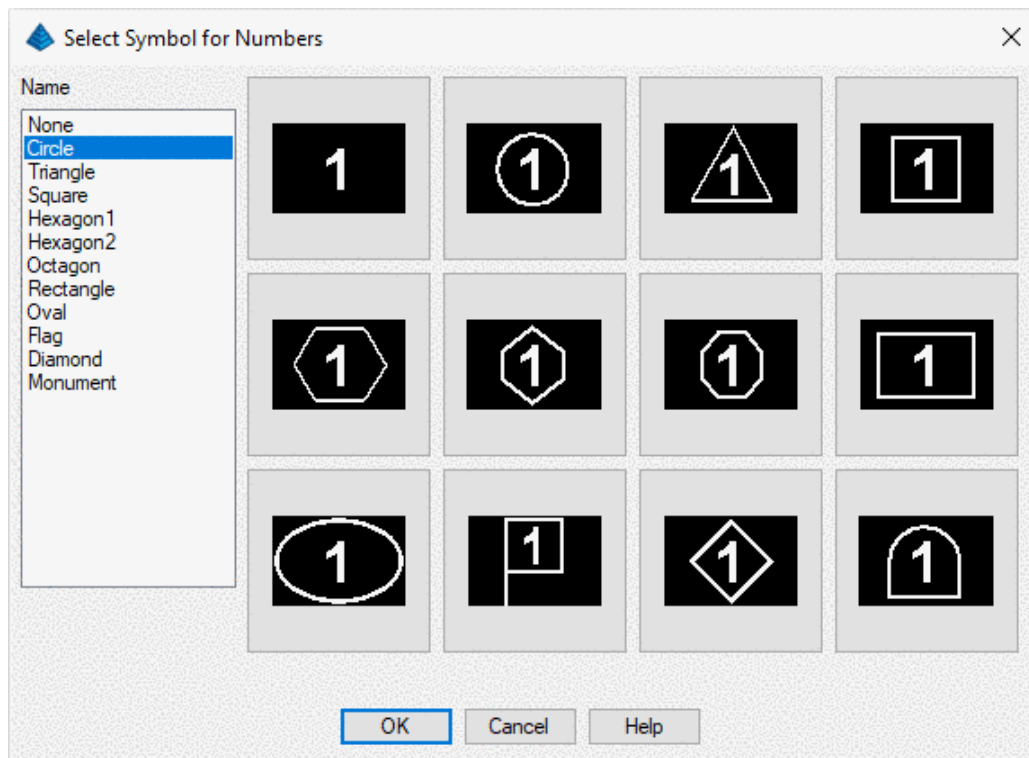
Select arcs, circles, faces, points, text, lines and polylines.
[FILter]/<Select entities>: type ALL and press Enter twice

All the point numbers for stakeout are created and stored into the active Coordinate File.

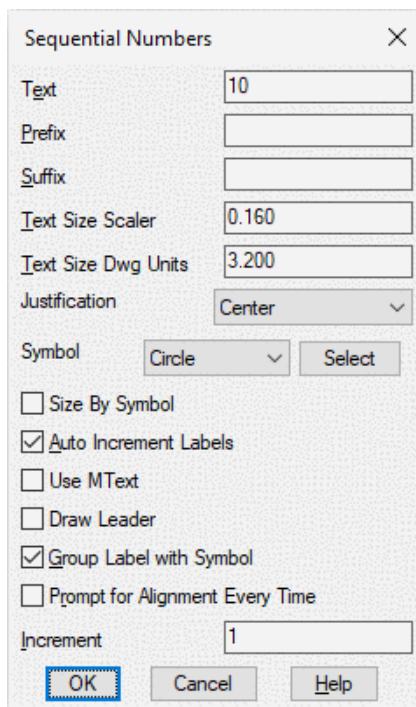
23. Our next goal is to number the lots, clockwise from the upper left. Use the Draw – Sequential Numbers command which displays the following dialog box:



Click on the **Select** button to select a desired symbol to circumscribe our sequential numbers as illustrated below:



Choose the circled text and click **OK** to return to the Sequential Numbers dialog box:

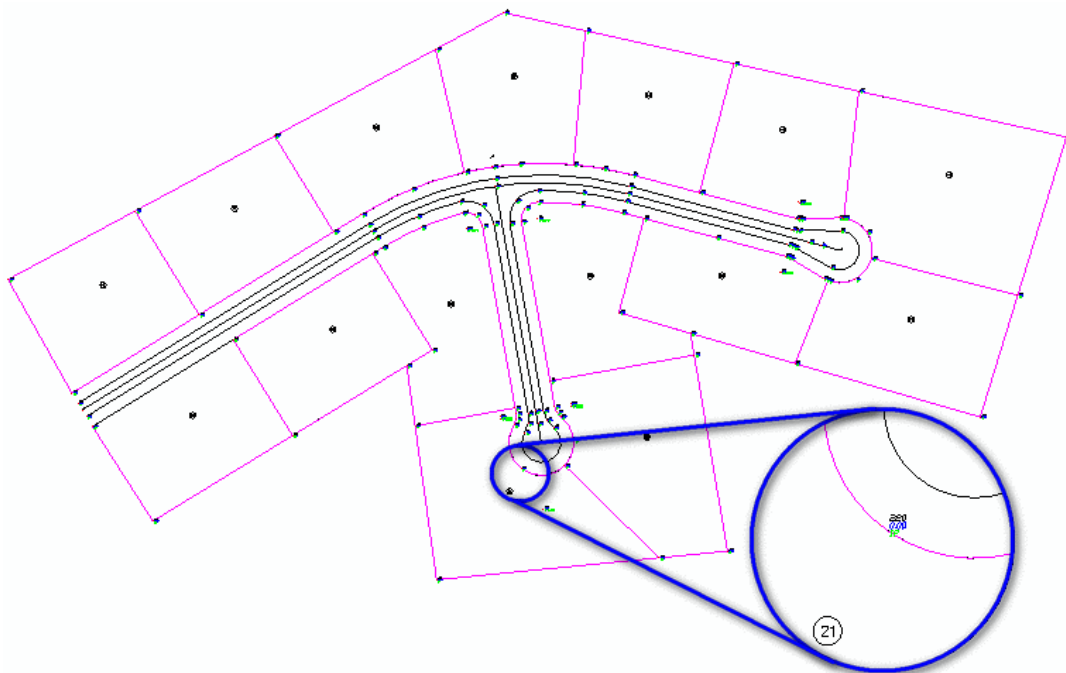


Set the Text value to **10** and the Text Size Scaler to **0.16** as shown and then click **OK**. When prompted:

Pick point for label position: pick near the center of the first upper left lot

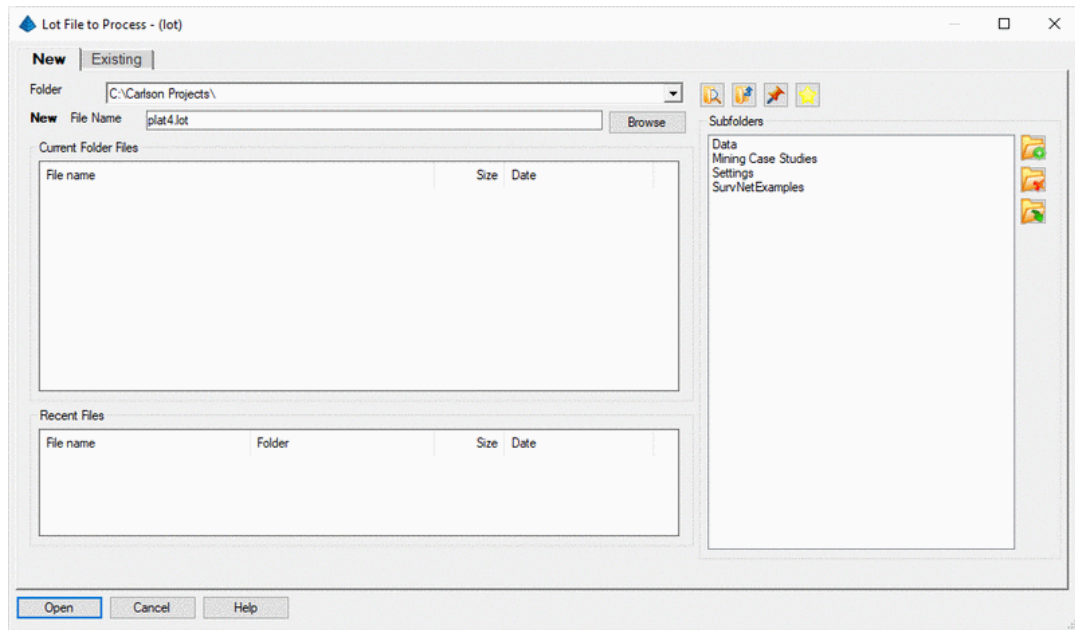
Pick point for label alignment: press F8 for <Ortho on>, pick to the right

Now pick near the center of all of the lots, going clockwise and press Enter at the conclusion of the last lot. When complete, press F8 again to set Ortho off. The resulting drawing, with point numbers, is shown below (enhanced for clarity):

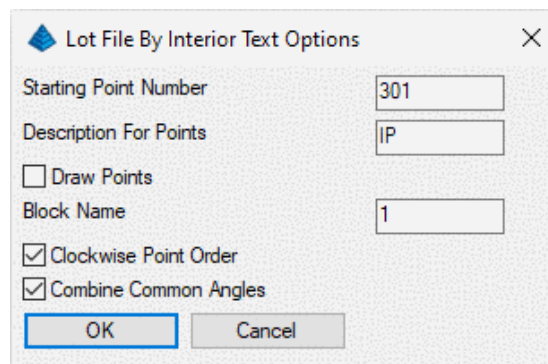


24. Official **Lot Files** can be created whenever a lot number or name exists within a lot as the sole text (other text may be present but could/would/should be frozen). We will play it safe and first freeze the points (PNTS) layer. Use the View – Freeze Layer by Selection command and pick on one of the points.

Next, issue the Area/Layout – Create Lots – Lot File by Interior Text. Supply the Lot File to Process value as shown below and click the Open button:



A dialog box will appear:



Set the values as shown above and click **OK**. When prompted:

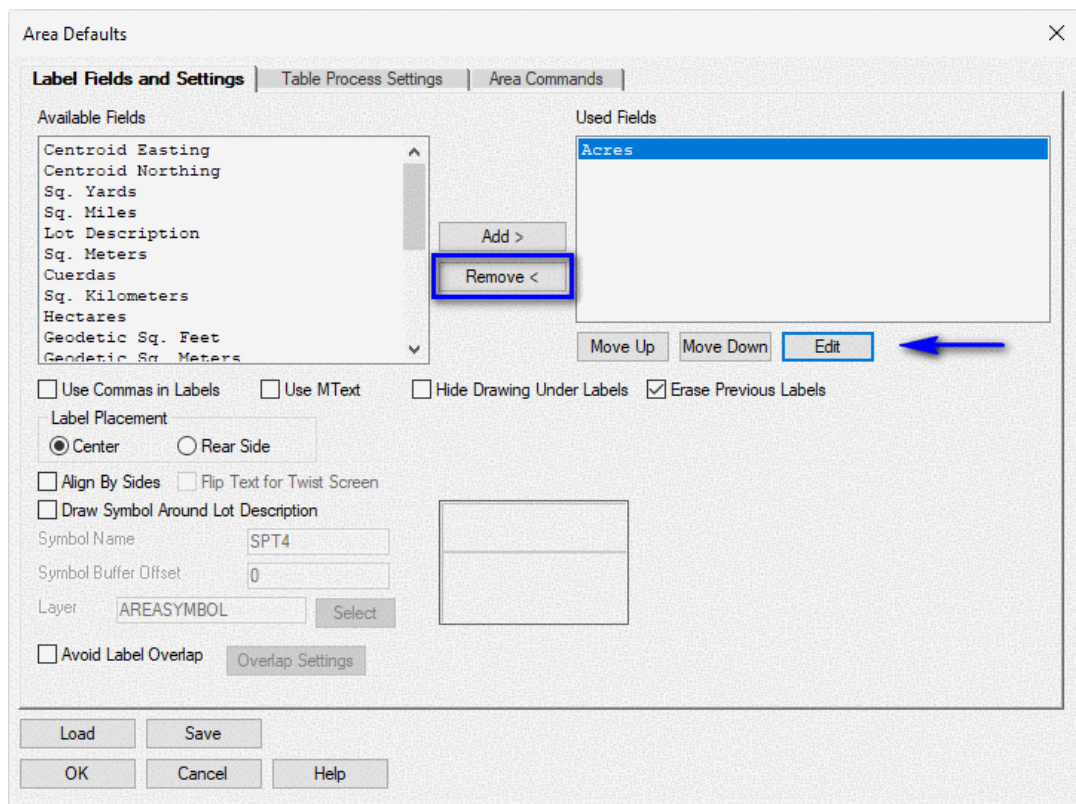
Select lot lines, polylines and text.

[FILter]/<Select entities>: type ALL and press the Enter button twice

The Lot File will be created.

NOTE: If we had not made points at all lot corners using **Create Points from Entities** command, the **Lot File by Interior Text** would create points at the Lot corners. This is the reason for the **Starting Point Number** prompt. If points are found, no new ones are created. Lot files must have points at all the corners.

25. Before we look at the Lot File, let's finish up and perform area annotation on the upper Lots. Select the Area/Layout – Area Defaults command:



Use the **Remove <** button to remove the **Sq. Feet** entry leaving only the **Acres** item in the *Items Used* portion of the dialog box. Use the **Edit** button to verify it is set to plot to 3 decimal places (*i.e.* 0.000) and click **OK** to complete the command.

Now choose the Area/Layout – Area by Interior Point command and when prompted:

Pick point inside area perimeter: pick inside Lot 10

Pick area label centering point (Enter for none): Pick near the center of Lot 10

Pick point inside area perimeter (Enter to end): repeat as prompted for Lots 11 through at least Lot 16 (continue with all Lots if you're so inclined) and press Enter when complete

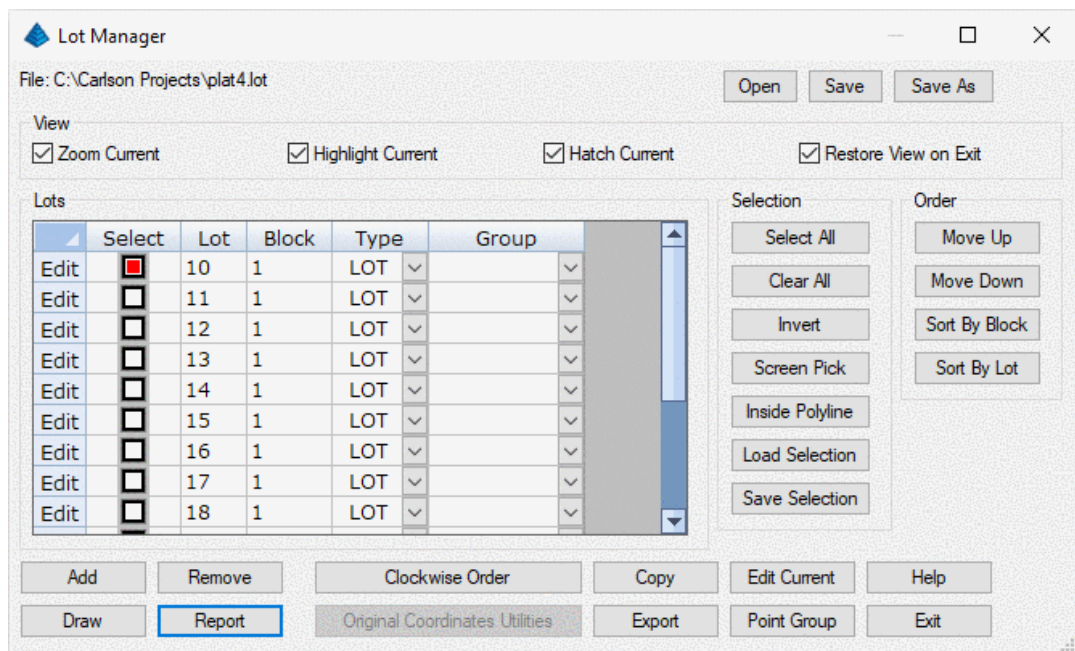
A detailed illustration is shown below:

21

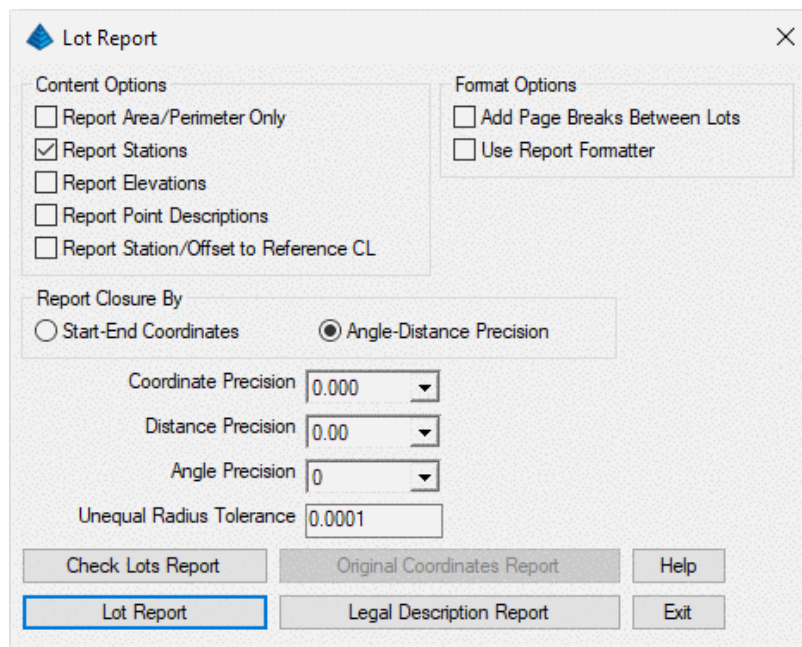
1.285 Acres

NOTE: For precise centroid placement, consider the use of the transparent Centroid snap ('centroid').

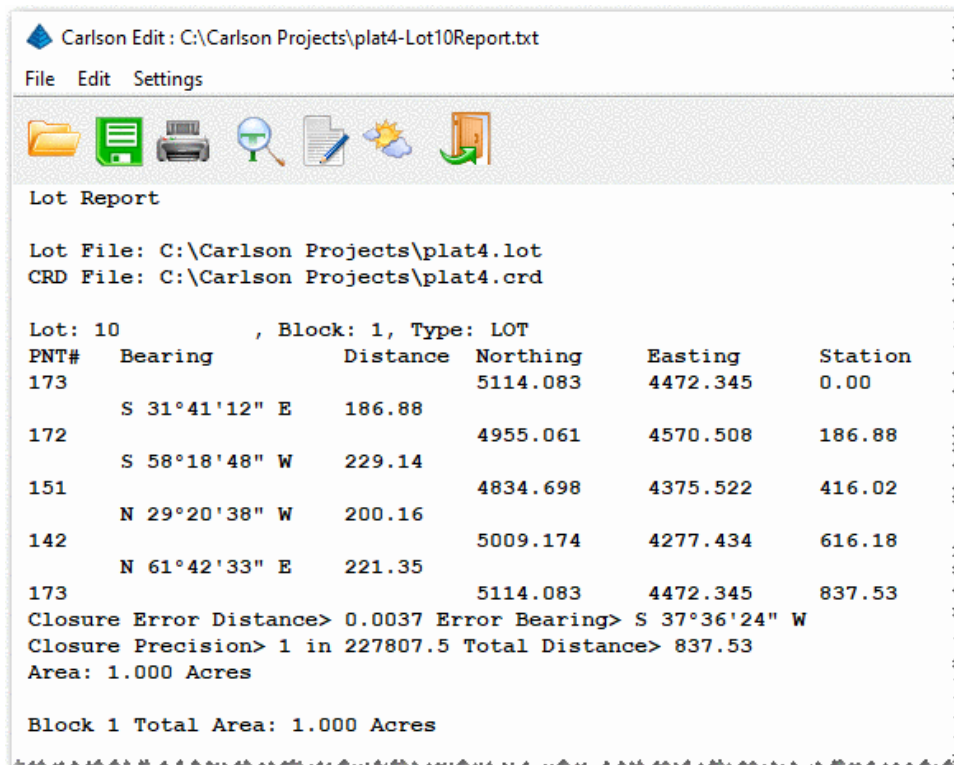
26. Let's explore the Lots created earlier. Run the Area/Layout – Lot File Manager command which displays the following dialog box:



Pick on Lot 10 and click **Report**. This will lead to the Lot Report dialog box:

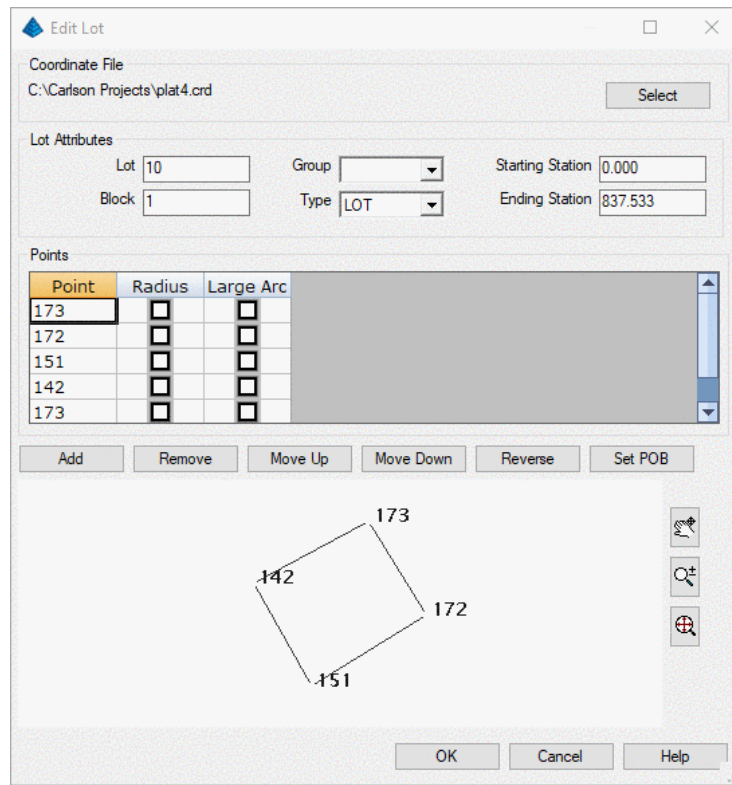


Be sure that your settings are as shown above and then click **Lot Report**:



This dialog is typical of the many Carlson Standard Report Viewer dialogs. You can click on one or more lines, highlight them and hit the delete key on the keyboard, and these lines will delete. You can edit lines directly in the dialog. You can also save the report to disk with the Save icon shown above. To exit, click the **Exit** (Doorway) button. The **Lot Manager** dialog box returns.

- The **Edit Current** option within the Lot Manager dialog box can be used to describe a lot by different point numbers, or to assign a lot to a different block. This is explained here and shown below for reference purposes only:



With Lot 10 selected, click **Edit Current** to display the dialog box above. Note the graphic display in the lower half, which maps the Points associated with the Lot. Also note the **Northwest corner** point number for Lot 10 as we'll use this information in the next step. Click Cancel and then Exit on the Lot Manager dialog box.

28. Presume you actually changed the point numbers that define Lot 10. That would cause the Lot to draw differently. Also, you could simply alter the coordinate values of a point in the current Lot File. That would also cause the Lot to draw differently. Let's take the latter approach.

Remember the **Northwest corner** point number for Lot 10? Let's change its coordinate value. Issue the Points – Edit Points command. A spreadsheet appears. Scroll down to your point number (142 in the image cited above):

Coordinate Data Sheet: C:\Carlson Projects\plat4.crd

File Edit Search Group Help

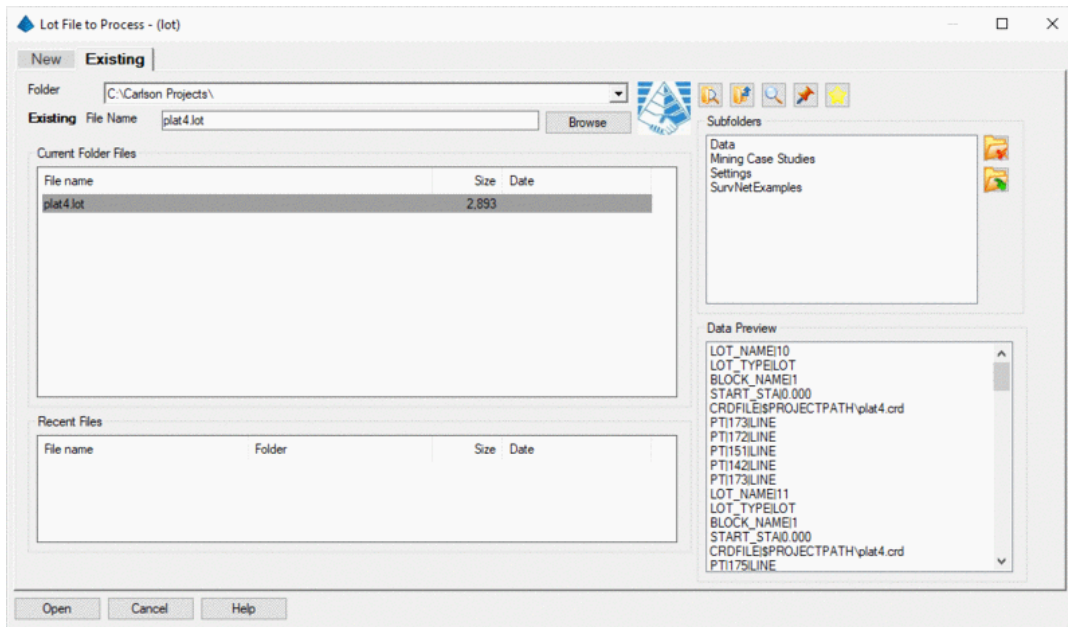
Point#	Northing	Easting	Elevation	Description
41	140	5072.8910	4841.3482	0.0000 IP
42	141	4798.9588	4397.5833	0.0000 IP
43	142	5009.1743	4277.4342	0.0000 IP
44	143	5418.3486	5037.6418	0.0000 IP
45	144	5227.5229	5900.6793	0.0000 IP
46	145	4798.1651	5768.4693	0.0000 IP
47	146	4926.6055	5324.0968	0.0000 IP
48	147	4783.6420	4407.0382	0.0000 IP
49	148	4916.9400	4622.9785	0.0000 IP
50	149	4769.7808	4713.8186	0.0000 IP
51	150	4638.5321	4499.6205	0.0000 IP
52	151	4834.6980	4375.5218	0.0000 IP
53	152	5108.6302	4819.2867	0.0000 IP

Point 142 Notes	
1	Original Northing = 5009.1743 Optional
2	
3	
4	

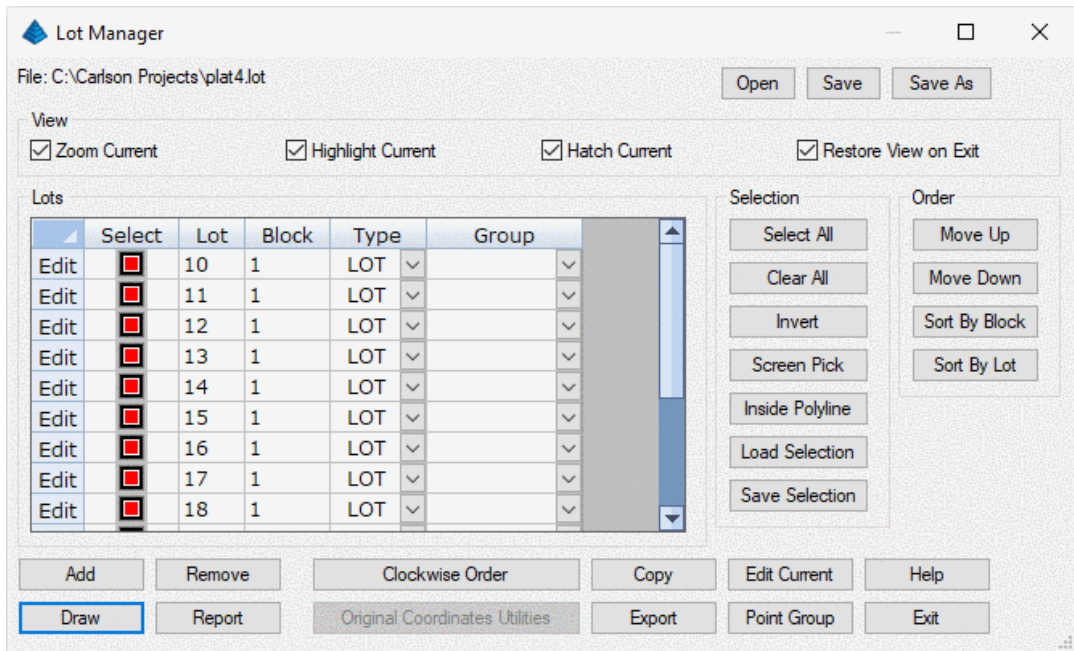
Click on the **Northing** and edit it to 5050. This is for illustration purposes. In reality, you might be fine-tuning your subdivision design points. As long as the same points define the lots, you are, in effect, able to make a ready-made new drawing. Once complete, issue (from the **Coordinate Data Sheet** dialog box) the File – **Save and Exit** option.

- Let's see what the change(s) look like in a brand new drawing. Before we draw the Lot File, save your drawing through the File – **Save** command.

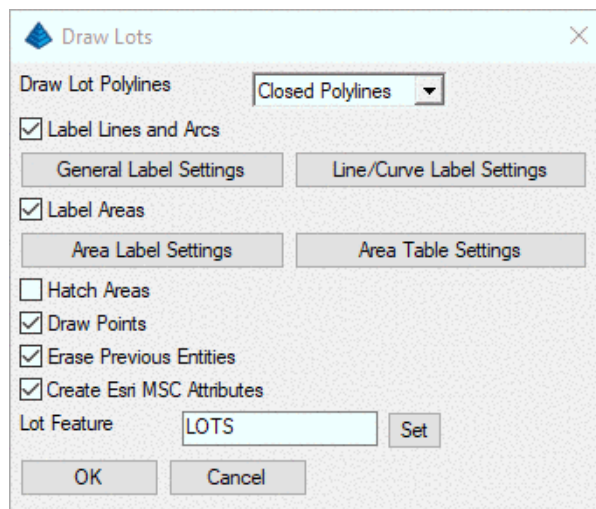
Then choose File – **New** and exit the Startup Wizard (if it appears), and go straight to the Area/Layout – **Lot File Manager** command as it also provides the tools for drawing Lot Files into the drawing:



From the **Existing**, select the *.lot file specified above and click Open. Follow this and select the existing **Plat4.crd** file from the earlier project. When the **Lot Manager** dialog appears, choose all Lots by clicking **Select All** and then click **Draw**.

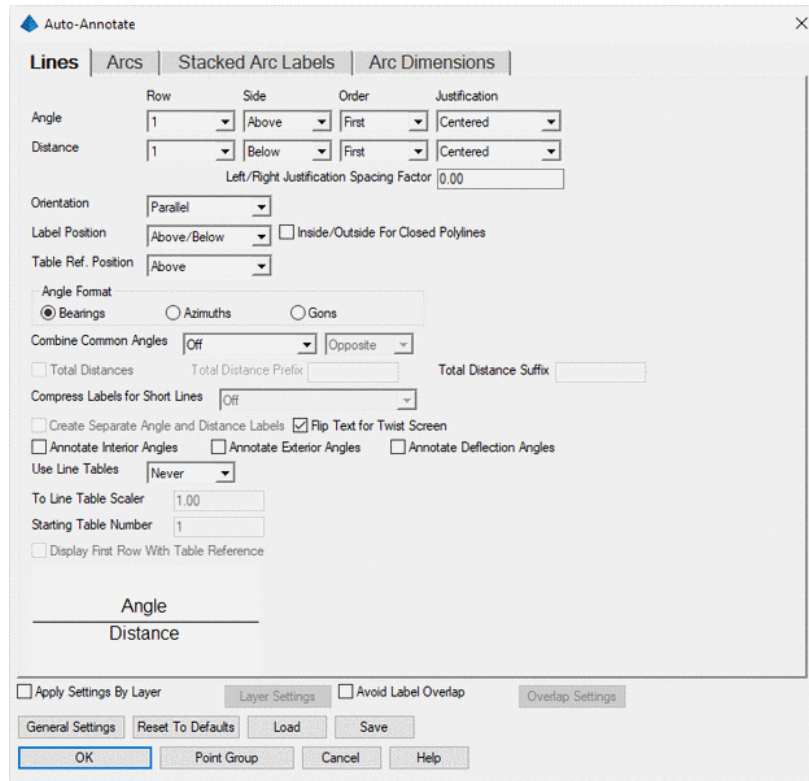


We have the ability to further define how the Lots are drawn and/or labeled:



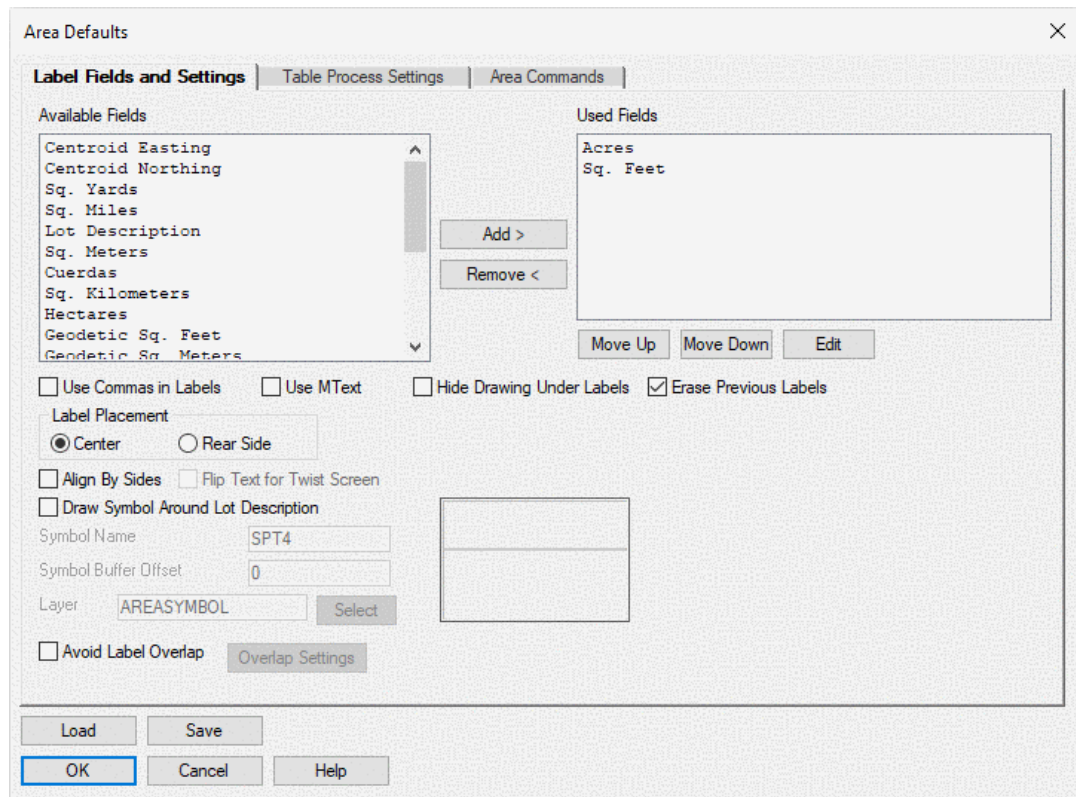
Apply settings required or as shown in the image above and:

- Click the **Line/Curve Label Settings** button to display the Auto-Annotate dialog box (also found under the **Annotate** menu) and set the values as shown below:



Click its **OK** button when complete.

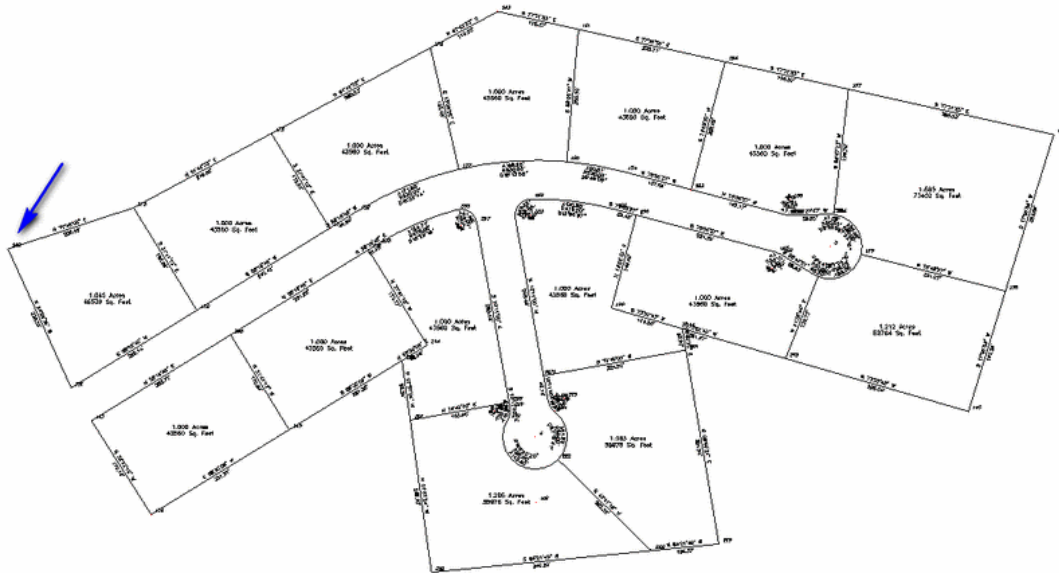
- Click the **Area Label Settings** button to re-confirm the display the **Area Defaults** options discussed earlier (also found under the **Area/Layout** menu). Let's reuse the **Sq. Feet** item to the **Used Items** as shown below:



Click its **OK** button when complete.

Click **OK** to draw the Lots and eventually return to the **Lot Manager** dialog box (click its **Exit** when complete).

This leads to the plot shown below (drawn at 1" = 60' for clarity), created entirely from stored Lot Files, and showing our revision of Lot 10:



This completes the Lesson 4 tutorial: Intersections and Subdivisions.

SurvNET

This tutorial is divided into two lessons:

1. Processing an Assumed Coordinate System 2D Total Station Network
2. Processing a 3D Network With Both Total Station Data and GPS Vectors

Least Squares is very flexible in terms of how the survey data needs to be collected. Generally speaking, any combination of angles and distances, combined with a minimal amount of control points and azimuths, are needed. This data can be collected in any order. But there needs to be at least some redundancy in the measurements.

Redundant measurements are measurements that are in excess of the minimum number needed to determine the unknown coordinates. Redundancy can be created by including multiple GPS, and other control points, within a network or traverse. Measuring angles and distances to points in the network that have already been located create redundancy. Running additional cut-off traverses, or additional traverses to existing control points, creates redundancy. Following are some general rules and tips in collecting data for least squares reduction:

- Backsights should be to point numbers. Some data collectors allow the user to backsight an azimuth not associated with a point number. SurvNET requires that all backsights be associated with a point number.
- There has to be at least a minimum amount of control. There has to be at least one control point. Additionally, there needs to be either one additional control point or a reference azimuth. Control points can be entered in either the raw data file, or there can be a supplemental control point file containing the control point. Reference azimuths are entered in the raw data file. The control points and azimuths do not need to be for the first points in the raw file. The control points and azimuths can be associated with any point in the network or traverse. The control does not need to be adjacent to each other. It is permissible to have one control point on one side of the project, and a reference azimuth on the other side of the project.
- At least one of the control points needs to be occupied. There may be situations where no control point is ever occupied in the network, but only backsighted. In these situations, a preliminary value for one of the occupied points needs to be computed and entered as a floating point control point.

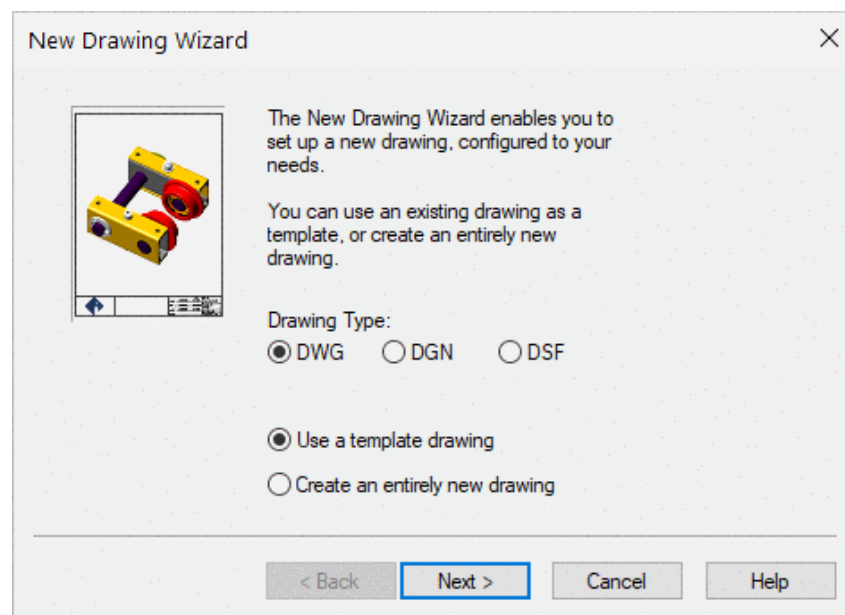
- Some data collectors do not allow the surveyor to shoot the same point twice using the same point number. SurvNET requires that all measurements to the same point use a single point number. The raw data may need to be edited after it has been downloaded to the office computer to ensure that points are numbered correctly.
- The majority of all problems in processing raw data are related to point number problems. Using the same point number twice to different points, not using the same point number when shooting the same point, misnumbering backsights or foresights, and misnumbering control points are all common problems.
- It is always best to explicitly define the control for the project. A good method is to put all the control for a project into a separate raw file. A big source of problems with new users is a misunderstanding in defining their control for a project.
- Some data collectors may have preliminary unadjusted coordinates included with the raw data. These coordinate records should be removed from the raw file. The only coordinate values that should be in the raw file are the control points.
- When a large project is not processing correctly, it is often useful to divide the project into several raw data files and debug and process each file separately, as it is easier to debug small projects. Once the smaller projects are processing separately, they can be combined for a final combined adjustment.

It is suggested that **Lesson 1** be completed before starting into **Lesson 2**. Throughout the lessons, we will cover the process of reducing and adjusting raw survey data into final adjusted coordinates, using the SurvNET program. The tutorial will describe the reviewing and editing of the raw data prior to the processing of the raw data. Next, the least squares project settings will be described, and then the final report generated from the least squares processing will be reviewed. Let's begin.

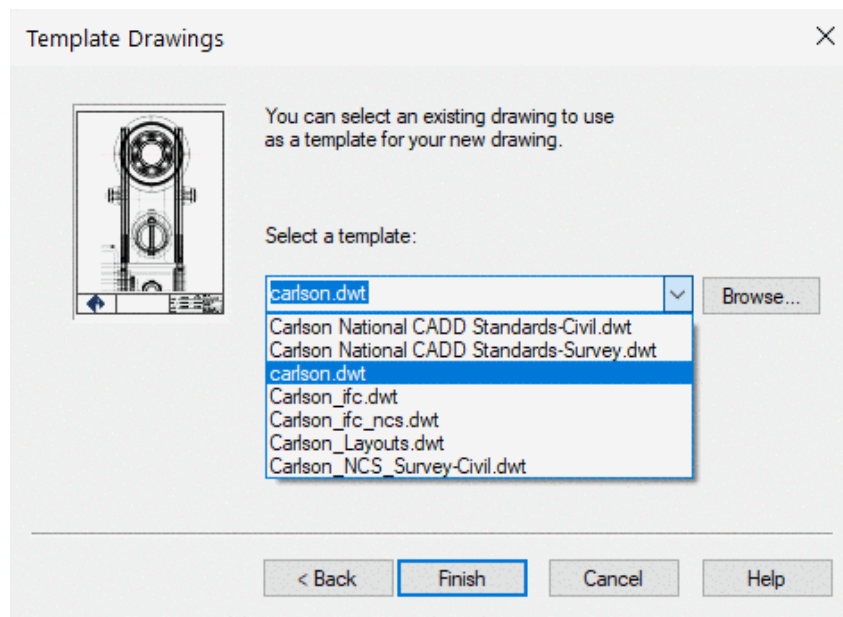
1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **New Drawing**.
 - If you get the Startup Wizard dialog box, click **New**.
 - If you are taken directly into CAD, click the File – **New** command.

The first of several Startup Wizard dialog boxes appears:

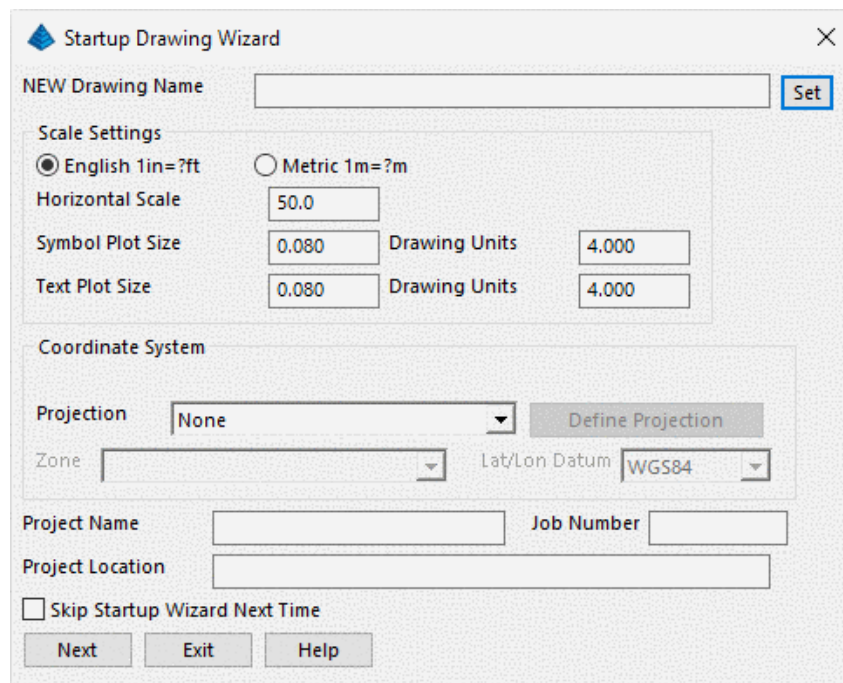
- (a) Choose the DWG document type and the desire to base the document on a **Drawing Template** as illustrated below and then click **Next >**:



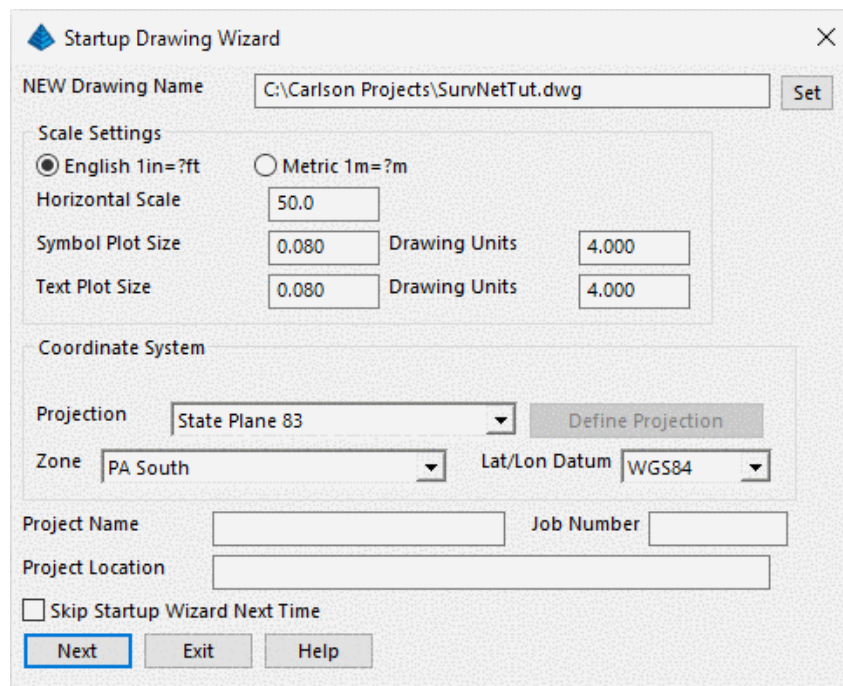
- (b) Choose the **carlson.dwt** as illustrated below (or *surv.dwt* if carlson.dwt is not available) and click **Finish**:



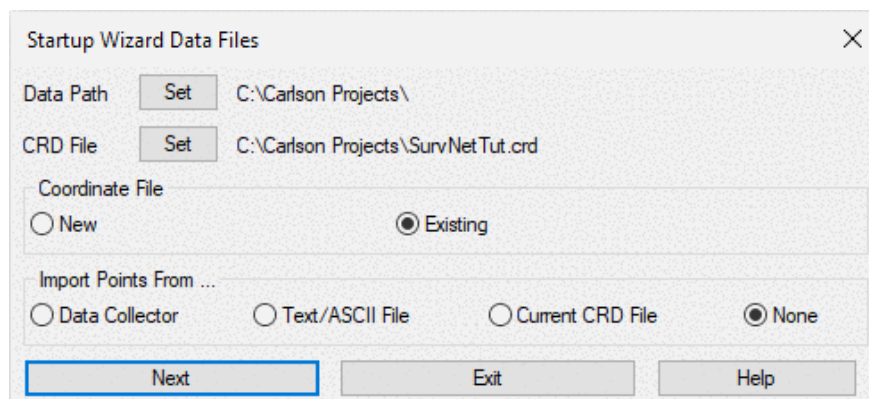
- (c) We can now begin the more pertinent settings for the project to come based on some preliminary settings that should be similar to the default scenario shown below:



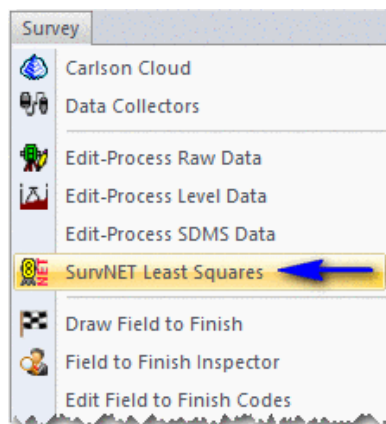
- (d) Click **Set** at the top of the dialog box, and enter in a NEW Drawing Name called **SurvNetTut**. Verify that the other settings match the settings shown below, and click **Next**:



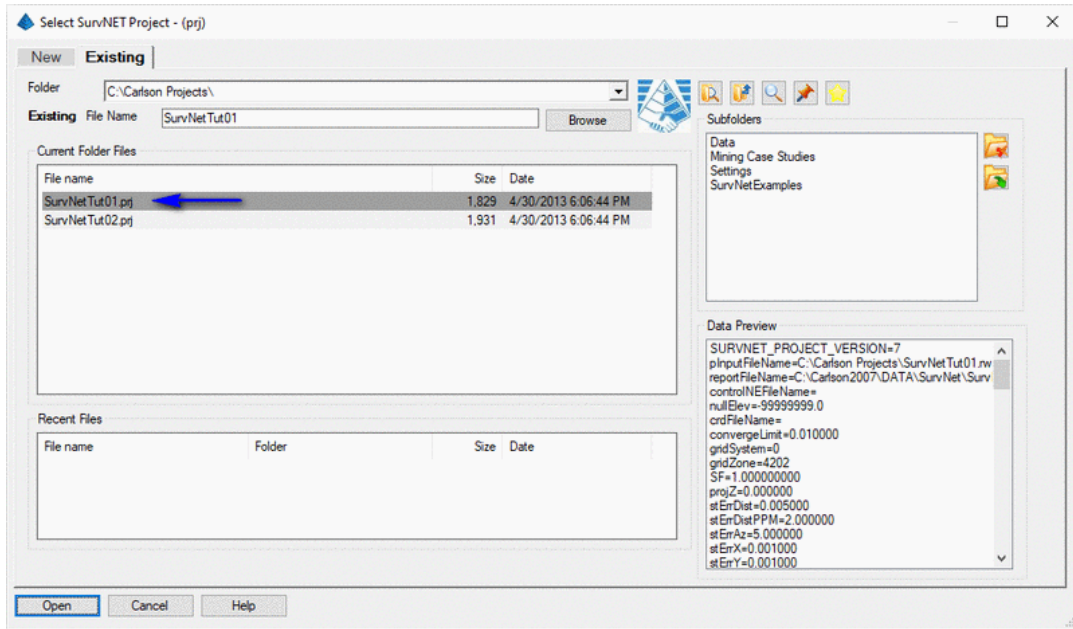
- (e) You will see the Startup Wizard Data Files dialog to set/confirm where to store data and indicate an information source for points/coordinates. Set/match the values as shown below and click **Exit**:



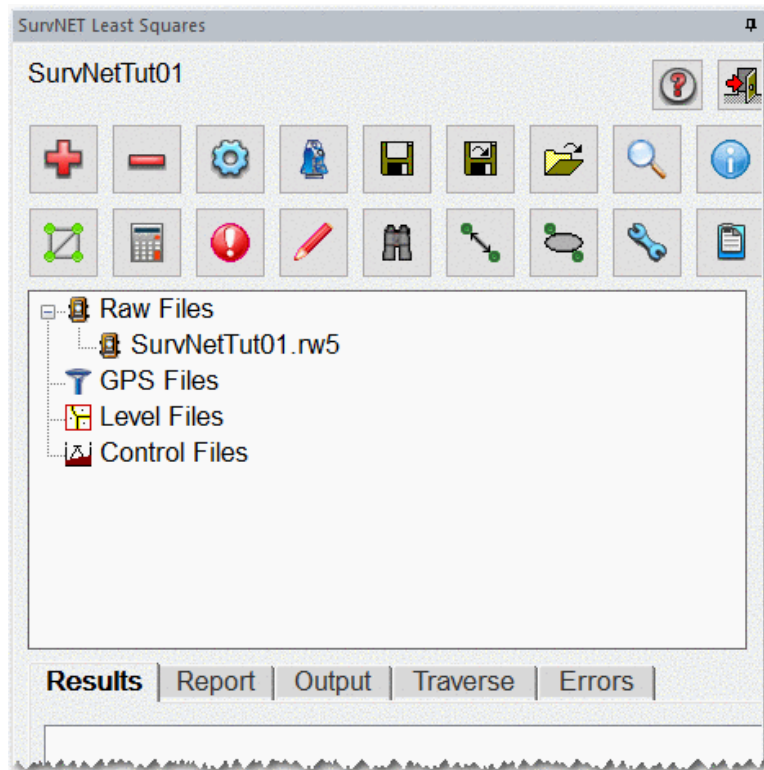
2. **Lesson 1 - Processing an Assumed Coordinate System 2D Total Station Network.** The raw data files associated with this tutorial are located in the *Carlson Projects* folder, under the installation folder on your computer (example: C:\Carlson Projects). Activate the Survey menu via Settings – Carlson Menu – **Survey Menu**.
3. The easiest way to start the program is to issue the Survey – SurvNET command as illustrated below:



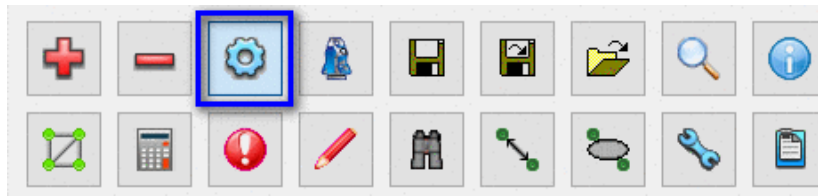
This, in turn opens the following dialog box:



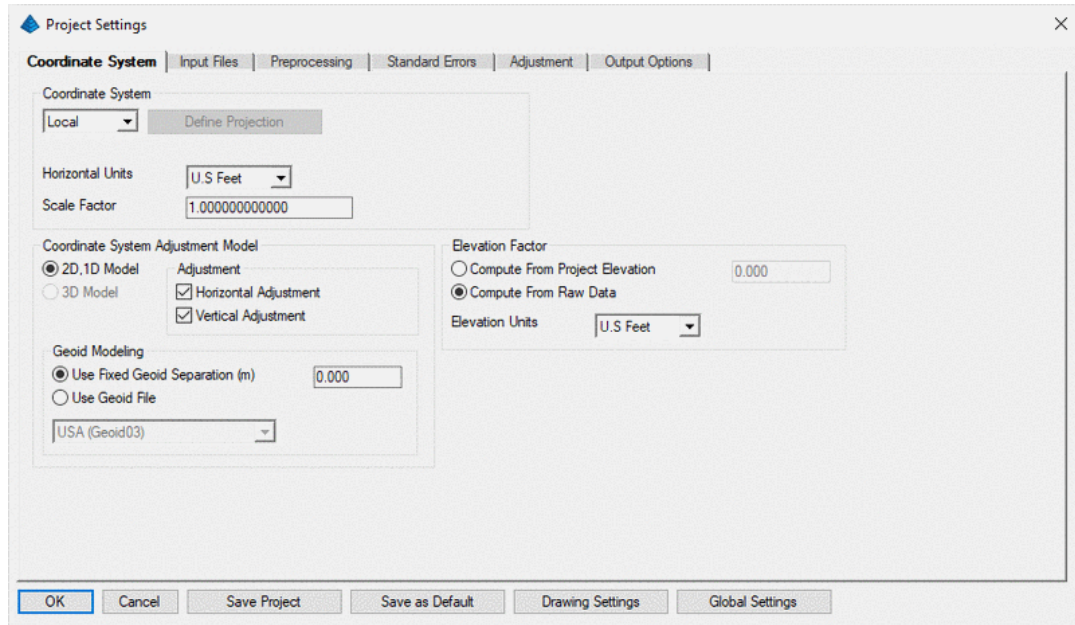
Select the **SurvNetTut01.prj** project and click **Open**. This will open the default SurvNET *project-tree* docked dialog box interface as shown below where we will process the contents of the **SurvNetTut01.rw5** "raw" file:



- Learning the meaning and implications of the different project settings is the most critical initial step in learning how to use SurvNET. Let's review the different project screens. Click the Settings (Gear) button as shown below:

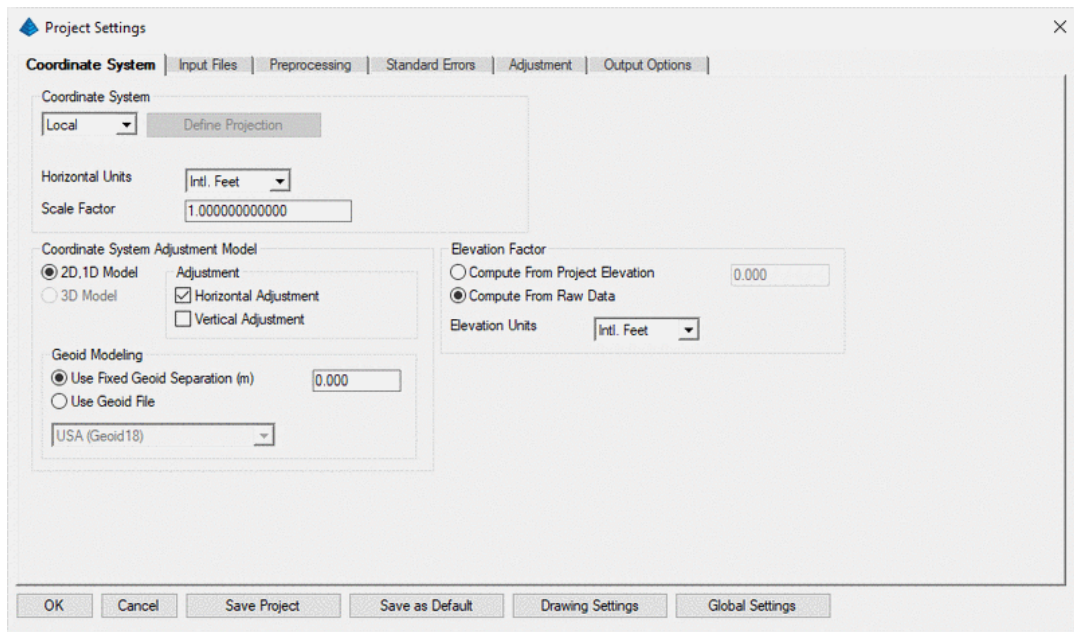


to display the dialog box below:

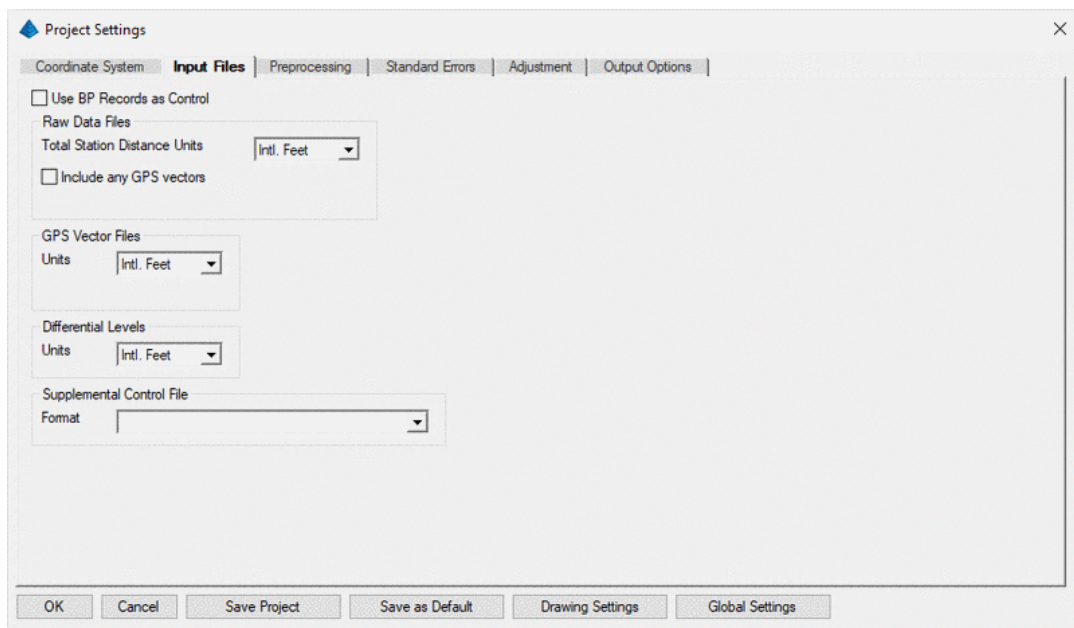


NOTE: In this dialog, the different settings required for the Least Squares reduction are available in the different tabs of the dialog box. When all of the settings are set as desired, click **OK** to save the changes.

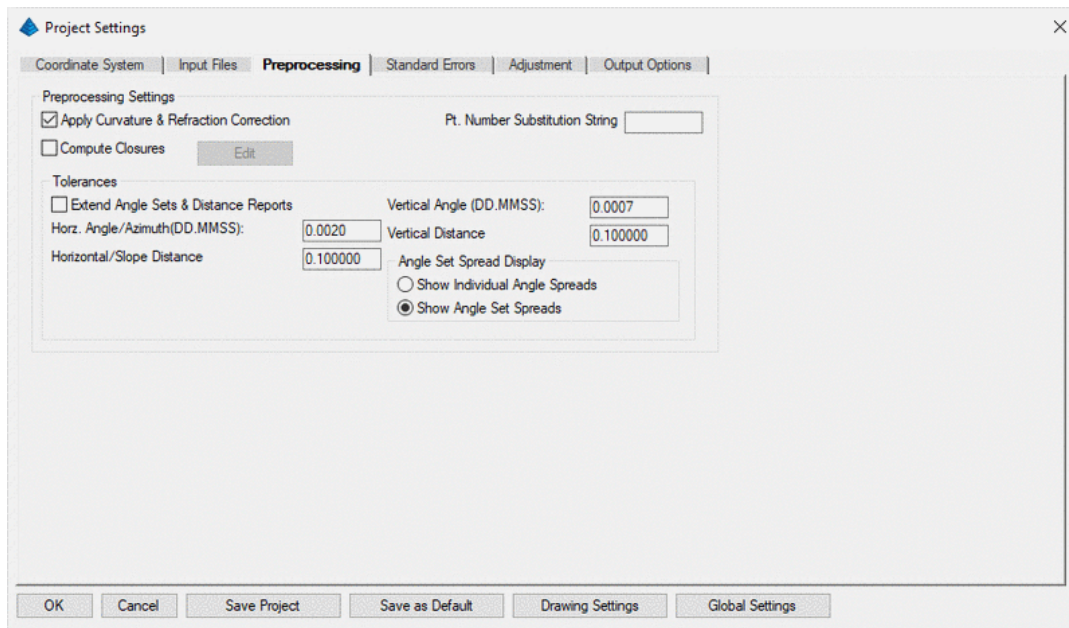
- Settings - Coordinate System: For the purpose of this tutorial, the **Coordinate System** settings tab should look as follows before proceeding to the next step. To use an assumed coordinate system, the **Local Coordinate System** needs to be selected, and the 2D,1D Adjustment Model must be chosen. When using a local coordinate system, the distance units are not important other than for display purposes in the report. Computing elevation factors and performing *Geoid* modeling is not applicable to assumed datums. Notice that in this example we are not performing a vertical adjustment (make this and other changes as applicable):



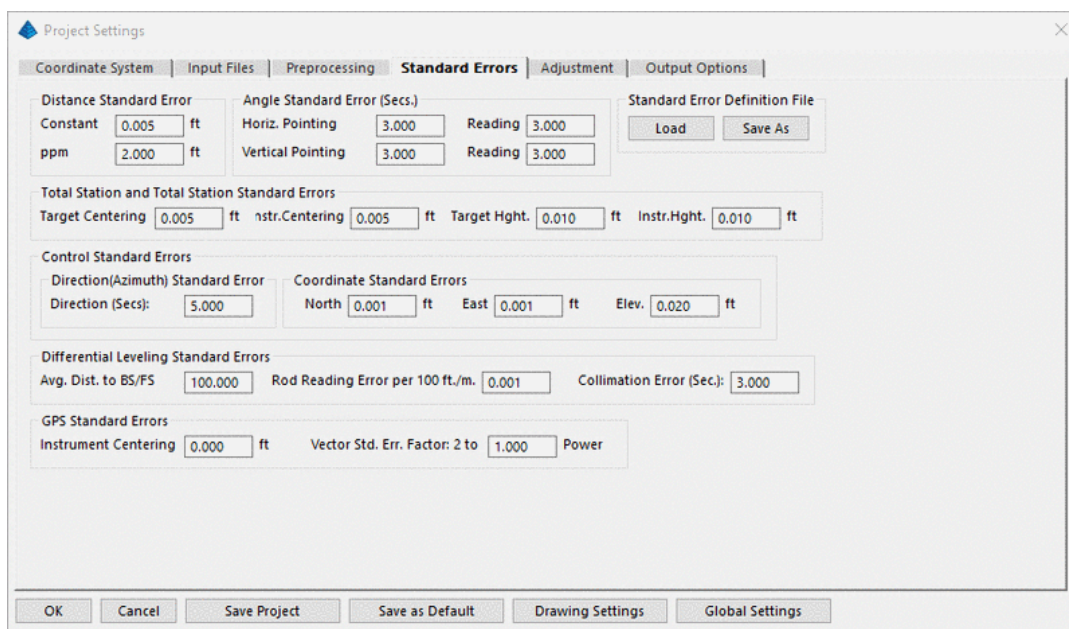
- Activate the Input Files tab. The Input Files settings is where you establish pertinent values for how the files are treated. SurvNET allows you to have multiple raw files in a single project. The ability for multiple raw files allows flexibility in collecting the data and processing large projects. It is typically easier in a large project to analyze and edit subsets of the total project, before combining all the data for a final adjustment. Notice that since we are working in a *Local* coordinate system and using the *2D, 1D Adjustment Model*, GPS vectors cannot be incorporated into this project. Make any necessary changes to match the values shown below:



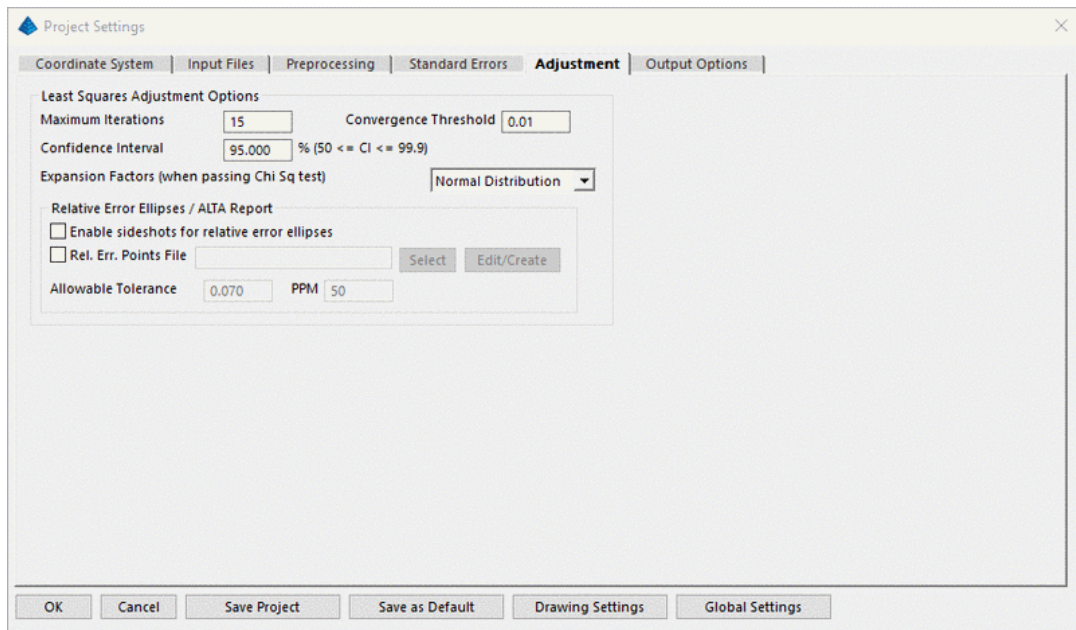
- Activate the Preprocessing tab to review the Preprocessing settings. Preprocessing consists of reducing and averaging all the multiple measurements, applying curvature and refraction correction, reducing the measurements to grid if appropriate, and computing unadjusted traverse closures if appropriate. Much of the data validation is performed during the preprocessing step. For the purpose of this tutorial, the **Preprocessing** settings should look as follows before proceeding to the next step:



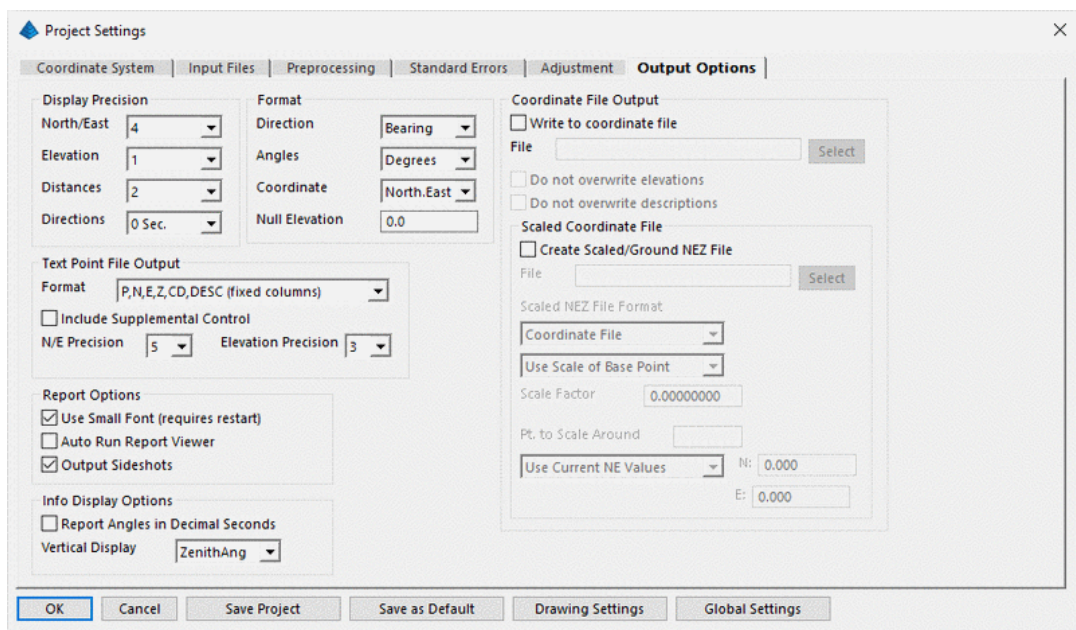
- Activate the Standard Errors tab to review the Standard Errors settings. Standard Errors are an estimate of the different errors you would expect to obtain based on the type equipment and field procedures you used to collect the raw data. For example, if you are using a 5 second theodolite, you could expect the angles to be measured within +/- 5 seconds (Reading error). The **Standard Errors** settings should look as follows before proceeding to the next step:



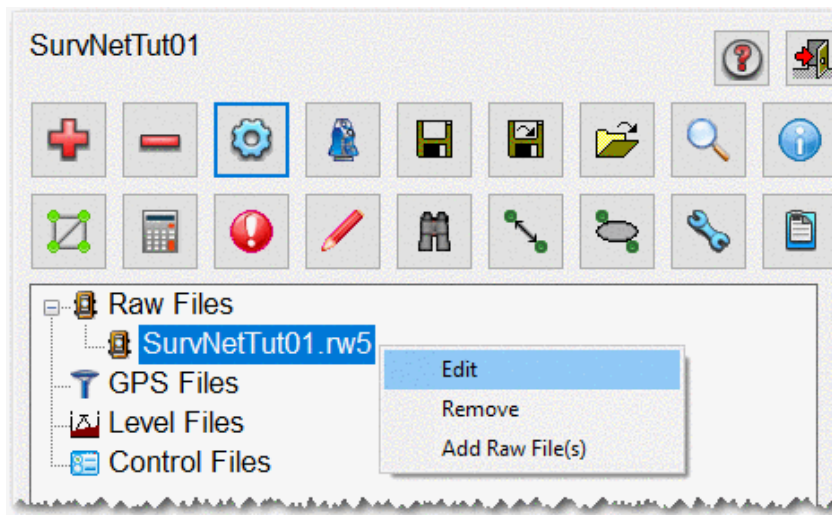
- Activate the Adjustment tab to review the Adjustment settings. The Adjustment settings affect how the actual Least Squares portion of the processing is performed. Additionally, from the screen the user can set whether ALTA reporting is performed. The Adjustment settings should look as follows before proceeding to the next step:



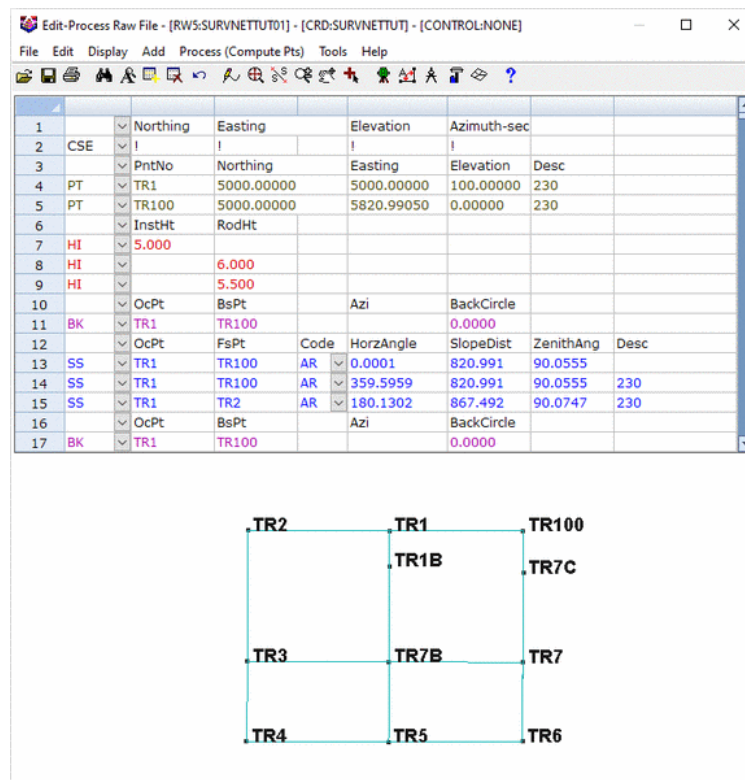
- Activate the Output Options tab to review the Output Options settings. These settings apply only to the output of data to the report files. These settings do not affect computational precision. For the purpose of this tutorial, the **Output Options** settings should look as follows before proceeding to the next step. Press **OK** to return to the main SurvNET screen.



5. Let's examine the underlying *raw* data associated with this project. Right-click on the file shown below and choose the **Edit** option:

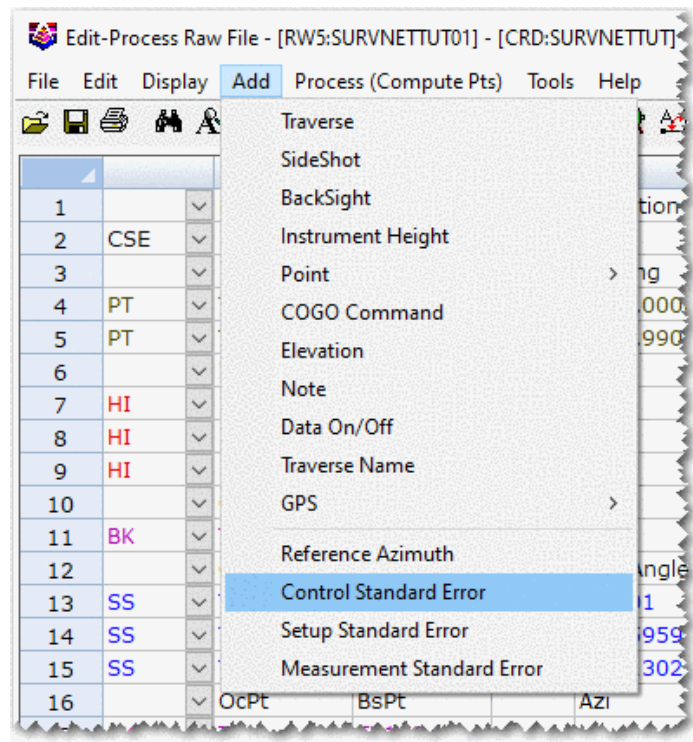


- The Carlson Edit-Process Raw File dialog box appears. If there are problems with the raw data (*i.e.* point numbering problems or incorrect rod heights), the raw data can be edited from this dialog:



Review the following Standard Errors and Control Points discussion before exiting the Raw File editor:

- The default **Standard Errors** for points are defined in the Standard Errors tab of the **Settings** command as discussed earlier. There are times when the default values may need to be overridden. For example, the control may be from GPS and the user has differing Standard Errors for various GPS points. Or maybe some of the control points were collected with RTK methods, and other GPS points collected with more accurate static GPS methods. Standard Error for individual points can be inserted into the raw data file that will supersede the otherwise default values. The following is the menu option used to insert *Control Standard Errors* into the raw file:

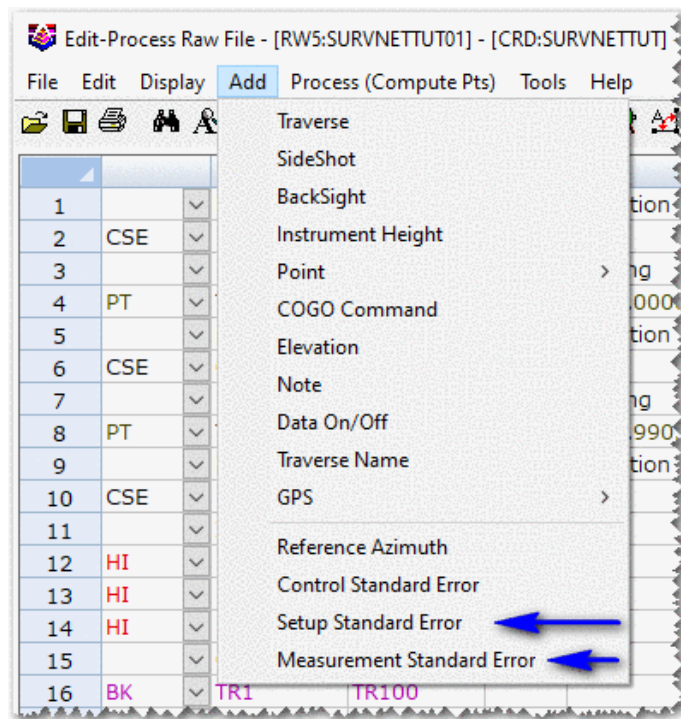


Notice in the above raw data file that points **TR1** and **TR100** are the *Control Points* for this project. Also, notice there is a Standard Error record (**CSE**) preceding the control points.

- The CSE record has the Exclamation (!) character in the N,E,& Z field. The '!' character designates that all following control points will be "fixed." Points that are fixed will not be adjusted during the adjustment. Placing a very small Standard Error on a control point is almost equivalent to fixing the point. Points can also be designated to be floating points by using the Number sign (#) character. The only practical use of creating a floating point is if SurvNET cannot compute preliminary coordinates because no control point is occupied. The surveyor can compute a preliminary value for one of the occupied points, and insert that point as a floating point. The floating point will be adjusted, and no weight will be given to the floating coordinate values.
- Standard Error records affect all the records that follow the Standard Error record. To revert the Standard Errors back to the default values, a CSE record can be inserted containing the Asterix (*) character. In the following example (shown for illustrative purposes), point **TR1** has been designated as a fixed point. **TR100** has a Northing Standard Error of **0.02** and an Easting Standard Error of **0.01**. Additionally, the Elevation for point **TR100** would not change. Following the **TR100** point record is a CSE record containing the '*' character in all fields. So, if there were any control points further down in the raw data file, they would use the default Standard Errors as set in the **Settings** dialog box.

		Northing	Easting	Elevation	Azimuth-sec			
1								
2	CSE	!	!	!	!			
3		PntNo	Northing	Easting	Elevation	Desc		
4	PT	TR1	5000.00000	5000.00000	100.00000	230		
5		Northing	Easting	Elevation	Azimuth-sec			
6	CSE	0.02	0.01	!	*			
7		PntNo	Northing	Easting	Elevation	Desc		
8	PT	TR100	5000.00000	5820.99050	0.00000	230		
9		Northing	Easting	Elevation	Azimuth-sec			
10	CSE	*	*	*	*			
11		InstHt	RodHt					
12	HI	5.000						
13	HI		6.000					
14	HI		5.500					
15		OcPt	BsPt	Azi	BackCircle			
16	BK	TR1	TR100		0.0000			
17		OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	Desc

- There may be times when non-Control Standard Errors need to be overridden for certain measurements. For example, if fixed tripods were used for backsights and foresights for part of the traverse, and hand-held rods were used for another portion of the traverse, it would be appropriate to have differing **Rod Ctr** Standard Errors for the different sections of the raw data.
- Standard Errors for angles and distances can also be inserted into the raw data file using **Setup Standard Error** and **Measurement Standard Error** commands as indicated below:



The Standard Errors set by these inserted records override the default Standard Errors. In the following example, a Setup Standard Error (SSE) record has been inserted in record 11. The SSE record affects all setup data that follow until another SSE record is inserted. In the following example, the Foresight Rod Centering error is set to 0.005, the Total Station Centering error is set to 0.005, the Total Station Measure-up error is set to 0.005 and the Foresight Measure-up error is set to 0.005:

1		▼ Northing	Easting	Elevation	Azimuth-sec		
2	CSE	▼ !	!	!	!		
3		▼ PntNo	Northing	Easting	Elevation	Desc	
4	PT	▼ TR1	5000.00000	5000.00000	100.00000	230	
5	PT	▼ TR100	5000.00000	5820.99050	0.00000	230	
6		▼ InstHt	RodHt				
7	HI	▼ 5.000					
8	HI	▼	6.000				
9	HI	▼	5.500				
10		▼ Rod Ctr	Inst Ctr	Inst Hgt	Rod Hgt		
11	SSE	▼ 0.005	0.005	0.005	0.005		
12		▼ OcPt	BsPt	Azi	BackCircle		
13	BK	▼ TR1	TR100		0.0000		
14		▼ OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng
15	SS	▼ TR1	TR100	AR	0.0001	820.991	90.0555
16	SS	▼ TR1	TR100	AR	359.5959	820.991	90.0555
17	SS	▼ TR1	TR2	AR	180.1302	867.492	90.0747

- The following is another example where it would be appropriate to insert a Measurement Standard Error (MSE) record into the raw data. If two different total stations with different accuracy specifications were used to collect the data, it would be appropriate to have different standard errors for the different sections of the raw file, depending on which total station was used to collect the data. In the following example, a MSE record has been inserted for record 27. The Horizontal Pointing and Reading error has been changed to 5 seconds, and the Vertical Pointing and Reading error has been changed to 10 seconds. The inserted MSE record will affect all following raw data until another MSE is inserted:

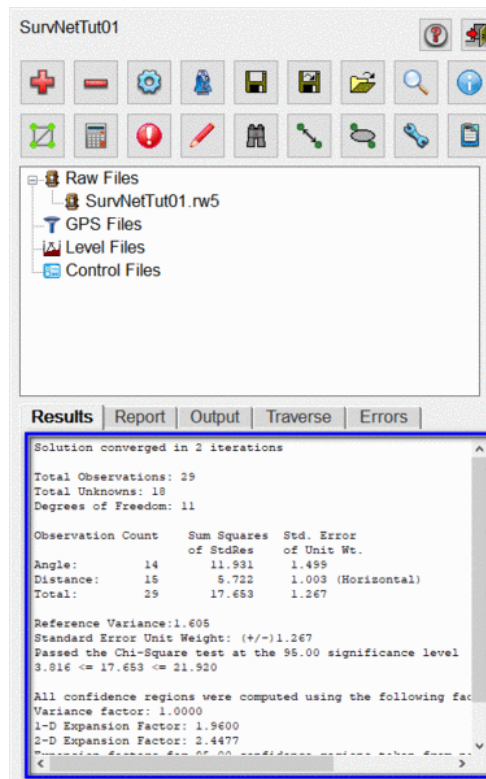
21		▼ OcPt	BsPt		Azi	BackCircle		
22	BK	▼ TR1	TR100			0.0000		
23		▼ OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	Desc
24	SS	▼ TR1	TR100	AR	0.0000	820.990	90.0600	230
25	SS	▼ TR1	TR2	AR	180.1308	867.493	90.0733	230
26		▼ HA Pt-sec	HA Rd-sec		VA Pt-sec	VA Rd-sec	Distance	Dist PPM
27	MSE	▼ 5	5		10	10	0.005	5
28		▼ OcPt	BsPt		Azi	BackCircle		
29	BK	▼ TR1	TR100			0.0000		
30		▼ OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	Desc
31	SS	▼ TR1	TR100	AR	0.0000	820.990	90.0600	230
32	SS	▼ TR1	TR2	AR	180.1310	867.489	90.0747	230
33		▼ OcPt	BsPt		Azi	BackCircle		
34	BK	▼ TR1	TR100			0.0000		
35		▼ OcPt	FsPt	Code	HorzAngle	SlopeDist	ZenithAng	Desc
36	SS	▼ TR1	TR100	AR	359.5959	820.991	90.0555	230
37	SS	▼ TR1	TR2	AR	180.1316	867.492	90.0745	230

Issue the File – **Exit** to dismiss the Raw File editor (discard any/all changes).

7. After exiting the raw data editor, we are ready to perform the Least Squares adjustment. Click the **Network Adjustment** (Double Triangle) button as shown below:



The least squares adjustment is performed, and the results from the adjustment are displayed. If the solution converged correctly, the report should look similar to the following window:



If there were errors or the solution did not converge, an error message dialog will be generated. If there are errors, you will need to return to the **Raw File Editor** to review and edit the raw data. Since the tutorial example should have converged, we will next review the various forms of output, including *graphical* representations of the adjusted network which can be:

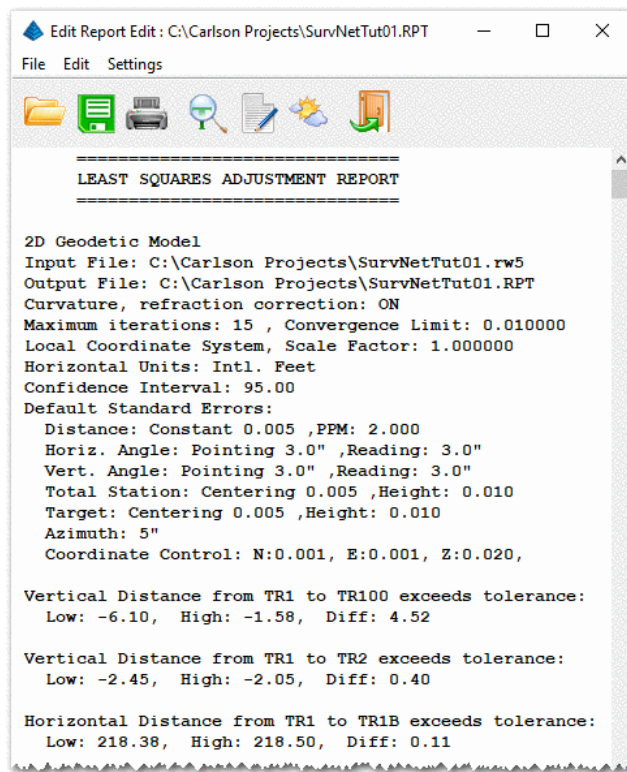
- Drawn into CAD through the **Draw** (Pencil) button, and/or
- Temporarily previewed through the **Quick View** (Binoculars) button.

Additionally, there are four reports created by the Least Squares program during processing. These include:

- The .rpt file (Main Report), which is the primary Least Squares report file summarizing the data and the results from the adjustment,
- The .out file (Output Report), which contains a listing of the final coordinates,
- The .err file (Error Report), which contains any errors or warnings that were generated during processing,
- The .res file (Results Report), which contains statistical data that was generated during processing.

To obtain these reports, click on the **Report** (Clipboard) button to specify the desired report which will display in the Standard Report Viewer which will be discussed below.

8. In this section, the different sections of the Least Squares reports are explained. As mentioned above, for the items to follow click on the **Report** (Clipboard) button and click on the:
 - **Main Report** button to display a report similar to that shown below:



The excerpt from the report above shows the header information and the preprocessing results. The header information consists of (among other things), the input and output file names, the coordinate system, the curvature/refraction setting, maximum iterations, and distance units. During the preprocessing process, multiple angles are reduced to a single angle and multiple slope distances, vertical angles, HI's, and rod heights are reduced to a single horizontal distance and vertical difference. During this process the horizontal angle, horizontal distance and vertical difference spreads are computed. If the spreads exceed the tolerance settings from the Settings dialog box, then a warning message is displayed showing the high and low measurement and the difference between the high and low measurement. Continue to explore this report...

- In the **Unadjusted Observations** section will be data similar to that shown below:

Edit Report Edit : C:\Carlson Projects\SurvNetTut01.RPT

File Edit Settings

Unadjusted Observations

Control Coordinates: 0 Observed Points, 2 Fixed Points, 0 Approx. Points

Station	Northing	Easting	StErr N	StErr E
TR1	5000.0000	5000.0000	FIXED	FIXED
TR100	5000.0000	5820.9905	FIXED	FIXED

Distances: 164 Observations

From Sta	To Sta	Distance	StErr
TR1	TR100	820.99	0.01
TR1	TR2	867.49	0.01
TR1	TR1B	218.43	0.01
TR1	100	20.66	0.01
TR1	101	23.26	0.01
TR1	102	124.11	0.01
TR1	103	64.60	0.01
		.	
		.	

Angles: 163 Observations

BS Sta	OC Sta	FS Sta	Angle	StErr (Sec)
TR100	TR1	TR2	180°13'10"	06
TR100	TR1	TR1B	090°15'52"	08
TR100	TR1	100	017°37'37"	61
TR100	TR1	101	015°29'18"	54
TR100	TR1	102	274°55'59"	12
TR100	TR1	103	279°19'36"	20
			.	
			.	

The excerpt from the *Unadjusted Observations* report above consists of some combination of control X, and Y, horizontal distances, horizontal angles, and azimuth measurements. These measurements consist of a single averaged measurement. For example, if multiple distances were collected between two points during data collection, only the single averaged measurement is used in the Least Squares adjustment. Also, Standard Errors for the measurements are displayed in this section of the report. The Standard Errors are computed from the Standard Error settings in the **Settings** dialog box using error propagation formulas. The Standard Error of an angle that was measured several times would typically be lower than an angle that was measured only once.

NOTE: If the data had been adjusted into NAD 83 coordinates both the ground distances and the grid distances would be displayed. The grid, elevation, and combined factor would also be displayed in this section of the report. Continuing into the report:

- In the **Adjusted Observations** section will be data similar to that shown below:

Edit Report Edit : C:\Carlson Projects\SurvNetTut01.RPT

File Edit Settings

Adjusted Observations

Adjusted Distances

From Sta	To Sta	Distance	Residual	StdRes	StdDev
TR1	TR100	820.99	-0.00	0.1	0.00
TR1	TR2	867.49	0.00	0.2	0.01
TR1	TR1B	218.43	0.00	0.0	0.01
TR2	TR3	804.57	0.00	0.1	0.01
			.		
			.		
			.		
Root Mean Square (RMS)			0.00		

Adjusted Angles

BS Sta	OC Sta	FS Sta	Angle	Residual	StdRes	StdDev(Sec)
TR100	TR1	TR2	180°13'08"	-02	0.3	04
TR100	TR1	TR1B	090°15'52"	00	0.0	08
TR1	TR2	TR3	089°55'07"	01	0.1	04
TR2	TR3	TR4	180°36'18"	-05	0.7	05
			.			
			.			
			.			
Root Mean Square (RMS)			05			

The excerpt from the *Adjusted Observations* report above shows the final adjusted measurements. This section is one of the most important sections to review when analyzing the results of the adjustment. In addition to the adjusted measurement, the Residual is displayed. The Residual is the amount of adjustment applied to the measurement and is computed by subtracting the unadjusted measurement from the adjusted measurement.

The Standard Deviation of the measurement is also displayed. Ideally, the computed Standard Deviation and Residual and the Standard Error displayed in the unadjusted measurement would all be of similar magnitude. The Standard Residual is a measure of the similarity of the Residual to the *a-priori* Standard Error. The Standard Residual is the measurements Residual divided by the Standard Error displayed in the unadjusted measurement section. A Standard Residual greater than 2 is marked with an "*". A high Standard Residual may be an indication of a blunder. If there are consistently a lot of high Standard Residuals it may indicate that the original Standard Errors set in the **Settings** dialog box were not realistic.

- If the **Enable sideshots for relative error ellipses** option is not set in the Adjustment section of the **Settings**, sideshots are computed separately after the adjustment is completed.

Edit Report Edit : C:\Carlson Projects\SurvNetTut01.RPT

File Edit Settings

Sideshots

From Sta	To Sta	Bearing	Distance	Northing	Easting	StDev N	StDev E
TR1	100	S 72°22'23"E	20.66	4993.7449	5019.6865	0.0064	0.0085
TR1	101	S 74°30'42"E	23.26	4993.7893	5022.4128	0.0063	0.0085
TR1	102	N 04°55'59"E	124.11	5123.6541	5010.6728	0.0088	0.0072
TR1	103	N 09°19'36"E	64.60	5063.7459	5010.4693	0.0087	0.0065
TR1	104	N 57°42'41"E	118.78	5063.4480	5100.4090	0.0075	0.0083
TR1	105	N 65°59'21"W	152.17	5061.9176	4861.0016	0.0081	0.0087
TR1	106	N 68°32'48"W	169.16	5061.8696	4842.5590	0.0083	0.0088

If the project had valid elevation benchmarks and measured HI's and rod heights the project could have been defined to adjust elevations. When using the **2D/1D Least Squares model**, the horizontal and the vertical adjustments are separate Least Squares adjustment processes. As long as there are redundant vertical measurements, the vertical component of the network can also be reduced and adjusted using Least Squares. In the vertical adjustment, benchmarks are held fixed. Click the **Exit** (Doorway) button to dismiss this report.

- **Output Report** button to display a report similar to that shown below:

Edit Output Edit : C:\Carlson Projects\SurvNetTut01.OUT

File Edit Settings

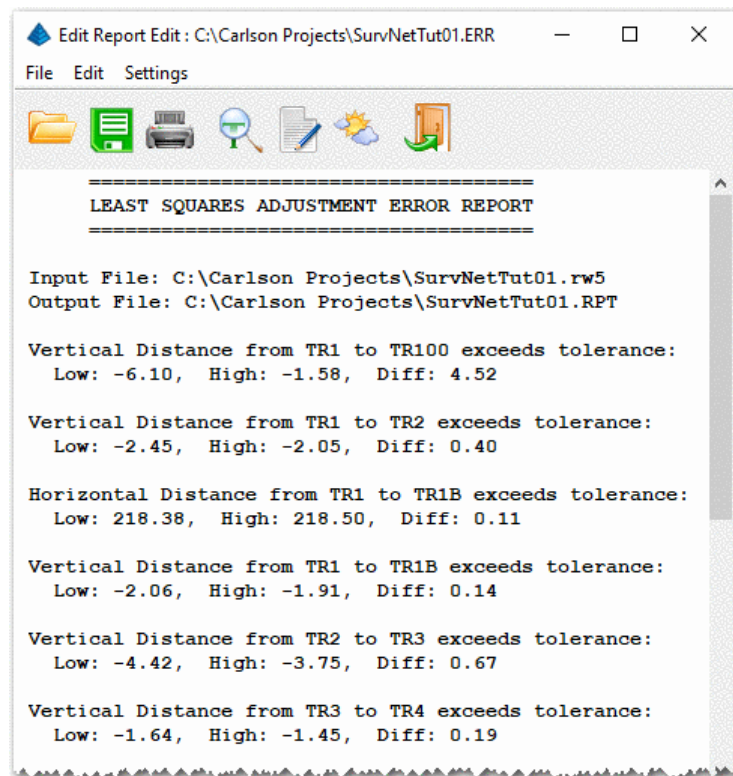
ADJUSTED COORDINATES

Input File: C:\Carlson Projects\SurvNetTut01.rw5
 Output File: C:\Carlson Projects\SurvNetTut01.RPT
 Curvature, refraction correction: ON
 Local Coordinate System, Scale Factor: 1.000000
 Horizontal Units: Intl. Feet

Pt.No.	Northing	Easting	Elevation	Description
100	4993.7449	5019.6865	105.4	-41
101	4993.7893	5022.4128	105.5	41
102	5123.6541	5010.6728	108.9	-41
103	5063.7459	5010.4693	104.9	41
104	5063.4480	5100.4090	105.8	41
105	5061.9176	4861.0016	105.4	-41
106	5061.8696	4842.5590	106.9	41
125	5065.1572	4270.2362	91.8	-41
126	5065.7646	4190.8053	91.1	41
127	5126.7771	4190.8334	91.4	41
128	5066.8710	4023.8238	88.8	-41
129	5066.8058	4056.0628	89.2	41
130	5007.0226	4128.9905	89.8	232

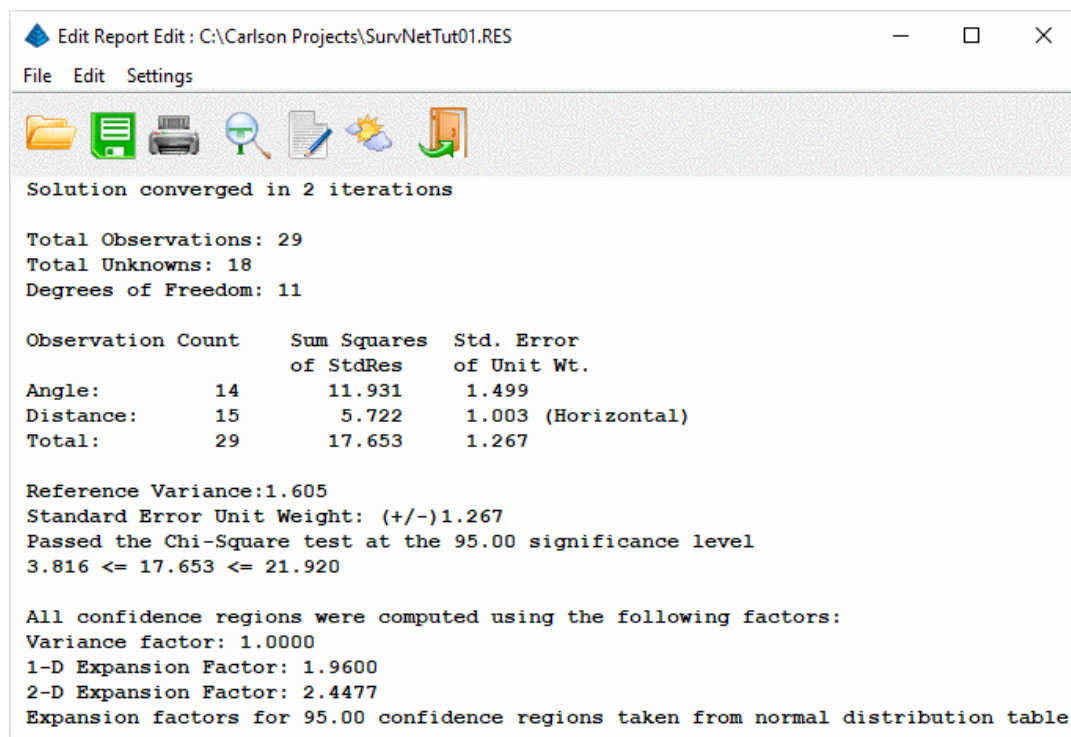
This section of the report shows the final adjusted coordinates. Click the **Exit** (Doorway) button to dismiss this report.

- **Error Report** button to display a report similar to that shown below:



This section of the report is one of the simplest and effective tools in finding blunders. Time spent learning how this function works will be well spent. If the project is not converging due to an unknown blunder in the raw data, this tool is one of the most effective tools in finding the blunder. Many blunders are due to point numbering errors during data collections, and the **K-matrix** analysis and **Point Proximity** search are great tools for finding these types of blunders. Click the **Exit** (Doorway) button to dismiss this report.

- **Results Report** button to display a report similar to that shown below:



This report displays some statistical measures of the adjustment including the number of iterations needed for the solution to converge, the degrees of freedom of the network, the reference variance, the standard error of unit weight, and the results of a Chi-square test.

The degree of freedom is an indication of how many redundant measurements are in the survey. Degree of freedom is defined as the number of measurements in excess of the number of measurements necessary to solve the network.

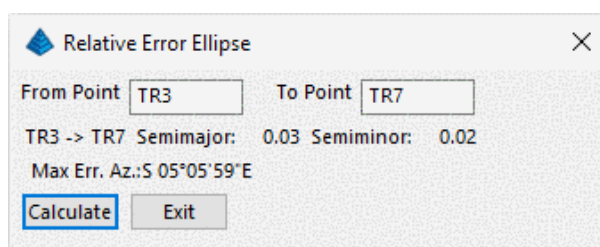
The standard error of unit weight relates to the overall adjustment and not to an individual measurement. A value of one indicates that the results of the adjustment are consistent with the prior standard errors. The reference variance is the standard error of unit weight squared.

The Chi-square test is a test of the "goodness" of fit of the adjustment. It is not an absolute test of the accuracy of the survey. The *a-priori* standard errors which are defined in the project settings dialog box or with the SE record in the raw data file are used to determine the weights of the measurements. These standard errors can also be looked at as an estimate of how accurately the measurements were made. The Chi-square test merely tests whether the results of the adjusted measurements are consistent with the *a-priori* standard errors. Notice that if you change the project Standard Errors and then reprocess the survey, the results of the Chi-square test change, even though the measurements themselves did not change.

In our example, the Chi-square test passed at the 95% significance level. Had our example failed the Chi-square test on the low end (e.g. a value < 3.816), indications would be that our data is actually better than expected compared to our *a-priori* standard errors. If we were to increase the Pointing and Reading Standard Error in the Settings menu by 10-20 seconds, we would probably fail the Chi-square. Also notice that if you change the Standard Errors by only 5-10 seconds and reprocess the data the final coordinates will not change significantly.

This is the final step in the adjustment. The final adjusted coordinates are now stored in the current project point database and can now be used for mapping and design.

9. Relative Error Ellipses are a statistical measure of the expected error between two points. *Regular* Error Ellipses are a measure of the absolute error of a single point. Some survey accuracy standards such as the ALTA standards state the maximum allowable error between any two points in a survey. Relative Error Ellipses can give you this information. Press the **Relative Error Ellipse** (Ellipse between Two Points) button to display a dialog box similar to that shown below:

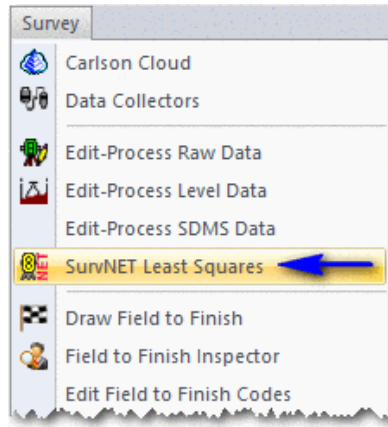


Enter **TR3** and **TR7** in the **From Pt.** and **To Pt.** fields, respectively, and click the **Calculate** button. At the 95% confidence level there should only be around 0.03 feet of error between points TR3 and TR7. If you need to compute Relative Error Ellipses for sideshots, make sure the **Enable Sideshots for Error Ellipse** toggle is set in the Adjustment tab of the Settings dialog box. Click the **Exit** button to dismiss the *Relative Error Ellipse* dialog box. Click the **Exit** (Doorway) button to dismiss the *SurvNET* docked dialog box.

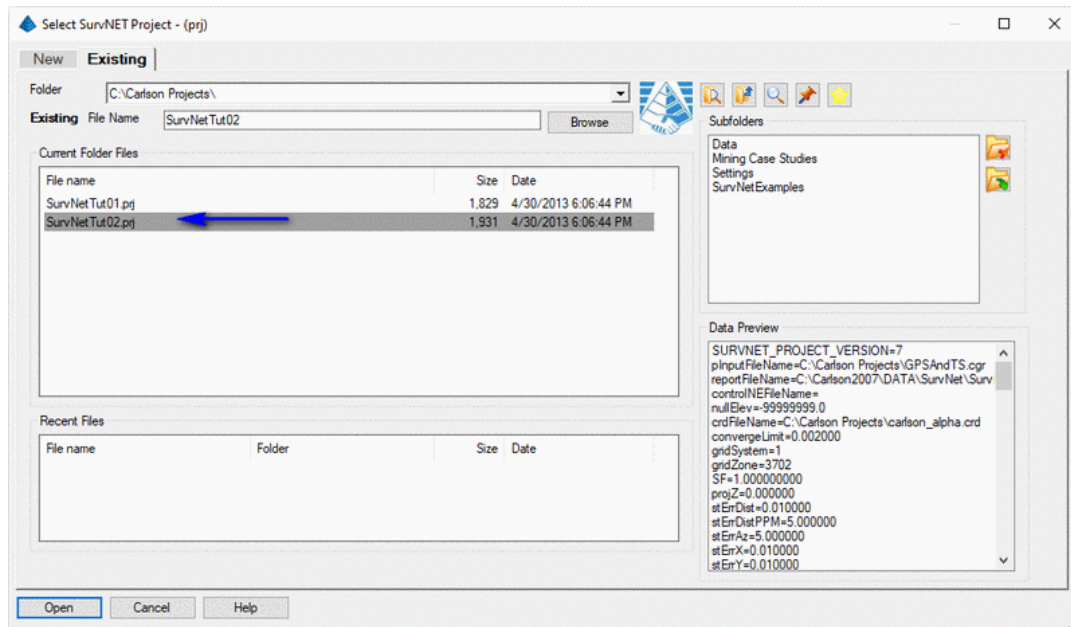
Lesson Two - Processing a 3D Network With Both Total Station Data and GPS Vectors

In this lesson we will process a project that contains both total station measurements and GPS vectors.

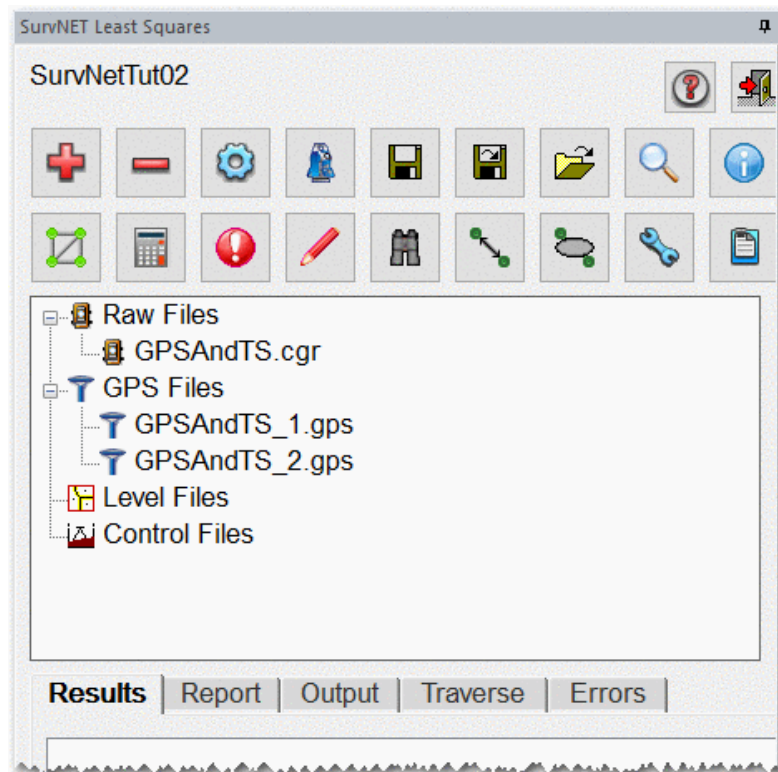
1. Issue the Survey – SurvNET command as illustrated below:



If the previous SurvNET project re-appears, click the **Load/New** (Open Folder) button. This, in turn opens the following dialog box:

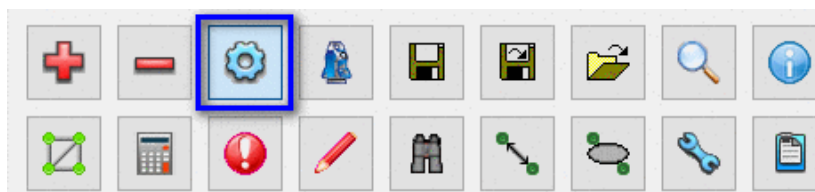


Select the **SurvNetTut02.prj** project and click **Open**. This will open the default SurvNET *project-tree* docked dialog box interface as shown below where we will process the contents of the **GPSAndTS.cgr** "raw" file and a pair of GPS vector files (**GPSAndTS_1.gps** and **GPSAndTS_2.gps**):

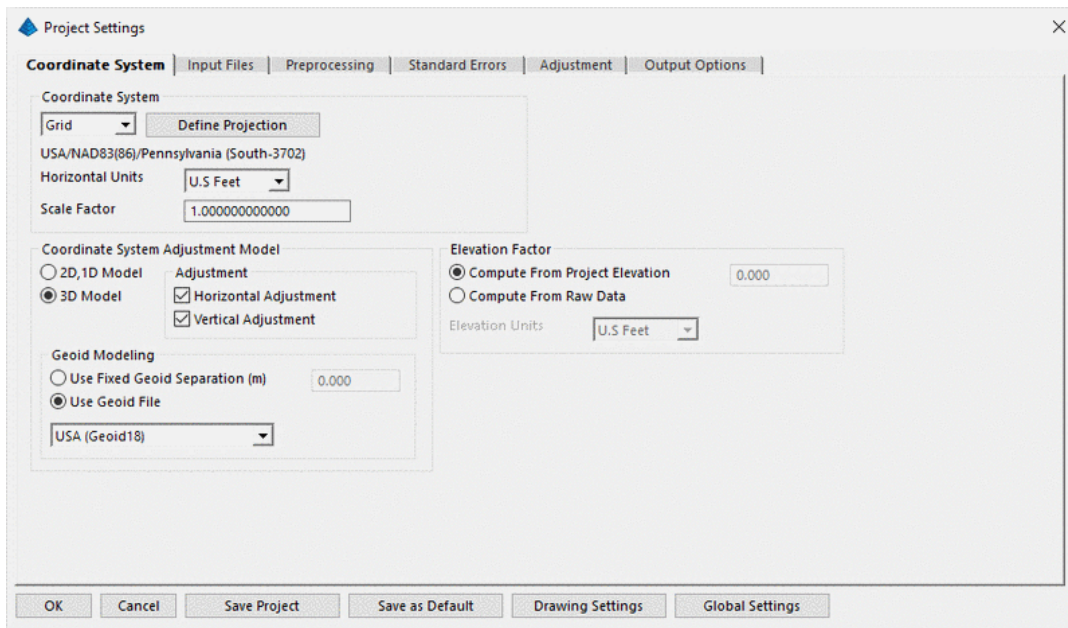


NOTE: The sample tutorial project has the input raw file in the default folder of **C:\Carlson Projects**. If you have a different data directory, then set the correct data file by highlighting the default file, pick Delete and then pick Add and select **GPSAndTS.cgr** (C&G format raw file) from your data folder. Do the same for the GPS Vector files of **GPSAndTS_1.gps** and **GPSAndTS_2.gps**.

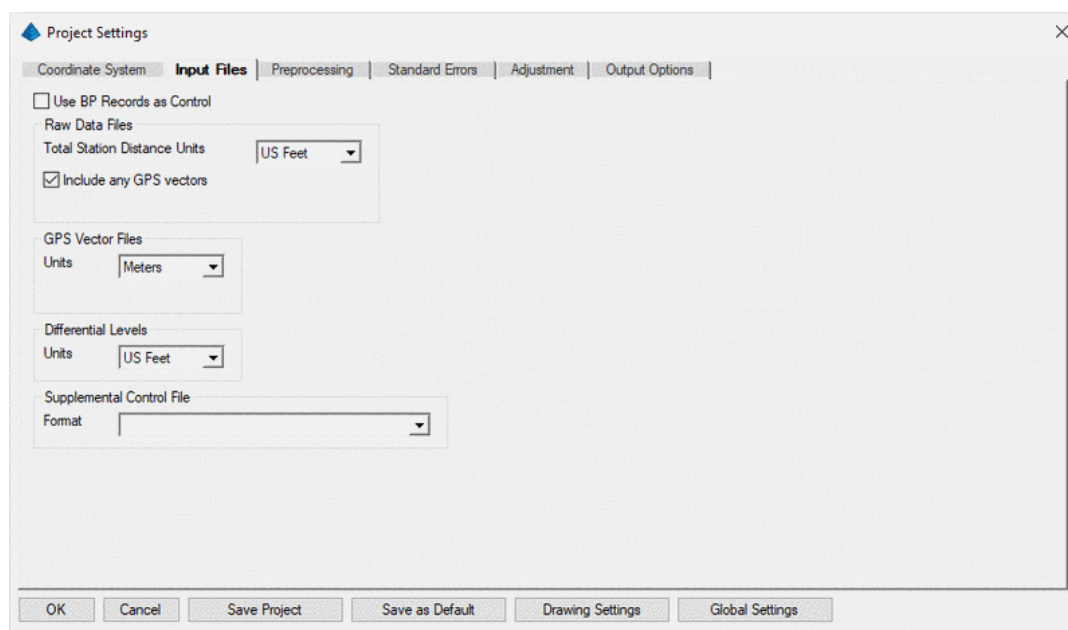
2. Let's review the project settings. Click the Settings (Gear) button as shown below:



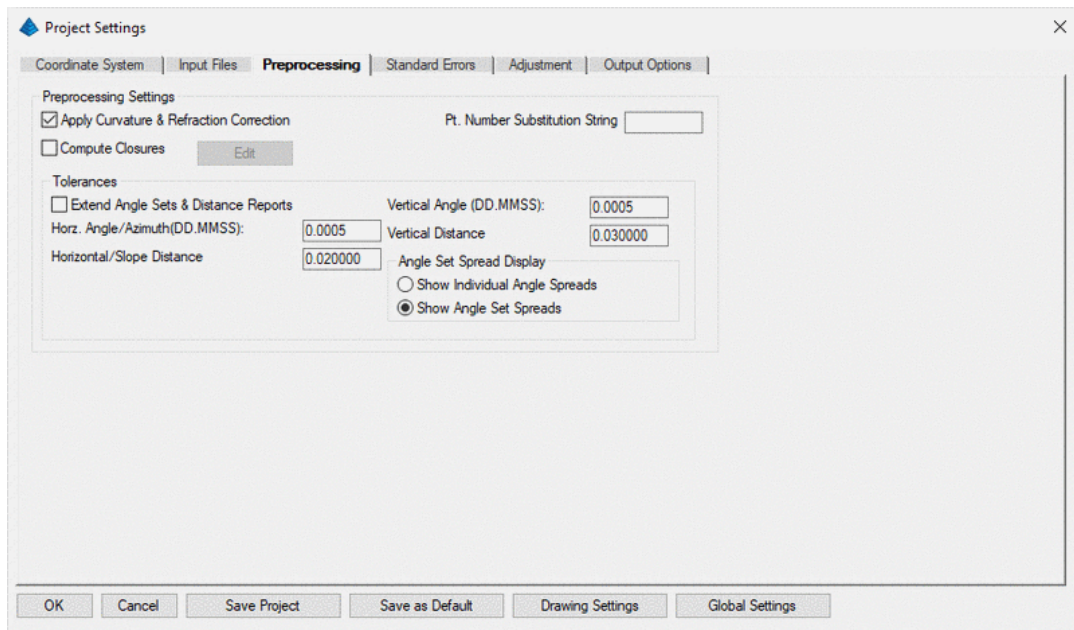
to display the dialog box similar to that shown below:



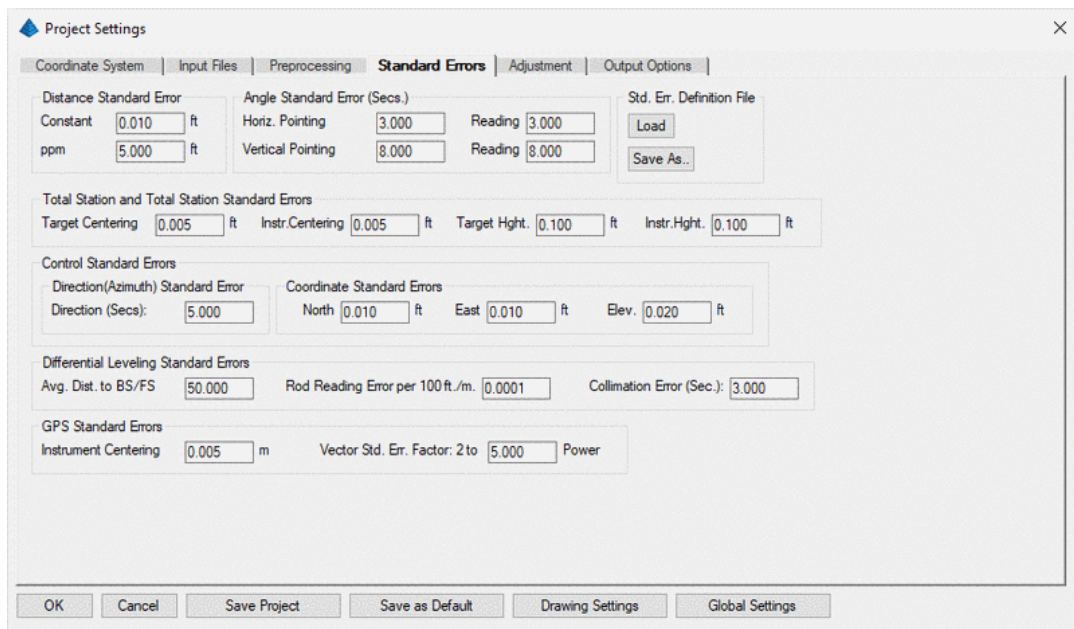
- Settings - Coordinate System: In order process GPS vectors, a coordinate system must be set (*e.g.* **SPC 1983**) with the appropriate State Plane zone. Further, the **Coordinate System Adjustment Model** must be set to the **3D Model**. With the 3D model, horizontal units and vertical units must be the same in regards to output and total station raw data. Geoid modeling may or may not be important depending on the extent of the project and the accuracies required. The most accurate results are typically obtained by using a **Geoid File**. Set the values as shown above.
- Activate the Input Files tab. Notice that the units need to be specified for both the GPS vector data and the total station raw data. Typically, but not always, GPS vectors are in meters while the total station and the final output may need to be in feet. Also make sure that the correct GPS vector format is correct. Some GPS formats are binary and cannot be edited easily. Sometimes it is needed to edit the GPS vectors, usually in terms of point numbers.



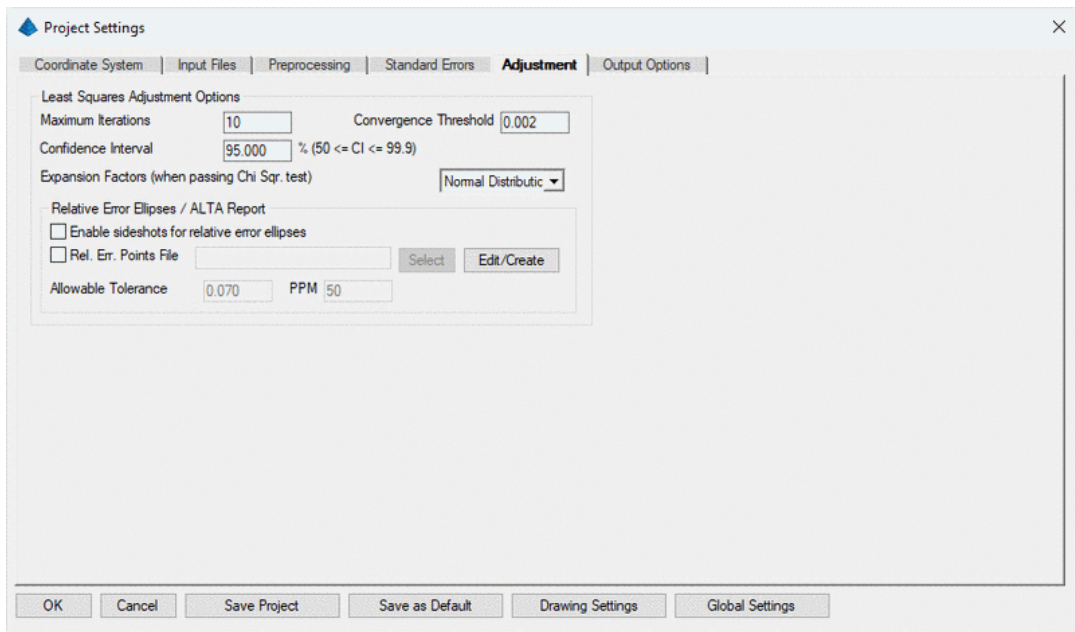
- Activate the Preprocessing tab. Though this tutorial does not cover the topic, it is from this screen that you would define the traverse file needed to compute either GPS Loop closures or total station traverse closures.



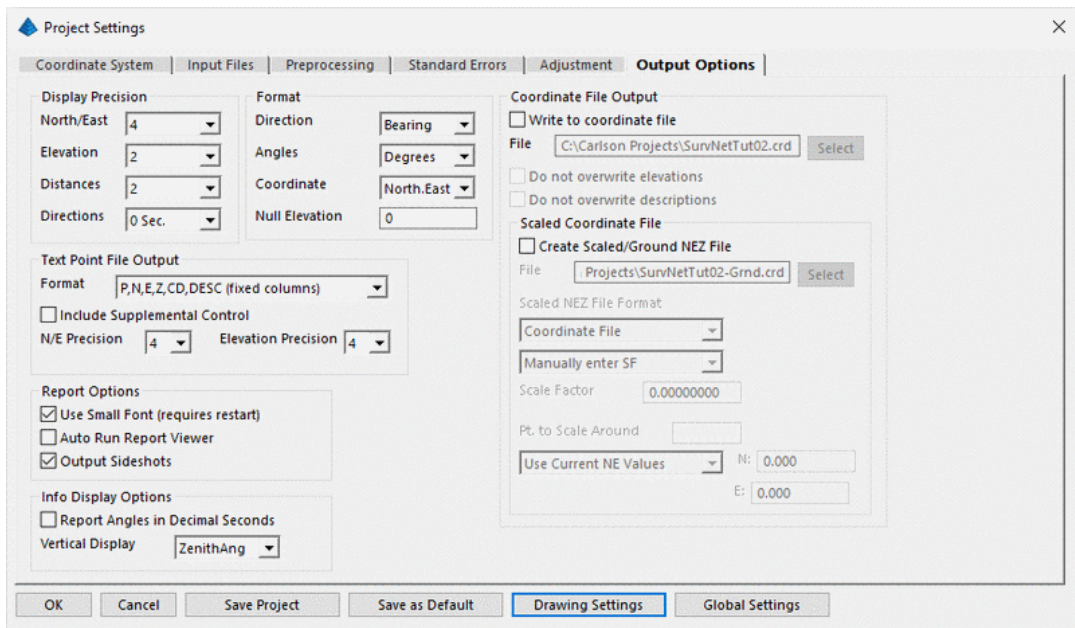
- Activate the Standard Errors tab. Notice the standard error settings related to GPS. The GPS instrument centering error can be defined. The vector standard error is a factor that can be used to increase the standard errors as defined in the GPS vector files.



- Activate the Adjustment tab. None of the settings in this screen are specific to processing GPS vectors:

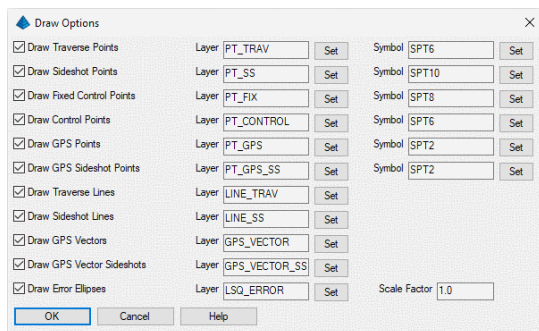


- Activate the Output Options tab. None of the settings in this screen are specific to processing GPS vectors.

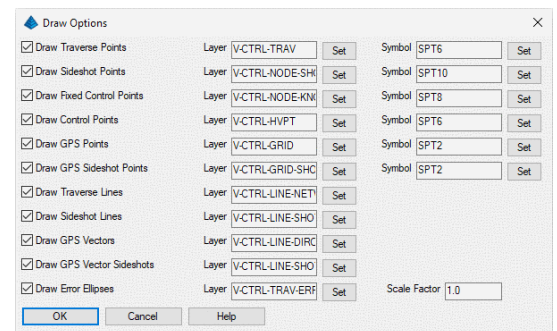


- Click the Drawing Settings button to display a dialog box similar to those shown below:

Carlson SurvNET Defaults



Suggested National CAD Standards Defaults

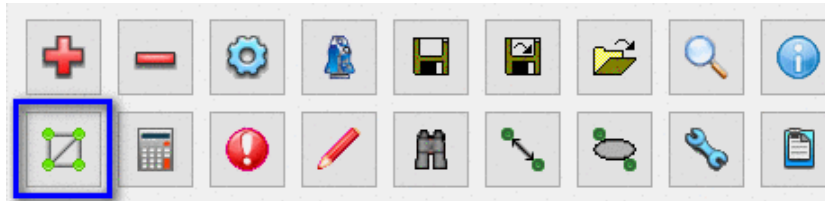


Suggested Drawing Settings

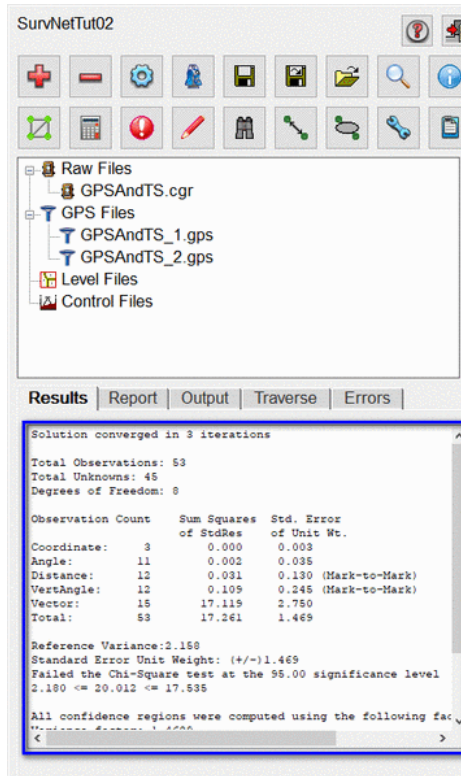
Set the desired value(s) as illustrated above and click **OK** when ready. We'll use this information later in

the Draw Results discussion.

3. Click the **Network Adjustment** (Double Triangle) button as shown below:



The Least Squares adjustment is performed. The project should process and converge and the following windows should be displayed:



Let's review sections of the report that are unique to the processing of GPS vectors and the 3D model.

4. Click the **Reports** (Clipboard) button and select the **Main Report**. Scroll through the report to the section similar to that shown below:

◆ Edit Report Edit : C:\Carlson Projects\SurvNetTut02.RPT
File Edit Settings

3-DIMENSIONAL ADJUSTMENT REPORT

Unadjusted Observations

Control Coordinates:	1 Observed Points,	0 Fixed Points,	0 Approx. Points			
Sta.	Latitude	Longitude	Z (Ellip.)	StErr N:	StErr E:	StErr Z:NE
200	40°18'38.58302"N	76°12'53.64149"W	744.12	0.0100	0.0100	0.0200

Grid XYZ	N:	E:	Z (Geoid):	StErr N:	StErr E:	StErr Z:
200	359769.1900	2396585.1700	858.21	0.0100	0.0100	0.0200

Geocentric XYZ	X:	Y:	Z:	StErr X(m):	StErr Y(m):	StErr Z(m):
200	1160578.5446	-4730339.6491	4104501.5919	0.0032	0.0050	0.0046

Notice that now that we are working with a specific datum (as opposed to an assumed coordinate system) the latitude/longitude, state plane coordinates and geocentric coordinates are all displayed. Continuing into the **Main Report** is the following:

◆ Edit Report Edit : C:\Carlson Projects\SurvNetTut02.RPT
File Edit Settings

Mark to Mark Slope Distances: 192 Observations

From Sta.	To Sta.	Dist.	StErr
200	201	352.78	0.02
200	202	344.09	0.02
200	203	170.89	0.01
200	204	149.60	0.01
.	.	.	.

Mark to Mark Vertical Angles: 192 Observations

From Sta.	To Sta.	Vertical Ang.	StErr (Sec.)
200	201	097°58'35"	11
200	202	077°56'37"	11
200	203	066°54'34"	11
200	204	079°12'50"	11
.	.	.	.

Horizontal Angles: 191 Observations

BS Sta.	Occ. Sta.	FS Sta.	Angle	StErr (Sec.)
201	200	202	131°40'05"	08
201	200	203	243°43'37"	11
201	200	204	176°40'38"	12
.

GPS Vectors: 17 Observations (Meters)
Multiplication factor applied to Variance and Covariance values: 32 (2 to 5 power)

From Sta.	Delta X	Variance Delta X	Covariance XY	Bearing
To Sta.	Delta Y	Variance Delta Y	Covariance XZ	H.Distance
	Delta Z	Variance Delta Z	Covariance YZ	V.Distance
BAS1	-219.104	1.038	0.007998	N 42°21'42"W
PIN9	109.035	1.047	-0.1474	277.25
	134.526	1.048	-0.0737	-33.53
FN11	86.739	0.0001678	-1.463E-06	N 25°22'14"E
BAS1	273.779	0.001389	6.92E-06	348.84
	205.297	0.001205	-0.0008282	-54.21

In the above *Unadjusted Observations* section of the report, notice that distances have been converted to mark to mark distances. Note that vertical angles are now treated as measurements in the 3D model. And lastly, notice that the GPS vectors are also displayed. The GPS vectors are displayed as Delta X,Y,Z in the geocentric coordinate system. Continuing further into the report is:

◆ Edit Report Edit : C:\Carlson Projects\SurvNetTut02.RPT

File Edit Settings

Adjusted Geographic Coordinates

Sta.	Latitude	Longitude	Z (Ellip.)	Conv. Ang.	Grid Factor	Z Factor	Combined Factor
200	40°18'38.58302"N	76°12'53.64149"W	744.12	000°59'45"	0.99996247	0.99996441	0.99992688
201	40°18'39.11637"N	76°12'58.09697"W	695.17	000°59'42"	0.99996246	0.99996675	0.99992921
202	40°18'40.69556"N	76°12'50.28719"W	815.99	000°59'47"	0.99996244	0.99996097	0.99992342
203	40°18'37.10066"N	76°12'53.03522"W	811.14	000°59'46"	0.99996249	0.99996120	0.99992369

Average Combined Scale Factor: 0.99993206

Adjusted Grid Coordinates, (US Feet)

Sta.	N:	E:	Z (Geoid):	StErr N:	StErr E:	StErr Z:	DN	DE	DZ
200	359769.1900	2396585.1700	858.21	0.0158	0.0158	0.0316	-0.0000	-0.0000	-0.0000
201	359817.1548	2396239.1359	809.26	0.0318	0.0398	0.0435			
202	359987.4480	2396841.2580	930.08	0.0372	0.0391	0.0434			
203	359620.0255	2396634.7363	925.23	0.0251	0.0269	0.0356			

In the above *Adjusted Coordinates* section of the report, notice that the grid, elevation, and combined factor are displayed with the adjusted geographic coordinates.

◆ Edit Report Edit : C:\Carlson Projects\SurvNetTut02.RPT

File Edit Settings

Adjusted Observations

Adjusted Mark to Mark Distances

From Sta.	To Sta.	Distance	Residual	StdRes.	StdDev
200	201	352.78	-0.00	0.0	0.03
200	202	344.09	-0.00	0.0	0.03

Root Mean Square (RMS) 0.00

Adjusted Horizontal Angles

BS Sta.	Occ. Sta.	FS Sta.	Angle	Residual	StdRes	StdDev(Sec.)
201	200	202	131°40'05"	-00	0.0	13
201	200	203	243°43'37"	-00	0.0	17

Root Mean Square (RMS) 00

Adjusted Mark to Mark Vertical Angles

From Sta.	To Sta.	Vertical Ang.	Residual	StdRes	StdDev(Sec.)
200	201	097°58'34"	01	0.1	17
200	202	077°56'37"	00	0.0	18

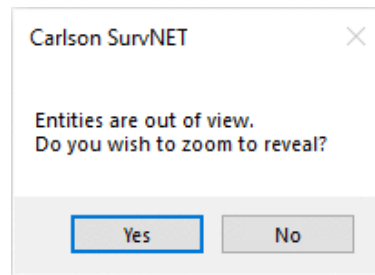
Root Mean Square (RMS) 01

GPS Vectors: 17 Observations (Meters)

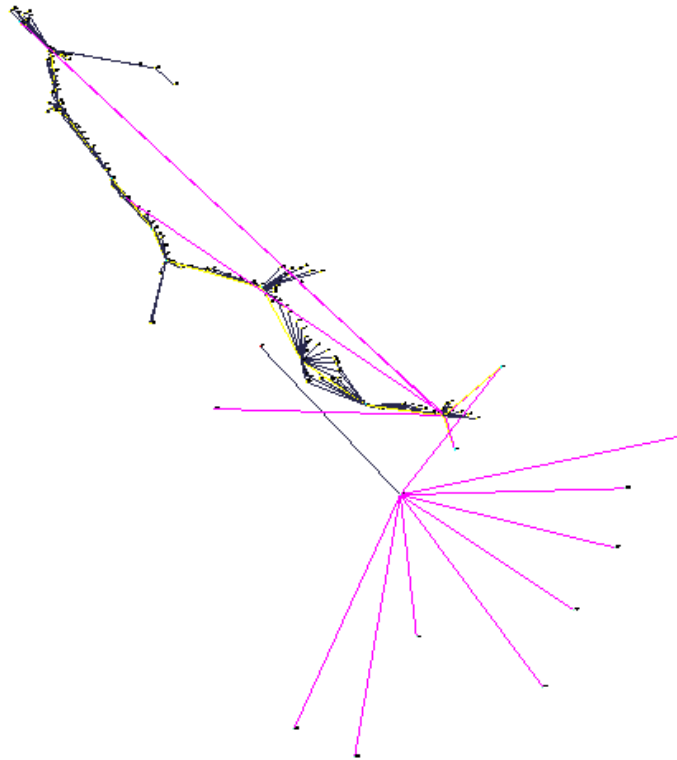
From Sta.	To Sta.	Delta X	Residual	StdRes	StdDev
202	BAS1	-117.4905	0.0000	0.0	0.0201
BAS1	BAS2	-106.6426	0.0000	0.0	0.0373
BAS1	202	-160.2261	0.0000	0.0	0.0465
BAS2	202	68.4197	1.1067	1.7	0.5616
202	BAS1	42.5571	1.1896	1.9	0.6975
202	BAS2	63.2396	-0.2167	0.3	0.7548

In the above *Adjusted Measurements* section, the adjusted measurements are shown along with their residuals, standard residuals, and standard deviation. Click the **Exit** (Doorway) button to dismiss this report.

- Let's look at the visual representation of the processed project. Click on the **Draw** (Pencil) button. You may encounter the following prompt:

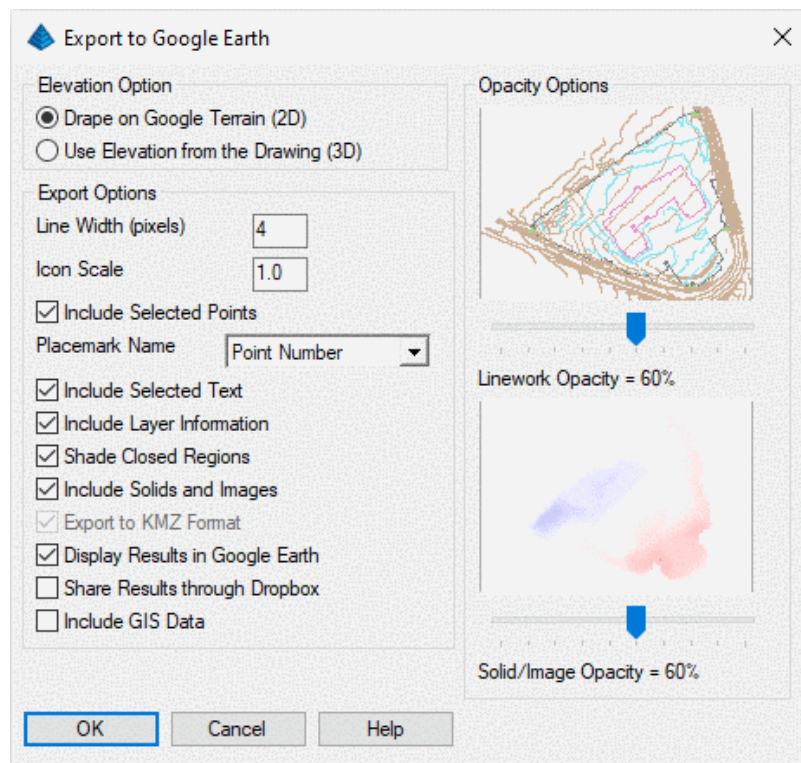


Click **Yes** to obtain graphical results similar to that shown below (National CAD Standards layer configuration shown):

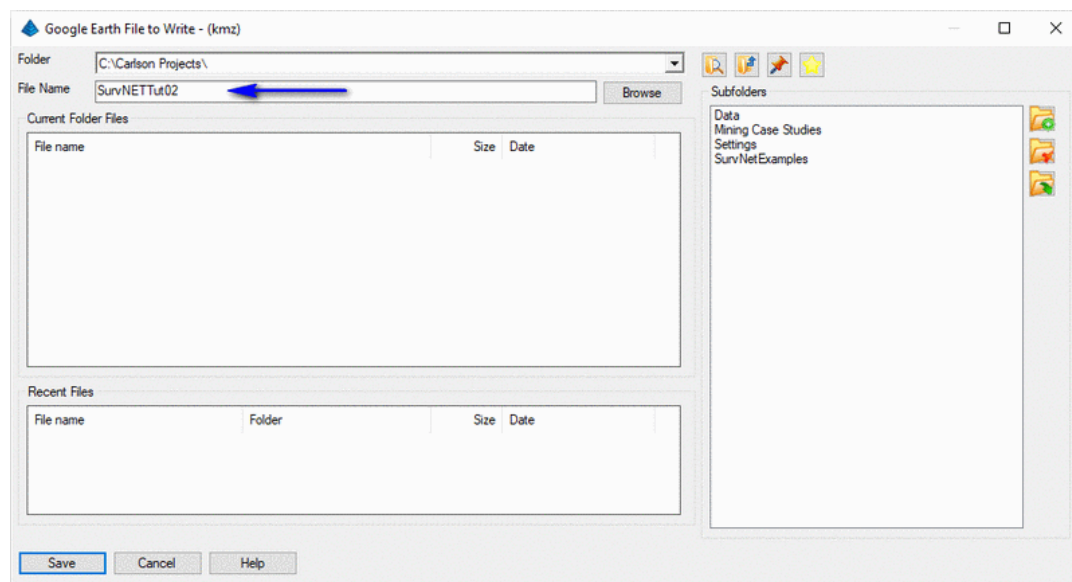


Click the **Exit** (Doorway) button to dismiss the SurvNET dialog box.

6. *Optional:* Lastly, for computers that have Google Earth™ Pro on desktop installed, let's see how the site looks when overlaid on this application (or any other application which can display Keyhole Markup Language (KML) files). As you may recall, we set a desired Coordinate System during the initial drawing setup and this system also matches the Coordinate System as specified in the **Settings** for this SurvNET project. Issue the File – Export – Google Earth File command to display the dialog box below:



Review and set the values as suggested above and click **OK** when ready to display a dialog box similar to that shown below:

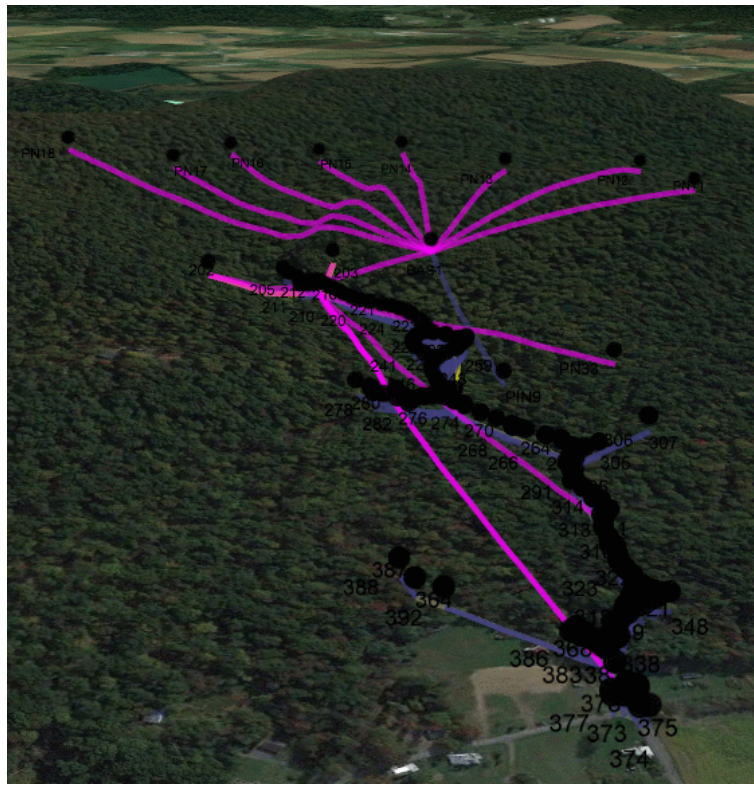


Provide the file name specified above and click the **Save** button when ready. When prompted:

Select points, polylines, text, solids, images, lines and arcs to write.

[FILter]/<Select entities>: type **all** and press Enter twice

The results are overlaid on Google Earth as illustrated below:



This completes the tutorial: SurvNET.

Contouring, DTM and Design

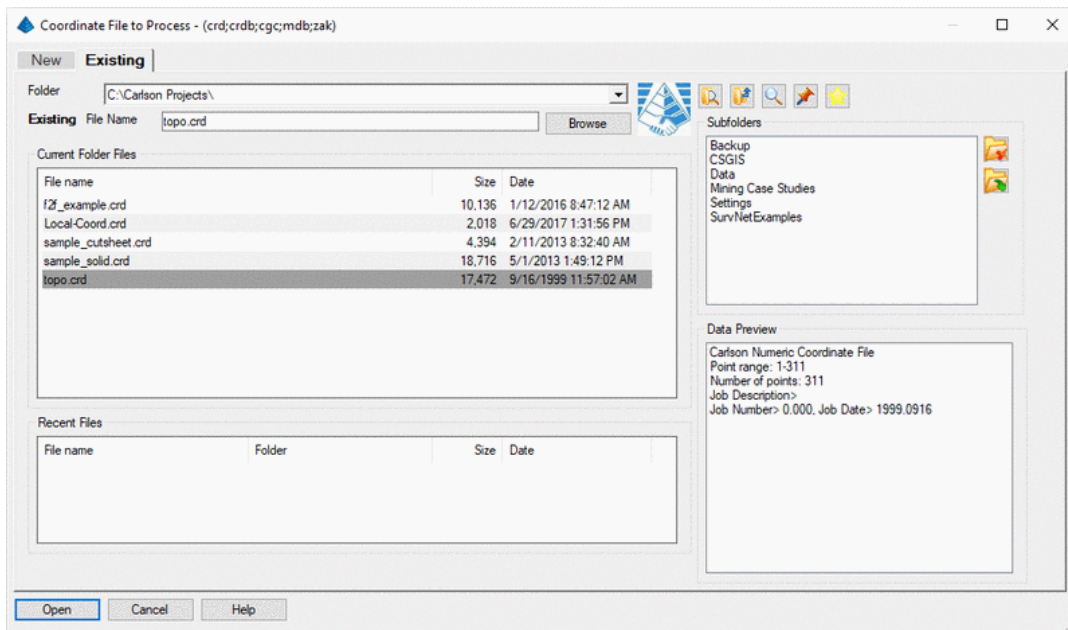
This is the easiest of the tutorials, and could be completed in as little as five minutes. If that is all the time you have, and you have purchased the Civil Design program, do this one first!

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **New Drawing**.
 - If you get the Startup Wizard dialog box, click the Exit button.
 - If you are taken directly into CAD, continue as is.

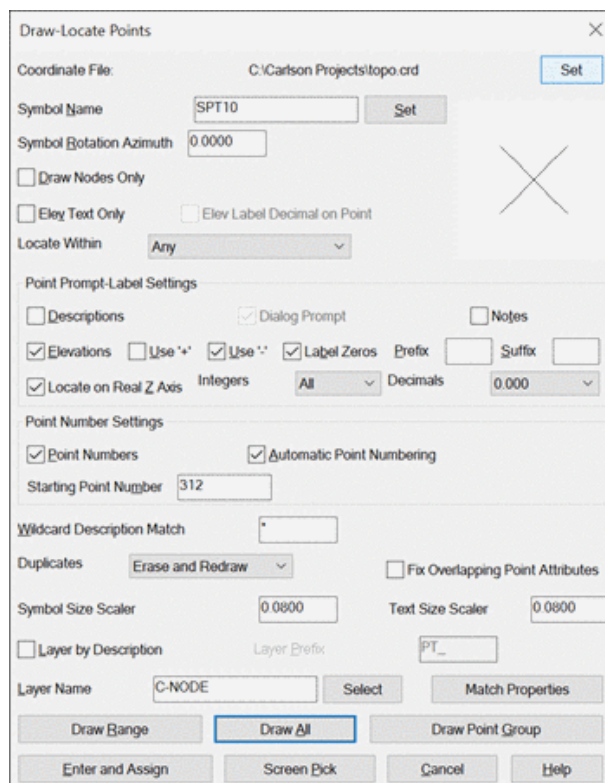
Ensure you are in the Civil Design program by issuing the Settings – Carlson Menu – **Civil Menu** command.

NOTE: Many of the routines below can also be accomplished with the Carlson Survey and/or Construction modules but our focus herein will presume use of the Civil module.

2. Once in Carlson, establish a coordinate (*.crd) file to work with via the Points – Set CoORDinate File command. You will then be asked to choose the coordinate file that you want to use. Select **Topo.crd** as shown below and click Open:

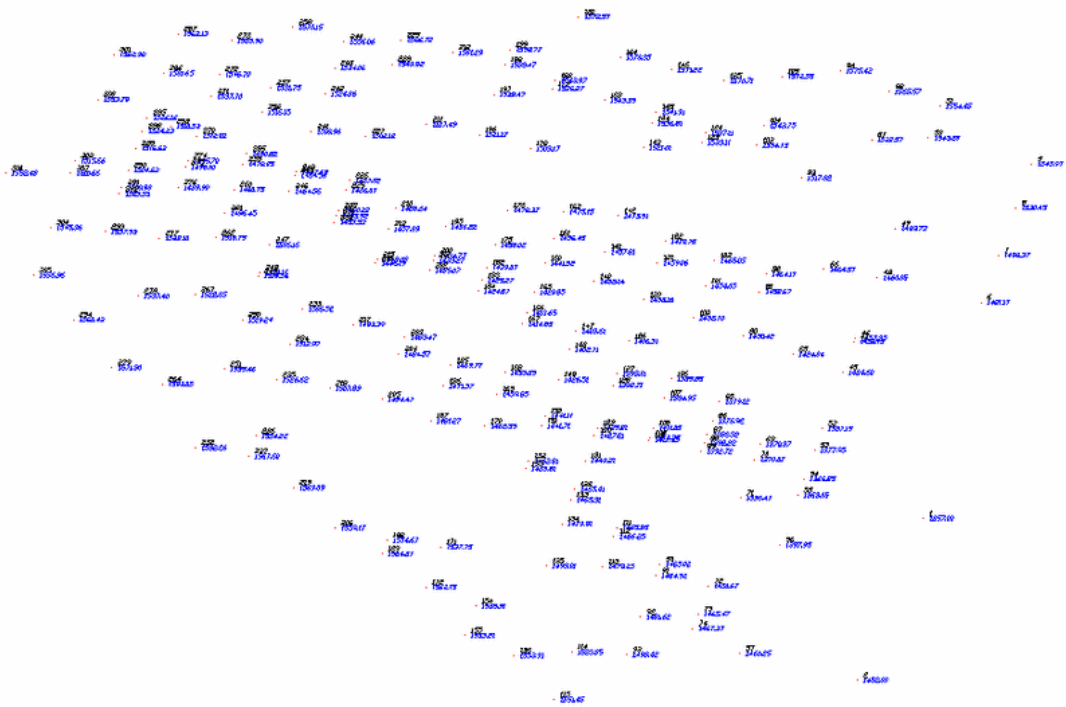


Next, issue the Points – Draw-Locate Points command which displays the dialog box similar to that shown below:

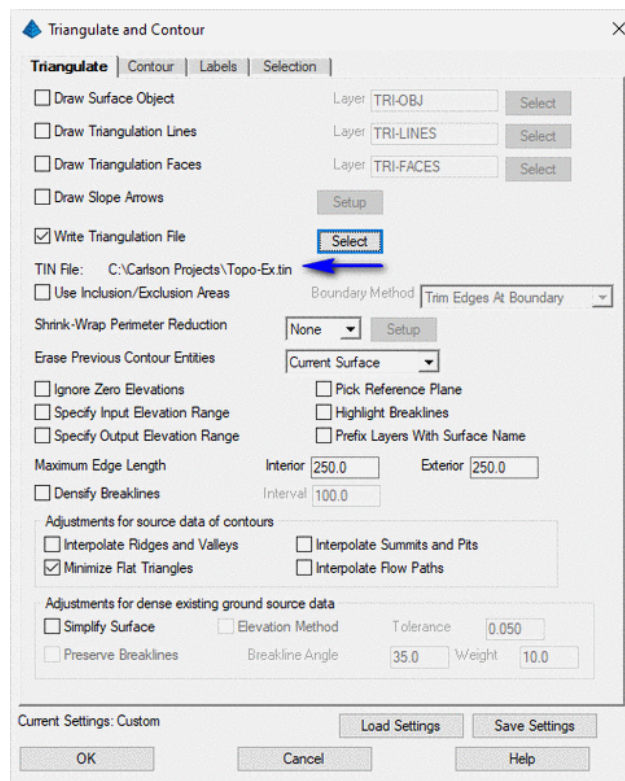


Choose Symbol "SPT0" by clicking **Set** to the right of Symbol Name then picking Symbol SPT0 from the options that appear. Establish the other settings as shown above and then click the **Draw All** button.

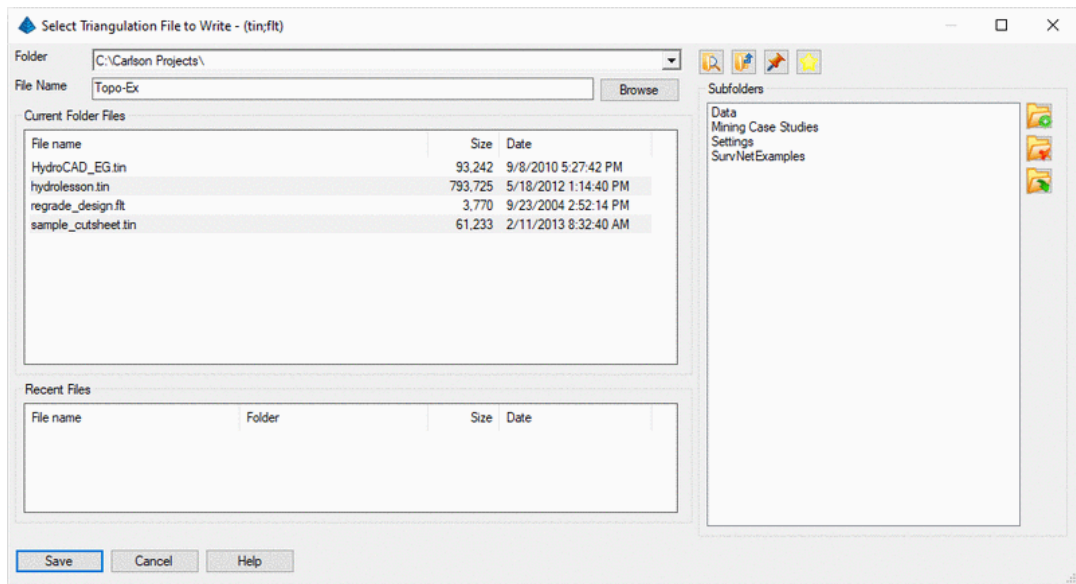
The points immediately plot on the screen, and the program should zoom to the extents of the points. If you don't see the points, issue the View – **Extents** command. Your drawing should resemble that shown below (drawn with a horizontal plot scale factor of **50** as established in Settings – Drawing Setup):



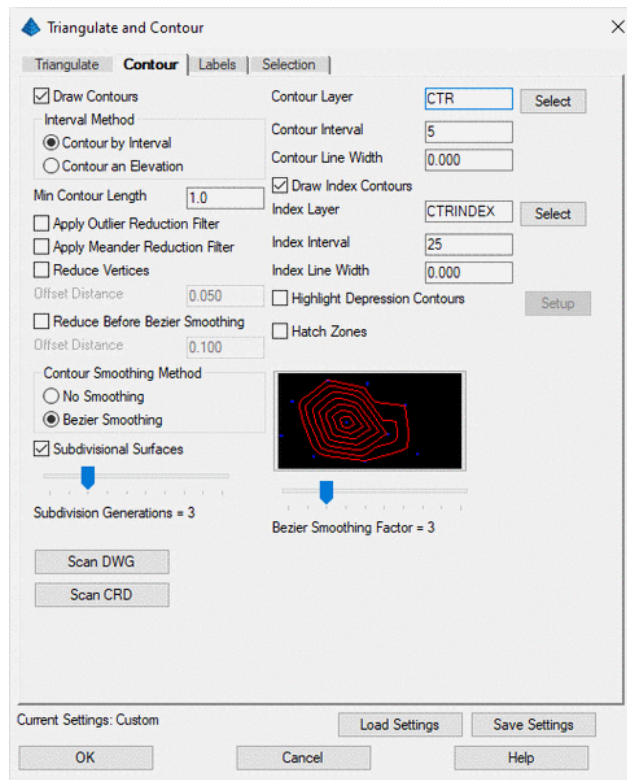
3. Issue the Surface – Triangulate & Contour command. The following dialog box (with several "tabs") will appear which you should fill out as discussed below:



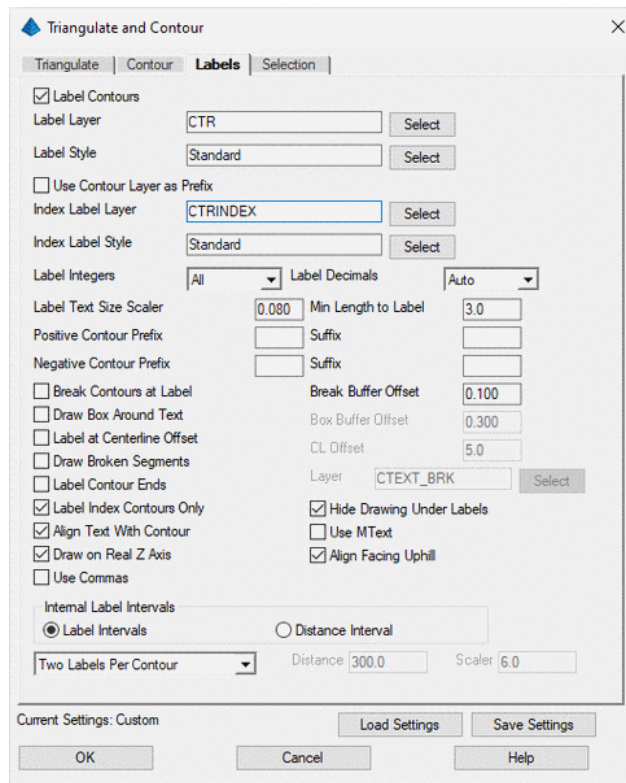
- In the Triangulate tab (as shown above), enable the **Write Triangulation File** toggle and click its **Select** button to set the name of a triangulation (*.tin) file as illustrated below. Click the **Save** button on this dialog box so that the results of the triangulation can be re-used for future purposes. Set the **Maximum Edge Length** values to 250 in all cases. If the goal of the field crew was never to shoot points further apart than 100 feet, then certainly triangulation lengths over 250 feet (or less) can be ignored. Set the **Shrink-Wrap Perimeter Reduction** to None (again, as illustrated above):



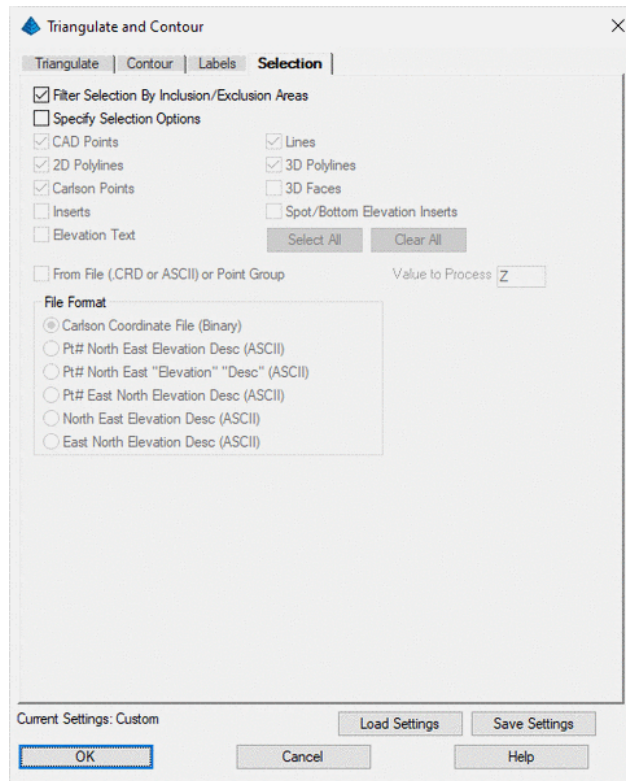
- In the Contour tab, as shown below, set the **Contour Interval** to 5, turn on **Draw Index Contours** and set its interval to 25 (index intervals are most often 5 times the standard contour interval):



- In the Labels tab, as shown below, enable the **Label Index Contours Only** toggle. Also set the Layers for the Intermediate and Index contour labels to be the same as that for the contours themselves:



- In the Selection tab, as shown below, clear the **Specify Selection Options** toggle:

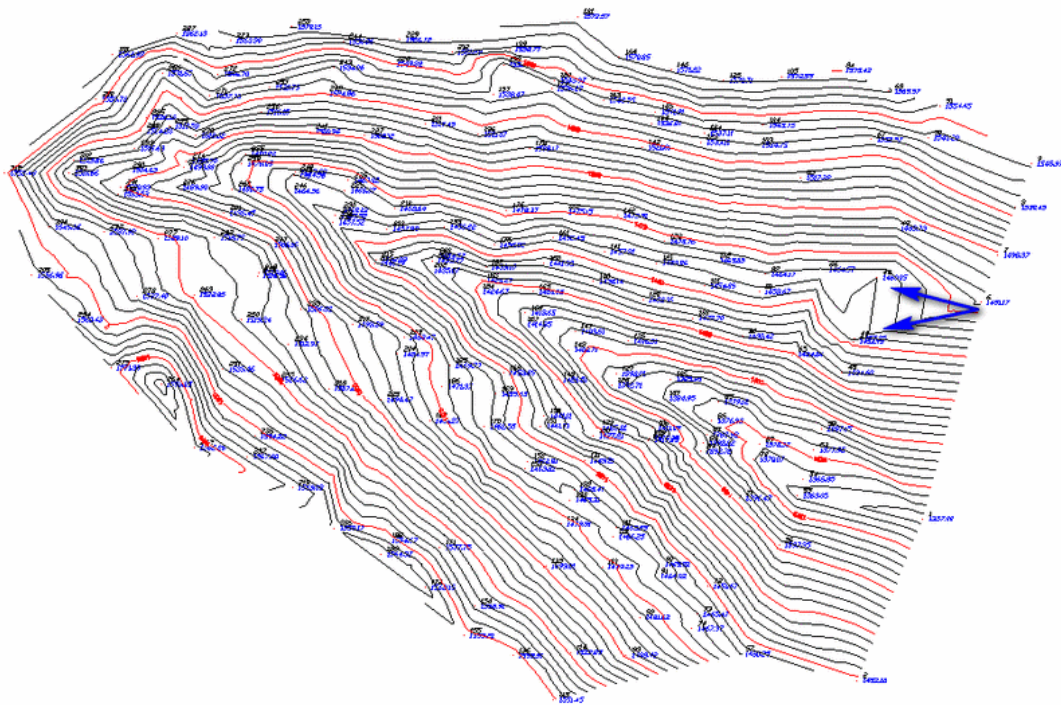


Click the **OK** button from any tab. When prompted:

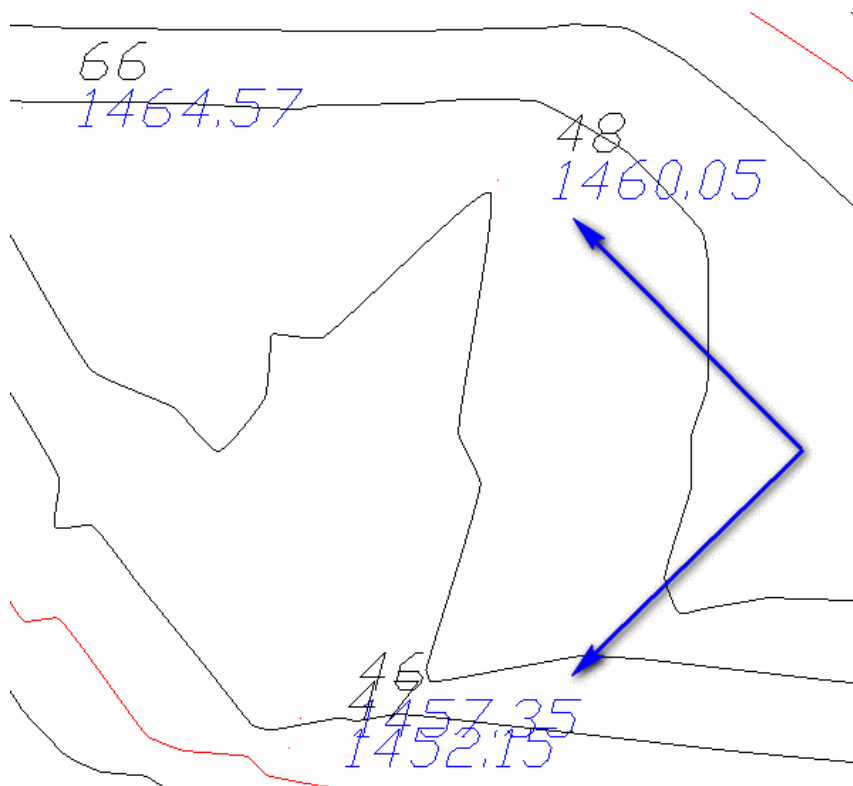
Select points and breaklines to Triangulate.

[FILter]/<Select entities>: type ALL and press Enter twice

A "TIN" file is written and the contours are drawn. Here is the plot so far:

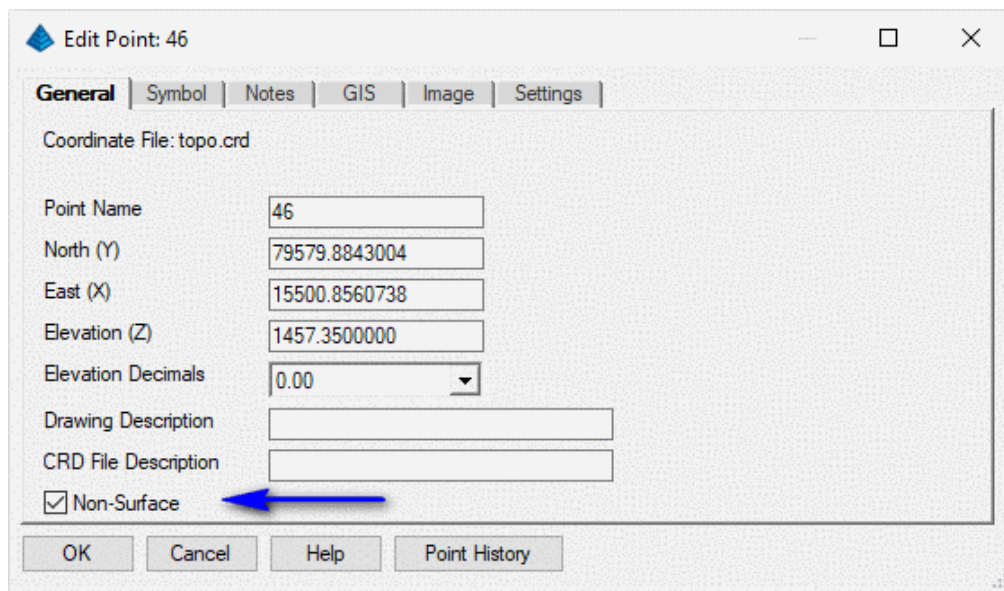


- The arrows above point to an area that needs editing. This area is enlarged below. You may want to zoom into this area for the next edit operation, and zoom back out when you are finished:



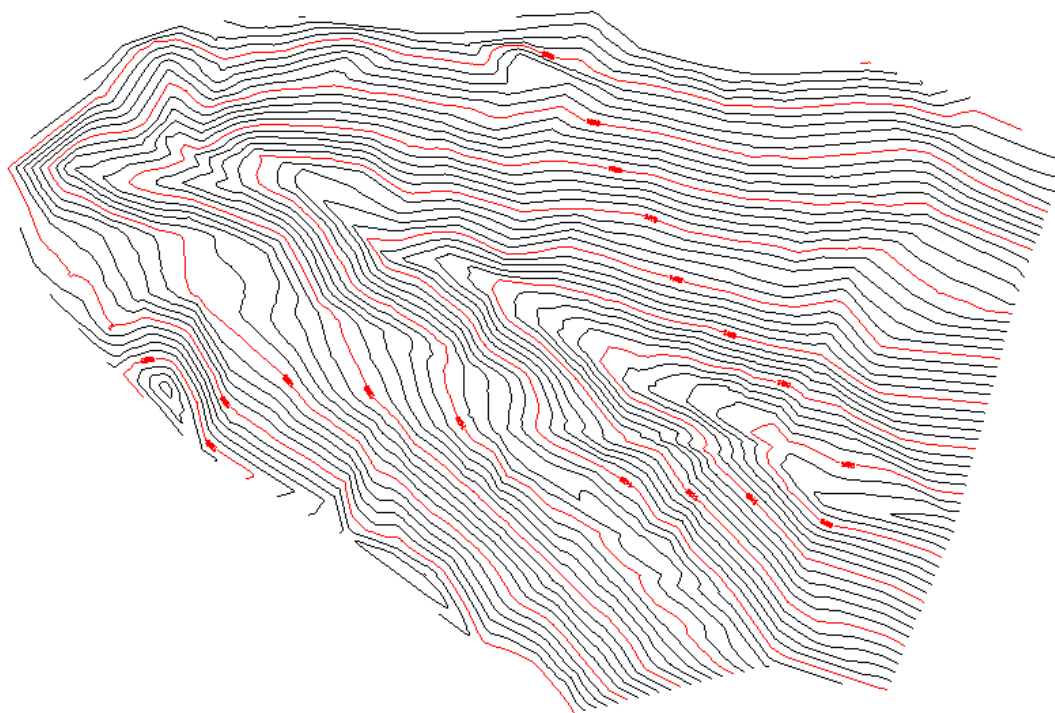
While some may prefer to directly change the contours (e.g. through the Surface – Modify Contours – Edit Contours command), we'll simply instruct Carlson to ignore the point for the purposes of contributing to a TIN.

"Double-click" (two left-mouse clicks in rapid succession) on Point 46 which will display the **Edit Point Attributes** dialog box (which can also be found under Points – Edit Point Attributes) as shown below:



Simply enable the **Non-Surface** option as shown above and click **OK**. This is essentially the same as using the 3D Data – Non-Surface Points/Entities – Tag Non-Surface Points command.

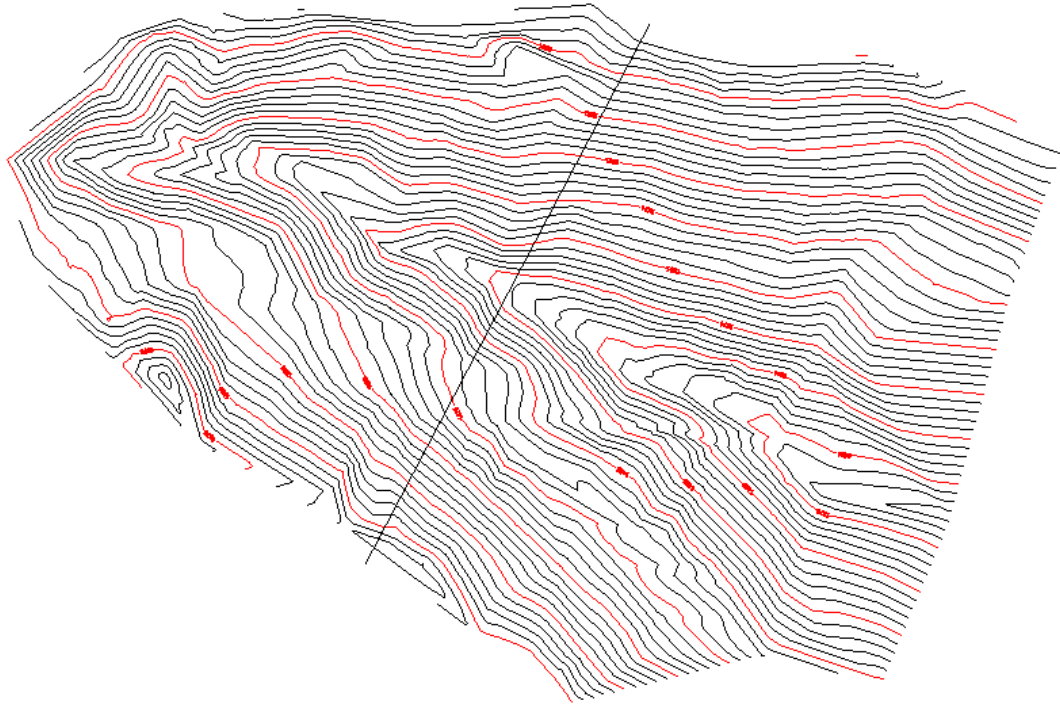
Also double-click on Point 48 to set its Non-Surface setting. When done, re-issue the Surface – **Triangulate & Contour** using the exact same settings and sequence as cited above. Issue the View – Freeze Layer by Selection and pick on one of the points (its number or its elevation), and press Enter. The points freeze. The results are below:



Tip: Whenever you make Carlson files, such as coordinate files (crd), TIN files (tin), and even pond capacity files (cap), they store to disk. When you do an Undo command (U for undo), you undo the graphics, but the files are safely stored and are not undone. Carlson does not like to overdo making files. Make them if you want, but we will not make any that are not needed.

5. Draw a 2D Polyline across the valley to represent the centerline of a dam through the use of the Draw – 2D Polyline command. Click the **OK** button on the **Polyline 2D Options** dialog box if it appears. The Carlson

"2dp" command is an enhanced version of the standard polyline command which draws the same polyline entity that you get when you type **PL** at the command prompt. Try to split the valley with the polyline as illustrated below:



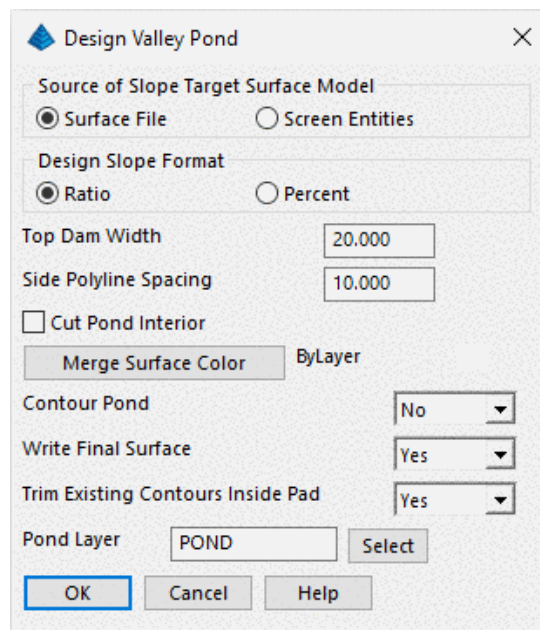
When prompted:

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: pick the southerly starting location

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: pick the northerly ending location

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: press Enter to terminate the command

6. Let's create an earthen dam from the polyline we just drew through the use of the Surface – Design Pond – Design Valley Pond command. Set the values as shown below and click **OK** when complete:



When prompted:

Pick top of pond polyline: pick the polyline as drawn earlier

Select the TIN file Topo-Ex.tin created earlier and then click Open.

Pick point within pond: pick upstream (northwesterly) of the polyline

Outslope ratio <2.00>: type 3 and press Enter

Interior slope ratio <3.00>: type 4 and press Enter

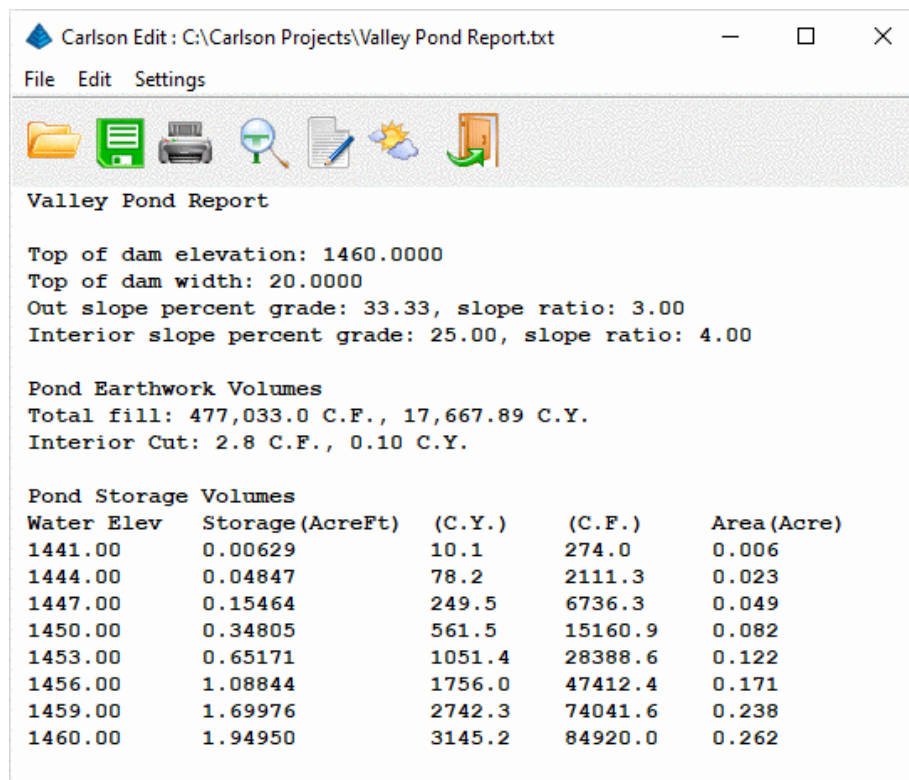
Range of existing elevations along dam top: 1420.63 to 1572.11

Top of dam elevation: type 1460 and press Enter

Calculate stage-storage values [<Yes>/No]? type Y and press Enter

Method to specify storage elevations [<Automatic>/Interval/Manual]? press Enter

The following report includes earthwork volumes and water storage volumes (your report will be slightly different):



Click the **Exit** (Doorway) button to continue forward in the process:

Output dam merged with existing or dam only [Merge/<Dam>]? press Enter and save the surface of just the dam in the file Topo-D.tin (for Topo with Dam)

Write stage-storage file [Yes/<No>]? type Y and press Enter to create a Topo-D.cap file

Adjust parameters and redesign pond [Yes/<No>]? type N and press Enter

Retain trimmed polyline segments [Yes/<No>]? type N and press Enter

Contour the pond [<Yes>/No]? type N and press Enter

The process is complete. At the Command line, enter **E** (for Erase) and when prompted:

[FILter]/<Select entities to delete>: pick on the centerline of the dam (there might be two of them)

[FILter]/<Select entities>: press Enter

Tip: The stage-storage curve that you save will plot in the **Carlson Hydrology** program. It makes a nice, handy plot for report purposes. See Tutorial - Hydrology and Watershed Analysis. Let's save the drawing through the File – **Save** and name the drawing **Topo-D.dwg**.

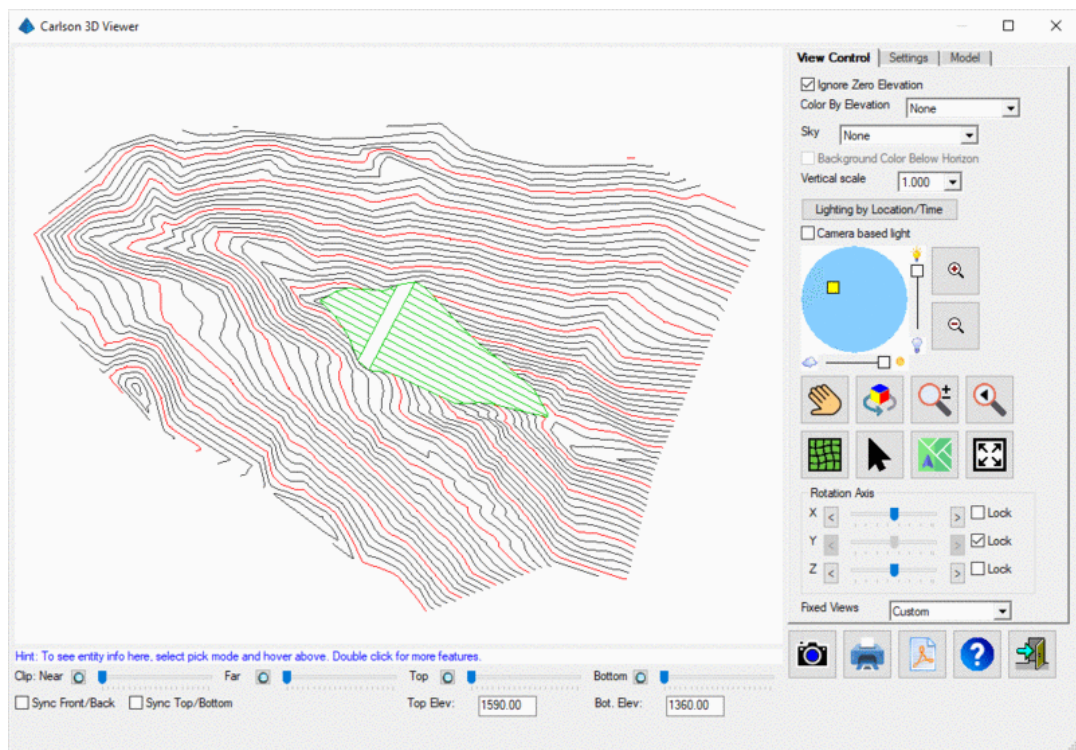
- Let's check out the data in 3D through the use of the View – 3D View – Drawing Viewer command. When prompted:

Select entities for the scene.

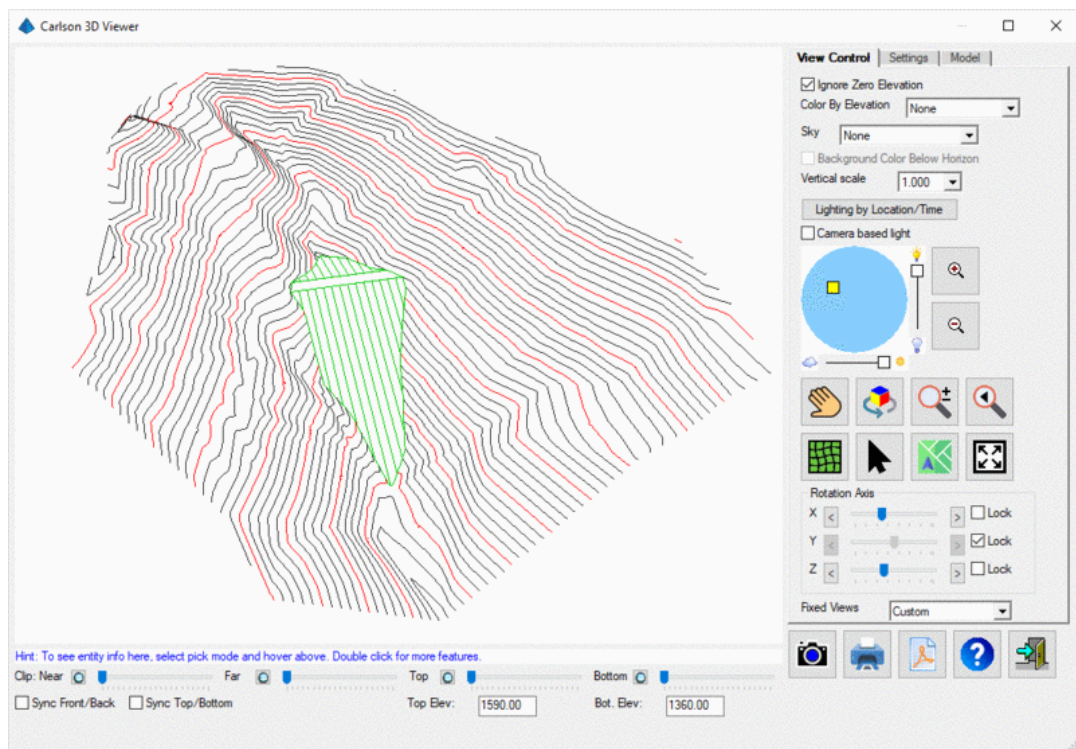
[FILter]/<Select entities>: type ALL and press Enter

[FILter]/<Select entities>: press Enter

This leads to the starter view (a plan view) shown below:

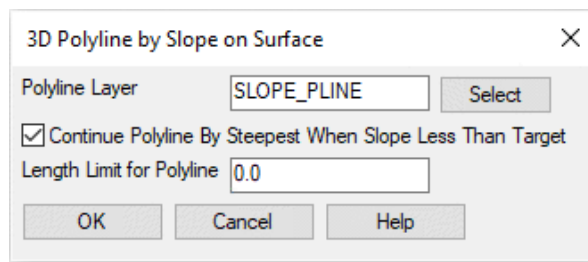


The main trick is to move the X-Axis bar to the left. Avoid the Y-Axis dial for now, and then grip on the Z-Axis dial and move it back and forth relatively fast, or just click on the Z-Axis arrows and watch things move slower. It's like you are in a helicopter over the site. Here's an example:

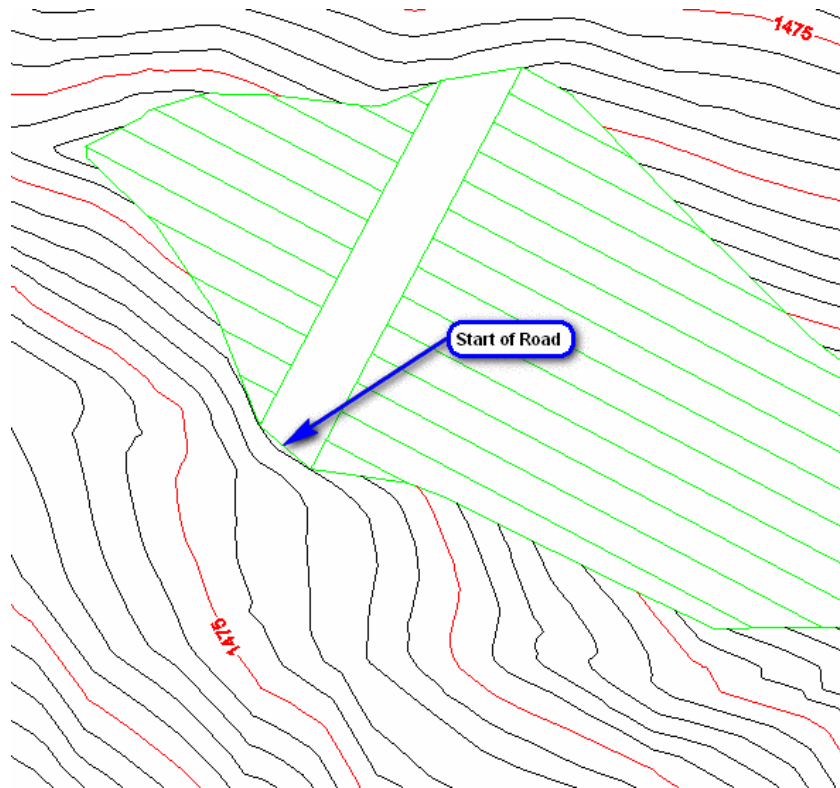


Click the **Exit** (Doorway) button to dismiss the 3D Viewer dialog box.

- Let's create a simple access road to service the dam via the 3D Data – 3D Polyline Utilities – 3D Polyline by Slope on Surface command. The following dialog box appears:



Complete the values as shown above and click **OK**. When prompted: **Choose a Grid or Triangulation File** specify the Topo-Ex.tin file created earlier



Pick origin point of 3D polyline: pick a point on the south side of the top of the dam, just before it contacts the ground as shown above

Direction of 3D polyline [<Up>/Down**]?** D

Direction of 3D polyline facing down slope [<Left>/Right**]?** R

Slope format [<Percent>/Ratio/Degree**]?** press Enter

Enter design slope percent: 10 (the values below should be similar to your results)

Horizontal distance: 415.34, **Slope distance:** 417.41

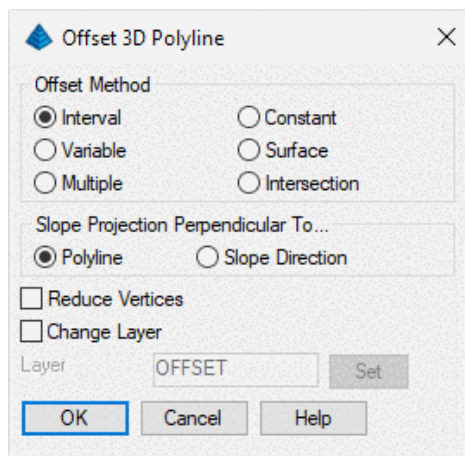
Vertical drop: 41.53, **Avg slope:** 10.00%, **Max slope:** 10.00%

Adjust slope [Yes/<No>**]?** N

Pick origin point of 3D polyline (Enter to end): press Enter

This created a smooth, 10% downhill grade 3D polyline which we can use to construct a maintenance road to service the dam.

9. Let's create the other side of the maintenance road through the use of the 3D Data – 3D Polyline Utilities – Offset 3D Polyline command (also found under Edit – 3D Polyline Utilities – Offset 3D Polyline):



This is a Carlson specialty; a high-powered Civil Design feature. In short, you can work in 3D because you can offset and manipulate 3D polylines using Carlson. Set the values shown in the dialog box above and click **OK**. When prompted:

Vertical/ <Horizontal offset amount>: 30 (for a 30' wide road)

Percent/Degree/Ratio/Vertical offset amount <0>: press Enter for a zero vertical offset from the original

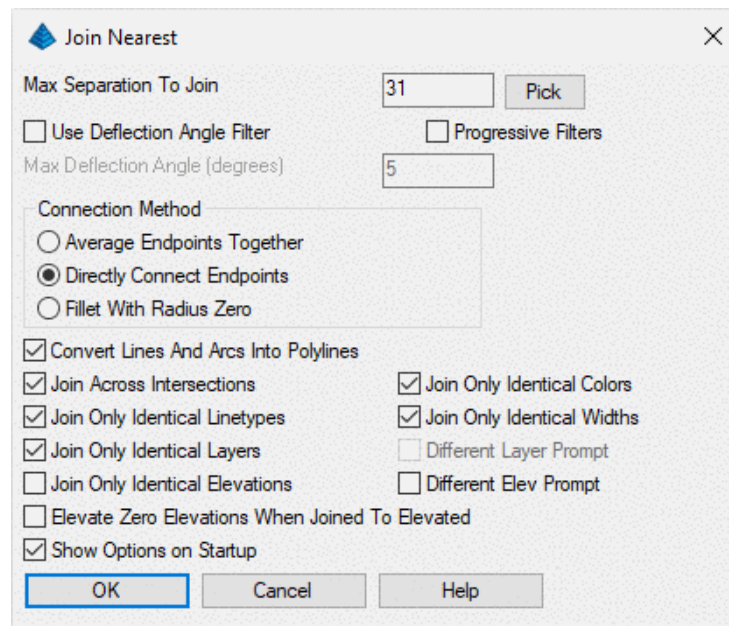
Select a polyline to offset: pick our new 3D polyline

Select side to offset [Both]: pick into the hill (to the left or southwesterly)

Select a polyline to offset (Enter to end): press Enter

This creates the other side of the road parallel, but not joined yet.

- Let's connect the two 3D polylines together through the use of the Edit – Join Nearest command. A dialog appears which you need to fill out as follows:



Because the two polylines are 30' apart, specify the **Max. Separation to Join** value to be 31. That way, they will join! The most important aspect is to specify the **Directly Connect Endpoints** option. Click **OK**. When prompted:

Select lines, arcs and unclosed polylines to join [Settings].

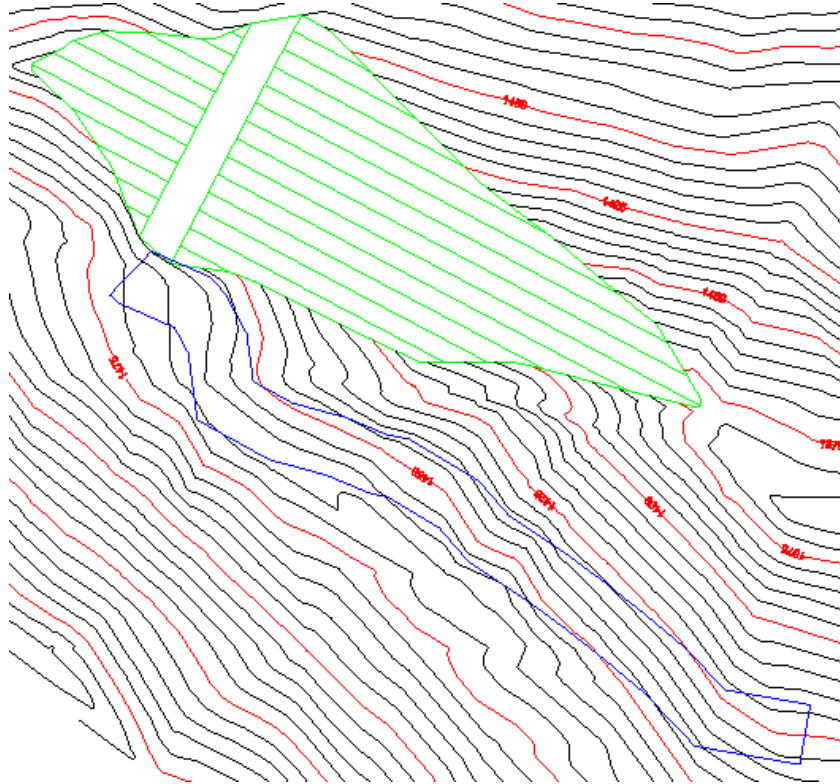
Select entities: pick both sides of the road one at a time, carefully avoiding picking a contour and press Enter

when complete

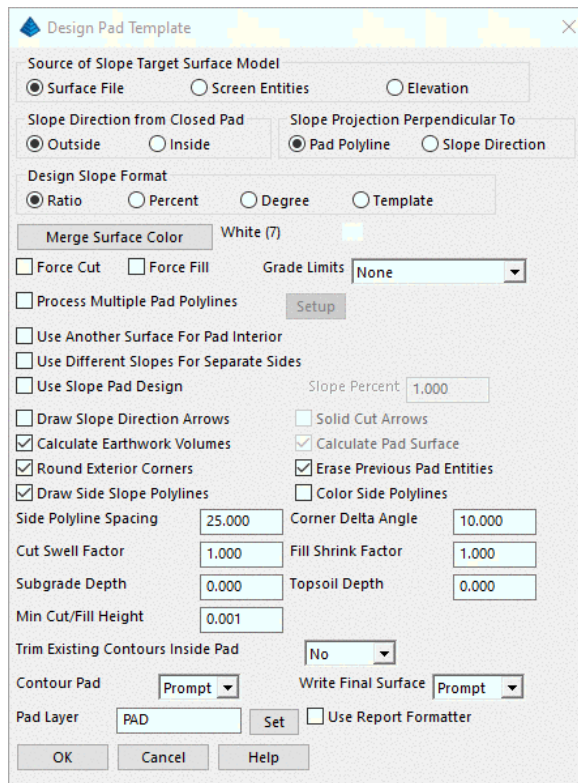
NOTE: If a contour is picked, press the Esc key to exit the command and start over or use Selection options (*e.g.* press R for remove, pick it to remove it from the selection set, then A to Add, and pick again on the road) to build the proper selection set.

Tip: This may be obvious, but when it is difficult to pick what you want (*e.g.* because several objects are nearby or are over top of what you want), it pays to do View – **Window** and zoom in closer, followed by View – **Previous** after you are done.

We have a road, or at least a sloping pad, seen below:



11. Let's project catch-slopes from the access road to the ground surface through the use of the Surface – Design Pad Template command to display the dialog box below:



Design Pad Template is one of the more diverse and powerful commands in Carlson. We will use it here to make a simple cut and fill slope from our road pad. We will go 0.5:1 in cut, but 1:1 in fill. You might think a 2:1 in fill is better, but remember, our hillside edge of the road (the original edge) follows very closely to the hill itself, as designed. If it cantilevers out a few inches, and the natural slope of the ground is 1.5:1 (which it is!), 2:1 will never catch, and we will create big fill areas. So we will go with 1:1 in fill, and get very tiny, quick tie-ins in those few cases where there is any fill at all.

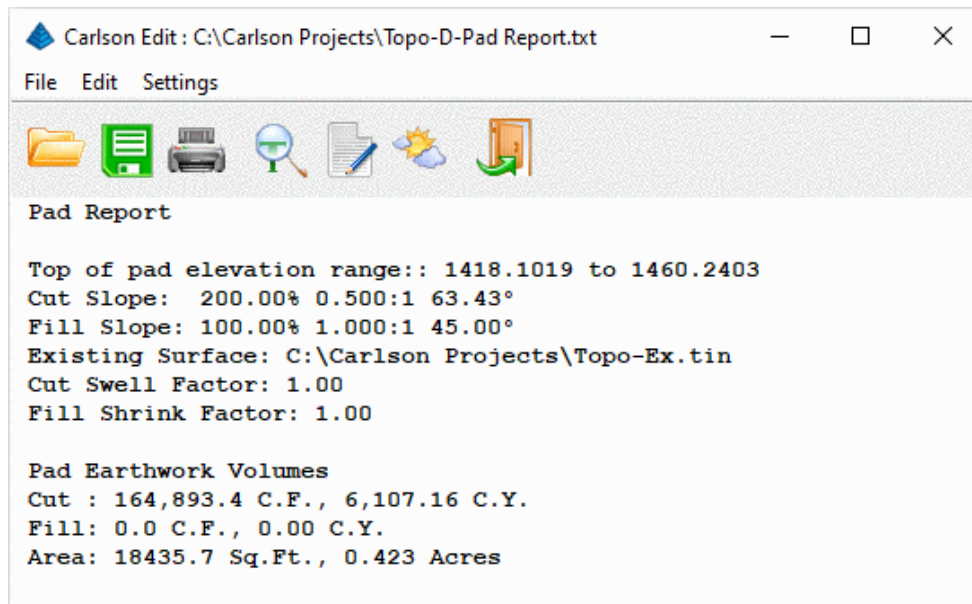
Set the entries as shown above and click **OK**. When prompted:

Pick pad polyline: pick the road pad

Select Slope Target Surface (dialog): specify the Topo-Ex.tin file created earlier

Fill outslope ratio <2.000>: 1

Cut outslope ratio <1.000>: 0.5 (a summary of the earthwork volumes will appear, your report will be slightly different):



Click the **Exit** (Doorway) icon to continue forward in the process:

Adjust parameters and redesign pad [Yes/<No>]? N

TIN Output: click Create New Surface

Merge Pad with Target Surface: click No

Output Triangulation to Write: use Topo-R.tin (R for Road)

Contour the pad [<Yes>/No]? N

Re-run the View – 3D View – **Drawing Viewer** command to view the drawing now with a road and click the **Exit** (Doorway) button when you're done reviewing the data.

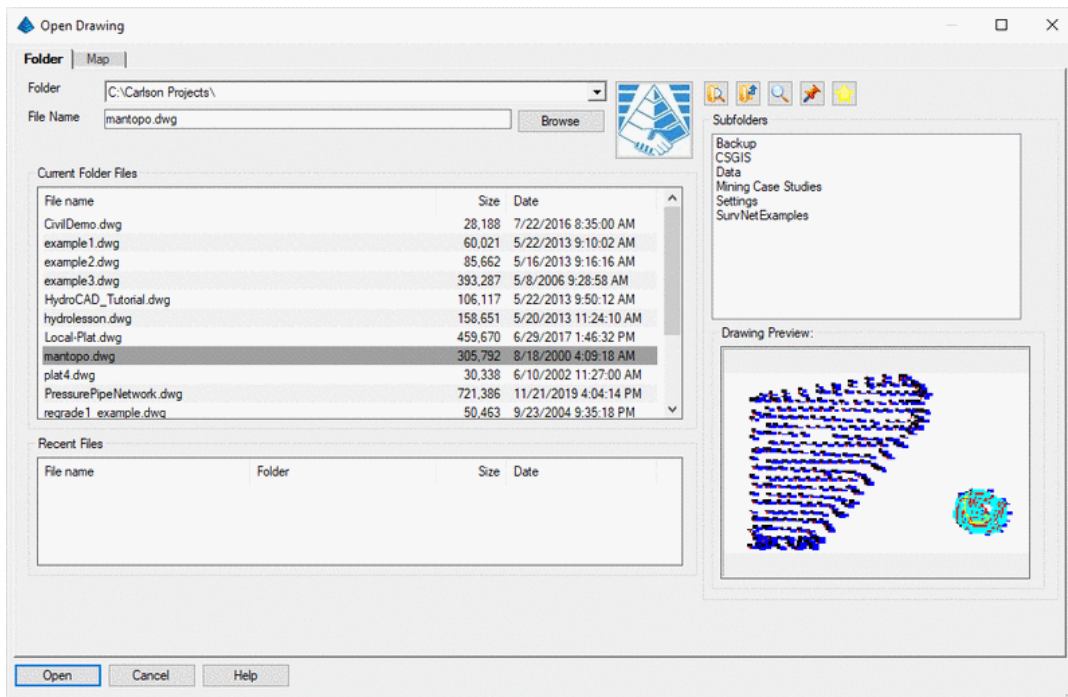
This completes the Lesson 5 tutorial: Contouring, DTM and Design.

Contouring, Break Lines and Stockpiles

In this lesson, you will explore routines centered around creating contours, working with breaklines and volumes from stockpiles.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

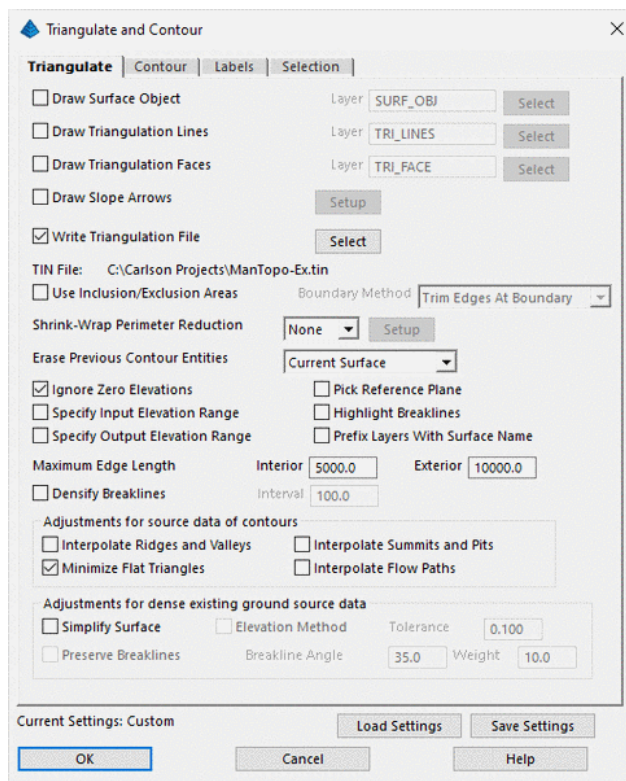
Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **Mantopo.dwg** file:



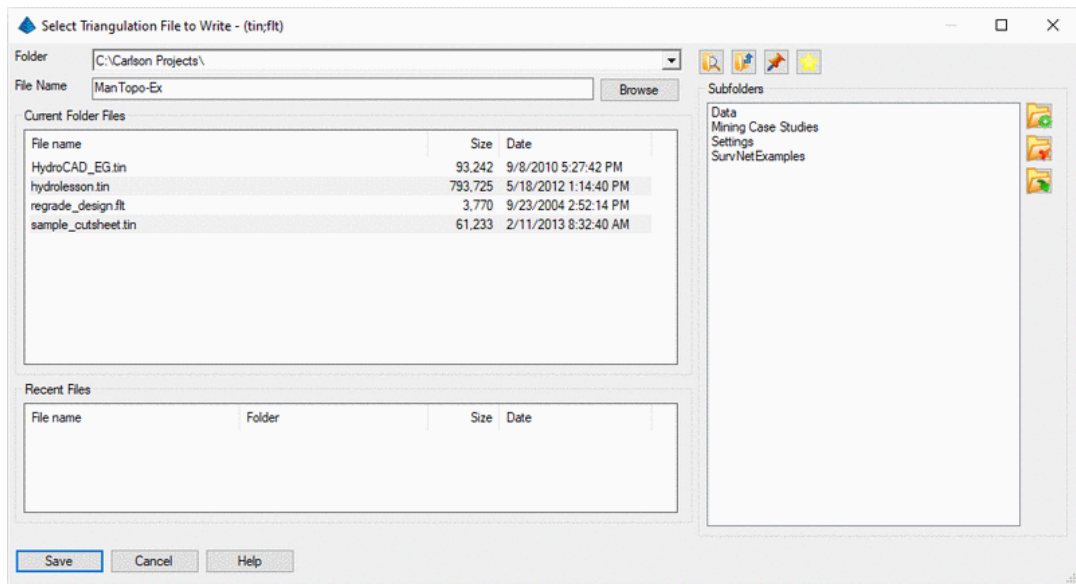
2. Ensure you are in the Survey program by issuing the Settings – Carlson Menus – **Survey Menu** command.

NOTE: Many of the routines below can also be accomplished with the Carlson Civil and/or Construction modules but our focus herein will presume use of the Survey module.

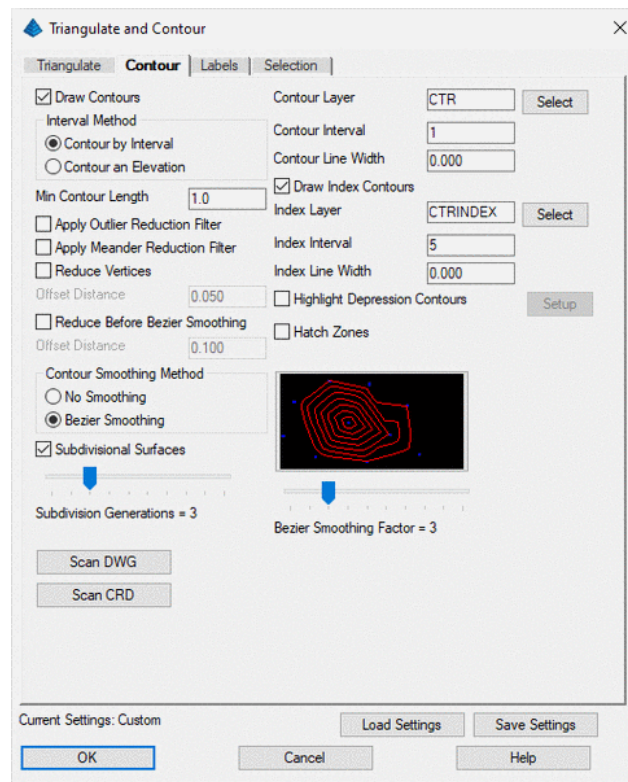
Issue the Surface – Triangulate & Contour command. The following dialog box (with several "tabs") will appear which you should fill out as discussed below:



- In the Triangulate tab (as shown above), enable the **Write Triangulation File** toggle and click its **Select** button to set the name of a triangulation (*.tin) file as illustrated below:



- Click the **Save** button on this dialog box so that the results of the triangulation can be re-used for future purposes. Back in the *Triangulate & Contour* dialog box, turn off the **Use Inclusion/Exclusion Areas** toggle and also turn off the various **Draw** options. Set any remaining options as illustrated above.
- In the Contour tab, as shown below, set the **Contour Interval** to 1, turn on **Draw Index Contours** and set its interval to 5:



Click the **OK** button from any tab. When prompted:

Select points and breaklines to Triangulate.

[FILter]/<Select entities>: type ALL and press Enter twice

If a secondary dialog box appears prompting for a coordinate file (*.crd), select **Mantopo.crd** as your coordinate file and then specify the following default options:

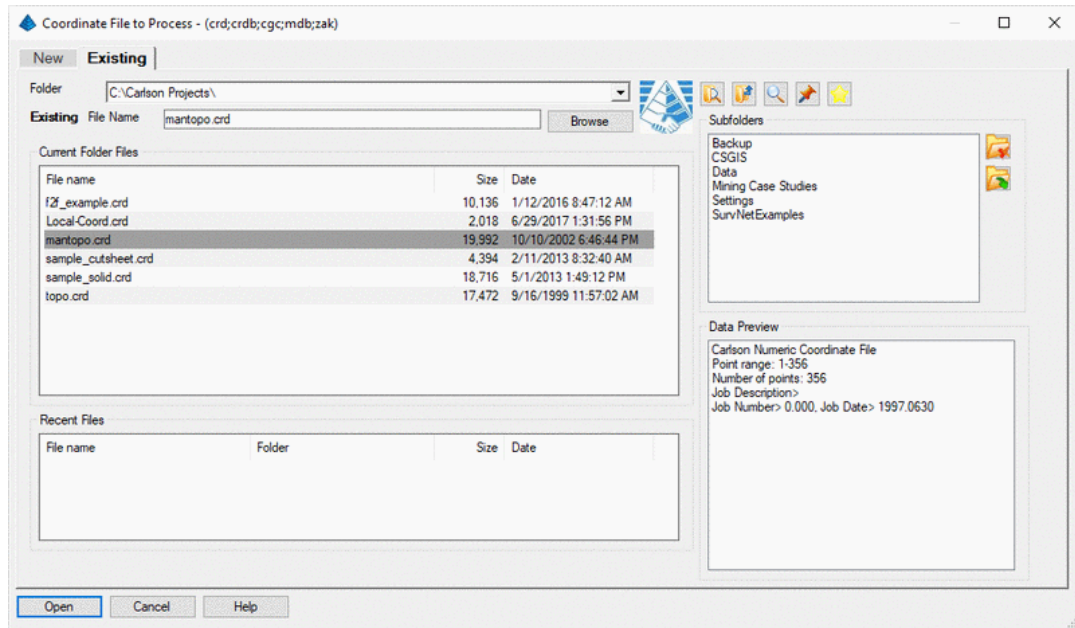
Range of Points: All

Description Match: *

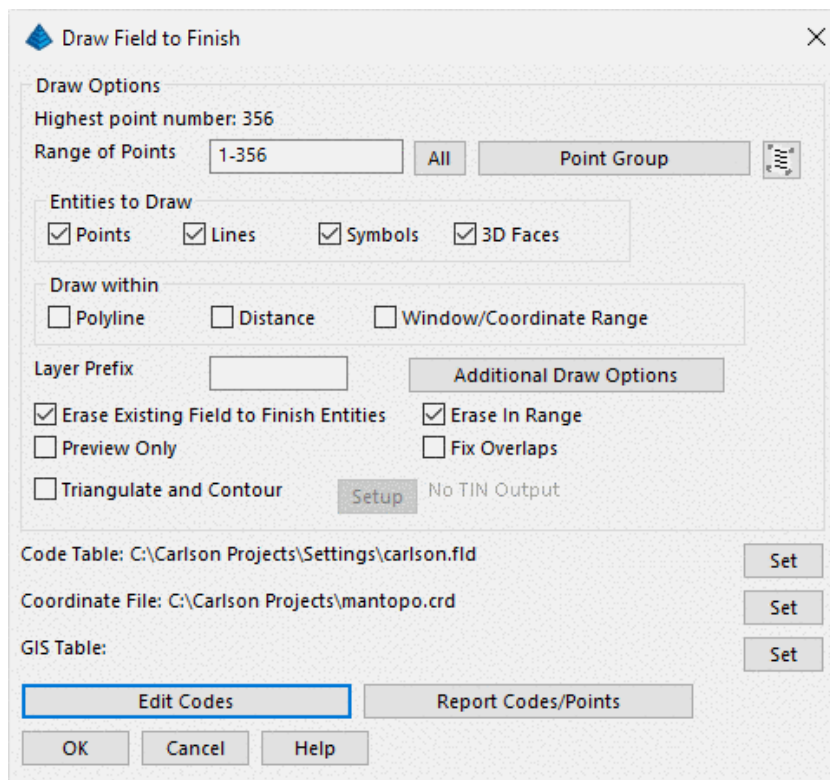
A "TIN" file is written and the contours are drawn. However, notice the unacceptable wavy look around the perimeter, an area which is meant to be a ditch. Let's clean this up through the use of *breaklines*.

3. We'd like to use some automated routines that will automatically generate the breaklines from description data encoded with points collected in the field. Issue the Survey – Draw Field-to-Finish command.

The program will recognize that you have not yet established a coordinate file, so click the **Existing** tab and pick the File as **mantopo.crd** as shown below:

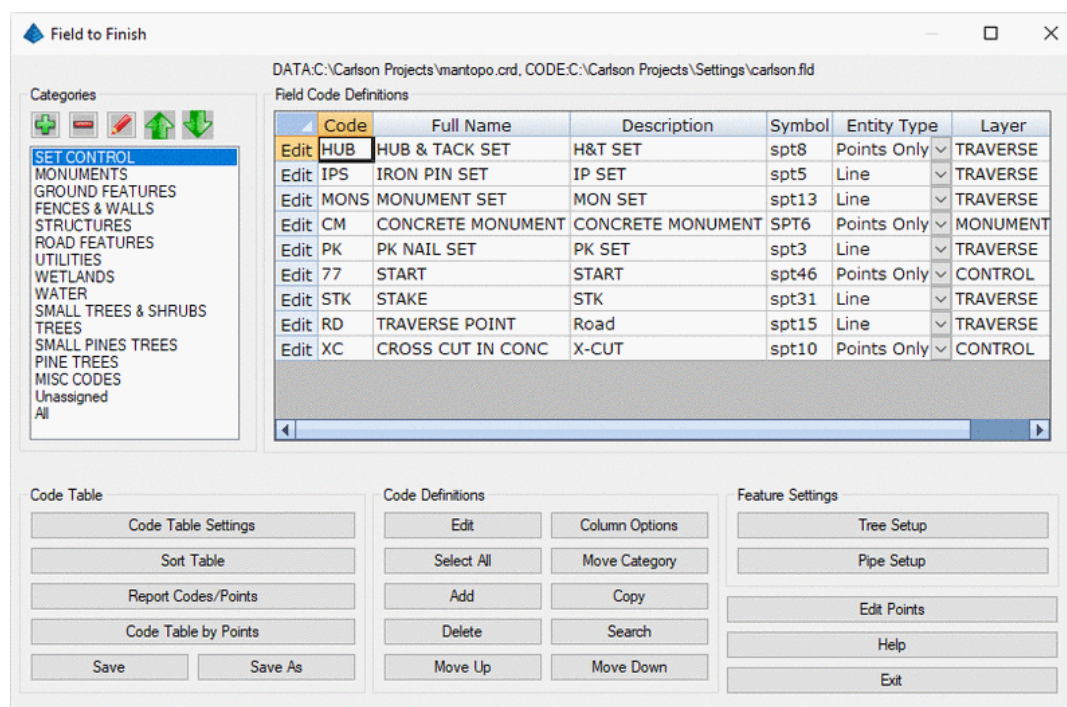


Click **Open**. The **Draw Field to Finish** dialog appears which may (should) resemble the following:

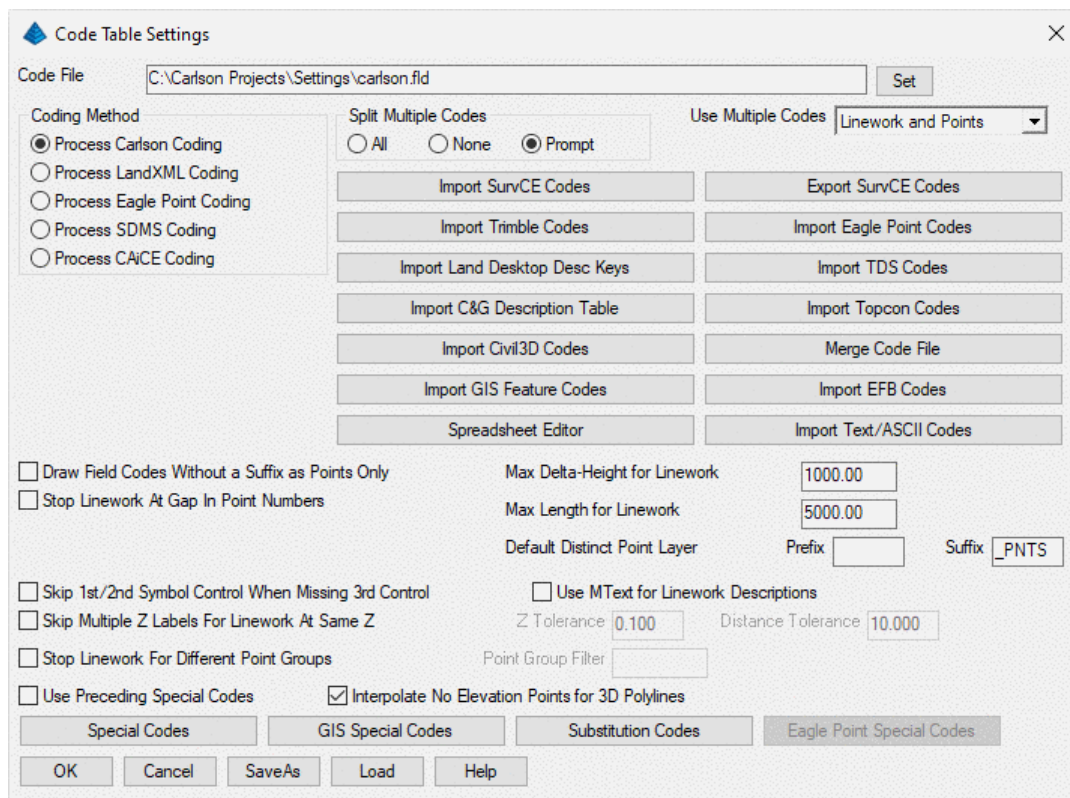


NOTE: You may need to click the **Set** button to the right of **Code Table:** to specify/use the "C:\Carlson Projects\Settings\Carlson.fld" file.

At the lower left of the Draw Field to Finish dialog, click **Edit Codes**. The Field to Finish dialog appears:

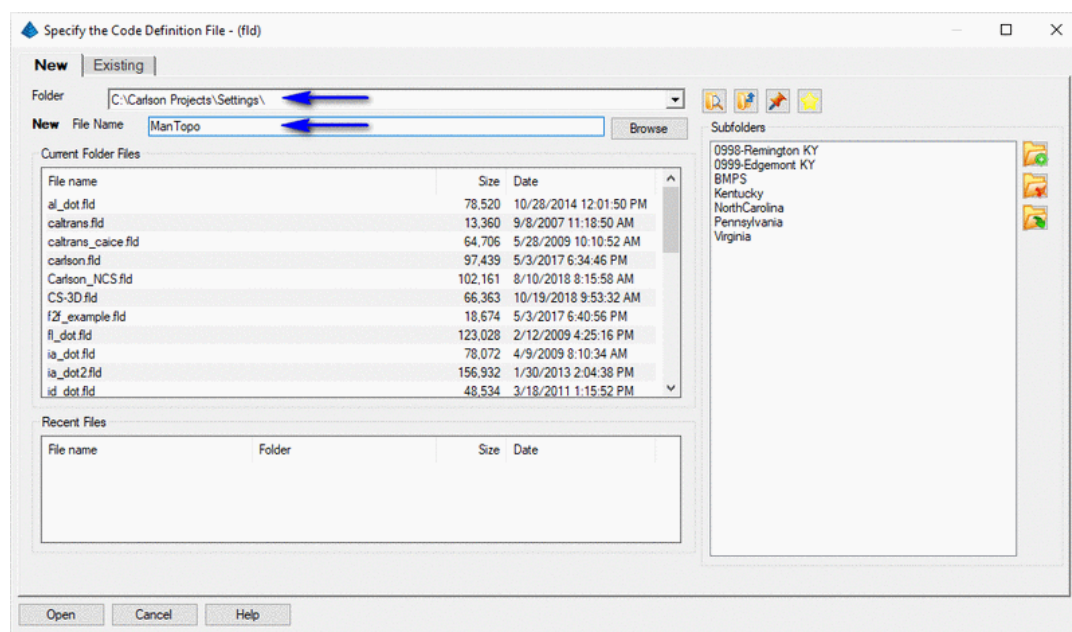


On the left side of the Field to Finish dialog, under the heading Code Table, there is an option called **Code Table Settings**. Click on it to display this dialog:

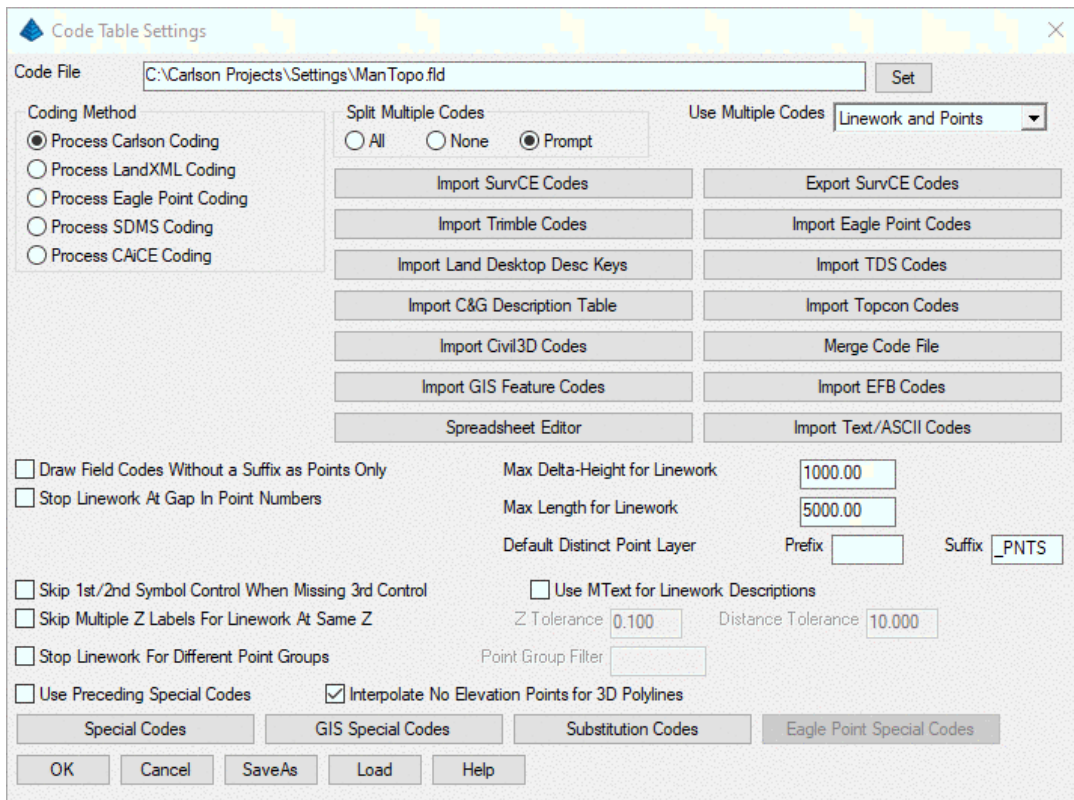


As you can see at the top of the Code Table Settings dialog, the current Field to Finish code definition (.FLD) file is **Carlson.fld**. We want to make a new code table because the coordinate file for the field survey includes special coding (e.g. 17 and 18) for ditch lines and top of banks.

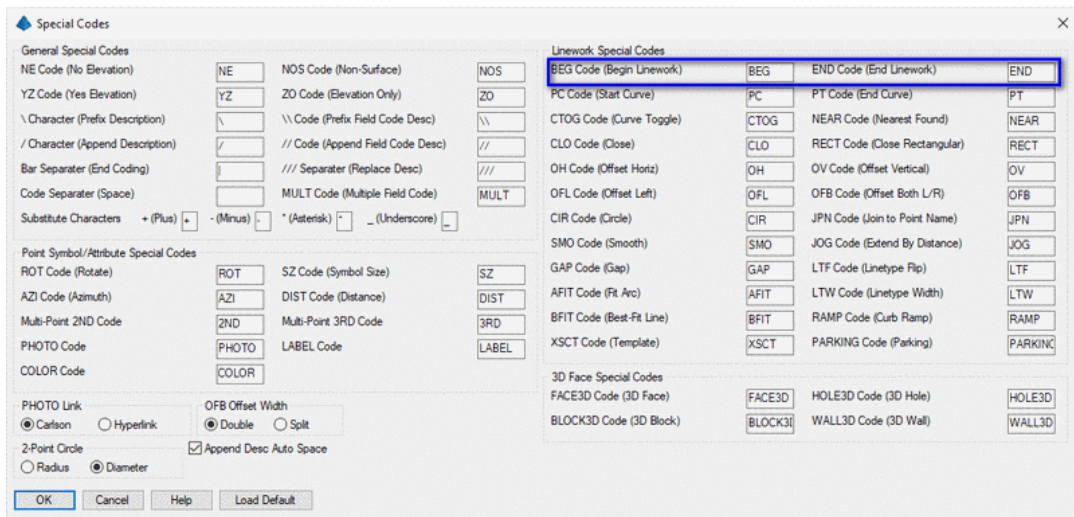
To do this, click **Set** at the upper-right of the Code Table Settings dialog, then choose the New tab (for new file) and name it **ManTopo**, as shown below (noting the "\\Carlson Projects\Settings" folder):



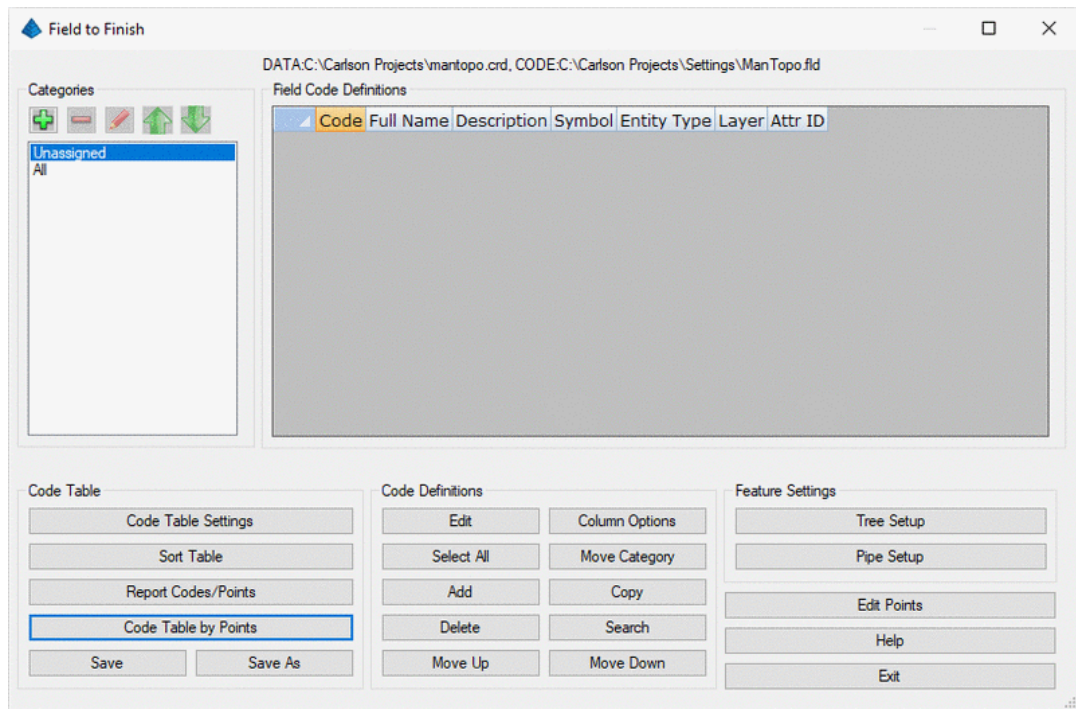
Click Open. You will be taken to the previous dialog.



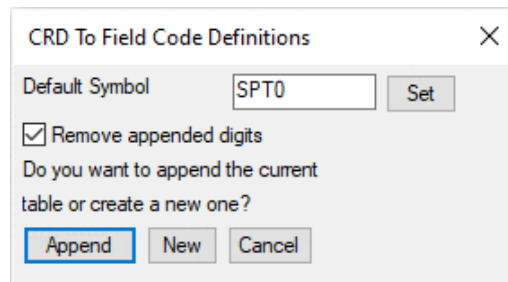
Notice how "C:\Carlson Projects\Settings\ManTopo.fld" is now listed at the top which is our new field code library. Let's also explore some additional capabilities. Click the **Special Codes** button to display the dialog box below:



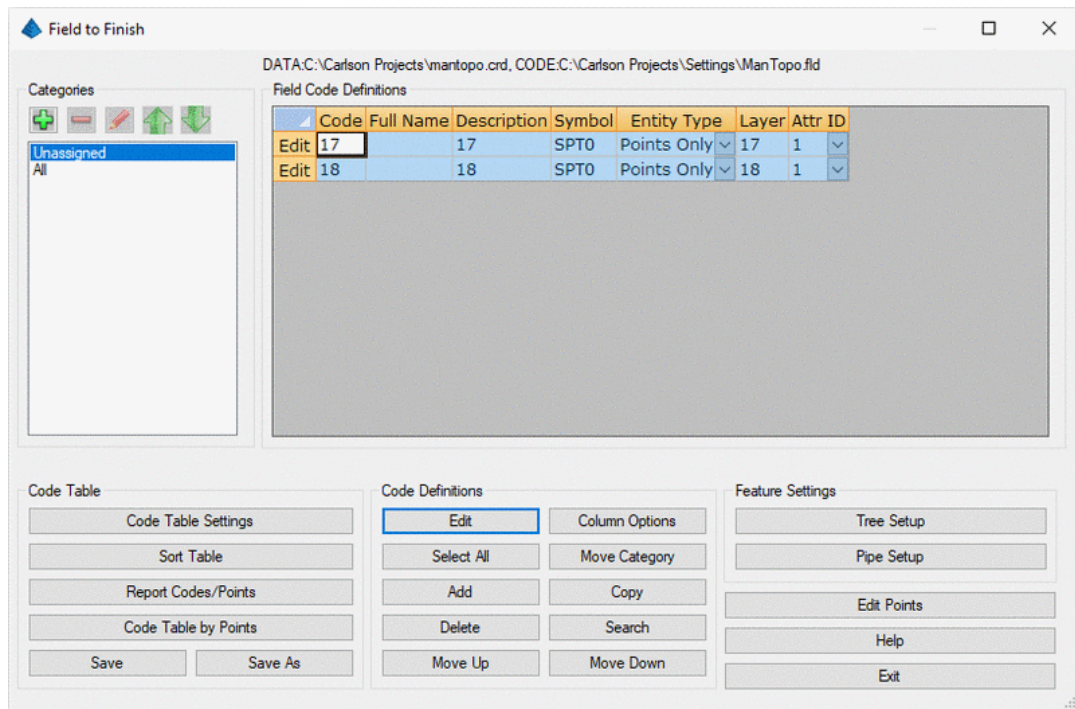
Make note of the **BEG Code (Begin Linework)** and the **END Code (End Linework)** special codes. These can be used to indicate special instances in the data set where linework is to begin and/or end, respectively. We'll use the END code later. Click **OK** to return to the previous dialog box. From it, click **OK**. You will return to the main Field to Finish table, completely empty, as shown below:



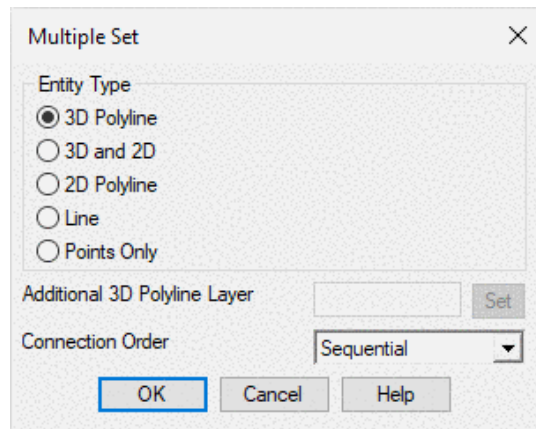
Now, get a jump start on the table by choosing the option **Code Table by Points** (located in the lower left of the dialog). Choose Append on the subsequent dialog box (shown below).



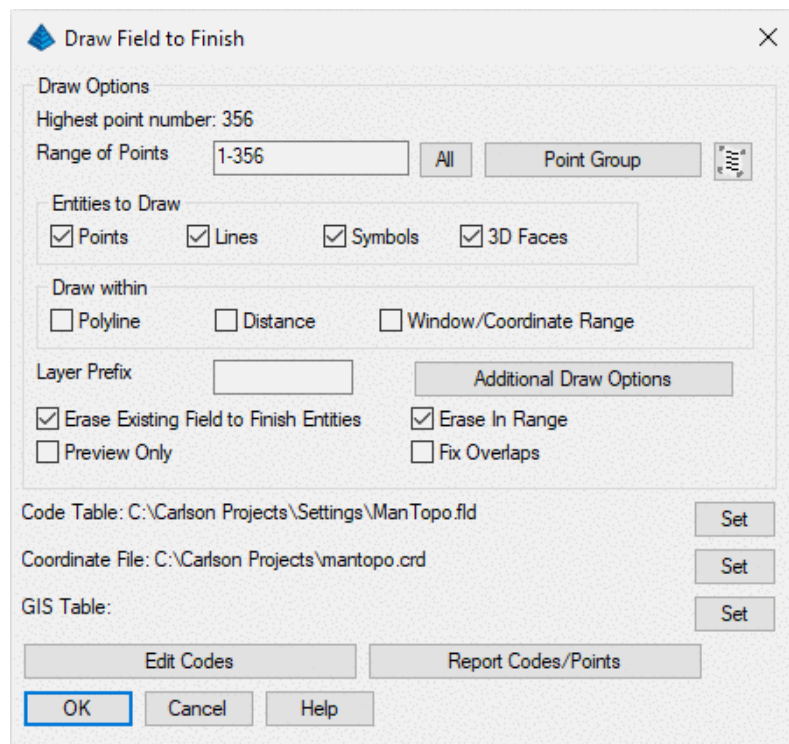
In this lesson, we only care about codes 17 and 18. Select all other codes (the *GROUND* code in this case) and click the **Delete** button to remove that Code from the list. Now select Codes 17 and 18 (by holding the CTRL or SHIFT key down while picking or using the **Select All** button) as shown below:



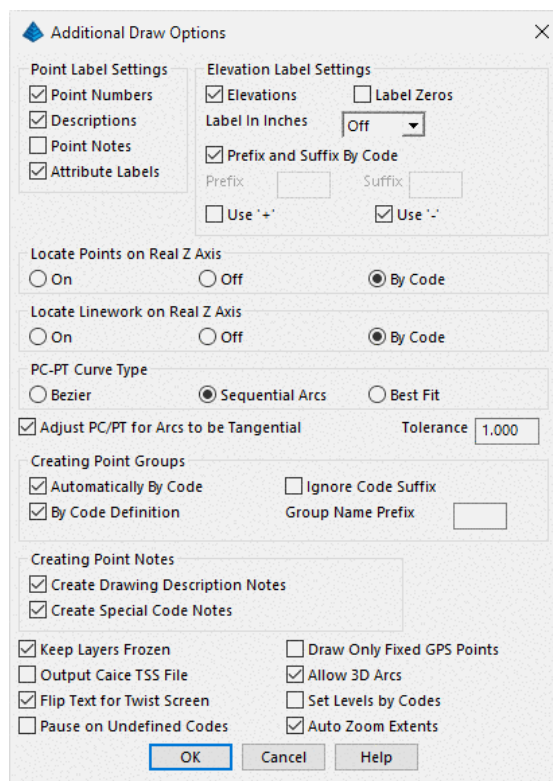
Pick the **Edit** button under Code Definitions. The *Multiple Set* dialog appears. Click the **Entity** button to establish the type of entity that should be drawn for codes 17 and 18 as shown below. Make all settings as shown in this box. We will instruct them to both generate 3D Polylines (which will act as breaklines or barrier lines for contouring). Accept the 3D Polyline choice by clicking **OK**:



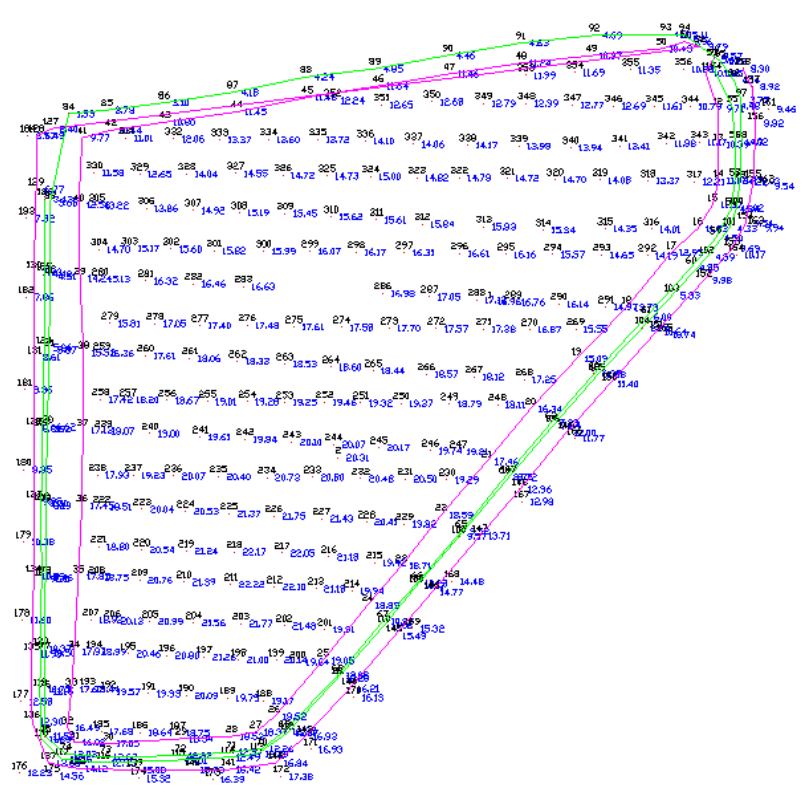
Click the **Layout ID** button to establish positions and style for the attributes (*i.e.* Point Number, Elevation and Description) for codes 17 and 18. Specify Attribute Layout ID **5** and click **OK**. Then click **Exit**, which will take you back to the Field to Finish dialog. The last steps are to first to save the **ManTopo.fld** Field-to-Finish by clicking the **Save** button. Then click **Exit** (lower right). You will return to the following dialog which allows you to control the details of what to draw:



Click the **Additional Draw Options** button and make sure that the **Point Label Settings** are set so that you can see the points properly as shown below:



Click **OK** to both dialog boxes. The following drawing is created. All the ditch lines and top of bank lines, because they were coded 17 and 18, are drawn in one quick procedure (previous contours removed and coloring added for clarity):

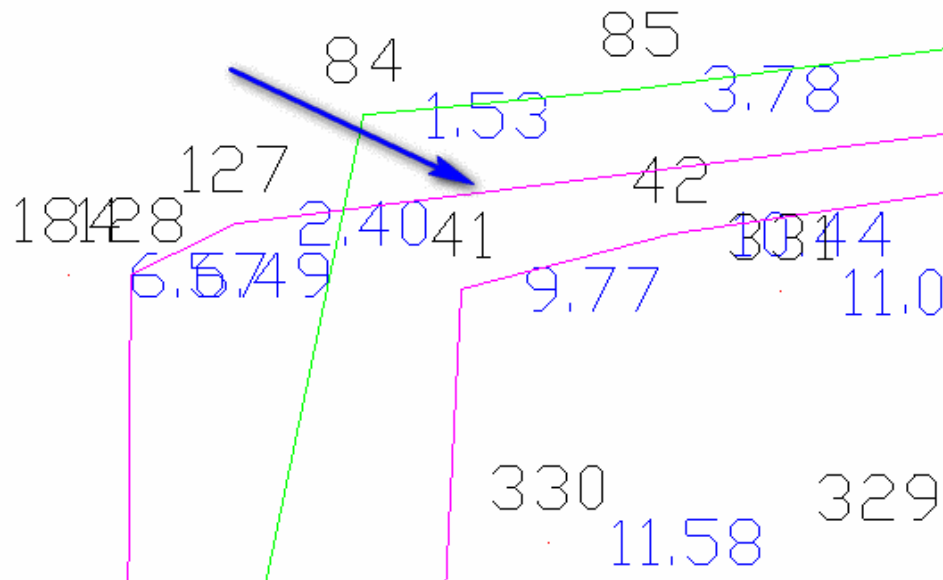


- Because the field crew did not use start and stop logic (e.g. appending **END** or some agreed upon code to a description could end a polyline and start another), some polylines connect that should not. In particular, the line near the NW corner is clearly crossing the ditch line. It must be removed. One approach would be to use the Edit – Polyline Utilities – Remove Polyline – Remove Polyline Segment command. When prompted:

Mode: break polyline at removal

Select polyline segment to remove [Options]: select the polyline segment to the right of point 127 indicated by the arrow below (you will recognize this as a long segment running from point 127 to point 52)

Select polyline segment to remove (Enter to end): press Enter



The problem with this approach is that if the **Draw Field to Finish** routine is ever re-run (e.g. perhaps due to

- additional data coming in from the field), the polyline segment would return. We'll address this shortly.
5. Re-run the Surface – **Triangulate & Contour** command and use the same settings as before and click **OK**. When prompted:

Select points and breaklines to Triangulate.

[FILter]/<Select entities>: type ALL and press Enter twice

NOTE: With breaklines added to the site, you may be alerted of a crossing breakline scenario located at the northeast corner of the site. Use the **Zoom To** and **Zoom In/Out** buttons to explore the data geometry at this area of the site. Let's permanently take care of this by modifying the descriptions of two key points... #10 and #52 (and optionally a third point, #94).

Use the Points – Edit Points command and make the following changes to the **Description** value:

Point Number	Old Description	New Description
10	17	GROUND
52	17	17 END
94	18	18 END

Edit Point Descriptions

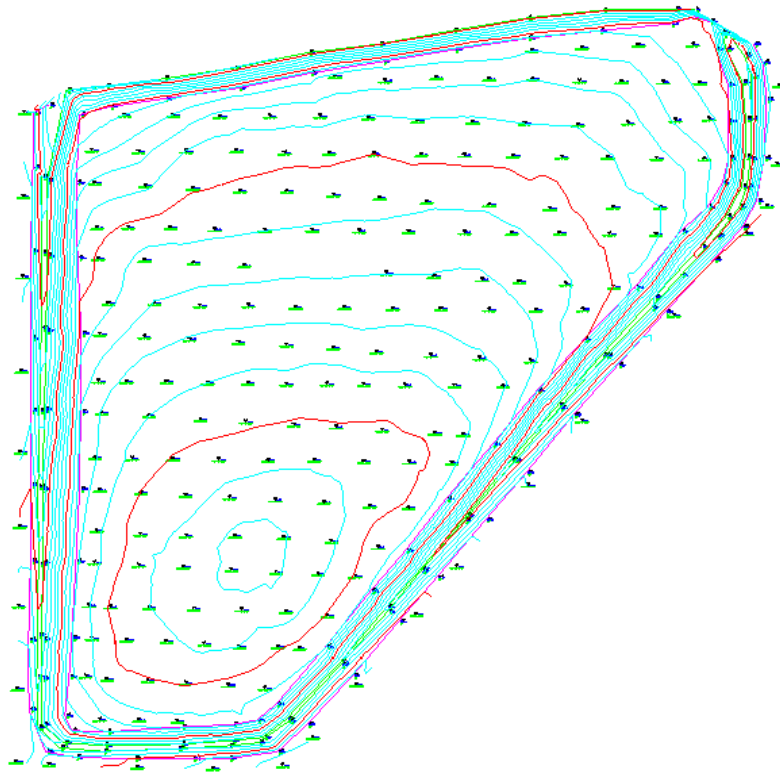
Re-run the Survey – **Draw Field to Finish** command and use the same settings as before and click **OK**.

Re-run the Surface – **Triangulate & Contour** command and use the same settings as before and click **OK**. When prompted:

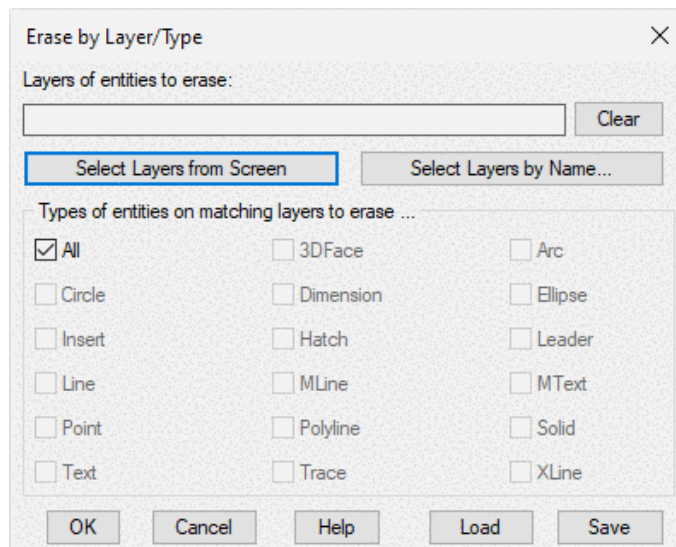
Select points and breaklines to Triangulate.

[FILter]/<Select entities>: type ALL and press Enter twice

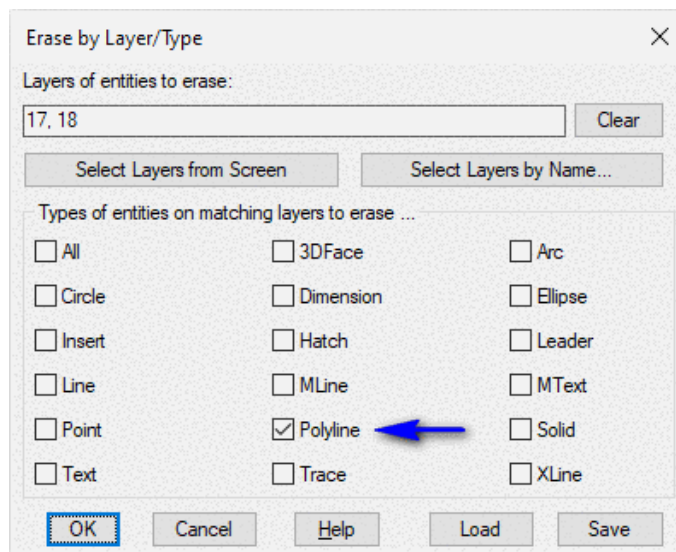
Now we get excellent contours, with a sharply defined ditch. Also note we did not get any processing errors when making the TIN. Use the View – Freeze Layer by Selection and pick on one of the points (its number or its elevation), and press Enter. The points freeze. Here is the improved drawing, helped out by 3D polylines that were produced by Draw Field-to-Finish:



6. Let's say that now you don't want the breaklines in the drawing (since they can be easily recreated with **Draw Field to Finish**). You don't want to even freeze them; you want to fully delete them. There is a command for that through the use of the Edit – Erase – Erase by Layer command. This dialog appears:



If you know the layer names, you can just type them in. If you know where they are but not their names, then click on **Select Layers from Screen**. If you'd recognize the layer name if you saw it in a list, click **Select Layers by Name**. Click on **Select Layers by Name** and pick 17 and 18, then **OK**. Set the Entities to Erase as shown below:



Notice the change in the drawing.

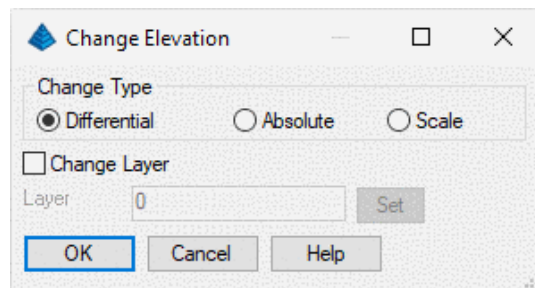
- To the east of the data set we've been working on is a "block" (one single "insert" entity composed of one or more source entities) which represents a stockpile. Because the stockpile data is currently packaged into this single block insert, erasing one component of the block would erase all other components within the block. We'd like to return it to its component parts. Issue the View – **Window** command and window in on the stockpile at the lower right of the drawing. Issue the Edit – Explode – **Standard Explode**. When prompted:

[FILter]/<Select entities to explode>: pick the stockpile block

[FILter]/<Select entities to explode>: press Enter

The exploded block reverts into individual components that can be worked on separately. It's also worth noting that while the block has been exploded, its "block definition" still exists in the drawing. This means that now that it's exploded, it is essentially taking up twice the amount of storage space in the drawing. As such, a common practice would be to "purge" (as obtained through the File – Drawing Utilities – **Purge** command).

- Let's assume our stockpile drawing is too high and should be lowered in elevations by 540 units (feet in this example). To best see the effect of this command, bring back the points by selecting View – Thaw/On All Layers command. Now run the Edit – Change – Elevations to display the dialog box below:



Set the value(s) as shown above and click **OK**. When prompted:

Positive number increases, negative number decreases elevation.

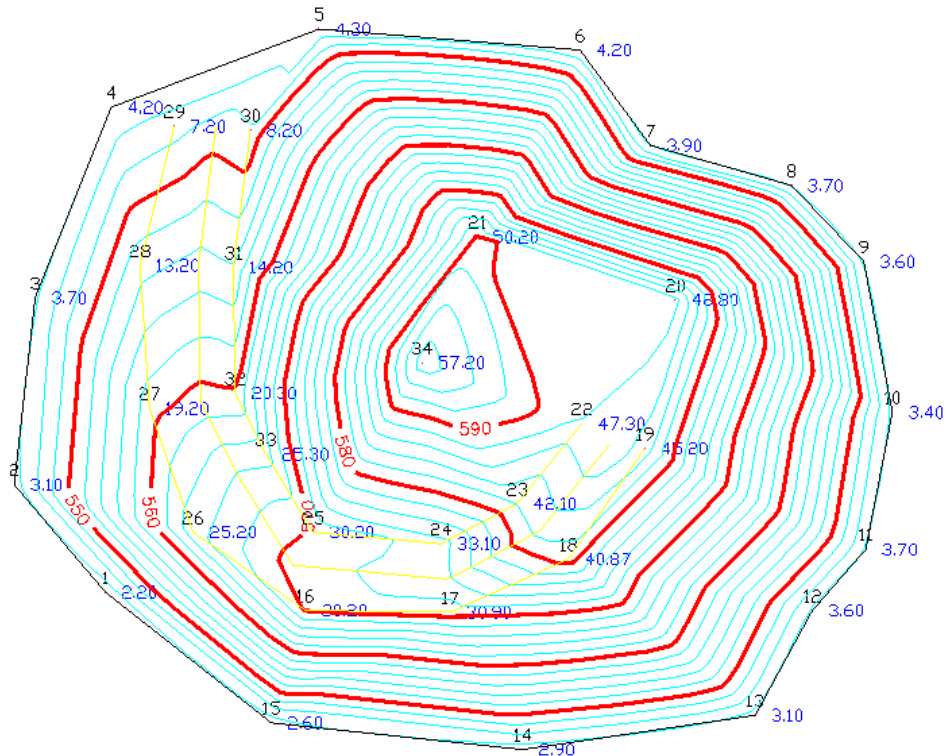
Elevation Difference <0.00>: -540

Ignore zero elevations [Yes/<No>]? Y

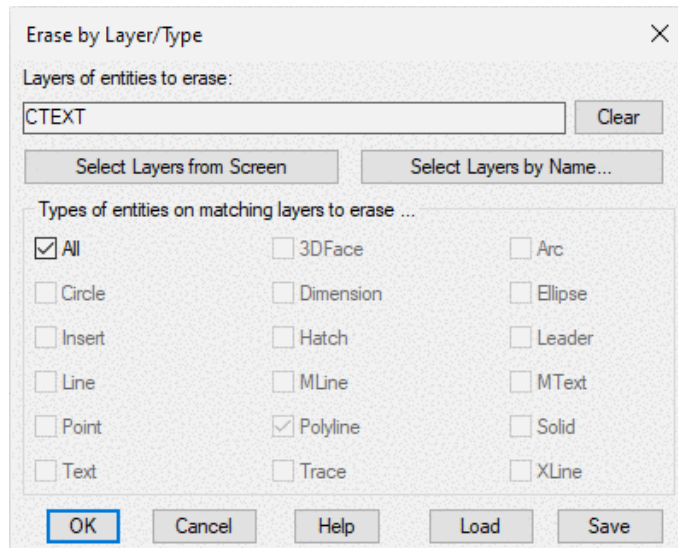
Select Entities for Elevation Change. do a lower right pick to upper left pick selection around the stockpile (automatic crossing)

[FILter]/<Select entities>: press Enter

Notice in the drawing below how the points, breaklines and contour polylines have changed in elevation with the exception of the contour text labels:



Issue the Inquiry – **List Elevation** command and pick on an index contour. Notice how the elevation has indeed changed. Reissue the Edit – Erase – **Erase by Layer** command and set the values as shown below to erase the 5 index contour labels which are no longer accurate.

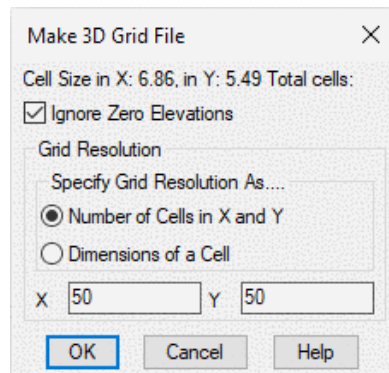


- One of the powerful commands of Carlson (Volumes by Layer) will produce accurate volumes without making any files. The only prerequisite is that the data for the existing and final surfaces exist in the drawing on separate layers. It is also very important to have an inclusion perimeter (boundary polyline) to define the limits where the volumes should be calculated. In our example, the original ground will be the 3D polyline connecting points 1 through 15, and everything else above will be the final ground (including the 3D perimeter itself). Issue the Surface – Volumes by Layers command. When prompted:

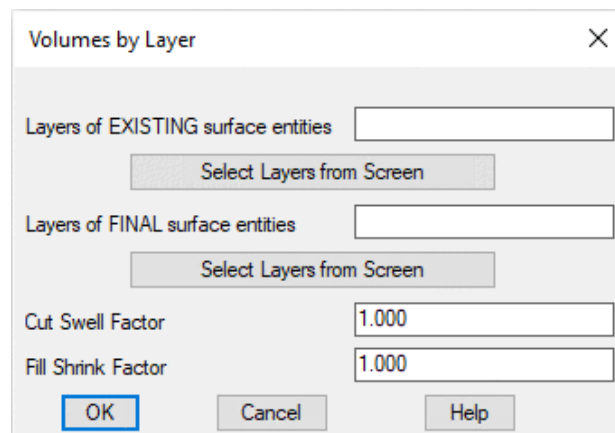
Pick Lower Left limit of surface area: pick below and to the left of the stockpile, but as close as possible to the stockpile without clipping it in the window (you want to totally include it, but with little wasted margin)

Pick Upper Right limit of surface area: pick above and to the right of the stockpile

A dialog appears:



We will stick with the defaults, as shown. Notice that we are using 50 grid cells within our window and since our window was not a perfect square, the cell sizes are not whole numbers. In this example it is 6.86 x 5.49. You may have slightly different sizes. Seeing this, if we wanted a 5 x 5 cell size, we could click the **Dimensions of a Cell** option and set the size to 5 x 5. Hundreds or thousands of cells in both directions will increase calculation time. You can experiment with larger cells or, if you prefer, smaller cells (which makes more cells). However, you will begin to see diminishing returns in terms of accuracy changes as your cell size continues to shrink. After a while (depending largely on the spacing of the source data), tighter, smaller cell sizes don't add any value to the precision of the calculation. Click **OK** to display a dialog box similar to that shown below:



Use the **Select Layers from Screen** buttons to select entities as outlined as follows:

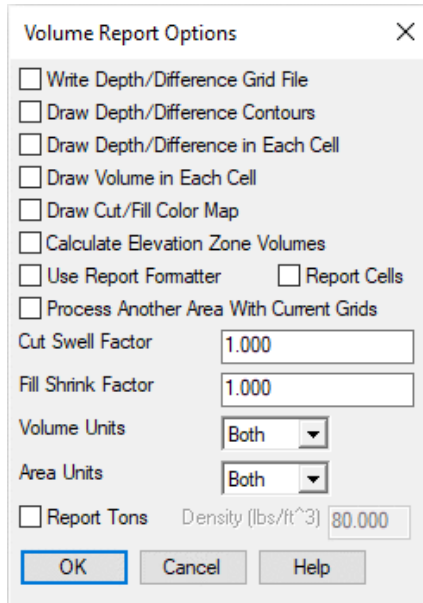
- **Layers of EXISTING surface entities:** PERIMETER
- **Layers of FINAL surface entities:** PERIMETER, BARRIER, CTRINDEX, CTR, PNTS

Notice how the Perimeter layer is common to both. If you want to be a master of volumes, remember this as a mantra: The perimeter should be a 3D polyline in a distinct layer, common to both surfaces. A stockpile is just a special case in that sometimes the 3D perimeter is all you know about the base surface. Click **OK** and when prompted:

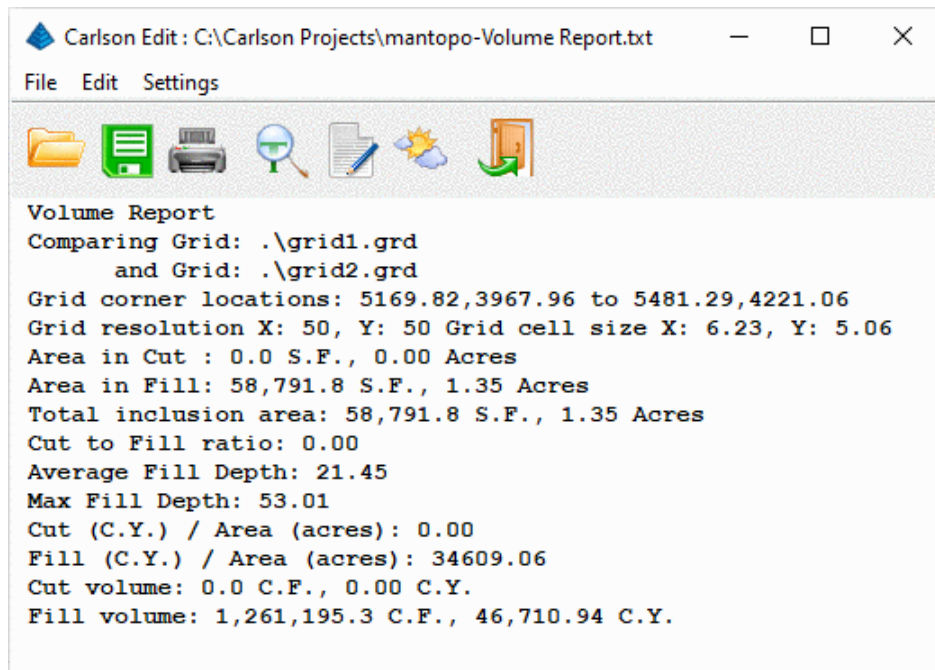
Select surface entities on corresponding layers.

[FILter]/<Select entities>: do a right-to-left (crossing) selection of the entire stockpile area
[FILter]/<Select entities>: press Enter
Select the Inclusion perimeter polylines or ENTER for none:
[FILter]/<Select entities>: pick the perimeter polyline
[FILter]/<Select entities>: press Enter
Select the Exclusion perimeter polylines or ENTER for none.
[FILter]/<Select entities>: press Enter

This leads to a flexible reporting and output dialog:



Elevation Zone Volumes, for example, would produce volumes in any desired increment from the base of the stockpile going up. If the stockpile consists of coal (80 lbs/c.f.), then **Report Tons** can be clicked on and a Density value entered. Click **OK** and the basic report is produced, as seen below:



Click the **Exit** (Doorway) button to return to the command prompt.

10. (Optional): The Stockpile is naturally well-suited for applying the simplest volume command of all **Stockpile Volumes**. It requires that the 3D perimeter polyline for the stockpile be placed in a layer called **Perimeter**, which ours is. So let's try it. Ensure you are in the Civil program by issuing the Settings – Carlson Menus – **Civil Menu** command. Issue the Surface – Stockpile/Ponds/Pit Volumes – Calculate Stockpile Volume. When prompted:

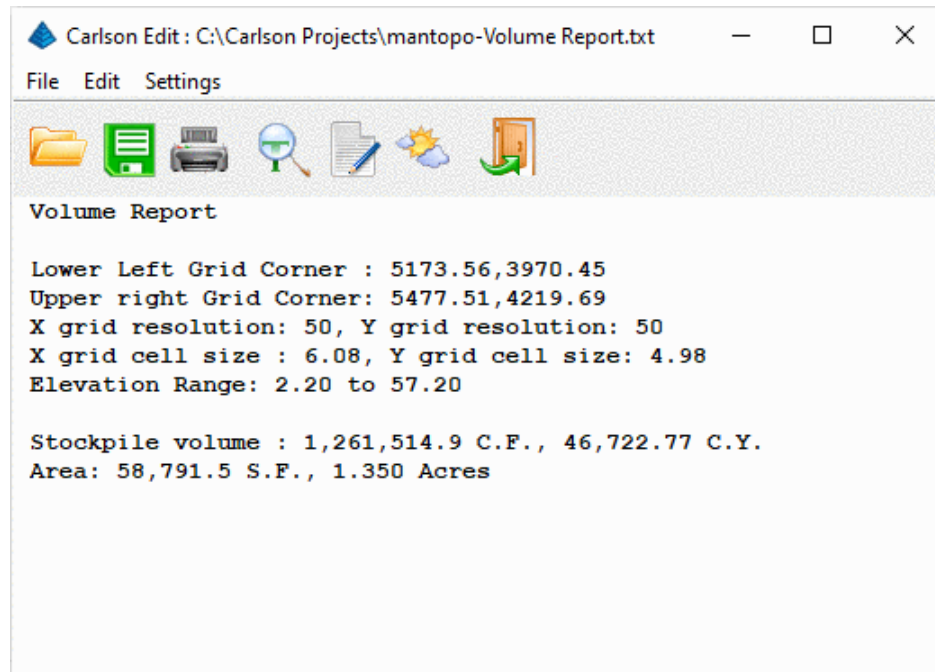
Select stockpile entities and perimeter.

[FILter]/<Select entities>: crossing select (right-to-left picks) the entire stockpile area

[FILter]/<Select entities>: press Enter

Stockpile Volume Report (dialog): accept the default values and click **OK**

The grid resolution dialog (note that it is still at 50x50 but the cell dimensions are likely different from before) appears again. Click **OK**. A report is generated:



This completes the tutorial: Contouring, Break Lines and Stockpiles.

A Dozen Tools for Surface Design

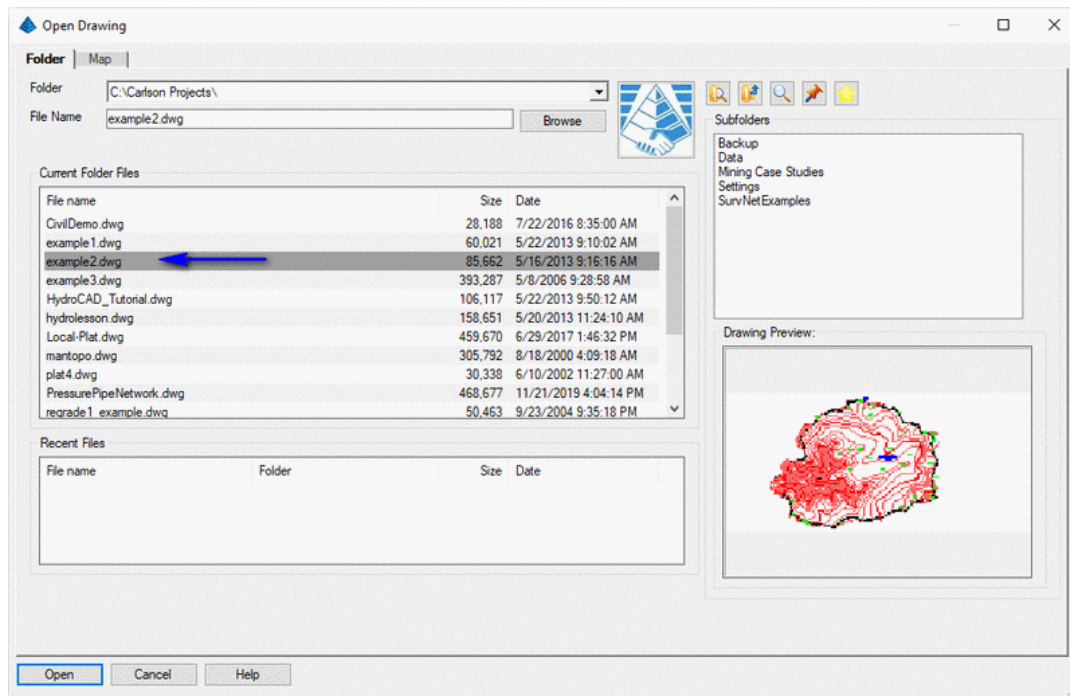
Throughout this tutorial, we'll focus on a variety of commonly used tools for site grading and surface design, including:

1. Draw 2D Polyline
2. Design Pad Template
3. Bench Pond
4. Draw 3D Polyline
5. Join Nearest
6. Offset 3D Polyline
7. Valley Pond
8. 3D Polyline by Slope on Surface
9. 2D to 3D Polyline by Surface Model
10. Profile Editor
11. Profile to 3D Polyline
12. Design Template

Let's begin.

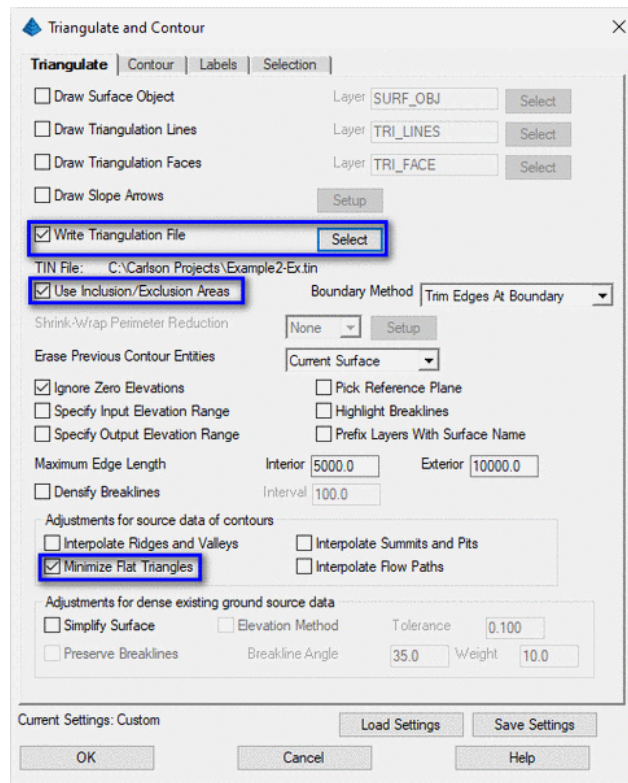
1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **example2.dwg** file.

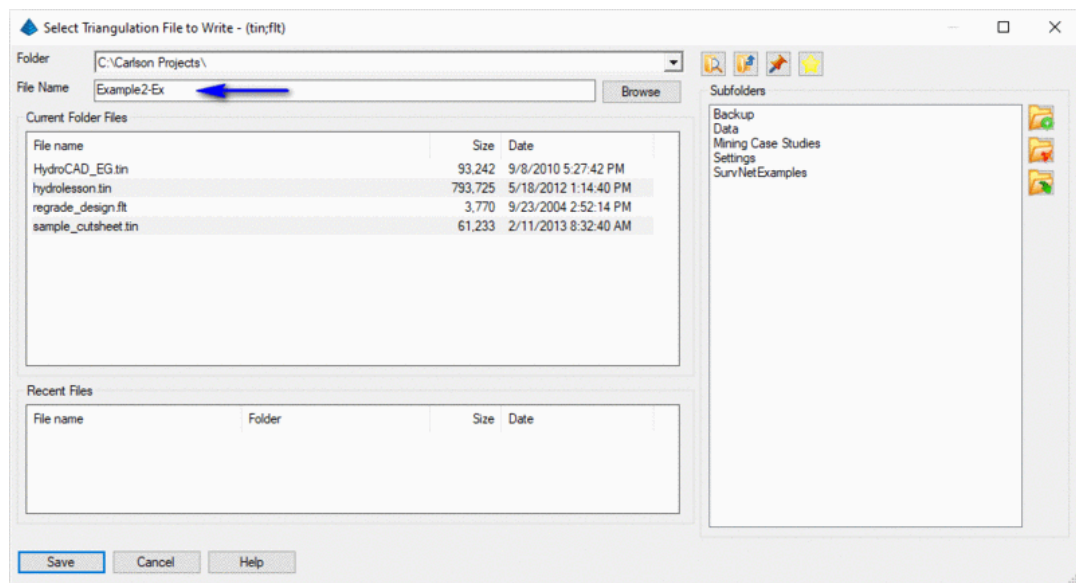


Use the File – **Save As** to save a copy named **example2-s.dwg**. Completing this tutorial will alter the drawing file and by renaming the file from the start, you'll keep the original file intact (allowing you to run through the tutorial a second time if desired). This is also a good practice to keep when working on drawings from 3rd parties.

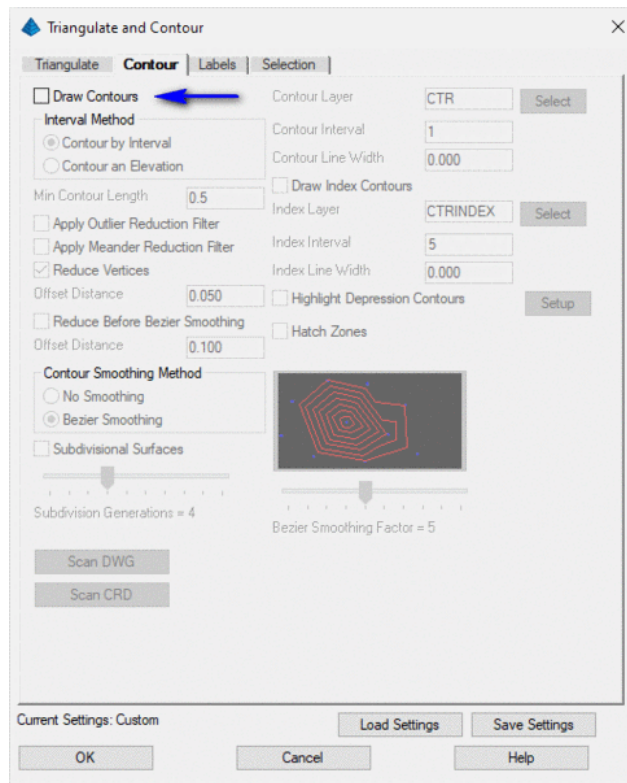
2. Ensure you are in the Civil program by issuing the Settings – Carlson Menus – **Civil Menu** command. Since the bulk of the following commands are designed to integrate into an existing surface, let's make one now. Issue the Surface – Triangulate & Contour command. The following dialog box (with several "tabs") similar to the one shown below will appear which you should fill out as discussed below:



- In the Triangulate tab (as shown above), enable the **Write Triangulation File** toggle and click its **Select** button to set the name of a triangulation (*.tin) file as illustrated below. Click the **Save** button on this dialog box so that the results of the triangulation can be re-used for future purposes. Enable the **Use Inclusion/Exclusion Areas** toggle and also turn on the **Minimize Flat Triangles** option. This will help form a more accurate model. Set any remaining options as illustrated above:



- In the Contour tab, as shown below, toggle off the **Draw Contours** option as there are already contours in the drawing:



Click the **OK** button from any tab. When prompted:

Select Inclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: pick the BORDER polyline and press Enter

[FILter]/<Select entities>: press Enter

Select Exclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: press Enter

Select points and breaklines to Triangulate.

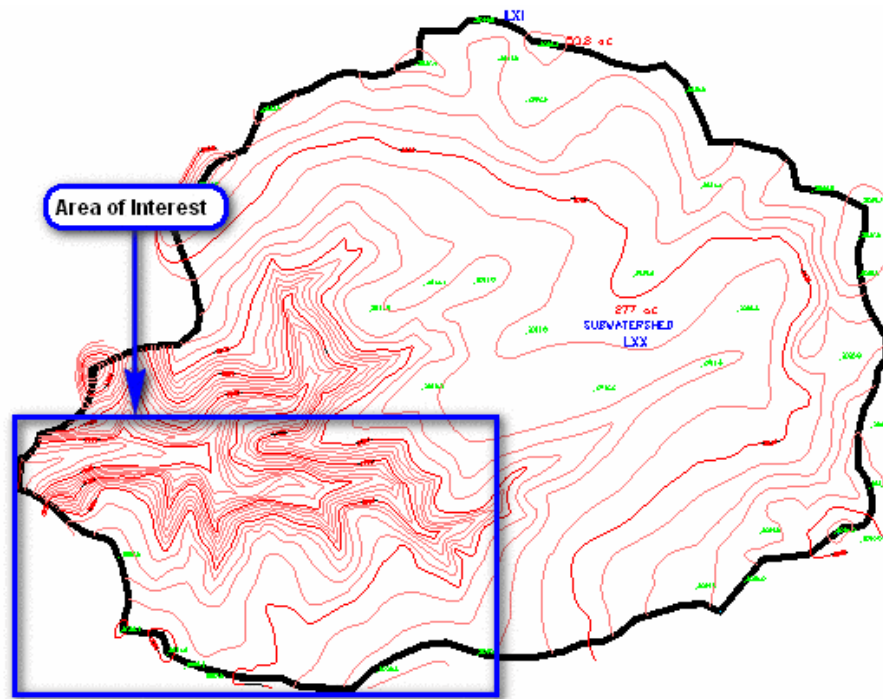
[FILter]/<Select entities>: type ALL and press Enter twice

A "TIN" file is written that we'll use again and again. Let's zoom in closer to our desired area of interest. Issue the View – **Window** command. When prompted:

Zoom: [In/Out/All/Center/Dynamic/Extents/Left/Previous/Right/Window/Entity/Scale]<Scale (nX/nXP)>: _w

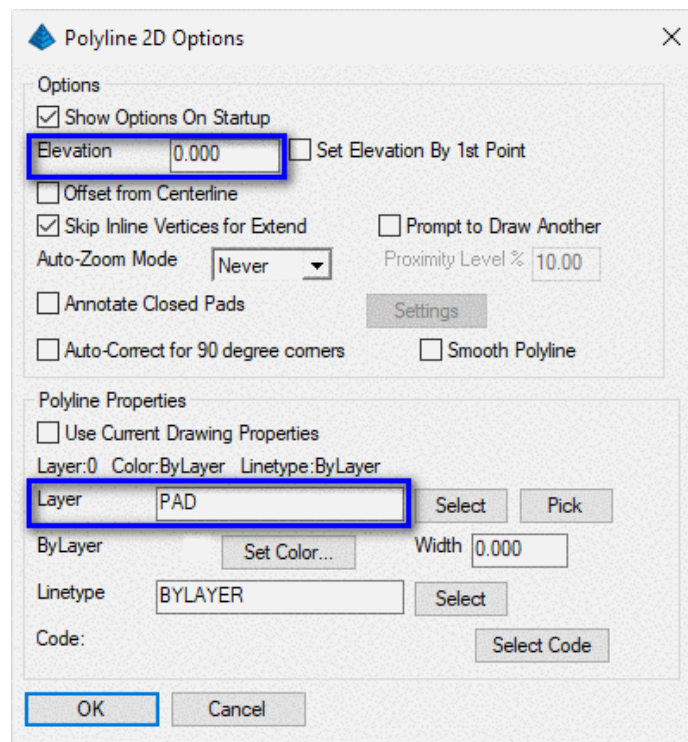
First corner: pick the lower left location of the area of interest shown below

Opposite corner: pick the upper right location of the area of interest shown below



NOTE: Many of the routines below can also be accomplished with the Carlson Survey and/or Construction modules but our focus herein will presume use of the Civil module.

- Tool 1:** The most basic use of a 2D polyline is for the creation of flat-bottomed pits and flat "building pads" at any desired elevation. Let's start with a "free-form" building pad at a specified elevation. Issue the Draw – 2D Polyline command to display a dialog box shown below:



Set the values as shown above making particular note of the highlighted items. Click **OK** when ready. When prompted:

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: pick a first pad location as shown below

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: pick the second pad location

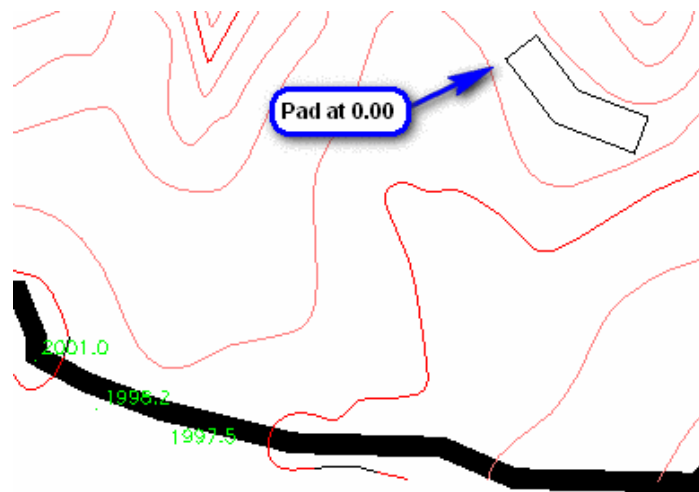
[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: pick the third pad location

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: pick the fourth pad location

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: pick the fifth pad location

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: pick the sixth pad location

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: press **C** to mathematically close the pad



Let's sketch out a simple rectangular pond bottom. Issue the Draw – **Rectangle** (key-in is *rectang*) and when prompted:

[Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width]/<Select first corner of rectangle>: type **end** (for an endpoint snap) and press Enter

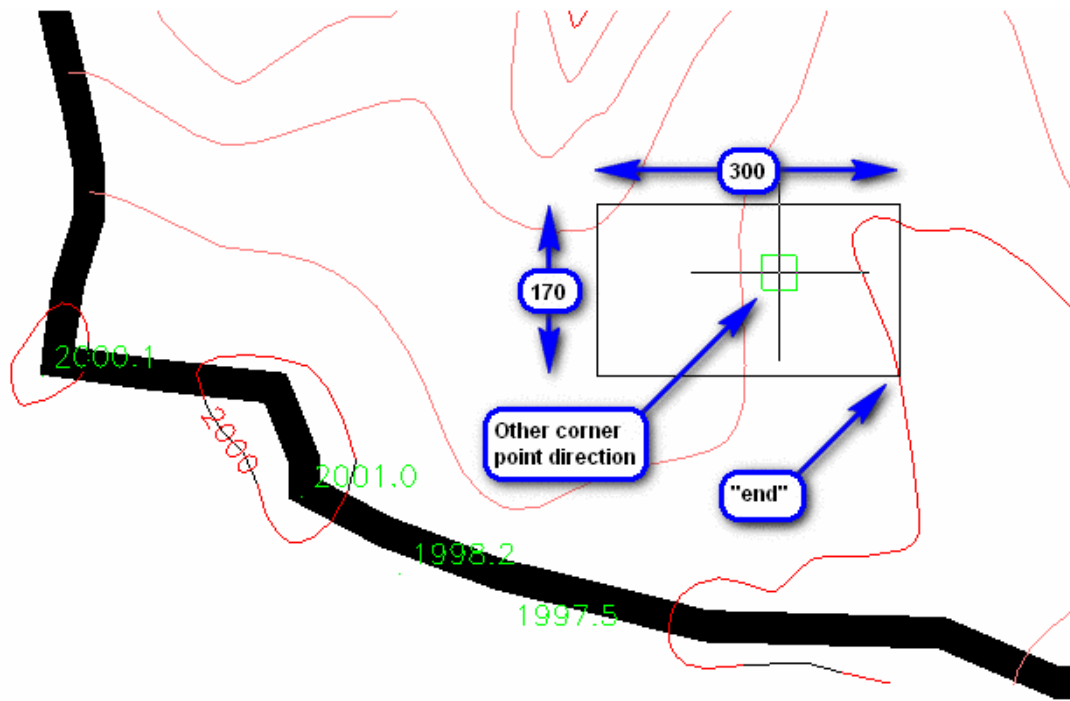
Snap to END of: pick the endpoint on the contour as shown below

Specify other corner point or [Area/Dimensions/Rotation]: type **D** and press Enter

Enter rectangle length <10.0000>: type **300** and press Enter

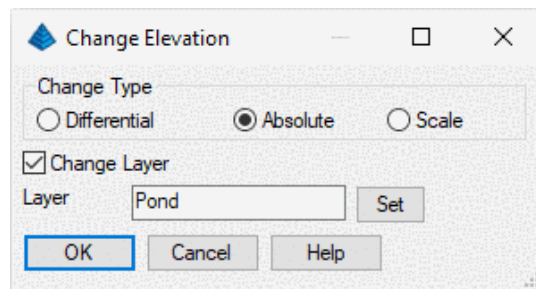
Enter rectangle width <10.0000>: type **170** and press Enter

Specify other corner point or [Area/Dimensions/Rotation]: pick a location northwest of the initial point



NOTE: Unlike the **2D Polyline** command discussed above which has added functionality permitting you to establish the elevation for the 2D polyline, the **Rectangle** command will inherit the elevation of the first pick point (a contour in this case).

Let's set the pond bottom polyline to a better target elevation. Issue the Edit – Change – Elevations command to display the dialog box as shown below:



Set the values as shown above and click **OK** when ready. When prompted:

Select/⟨Enter Elevation <0.0000>⟩: if the default value is other than 0, type 0 and press Enter

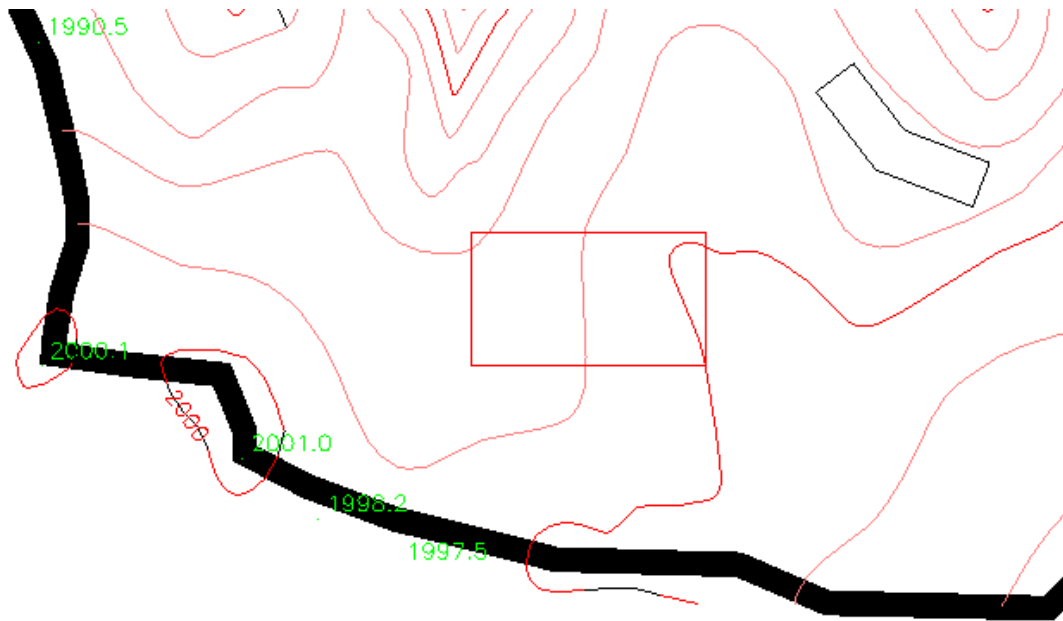
Select Entities for Elevation Change.

[FILter]/⟨Select entities⟩: pick the pond bottom polyline just drawn

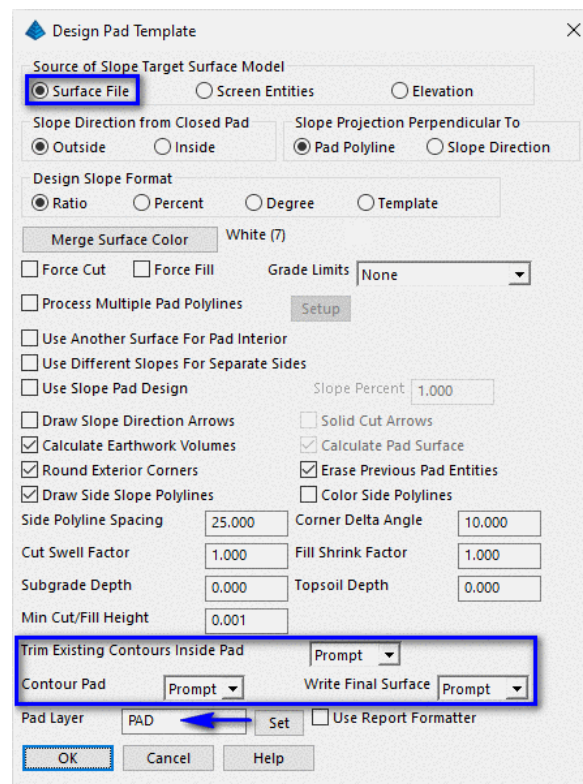
[FILter]/⟨Select entities⟩: press Enter

Select/⟨Input another Elevation (Enter to end)⟩: press Enter

Our resulting polylines should resemble that shown below:



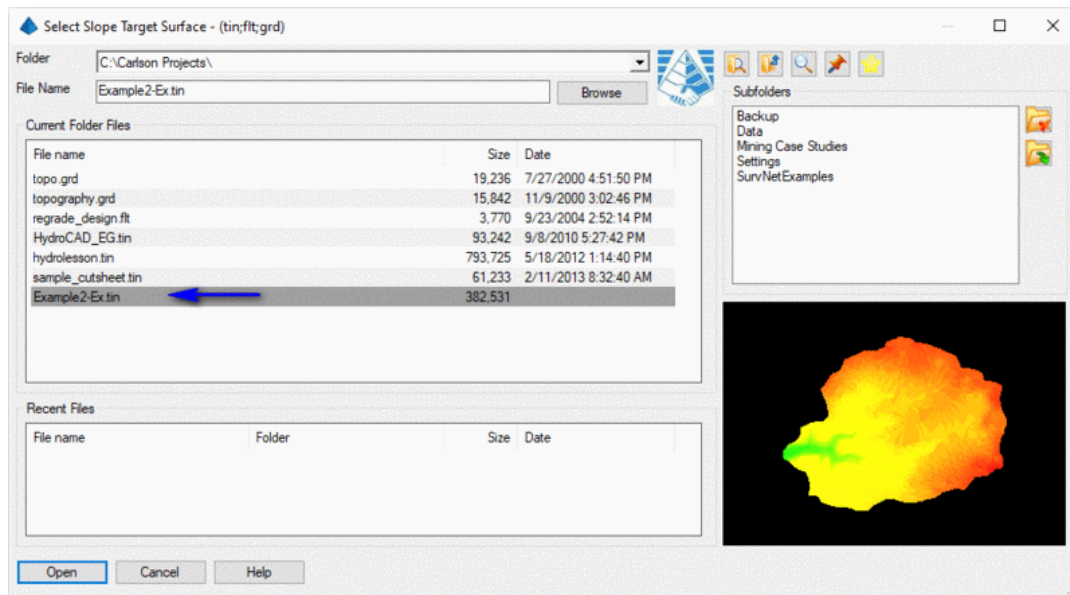
4. **Tool 2:** With the **pad** and **pond** polylines drawn, we'd next like to project slopes to tie back to the *existing ground surface* to produce volumes. Issue the Surface – Design Pad Template routine. A dialog box similar to that shown below appears:



Set the values as shown above and click **OK** when ready. When prompted:

Pick pad polyline: pick the non-rectangular pad polyline (the first that was drawn)

A dialog box similar to that shown below appears. Select the surface model file created earlier and click **Open** when ready. Prompting will resume:



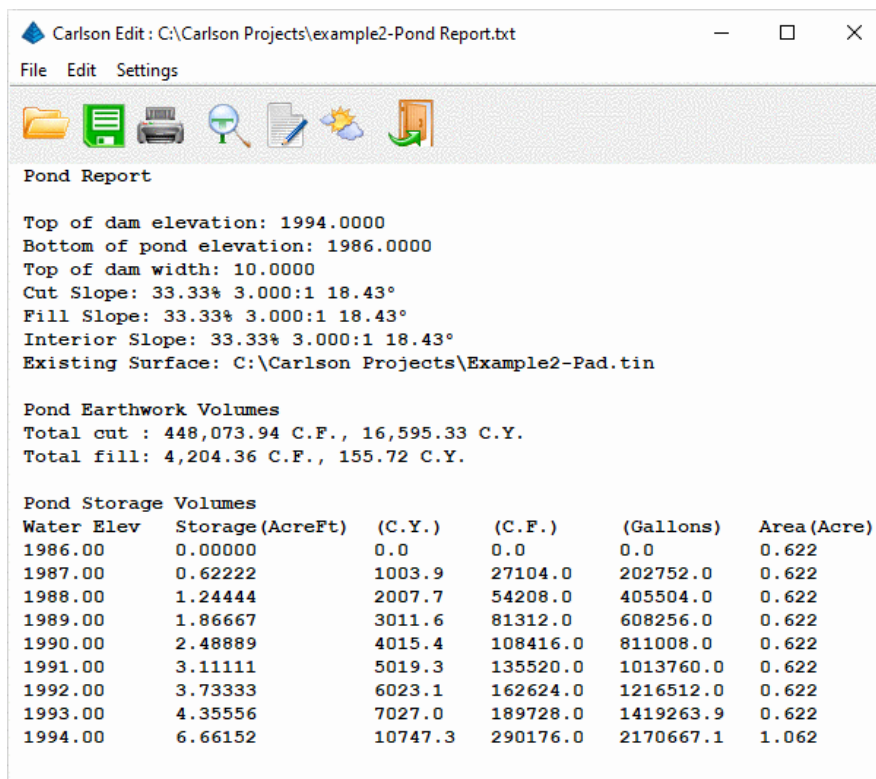
Fill outslope ratio <3.000>: press Enter

Cut outslope ratio <3.000>: press Enter

Range of existing elevations along pad: 1992.33 to 1995.37 (your values may vary)

Pad elevation <1992.33>: type 1993.5 and press Enter (we'll try to target a balanced Cut vs. Fill)

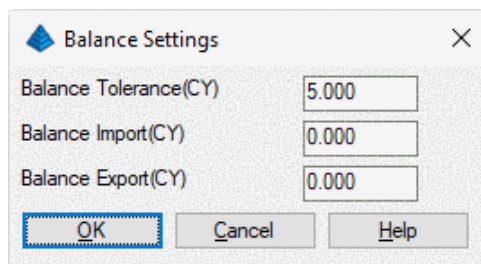
Slope projections from the source polyline are projected to the surface model file and preliminary results are displayed in the Standard Report Viewer similar to that shown below:



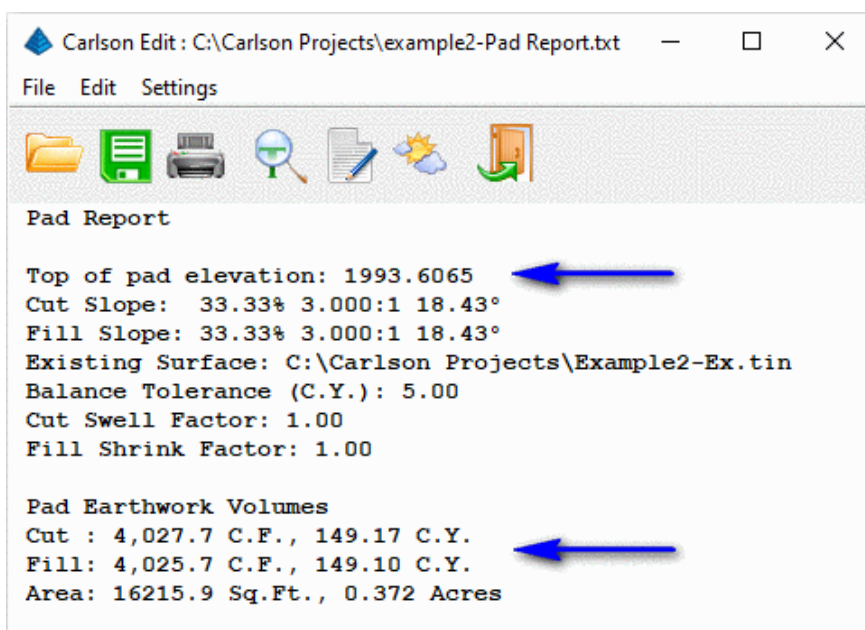
Review the report and notice the *Cut Volume* will be different from that of the *Fill Volume*. Click the **Exit** (Doorway) button to dismiss the report. Prompting resumes:

Adjust parameters and redesign pad [Yes/<No>]? type Y and press Enter
Balance cut/fill [Yes/<No>]? type Y and press Enter

A dialog box similar to that appears. Set the values as shown below and click **OK** when ready:



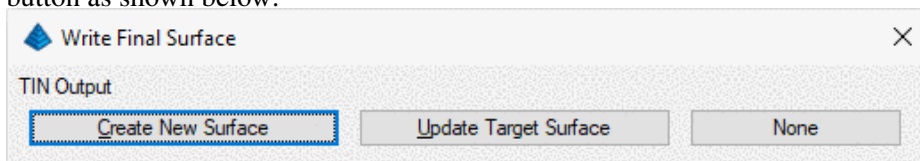
The solution is iterated until the balance tolerance is achieved and an updated report similar to that shown below appears:



Let's accept the solution. Click the **Exit** (Doorway) button to dismiss the report. Prompting resumes:

Adjust parameters and redesign pad [Yes/<No>]? type N and press Enter

A dialog box as shown below appears, prompting us with an option to commit the **Final** surface to an external *triangulation* file which could ultimately be used again in the future. Click the **Create New Surface** button as shown below:



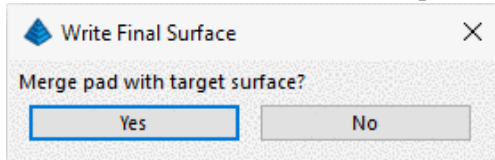
A secondary dialog box as shown below appears prompting if we'd like to merge the **Final** surface into the **Existing** surface file produced earlier.

NOTE:

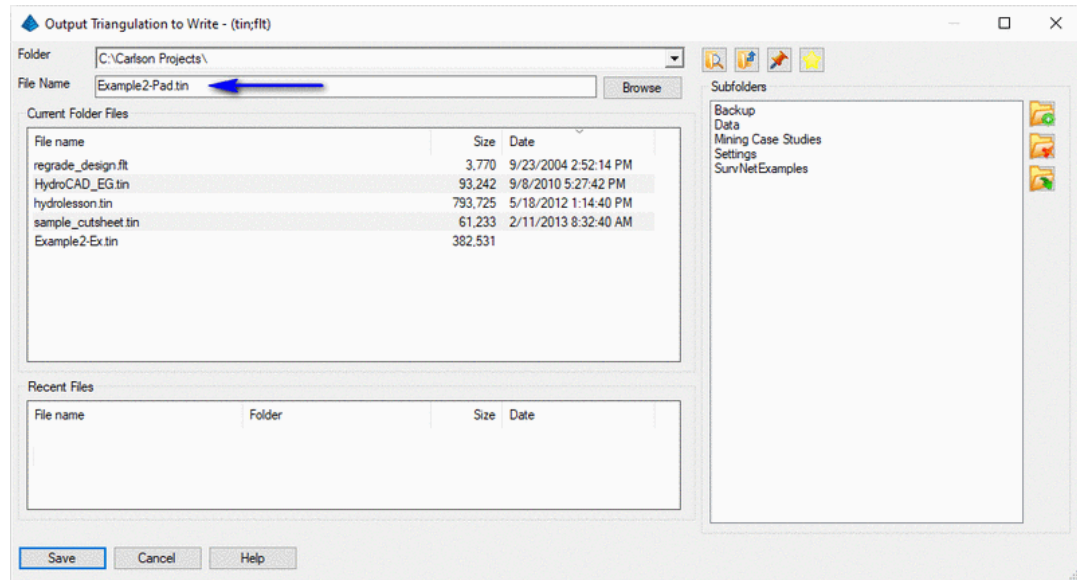
- Clicking **Yes** takes the *existing* surface model and "stitches in" the pad polylines to form a composite surface model which is typically given a new name (thereby preserving the original *existing* surface).

- Clicking **No** takes the pad polylines to form an individual model of the design features which is typically given a new name (thereby preserving the original *existing* surface).

Click the **Yes** button as this will help us later:



You are then prompted for the name of a *triangulation* (*.tin) to write. Specify the value as shown below and click the **Save** button when ready:



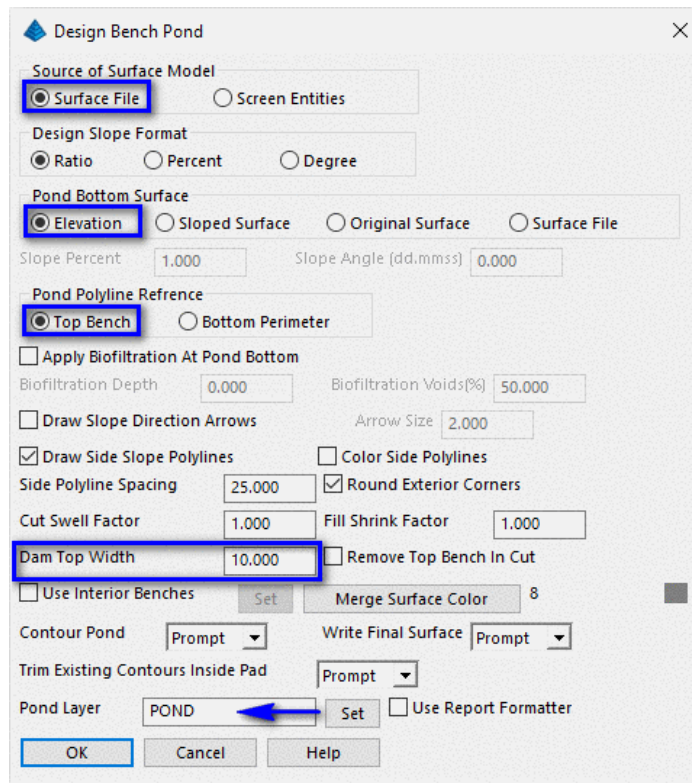
Prompting resumes:

Trim existing contours inside pad perimeter [Yes/<No>]? type N and press Enter

Contour the pad [<Yes>/No]? type N and press Enter

With the pad polyline completed, let's work on the pond.

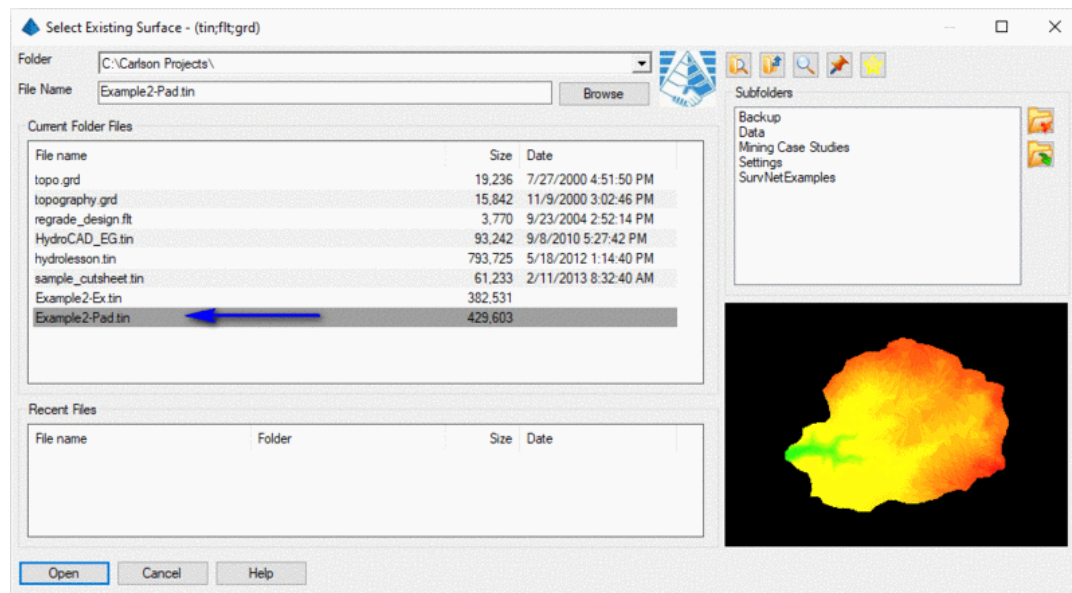
5. **Tool 3:** In certain situations, we might desire a "bench" (or "bermed") pond. Issue the Surface – Design Pond – Design Bench Pond command to display a dialog box similar to that shown below:



Set the values as shown above paying particular attention to the highlighted items and click **OK** when ready. When prompted:

Pick top of pond polyline: pick the rectangular pond polyline

A dialog box appears prompting for the **Existing** surface. Specify the surface model we just made as illustrated below and click **Open** when ready:



Prompting resumes:

Fill outslope ratio <3.000>: press Enter

Cut outslope ratio <3.000>: press Enter

Interior slope ratio <3.000>: press Enter

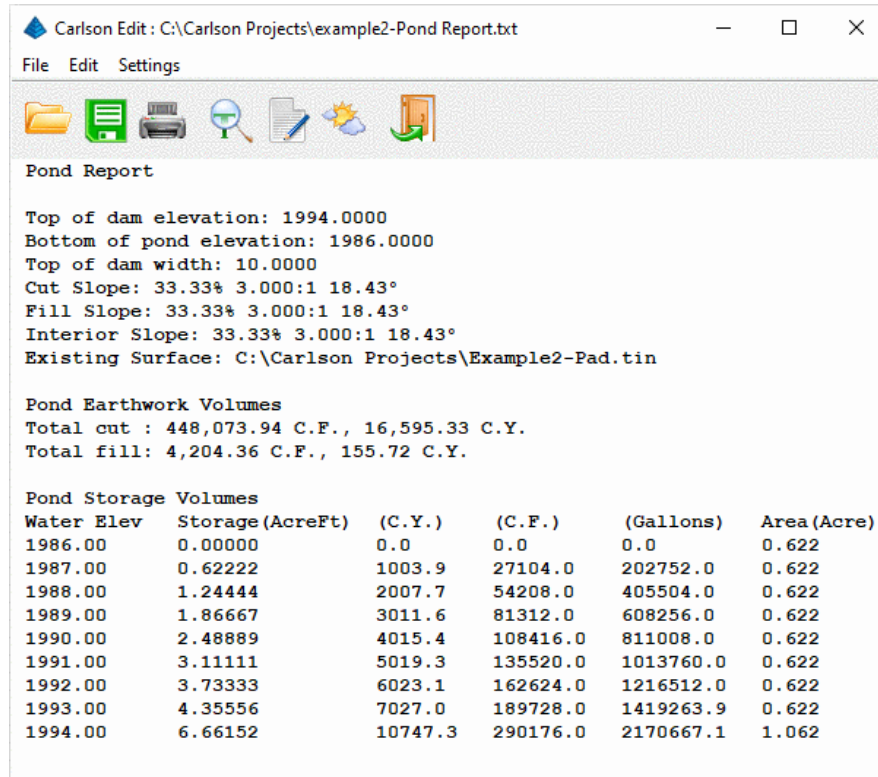
Range of existing elevations along dam top: 1991.51 to 2000.56 (your results may vary)

Top of bank elevation <1991.51>: type 1994 and press Enter

Pond bottom elevation: type 1986 and press Enter

Method to specify storage elevations [<Automatic>/Interval/Manual**]?** type A and press Enter

Slope projections from the source polyline are projected to the surface model file and preliminary results are displayed in the **Standard Report Viewer** similar to that shown below:



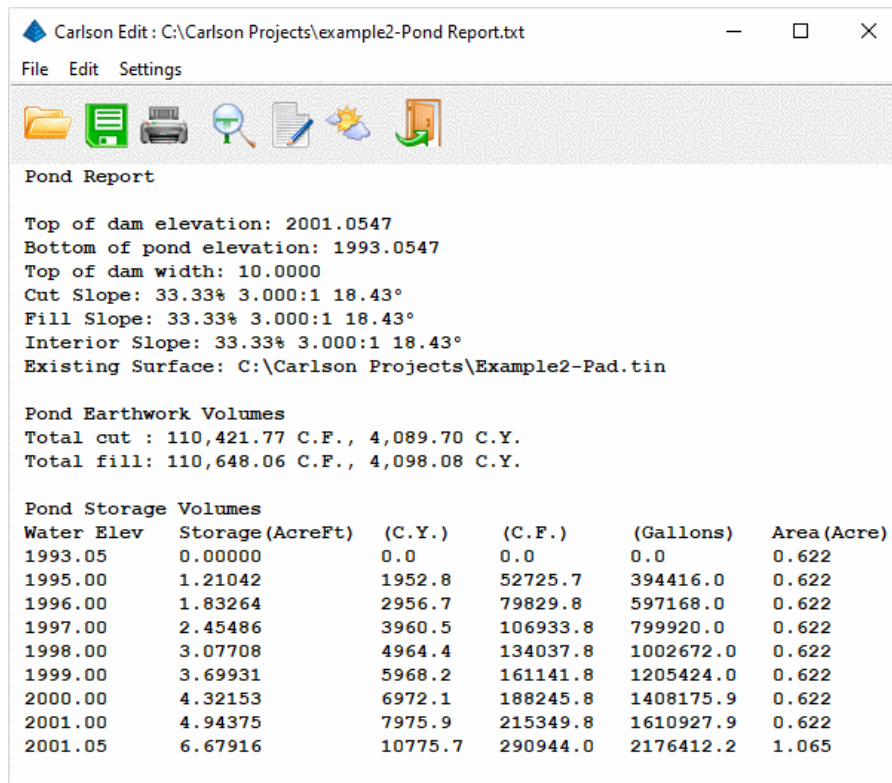
Review the report and notice the *Cut Volume* will be different from that of the *Fill Volume*. Click the **Exit** (Doorway) button to dismiss the report. Prompting resumes:

Adjust parameters and redesign pond [Yes/<No>]? type Y and press Enter

Balance cut/fill [Yes/<No>]? type Y and press Enter

Method to specify storage elevations [<Automatic>/Interval/Manual**]?** type A and press Enter

The solution is iterated until the balance tolerance is achieved and an updated report similar to that shown belows appears:



Let's accept the solution. Click the **Exit (Doorway)** button to dismiss the report. Prompting resumes:

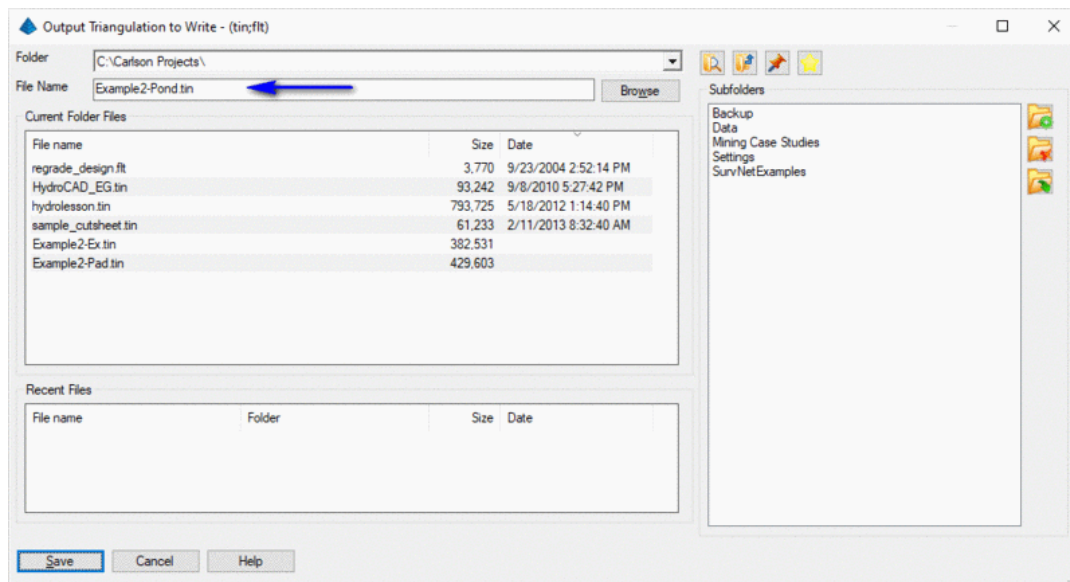
Adjust parameters and redesign pond [Yes/<No>]? type N and press Enter

Write stage-storage file [Yes/<No>]? type N and press Enter

Write final surface to triangulation file [Yes/<No>]? type Y and press Enter

Merge pond with target surface [<Yes>/No]? type Y and press Enter

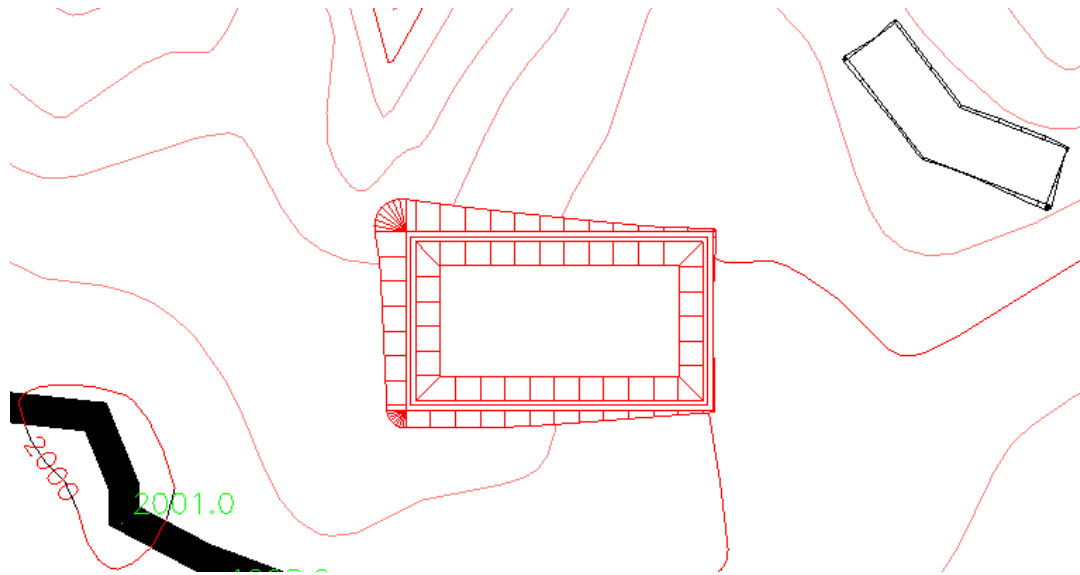
You are prompted for an output file. Specify the value as shown below and click the **Save** button when ready:



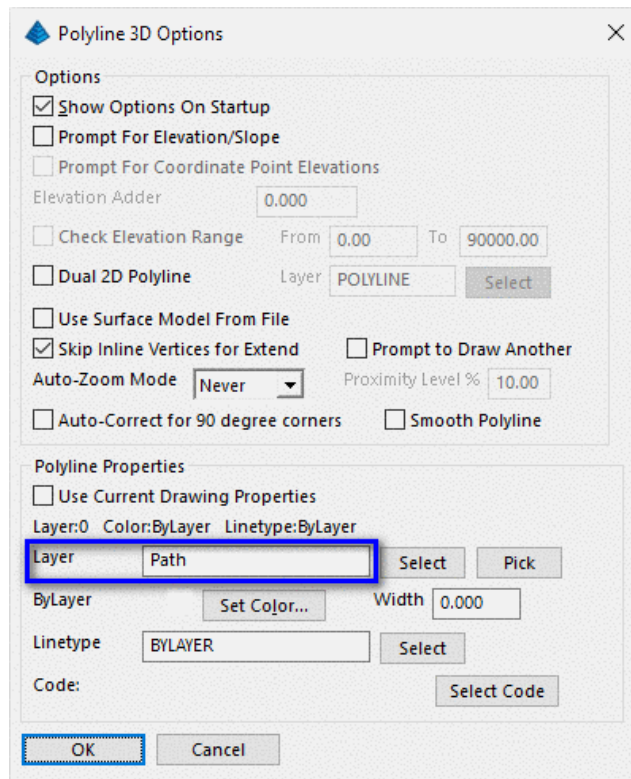
Prompting resumes:

Trim existing contours inside pad perimeter [Yes/<No>]? type Y and press Enter
Retain trimmed polyline segments [Yes/<No>]? type N and press Enter
Contour the pond [<Yes>/No]? type N and press Enter

The result is a partial Fill, partial Cut Bench Pond as shown below:



- Tool 4:** With the two primary structures designed, let's try a graded path between the two with an interior angle point that allows each end to be perpendicular to its intended structure. Issue the Draw – 3D Polyline command to display a dialog box similar to that shown below:



Set the values as shown above paying particular attention to the highlighted information and click **OK** when ready. When prompted (using the illustration below as a guide):

[Continue/Extend/Follow/Options/<Pick point or point numbers>]:type END and press Enter

Snap to END of: pick northeast rectangular corner of pond structure

[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: press **F8** function key to enable "ortho" mode ('_ORTHOgonal)

ORTHOMODE is currently off: [ON/OFF/Toggle]<Off>: **_T**

(ORTHOMODE is currently On):

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: pick the hinge location then press **F8** function key to disable "ortho" mode

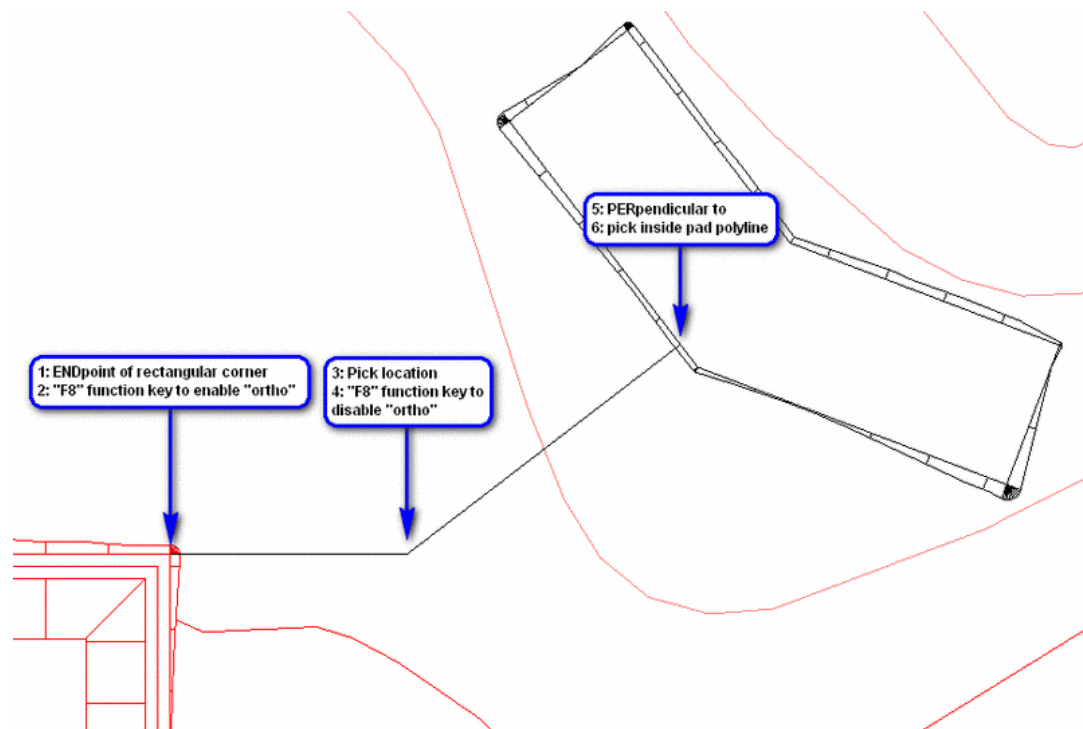
ORTHOMODE is currently on: [ON/OFF/Toggle]<On>: **_T**

(ORTHOMODE is currently Off): type **PER** and press Enter

Snap to PER of: pick inside pad polyline

Warning: last 3D polyline point at 0 elevation

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: press Enter

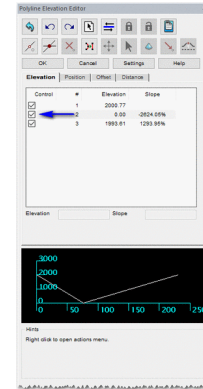


This is an obvious tool for creating design considerations. Our 3D polyline has one problem... the hinge vertex may not have the elevation we desire. Let's fix this.

NOTE: *Bonus Tool!* Issue the 3D Data – Polyline Elevation Editor command. When prompted:

Select a polyline to edit: pick the polyline you just drew

Before Edit

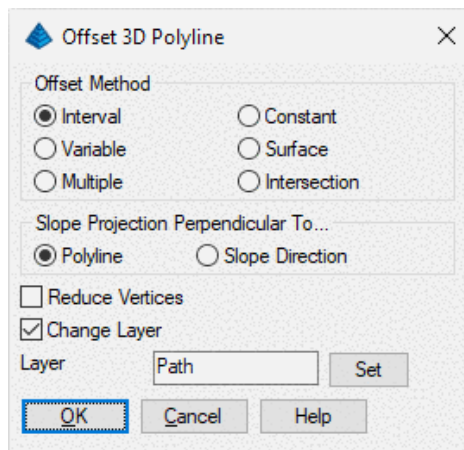


A "docked" dialog box similar to that shown below appears:

Polyline Elevation Editor

You'll note that each of the three vertex locations are toggled as **Control** locations. Presuming your snap locations were correct, vertex 2 may very well be at elevation 0. Clear its **Control** toggle so its elevation is interpolated from the two surrounding **Control** locations. Experiment with this "Swiss Army Knife" tool with the end goal of a uniform grade between the two structures. Click **OK** when ready.

- Tool 5:** We'd like to offset this graded polyline by the same width as the **Bench Pond Width** as created earlier. Issue the Edit – 3D Polyline Utilities – Offset 3D Polyline to display a dialog box similar to that shown below:



Set the values as shown above and click **OK** when ready. When prompted:

Percent/Ratio/<Horizontal offset amount>: type 10 and press Enter

Select/<Enter the common elevation <0.0000>: press Enter

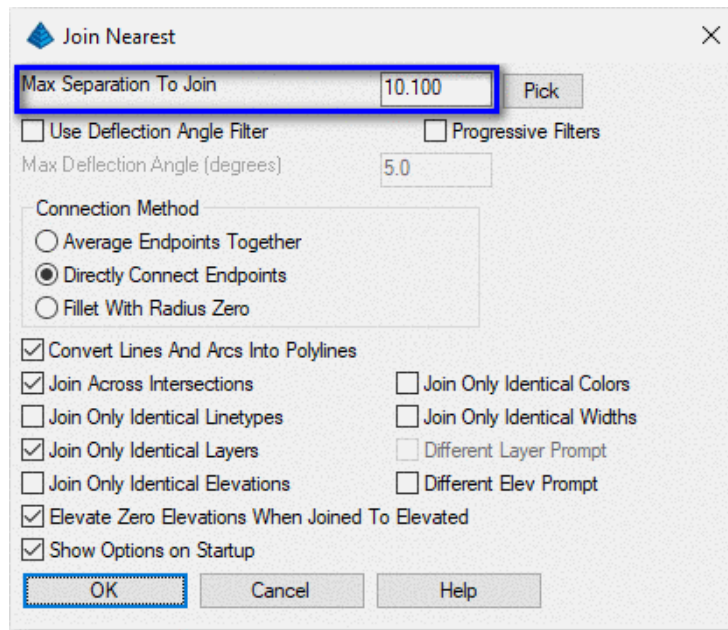
Select a polyline to offset: pick the polyline you just edited

Select side to offset [Both]: pick to the South of the polyline

Select a polyline to offset (Enter to end): press Enter

We now have a 3D-parallel offset of the initial polyline.

- Tool 6:** We'd like to connect these two polylines together to form one closed region. Issue the Edit – Join Nearest command to display a dialog box similar to that shown below:



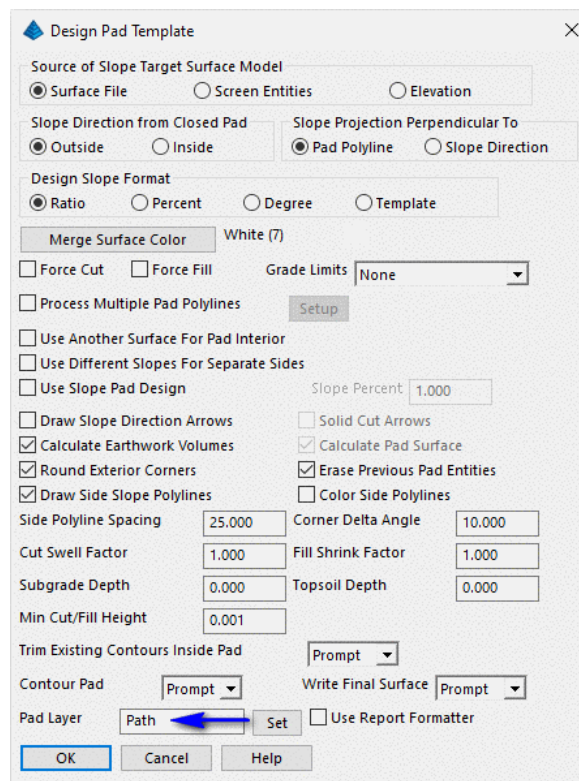
Note the **Max Separation to Join** value. Common practice is to pick the distance to span and then increase slightly to ensure the desired items will join. Set the values as shown above and click **OK** when ready. When prompted:

Select lines, arcs and unclosed polylines to join [Settings].

Select entities: pick the two polylines

Select entities: press Enter

Now we have a 3D polyline closed perimeter, which can act as a pad. Re-issue the Surface – **Design Pad Template** command to display a dialog box similar to that shown below:



Set the values as shown above and click **OK** when ready. When prompted:

Pick pad polyline: pick the closed path polyline

Select Slope Target Surface (dialog): select the surface model recently created in the Bench Pond discussion and click **Open**

Fill outslope ratio <3.000>: press Enter

Cut outslope ratio <3.000>: type 5 and press Enter

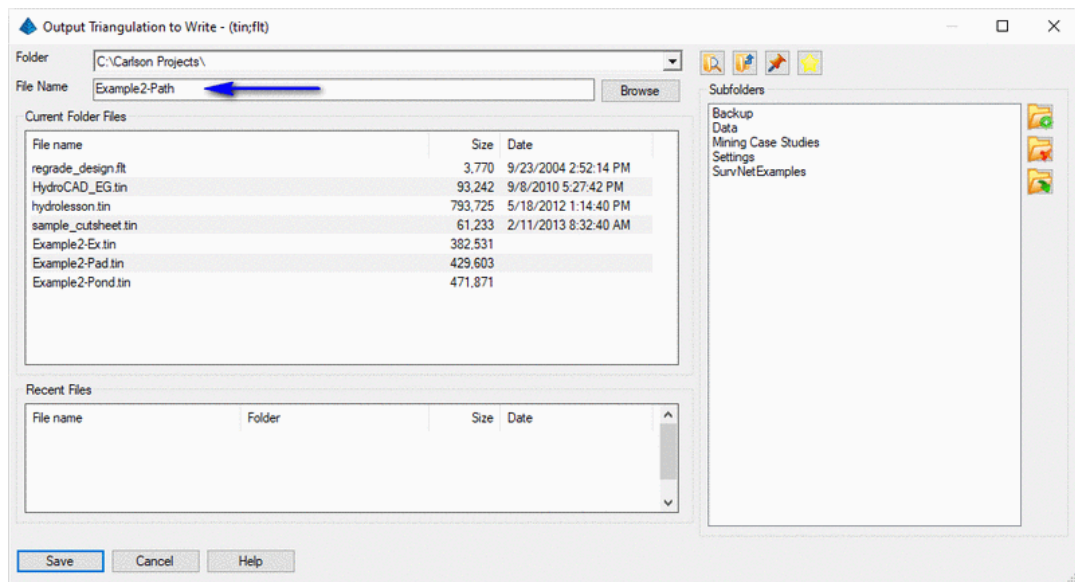
Slope projections from the source polyline are projected to the surface model file and preliminary results are displayed. Review the report and click the **Exit** (Doorway) button to dismiss the report. Prompting resumes:

Adjust parameters and redesign pad [Yes/<No>]?: type N and press Enter

Write Final Surface (dialog): click **Create New Surface**

Merge Final Surface (dialog): click **Yes**

Write Triangulation Surface (dialog): set the value as shown below and click **OK**



Prompting resumes:

Trim existing contours inside pad perimeter [Yes/<No>]?: type Y and press Enter

Retain trimmed polyline segments [Yes/<No>]?: type N and press Enter

Contour the pad [<Yes>/No]?: type N and press Enter

With the path completed, let's work on a new pond structure.

9. **Tool 7:** We'd like to create a berm utilizing the walls of an existing valley to form a *Valley Pond*. We have a small amount of "prep" work:
 - (a) To help distinguish from the **Bench Pond** we created earlier, create a new layer called **POND2** and give it a color of **Magenta** and set it as the *current* layer. Dismiss the layer properties dialog box.
 - (b) Issue the Draw – Arc – 3 Point command and when prompted:

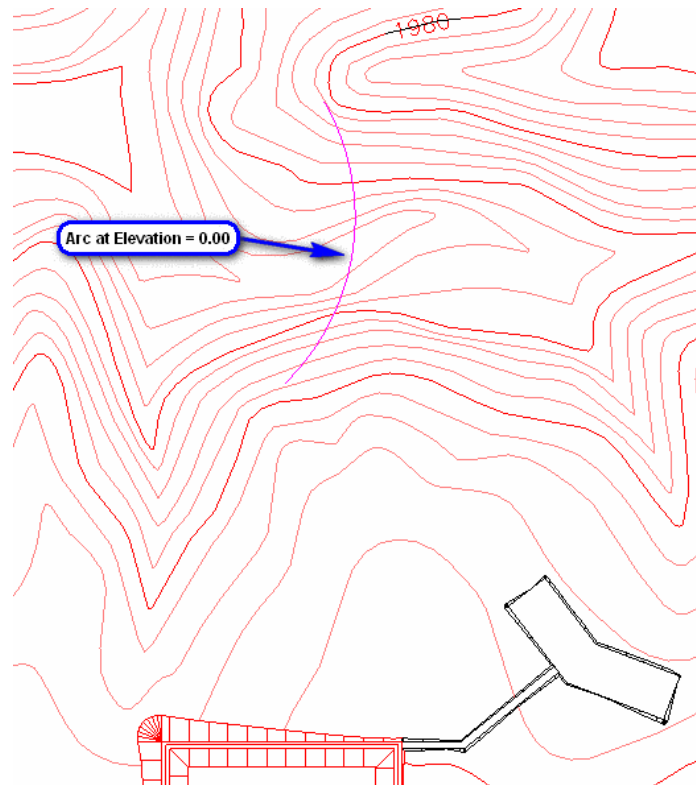
Pick PC point or point numbers: pick a south end of the arc as shown below

Point on arc.

Pick point or point number: pick an interior arc point

PT point.

Pick point or point number: pick a north end of the arc as shown below



(c) Issue the Edit – Polyline Utilities – Entities to Polylines command. When prompted:

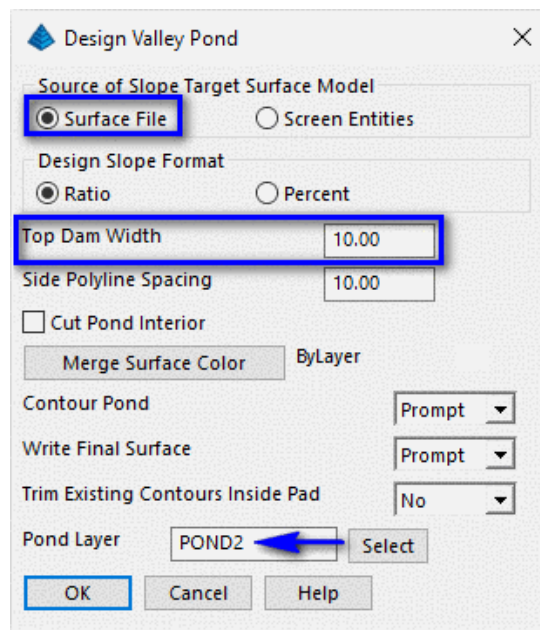
Select lines, arcs, circles, 3DFaces, ellipses, splines, multilines, regions and solids to convert.

[FILter]/<Select entities>: pick the arc you just drew and press Enter twice

Unlike Design Bench Pond, the Design Valley Pond routine requires only a polyline axis line for the center of the dam. The polyline can be a 2-point polyline, or it can have several vertices along its length to create a concave or convex dam structure. The main thing is to "overdraw" the axis polyline, make it ride up on the left and right hillside, well beyond the desired top of dam elevation. This allows the routine to look inward and find the extents of the dam on each hillside, without doing an artificial extension of the polyline. Just "overdo" the length of the axis line and you are in business.

Our curved polyline runs from approximately 1976 on the south side of the valley to approximately 1976 on the north side. It crosses the valley at 1947. Let's decide to put the top of the dam at 1974 even. We will make the dam top 10' wide, with 4:1 downstream and 3:1 upstream slopes.

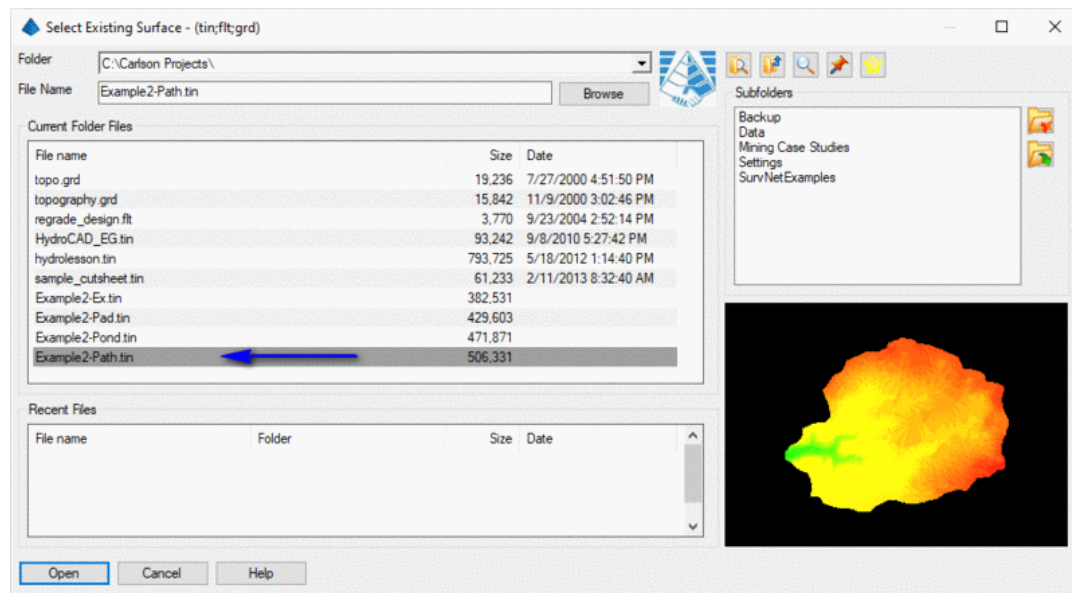
Issue the Surface – Design Pond – Design Valley Pond command. For this pond, use the starting values as shown below and click **OK** when ready:



When prompted:

Pick top of pond polyline: pick the polyline we just created

Select Existing Surface: specify the TIN file shown below and click the **Open** button when ready:



Pick point within pond: pick within the storage area (east of the arc polyline)

Outslope ratio <2.00>: type 4 and press Enter

Interior slope ratio <2.00>: type 3 and press Enter

Top of dam elevation: 1974

Calculate stage-storage values [<Yes>/No**]?** press Enter

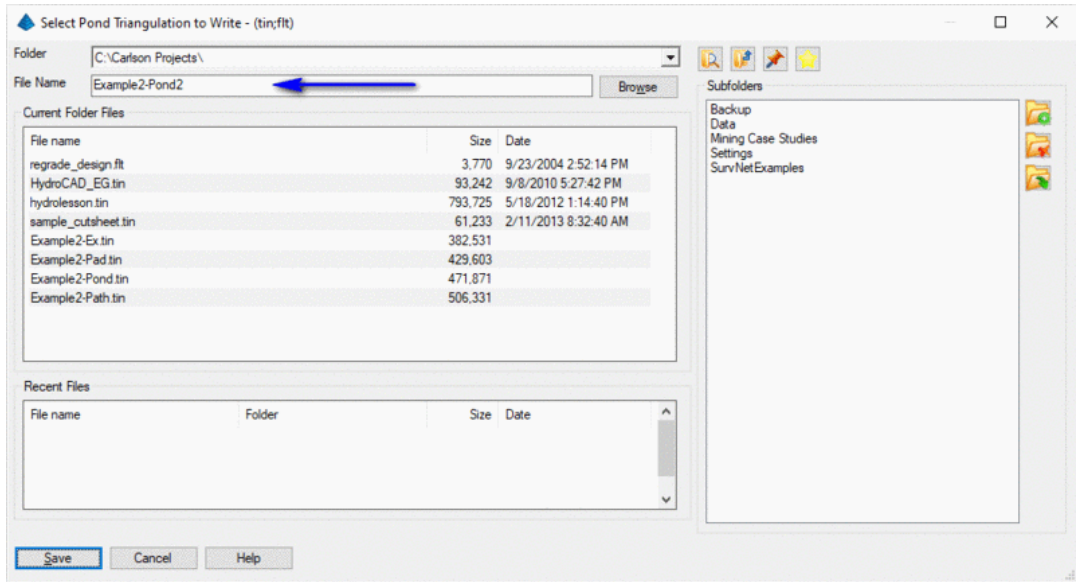
Method to specify storage elevations [<Automatic>/Interval/Manual**]?** specify the Interval (i) method

Starting elevation <1947.68>: 1946 (your values may vary, we'll start just below the lowest elevation)

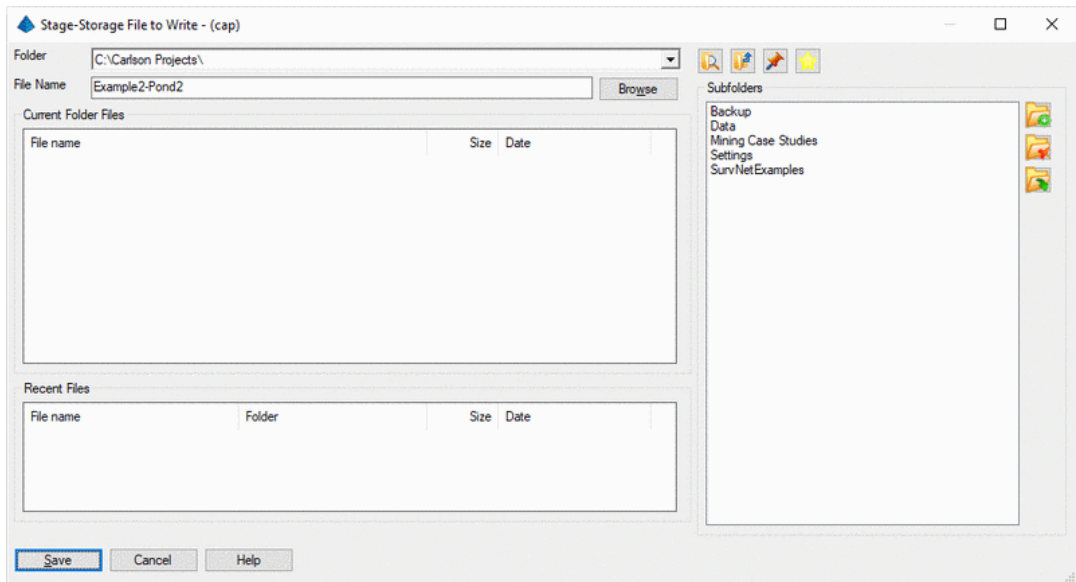
Elevation interval <2.00>: press Enter (review the report, click the **Exit** (doorway) button when ready)

Output triangulation file of final pond surface [Yes/<No>**]?** indicate Yes

Output dam merged with existing or dam only [Merge/<Dam>**]?** indicate M and press Enter (indicate the file name below and click **Save** when ready)

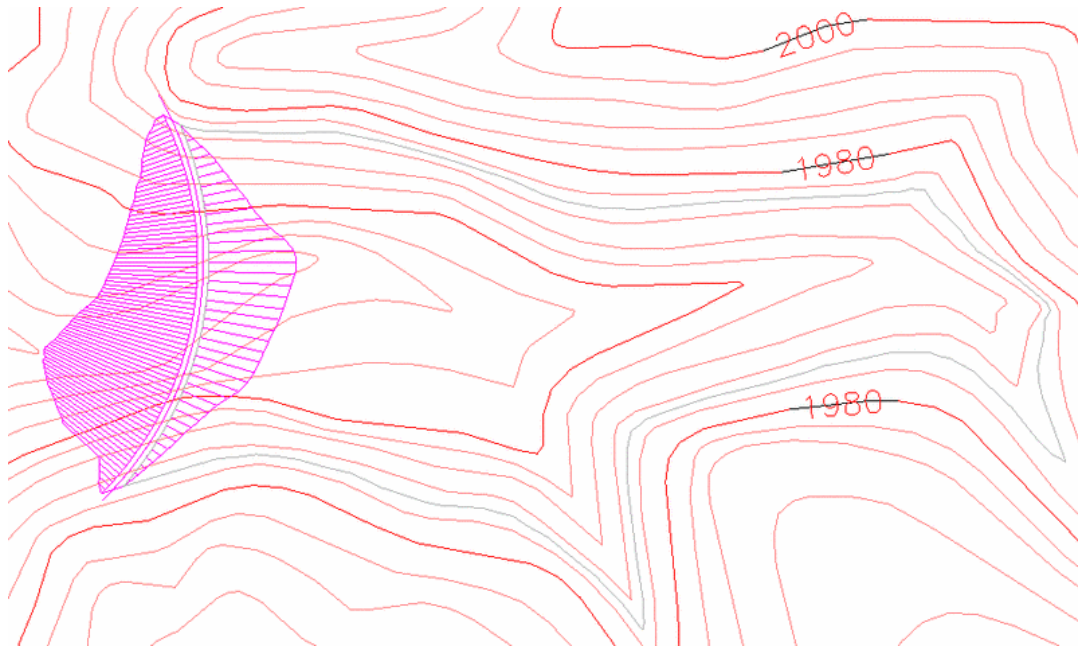


Write stage-storage file [Yes/<No>]? indicate Yes (indicate the file name below and click **Save** when ready)

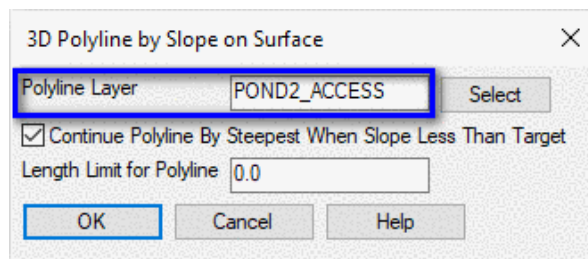


Adjust parameters and redesign pond [Yes/<No>]? press Enter
Contour the pond [<Yes>/No]? indicate No

You should now have a pond that looks like the one in the following image:



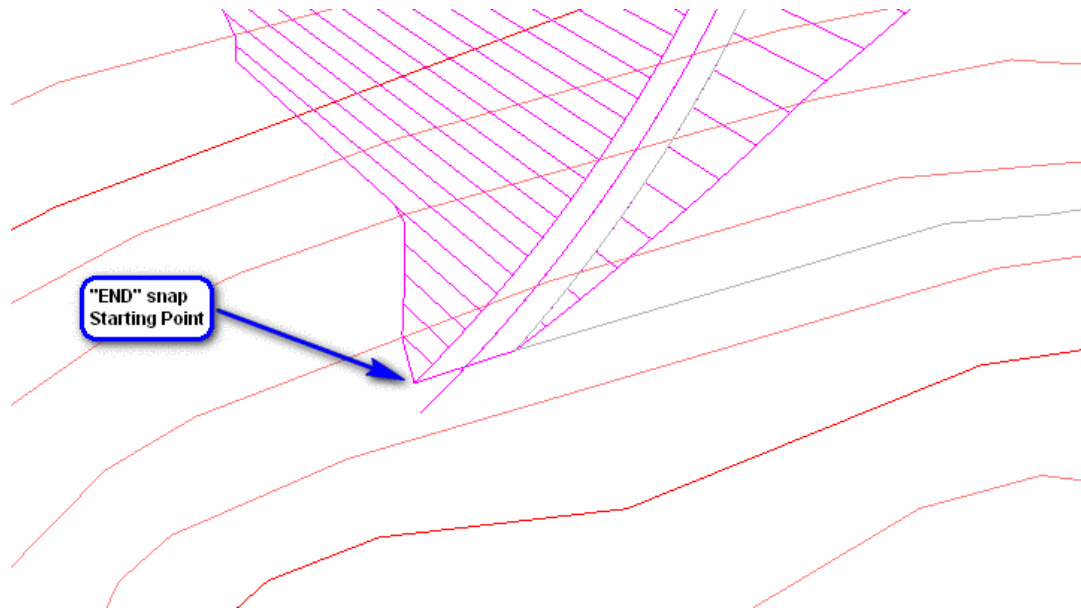
10. **Tool 8:** How would you start at the top of the dam (elevation 1974) and build a road running uphill (or downhill) at a 6% grade? Or, in general, how would you obtain 3D polylines for roads and diversion ditches that follow the terrain at prescribed grades starting at desired points? The answer is to issue the 3D Data – 3D Polyline Utilities – 3D Polyline by Slope on Surface command to display the following dialog box:



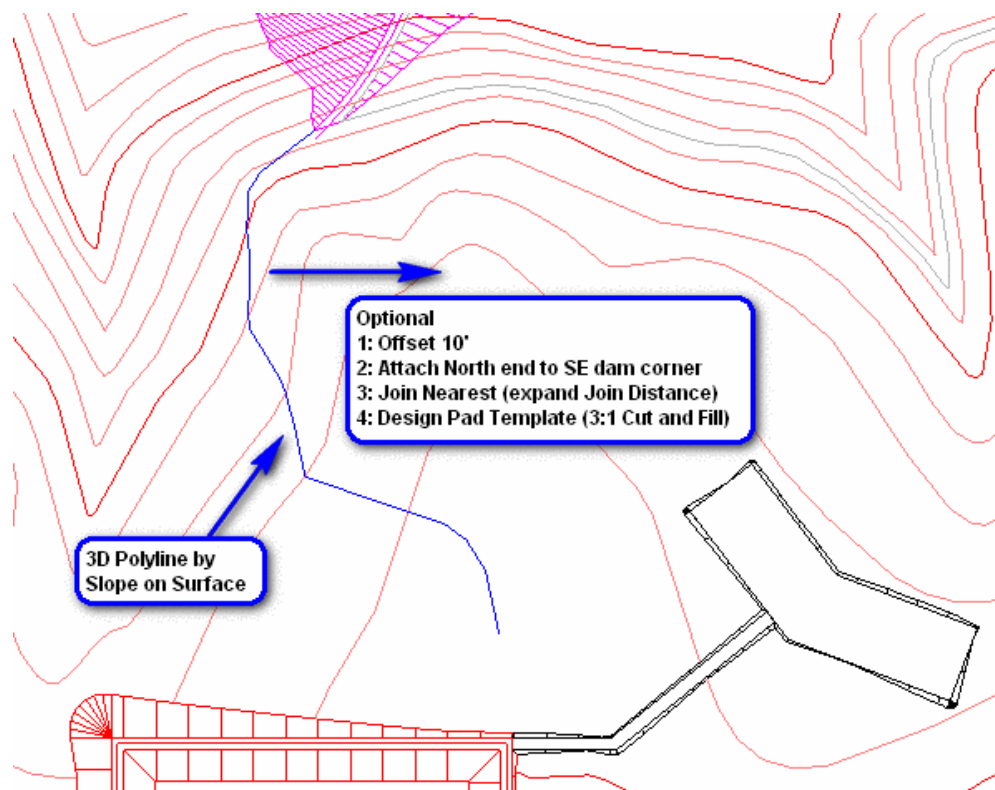
Set the values as shown above and click **OK** when ready. When prompted:

Choose Grid or Triangulation File (dialog): open the surface created in the previous step

Pick origin point of 3D polyline: pick the starting location as shown below

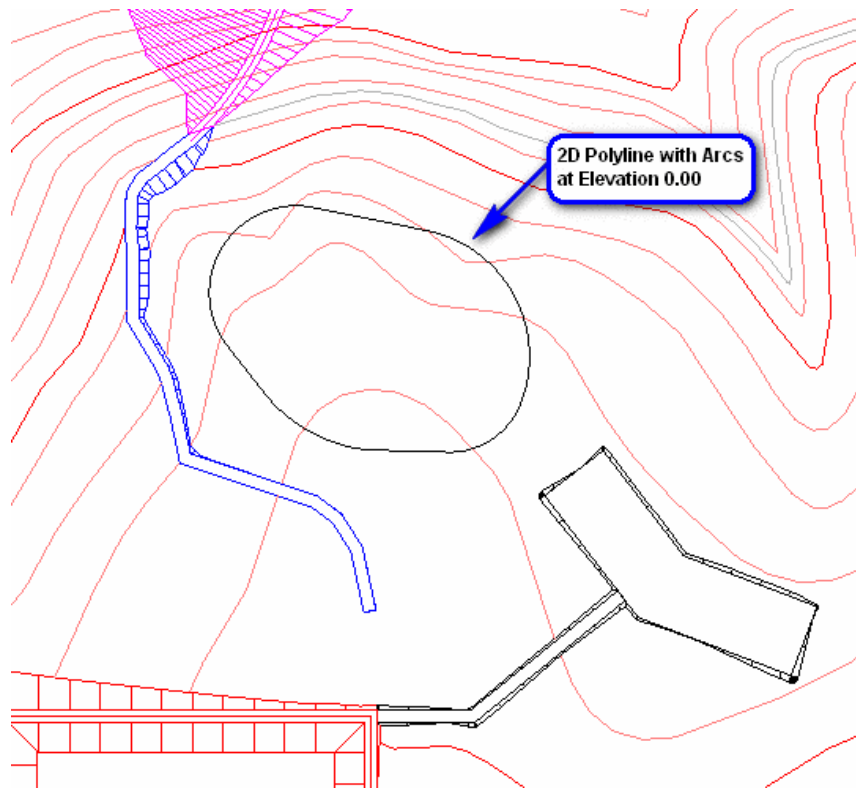


Direction of 3D polyline [\langle Up \rangle / \langle Down \rangle]? type U and press Enter
Direction of 3D polyline facing up slope [\langle Left \rangle / \langle Right \rangle]? type R and press Enter
Slope format [\langle Percent \rangle / \langle Ratio \rangle / \langle Degree \rangle]? press Enter
Enter design slope percent: type 6 and press Enter (a 3D polyline at the cited slope is drawn as illustrated below, your results will vary)
Adjust slope [Yes/ \langle No \rangle]? type N and press Enter
Pick origin point of 3D polyline (Enter to end): press Enter



As optional exercises, construct the **Haul** surface through steps we've already discussed:
 (a) Offset 3D Polyline (offset 10' to the East)

- (b) Perform some minor clean-up (at the north end through the use of grips)
 - (c) Join Nearest (connect the two lines, an expanded Join Distance is warranted if the clean-up work above is performed)
 - (d) Design Pad Template (3:1 Cut and 3:1 Fill slopes, create a surface called *Example2-Haul*)
11. **Tool 9:** In our next example, we wish to create a borrow-pit whose outer perimeter we know and we desire to excavate down to a given elevation with stable side slopes. For the "prep" work, use the image as a guide below to create a **2D polyline** at elevation 0:



NOTE: For this type of shape, it is recommended you use the standard CAD "polyline" (key-in **pline**) with alternating combinations of:

- line
- arc
- line
- follow
- arc
- line
- follow
- (*etc* repeating)
- close

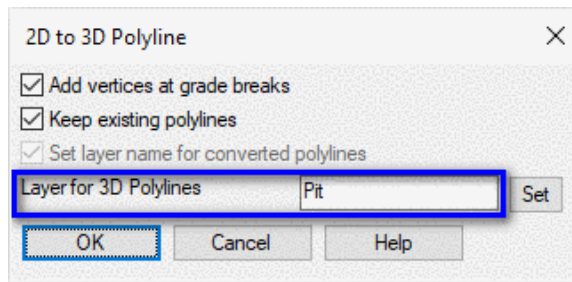
Once complete, issue the 3D Data – 2D to 3D Polyline – By Surface Model command (sometimes called the "drape" command) and when prompted:

Choose Grid or Tmesh File to Process (dialog): open the surface created in the previous step

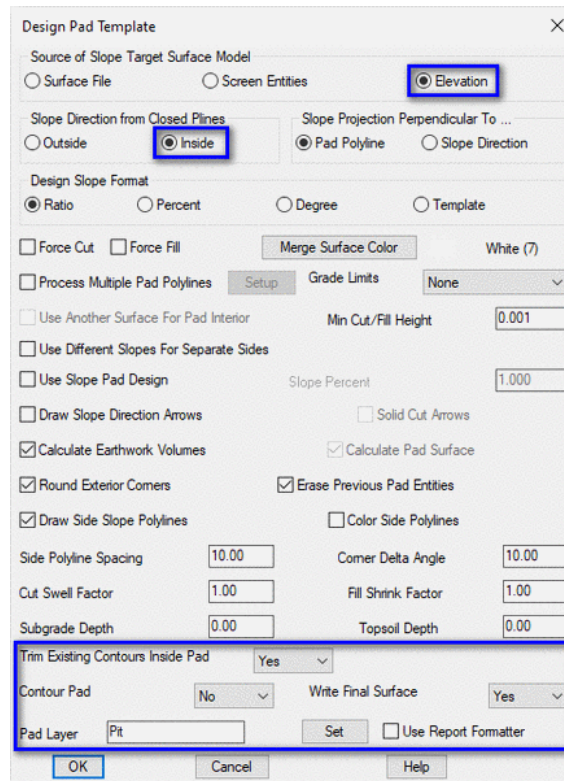
Select polylines and lines to convert.

[FILter]/<Select entities>: pick the 2D polyline

[FILter]/<Select entities>: press Enter



Set the values as shown above and click **OK** when ready. Re-run the Surface – **Design Pad Template** command to display a dialog box similar to that shown below:



Set the values as shown above paying particular attention to the highlighted items and click **OK** when ready. When prompted:

Pick pad polyline: pick the 3D polyline just created

Fill outslope ratio <3.000>: press Enter

Cut outslope ratio <3.000>: press Enter

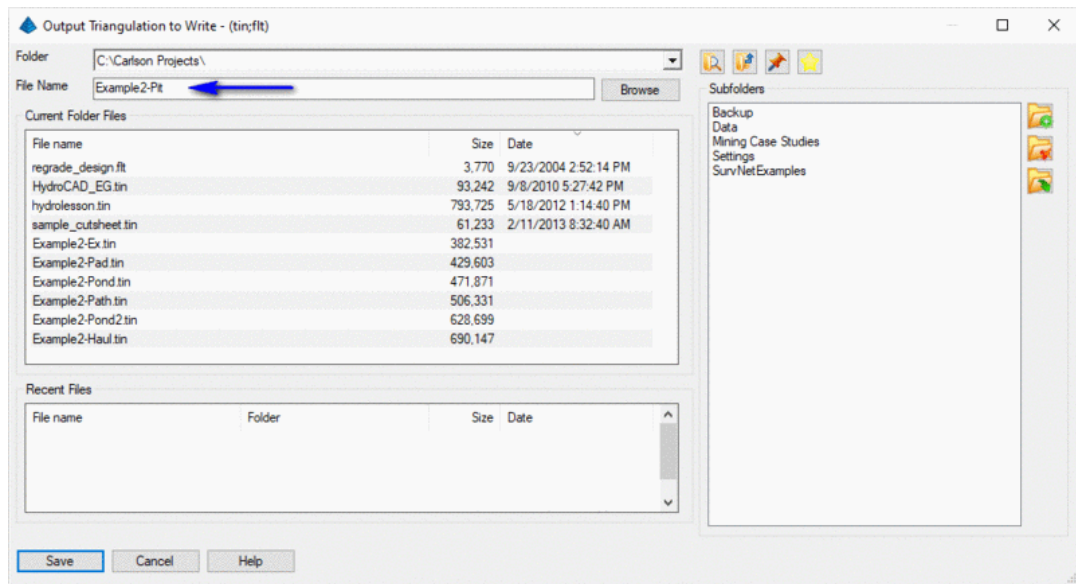
Target surface elevation: type 1976 and press Enter

Select Existing Surface for Volumes (Cancel for None) (dialog): open the surface created in the previous step

Review the **Pad Report** and click the **Exit** (Doorway) button when ready. Prompting resumes:

Adjust parameters and redesign pad [Yes/<No>]? press Enter

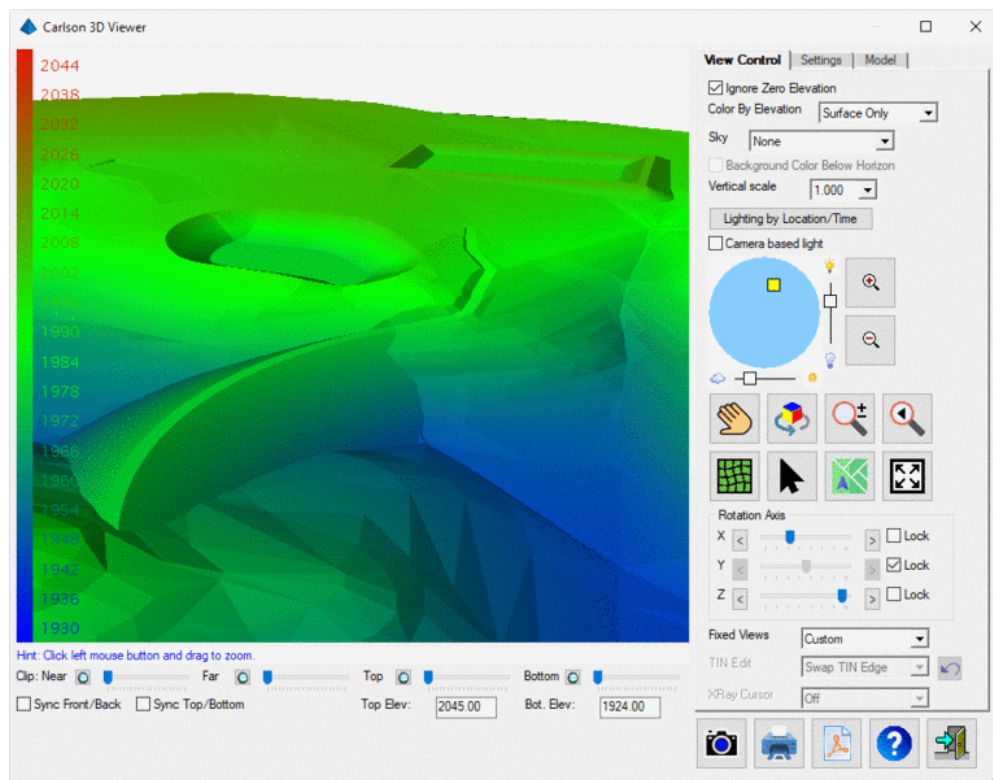
Output Triangulation to Write (dialog): set the value as shown below and click **Save**



Prompting resumes:

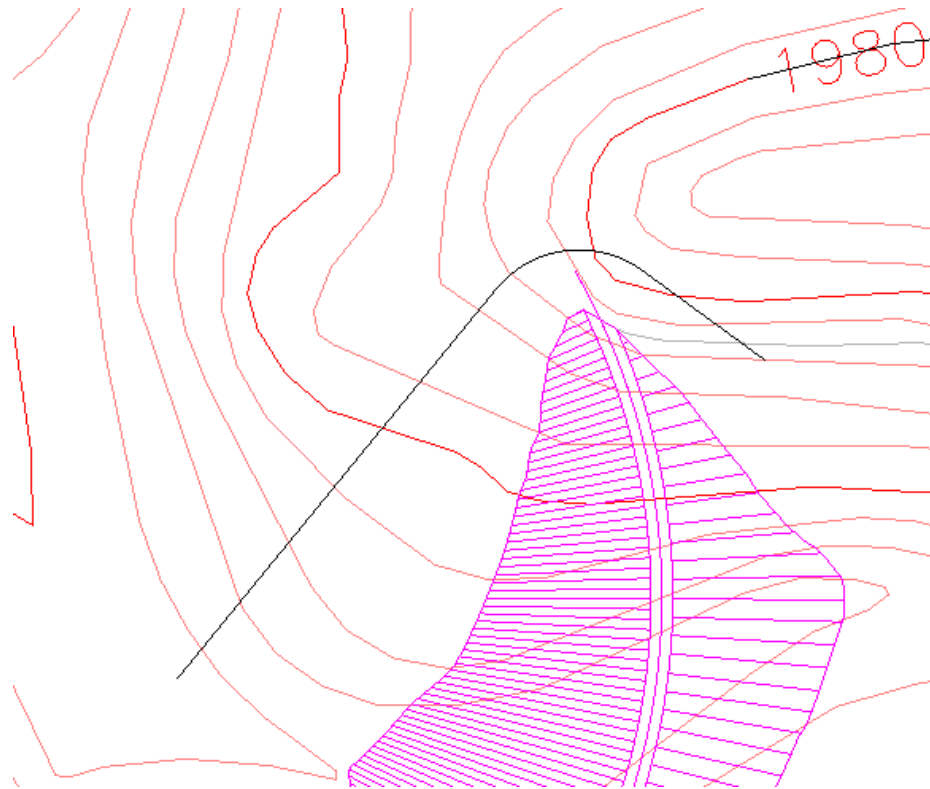
Retain trimmed polyline segments [Yes/<No>]? type N and press Enter

Issue the View – 3D View – Surface File Viewer command to examine the results of the most recently created surface model thus far as illustrated below:



Click the **Exit** (Doorway) button when ready.

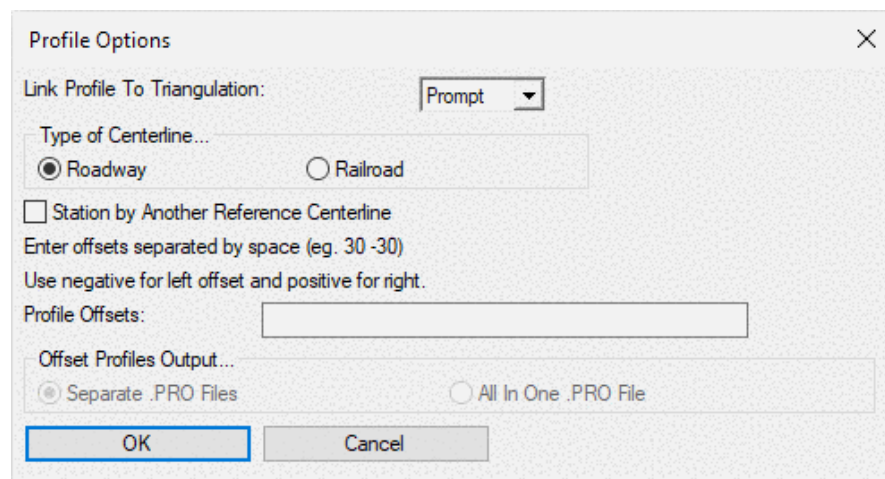
12. **Tool 10:** Our next goal is to build a diversion ditch around the dam. Re-issue the **Draw 2D Polyline** command that starts on the water-side of the valley dam near the 1972 contour (as you may recall, the valley dam has a top elevation of 1974) and curves around into the hill and into the drainage below the dam as shown below:



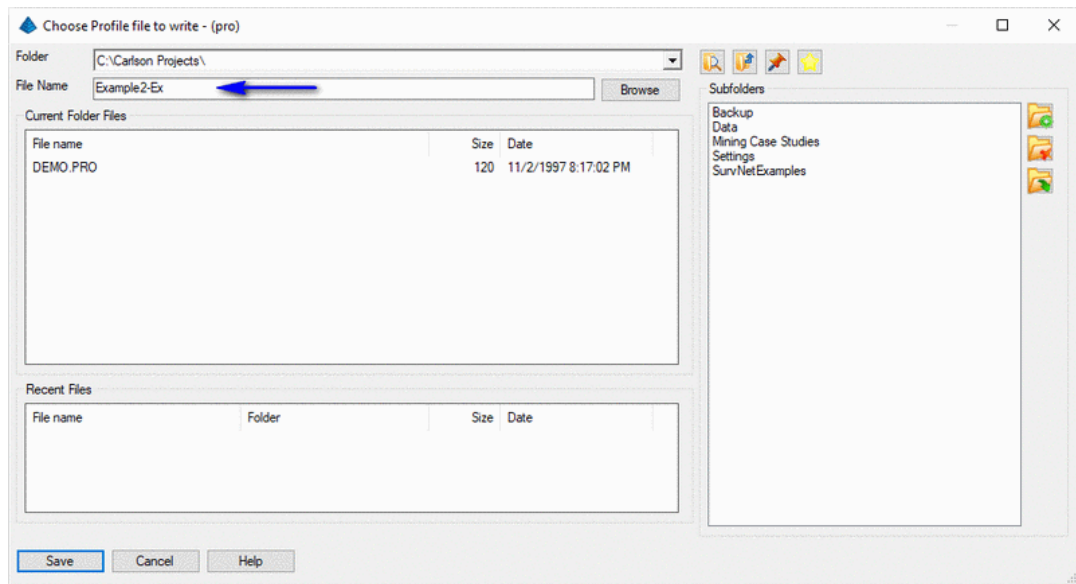
Let's create an *existing ground* ground profile from the TIN so that we can use this profile as a reference for the *design* profile to follow. Issue the Profiles – Create Profile From – Profile from Grid or Triangulation Surface command. When prompted:

Choose Grid or Triangulation File to Process (dialog): open the surface created in the previous step

Profile Options (dialog): set the options as shown below and click **OK**



Choose Profile File to Write (dialog): set the options as shown below and click **Save**

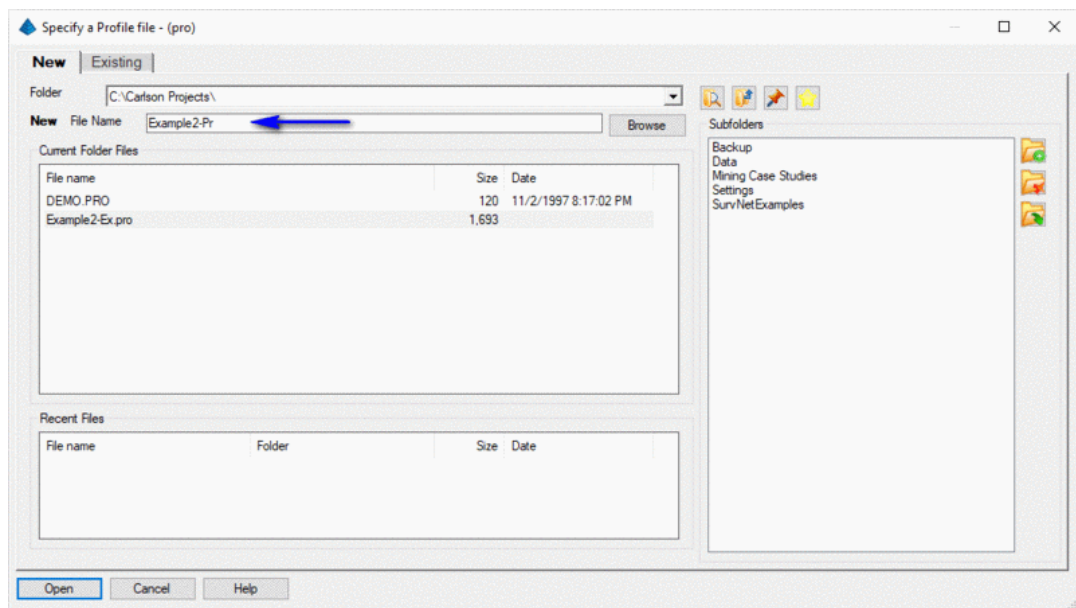


Polyline should have been drawn in direction of increasing stations.

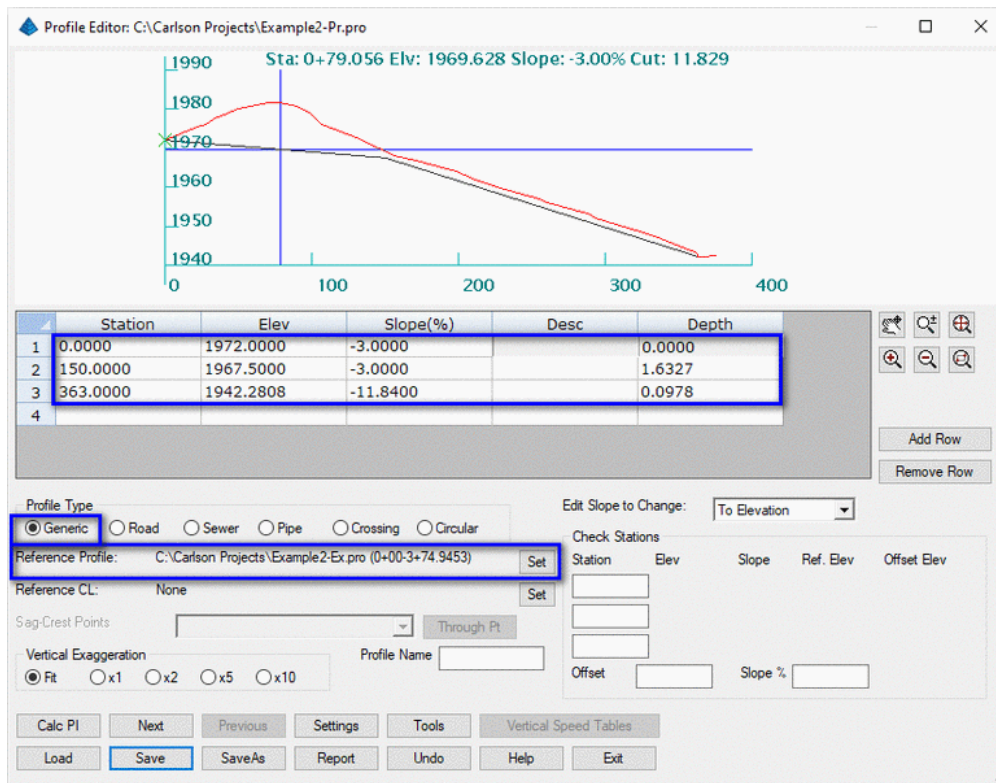
CL File/⟨Select centerline polyline⟩: pick the polyline you just drew

Starting station <0.0>: press Enter

Issue the Profiles – Profile Editor and when prompted, create the new file as illustrated below and click the **Open** button ready:

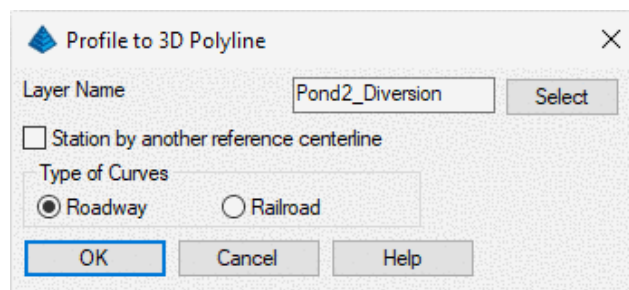


This displays a dialog box similar to that shown below:



Within this dialog box, perform the following actions:

- Use the **Set** to set the **Reference Profile** of the *existing ground* profile you just created
 - Specify a starting **Station** of **0** and give it a depth of **0**
 - Specify an ending **Station** of approximately that shown in the *Reference Profile* and give it a depth of **0**
 - Use the **Add Row** button to specify an internal vertical hinge point at a station of your choosing and a depth ≥ 0.75 . With the elevation/grade calculated, adjust the grade for the first record to be the next even percent as illustrated above
 - In all cases, make sure the new profile is always completely below *existing ground* so that the diversion ditch is always in cut
 - Alternatively, end the profile at a station where desired so that any pond overflow is clear of the dam and simply flows over hill
 - Save and exit from the **Profile Editor** when complete
13. **Tool 11:** Our next topic of discussion will focus on converting/combining the 2D polyline we drew along with that of the design profile to create a properly sloped 3D polyline representing one edge of the diversion ditch. Issue the Profiles – Profile to 3D Polyline command to display a dialog box similar to that shown below:



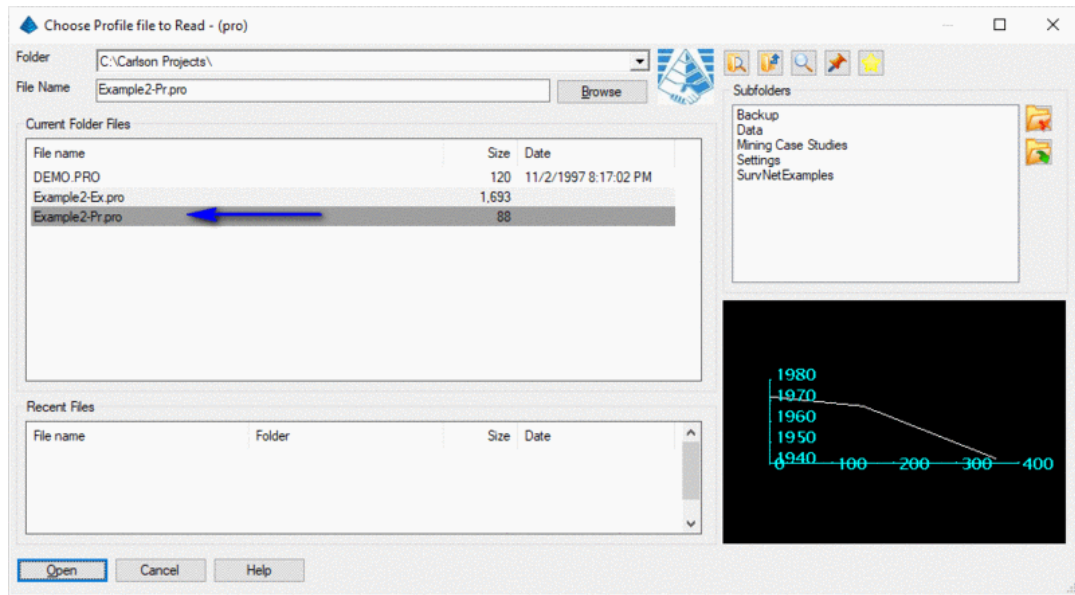
Set the values as shown above and click the **OK** button when ready. When prompted:

Polyline should have been drawn in direction of increasing stations.

CL File/<Select centerline polyline>: pick the 2D polyline we created earlier

Starting station <0.0>: press Enter

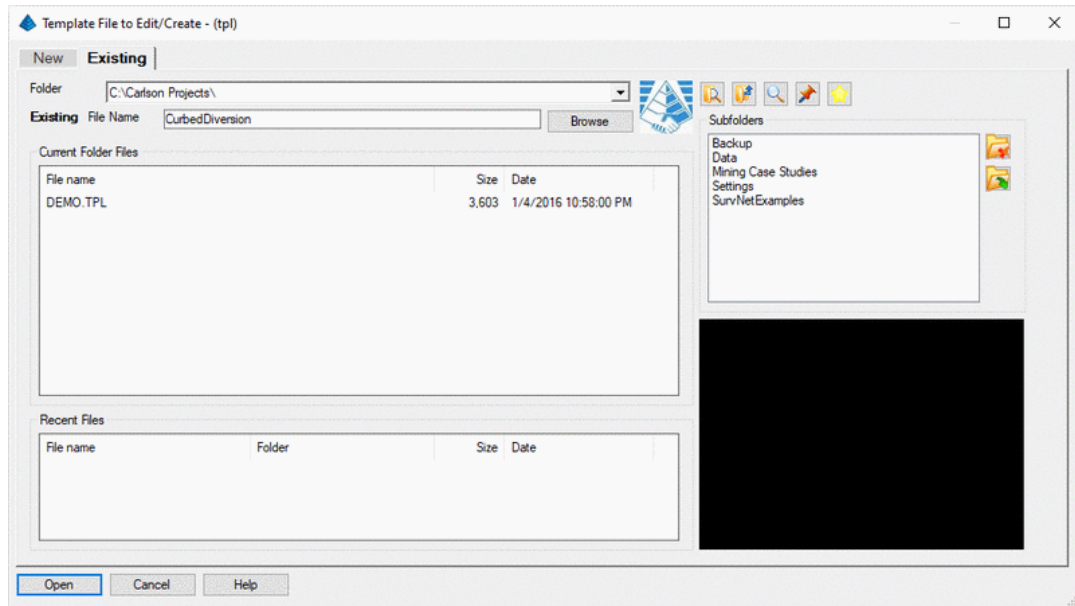
Select Profile to Read (dialog): select the design profile we created earlier and click **Open**



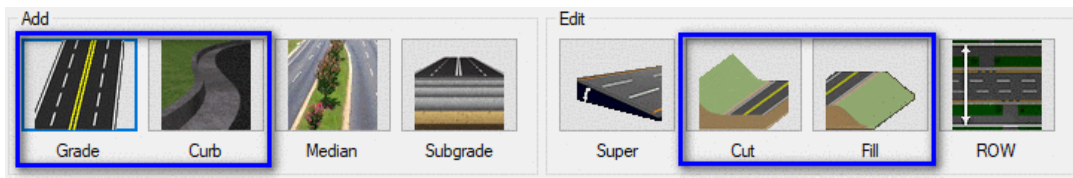
Erase centerline [Yes/<No>]? type Y and press Enter

The result is a 3D polyline along the length of the 2D polyline but at elevations defined by the design profile.

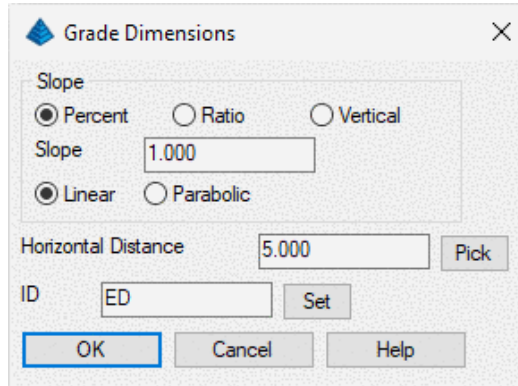
- 14. Tool 12:** Our next task will be to line the diversion ditch with concrete, include a traditional roadway curb element (to help control erosion) and create tie-slopes from the back of the curb feature. Issue the Roads – Design Template command to display a dialog box similar to that shown below:



We'll focus on this horizontal strip of icon options:

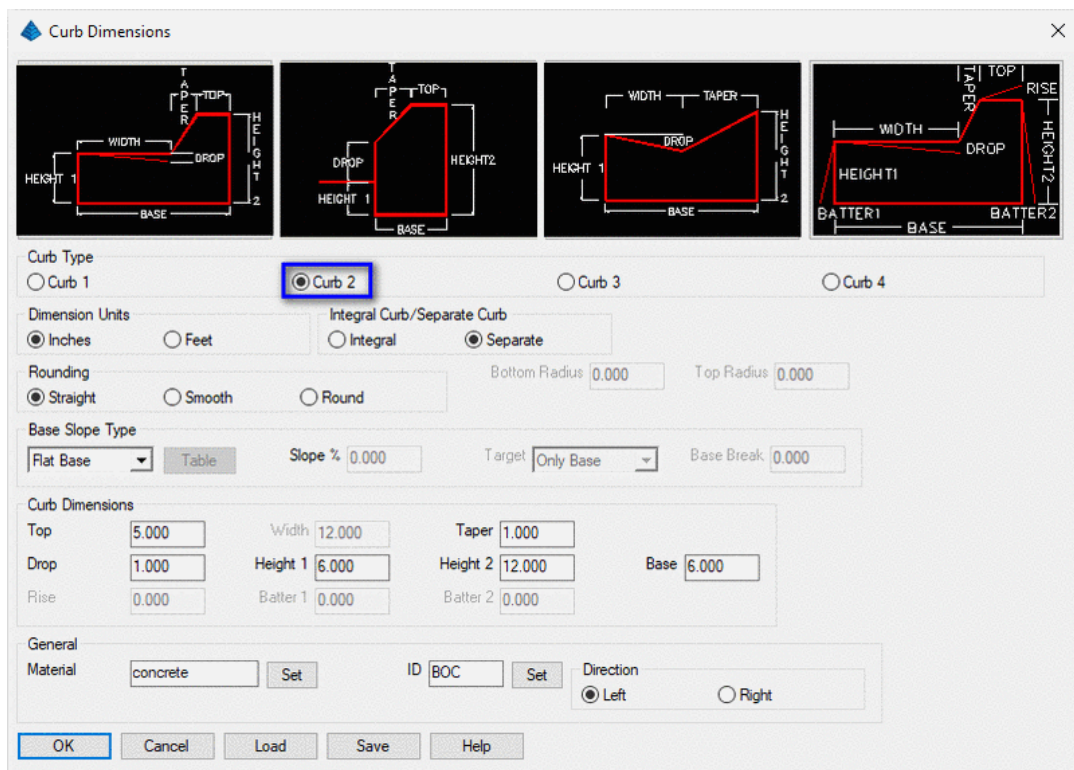


- To begin, click on the **Grade** button to display a dialog box similar to that shown shown below:



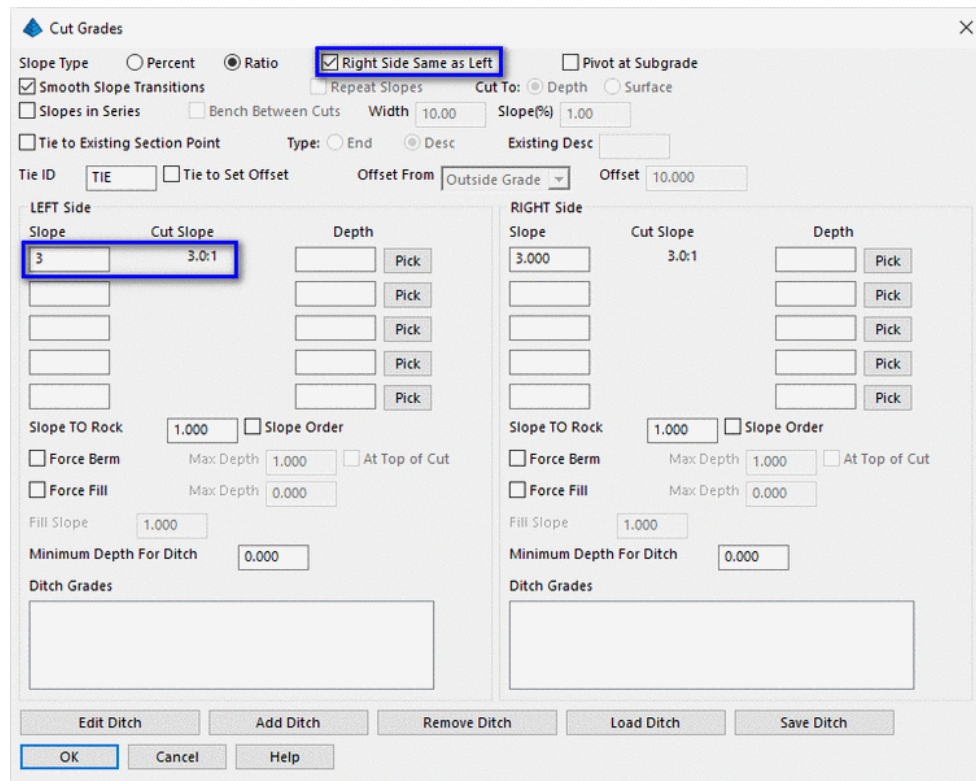
Set the values as shown above to ultimately create a 10' wide surface that drains to the center at a 1% slope and terminates with an **ID** of **ED** (edge drainage). Click **OK** when ready.

- Click on the **Curb** button to display a dialog box similar to that shown shown below:



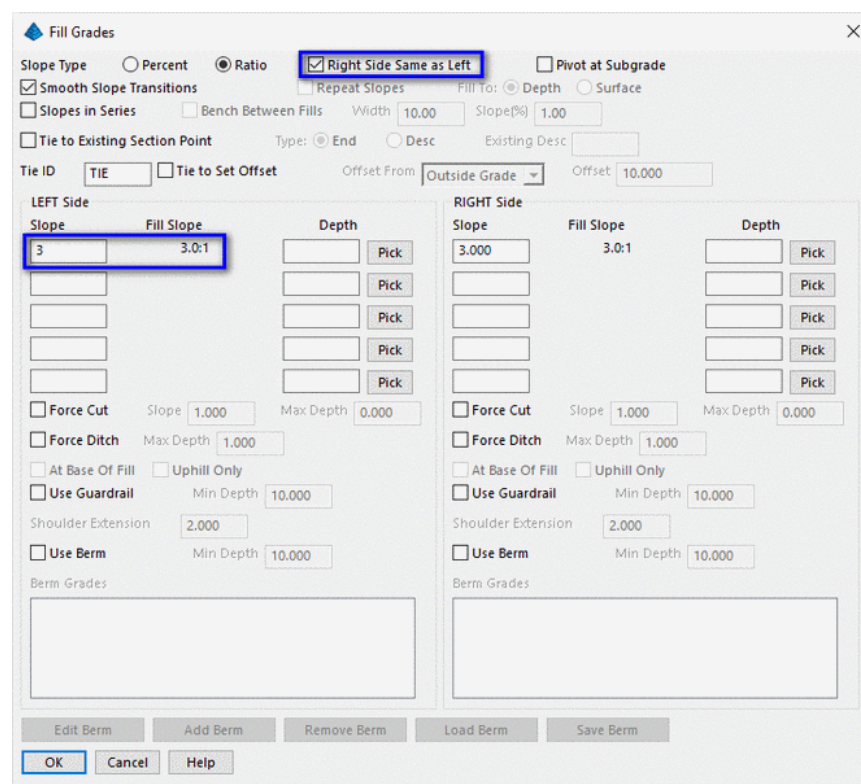
Set the values as shown above to ultimately create a simple concrete curb with an **ID** of **BOC** (back of curb). Click **OK** when ready.

- Click on the **Cut** button to display a dialog box similar to that shown shown below:



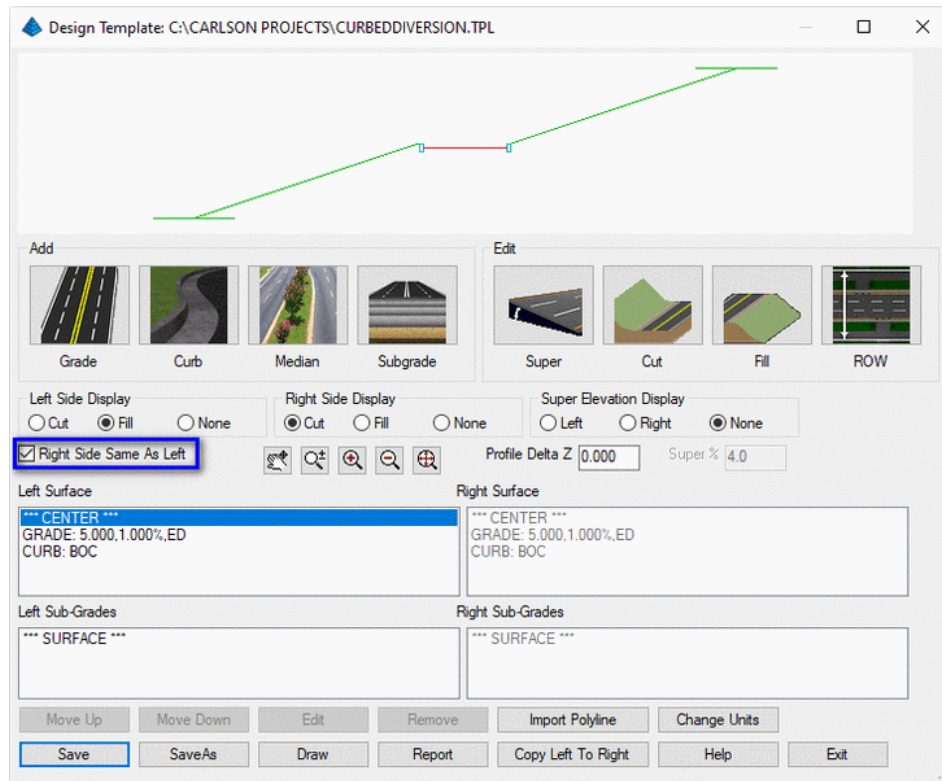
Set the values as shown above to ultimately create a single catch-slope used on both sides of the template for Cut scenarios with an **ID** of **TIE** (tie point). Click **OK** when ready.

- Click on the **Fill** button to display a dialog box similar to that shown shown below:

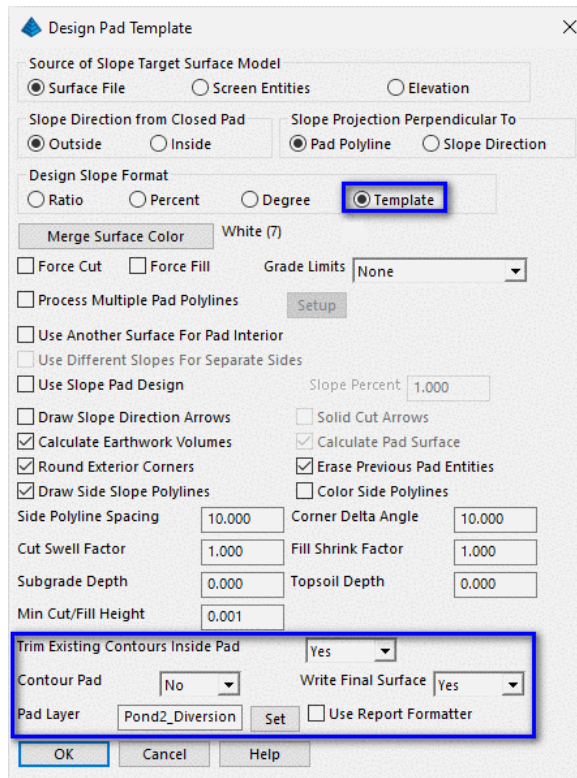


Set the values as shown above to ultimately create a single catch-slope used on both sides of the template for Fill scenarios with an **ID** of **TIE** (tie point). Click **OK** when ready.

Verify the completed template resembles that shown below (paying attention to the highlighted item) and click **Save** and **Exit** when ready:



Now let's use the Surface – **Design Pad Template** routine one more time as illustrated below:



For the first time, we have selected the **Template** option as illustrated above. Set the values as shown above

paying particular attention to the highlighted items and click **OK** when ready. When prompted:

Pick pad polyline: pick the 3D polyline created earlier

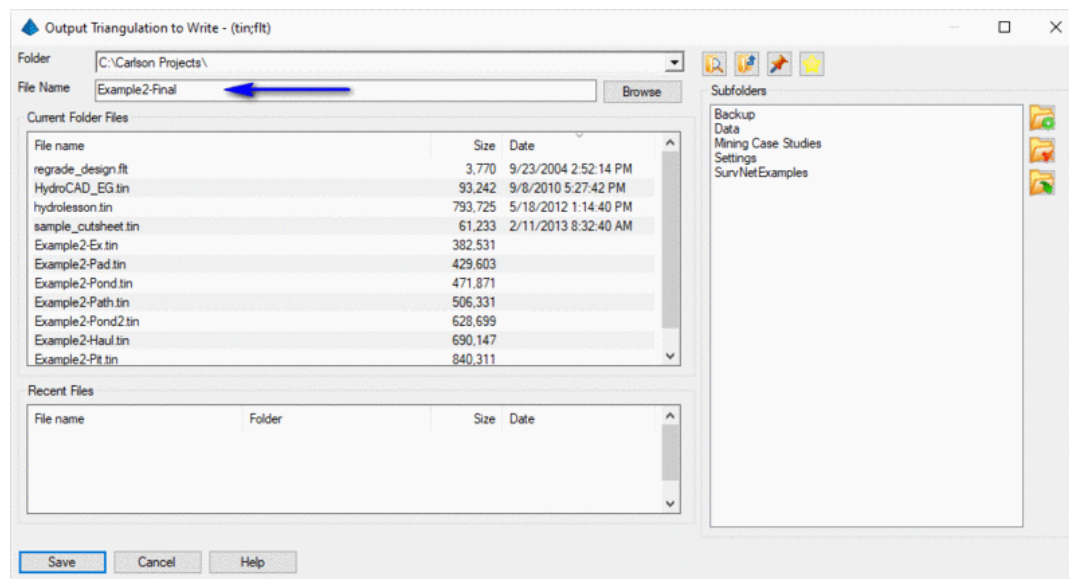
Select Slope Target Surface (dialog): open the most recent surface for the project

Select Template (dialog): pick the template file created earlier

Review the **Pad Report** and click the **Exit (Doorway)** button when ready. Prompting resumes:

Merge Final Surface (dialog): click **Yes**

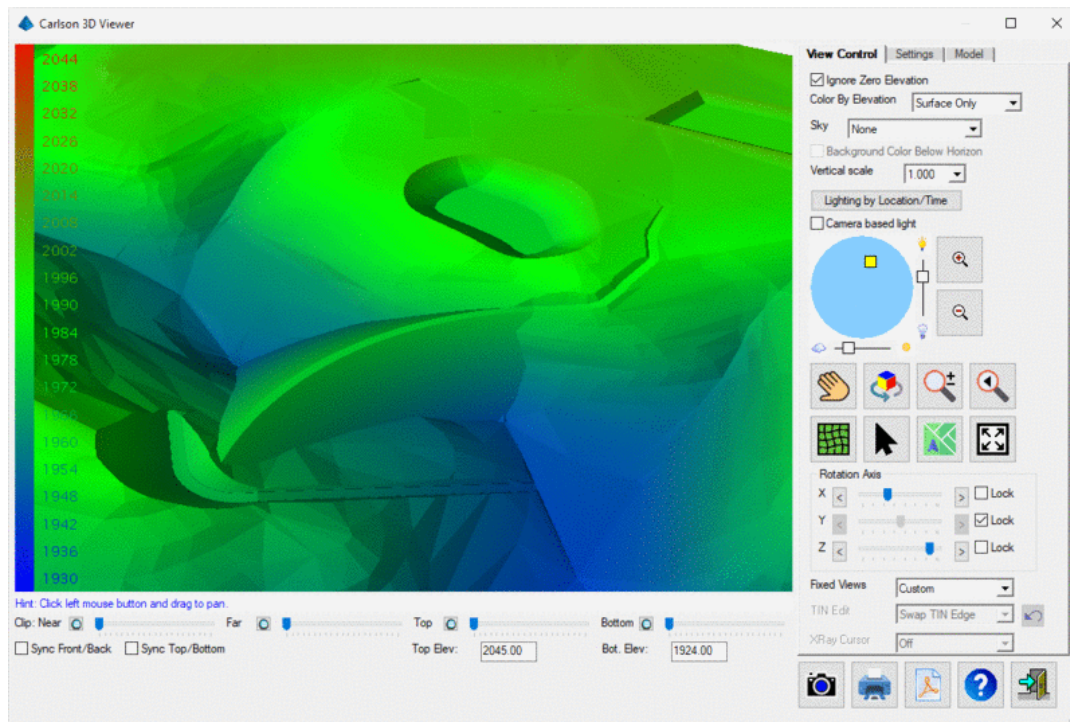
Output Triangulation to Write (dialog): set the value as shown below and click **Save**



Prompting resumes:

Retain trimmed polyline segments [Yes/<No>]? type N and press Enter

Issue the View – 3D View – **Surface File Viewer** command to examine the results as illustrated below:



Click the **Exit (Doorway)** button when ready.

Finally, let's see how much earthwork has been moved since we started. Issue the **Surface – Volumes by Triangulation – Two Triangulation Surface Volumes** command. When prompted:

Select EXISTING Surface Triangulation File (dialog): open the first surface model created in this project

Select FINAL Surface Triangulation File (dialog): open the last surface for the project

Volume Report Options (dialog): accept the default values

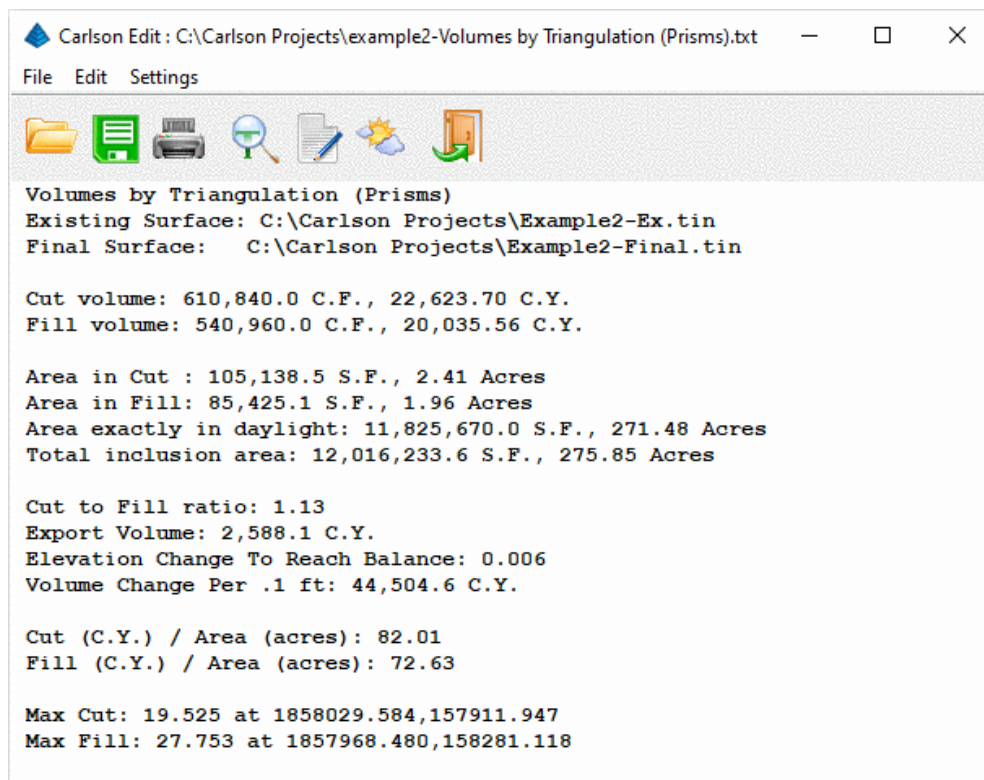
Select Inclusion polylines or press Enter for none.

[FILter]/<Select entities<: press Enter

Select Exclusion polylines or press Enter for none.

[FILter]/<Select entities<: press Enter

Review the results as illustrated below (your results will be different):



Click the **Exit** (Doorway) button when ready.

This completes the tutorial: A Dozen Tools for Surface Design.

Calculate Volumes by Five Methods

For this tutorial, we'll be using a sample stockpile drawing and calculating its volume using five distinct techniques:

1. **Stockpile Volumes:** Creates grid surfaces from perimeter polyline and surface entities and calculates volumes in one command
2. **Volumes By Layers:** Creates grid surfaces from existing and design layers and calculates volumes in one command
3. **Two Grid Surface Volumes:** Calculates volumes between two grid files
4. **Volumes By Triangulation:** Calculates volumes between two triangulation files
5. **Calculate Sections Volumes:** Calculates volumes between two section files using volumes by average end areas

Each of these routines has its own advantages. You can choose the routine that best suits your data or run multiple methods as a check of the volumes. The volume reports for each of these routines will vary slightly due to using different types of surface models that have different resolutions. These volume differences should be less than 2%.

A Results Summary of the various volume methodologies is also provided.

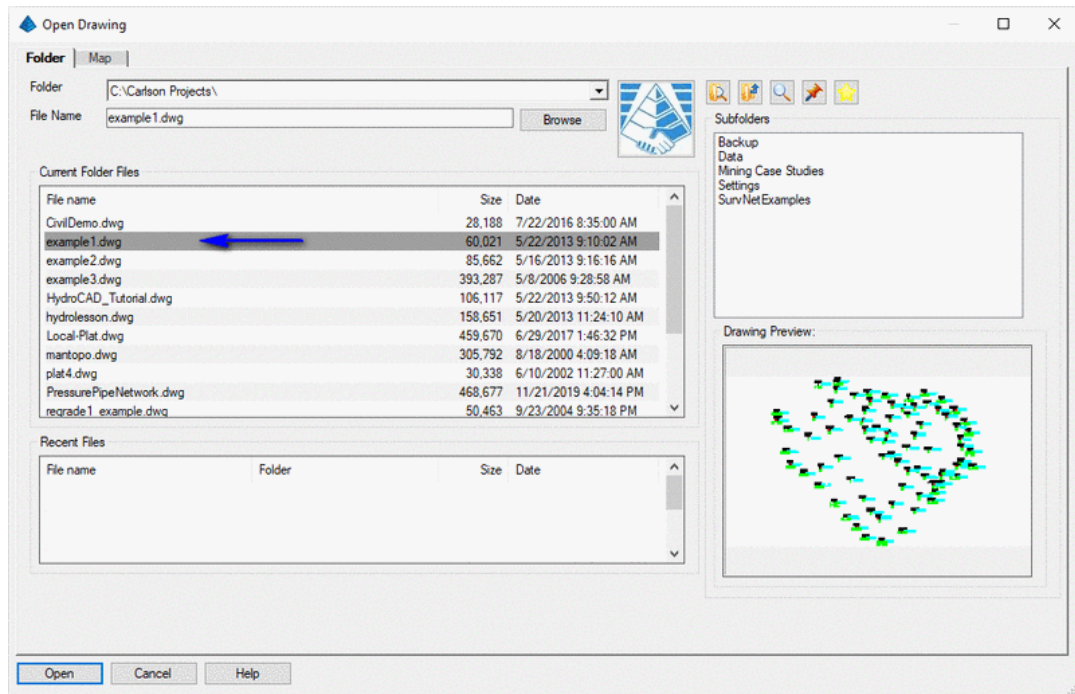
If there is a greater difference, try increasing the resolution of the surface models. For grids, make the grid cell size smaller and for sections, make the station interval smaller. If there is still a significant difference, then the cause should be investigated by checking the source data.

Common Steps

The following are common preparation steps for all five volume methods:

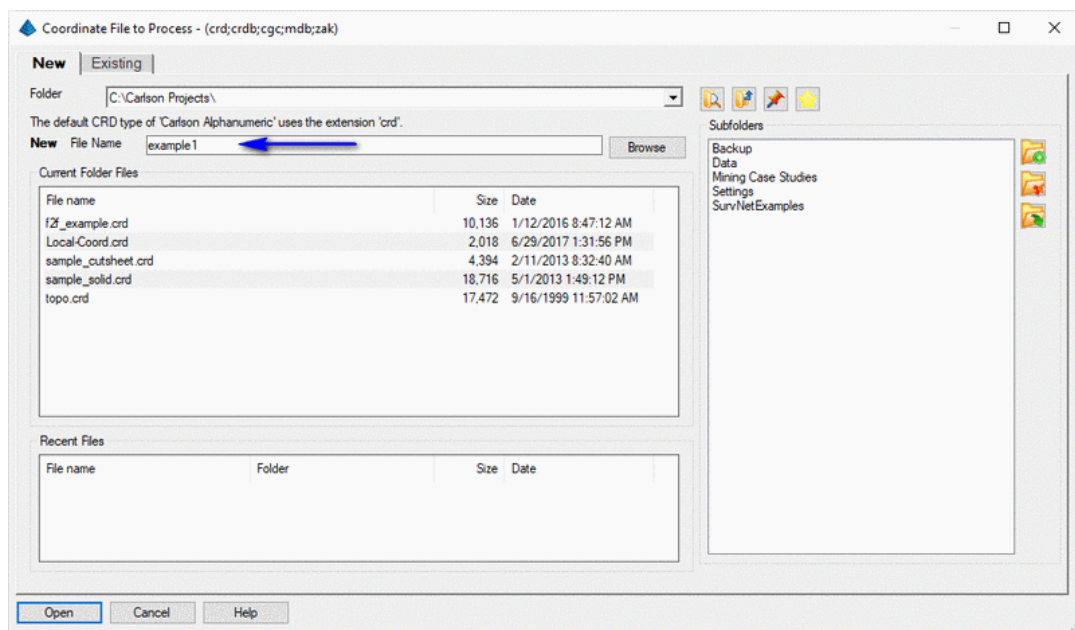
1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.

- If you are taken directly into CAD, click File – **Open**.
Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **example1.dwg** file.



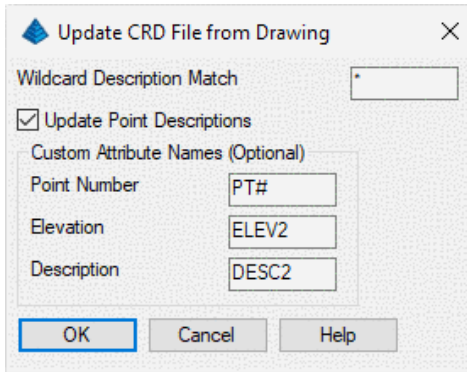
Use the File – **Save As** to save a copy named **example1-v.dwg**. Completing this tutorial will alter the drawing file and by renaming the file from the start, you'll keep the original file intact (allowing you to run through the tutorial a second time if desired). This is also a good practice to keep when working on drawings from 3rd parties.

2. Ensure you are in the Civil program by issuing the Settings – Carlson Menus – **Civil Menu** command. Since our drawing contains point data, let's house it in a related *coordinate file* for future use. Issue the Points – Set Coordinate File command to display a dialog box similar to that shown below:



Set the value as shown above and click **Open** when ready. Next, issue the Points – Point Utilities – Update CRD File from Drawing command (also found under Points – Coordinate File Utilities) to display a dialog

box similar to that shown below:



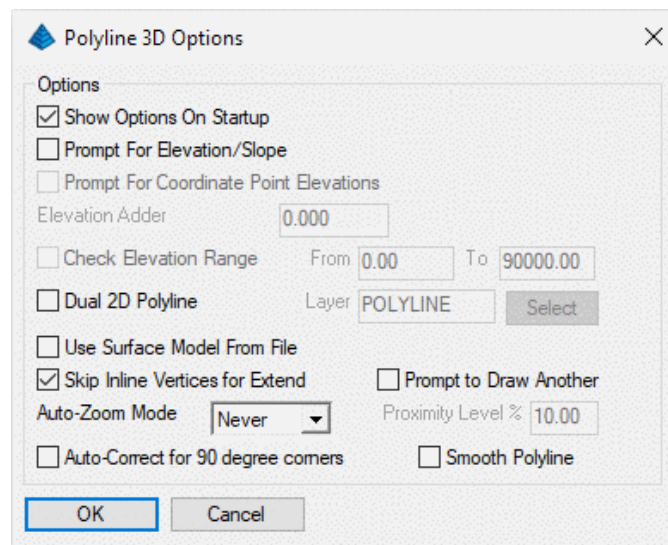
Accept the default values and click **OK** when ready. When prompted:

Select Carlson, LDD, Softdesk, InRoads, EaglePt, Geodimeter, Leica or MicroSurvey Points.

[FILter]/<Select entities>: type ALL and press Enter

[FILter]/<Select entities>: press Enter

3. Due to the shape of the data (and in an effort to get consistent results), we'd like to develop a boundary (or *inclusion*) polyline around the data set. There are several ways to accomplish this:
 - Through the use of the **Survey** module and its Survey – Draw Field to Finish command, or,
 - Through the use of the Draw – 3D Polyline command, or,
 - Through the use of the Draw – Shrink-Wrap Entities command, or,
 - Through the use of the Surface – Stockpile/Pond/Pit Volumes – Draw 3DPoly Perimeter command (run it now) to display a dialog box similar to that shown below:

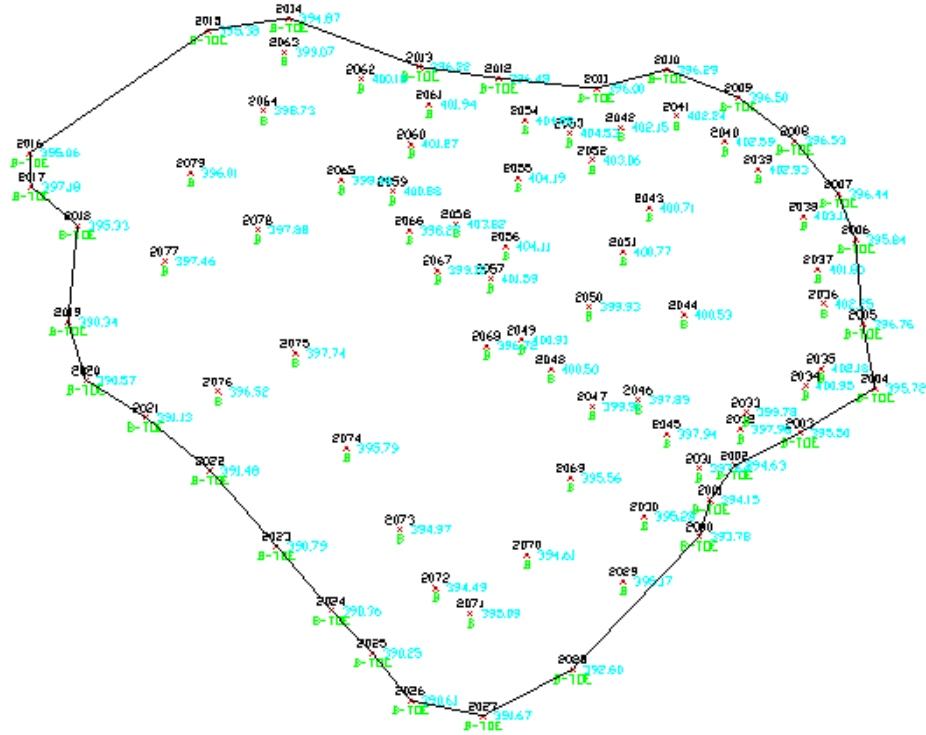


Set the values as shown above (click **OK** when ready) and when prompted:

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: type the range of 2000-2028 and press Enter

[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: type C and press Enter

The drawing should resemble that shown below:



Let's begin the volume calculation processes.

Volume Method 1 - Calculate Stockpile Volume

The **Calculate Stockpile Volume** routine is based on a *grid* surface methodology like the Two Surface Volumes discussion below. The difference with this routine is that it builds the grid surfaces within the routine to save time otherwise needed to build the grid files. The fewer steps make this routine faster and easier but it doesn't have options for checking surfaces. Instead the input data entities should be checked before running this routine. Also, **Calculate Stockpile Volume** only applies to volumes calculations when the volume is all Fill.

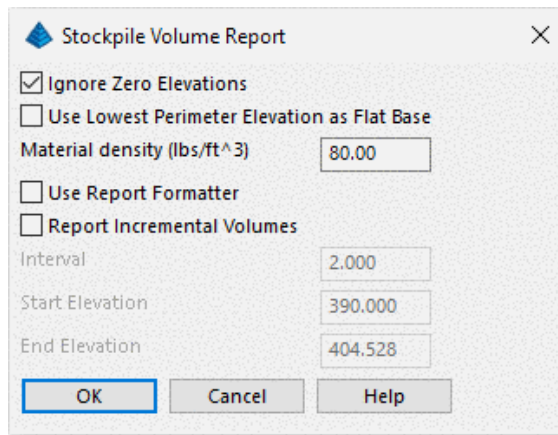
1. Issue the Surface – Stockpile/Pond/Pit Volumes – Calculate Stockpile Volume command. When prompted:

Select stockpile entities and perimeter.

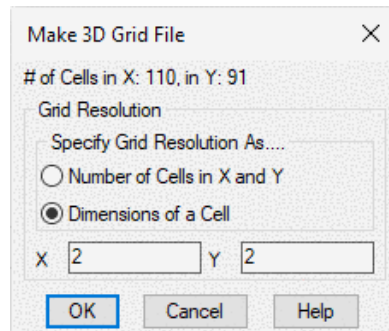
[FILTER]/<Select entities>: type ALL and press Enter (The program looks for a closed 3D polyline on the PERIMETER layer to use as the inclusion perimeter and the base surface model. If this polyline is not found, then the program will prompt to select the perimeter polyline. All the selected entities including the perimeter are used to model the second surface of the stockpile top).

[FILTER]/<Select entities>: press Enter

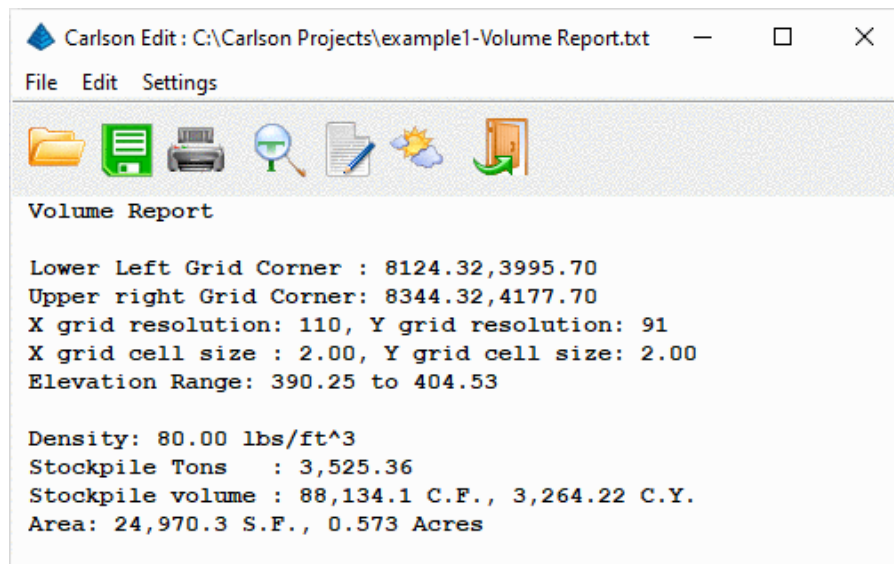
The routine offers various reporting capabilities. Let's assume a typical density for a pile of road salt as illustrated below and click **OK** when ready:



Specify the grid resolution as shown in the dialog below and click **OK** when ready:



The result is summarized in a volume report in the Standard Report Viewer similar to that shown below:



Click the **Exit** (Doorway) button to dismiss the report.

NOTE: The limits of the grid are determined automatically and shown in the report. For the sake of accuracy, the next two methods will use the grid parameters shown above.

NOTE: For situations involving a Cut-only scenario (such as a pond or pit), use the Surface – Stockpile/Pond/Pit Volumes – Calculate Pond/Pit Volume command.

Volume Method 2 - Volumes By Layers

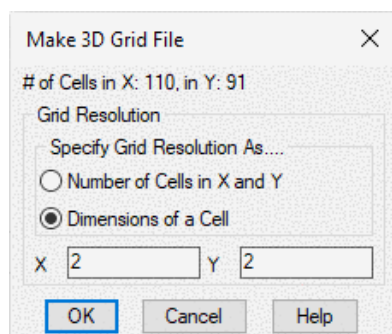
Let's explore a simple routine that handles both Cut and Fill scenarios.

1. Issue the Surface – Volumes By Grid Surface – Volumes By Layers command. When prompted:

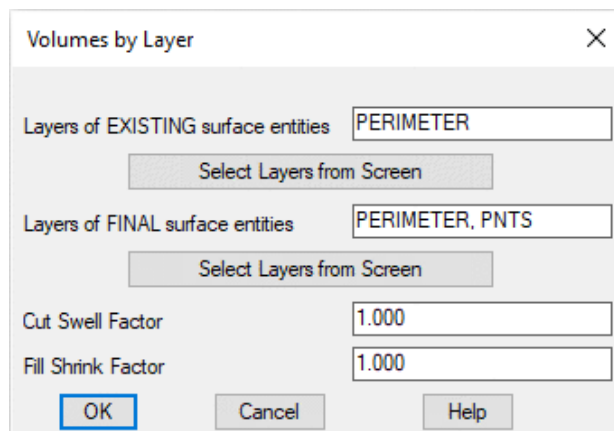
Pick Lower Left limit of surface area <8124.32,3995.70>: indicate the values shown to the left and press Enter (or pick a lower left limit)

Pick Upper Right limit of surface area <8344.32,4177.70>: indicate the values shown to the left and press Enter (or pick an upper right limit)

Specify the grid resolution as shown in the dialog below and click **OK** when ready:



A dialog box similar to that shown below appears:



Click the **Select Layers from Screen** buttons and when prompted:

- **Select entities on layers of Existing surface.**
[FILter]/<Select entities>: pick the PERIMETER polyline and press Enter
- **Select entities on layers of Final surface.**
[FILter]/<Select entities>: type ALL and press Enter twice

Once the values are as shown above, click **OK** when ready. Prompting resumes:

Select surface entities on corresponding layers.

[FILter]/<Select entities>: type ALL and press Enter twice

Select the Inclusion perimeter polylines or ENTER for none:

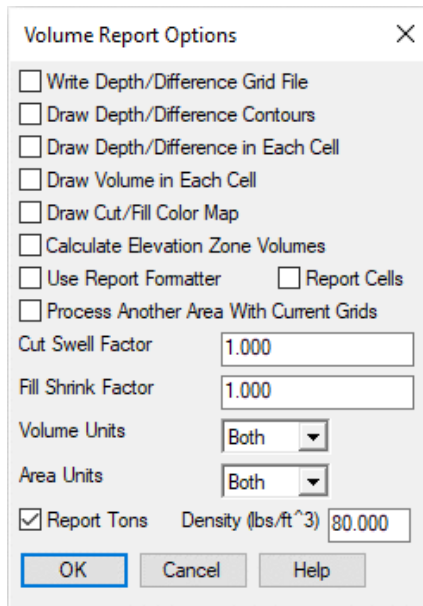
[FILter]/<Select entities>: pick the PERIMETER polyline and press Enter

[FILter]/<Select entities>: press Enter

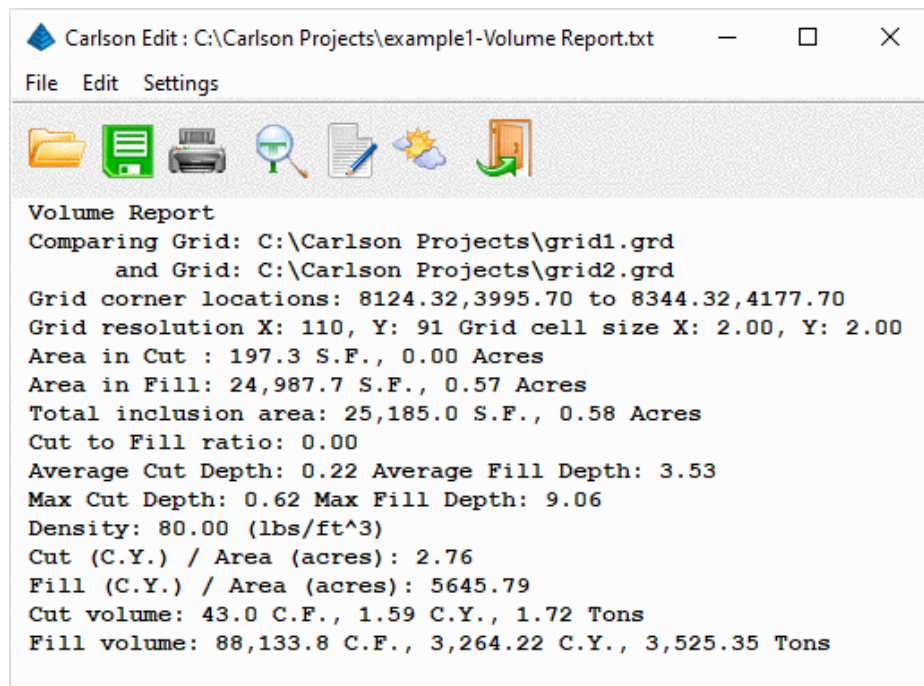
Select the Exclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: press Enter

Volume Report Options (dialog): set the values as shown below and click **OK** when ready:



The volume report similar to that shown below appears:



Click the **Exit** (Doorway) button to dismiss the report.

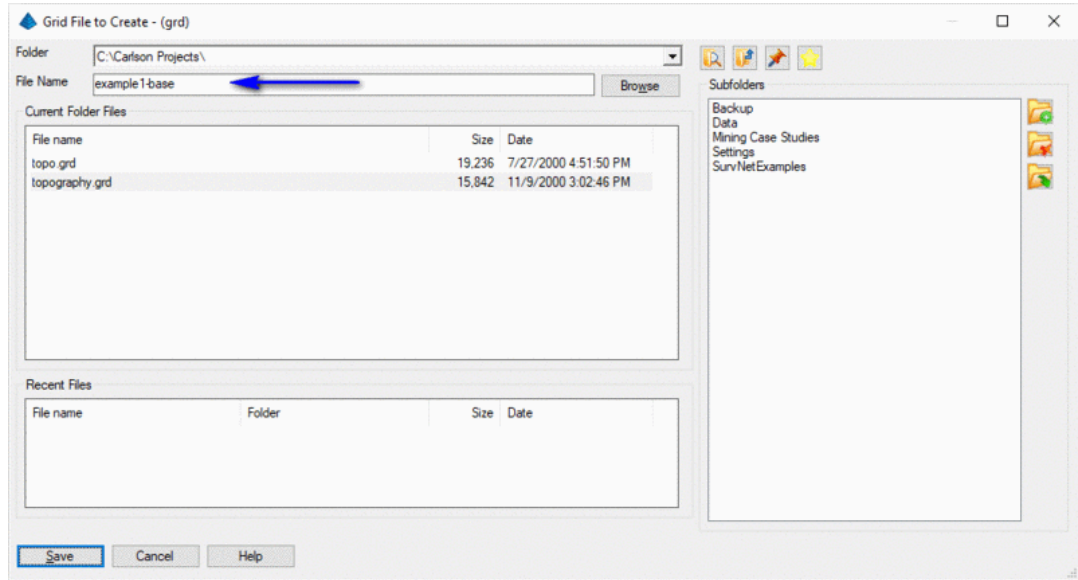
NOTE: Similar to Calculate Stockpile Volume, this routine builds the grid surfaces within the routine to save the steps of creating grid files. The difference between this routine and the *Calculate Stockpile Volume* routine is that **Volumes By Layers** uses entities on specified layers for **Existing** and **Final** scenarios to build the surfaces and it will calculate both Cut and Fill volumes.

Volume Method 3 - Two Grid Surface Volumes

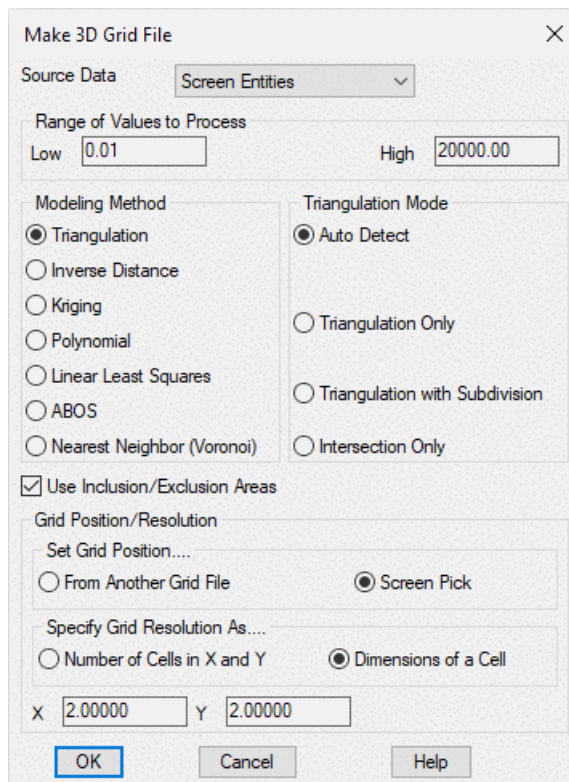
For the next example, we must create the two grid files; one representing an assumed base-ground (*Existing*) surface, the other representing the surface of the pile (*Final*).

1. Issue the Surface – Make 3D Grid File command. When prompted:

Grid File to Create (dialog): supply the file name indicated below and click **Save** when ready:



A dialog box similar to that shown below appears. Set the values as shown and click **OK** when ready:



Prompting resumes:

Pick first grid corner <8124.32,3995.70>: press Enter (notice the grid limit from the previous example is defaulted)

Pick opposite grid corner <8344.32,4177.70>: press Enter

Select the Inclusion perimeter polylines or ENTER for none.

[Filter]/<Select entities>: pick the PERIMETER polyline

[FILter]/<Select entities>: press Enter

Select the Exclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: press Enter

Select points, lines, polylines and faces to grid from.

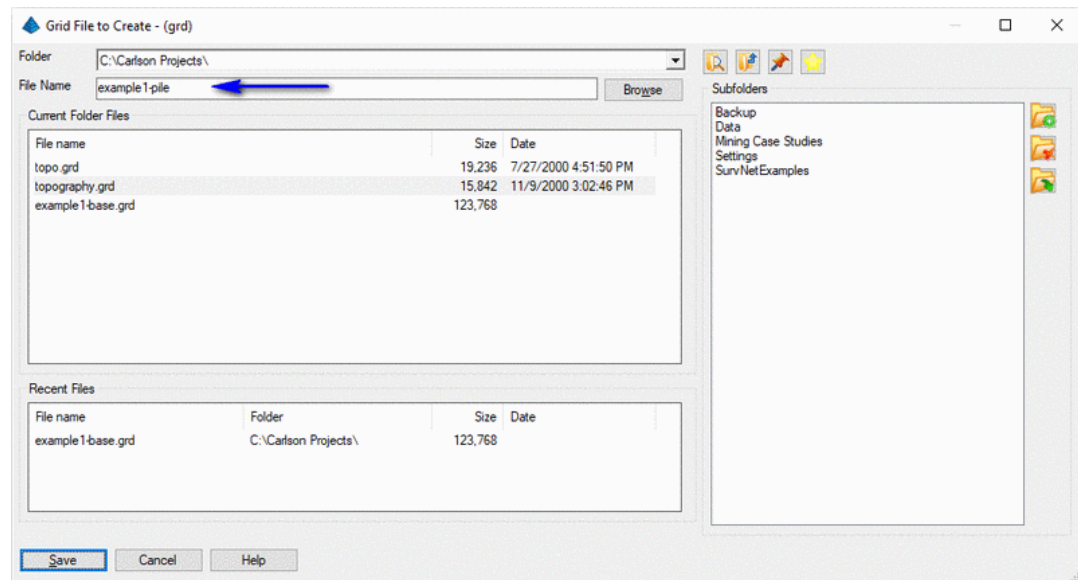
[FILter]/<Select entities>: pick the PERIMETER polyline again and press Enter

[FILter]/<Select entities>: press Enter

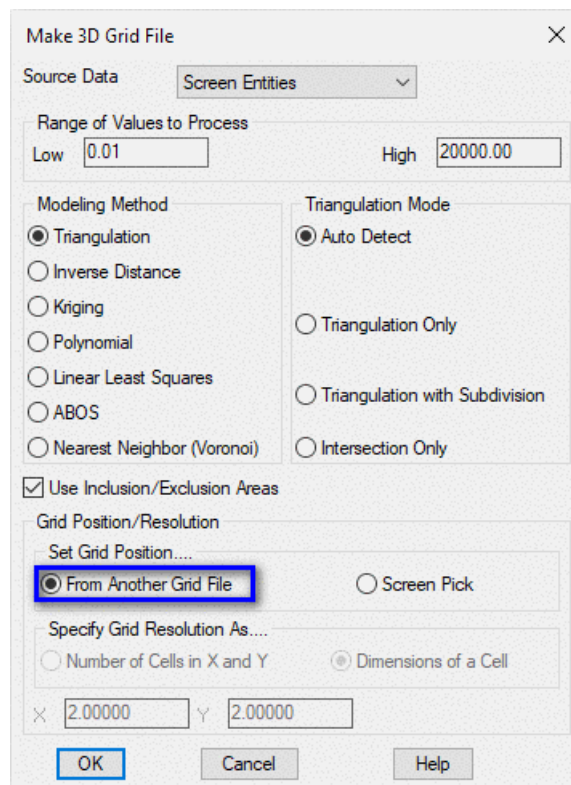
A grid file representing an assumed base-ground (*Existing*) surface is written.

2. Re-run the Surface – **Make 3D Grid File** command. When prompted:

Grid File to Create (dialog): supply the file name indicated below and click **Save** when ready:



A dialog box similar to that shown below appears. Set the values as shown and click **OK** when ready:



NOTE: For the volume routine to follow, the two grids to compare should have matching grid positions and resolution. Prompting resumes:

Grid File to Get Position From (dialog): select the grid file created earlier and click **Open** when ready
Select the Inclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: pick the PERIMETER polyline

[FILter]/<Select entities>: press Enter

Select the Exclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: press Enter

Select points, lines, polylines and faces to grid from.

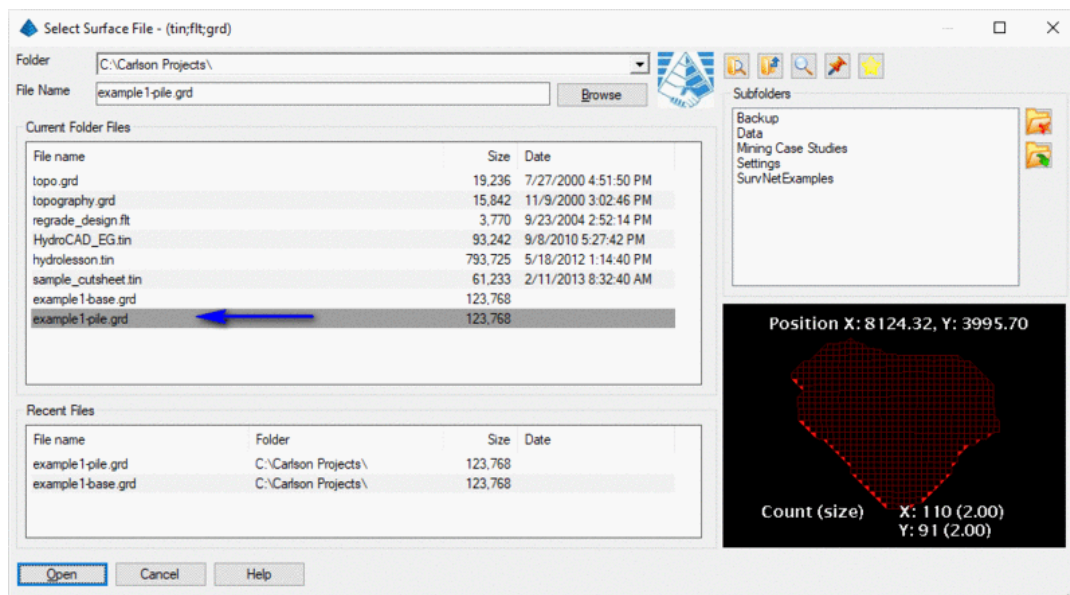
[FILter]/<Select entities>: type ALL and press Enter

[FILter]/<Select entities>: press Enter

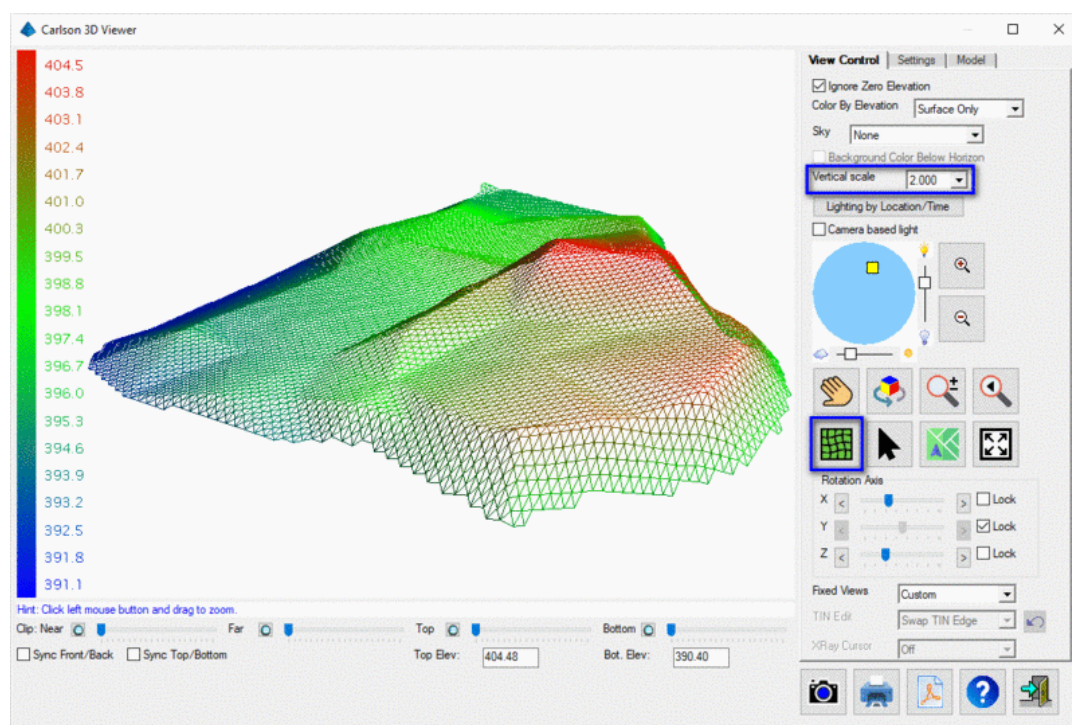
A grid file representing the surface of the pile (*Final*) is written.

3. *Optional:* Presume we might want to check the surface(s) just created. This step is optional to verify that the surfaces are good by checking for bad elevation data points and that the surfaces follow the data points. Issue the View – 3D View – Surface File Viewer and when prompted:

Select Surface File (dialog): select the grid file that was just created and click **Open** when ready:

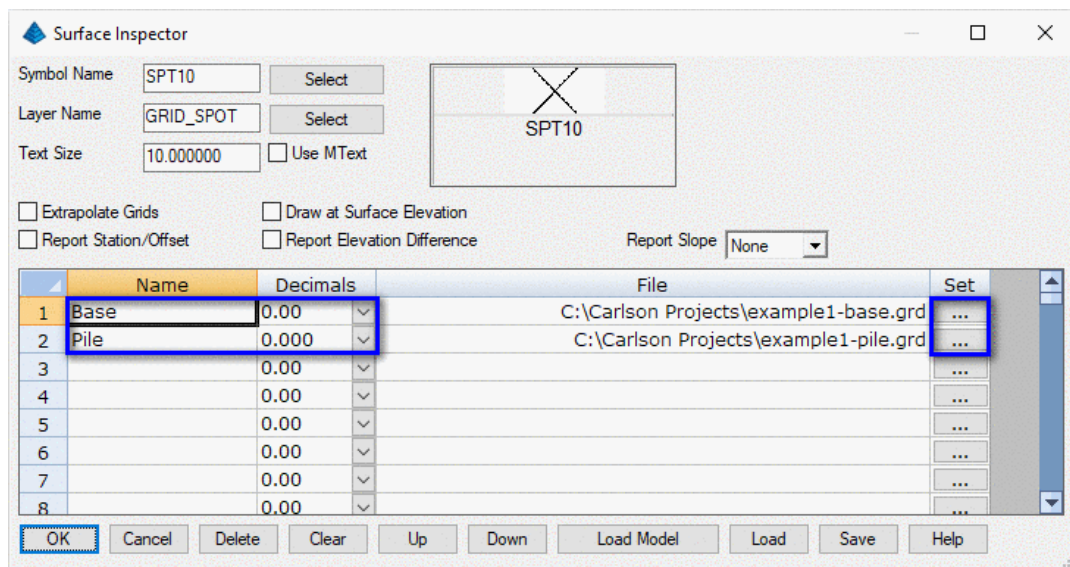


A 3D Viewer window similar to that shown below appears:

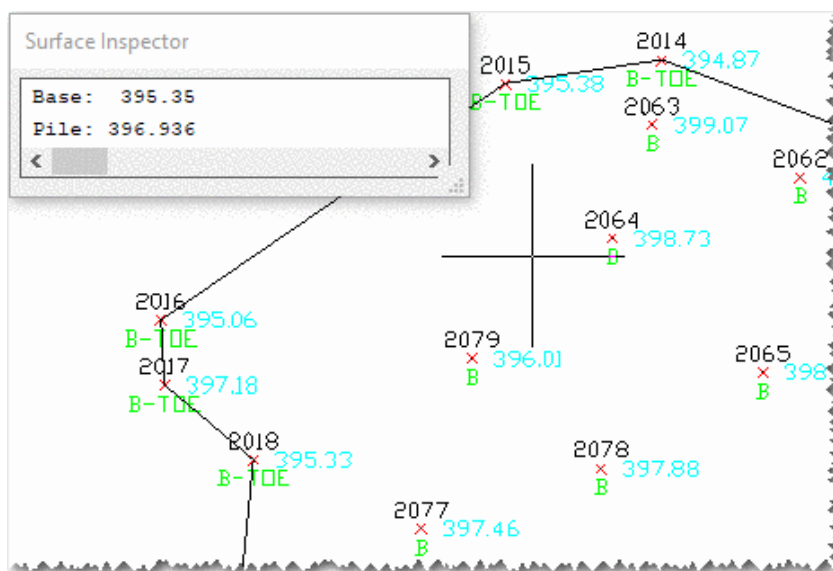


In the 3D Viewer dialog, move the pointer near the outer edge of the graphic and the cursor will change to an Z symbol which orients the view about the vertical axis. Moving the cursor to the interior of the graphic and the cursor will change to an X/Y symbol. Click and drag from the bottom upward to change the isometric viewing angle. You can also set the **Vertical Scale** to 2.0 and enable the **Color By Elevation** toggle for better viewing of the elevation difference. Experiment with any desired remaining options. When ready, click the **Exit** (Doorway) button.

- Now, let's check some spot elevations. Issue the Surface – Surface Inspector command to display a dialog box similar to that shown below:



Use the highlighted controls to set the values as shown above. When ready, click the **OK** button. As you position the cursor into the data set, notice a real-time reporting of the two surface elevations as illustrated below:



Check that the grid elevations match the point elevations reasonably well. Remember that the base elevations are using only the B-TOE points. The elevations won't match exactly with grid surfaces because the grid model is at the resolution of the grid cells. When finished checking, press **Enter**.

- Now that you have the *Base* and *Pile* grid files, issue the Surface – Volumes By Grid Surface – Two Grid Surface Volumes command. When prompted:

Select the Inclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: pick the PERIMETER polyline

[FILter]/<Select entities>: press Enter

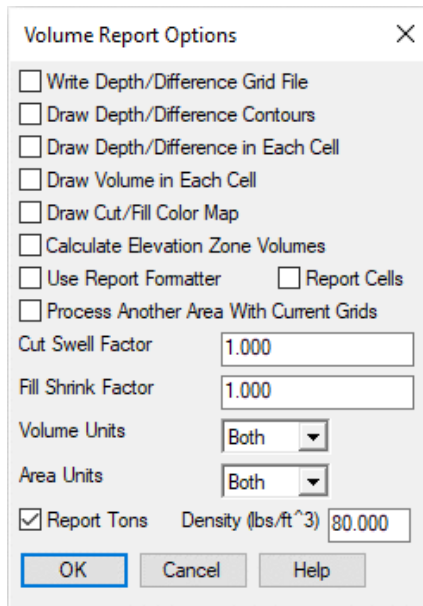
Select the Exclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: press Enter

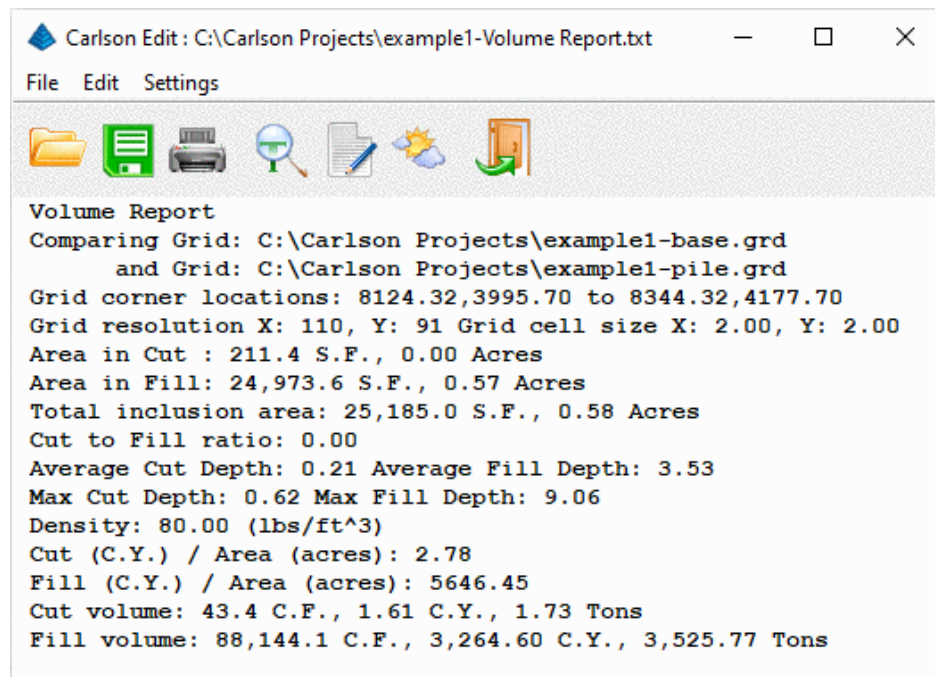
Select Base Grid File (dialog): select the *Base* grid file and click **Open** when ready

Select Final Grid File (dialog): select the *Pile* grid file and click **Open** when ready

Volume Report Options (dialog): set the values as shown below and click **OK** when ready:



The volume report similar to that shown below appears:



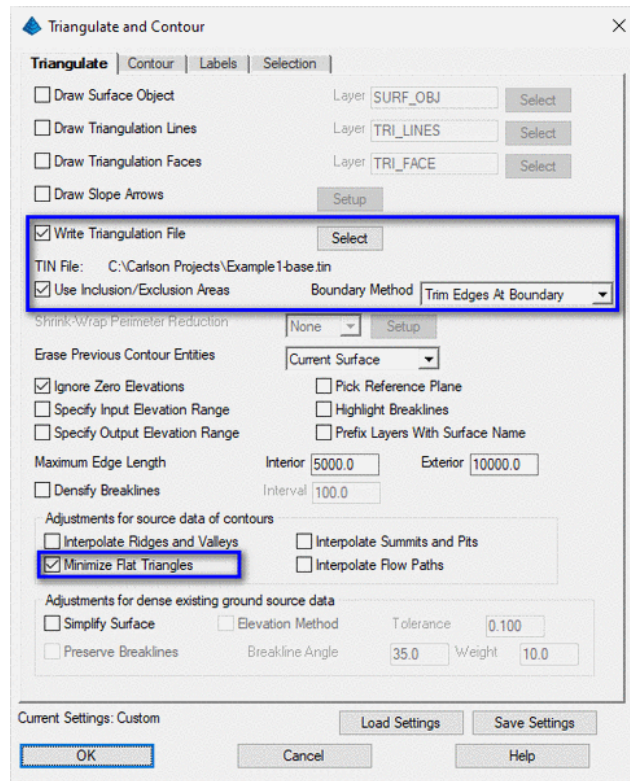
Click the **Exit** (Doorway) button to dismiss the report.

NOTE: Although *grid* files produce relatively accurate volumes (based on the *grid resolution*), their primary value lies in the ability to hand-calculate the results for validation purposes. With a grid resolution (*e.g.* 2' x 2') and the average elevation of each grid cell (resulting in a grid depth), a grid cell volume can be computed through traditional means.

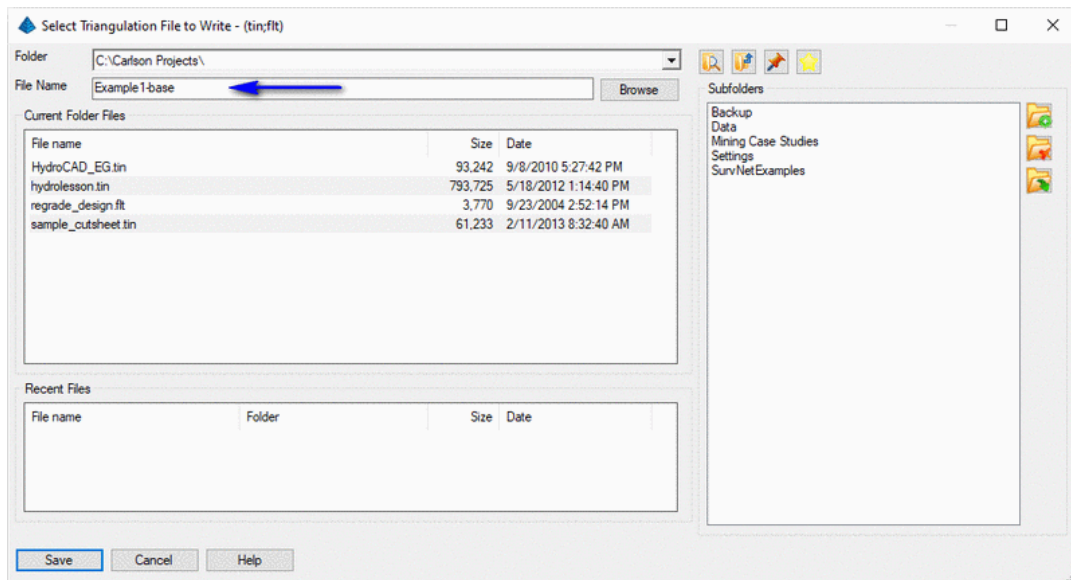
Volume Method 4 - Volumes By Triangulation

For precise surface-to-surface volumes, creating *triangulated irregular network* (TIN) files is the preferred methodology.

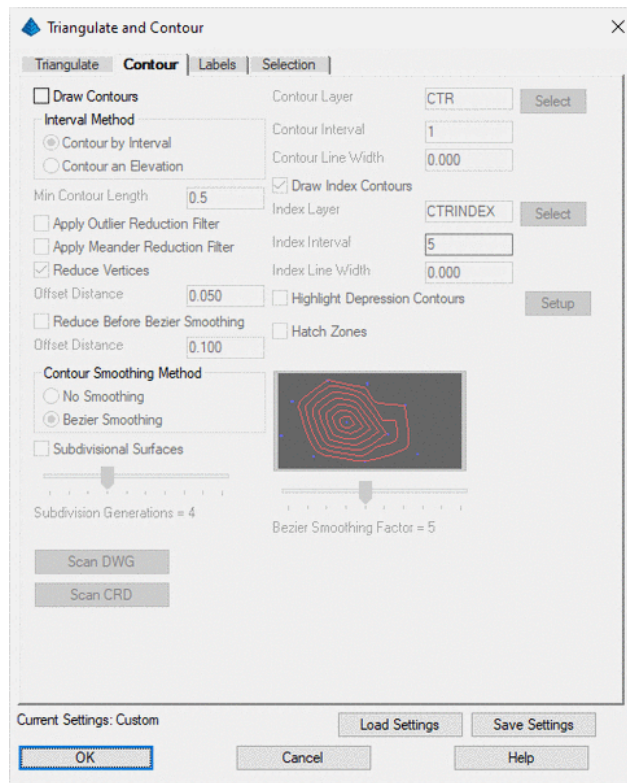
- Issue the Surface – Triangulate & Contour command. The following dialog box (with several "tabs") will appear which you should fill out as discussed below:



- In the Triangulate tab (as shown above), enable the **Write Triangulation File** toggle and click its **Select** button to set the name of a triangulation (*.tin) file as illustrated below. Click the **Save** button on this dialog box so that the results of the triangulation can be re-used for future purposes. Turn on the **Use Inclusion/Exclusion Areas** toggle and set any remaining options as illustrated above:



- In the Contour tab, as shown below, toggle off the **Draw Contours** option:



Click the OK button from any tab. When prompted:

Select Inclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: pick the PERIMETER polyline

[FILter]/<Select entities>: press Enter

Select Exclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: press Enter

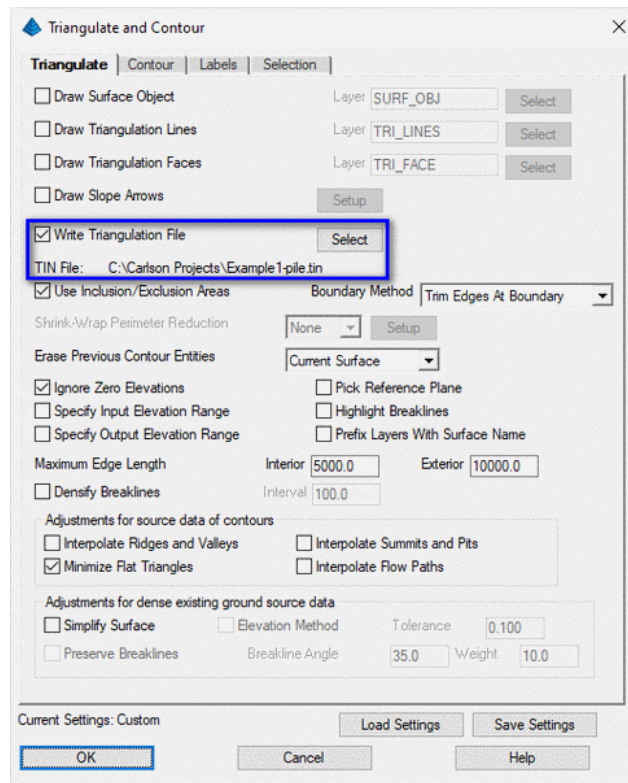
Select points and breaklines to Triangulate.

[FILter]/<Select entities>: pick the PERIMETER polyline again and press Enter

[FILter]/<Select entities>: press Enter

If a secondary dialog box appears prompting for a coordinate file (*.crd), click **Cancel**. A "TIN" file representing an assumed base-ground (*Existing*) surface is written.

- For the *pile* TIN, we'll follow the same general procedure as above but with a few small tweaks. Re-issue the Surface – **Triangulate & Contour** command and set the values as shown below (click **OK** when ready):



When prompted:

- Select Inclusion perimeter polylines or ENTER for none.**
- [FILter]/<Select entities>:** pick the PERIMETER polyline
- [FILter]/<Select entities>:** press Enter
- Select Exclusion perimeter polylines or ENTER for none.**
- [FILter]/<Select entities>:** press Enter
- Select points and breaklines to Triangulate.**
- [FILter]/<Select entities>:** type ALL and press Enter
- [FILter]/<Select entities>:** press Enter

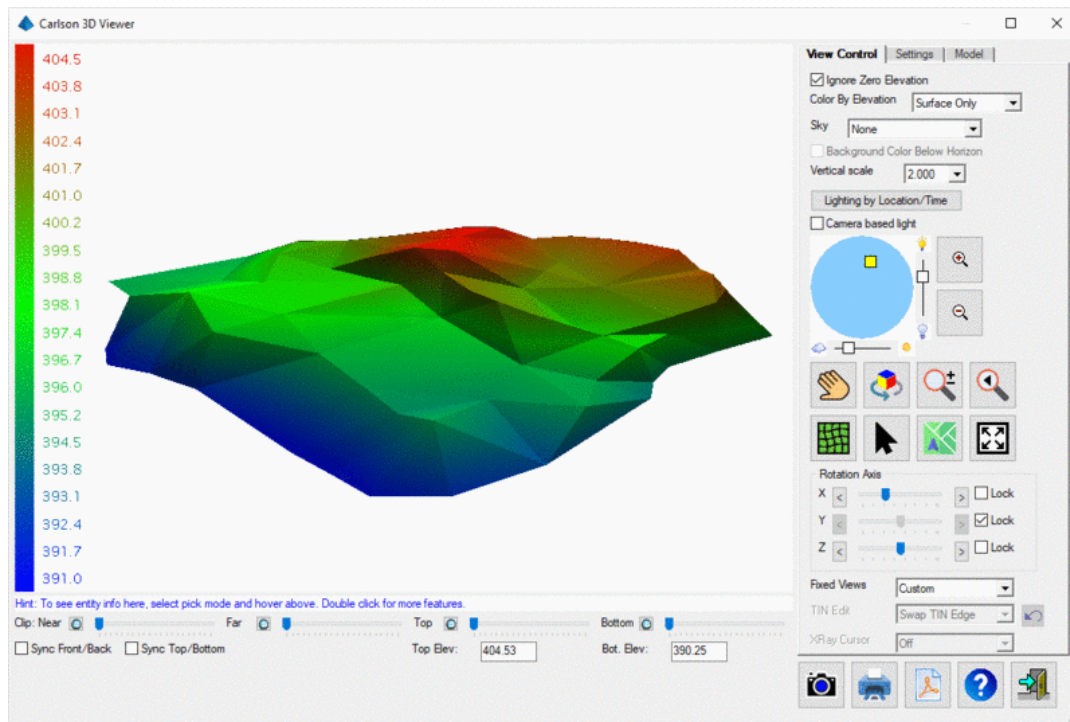
If a secondary dialog box appears prompting for a coordinate file (*.crd), click **Cancel** or select the coordinate file created earlier in this lesson if you so choose and accept the default parameters that follow.

A "TIN" file representing the pile (*Final*) surface is written.

3. *Optional:* Similar to the Check Surfaces step under Two Grid Surface Volumes, this is an optional step to check that the surface(s) are correct. Re-issue the View – 3D View – **Surface File Viewer** command and when prompted:

Select Surface File (dialog): select the TIN file that was just created and click **Open** when ready.

A 3D Viewer window similar to that shown below appears:



When ready, click the **Exit** (Doorway) button.

4. Now that we have our two triangulation files, we can generate a TIN to TIN prismatic volume calculation. Of all the volume methods, this one is the most accurate since all the source data points are used in the volume model. Issue the Surface – Volumes By Triangulation – Two Triangulation Surface Volumes command. When prompted:

Select EXISTING Surface Triangulation File (dialog): select the *Base* TIN file and click **Open** when ready

Select FINAL Surface Triangulation File (dialog): select the *Pile* grid file and click **Open** when ready

Select Inclusion perimeter polylines or ENTER for none.

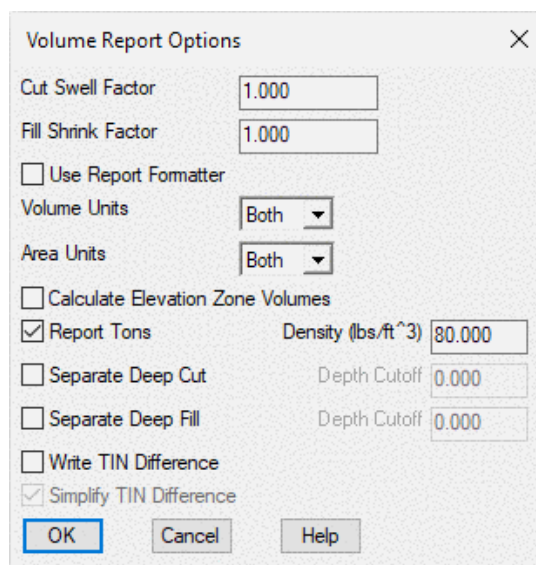
[FILter]/<Select entities>: pick the PERIMETER polyline

[FILter]/<Select entities>: press Enter

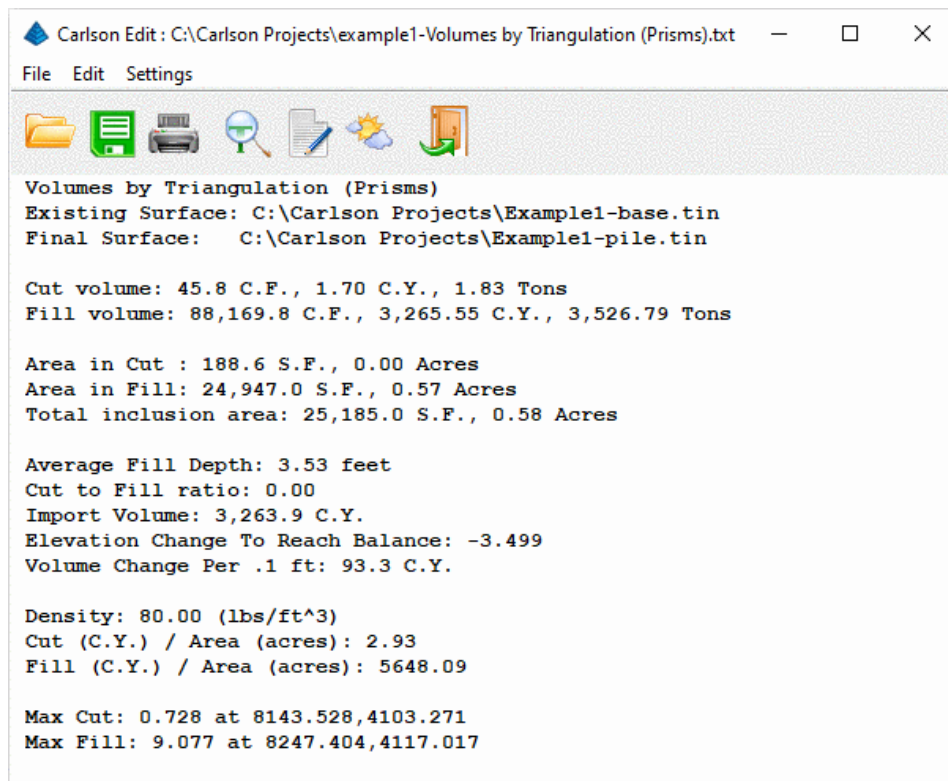
Select Exclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: press Enter

Volume Report Options (dialog): set the values as shown below and click **OK** when ready:



The volume report similar to that shown below appears:



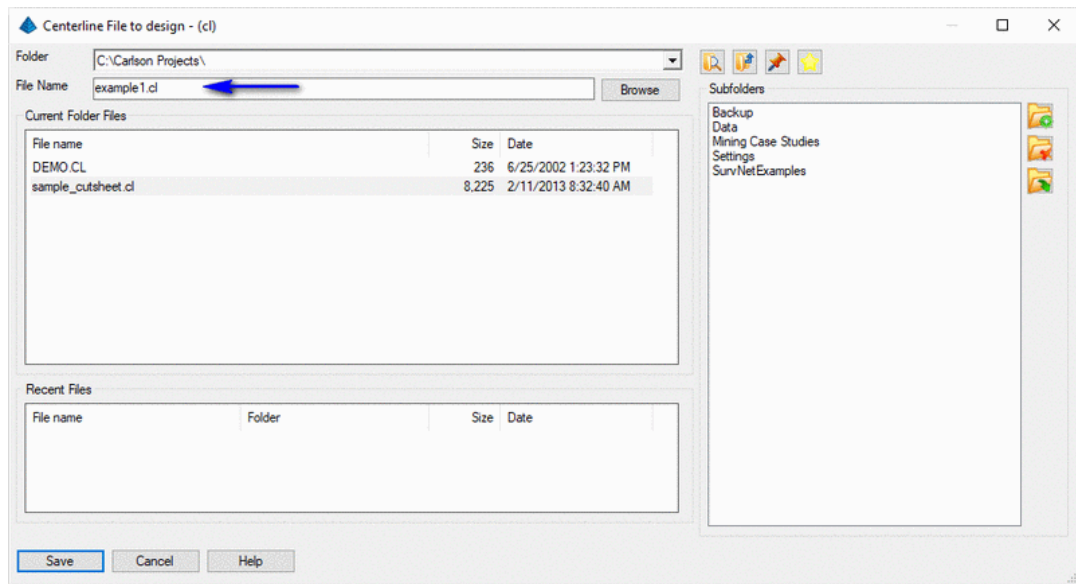
Click the **Exit** (Doorway) button to dismiss the report.

Volume Method 5 - Calculate Section Volumes

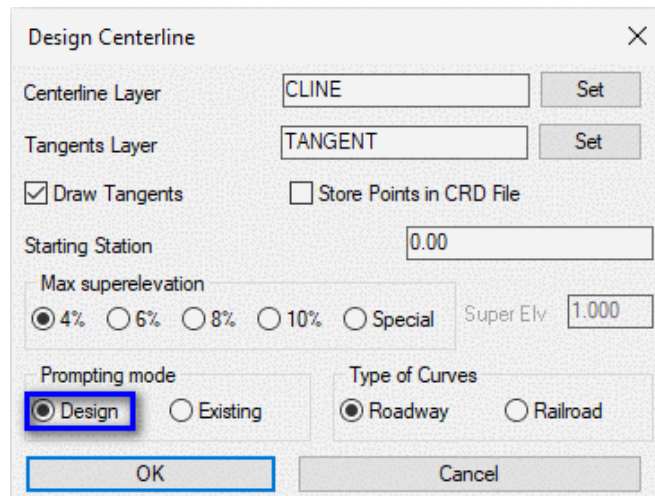
In roadway/corridor types of projects, volumes by *Average End Area* are commonly used. Let's see the results of this process.

1. Our first task will be to draw a centerline (baseline) upon which the corridor will be based. Issue the Centerline – Design Centerline command and when prompted:

Centerline File to Design (dialog): set the file name as shown below and click **Open** when ready



Design Centerline (dialog): set the values as shown below and click **OK** when ready



Pick Point or Point number: type 2016 and press Enter

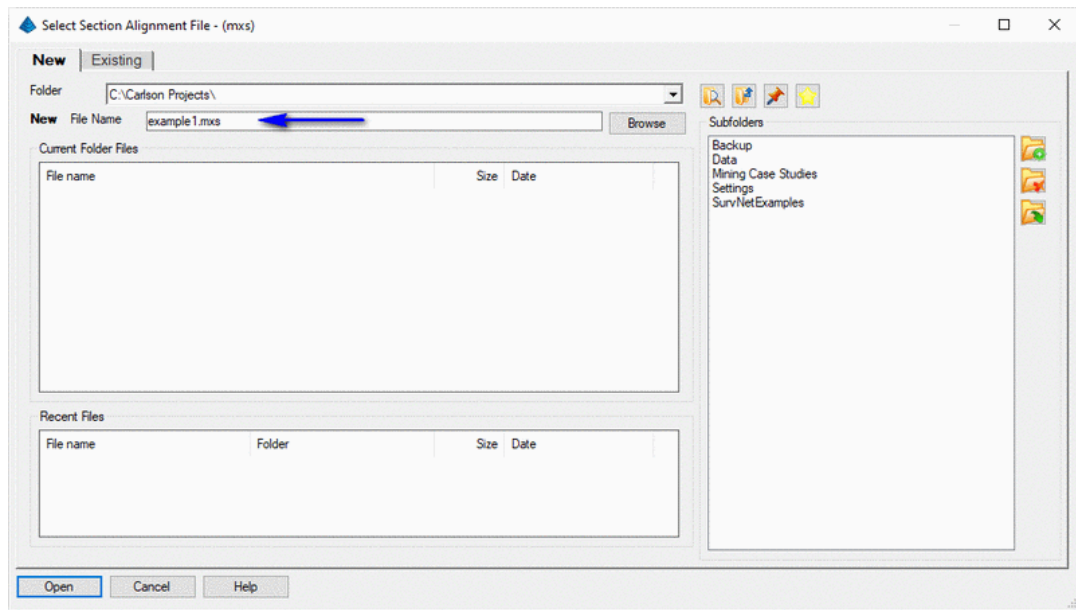
Bearing/PI/Exit/Undo/<Pick Point or Point number>: type 2004 and press Enter

Bearing/PC/PI/Exit/Undo/<Pick Point or Point number>: press Enter

A centerline is generated between the two points.

NOTE: Double-clicking on the polyline on the screen will open the Centerline Editor command.

- Our next task will be to establish the cross-section limits along this corridor centerline. Issue the Sections – Section Alignment Editor to display a dialog box similar to that shown below:



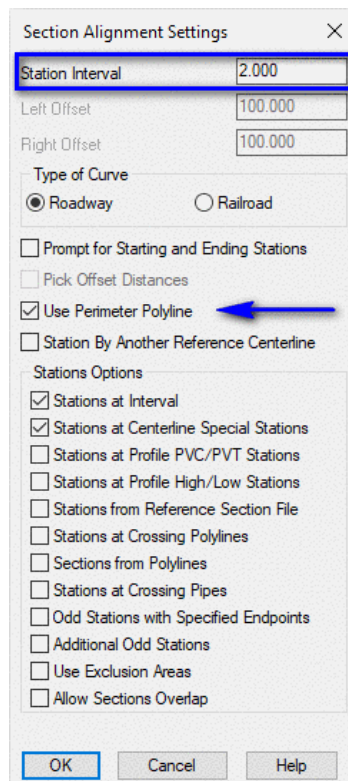
Set the file name as indicated above and click **Open** when ready. When prompted:

Polyline should have been drawn in direction of increasing stations.

CL File/<Select polyline that represents centerline>: pick the centerline polyline

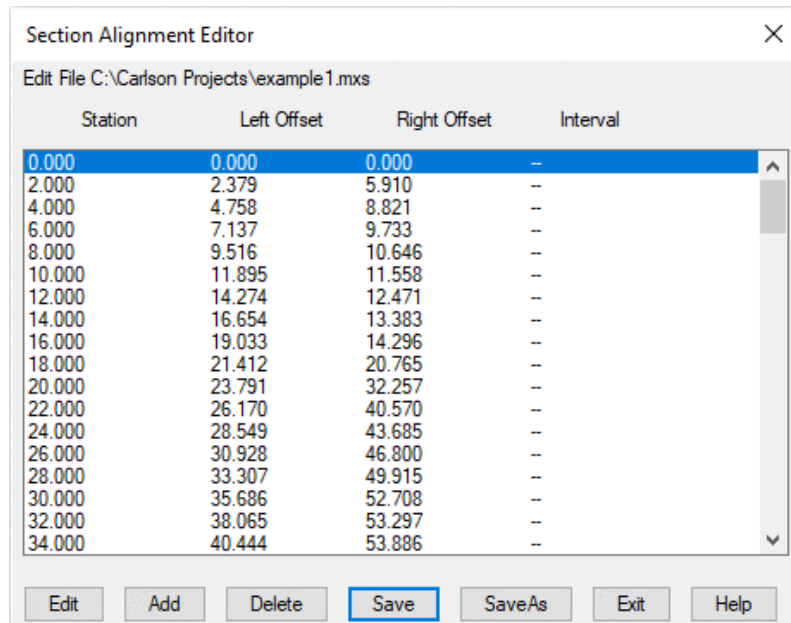
Enter Beginning Station of Alignment <0.00>: press Enter

Section Alignment Settings (dialog): set the values as shown below and click **OK** when ready



Select boundary polyline: pick the PERIMETER polyline

The program draws temporary lines in the drawing to show the positions of the sections and displays a summary of the section alignment as illustrated below:



Click the **Save** button followed by the **Exit** button.

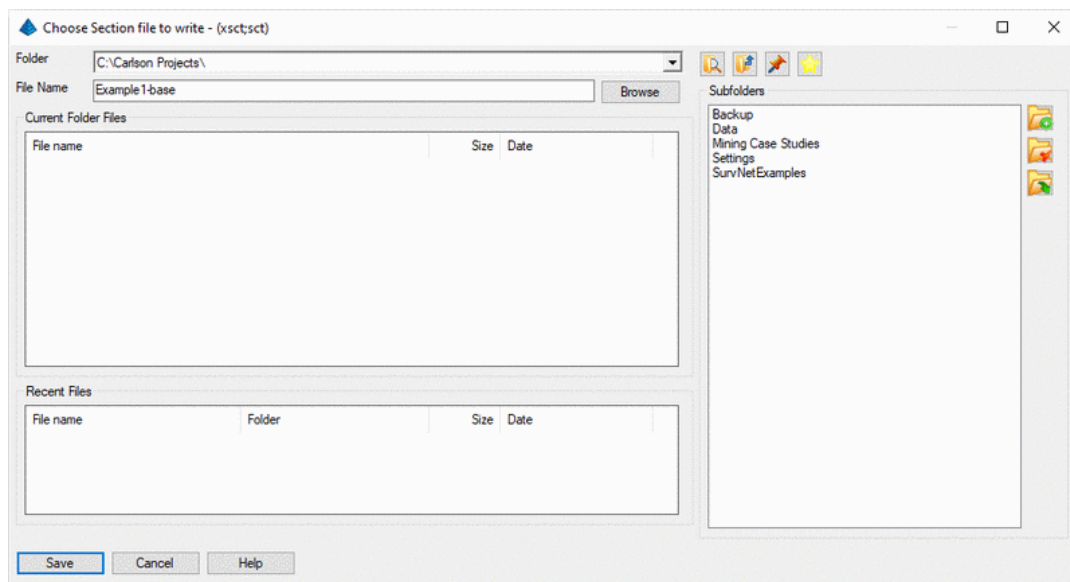
- To create the section file for the *Existing* (base) surface, issue the Sections – Create Sections from... – Sections from Grid or Triangulation Surface command. When prompted:

Choose Grid or Triangulation file to Process (dialog): specify the base.tin created earlier and click **Open** when ready

Section Options (dialog): set the option to **Prompt** and click **OK** when ready (with *Prompt* specified, you'll be prompted if the Section should change if there is an underlying change to the source TIN file)

Section Alignment File to Process (dialog): specify the section alignment (*.MXS) created earlier and click **Open** when ready

Choose Section File to Write (dialog): specify a file name as illustrated below and click **Save** when ready



The first of two section files is written.

- Let's follow the same general process for the *Final* (pile) surface with a few minor adjustments. Re-issue

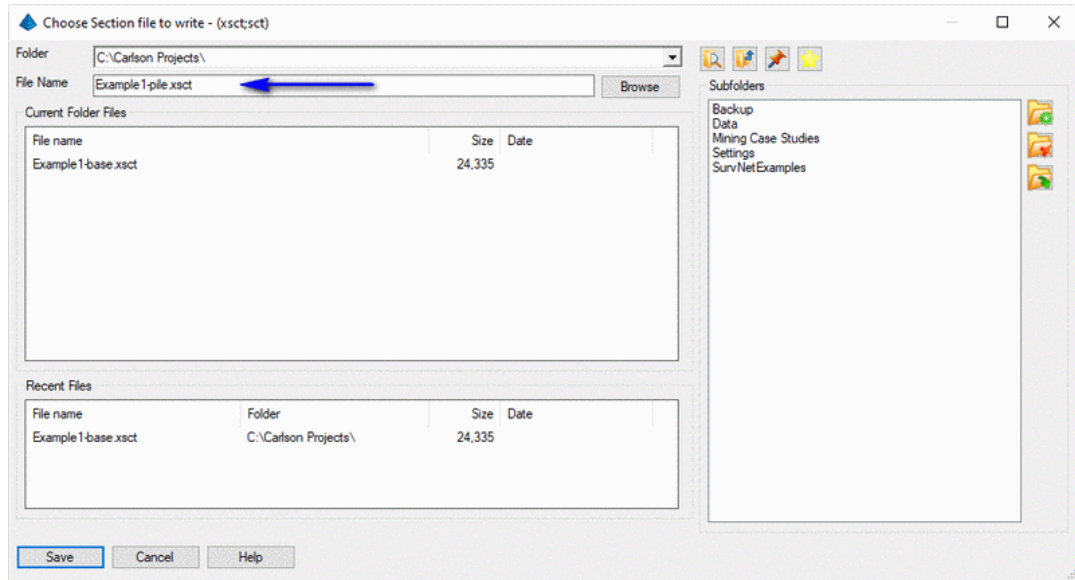
the Sections – Create Sections from... – **Sections from Grid or Triangulation Surface** command. When prompted:

Choose Grid or Triangulation file to Process (dialog): specify the pile.tin created earlier and click **Open** when ready

Section Options (dialog): set the option to **Prompt** and click **OK** when ready

Section Alignment File to Process (dialog): specify the section alignment (*.MXS) created earlier and click **Open** when ready

Choose Section File to Write (dialog): specify a file name as illustrated below and click **Save** when ready

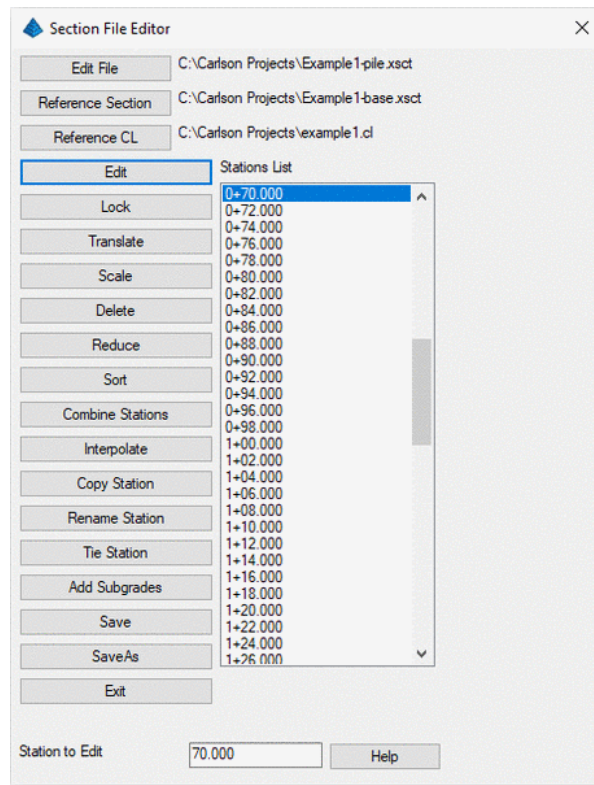


The second of two section files is written.

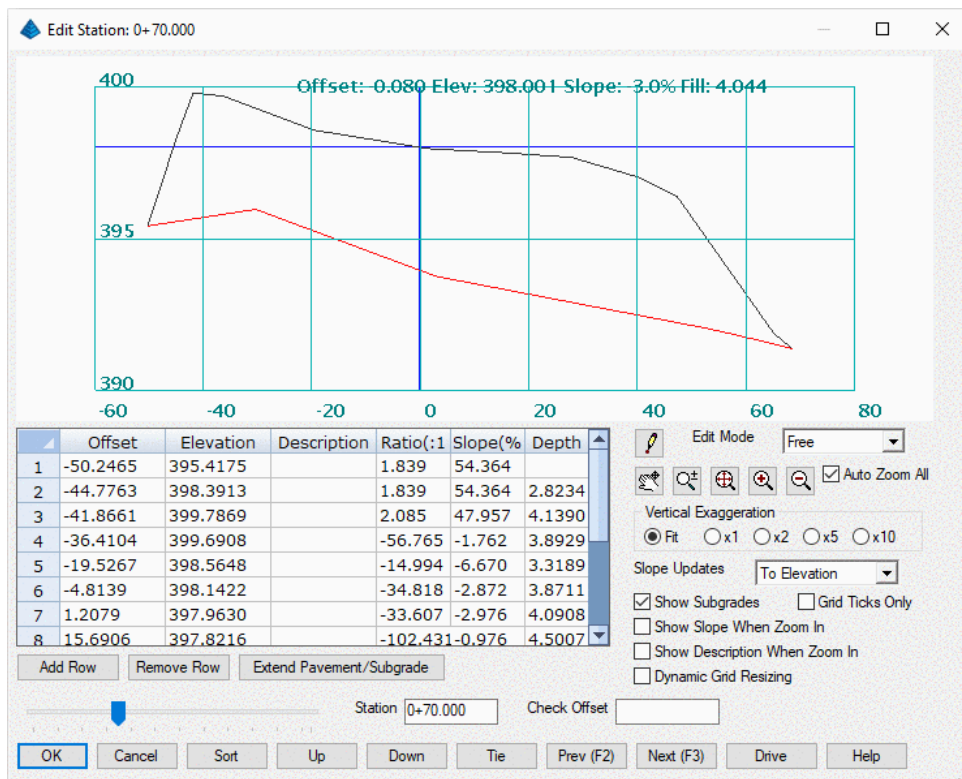
5. *Optional:* Similar to the Check Surfaces (grid) and the Check Surfaces (TIN) steps discussed earlier, this is an optional step to check that the section(s) are correct. Issue the Sections – Section **File** Editor command and when prompted:

Section File to Process (dialog): select the **pile** section file that was just created and click **Open** when ready.

A Section File Editor dialog box similar to that shown below appears:



Set the remaining values as shown above. To examine a specific section, scroll through the list and select a desired station (e.g. 0+70.00) and click the **Edit** button to display a dialog box similar to that shown below:



Review the controls in this dialog box and click **OK** when ready and then click **Exit** on its parent dialog box.

- Now that we have two section files, we can derive the volume between them. Issue the Sections – Calculate Section Volumes command. When prompted:

Section File (Existing Ground) to Read (dialog): select the *Base* section file and click **Open** when ready

Section File (Final Ground) to Read (dialog): select the *Pile* section file and click **Open** when ready

Calculate Section Volumes (dialog): set the values as shown below and click **OK** when ready

The volumes are calculated and reported, along with the cut and fill end-areas at each station as illustrated below:

Station	Cut(sf)	Fill(sf)	Interval	Cut(cy)	Fill(cy)
0+00.00	0.00	0.00			
0+02.00	0.00	0.04	2.00	0.00	0.00
0+04.00	0.00	0.17	2.00	0.00	0.01
0+06.00	0.00	0.39	2.00	0.00	0.02
0+08.00	0.00	0.69	2.00	0.00	0.04
0+10.00	0.00	1.07	2.00	0.00	0.07
0+12.00	0.00	1.55	2.00	0.00	0.10
0+14.00	0.00	2.10	2.00	0.00	0.14
0+16.00	0.00	2.75	2.00	0.01	0.18
0+18.00	0.16	3.64	2.00	0.06	0.24
0+20.00	1.37	5.63			

Review the results and click the **Exit** (Doorway) button to dismiss the report.

Results Summary

As mentioned earlier, each volume methodology has a particular use and knowing the application(s) for each method will serve you well. Here is a summary of our results:

Method	Fill (c.y.)	Cut (c.y.)	When to use
Stockpile Volumes	3,264.22	- na -	Quick and approximate Fill-only volumes are sufficient
Volumes By Layers	3,264.22	1.59	Quick and for large data sets when approximate Cut and Fill volumes are sufficient
Two Grid Surface Volumes	3,264.60	1.61	Large data sets when approximate Cut and Fill volumes are sufficient
Volumes By Triangulation	3,265.55	1.70	Large or small data sets when precise surface-to-surface Cut and Fill volumes are needed
Calculate Sections Volumes	3,265.59	1.70	Corridor studies where end-area averages are sufficient

Volume Summary/Comparison

This completes the tutorial: Calculate Volumes By Five Methods.

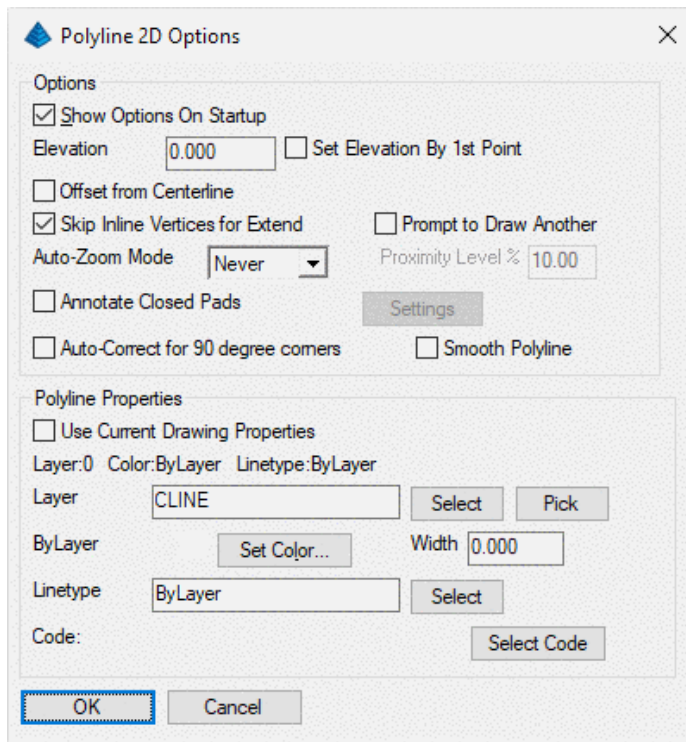
Basic Road Design with Volumes

This tutorial requires the Civil Design program.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **Example2.dwg** file. It will look like the illustration below (without the curved road).

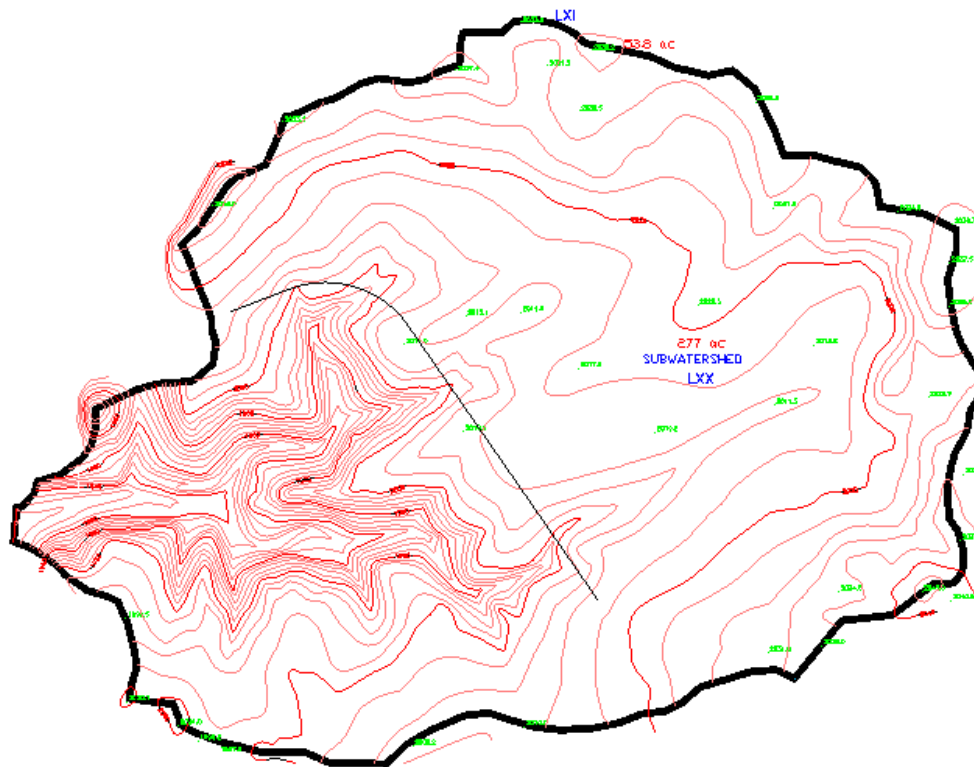
2. Activate the Civil module via Settings – Carlson Menus – **Civil Menu**. Since *Example2.dwg* will also be used multiple times throughout this tutorial, you might want to make a copy so that you don't overwrite the initial drawing. Issue the File – **Save As** command and specify a different name, such as **Example2r.dwg**.
3. We'd like to generate a sample roadway centerline. Pick the Draw – 2D Polyline command (or simply type-in **2DP** at the command line) which starts the Carlson enhanced Polyline command. If the Polyline 2D Options dialog box appears, set the values shown below and click the **OK** button:



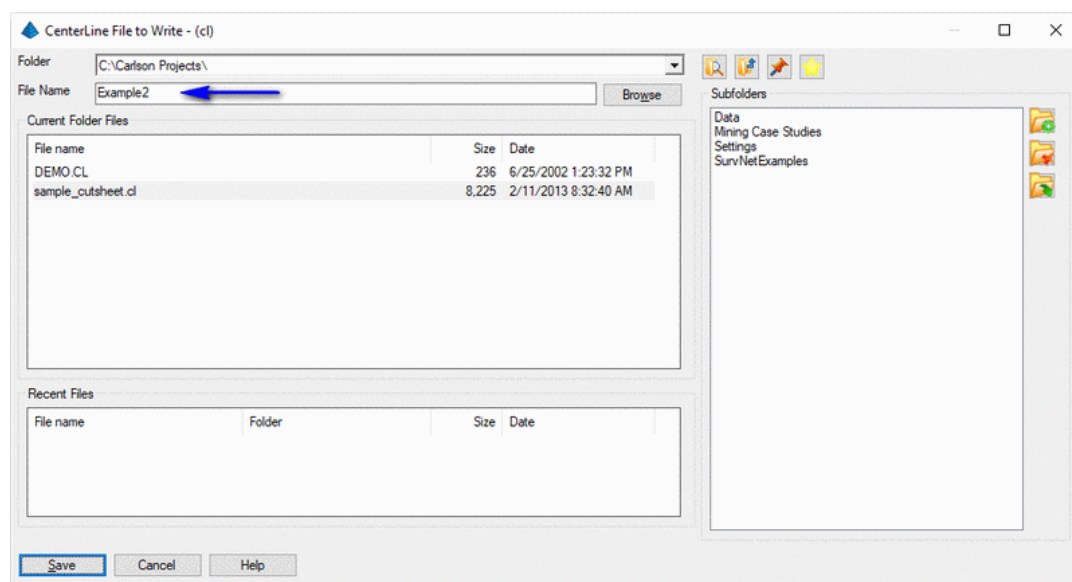
When prompted:

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: 1857700,159400
[Arc/Close/Distance/Follow/Undo/<Pick point or point numbers>]: D
Enter Distance [Meters/<Feet>/Chains/Links/Rods/Pick/Quit]: F
Enter Distance [Meters/<Feet>/Chains/Links/Rods/Pick/Quit]: 310
Define direction method [Cursor/Line/Pick/<Angle>]? press Enter
Code: 1-NE 2-SE 3-SW 4-NW 5-AZ 6-AL 7-AR 8-DL 9-DR
Enter angle code (1-9) <4>: 1 (for a northeast bearing)
Enter bearing (dd.mmss): 68.5525
Enter Distance [Meters/<Feet>/Chains/Links/Rods/Pick/Quit]: Q
[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: A
[Radius pt/radius Length/Arc length/Chord/Second pt/Undo/<Endpoint or point number>]: L
Specify radius length: 500
Curve direction [Left/<Right>]? press Enter
[Arc length/Chord length/Delta angle/Tangent-out/<End point or point number>]: D
Specify delta angle (ddd.mmss): 76.2405
[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: L
<Enter or pick distance>: 1663.2721
Segment length: 1663.27, Total length: 2640.00
[Arc/Close/Distance/Extend/Follow/Line/Undo/<Pick point or point numbers>]: press Enter

The result should resemble the image below:



- Our next task will be to create a centerline file necessary for the final road design routine. Issue the Centerline – Polyline to Centerline File command (other methods are available for Centerline creation as described in the Centerline Menu of the documentation). Provide a centerline (*.cl) file name as shown below and click **Save** when ready:



When prompted:

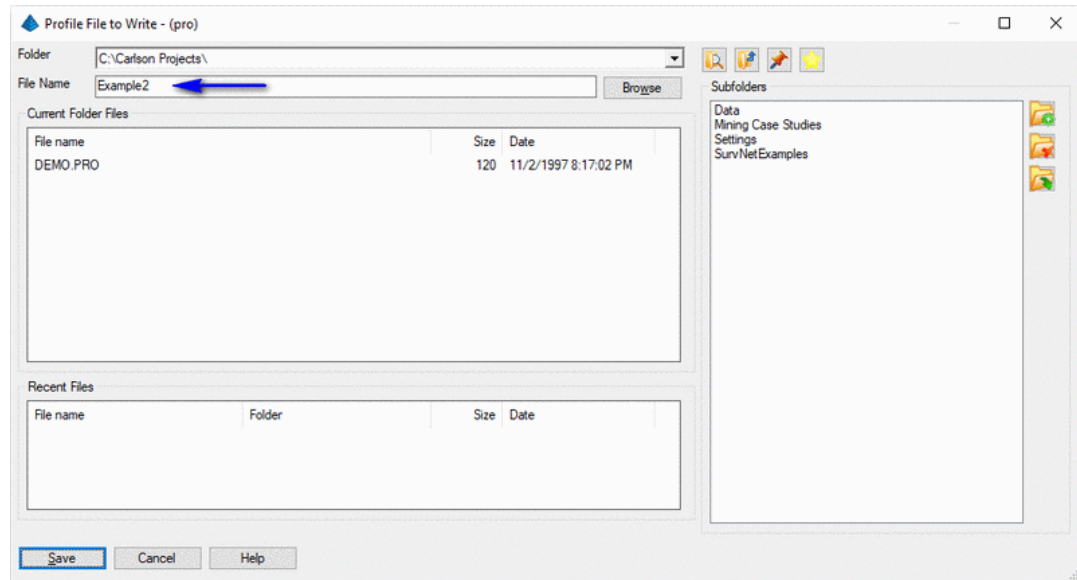
Polyline should have been drawn in direction of increasing stations.

Select polyline that represents centerline: pick polyline representing the centerline

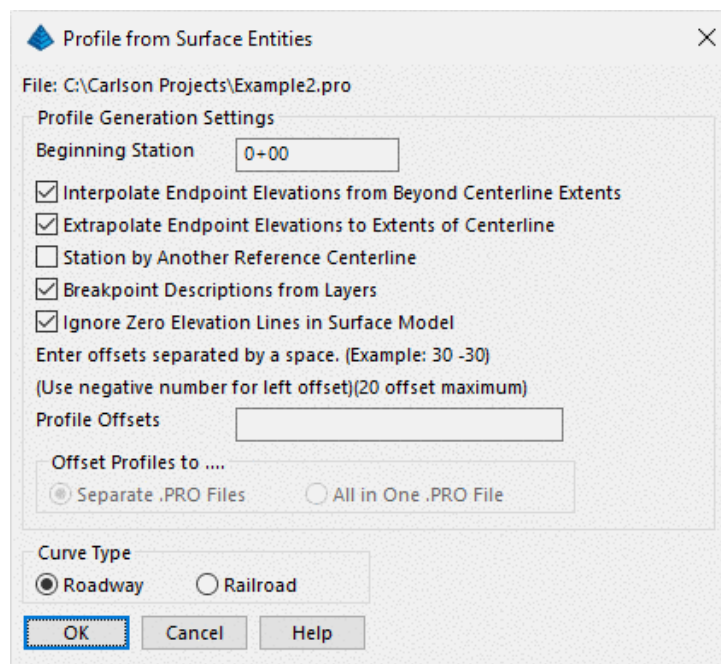
Centerline station [Reverse/Ending/<Beginning: 0+00>]: press Enter

Press ENTER to continue. press Enter

- Now we will make a profile file (*.pro) based on the centerline geometry we just created. Issue the Profiles – Create Profile From – Profile from Surface Entities. Supply the new profile name as shown below and click **Save** when ready:



A settings dialog as shown below appears. Set the values as shown and click **OK**:



When prompted:

Polyline should be drawn in direction of increasing stations.

CL File/<select centerline polyline>: C (and select the *.cl file created above)

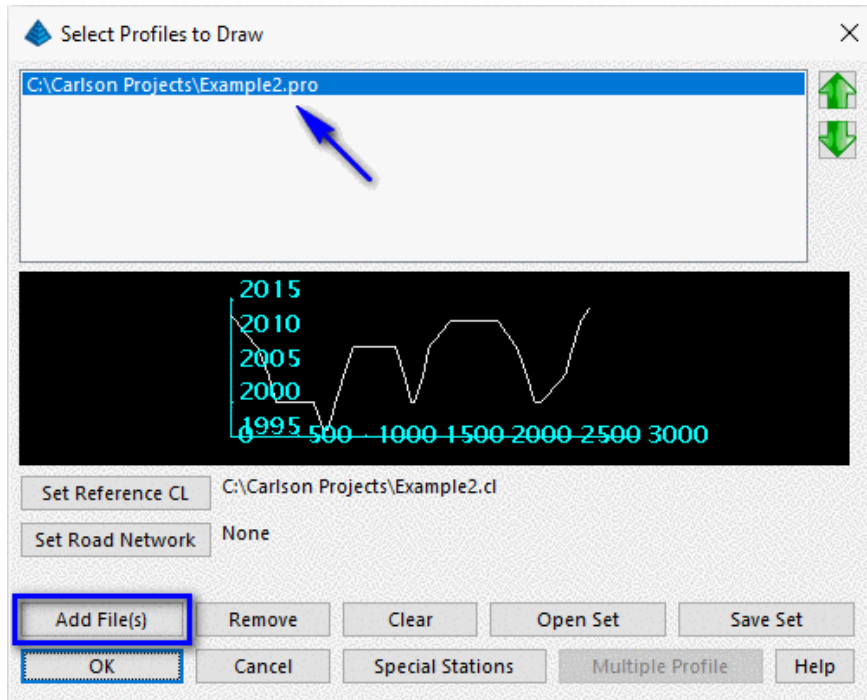
Select Lines, PLines, and/or 3DFACEs that define the surface for profiling.

[FILter]/<Select entities>: type ALL and press Enter twice

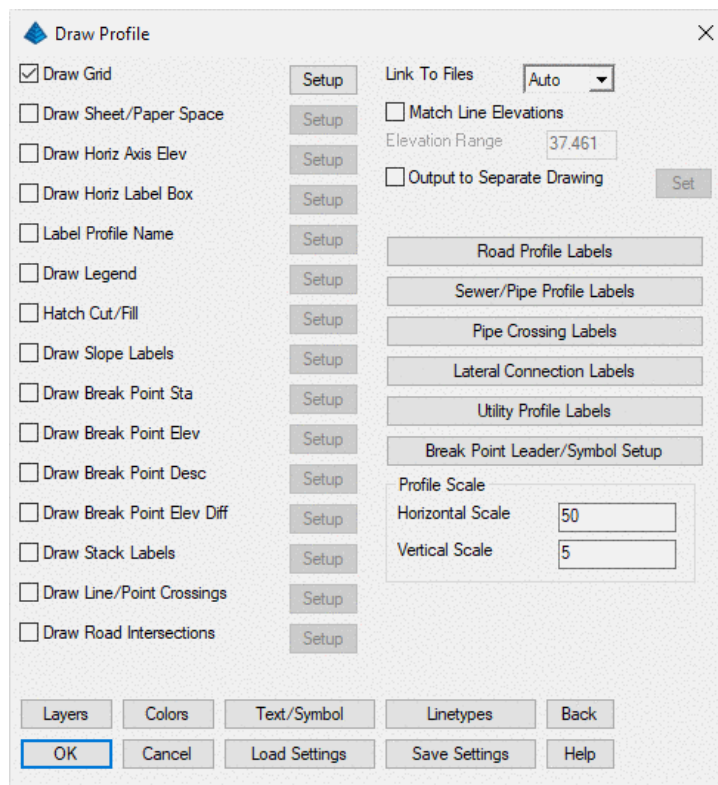
The profile data is written to file.

NOTE: Common practice is to build a surface model from any and all data that carries an elevation. However, there are several Carlson **Create Profile from...** routines and we opted to work with a routine that gets its information "direct from the source" (*i.e.* the contours themselves).

6. With the profile created, let's place the data into the drawing for visualization purposes. Use the Profiles – Draw Profile command and use the **Set Reference CL** and **Add** buttons to set the values as shown below. Click **OK** when ready:



The window below will appear. Set the values as shown making note to set the **Horizontal Scale** to **50** and the **Vertical Scale** value to **5**. This will establish a 10X vertical exaggeration of the profile. Click **OK** when ready:

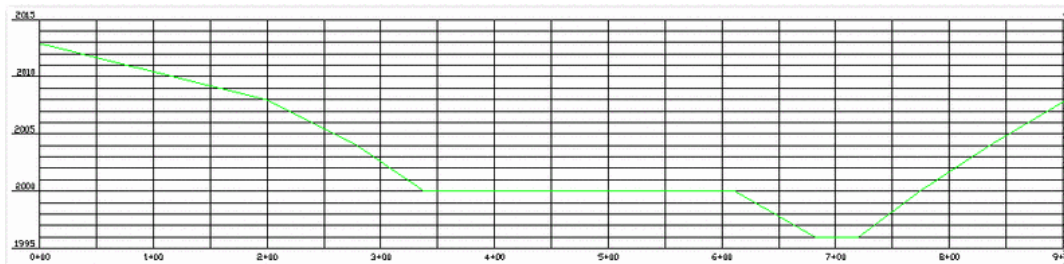


When prompted:

Profile Grid Range (Dialog):

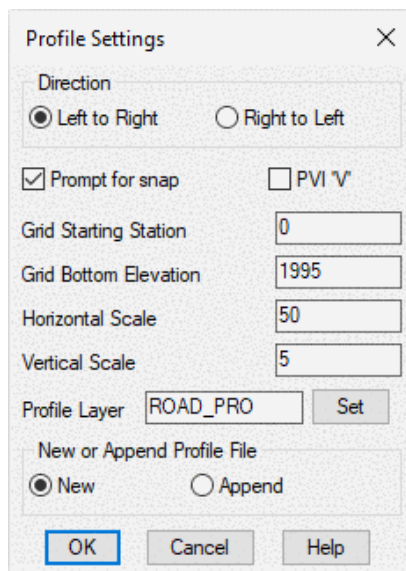
Pick Starting Point for Grid <0.00,0.00>: pick an arbitrary and desirable location on screen

Use the View – **Window** command to study the result as desired. Your profile should look similar to this.



NOTE: The "flat spots" shown in this profile are the result of extracting the profile data directly from the contours. Extracting a profile from a surface model is a more common approach in today's computer age.

- Now we will design a proposed road centerline profile in relation to the existing ground profile. Use the Profiles – Design Road Profile – Design Road On Profile Grid. The following dialog box will appear. Since we followed up the **Draw Profile** command with this one, it was able to determine proper startup values for the dialog:



Choose **OK** on this dialog. A new file creation dialog box will appear, asking for an output file name. Enter a name such as **Example2-Pr**, and click **Save**. When prompted:

Pick Lower Left Grid Corner <0.00,0.00>[endp on]: if you have not moved your profile grid, press Enter (Carlson has *endpoint* osnap active to make the pick accurate)

At this point another dialog will appear in the upper left corner. Initially, it will display only station and elevation. Once a beginning point has been designated, it will also display the relative difference from the last point to the cursor position (illustrated below). This can be an aid in determining acceptable slopes for your design.

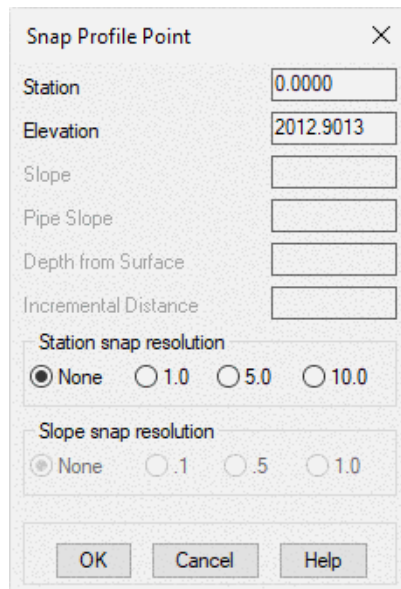


When prompted:

Enter station or pick a point (Enter to End): end

Snap to END of: pick the left-most endpoint of the existing ground profile as a tie in point

The following dialog appears. Choose **OK** to accept the defaults:



prompting resumes:

Station of second PVI or pick a point (U,E,D,Help): 1111.01

Percent grade entry/<Elevation of PVI>: 1999.37

Station of next PVI or pick a point (U,E,D,Help): 1911.64

Percent grade entry/<Elevation of PVI>: 2002.66

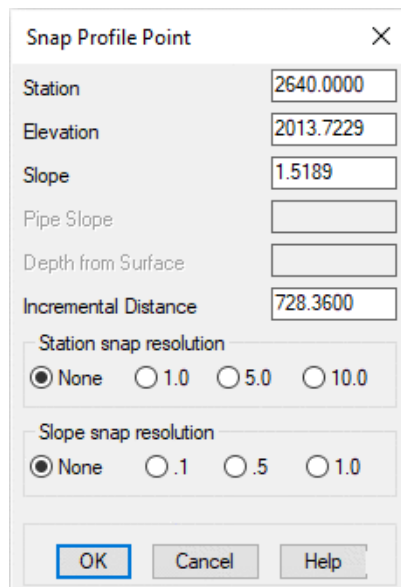
View table/Unequal/Through pt/Sight dist/K-value/<Vert Curve Length>: 500

Use these values [<Yes>/No]? Y

Station of next PVI or pick a point (U,E,D,Help): end

Snap to END of: pick the far-right endpoint of the existing ground profile as a tie in point

The following dialog appears. Choose **OK** to accept the defaults:



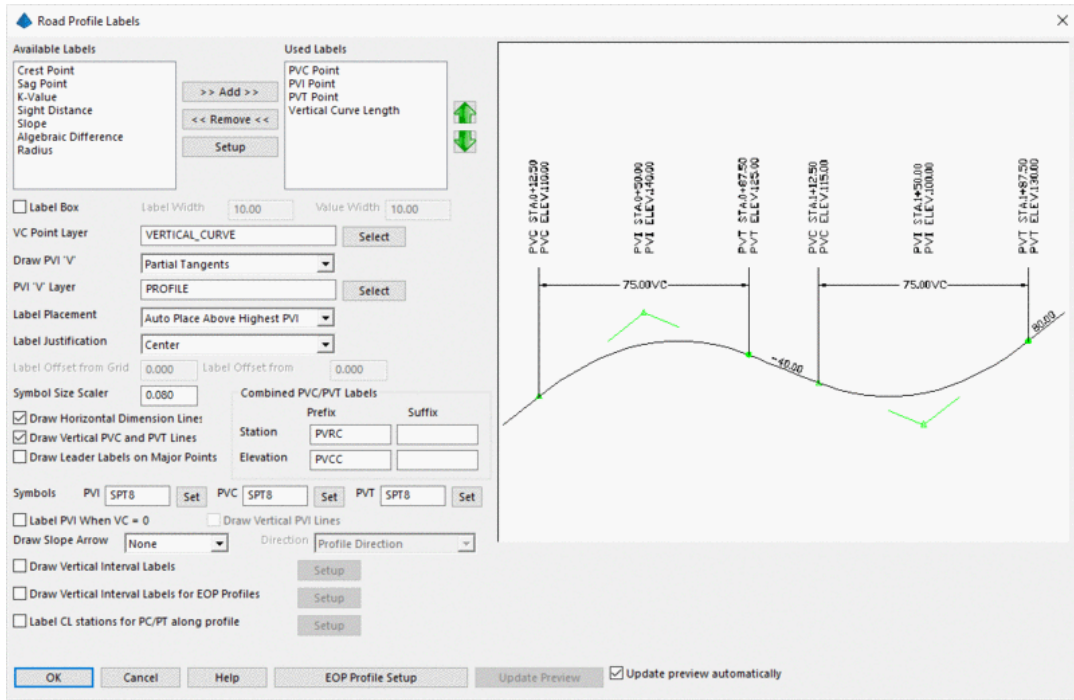
prompting resumes:

View table/Unequal/Through pt/Sight dist/K-value/<Vert Curve Length>: 500

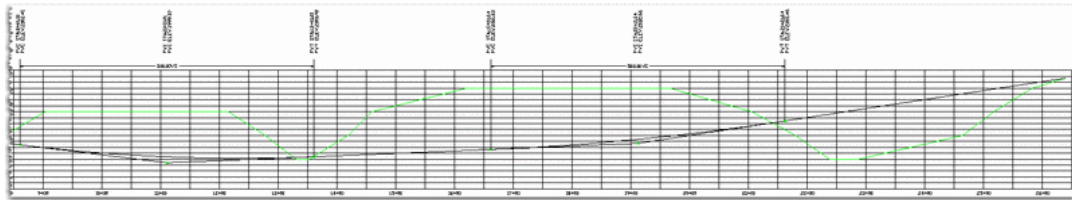
Use these values [<Yes>/No]? Y

Station of next PVI or pick a point (U,E,D,Help): press Enter

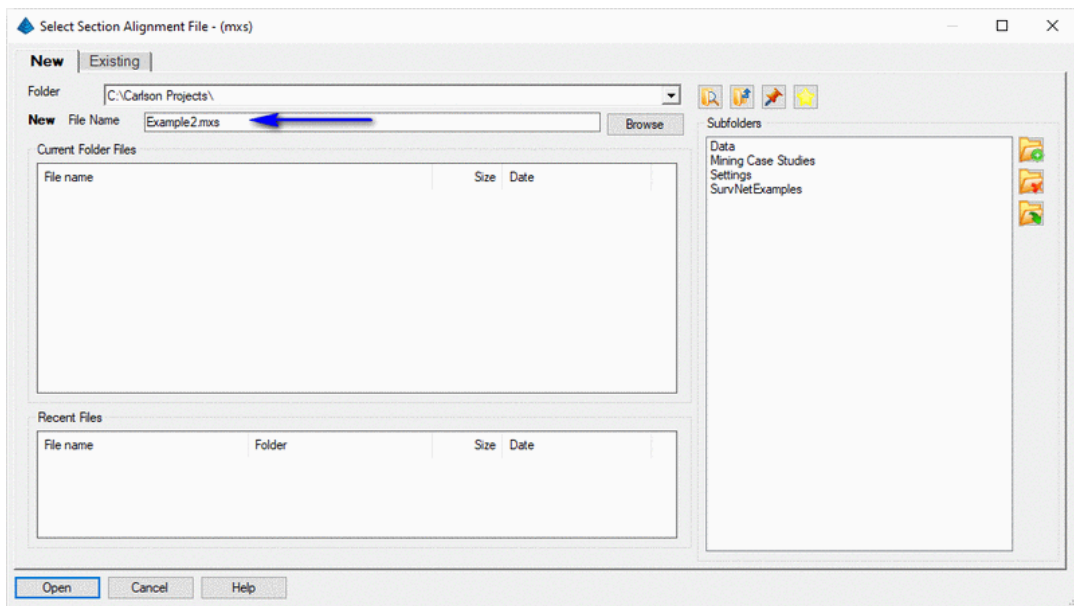
At this point the following dialog appears. Change settings to match, and choose **OK**:



Carlson will now finish the road design, and your drawing should like the following:



- Now we will create the alignment for our cross-section file. This step gives the section interval and the offset left and right from our centerline. Use the Sections – Section Alignment Editor command and choose the **New** tab. Type the name indicated below and click **Open**:

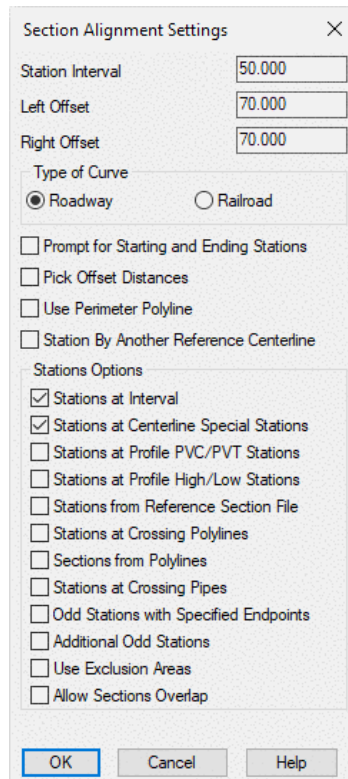


Notice how all files can have the same name in this road design portion, as they all have a unique file extension. So for the organization of various jobs, it is sometimes helpful to have all of the files with the same name. When prompted:

Polyline should have been drawn in direction of increasing stations.

CL File/<Select centerline polyline>: C (and select the Centerline file created earlier)

The dialog will appear as shown, enter in the stations and offsets exactly as they appear here. This will give the needed detail for the road design routine:

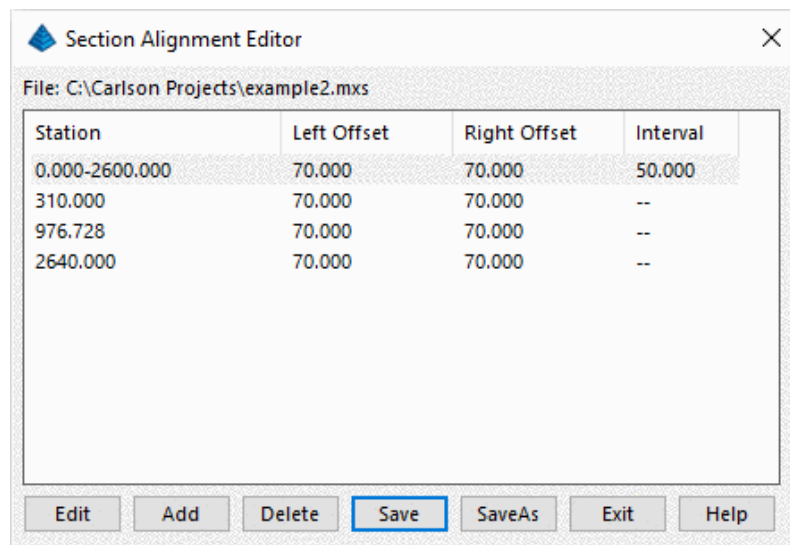


The dialog box 'Section Alignment Settings' contains the following fields and options:

- Station Interval: 50.000
- Left Offset: 70.000
- Right Offset: 70.000
- Type of Curve: Roadway, Railroad
- Prompt for Starting and Ending Stations
- Pick Offset Distances
- Use Perimeter Polyline
- Station By Another Reference Centerline
- Stations Options:
 - Stations at Interval
 - Stations at Centerline Special Stations
 - Stations at Profile PVC/PVT Stations
 - Stations at Profile High/Low Stations
 - Stations from Reference Section File
 - Stations at Crossing Polylines
 - Sections from Polylines
 - Stations at Crossing Pipes
 - Odd Stations with Specified Endpoints
 - Additional Odd Stations
 - Use Exclusion Areas
 - Allow Sections Overlap

Buttons: OK, Cancel, Help

Choose **OK**, and another window appears that allows for any station editing or changes. It all looks good here, so click **Save** and then **Exit**:



The dialog box 'Section Alignment Editor' shows the following table:

Station	Left Offset	Right Offset	Interval
0.000-2600.000	70.000	70.000	50.000
310.000	70.000	70.000	--
976.728	70.000	70.000	--
2640.000	70.000	70.000	--

Buttons: Edit, Add, Delete, Save, SaveAs, Exit, Help

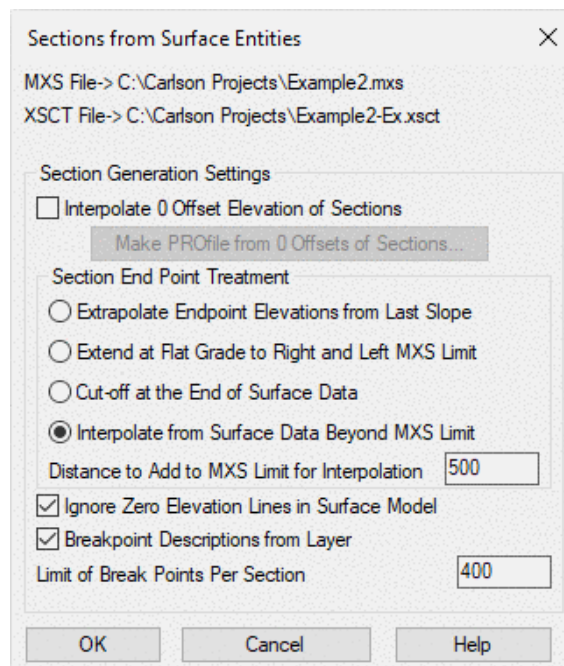
The Section Alignment file (*.xms) is now written. There is now a preview of the section alignment lines shown on the centerline. These are just images, if the drawing is regenerated, they will disappear (they can be drawn permanently if desired).

9. Next, we will create the actual section file (*.SCT) from the contours, in combination with the Section Alignment file (*.MXS). Run the Sections – Create Sections from – Sections from Surface Entities command. We will use the contours and breaklines for surface elevations, as we did with generating the profile. When prompted:

MXS File to Process (Dialog): Specify the MXS file that we just created to read for the alignment

Section File to Write: Provide a file name such as Example2-Ex (e.g. the "-Ex" portion of the name signifying *Existing* section conditions)

A dialog box with additional settings appears:



We'll enter a distance of 500 feet to add to our MXS limit of 70. This will search farther for contour elevations, then choose **OK**. When prompted:

Select Lines, Polylines, and/or 3DFaces that define the surface.

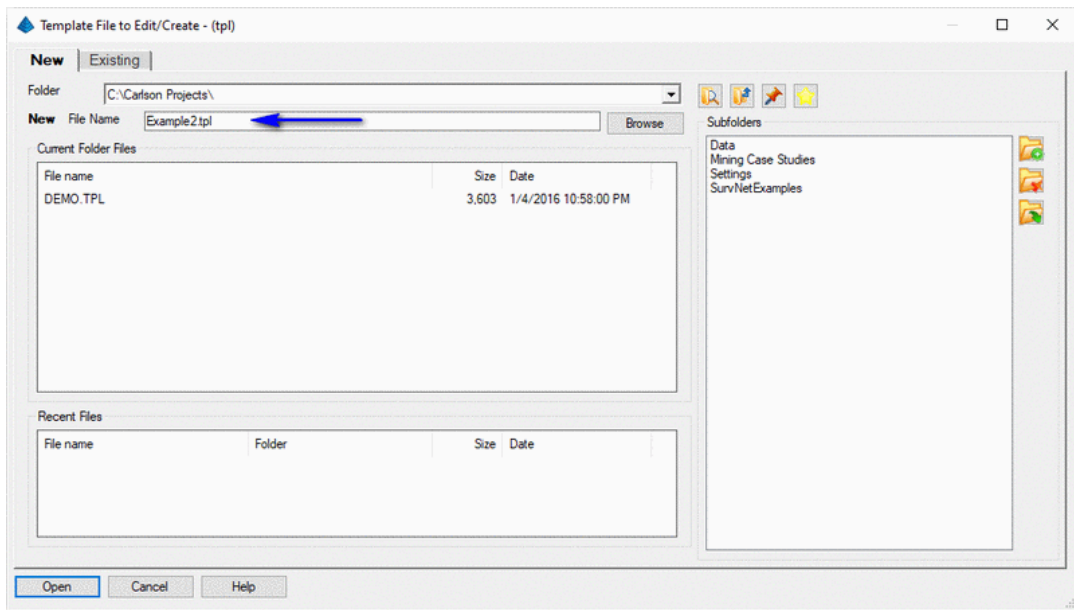
[FILter]/<Select entities>: type ALL and press Enter

[FILter]/<Select entities>: press Enter

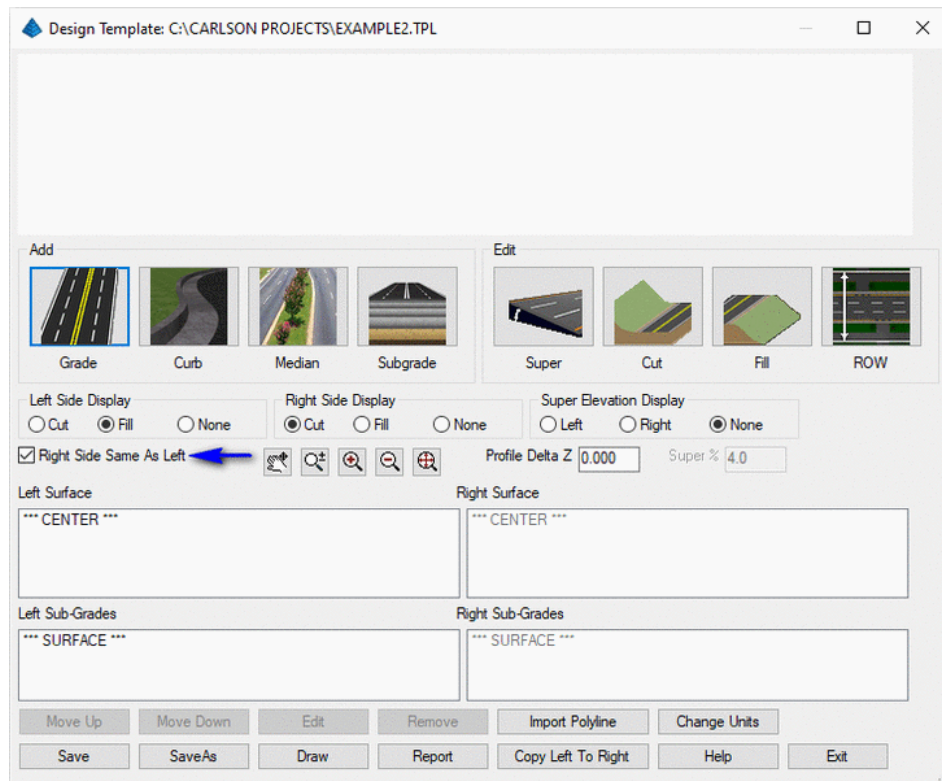
The section file is generated.

10. To continue the road design, let's design a wide boulevard, 27' of drivable pavement, with curb and gutter on the outside. Whenever a cut is within rock, the cut slope will employ a 0.5:1 slope rather than the typical 3:1 slope. At the top of rock, the cut will revert to 3:1. In fill, the condition will be 3:1 for fill under 6' and 2.5:1 for fill over 6' in depth. Pavement depths will be 6" of asphalt (consisting of a 2" wearing coat and a 4" asphalt base) and 8" of crushed aggregate.

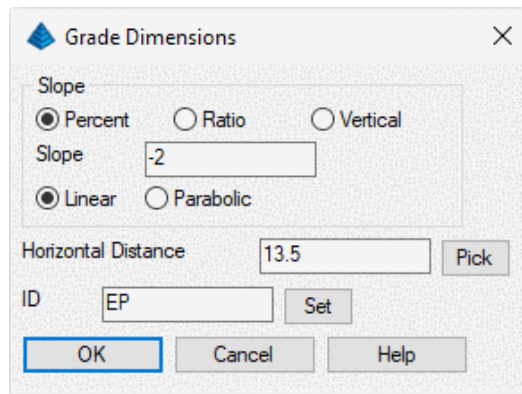
Issue the Roads – Design Template command and click on the **New** tab to create the file name as shown below. Click on **Open** when ready:



A large dialog box appears as shown below. In it, you enter segments of the template, which typically work outwards from the middle as you add more lanes, curbs and shoulders. We will enter a symmetrical template, with 13.5' pavement sections either side of centerline, connecting to a curb and gutter. Then we'll add an 8' shoulder.



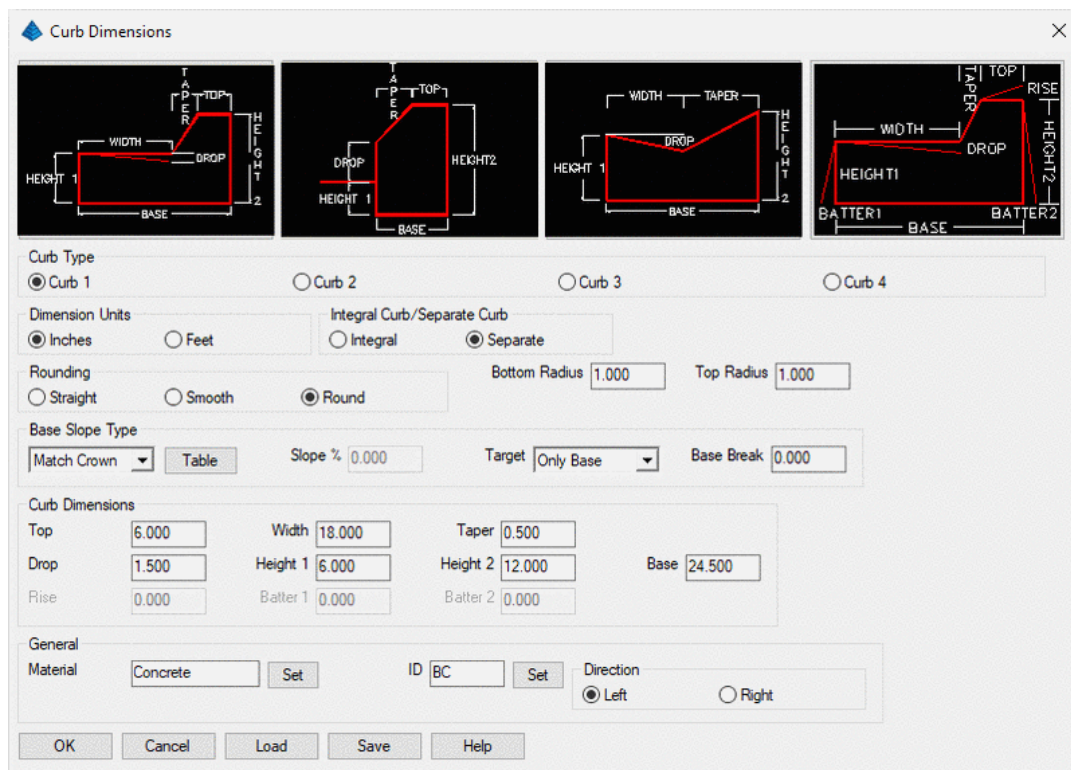
For the lanes, click the **Grades** icon. This leads to a child dialog as shown next:



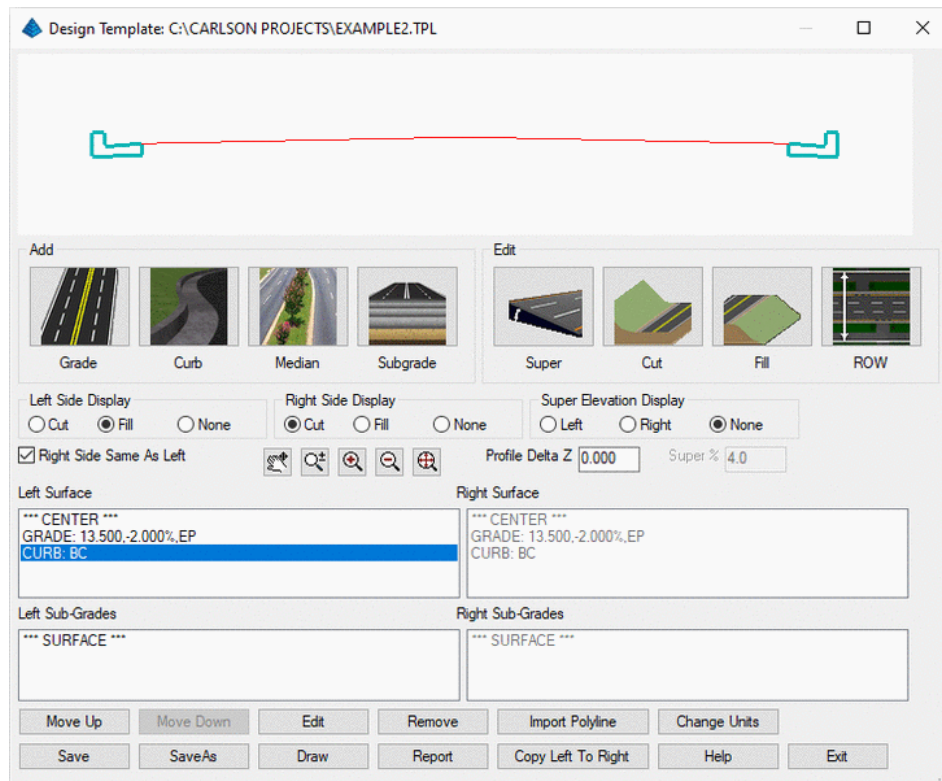
Fill out as shown. It's important to note that a downhill pavement from a crown in the middle is entered as a negative slope. That is, it is -2% heading from centerline outward, regardless of which side of centerline we are speaking of. Slope is independent of the profile grade point. It is also important to enter an ID whenever requested as these can be referenced later for advanced placement control.

A breakpoint in a shoulder in superelevation could be defined as occurring at $EP+3$, as opposed to the exact offset distance from centerline. The advantage of the parametric $EP+3$ equation is that if the road lane width expands (e.g. for a passing lane), but the shoulder always breaks 3 feet beyond the edge of pavement, then $EP+3$ is the most effective way to reference the breakpoint. Click **OK** and note that the lanes show up in the preview window at the top.

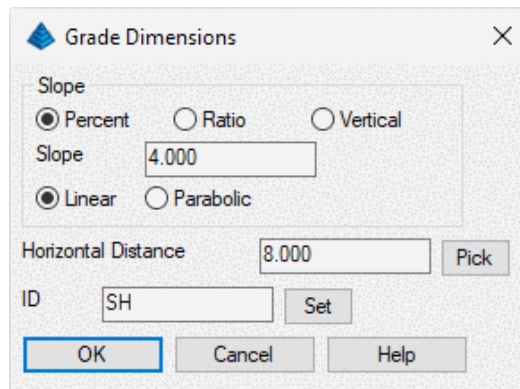
Next, we will add a curb. Click the **Curb** icon and fill out as shown:



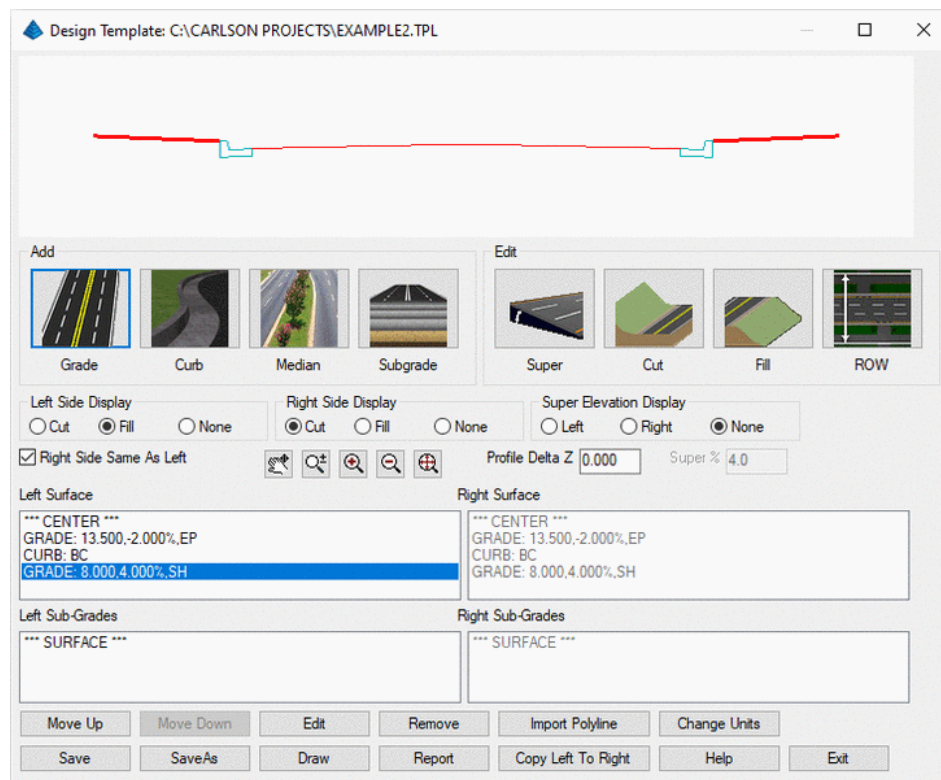
It is often a good idea to "match crown"... to make the curb match the slope of the last pavement lane (e.g. the 2% pavement slope above). But if your curb tilts downward more (like -3%), then use a *Special Base Slope Type*. If it is flat, by all means click on *Flat Base*. Click **OK**. Here's what our screen looks like so far:



Next, we will add a shoulder, going uphill at 4% for 8'. Notice what is happening. You are "current" on the *Curb* entry, so if you add another *Grade*, it will append after the curb, and add to the back of the curb. If you were to click on the *GRADE: 13.500, -2.000%, EP* segment, you would add a second "lane" before the curb, which is NOT what you want. Click on the **Grades** icon with the *CURB: BC* item highlighted. Fill out the dialog as shown:



That's it for the surface! Here's what our screen looks like now:



NOTE: As you select the different items within the component list, the viewer window will highlight the current selection.

Now we have subgrade material(s) and catch-slopes to consider. Let's turn our attention to subgrade materials and consider: if our road materials are a total of 14" deep (2" wearing, 4" asphalt, 8" stone) and our concrete gutter is 6" deep, do we want an aggregate stone "bed" to extend past the curb & gutter which is then covered by dirt? The most complex concept is the "wrap-around" subgrade, so let's go for it.

Select the **Subgrade** icon. We'll do three subgrade surfaces: first asphalt-wearing, which will run straight out and hit the curb, then asphalt-base which will extend to the gutter face and then tie vertically upward and then aggregate, which will run out past the back-of-curb and wrap back.

For any subgrade, we still do the vertical offset as a negative distance (negative meaning down). But follow this concept: we start it from the offset 0 and keep going at "Continue Slope" until it hits something (the curb). This approach isn't recommended if there is nothing to intersect. But it will run into the curb in this case. This *Continue Slope* concept works perfectly for shallow asphalt and concrete materials that will bump into a curb, when extended.

Subgrade Dimensions

Subgrade Type: Roadway Cut Slopes Fill Slopes

Distance Direction: Out In

Slope Type: Match Surface Special

Slope (%) 0.000 Tie Undershoot Vertically at Hinge

Intersect Surface: Continue Slope

Wrap Height (in) 0.000 Tie Slope (%) 0.000

Horizontal Offset: 0

Vertical Offset (in): -2.000 Units: Feet Inches Match Existing Surface

Distance: EP

Material: Asph-Wearing Coatings/Notes:

Super Elevation Settings

Low Side	High Side
Pivot Offset: <input type="text"/>	Pivot Offset: <input type="text"/>
Max Slope After Pivot (%): <input type="text"/>	Max Slope After Pivot (%): <input type="text"/>
Slope Type After Pivot: <input checked="" type="radio"/> Normal <input type="radio"/> Special	Slope Type After Pivot: <input checked="" type="radio"/> Normal <input type="radio"/> Special
Standard Slope Percent: 0.000	Standard Slope Percent: 0.000
Minimum Slope Percent: 0.000	Minimum Slope Percent: 0.000

Complete as shown above, and click **OK**. Repeat the **Subgrade** option to place the base asphalt material as shown below and click **OK** when complete:

Subgrade Dimensions

Subgrade Type: Roadway Cut Slopes Fill Slopes

Distance Direction: Out In

Slope Type: Match Surface Special

Slope (%) 0.000 Tie Undershoot Vertically at Hinge

Intersect Surface: Straight Up

Wrap Height (in) 0.000 Tie Slope (%) 0.000

Horizontal Offset: 0

Vertical Offset (in): -6.000 Units: Feet Inches Match Existing Surface

Distance: EP

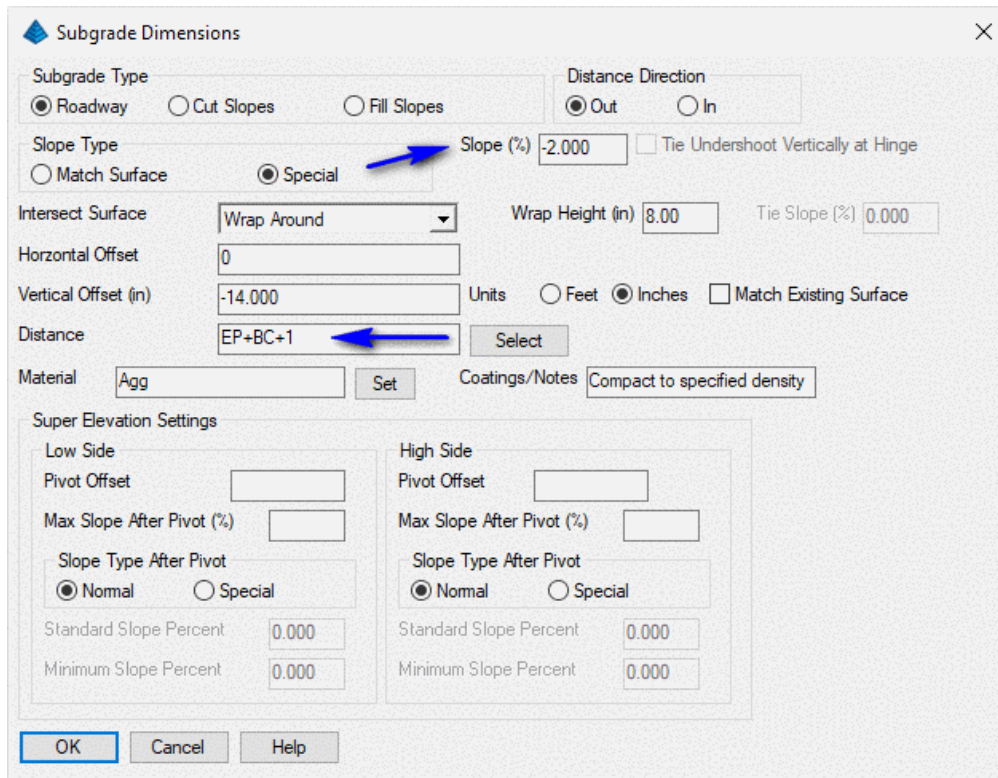
Material: Asph-Base Coatings/Notes: Apply tack coat

Super Elevation Settings

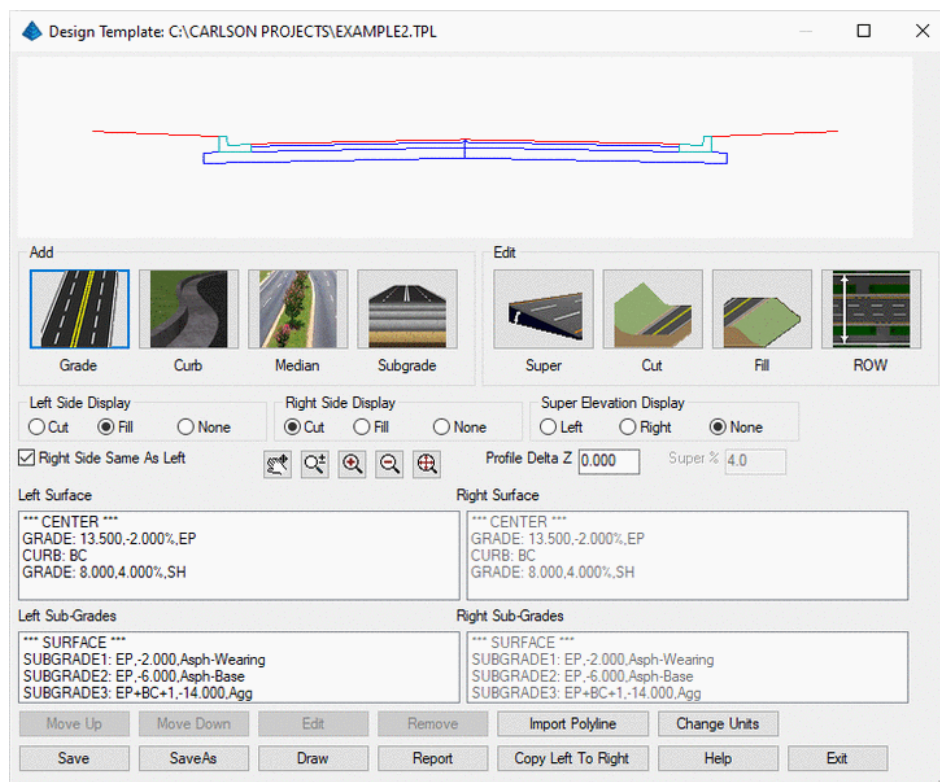
Low Side	High Side
Pivot Offset: <input type="text"/>	Pivot Offset: <input type="text"/>
Max Slope After Pivot (%): <input type="text"/>	Max Slope After Pivot (%): <input type="text"/>
Slope Type After Pivot: <input checked="" type="radio"/> Normal <input type="radio"/> Special	Slope Type After Pivot: <input checked="" type="radio"/> Normal <input type="radio"/> Special
Standard Slope Percent: 0.000	Standard Slope Percent: 0.000
Minimum Slope Percent: 0.000	Minimum Slope Percent: 0.000

Now for the final subgrade... the aggregate bed beneath the asphalt. Follow this: if the stone can't "Match Surface" (note this option under *Slope Type*), it will start uphill with the shoulder as it passes beyond the curb. So it must have a *Special Slope Type*, a consistent 2% cross-slope all the way. The "Wrap Height" is the vertical rise at the end of the aggregate base before it wraps back and returns back to the curb. Select the

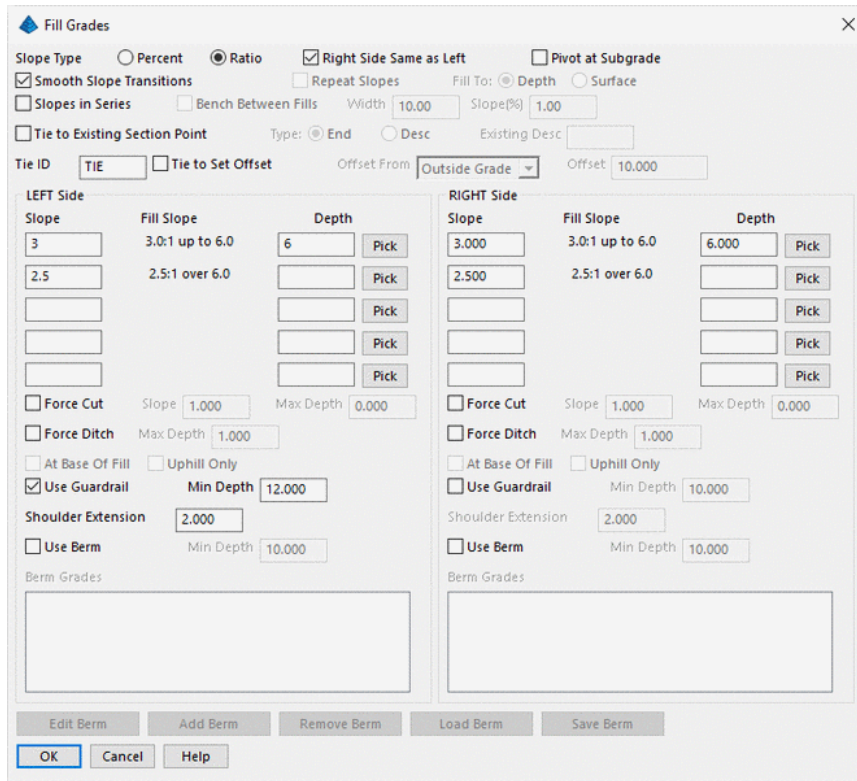
Subgrade icon again, complete the values as shown and click **OK** when ready:



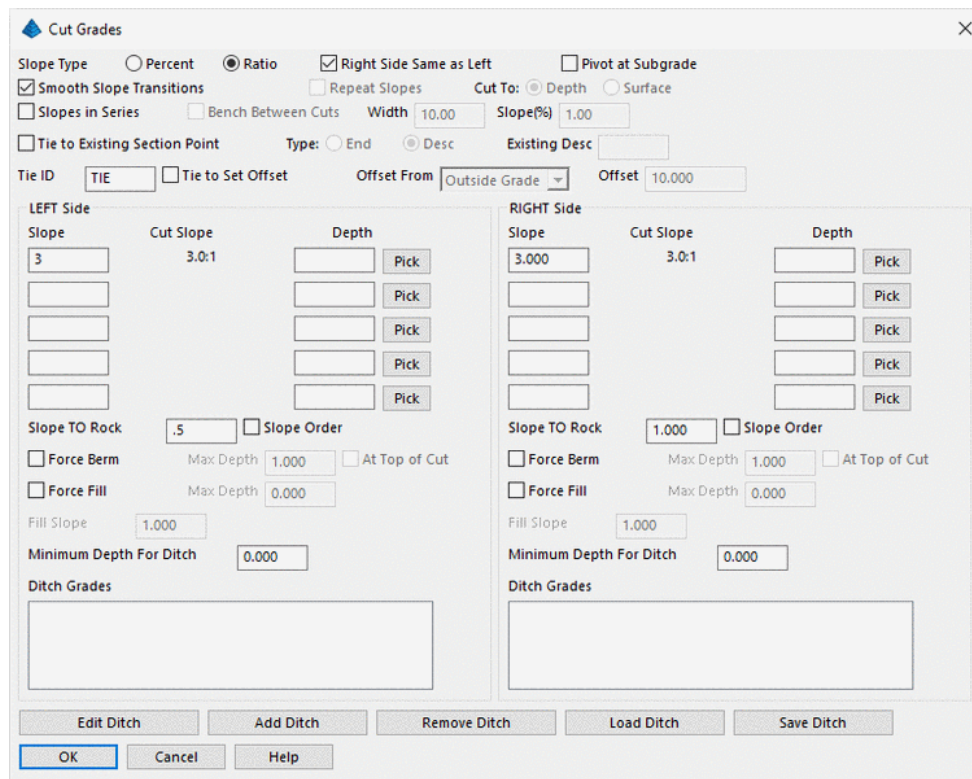
Note the preview screen:



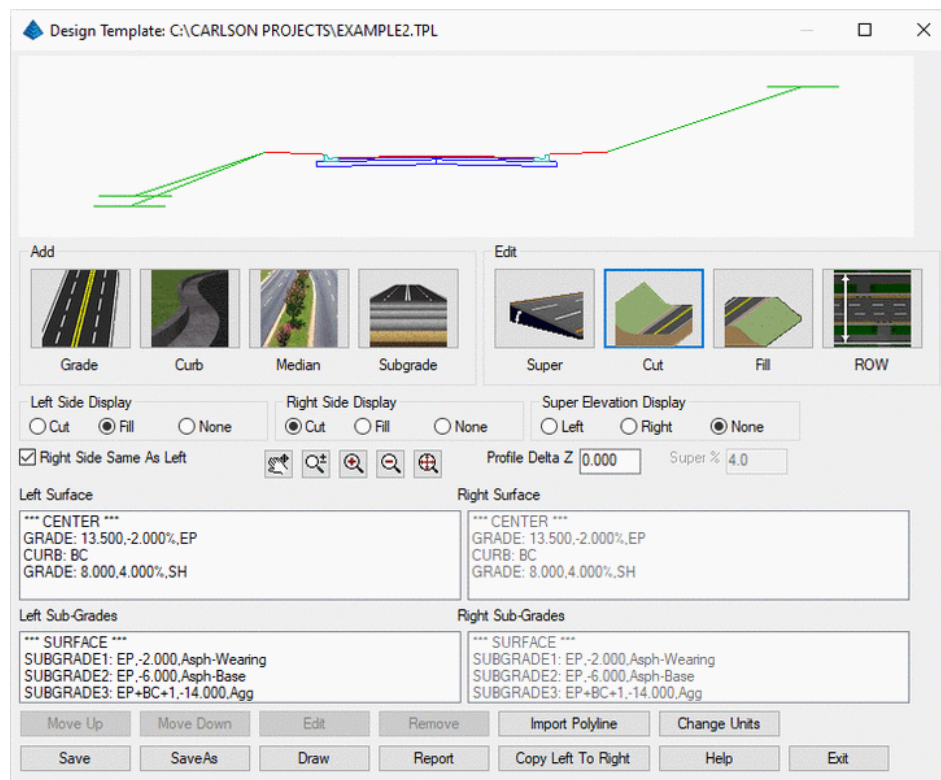
We still need to enter the tie-slope conditions. Let's continue and click the **Fill** as it is easy in our example:



Just four entries total: 3 (for 3:1), 6 (up to 6'), 2.5 (for 2½:1 over 6'), then enable the **Use Guardrail** toggle as shown. Click **OK**. Next, click the **Cut** button:

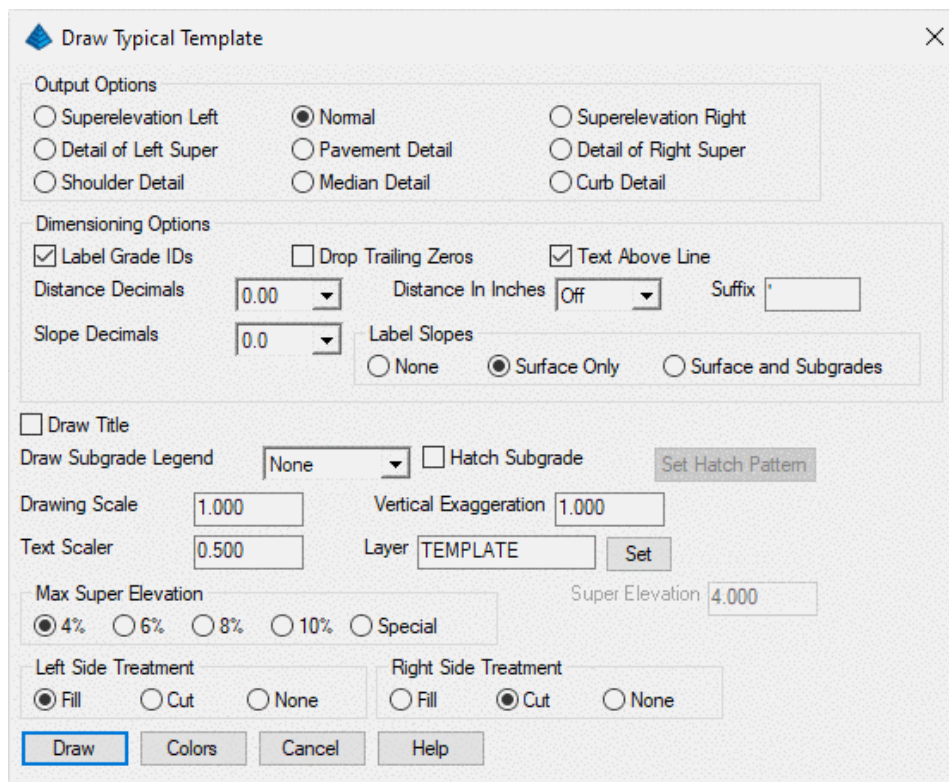


This too, is easy (in terms of total entries). Just two entries do it: 3 (for 3:1 normal cut) and in the **Slope TO Rock** area, specify 0.5 (for 0.5:1 cut when in rock). Click **OK**.



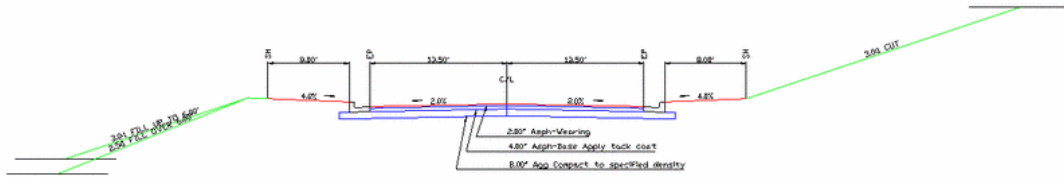
The template is complete. Click **Save** and then click **Exit** to close the dialogue box.

- Now let's prove we have a good template by issuing the Roads – Draw Typical Template command (this is the same as the *Draw* button as found in the previously discussed **Design Template** command). The file extension for templates will be *.tpl. Select the template file as named above and click **Open** to display the following dialog box:

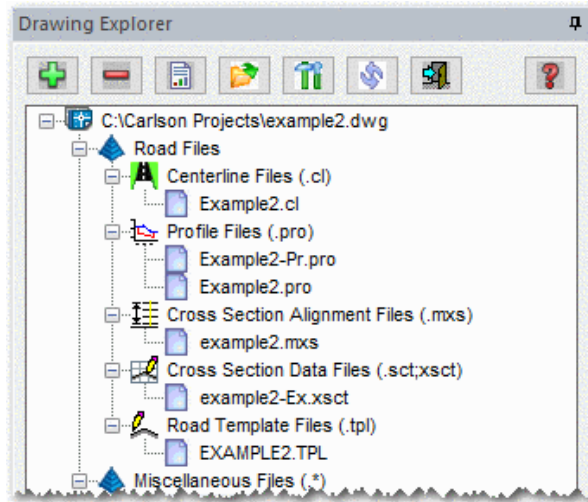


We have doubled the text scaler to **0.5** for better appearance in this tutorial. Click on **Draw** and pick a starting

position point. Here is the look of the plotted template:

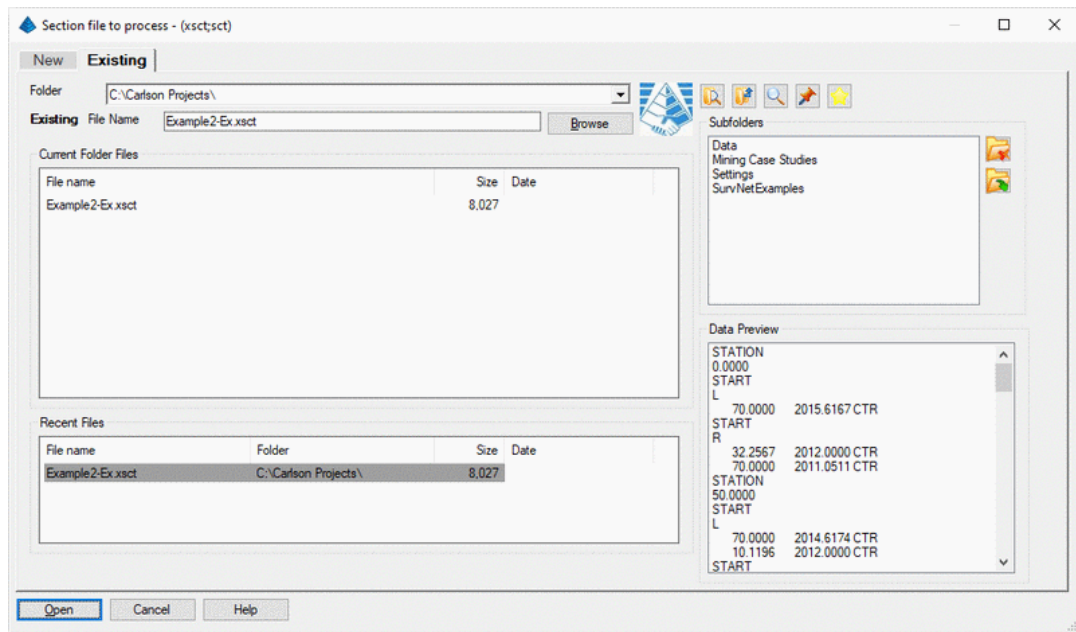


- As more files are created, edited, loaded and reviewed within a work session, the *.ini file associated with the active drawing takes note. You can review your active files as you work, or days later, because they save to the *.ini file that shares the same name as the drawing file. To see the files associated with this tutorial drawing file, issue the File – Drawing Explorer command to display a docked dialog box similar to that shown below:

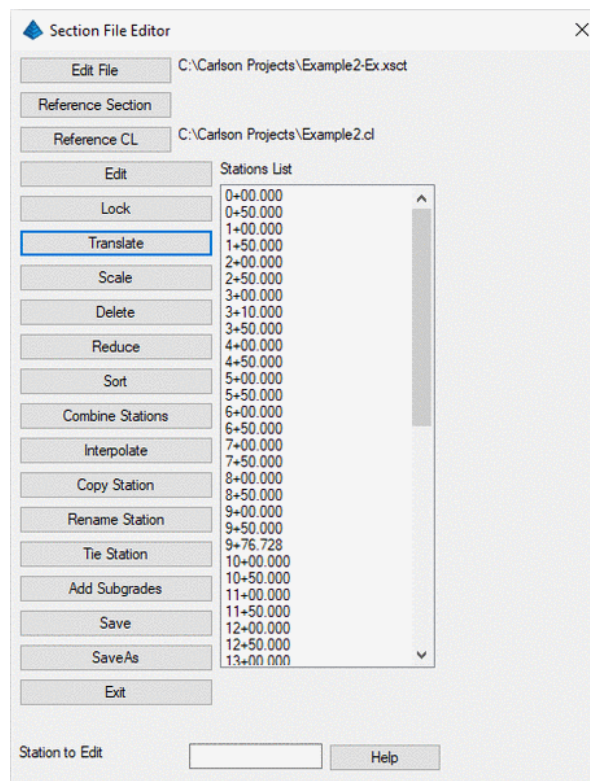


To edit any given file, simply double-click it or right-click it to show a series of sub-options. Click the **Exit** (Doorway) button when complete.

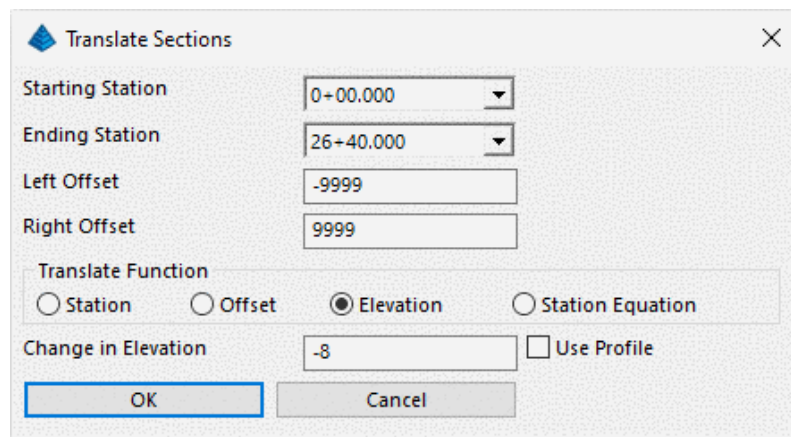
- Although Carlson has dedicated technology for using soil-boring logs to determine subsurface strata (*e.g.* topsoil thickness, depth to rock, *etc.*), we'll develop an approximate rock surface to demonstrate rock-section cuts. Issue the Sections – Section File Editor command, select the section file you created earlier and click **Open** as illustrated below:



Section File Editor has many uses. One of them is to translate or lower the elevations of a file and re-save. If we lower the elevations of our ground sections 8 feet, we can call that the rock line. Rock lines react with templates and profiles to create rock cuts and rock quantities when we process the road corridor template through the project. The next dialog that appears is shown below:

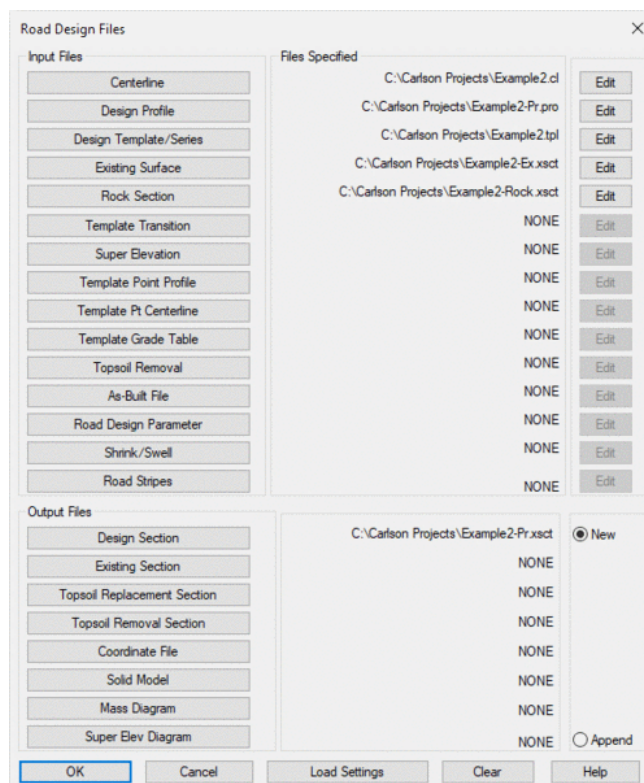


Click the **Translate** button to display the dialog box shown below. Make sure the dialog looks that same as shown below, and click **OK**:



Control returns to the *Section File Editor* dialog box. Click **SaveAs**, and enter a different name, such as *Example2-Rock*, and save the file. Then click **Exit**. *Section File Editor* can do much more through its **Edit** option (although we will not be making any other edits to either section file). However, when using *Edit*, you would first highlight one station, then click *Edit* to review and/or revise it.

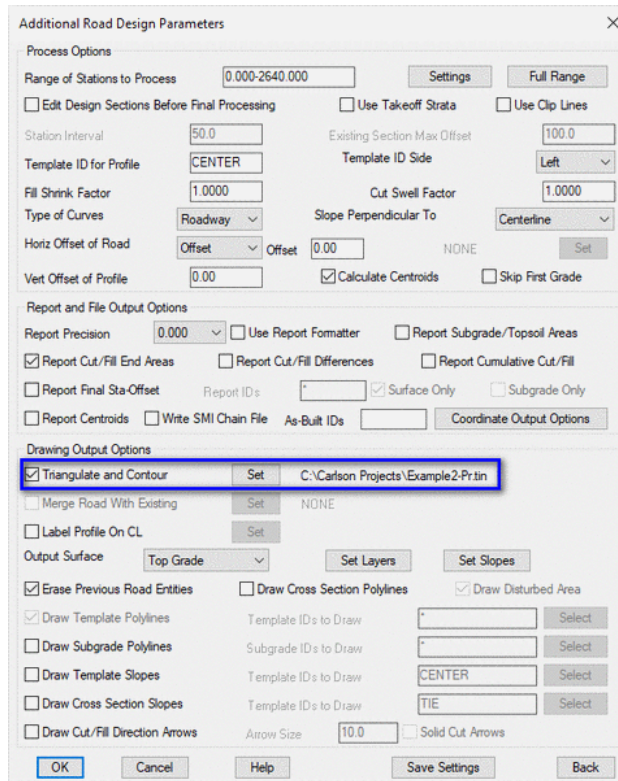
14. It's time to pull everything together. Issue the Roads – Process Road Design to display the dialog box shown below:



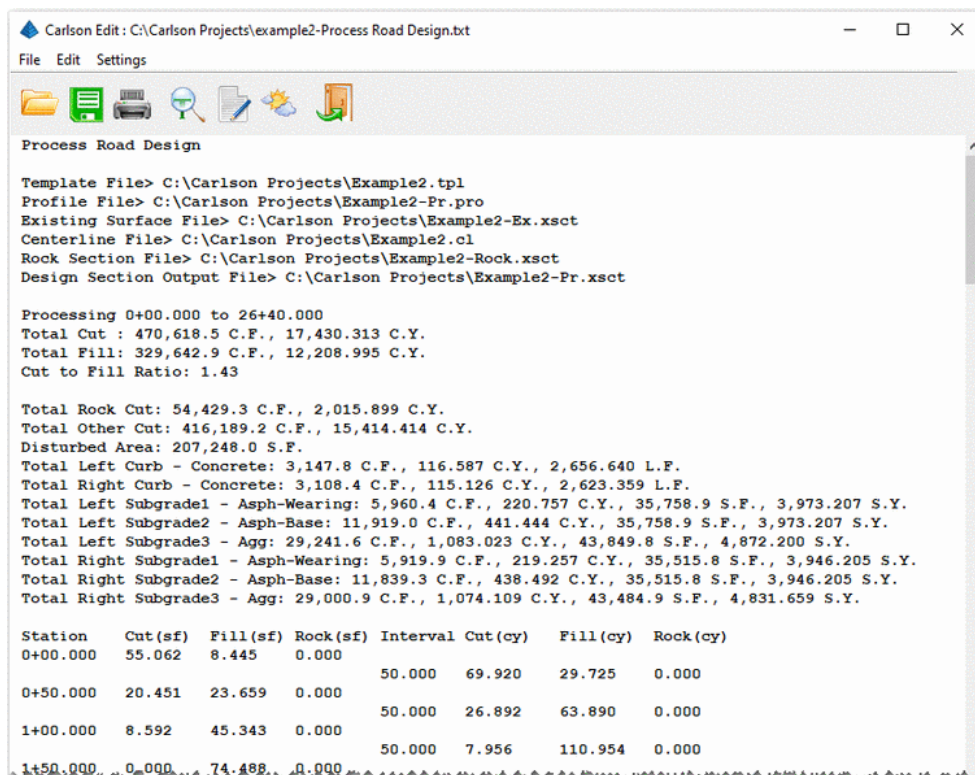
Fill out the dialog as shown above, making sure to:

- (a) correctly specify your **Design Profile**, and,
- (b) specify a *New* **Design Section** file as found in the Output Files section of the dialog box.

Click **OK** to initiate the design process. On the next dialog, be sure to enable the **Triangulate & Contour** and specify an output TIN file and contours of the completed road as illustrated below:



Now click **OK**. Here is a partial view of the final report, with itemized quantities:



Click the **Exit** (Doorway) button when finished reviewing the report. Depending on your *Triangulate & Contour* settings, you may be prompted:

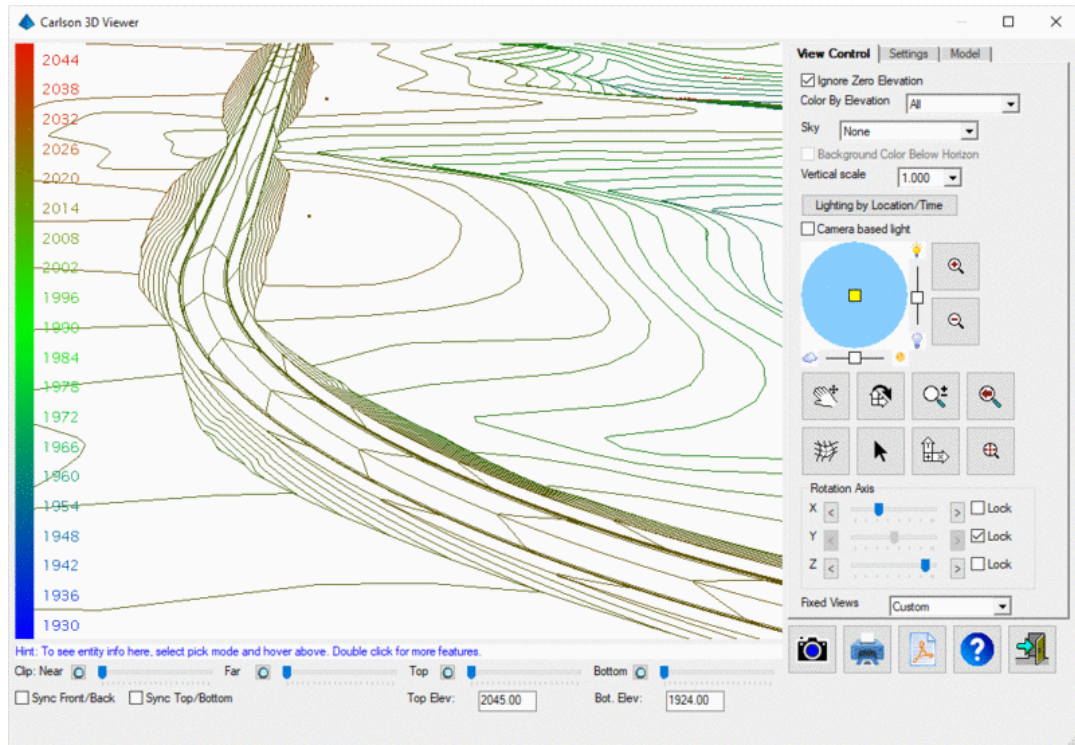
Retain trimmed polyline segments [Yes/<No>]? press Enter

15. Let's inspect the result visually through the use of the View – 3D View – Drawing Viewer command. When prompted:

Select entities for the scene.

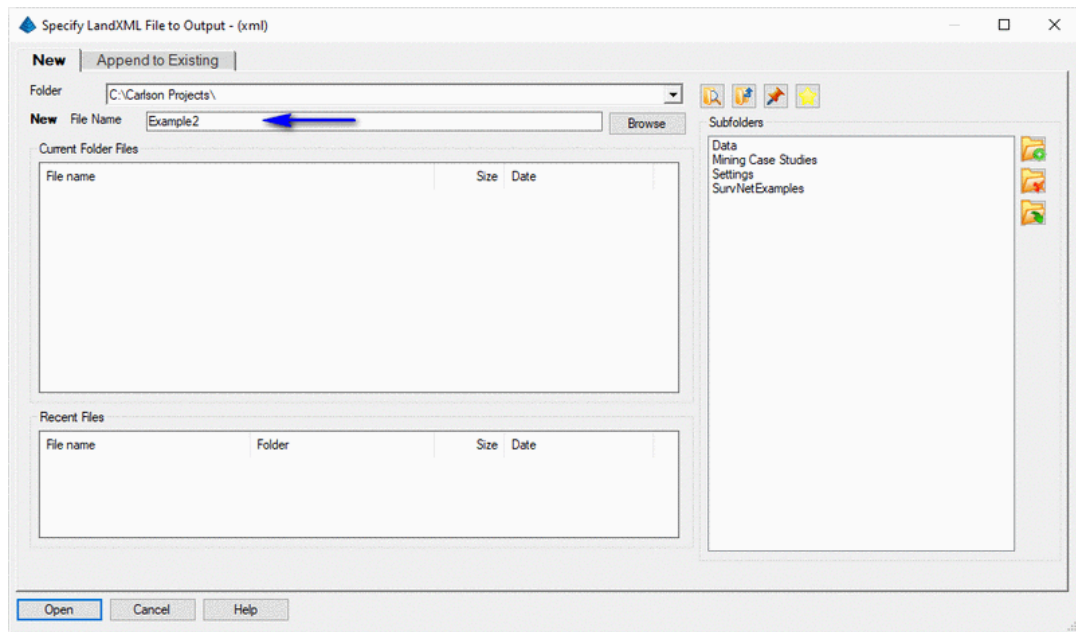
[FILter]/<Select entities>: type ALL and press Enter twice

Here is a resulting graphic:

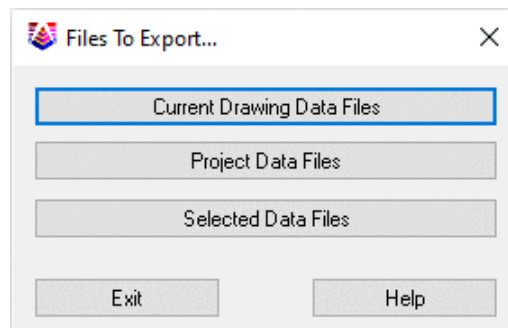


Click the **Exit** (Doorway) button when finished.

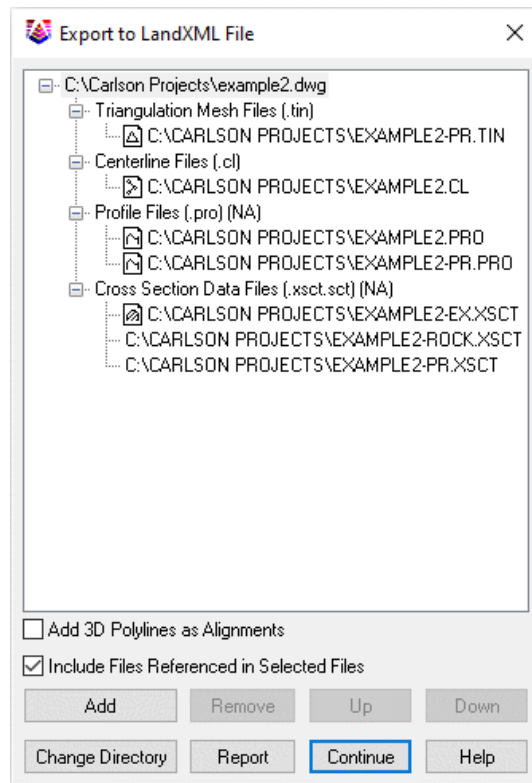
16. Our last task will be to package up the various files used in this example to a CAD-neutral, vendor-neutral *LandXML* format that is often used when needing to convey (or archive) the intent of the design to another party (*i.e.* stake-out crews, machine-control paving crews, *etc.*). Issue the File – Export – LandXML File command to create a LandXML file as shown below and click **Open** when ready:



We'll automate the process of the data files that should be packaged into the LandXML file through the use of the **Current Drawing Data Files** option as highlighted below:



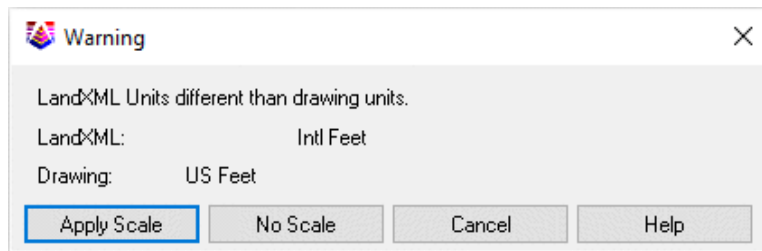
A summary of the files associated with the drawing should look similar to that shown below:



Click on **Continue** once you are satisfied with the list to display the dialog box below:



Set the values as shown above and click **Export**. In the event you are notified about a potential *Units* conflict (for this example), initiate the desired response (as offered below):



The LandXML file is written and can be validated through the LandXML Validator.

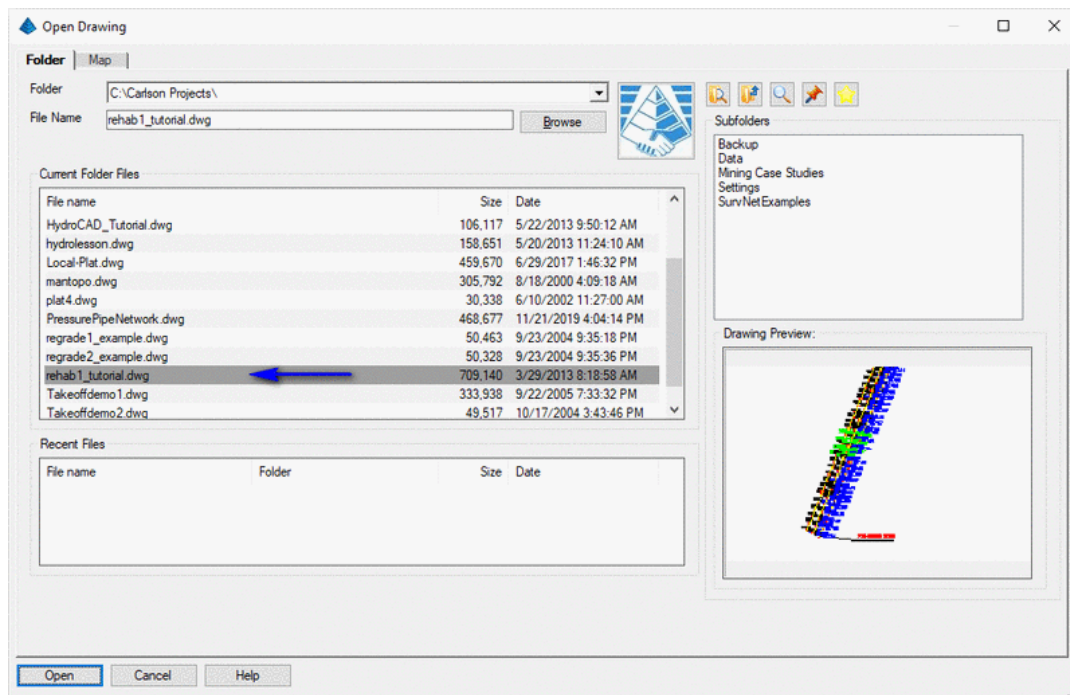
This completes the tutorial: Basic Road Design with Volumes.

Road Rehabilitation

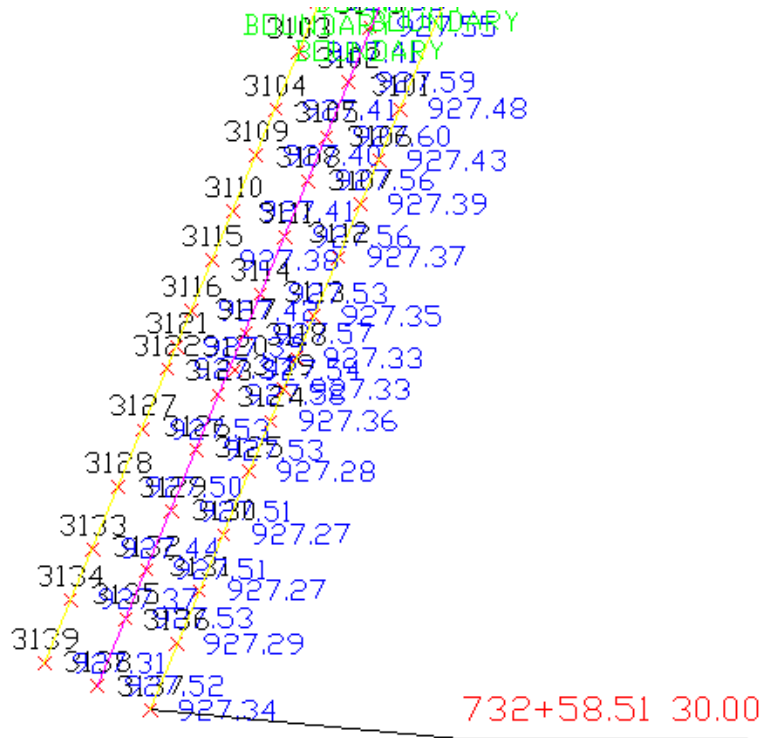
This lesson designs a road rehabilitation project for two lanes of an Interstate highway. The top surface of the existing road has already been milled and the top of the milled surface has been surveyed. The two lanes are crowned in the center with a typical cross slope of approximately 1%. The design is to add 8" of concrete with a minimum cross slope of 1.56%. Let's get started.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

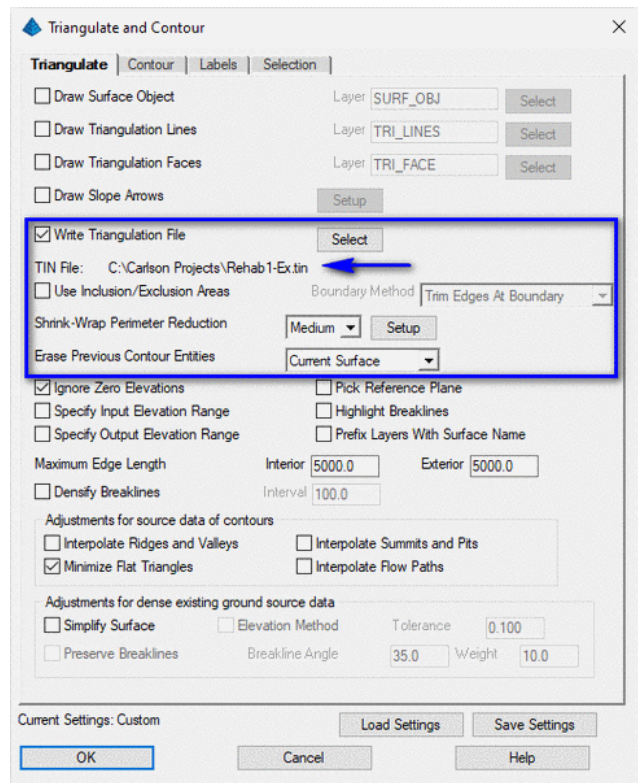
Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **rehab1_tutorial.dwg** file.



Use the File – **Save As** to save a copy named **rehab2_tutorial.dwg**. Completing this tutorial will alter the drawing file and by renaming the file from the start, you'll keep the original file intact (allowing you to run through the tutorial a second time if desired). This is also a good practice to keep when working on drawings from 3rd parties. The drawing should resemble that shown below:



- Ensure you are in the Civil program by issuing the Settings – Carlson Menu – **Civil Menu** command.
- Our immediate goal will be to create an existing ground surface model (TIN). Issue the Surface – Triangulate & Contour and set the values shown in the dialog box below and click **OK** when ready:



When prompted:

Select the points and breaklines to Triangulate.
[FILTER]/<Select entities>: type ALL and press Enter

[FILter]/<Select entities>: press Enter

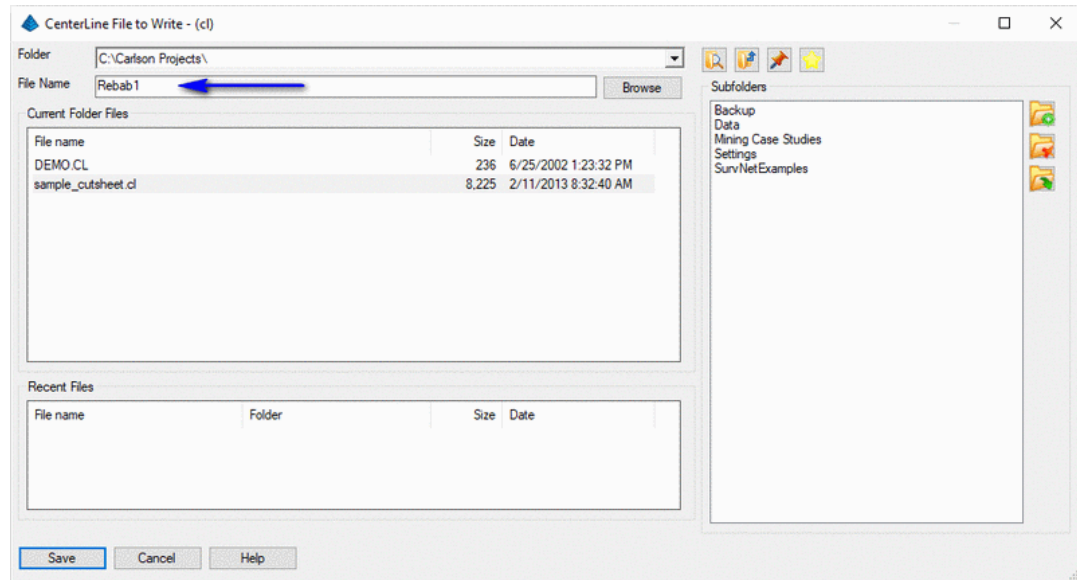
A surface model (TIN) file representing existing conditions is created.

3. Our next task will be to create a horizontal centerline from polyline data found in the drawing. Before we complete this task, we'll want to freeze the *BREAKLINE* layer to expose the polylines on the **PROPOSED PAVEMENT** layer. Issue the View – Freeze Layer by Pick and when prompted:

Pick entities on layers to be frozen: pick one of the pavement edge polylines

Pick entity on layer to be frozen (U-Undo,Enter to end): press Enter

Issue the Centerline – Polyline To Centerline File to display the dialog box shown below:



Set the desired file name as shown above and click **Save** when ready. When prompted:

Polyline should have been drawn in the direction of increasing stations.

Select polyline that represents centerline: pick the centerline polyline

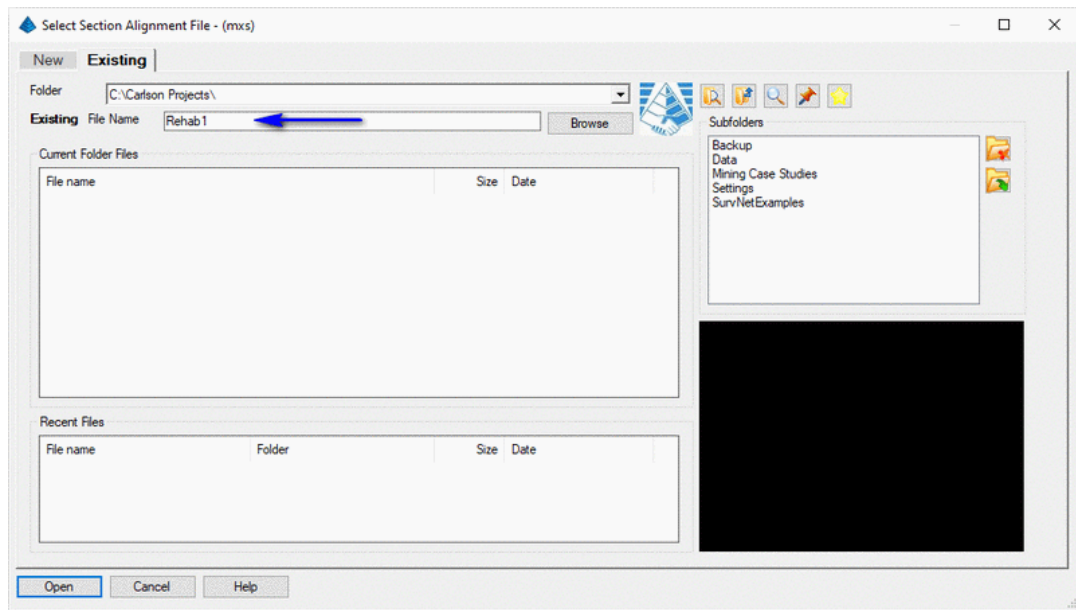
Centerline station [Reverse/Ending/<Beginning: 0+00>]: type 73258.51 and press Enter

Press ENTER to continue. press Enter

A centerline alignment is created.

NOTE: Double-clicking on the polyline on the screen will open the Centerline Editor command.

4. Now that the horizontal path of the road has been established, the next step will be to generate cross-section information of existing conditions. Issue the Sections – Section Alignment Editor to display a dialog box similar to that shown below:



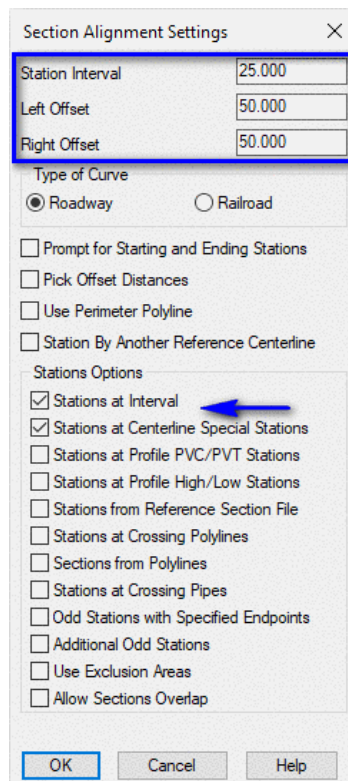
Set the file name as indicated above and click **Open** when ready. When prompted:

Polyline should have been drawn in the direction of increasing stations.

CL File/ <Select polyline that represents centerline>: indicate the Centerline option

Specify a CL File (dialog): select the Centerline created in the previous step and click **Open**

Section Alignment Settings (dialog): set the values as shown below and click **OK** when ready



- The program draws temporary lines in the drawing to show the positions of the sections and displays a summary of the section alignment cross-section stations. Click the **Save** button followed by the **Exit** button.
- To create the section file for the *Existing* (base) surface, issue the Sections – Create Sections from... –

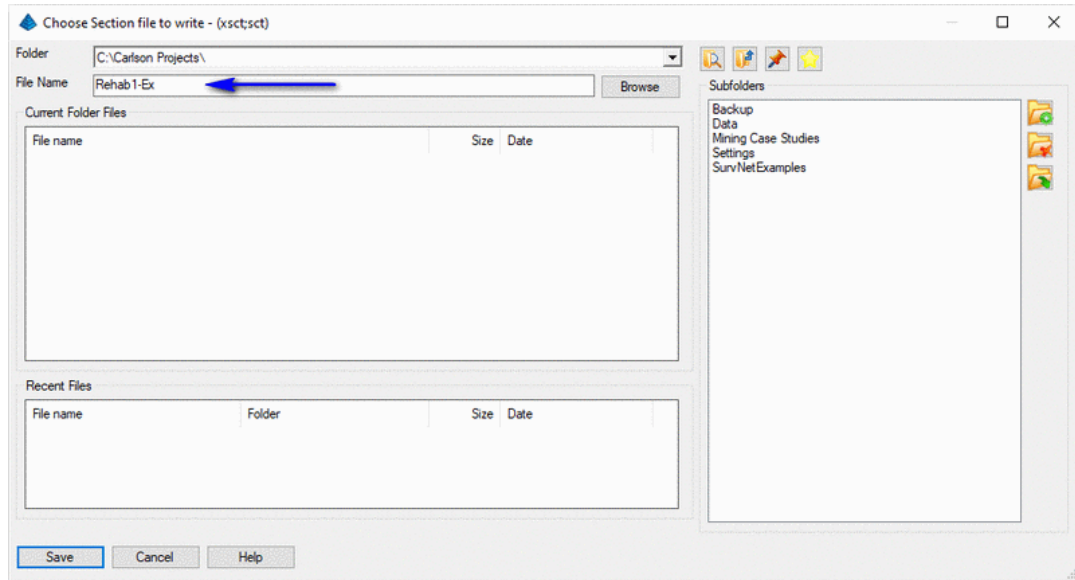
Sections from Grid or Triangulation Surface command. When prompted:

Choose Grid or Triangulation file to Process (dialog): specify the TIN file created earlier and click **Open** when ready

Section Options (dialog): set the option to **Prompt** and click **OK** when ready (with *Prompt* specified, you'll be prompted if the Section should change if there is an underlying change to the source TIN file)

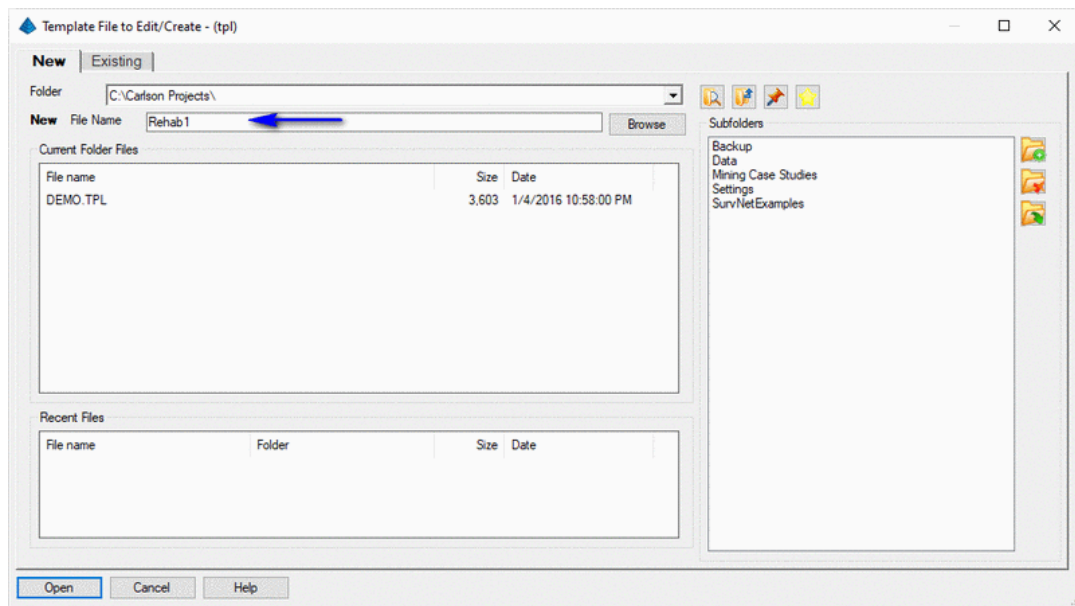
Section Alignment File to Process (dialog): specify the section alignment (*.MXS) created earlier and click **Open** when ready

Choose Section File to Write (dialog): specify a file name as illustrated below and click **Save** when ready



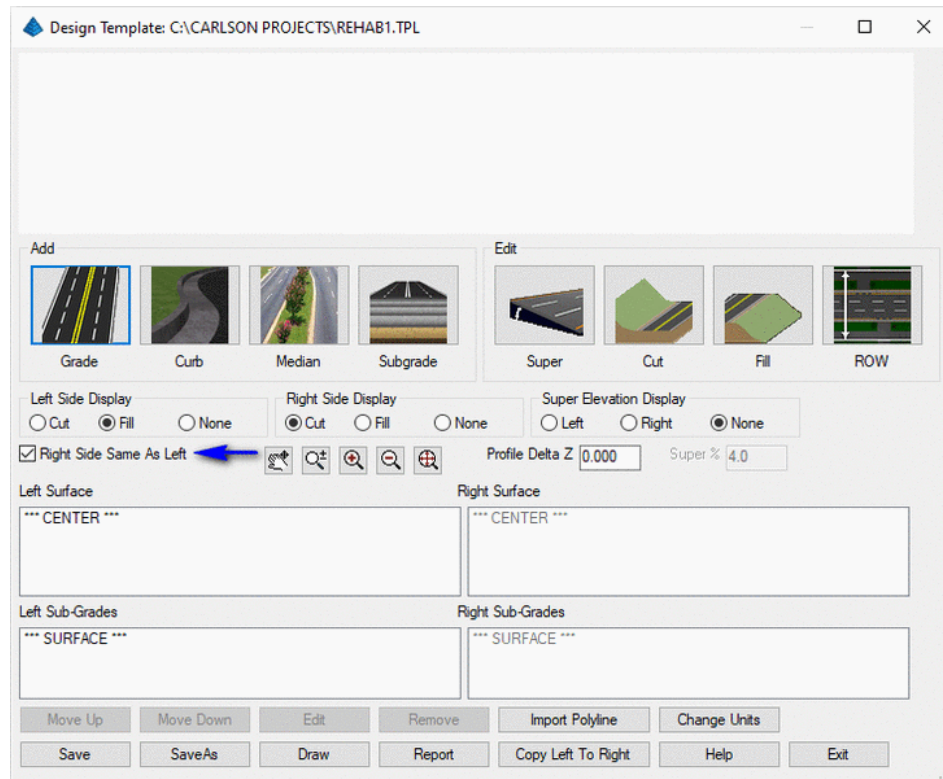
The section file is written.

6. Now that we've generated existing cross-section data, our next task will be to create a design template "overlay" for the road surface we wish to construct. Issue the Roads – Design Template and click on the **New** tab to create the file name as shown below. Click on **Open** when ready:

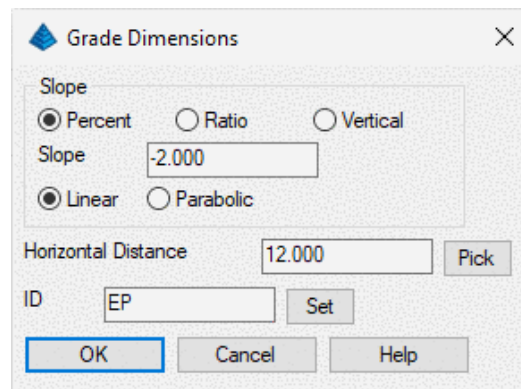


A large dialog box appears as shown below. In it, you enter segments of the template, which typically work

outwards from the middle as you add more lanes, curbs and shoulders. We will enter a symmetrical template, with 12' pavement sections either side of centerline with an 8" subgrade.



For the lanes, click the **Grades** icon. This leads to a child dialog as shown next:



Fill out as shown. It's important to note that a downhill pavement from a crown in the middle is entered as a negative slope. That is, it is -2% heading from centerline outward, regardless of which side of centerline we are speaking of. Slope is independent of the profile grade point. It is also important to enter an ID whenever requested as these can be referenced later for advanced placement control. Click **OK** and note that the lanes show up in the preview window at the top.

Click the **Subgrade** button and indicate the following desired values for the subgrade surface:

The image shows a software dialog box titled "Subgrade Dimensions". It contains several sections for configuring subgrade parameters:

- Subgrade Type:** Radio buttons for "Roadway" (selected), "Cut Slopes", and "Fill Slopes".
- Distance Direction:** Radio buttons for "Out" (selected) and "In".
- Slope Type:** Radio buttons for "Match Surface" (selected) and "Special".
- Slope (%):** Text input field with "0.000".
- Tie Undershoot Vertically at Hinge:** Check box (unchecked).
- Intersect Surface:** Dropdown menu with "Straight Up" selected.
- Wrap Height (in):** Text input field with "0.000".
- Tie Slope (%):** Text input field with "0.000".
- Horizontal Offset:** Text input field with "0".
- Vertical Offset (in):** Text input field with "-8.000".
- Units:** Radio buttons for "Feet" and "Inches" (selected).
- Match Existing Surface:** Check box (unchecked).
- Distance:** Text input field with "EP" and a "Select" button.
- Material:** Text input field with "Concrete" and a "Set" button.
- Coatings/Notes:** Text input field.
- Super Elevation Settings:** A sub-dialog box with settings for "Low Side" and "High Side", including "Pivot Offset", "Max Slope After Pivot (%)", "Slope Type After Pivot" (Normal/Selected, Special), "Standard Slope Percent" (0.000), and "Minimum Slope Percent" (0.000).

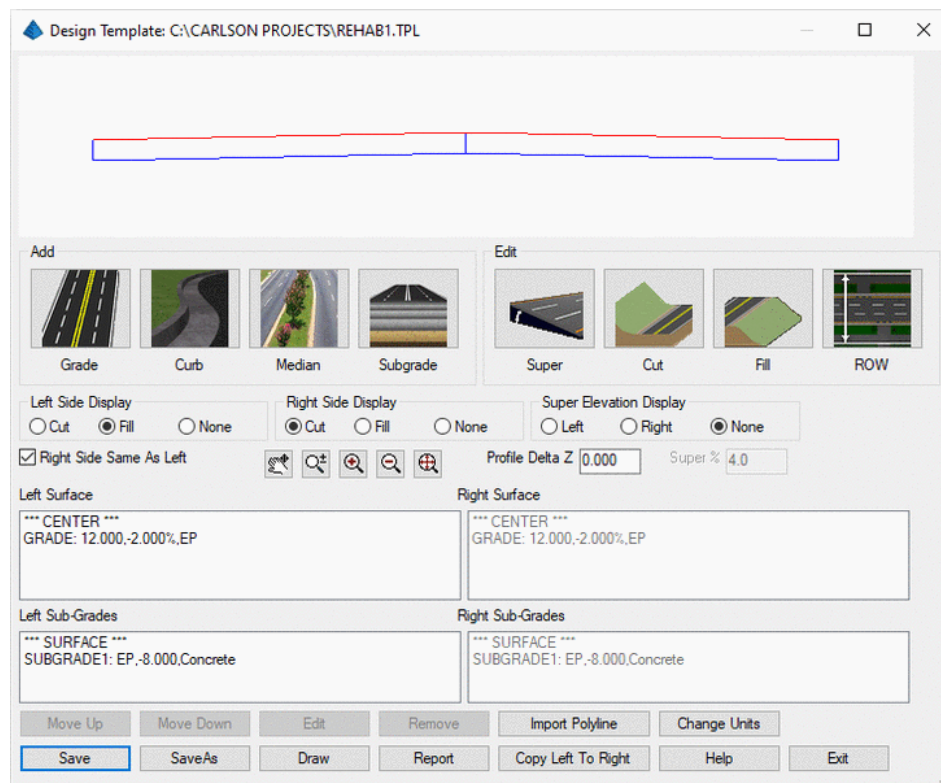
At the bottom are "OK", "Cancel", and "Help" buttons.

NOTE:

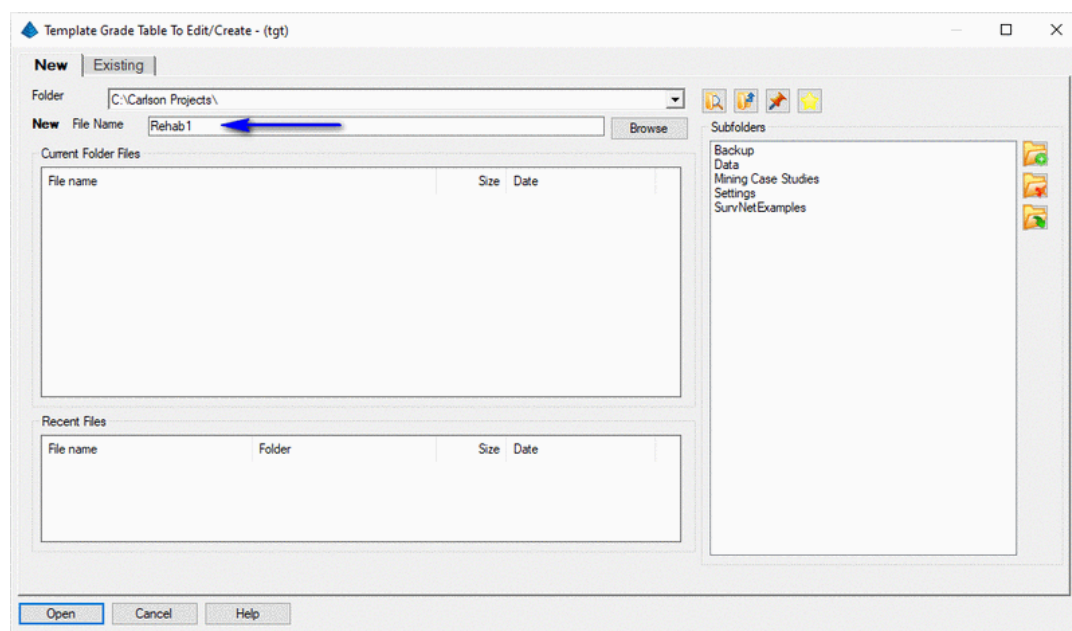
- **Straight Up** for the Intersect Surface option instructs the subgrade surface to intersect vertically to the grade surface above.
- **0** for the horizontal offset distance instructs the subgrade surface to start directly under the centerline of the road.
- **8 inches** for the Vertical Offset instructs the subgrade to be 8 inches below the grade surface above and will generate the minimum 8" pavement thickness specified earlier in this lesson.
- **EP** for the Distance instructs the subgrade width to be the same width as the EP (edge-of-pavement) width specified in the *Grade Dimensions*.

NOTE: By using the parametric/variable ID points (*e.g.* the "EP" code *vs* the 12' Horizontal Distance, the subgrade width will vary as needed to accommodate the pavement grade width!

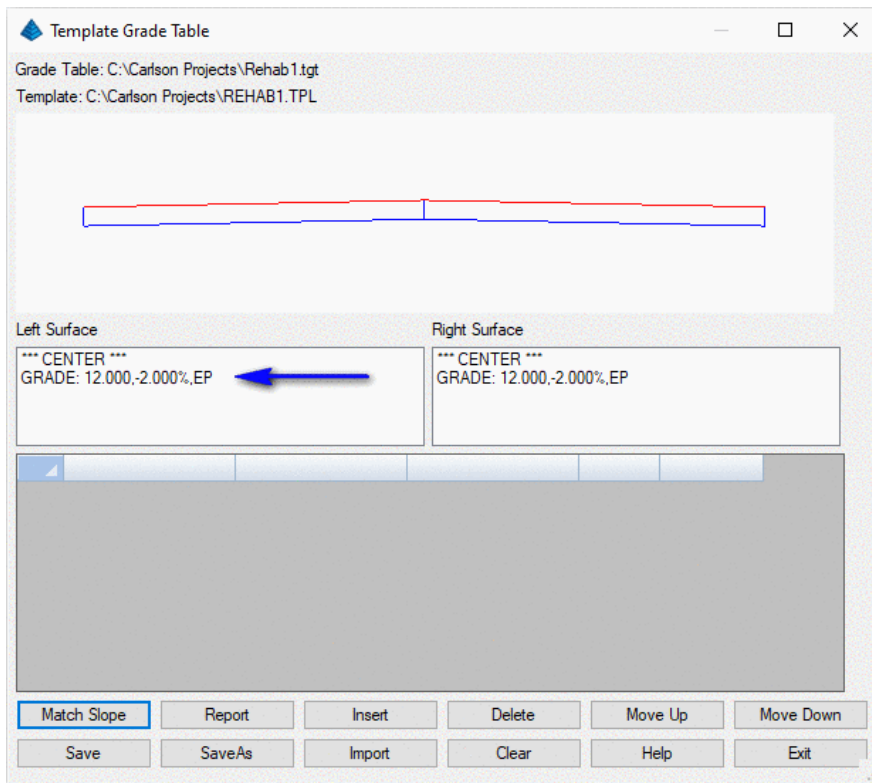
Click OK when ready to dismiss the Grade Dimensions dialog box. The proposed road template should resemble the following:



- That's all that needs to be defined in the template. Click the **Save** button and then click **Exit**.
7. The next step is to design the proposed cross-slopes to follow the existing section grades as much as possible. Issue the Roads menu – Template Grade Table to display a dialog box similar to that shown below:

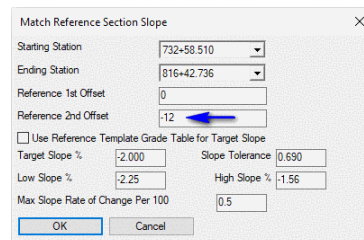


Set the file name as shown above and click when ready. A dialog box similar to that shown below appears:

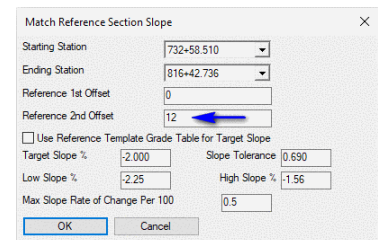


- Highlight the **GRADE** surface from the Left Surface list, then
- click the **Match Slope** button, then
- select the **section** file created earlier as the *Section File to Process* and click **Open** when ready. A dialog box similar to that shown below appears:

Left Side Settings



Right Side Settings



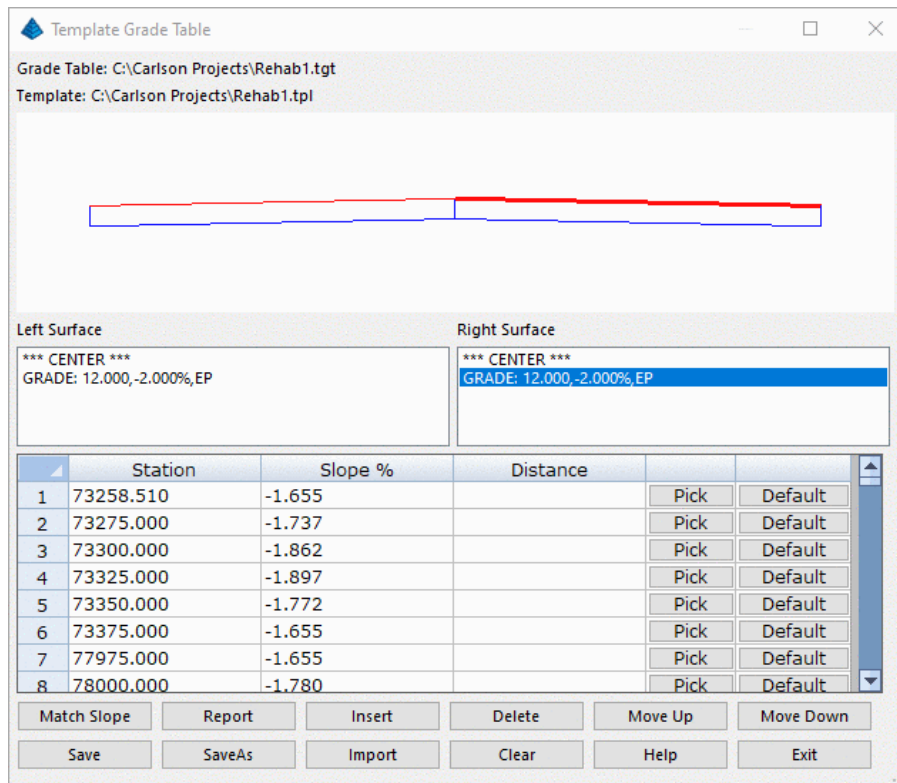
Match Reference Section Slope

Set the appropriate values for the Left/Right side as shown above and click **OK** when ready.

NOTE:

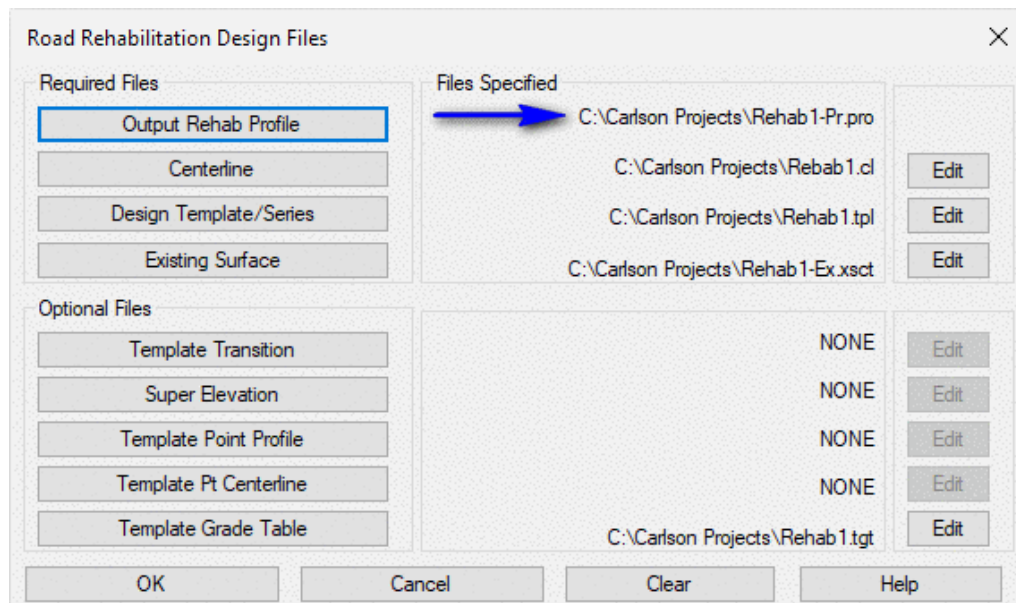
- The **Reference 1st/2nd Offset** values are used to sample the existing sections at the specified offsets to get the existing cross slope.
- The **Lowest/Highest Slope %** values restrict the design cross-slopes to be within the range specified.
- The **Max Slope Rate of Change Per 100** restricts how much the design cross-slope can change between stations. For example, with the maximum set to 0.5 and a cross-slope of -2.0% at station 1+00, the cross-slope at station 2+00 must be between -2.5% and -1.5%.

After picking **OK** on the dialog box above, highlight the **GRADE** surface from the Right Surface list and repeat the steps above. The resulting *Template Grade Table* should resemble that shown below:

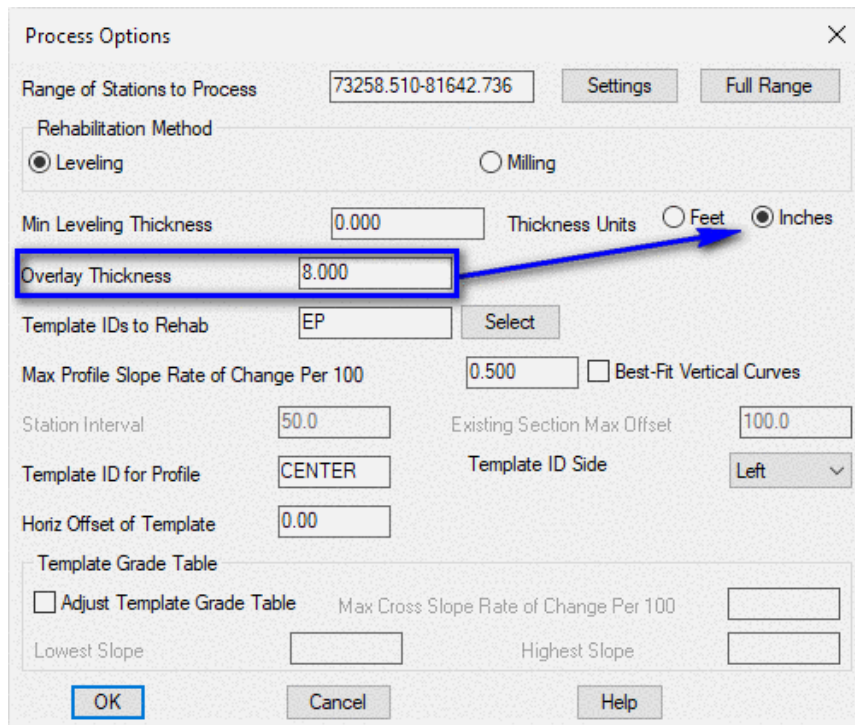


Click **Save** and then click **Exit**.

- The next step is to define the design profile that minimizes quantities while satisfying the design parameters. Issue the Roads – Road Rehabilitation Profile command to display a dialog box similar to that shown below:



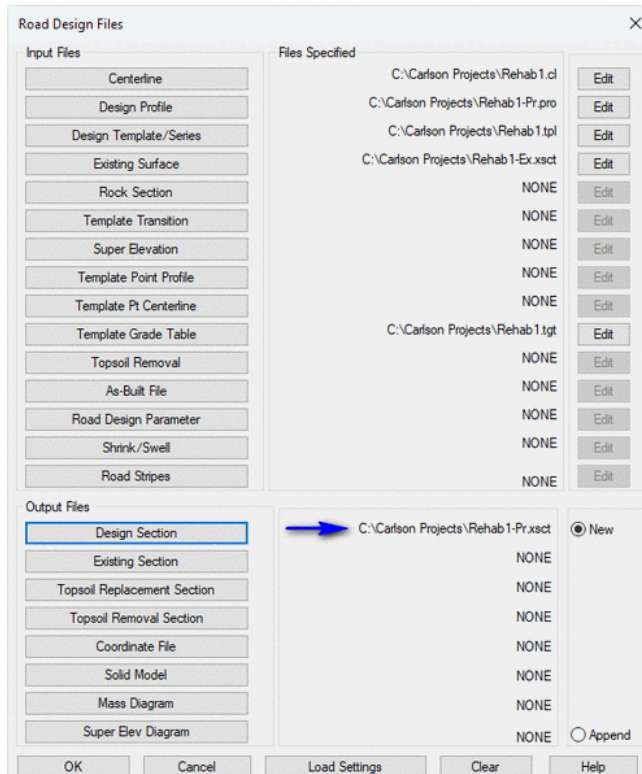
Set the values as shown above and click **OK** when ready to display a dialog box similar to that shown below and click **OK** when ready:



NOTE: The **Max Profile Slope Rate of Change Per 100** option is essentially identical to the *Max Slope Rate of Change Per 100* value found in *Template Grade Table* except this one applies to the profile slope instead of the cross-slope.

A profile for the data is generated and any adjustments to the profile can be made through the Profiles – Design Road Profile – Road Profile Editor command.

9. The road design is now ready to be computed. Issue the Roads menu – Process Road Design command to display a dialog box similar to that shown below. Make sure the **Centerline**, **Profile**, **Design Template**, **Existing Surface** and **Template Grade Table** options are each utilizing the files created earlier. Click on the **Design Section** button to create a new file as illustrated below:



Click **OK** when ready to continue. On the **Additional Road Design Parameters** dialog box, accept the default values as illustrated below and click **OK** to initiate the processing of the design:

The program reports the quantities per station and totals in a report similar to that shown below:

```

Carlson Edit : C:\Carlson Projects\rehab1_tutorial-Process Road Design.txt
File Edit Settings

Process Road Design

Template File> C:\Carlson Projects\Rehab1.tpl
Profile File> C:\Carlson Projects\Rehab1-Pr.pro
Existing Surface File> C:\Carlson Projects\Rehab1-Ex.xsct
Centerline File> C:\Carlson Projects\Rehab1.cl
Template Grade Table File> C:\Carlson Projects\Rehab1.tgt
Design Section Output File> C:\Carlson Projects\Rehab1-Pr.xsct

Processing 732+58.510 to 816+42.736
Total Cut : 0.0 C.F., 0.000 C.Y.
Total Fill: 12,504.8 C.F., 463.142 C.Y.
Cut to Fill Ratio: 0.00
Disturbed Area: 201,221.6 S.F.
Total Left Subgrade1 - Concrete: 67,064.5 C.F., 2,483.869 C.Y., 100,603.4 S.F., 11,178.153 S.Y.
Total Right Subgrade1 - Concrete: 67,072.2 C.F., 2,484.157 C.Y., 100,601.3 S.F., 11,177.920 S.Y.

Station    Cut(sf)    Fill(sf)    Interval Cut(cy)    Fill(cy)
732+58.510 0.000      0.080      16.490 0.000      0.399
732+75.000 0.000      1.226      25.000 0.000      2.114
733+00.000 0.000      3.340      25.000 0.000      3.691
733+25.000 0.000      4.632      25.000 0.000      5.087
733+50.000 0.000      6.356      25.000 0.000      5.765
733+75.000 0.000      6.096      25.000 0.000      5.282
734+00.000 0.000      5.313

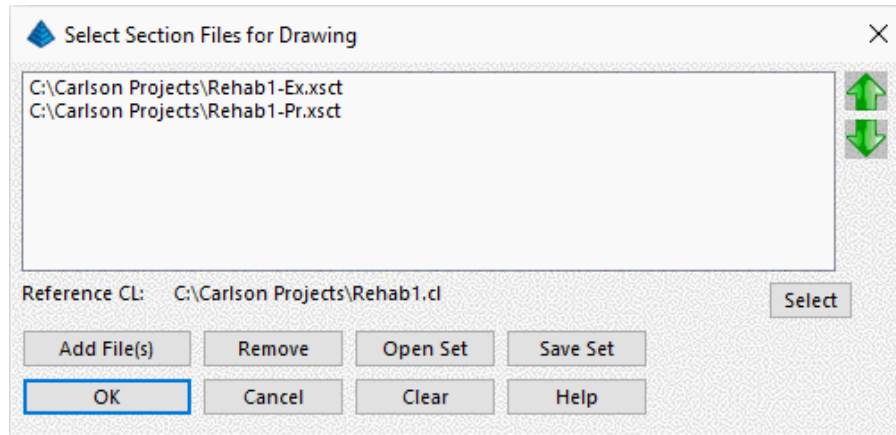
```

Review the report and click the **Exit** (Doorway) button when ready.

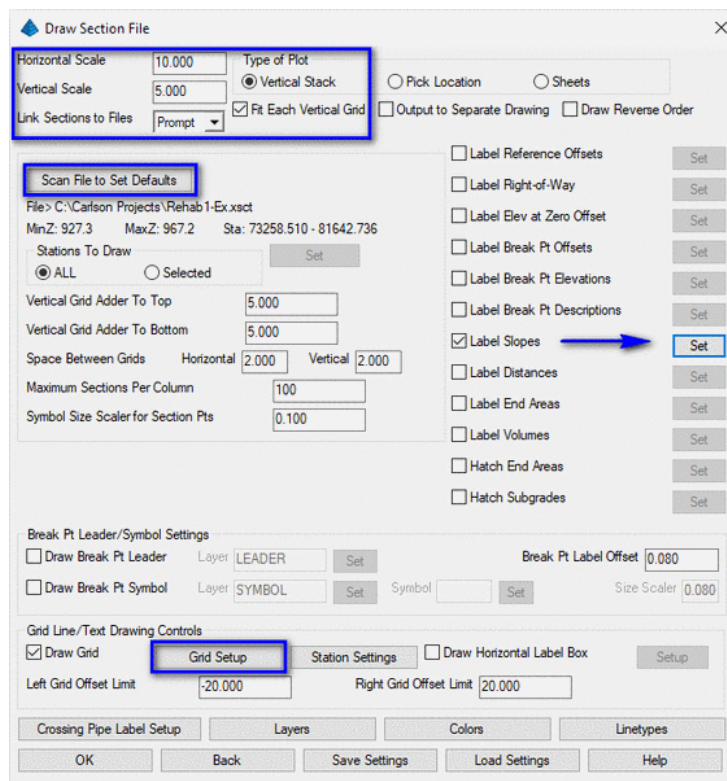
NOTE: Be aware of the following important values:

- The *Total Fill* is the amount of extra leveling.
- The *Subgrade Volumes* are the amount of overlay.

10. Our next task will be to visually verify the design by generating cross-section plots. Issue the Sections – Draw Section File command to display a dialog box similar to that shown below:



Click the **Add File(s)** button to add the two section files we've created earlier and use the *Up/Down* arrow buttons to set them in the order shown. Click **OK** when ready to display a dialog box similar to that shown below:



Set the values as shown above paying particular note to:

- Set **10** for the Horizontal Scale value.
- Set **5** for the Vertical Scale value.
- Click the **Scan Files to Set Defaults** button to "read" the section files specified earlier and retrieve data from them.

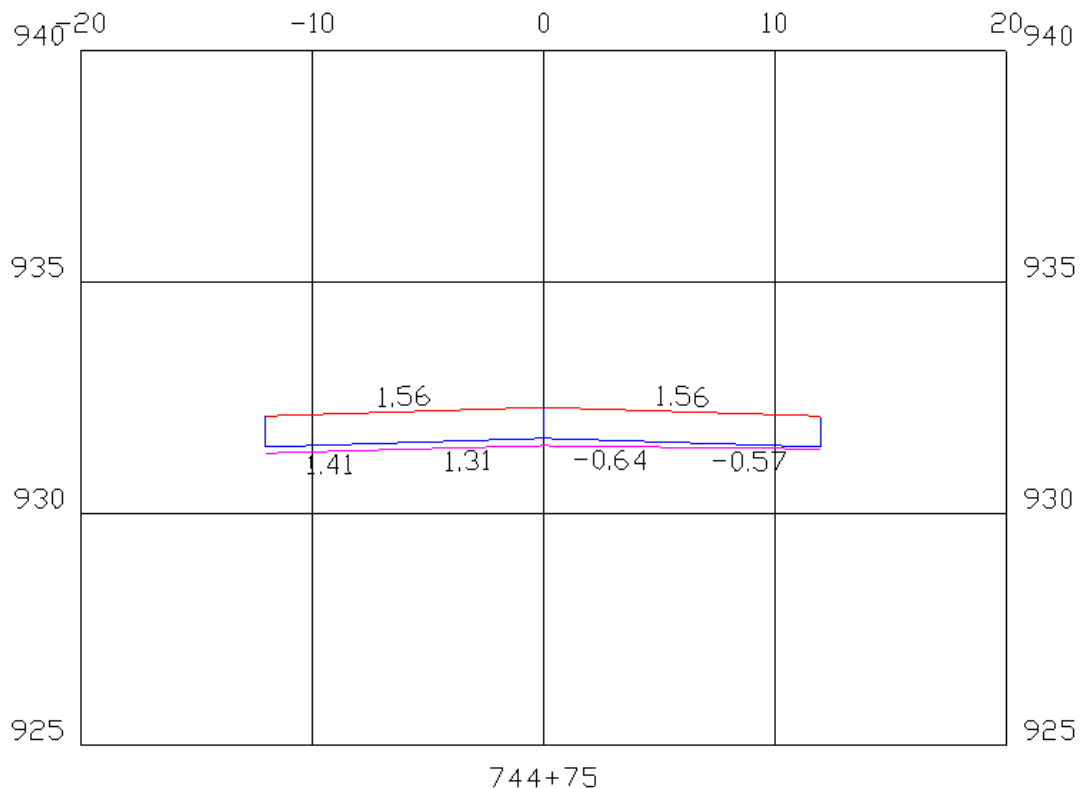
- **Vertical Stack** for the Type of Plot.
- Enable the **Label Slopes** option and then click the **Set** button to ensure:
 - (a) **Label Section 1** is enabled (use its **Set** button to enable the *Label Below* option).
 - (b) **Label Section 2** is enabled (use its **Set** button to enable the *Label Below* option).

NOTE: You can click the **Grid Setup** to set the **Horizontal/Vertical Grid Spacing** values to be the same as that of the *Horizontal/Vertical Scale* values as cited above.

Click the **OK** button and when prompted:

Select Starting Point for Row of Sections <0.00,0.00>: pick a location to the left of the roadway shots

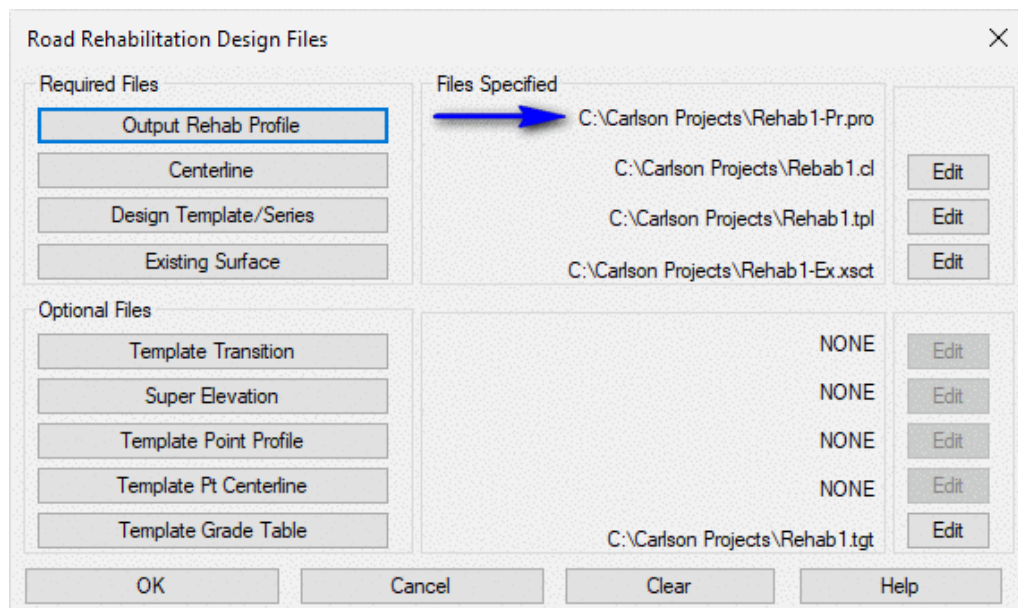
The collection of cross-section stations will be placed into the drawing similar to that shown below:



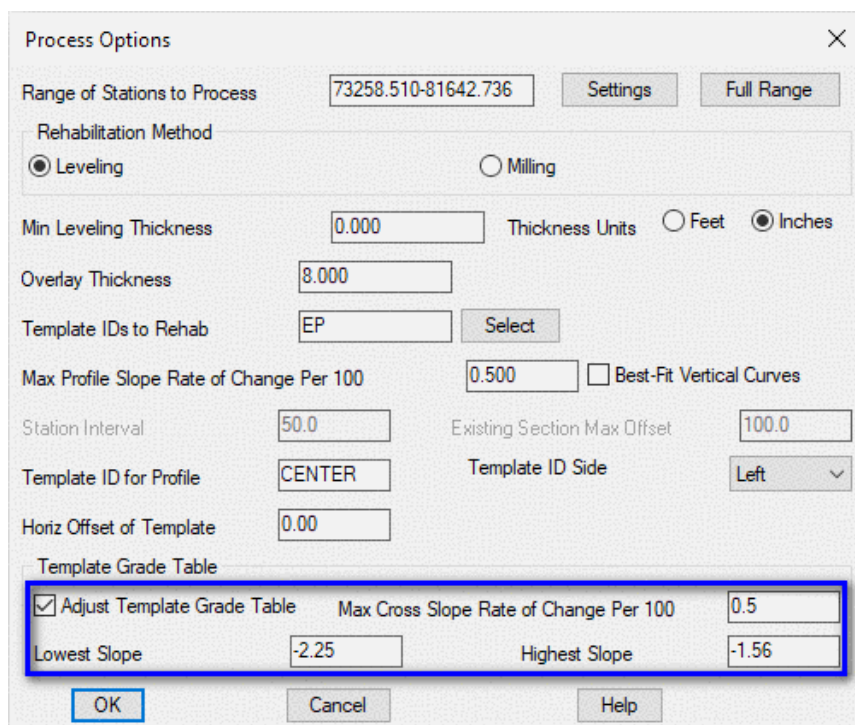
This station shows the existing surface with slopes ranging from 1.41% left to 0.57% right and the design with slopes 1.56% left and right. Since 1.56% was the minimum design cross slope, the design is steeper than existing and there is some leveling needed between the design overlay and the existing.

The profile elevation for this station was set by the right edge since this was the highest point. There is some extra leveling fill needed on the left side. This left side slope can be steeper because our design cross-slope range is 1.56% to 2.25%.

11. Let's continue with additional steps to refine the design! Re-issue the Roads – **Road Rehabilitation Profile** command again. Use all the same files as we did earlier:

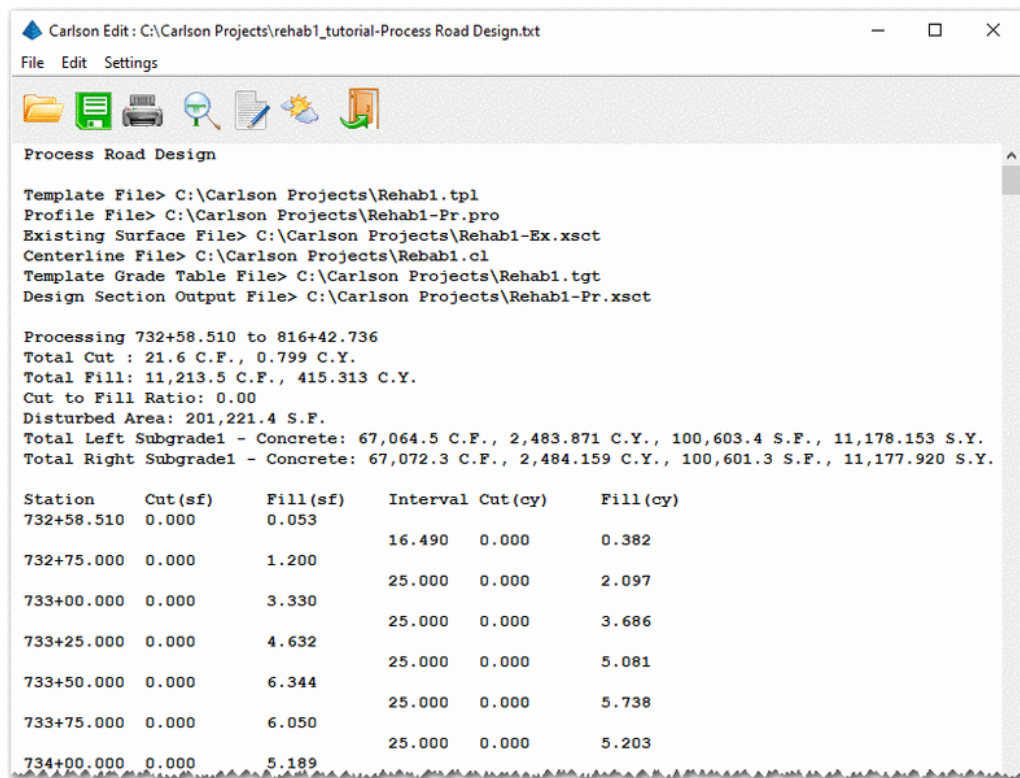


Set the values as shown above and click **OK** when ready to display a dialog box similar to that shown below. Enable the **Adjust Template Grade Table** toggle and set the **Lowest/Highest Slopes** and the **Max Cross Slope Rate** to the same design parameters used during *Template Grade Table* as shown below and click **OK** when ready:



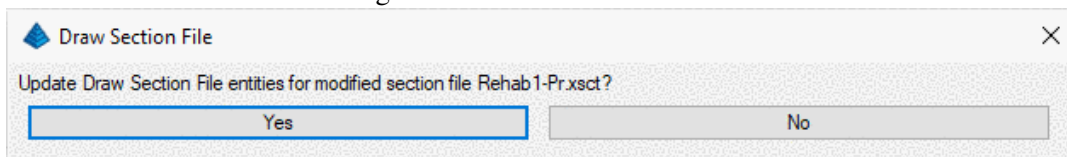
NOTE: The **Adjust Template Grade Table** option checks the left and right grades while creating the rehabilitation profile and adjusts the slope on the side that is not controlling the profile grade.

12. Re-issue the Roads – **Process Road Design** command again with all the same settings from the previous run to display a report similar to that shown below:

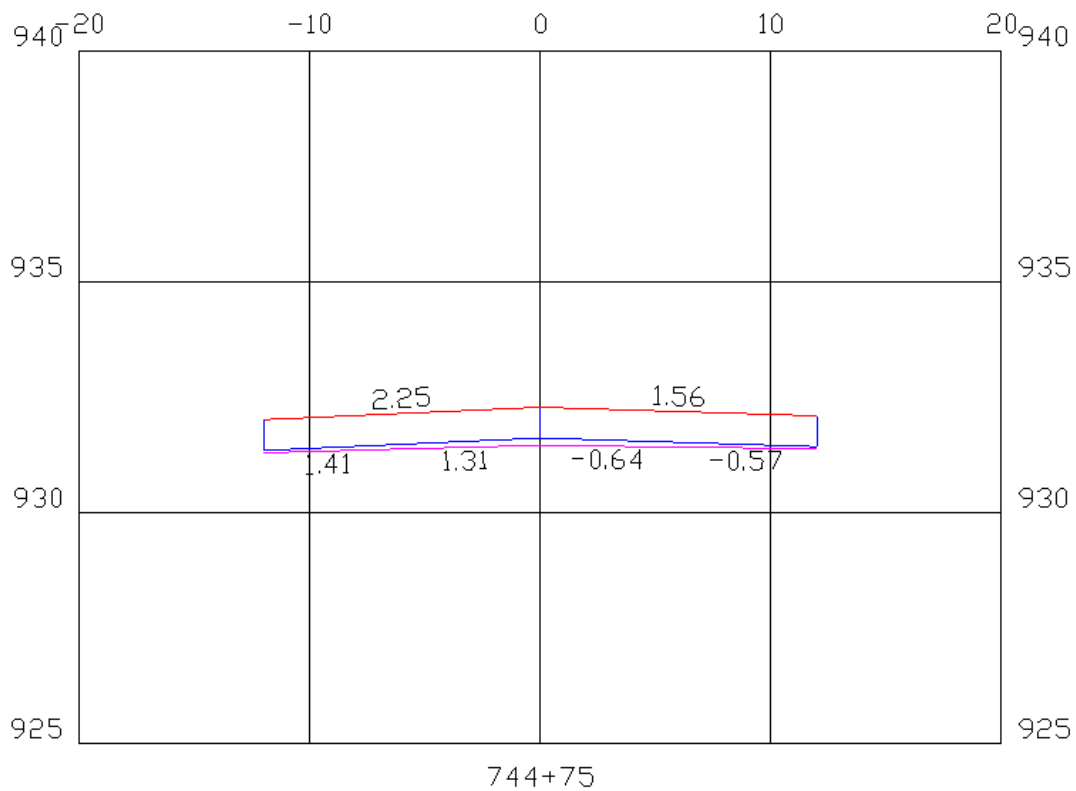


Review the report and click the **Exit (Doorway)** button when ready.

NOTE: Because we had the **Link Sections to Files** option set to **Prompt**, let's update the already placed sections! Click **Yes** on the dialog box shown below:



For this run, the **Template Grade Table** has been adjusted. In the report, the overlay quantities are the same and the fill quantity for the extra leveling has been reduced as expected. In this case, the *Template Grade Adjustment* reduced the Total Fill (extra leveling) amount from 463.142 C.Y. to 415.313 C.Y. (about a 10% reduction in needed leveling material) with an updated cross-section representation similar to that shown below:



The only difference in the final section is that the left side is steeper to better match the existing surface.

This concludes this tutorial lesson: Roadway Rehabilitation.

Hydrology and Watershed Analysis

This tutorial requires the Hydrology module.

This lesson will step through some of the more common *Hydrology* module routines, and design structures based on the analysis of a watershed.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

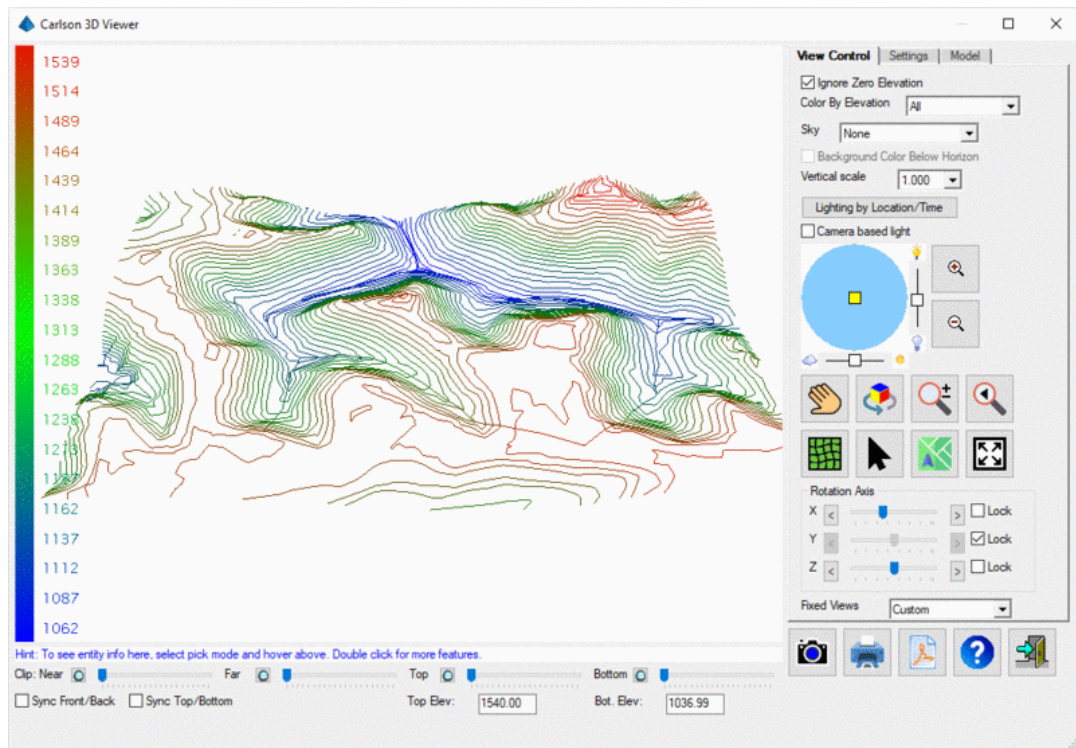
Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **HydroLesson.dwg** file. Activate the Hydrology module via Settings – Carlson Menus – **Hydrology Menu**.

2. The drawing file **HydroLesson.dwg** is a nice example to show the features of the Hydrology module. A surface file (**hydrolesson.tin**) is needed for these routines and is also supplied. Use the View – **Layer Control** command to become familiar with the various layers, and then go to the View – 3D View – Drawing Viewer command to see the change of elevations in the surface. When prompted:

Select entities for the scene.

[FILter]/<Select entities>: type ALL and press Enter twice

Here is a resulting graphic:

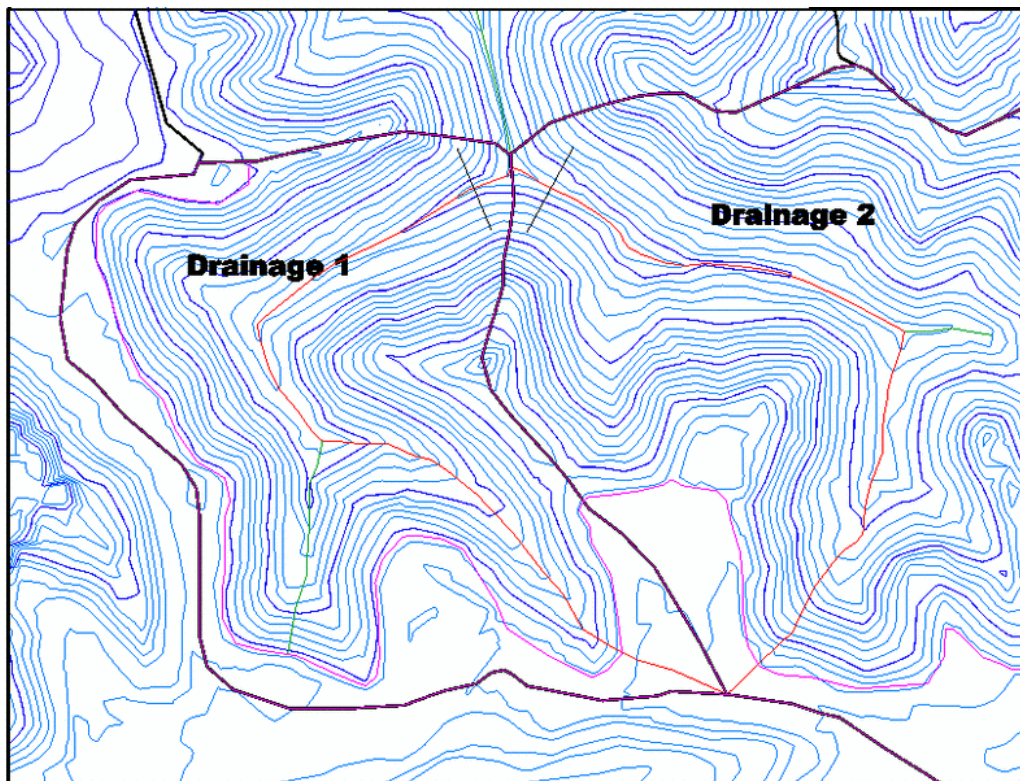


Click **Exit** (the doorway button) when finished.

There are two main drainage areas that we will be looking at:

- Drainage 1
- Drainage 2

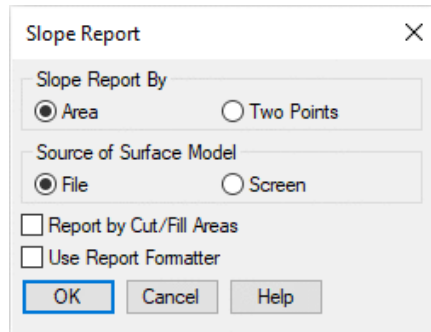
They are labeled in the below graphic. The other drainage areas in this region will be ignored, as they do not drain to the same area we are looking at, the north central low spot:



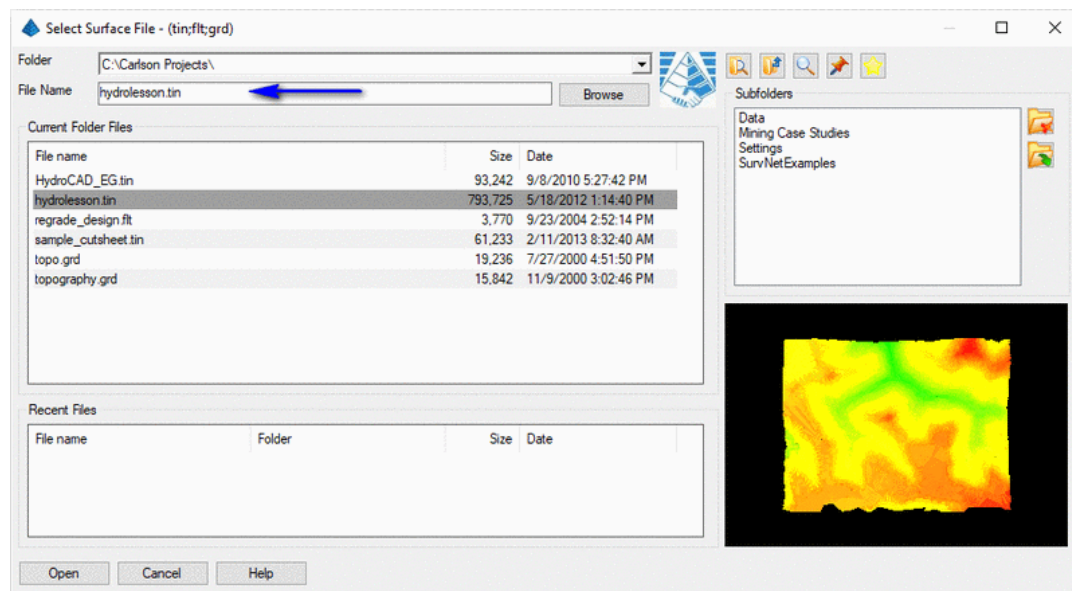
There are routines for finding these watersheds based on *grid* (*.grd) or *TIN* files, but this drawing has the closed polylines already generated. We will walk through some of the steps to gather the slope and area information.

- Issue the View – Freeze Layer by Pick command to freeze all contours, which are on two separate layers. Also pick the Magenta-colored **PILLARS** layer.

We would next like to evaluate the terrain data within each of the two watersheds available to us. Issue the Surface – Slope Report command which displays the dialog box below:



Set the values as shown above and click the **OK** button. On the subsequent dialog box, select the surface shown below and click the **Open** button when ready:



When prompted:

Select entities for the scene.

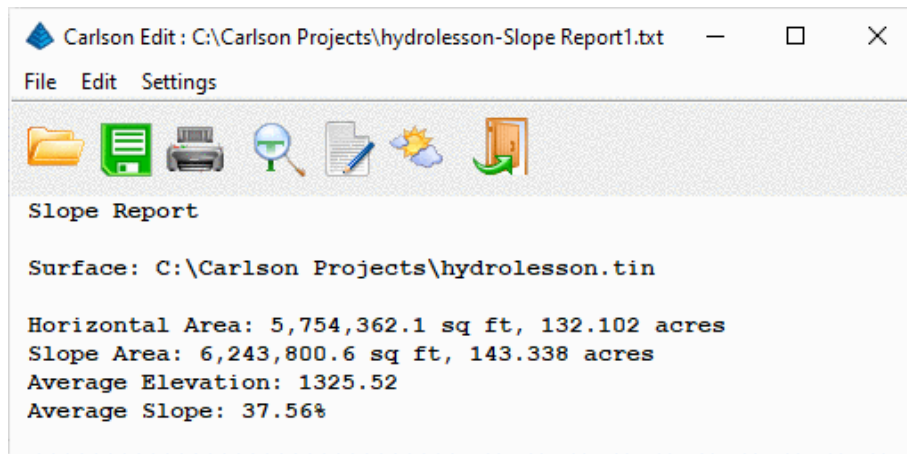
Select the Inclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: select the closed perimeter line that runs around Drainage 1 (press Enter when complete)

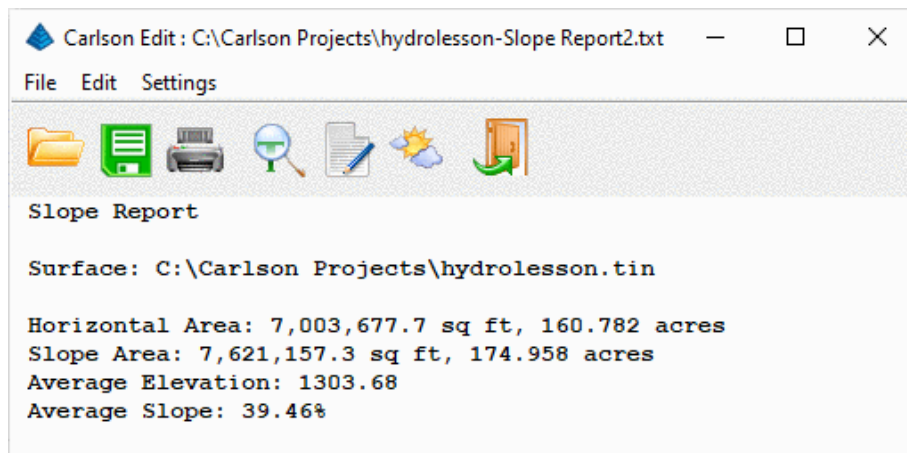
Select the Exclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: press Enter

The surface information from the selected TIN that is enclosed by the watershed perimeter polyline is echoed to the Standard Report Viewer as shown below:

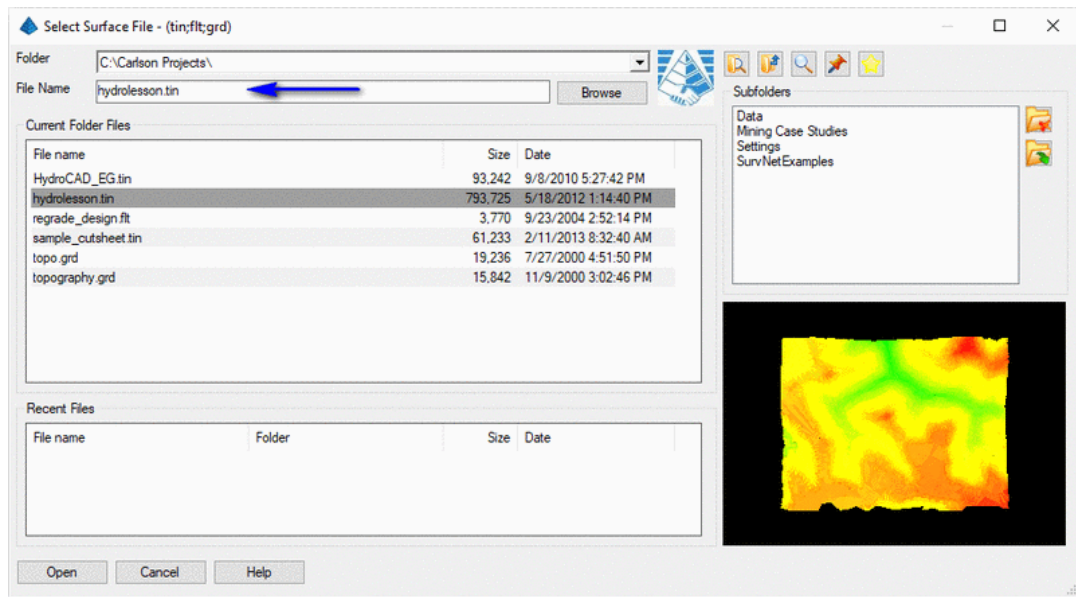


Click the **Exit** (Doorway) button when finished. Immediately press **Enter** to re-run the command and this time, select the closed perimeter line that runs around **Drainage 2**. Contrast its results to that shown above:

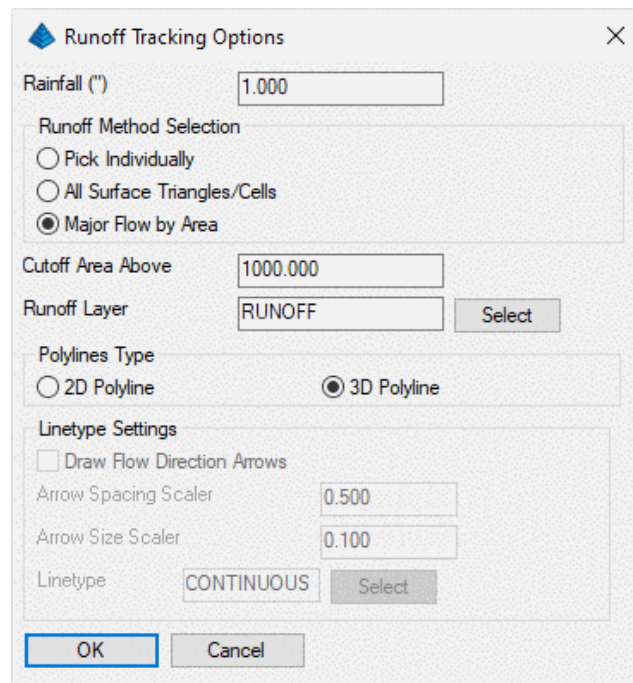


Notice that the watersheds are similar in size, and have approximately the same average slope. Click the **Exit** (Doorway) button when finished. Use the View – Thaw/On All Layers command to thaw the layers that you froze earlier.

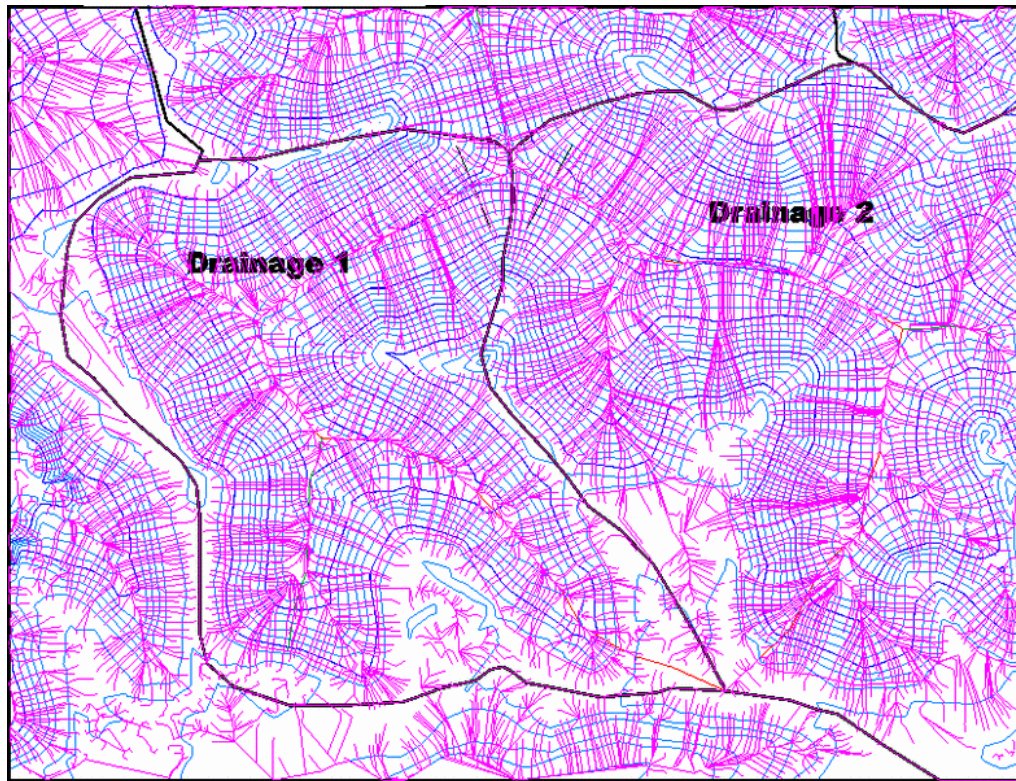
4. Our next task will be to evaluate the drainage characteristics of the site. Issue the Watershed – Runoff Tracking command. On the subsequent dialog box, select the surface shown below and click the **Open** button when ready:



On the following dialog box, accept the parameters as shown below and click **OK** when ready:



The result will be a collection of 3D polylines showing the path(s) water will take from a given rainfall intensity amount. Among other things, the command is useful to fine-tune a watershed boundary. For example, the *Pick Individually* option can be used to spot-pick locations near a boundary line. You can see which direction the water will flow and adjust the watershed perimeter accordingly. Shown is an example of the drawing with the runoff tracking lines falling within their respective watersheds:



Issue the View – **Freeze Layer by Pick** command and pick one of the previously generated flowlines and the two types of contour lines to remove them from the display.

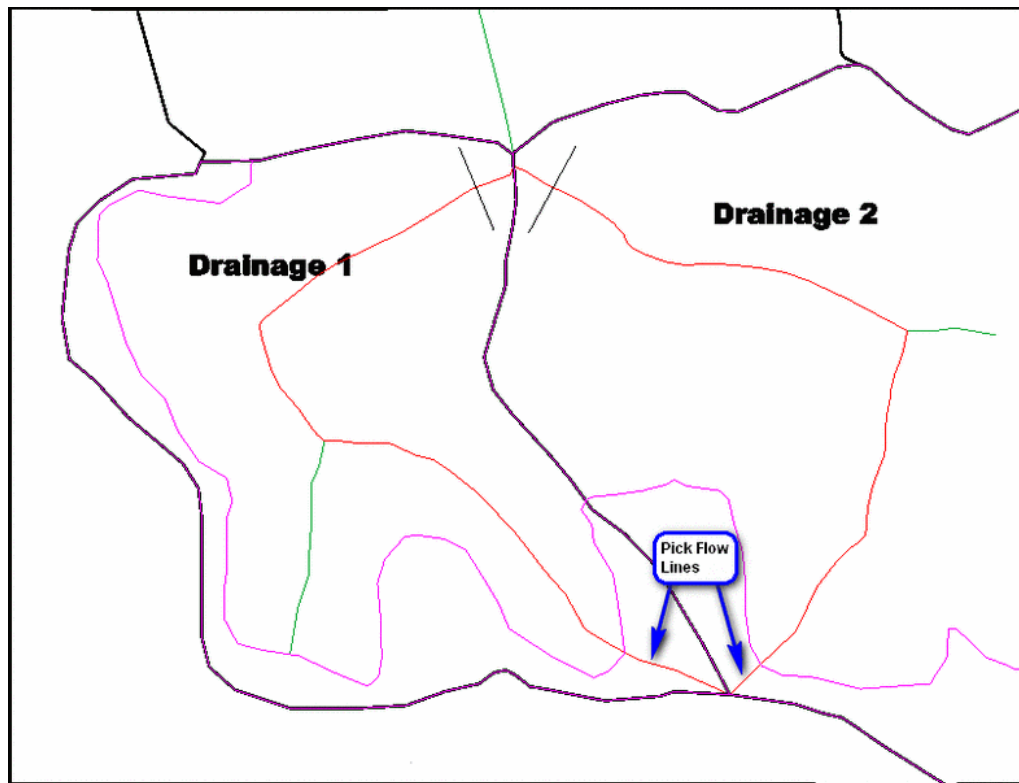
5. The next step is to see information for the longest flow line data within the watersheds. Issue the Watershed – Report Flow Values command. When prompted:

Type of flow line [<3DPolyline>/Profile]? press Enter for the 3D Polyline option

Select 3D polyline flow line: pick the Flowline for Drainage 1 as shown below

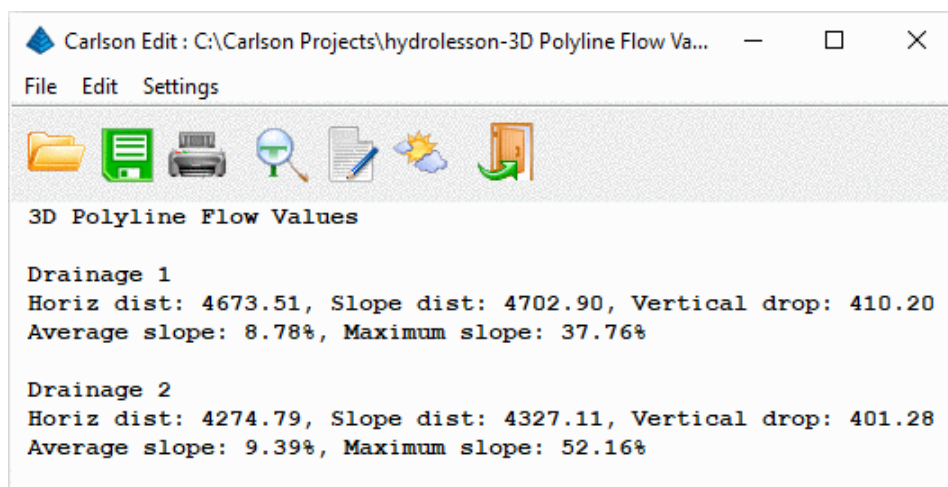
Select 3D polyline flow line: pick the Flowline for Drainage 2 as shown below

Select 3D polyline flow line: press Enter



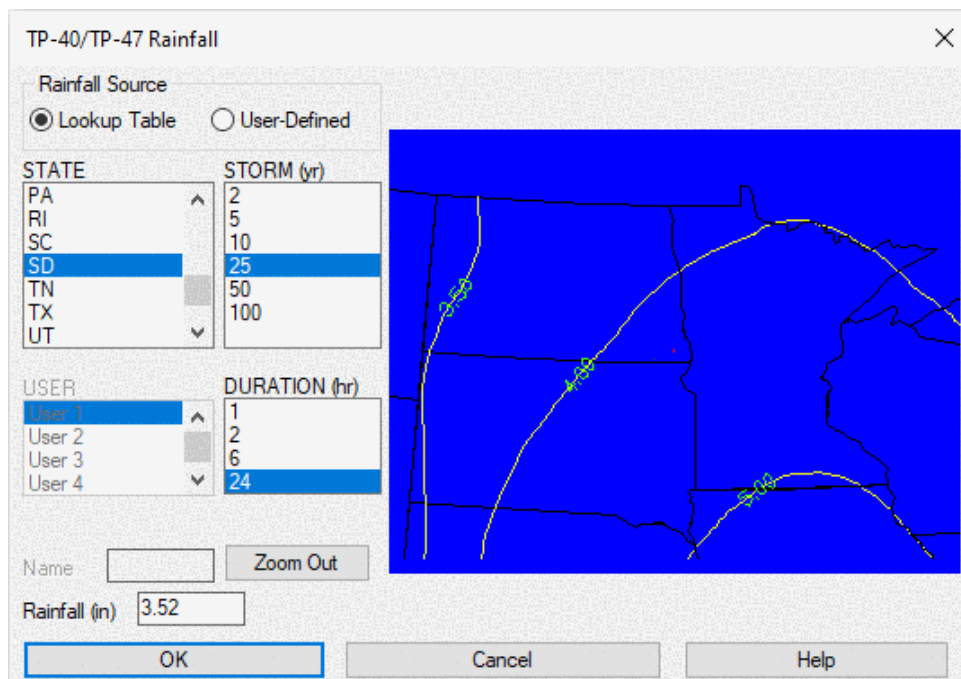
NOTE: The 3D polylines in the drawing were created (with runoff tracking and some editing) to represent the longest flow line.

The results are summarized in a nice report as illustrated below (with additional annotation added for clarity) showing the slopes and vertical drop. The report can be saved for future reference but "screen picks" of these polylines can be used in other aspects of the software:



Click the **Exit** (Doorway) button when finished.

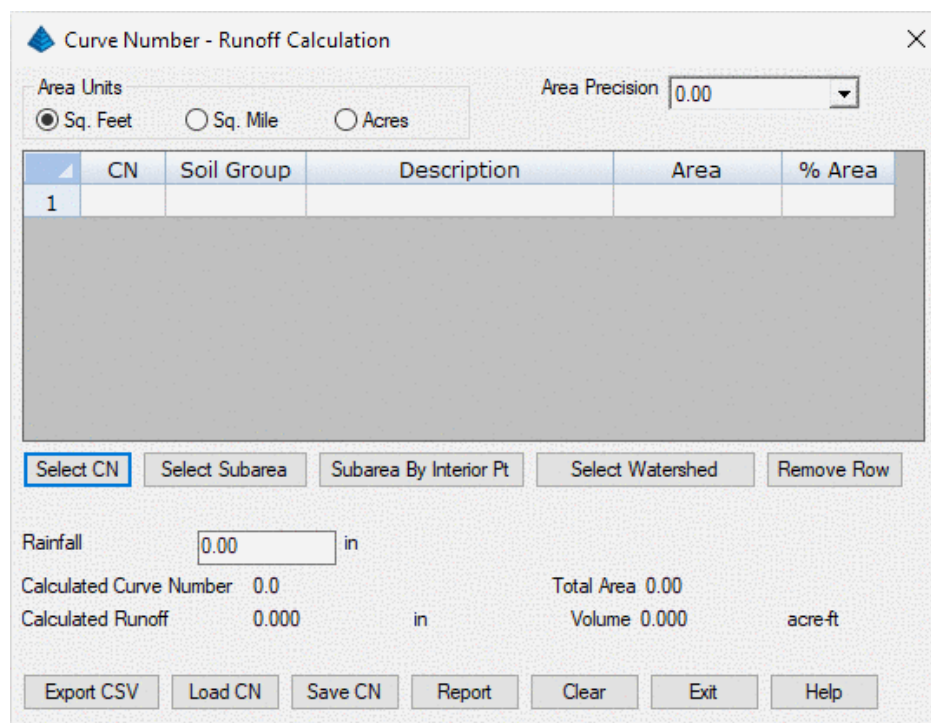
- Our next task will be to determine (or establish) the rainfall intensity/depth based on a given location, storm event and durations. Issue the Watershed – Rainfall Frequency & Amount – TP-40/TP-47 command to display the dialog box similar to that below:



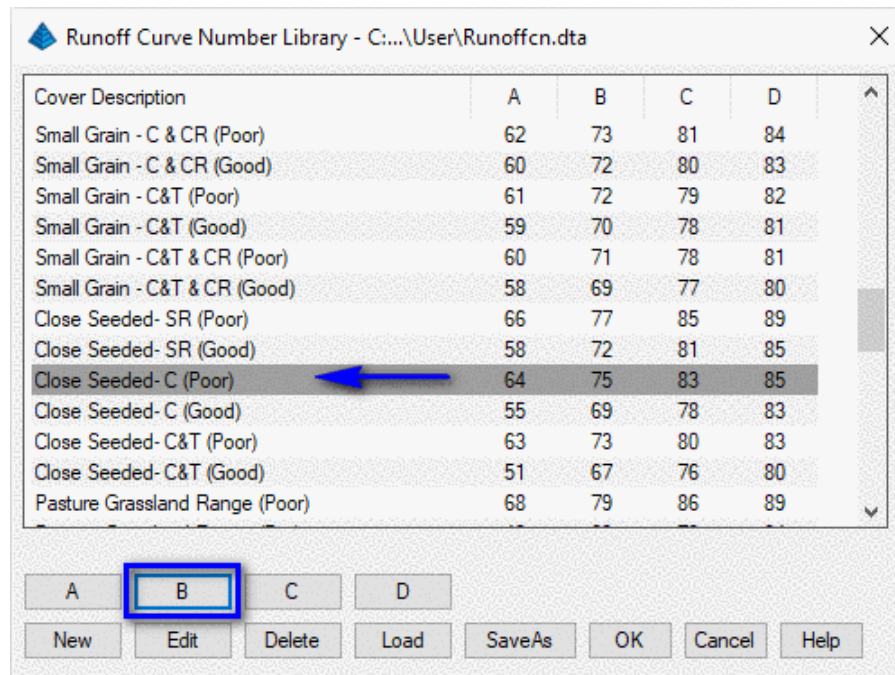
Set the values as shown above and note that as you click on the map, the rainfall intensity of the selected location will be reflected into the dialog box. For the sake of consistency, we will use the value of **3.52** inches (key this value in if you can't select it from the screen). For customizing this table to suit your needs, there is the **User Defined** toggle that permits individual entry in the lower left portion of the dialog box allowing for future retrieval if desired. Click **OK** when finished.

- Next, we'd like to obtain the runoff and volume of the storm based on the curve number and acreage of each sub-watershed. Use the View – Isolate Layers and click on the **Magenta "PILLARS"** layer and the **Black (or White) "Watershed.Perim"** layer.

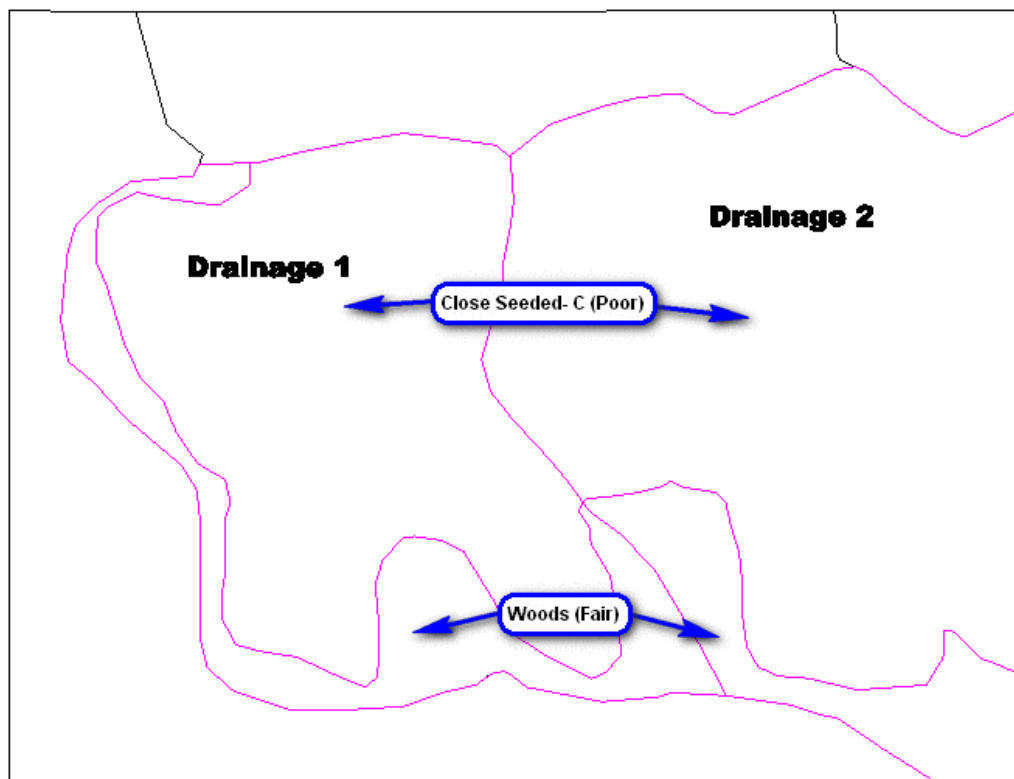
We will calculate a weighted curve number representing two land use types and will run this twice, once for each watershed. Issue the Watershed – Curve Numbers (CN) & Runoff command. The following dialog box appears:



Let's specify the first of our two Curve Numbers. Click the **Select CN** button to display a dialog box similar to that below:



Scroll through the list to locate and highlight the Land Use cited above and then click soil type button **B**. The entry is loaded into the *Curve Number - Runoff Calculation* dialog box. Use the land use guide below for the exercise to follow:



With this entry active in the list, click the **Subarea By Interior Point** button and when prompted:

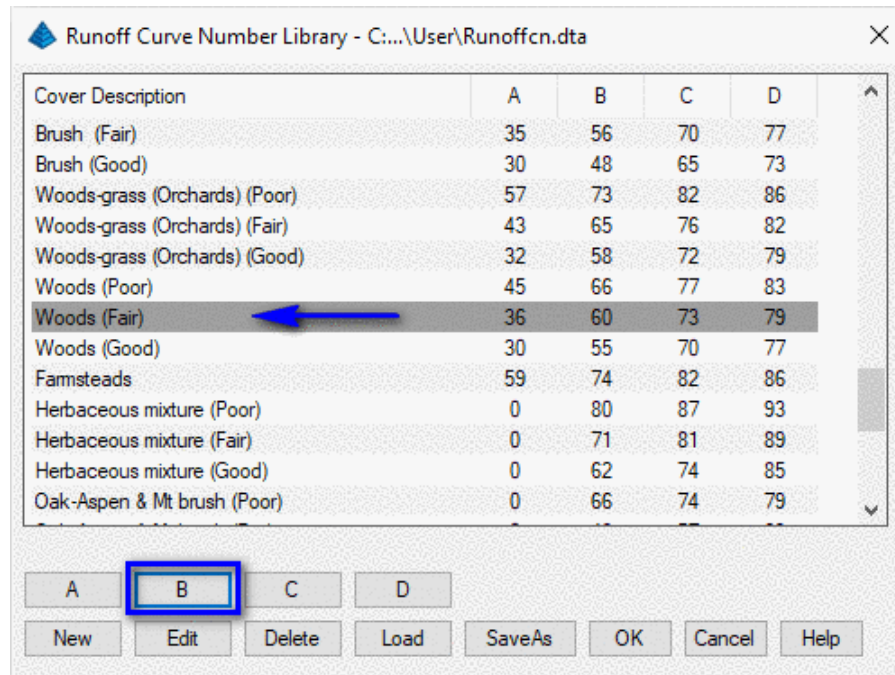
Pick point inside area perimeter: click within the *Drainage 1* area for the land use identified as **Close Seeded- C (Poor)** as shown above

(the selected area will shade)

Use the Selected Area? click Yes when you have the correct watershed sub-area

Pick point inside area perimeter (Enter to end): press Enter

Let's specify the second of our two Curve Numbers. **Highlight/select** the empty Row 2, and click the **Select CN** button to display a dialog box similar to that below:



Scroll through the list to locate and highlight the Land Use cited above and then click soil type button **B**. The entry is loaded into the *Curve Number - Runoff Calculation* dialog box. Again, with the *Row 2* entry active in the list, click the **Subarea By Interior Point** button and when prompted:

Pick point inside area perimeter: click within the *Drainage 1* area for the land use identified as **Woods (Fair)** as shown above

(the selected area will shade)

Use the Selected Area? click Yes when you have the correct watershed sub-area

Pick point inside area perimeter (Enter to end): press Enter

Set the remaining values as shown below:

Curve Number - Runoff Calculation

Area Units: Sq. Feet Sq. Mile Acres

Area Precision: 0.00

	CN	Soil Group	Description	Area	% Area
1	75	B	Close Seeded- C (Poor)	98.93	74.89
2	60	B	Woods (Fair)	33.17	25.11
3					

Select CN Select Subarea Subarea By Interior Pt Select Watershed Remove Row

Rainfall: 3.52 in

Calculated Curve Number: 71.2 Total Area: 132.10

Calculated Runoff: 1.090 in Volume: 11.997 acre-ft

Export CSV Load CN **Save CN** Report Clear Exit Help

Click on the **Save CN** button to save this weighted Curve Number data to a file as shown below:

Select a CN factors file - (.cn)

Folder: C:\Carlson Projects\

File Name: HydroLesson-D1-Ex

Current Folder Files:

File name	Size	Date

Recent Files:

File name	Folder	Size	Date

Subfolders:

- Data
- Mining Case Studies
- Settings
- SurvNetExamples

Save Cancel Help

To obtain the weighted Curve Number for the *Drainage 2* area, **select/highlight** the desired row/land use entry and click the **Subarea By Interior Point** button. Repeat for the other land use and the results should resemble that shown below:

Curve Number - Runoff Calculation

Area Units: Sq. Feet Sq. Mile Acres

Area Precision: 0.00

	CN	Soil Group	Description	Area	% Area
1	75	B	Close Seeded- C (Poor)	142.13	87.23
2	60	B	Woods (Fair)	20.81	12.77
3					

Select CN Select Subarea Subarea By Interior Pt Select Watershed Remove Row

Rainfall: 3.52 in

Calculated Curve Number: 73.1 Total Area: 162.94

Calculated Runoff: 1.198 in Volume: 16.269 acre-ft

Export CSV Load CN **Save CN** Report Clear Exit Help

Click on the **Save CN** button to save this weighted Curve Number data to a file as shown below:

Select a CN factors file - (.cn)

Folder: C:\Carlson Projects\

File Name: HydroLesson-D2-Ex.cn

Current Folder Files:

File name	Size	Date
HydroLesson-D1-Ex.cn	122	

Recent Files:

File name	Folder	Size	Date
HydroLesson-D1-Ex.cn	C:\Carlson Projects\	122	

Subfolders: Data, Mining Case Studies, Settings, SurvNetExamples

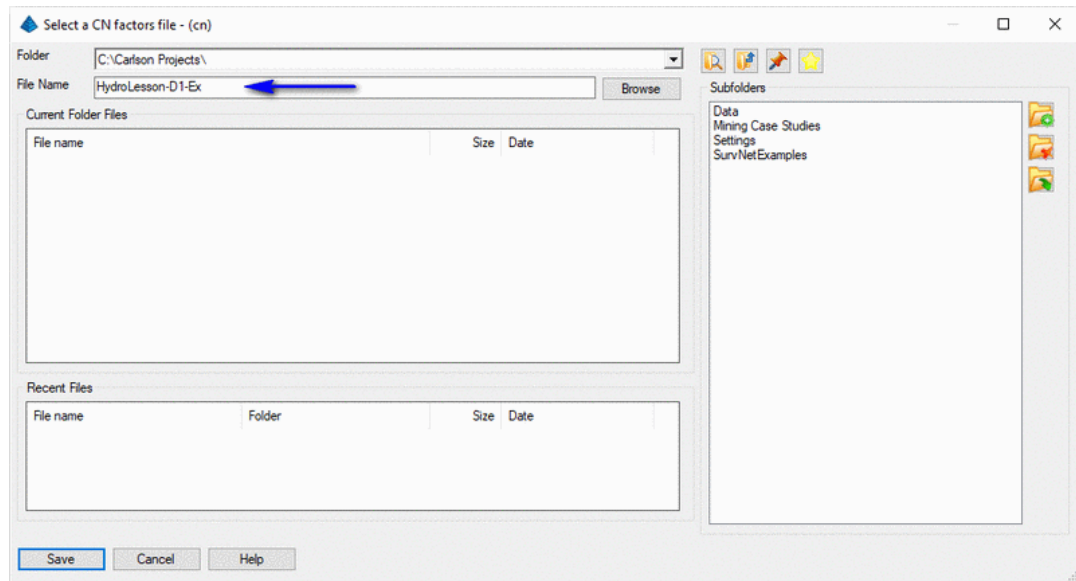
Save Cancel Help

Control returns to the *Curve Number - Runoff Calculation* dialog box and click the **Exit** button to complete the command.

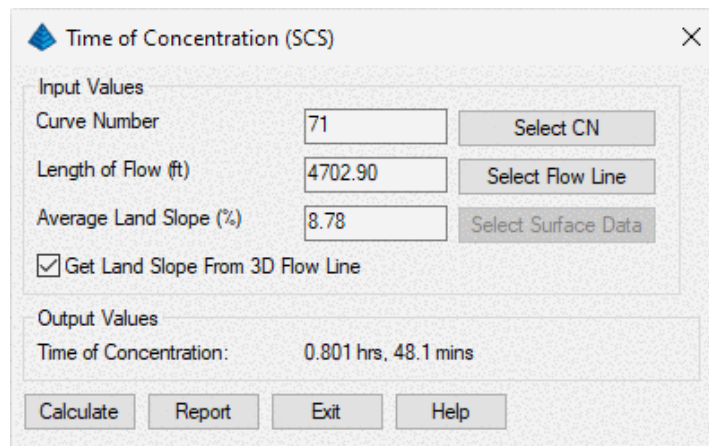
- The next thing we'd like to do is establish the Time of Concentration (T_c) for the two watersheds. This is a quick step necessary for ultimately getting the *Peak Flow*. Issue the View – Restore Layers command to bring back the visibility of the **Drainages** (flowlines) layer previously frozen as a result of the *Isolate Layers* command.

NOTE: The *CN number* from the previous exercise (for *Drainage 2*) will be considered "active" and we'll need to exercise care to make sure the proper flowline is selected. Let's follow the Drainage 1/Drainage 2 process to make sure!

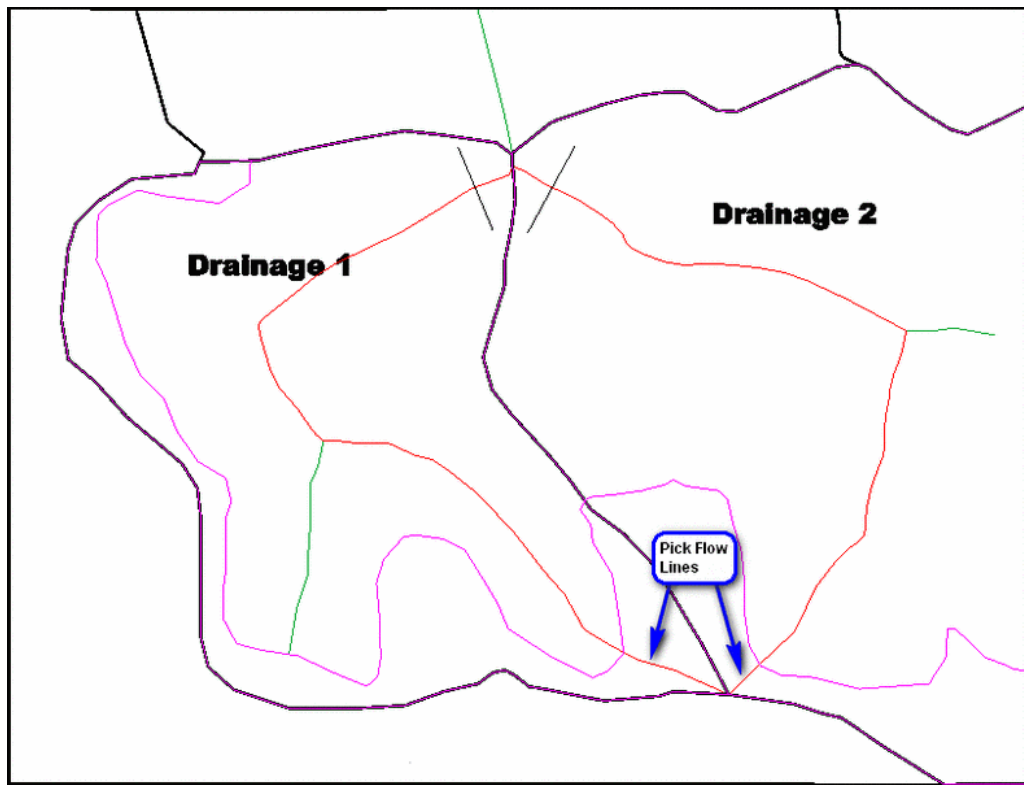
- (a) Recall the Curve Number information for *Drainage 1* by re-issuing the Hydrology – **Curve Numbers (CN) & Runoff** command.
- (b) Click the **Load CN** to load the *.cn file as illustrated below (use the **Open** button rather than the button shown here):



- (c) Click the **Exit** button.
- (d) Issue the Watershed – Time of Concentration (Tc) – SCS Method command. This will launch the *Time of Concentration (SCS)* dialog box similar to that shown below:



- (e) Click the **Select Flow Line** button and graphically select the *Red* flowline that contributes to **Drainage 1**:



(f) With the settings shown, click the **Calculate** to obtain the T_c for *Drainage 1*:

Time of Concentration (SCS) ✕

Input Values

Curve Number	71	Select CN
Length of Flow (ft)	4702.90	Select Flow Line
Average Land Slope (%)	8.78	Select Surface Data

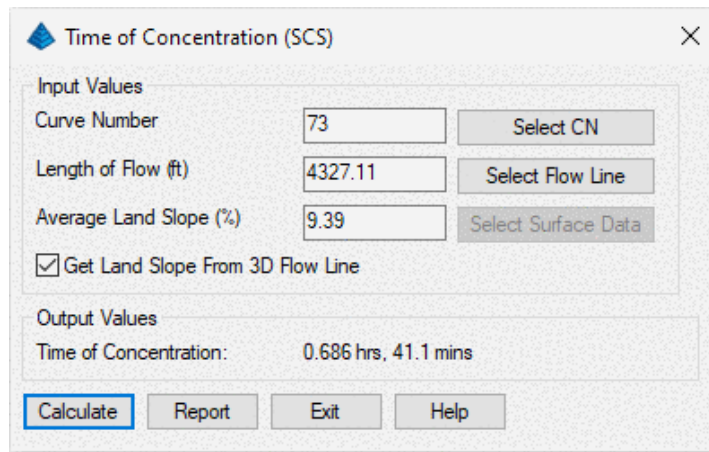
Get Land Slope From 3D Flow Line

Output Values

Time of Concentration: 0.801 hrs, 48.1 mins

Calculate
Report
Exit
Help

Repeat the steps above (loading the Curve Number information for *Drainage 2* and graphically selecting the *Red* flowline that contributes to **Drainage 2**), obtaining the results shown below:



Note the two results and click **Exit** when ready.

9. Now, let's see what the peak flow will be in each drainage. Issue the Watershed – Run Peak Flow – Graphical Method command. The drainage area of the last watershed calculated should appear in the **Drainage Area** window. If not, then use either:

- the View – **Freeze Layer by Pick** command to leave the *Watershed_Perim* layer visible and follow with the **Select Area from Screen** button, or
- keying-in the value as reported in the Curve Number Runoff Calculation 1 exercise from above

Set the *Rainfall Depth*, *24 hr. Rainfall Frequency*, *Runoff Curve Number* and *Time of Concentration* values as shown in the table below (summarized for convenience):

	132.10198	162.94296	
Rainfall Depth (Inches)			
	3.52	3.52	
24 hr. Rainfall Frequency (Year)			
	25	25	
Runoff Curve Number			
	71	73	
Time of Concentration (Minutes)			
	48.1	41.1	
Percent of Pond and Swamp Areas			
	0.00	0.00	
Pon and Swamp Adjustment Factor			
	1.00	1.00	
Storm Type			
	II	II	
Peak Discharge		80.726	123.399

Peak Flow Data Values

With the appropriate values entered, click **Calculate** to obtain the peak discharge values as shown below:

Drainage 1

Drainage 2

Peak Discharge Values

Notice that all of these routines have a **Report** button to keep a running log of all the calculated data.

Now we need to see how large the ponds need to be to detain this size of a storm event. Initiate the Structure – Detention Pond Sizing – TR-55 Method command to achieve this.

The *Drainage Area* and *Peak Inflow Discharge* values from the last area calculated will appear. In this example, we will allow for a combined maximum 10 ft³/sec (cfs) to be discharged from the ponds combined (in other words, 5 cfs from each will be our **Desired Peak Outflow**). Use the summary of values below and click **Calculate**. The *Runoff Volume* and the *Storage Volume* will appear at the bottom of the window:

	132.10198	162.94296
Peak Inflow Discharge (cfs)		
	80.726	123.399
Desired Peak Outflow (cfs)		
	5.00	5.00
Runoff Q - Post-Dev (inches)		
	1.090	1.198
Storm Type		
	II	II

Detention Pond Sizing Data Values

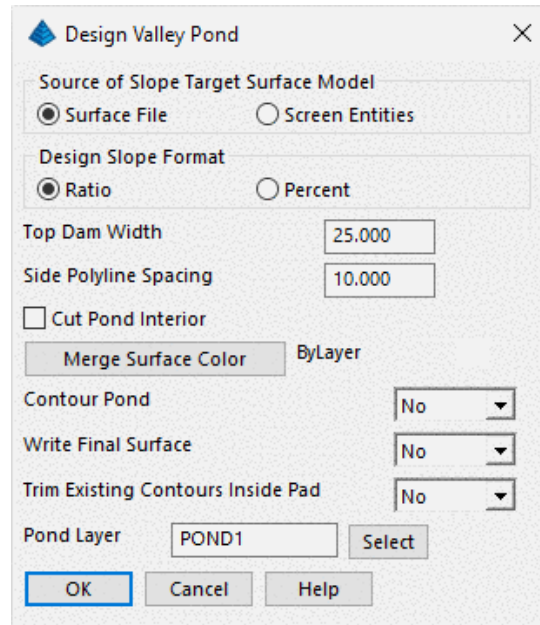
Drainage 1

Drainage 2

Detention Pond Size

With the storage areas determined, we now have a starting point and we can now create the ponds with these sizes.

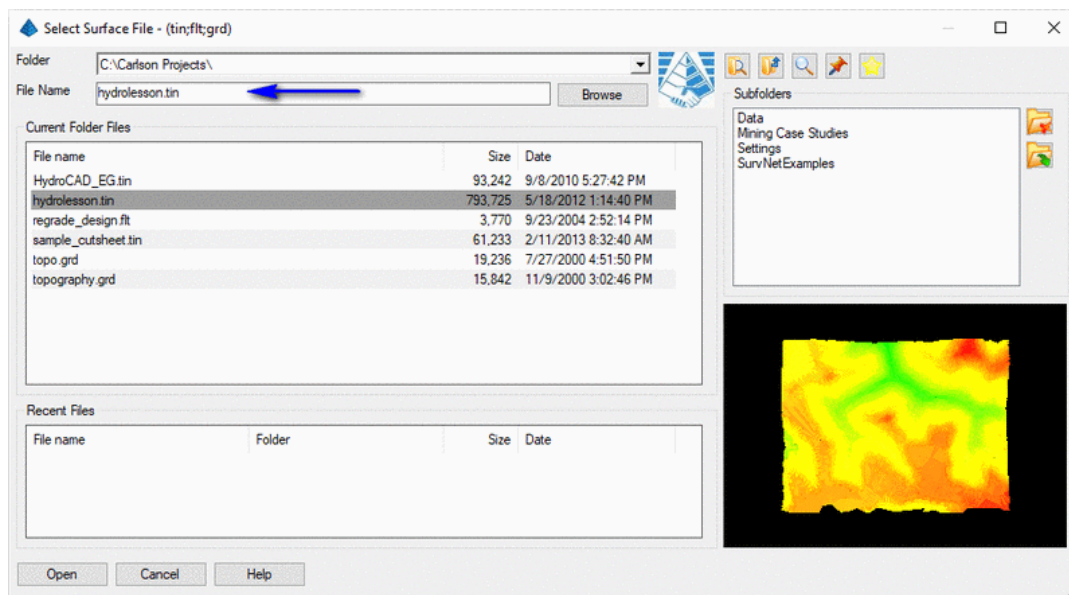
We know approximately where we want the two ponds, and have the dam centerline polylines drawn in the drawing already. They are in the *Top-of-Dam* layer. If it is frozen, use the View – **Layer Control** command to thaw it now. Issue the Structure – Design Valley Pond command. For Pond 1 (corresponding to Drainage 1), use the starting values as shown below and click **OK** when ready:



When prompted:

Pick top of pond polyline: pick the polyline for the Drainage 1 region

Select Existing Surface: specify the TIN file shown below and click the **Open** button when ready:



Pick point within pond: pick within the storage area (southwest of the dam centerline for Drainage 1)

Outslope ratio <2.00>: press Enter

Interior slope ratio <2.00>: press Enter

Top of dam elevation: 1109

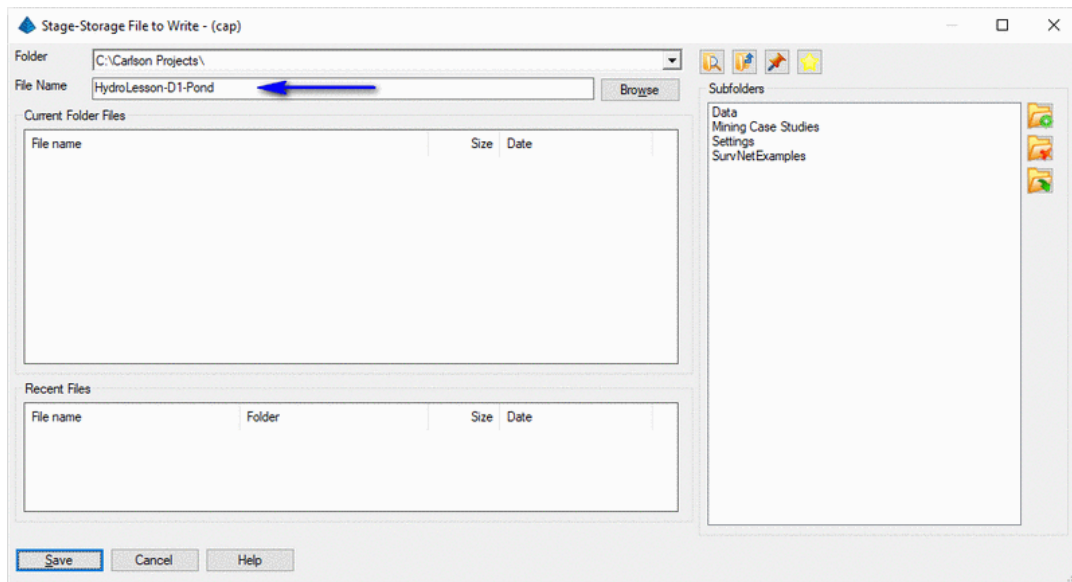
Calculate stage-storage values [<Yes>/No**]?** press Enter

Method to specify storage elevations [<Automatic>/Interval/Manual**]?** specify the Interval (i) method

Starting elevation **<1078.65>:** 1078 (we'll start just below the lowest elevation)

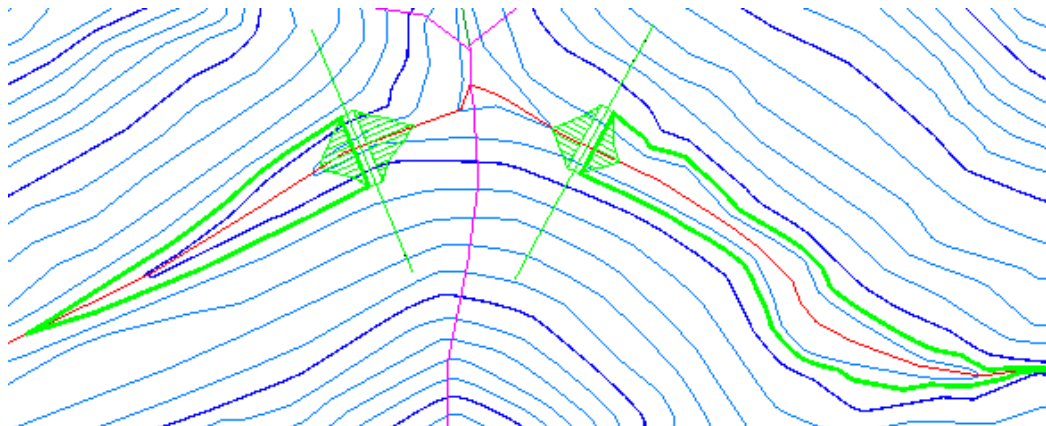
Elevation interval **<2.00>:** press Enter (notice the **Storage** capacity of the pond is greater than the desired *Storage Volume*, click the **Exit** (Doorway) button when ready)

Write stage-storage file [Yes/<No>**]?** indicate Yes (indicate the file name below and click **Save** when ready)



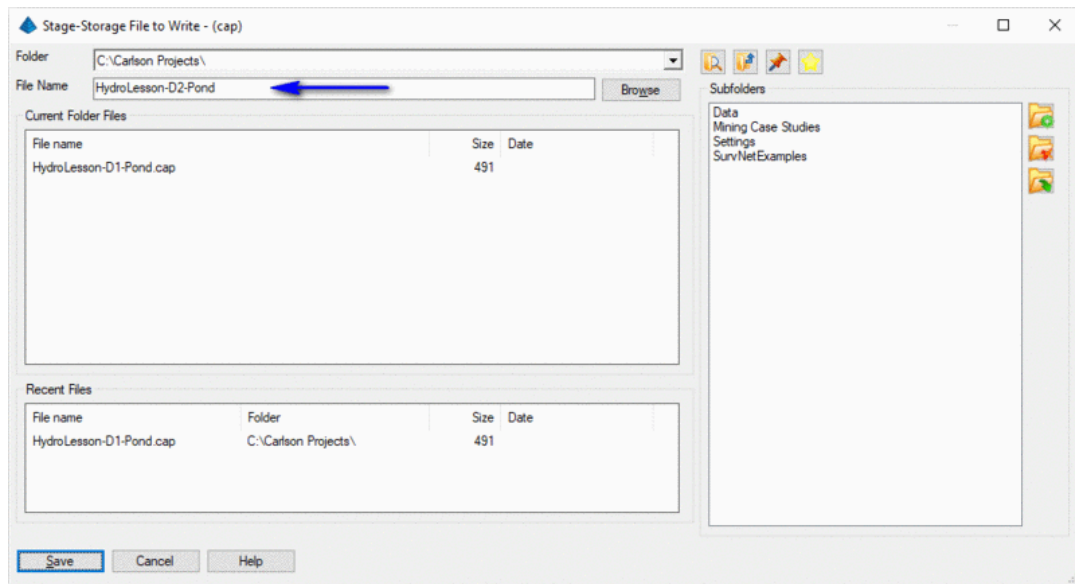
Adjust parameters and redesign pond [Yes/<No>**]?** press Enter

You should now have a pond that looks like the one on the left in the following image (emphasis added for clarity):



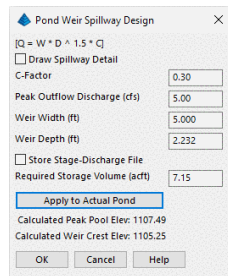
Repeat the routine for Pond 2 using the same parameters except:

- **Top of dam elevation:** 1087
- **Starting elevation **<1063.01>**:** 1062
- **Write stage-storage file [**Yes/<No>**]?** indicate Yes (indicate the file name below and click **Save** when ready)

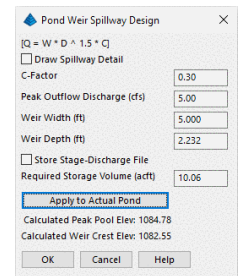


Now let's see what the spillway will need to be for the storage calculated. Run the Structure – Rectangular Weir Design command to display the dialog similar to that below:

Drainage 1/Pond 1



Drainage 2/Pond 2



Rectangular Weir Design Values

Supply the following values (pressing the **Tab** key between entries):

- **C-Factor:** 0.30
- **Peak Outflow Discharge (cfs):** 5.00
- **Weir Width (ft):** 5.00 (the Weir Depth will be found, however, we'll use a Drop Pipe in Pond 1, and a channel in Pond 2)
- **Required Storage Volume (acft):** provide the respective values shown above

Click the **Apply to Actual Pond** button and choose the Pond 1 stage-storage capacity file created earlier. This should yield two elevations shown in the dialog: the **Peak Pool Elevation** and the **Weir Crest Elevation** (bottom of spillway). This will be our principal spillway. Our emergency spillway will be assumed to be 1.5 feet higher.

Now that we have the spillway elevations and a capacity file (*.CAP) for each pond, let's draw the *Stage Storage/Area Curve Graphs* to get a graphic of the curves. Issue the Structure – Stage-Storage – Draw Stage-Storage Curve which displays a dialog box similar to that below. Use the settings below for Pond 1 and click **OK** when ready:

Draw Stage-Storage Curve

Task Information

Name of Pond: Sediment Control Structure No. 1

Permit Application Number (eg. 898-5252): 900

Permit Item Number (eg. 30.3): 30.3

Page Number: 1

Stage-Storage Input Options

From a Stage-Storage File

Manually Input Elevation and Area / Dimension(Width,Length) / Volume

Select File: C:\Carlson Projects\HydroLesson-D1-Pond.cap

Stage-Storage Data Preview Window

Elev (Ft)	Area (Acre)	Interval (Ft)	Avg. Area ...	Inc. Vol (A...	Acc. Vol (A...	Stage
1078.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1080.0000	0.0061	2.0000	0.0031	0.0033	0.0033	2.0000
1082.0000	0.0147	2.0000	0.0104	0.0201	0.0234	4.0000

Pond Information

Elev of Top of Structure: 1109.0000

Elev of Principal Spillway: 1104.50

Elev of Emergency Spillway: 1106.00

Elev of Clean Out Level: 1079.00

Calculate Storage or Elevation Points

Storage Units: Acre-Ft Percent of Total Storage

Elevation (Ft): 0.0000

Storage: 1.0000

Calculate Permanent Pool Elevation

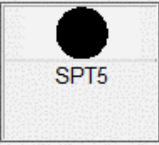
Total Runoff (Acre-Ft): 0.0000

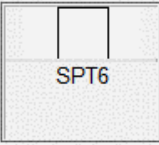
Permanent Pool Elevation (Ft):

OK Cancel Help

A secondary dialog box appears. Complete as follows and click **OK** when ready:

Draw Stage-Storage Setup

 SPT5

 SPT6

Symbol Name: SPT5

Symbol Name: SPT6

Scale of Drawing: 1.0

Draw Stage-Area Curve

Range of Elevation (Ft): 31.00

Total Storage Range (Acre-Ft): 9.12

Elev. Grid Interval: 2

Storage Grid Interval: 2

Elev. Grid Decimals: 0.00

Storage Grid Decimals: 0.00

Draw Square Grids

Horiz. Axis Label: Accumulative Storage

Vert. Axis Label: Elevation MSL

Title of Plot: Stage Storage Curve

Data Title: Storage volume computations

Skip every 2nd table entry

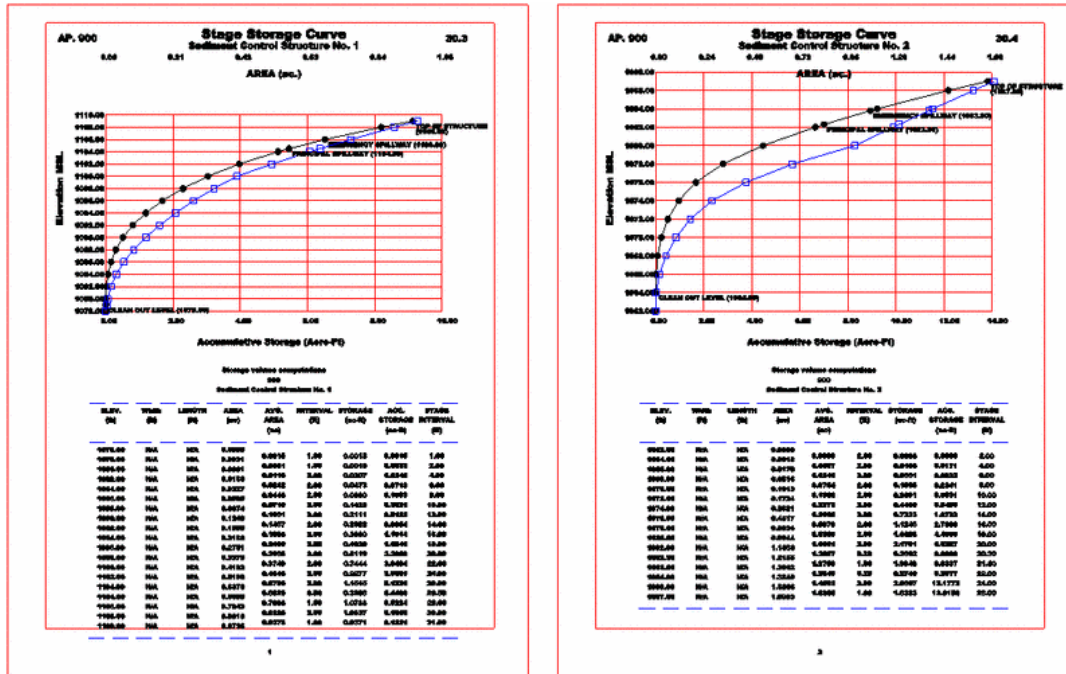
Footnotes

None Certification Business Address

OK Cancel Help

When prompted: **Pick starting position:** pick the lower left corner of the report

Do the same for *Pond 2*, with the other elevations from the **Spillway** and **Top of dam** as calculated above and choose to put this on Page Number 2. Your graphs should look like the two pictured below (emphasis added):



Let's add a drop-pipe to *Pond 1*. This will give us a Stage-Discharge file that we will add to our structure in the routing of the storm through our ponds. Launch the Structure – Drop Pipe Spillway Design command to display a dialog box similar to that shown below:

Drop Pipe Design

Riser Parameters

Shape: Circular

Top Elev: 1106 ft

Base Elev: 1095 ft

Diameter: 10 in

Width: in

Orifice Coef.(0.4-0.6): 0.60

Weir Coef.(3.33 in general): 3.33

Culvert Parameters

Calculation Method: Carlson Legacy HDS-5

Diameter: 250 in

Length: 24 ft

Outlet Invert Elev: 1080.000 ft

Slope: 62.500 %

Friction Coefficient: 0.013

Entrance Loss Coefficient: 1.50 Set

Riser Inlets

Inlet Name	Inlet Type	Invert (ft)

Add
Edit
Remove

Calculation

Headwater Elev: 1109.004 ft

Discharge: 4.55 cfs

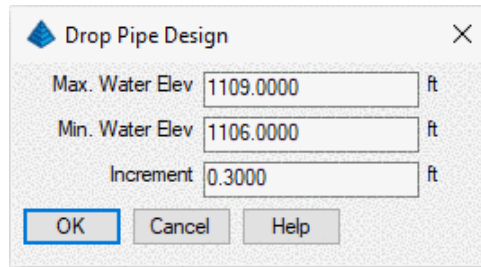
Calc Discharge
Calc Headwater Elevation

Report Setup
Report
Stage-Discharge Result
Draw

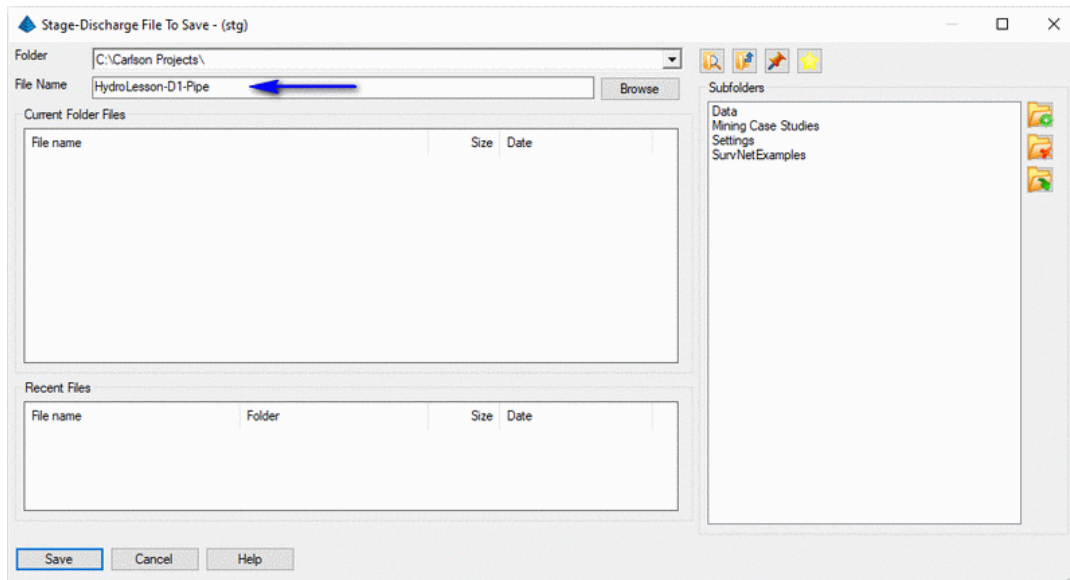
OK
Cancel
Load
Save
Help

We will design one for the 5 cfs flow we need. Enter in the values shown in the window and click the **Calc Discharge**

to see the results. We appear to be near the 5 CFS discharge we are looking for. Click the **Stage-Discharge Result** button to display a dialog box similar to that shown below and click **OK** when ready:



The results are summarized as illustrated below. Click the **Write Stage-Discharge File** button to create a *.stg file as suggested below:



Click **Close** and/or **OK** to dismiss any remaining dialog boxes.

As mentioned earlier, *Pond 2* will have a channel which will create a different Stage-Discharge curve and resulting Stage-Discharge file. Run the Structure – Channel Design – Non-Erodible command to display a dialog box similar to that shown below:

Channel Design (Manning's Eq)

Channel Type

- Triangular, Equal Side Slopes
- Triangular, Unequal Side Slopes
- Trapezoidal, Equal Side Slopes
- Trapezoidal, Unequal Side Slopes
- Rectangular Channel
- Irregular Shaped Channel
- Parabolic Shaped Channel

Channel Dimensions

Ratio Left: 1.0

Ratio Right: 1.0

Base Dimension: 5.0

Wetted Perimeter:

Cross-Sectional Area:

Top Dimension: 10.0

Channel Design Parameters

Specify 2nd Manning's n for Upper Banks

Channel Slope (%): 0.1

Manning's n for Channel:

Depth of 1st Manning's n from Channel Floor: 0.0

Manning's n for Upper Banks:

Channel Design

Discharge (cfs): 4.9971

Depth (feet): 0.7648

Velocity (fps): 1.1334

Standard Parameters

Required Freeboard (ft): 0.30

Channel Lining:

Specify the parameters shown *except* the **Discharge (cfs)** item... set its value to **5** and click on the **Calculate** button to determine the Channel *Depth* and Flow *Velocity*. Once satisfied, click on the **Write Stage-Discharge** button to create a *.stg file as suggested below and click **Save** when ready:

Stage-Discharge File to Write - (.stg)

Folder: C:\Carlson Projects\

File Name: HydroLesson-D2-Chan

Current Folder Files

File name	Size	Date
HydroLesson-D1-Pipe.stg	228	

Recent Files

File name	Folder	Size	Date
HydroLesson-D1-Pipe.stg	C:\Carlson Projects\	228	

Subfolders

- Data
- Mining Case Studies
- Settings
- SurvNetExamples

When prompted:

Maximum Depth <5>: 1.5

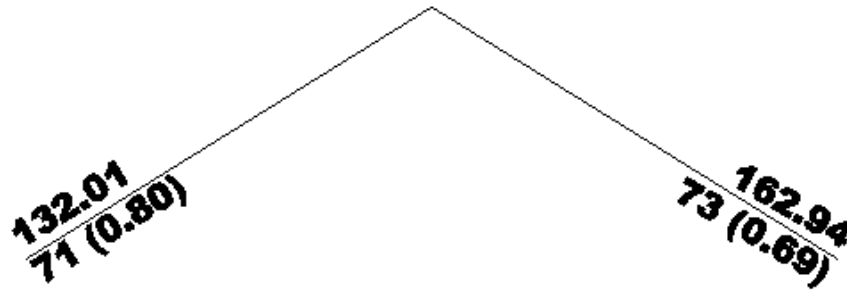
Interval <1>: 0.30

Base Elevation: 1082.30

Click **Exit** to dismiss the dialog box.

Now we will look at creating a schematic illustration of the project with annotation placed on them. We will first produce a hydrograph of the two drainages without the ponds, then add the ponds to the flow polylines and regenerate the hydrographs. We hope to reduce the discharge to a much smaller amount, but over a longer period of time.

Issue the Watershed – TR-20 Routing – Draw Flow Polylines to approximate the direction of flow: As seen in the diagram, pick from SW to NE to simulate the general direction of flow for *Drainage 1*:



When prompted:

Text size <4.00>: press Enter

Draw from highest to lowest elevation (direction of flow).

Exit/Pick point: pick a starting location

Undo/Exit/Join/Pick point: pick an ending location to the northeast

Undo/Exit/Join/Pick point: press Enter and complete the values as shown and click **OK** when ready:

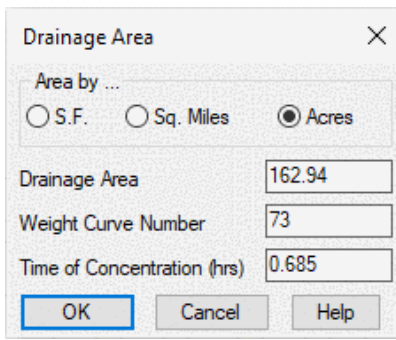
Drainage Area		
Area by ...		
<input type="radio"/> S.F.	<input type="radio"/> Sq. Miles	<input checked="" type="radio"/> Acres
Drainage Area	132.10	
Weight Curve Number	71	
Time of Concentration (hrs)	0.80	
OK Cancel Help		

Draw another flow polyline [<Yes>/No]? press Enter

Exit/Pick point: pick a starting location

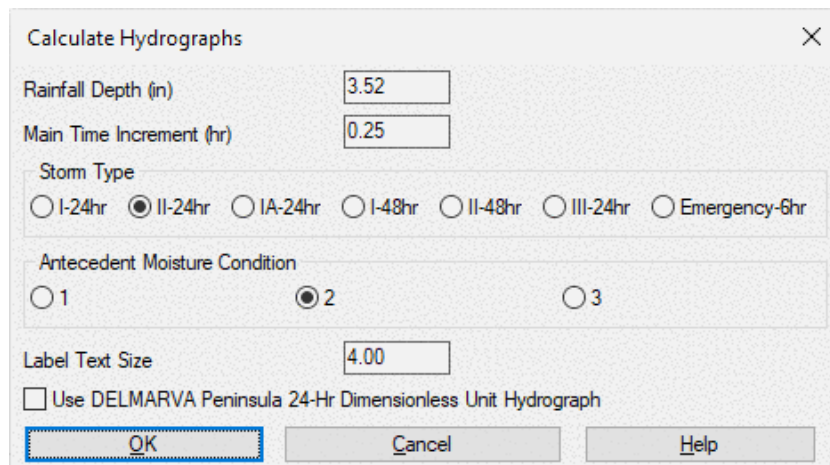
Undo/Exit/Join/Pick point: pick an ending location to the northwest

Undo/Exit/Join/Pick point: press Enter and complete the values as shown and click **OK** when ready:



Draw another flow polyline [<Yes>/No**]? indicate No**

Our next task will be to run the TR-20 program and generate a hydrograph file that we can draw on screen. Run the Watershed – TR-20 Routing – Hydrograph Development command supply the values as shown below and click **OK** when ready:



When prompted:

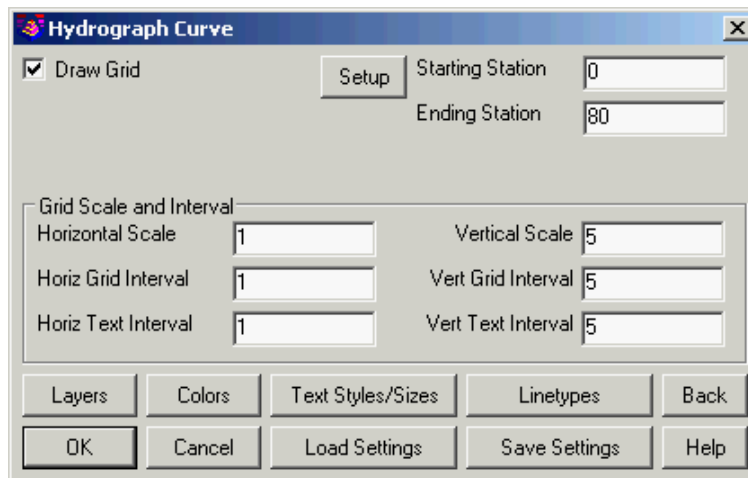
Select flow polylines, structure and reach symbols.

[FILter]/<Select entities>: *crossing Window* the flow line entities and annotation

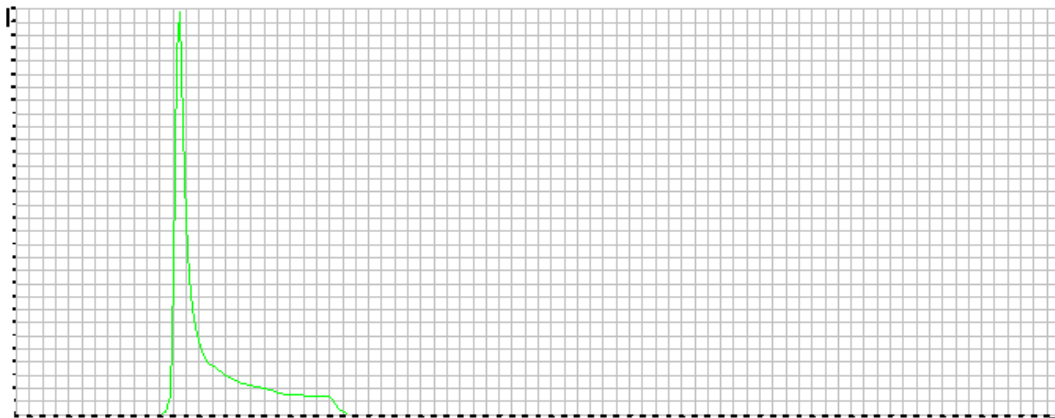
[FILter]/<Select entities>: press Enter

The routine will run TR-20 and give a **Standard Report Viewer** report. There are now some hydrograph files that we can draw in the next step.

We'd now like to draw our hydrographs. Issue the Watershed – Draw Hydrograph and select the J1ADD.h1. This is the file of both drainages combined.



The scales to be used should be about 1,1,1,5,5,5 and we will draw the grid on the first one, and turn off the grid for additional hydrographs. Choose starting time of 0, and an ending time of 80 (the next one will go that long):



Our next task will be to create a schematic representation of the ponds and spillways. Issue the Watershed – TR-20 Routing – Locate Structures command which will create a small triangle at the end of each flow line where they were picked. You will run this on each flow line individually:

	Elevation (Ft)	Discharge (CFS)	Storage (Acre-Ft)
1	1105.0000	0.0100	0.0000
2	1106.0000	4.1560	0.0000
3	1107.0000	5.8774	0.0000
4	1108.0000	7.1984	0.0000
5	1109.0000	8.3120	0.0000
6	1110.0000	9.2931	0.0000
7			
8			
9			

Water Elev at T = 0: 0.00 ft

Click the **Load** button twice:

1. once to pick the *.cap file, and,
2. once again to pick the *.stg file

Click **OK** and re-issue the **Hydrograph Development** command again to generate the new hydrographs.

This completes the tutorial: Hydrology and Watershed Analysis.

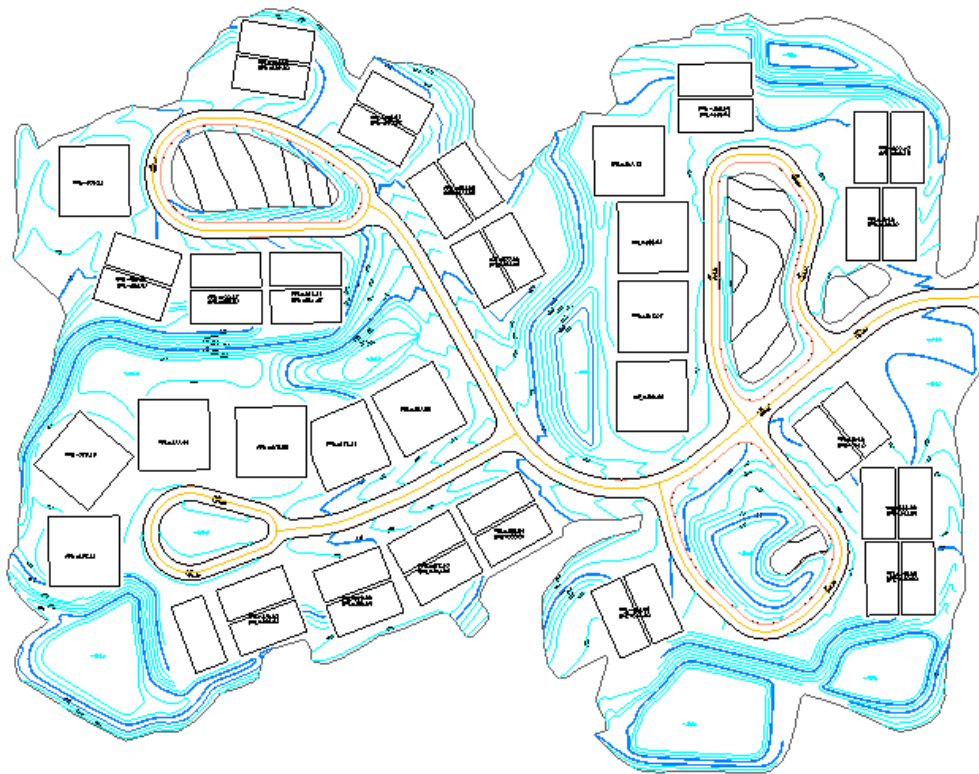
Stormwater Network Design

In this tutorial, we'll lay out the structure and pipes for the stormwater drainage and analyze the flow for a portion of a site. We'll use the tools to automatically calculate the drainage and runoff coefficients. These automated methods require setup of a surface and runoff regions. Alternatively, these tools can be skipped in which case the drainage areas and runoff coefficients for the inlets can be entered manually into the sewer network.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

Browse/navigate to the default folder location of C:\Carlson Projects and open the **Example3.dwg** file.

2. Activate the Hydrology menu via Settings – Carlson Menus – **Hydrology Menu**. Your drawing should resemble that shown below:

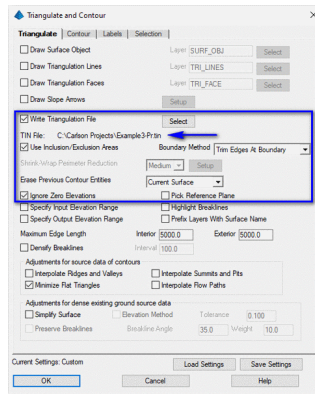


3. We'd like to create a surface model of proposed site conditions. The drawing entities for the design surface that we will use to model drainage have already been prepared. These entities consist of:
 - design contours
 - elevated pad perimeter polylines
 - spot elevations
 - 3D polylines for the road centerlines and face of curbs

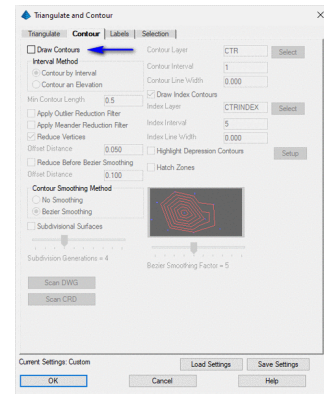
To model the drainage watershed surface, it is suggested that a **triangulated irregular network (TIN)** file be used (as opposed to a *grid* surface) so that the flow can properly follow the edges of the road. Issue the

Surface – Triangulate & Contour command to display a dialog box similar to that shown below:

Triangulate Tab



Contour Tab



Triangulate & Contour

NOTE: Use the **Select** button on the *Triangulate* to create an output TIN file as illustrated above. Once set, click the **OK** button and when prompted:

Select Inclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: pick the perimeter polyline

[FILter]/<Select entities>: press Enter

Select Exclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: press Enter

Select points and breaklines to Triangulate.

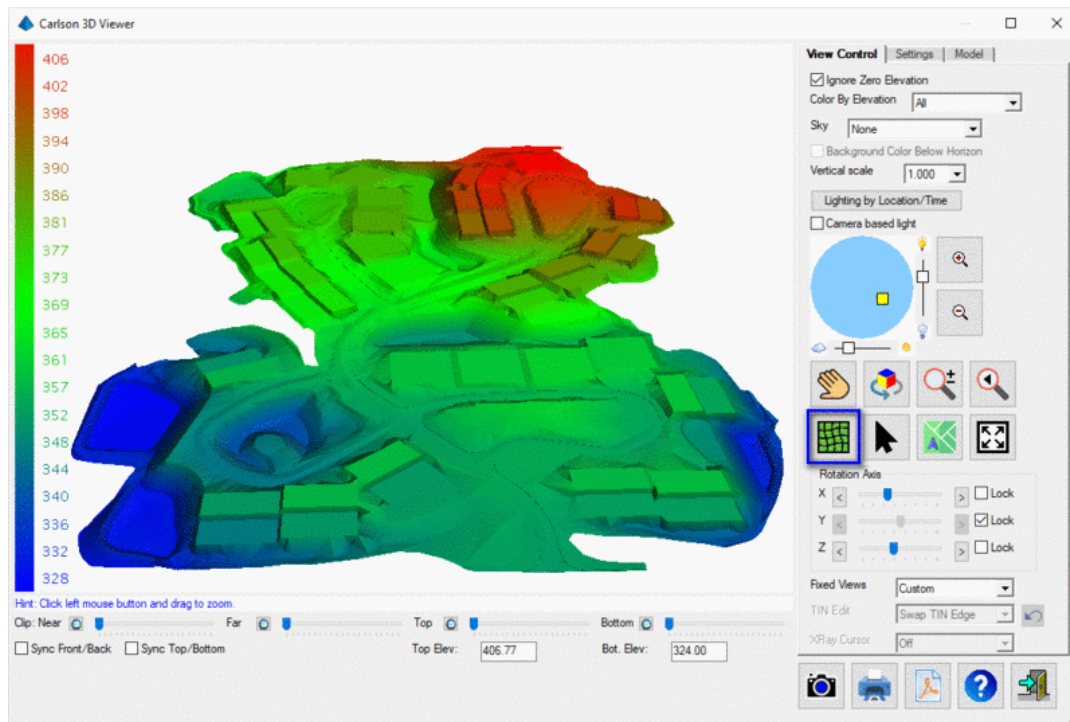
[FILter]/<Select entities>: type ALL and press Enter twice

The surface model *.TIN file is created.

4. *Optional:* Let's verify that the surface is good by checking for bad elevation data points and that the surface follows the data points. Issue the View – 3D View – Surface 3D Viewer command. When prompted:

Select Surface File (dialog): pick the TIN file created in the previous step and click **Open**

A display similar to that shown below appears:



NOTE: In the 3D Viewer dialog, move the pointer near the outer edge of the graphic and the cursor will change to an Z symbol which orients the view about the vertical axis. Moving the cursor to the interior of the graphic and the cursor will change to an X/Y symbol. Click and drag from the bottom upward to change the isometric viewing angle. You can also set the **Vertical Scale** to 2.0 and enable the **Color By Elevation** toggle for better viewing of the elevation difference and the *Surface Shading* toggle (highlighted above) can be used to shade/unshade the surface. Experiment with any desired remaining options.

The surface looks correct in the 3D Viewer. The site has a slope from the top road circle down towards the detention ponds at the bottom. When ready, click the **Exit** (Doorway) button.

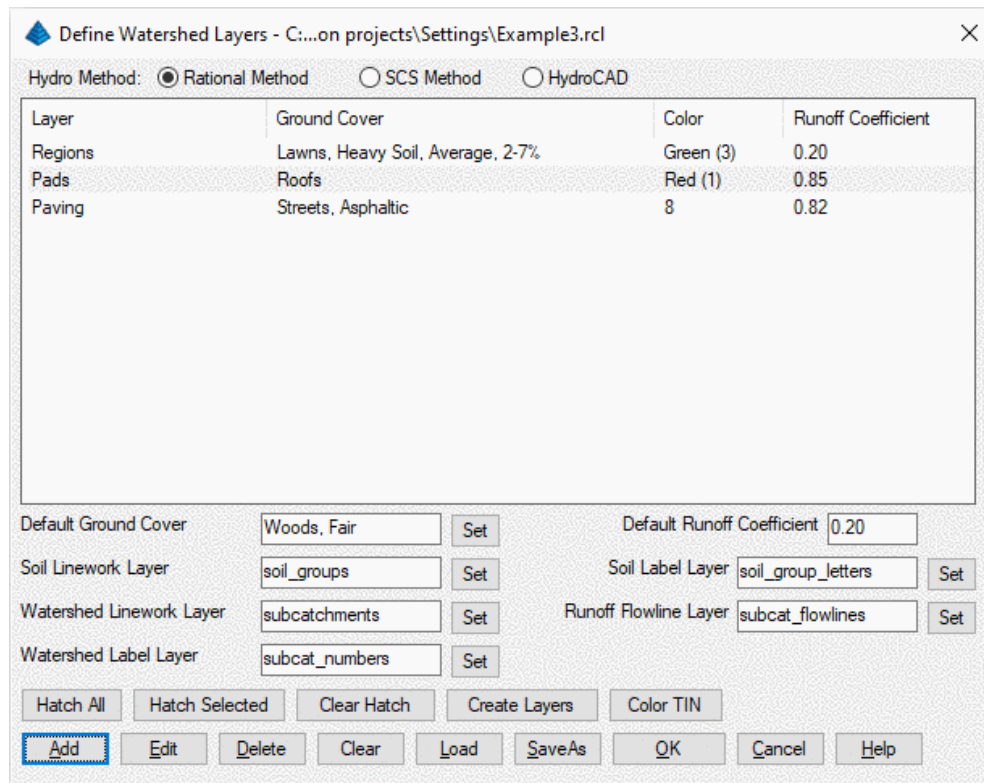
5. Step 4 - Runoff Coefficients. This step sets up layers that are assigned Rational Method runoff coefficients and applied to closed polylines on the specified layers. The runoff coefficients are the C-Factors in the Rational Method formula $q = CiA$, where:

- Q is flow
- C is a runoff coefficient
- i is rainfall intensity
- A is area

The runoff polyline areas use *region logic* where a polyline inside another on the same layer is used as an exclusion. A limitation is that polylines on the same layer must not intersect each other. For polylines on different layers, there can be polylines within other polylines and for any given point; the smallest enclosing polyline is used to determine the runoff coefficient. In this example, the:

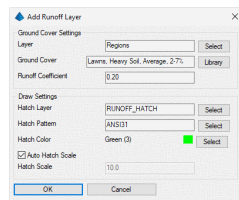
- site perimeter polyline is on the *Regions* layer
- building pads are on the *Pads* layer
- edge of pavement polylines are on the *Paving* layer

All these polylines are already closed polylines so we're ready to assign the runoff coefficients to the layers. Issue the Watershed – Define Watershed Layers command to display a dialog box similar to that shown below:

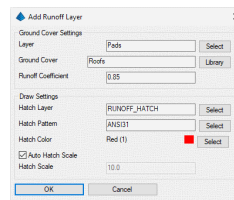


Begin with an empty dialog by clicking the **Clear** button to delete any existing layers from the table. Click the **Add** button and use the *Runoff Summary* information below for each of the specified layers:

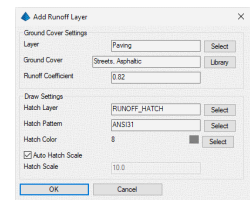
Regions Layer



Pads Layer



Paving Layer



Runoff Layers

For each entry:

- Click the **Select** button associated with *Layer* to set each layer
- Click the **Library** button associated with *Ground Cover* to set the associated ground cover
- Click the **Select** button associated with *Hatch Color* to set the color for each layer ground cover
- Click **OK** to save each item

Repeat the above sequence for each ground cover. Let's visually check the results. Click the **Hatch All** button that should generate an image similar to that shown below:



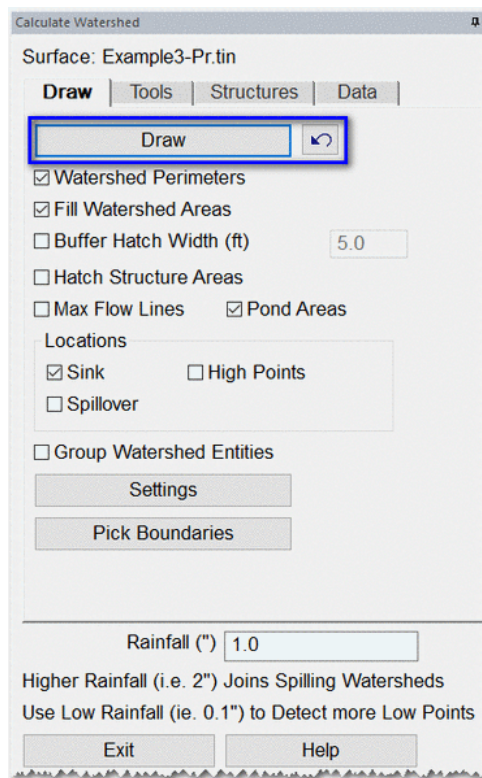
NOTE: The areas within the buildings are inside both the Regions and Pads polylines and the Pads govern because they are the smaller area. Likewise the road areas are governed by the Paving layer and road interior islands are not counted for Paving because the interior Paving polylines act as an exclusion perimeters. The rest of the area is set to the Regions layer.

We don't need to keep the runoff hatches. Remove them by clicking the **Clear Hatch** button and click **OK** when ready to dismiss the main dialog box (save the configuration using the file name shown in the top of the image cited earlier).

6. The next thing we'd like to do is analyze the watershed characteristics of the site. Issue the Watershed – Watershed Analysis command and when prompted:

Select Surface File (dialog): pick the TIN file created in the previous step and click **Open**

A "docked" dialog box similar to that appears:



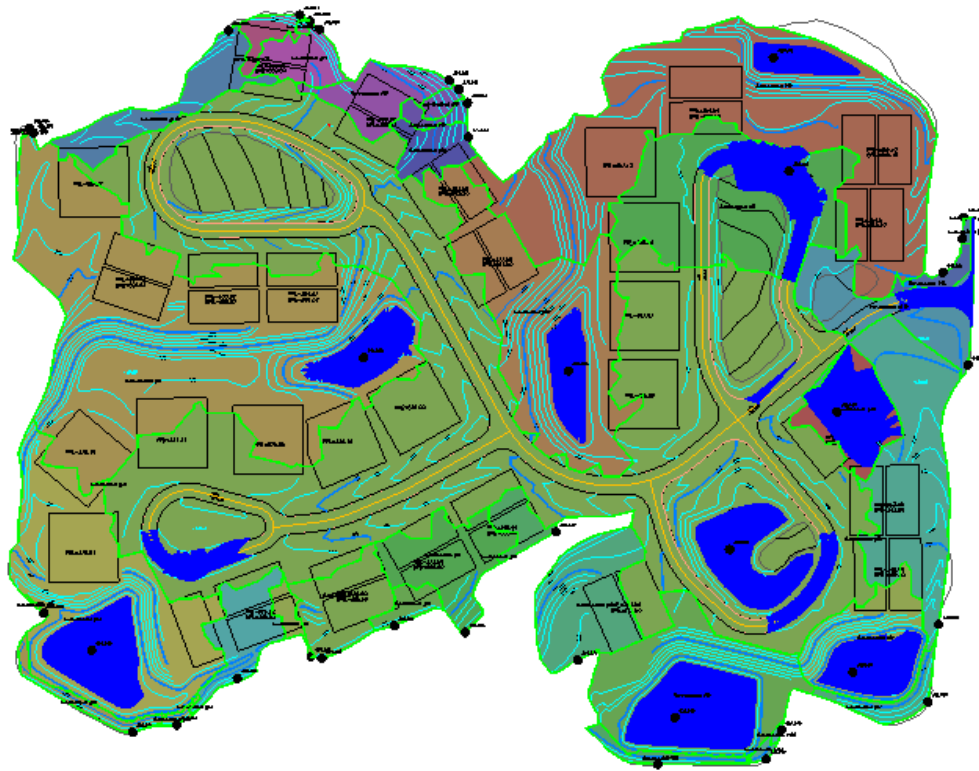
On the **Draw** tab, enable the toggles for:

- Watershed Perimeters
- Fill Watershed Areas
- Sink Locations (low points)
- Pond Areas

Before processing the watersheds, set the **Rainfall** to 1 inch.

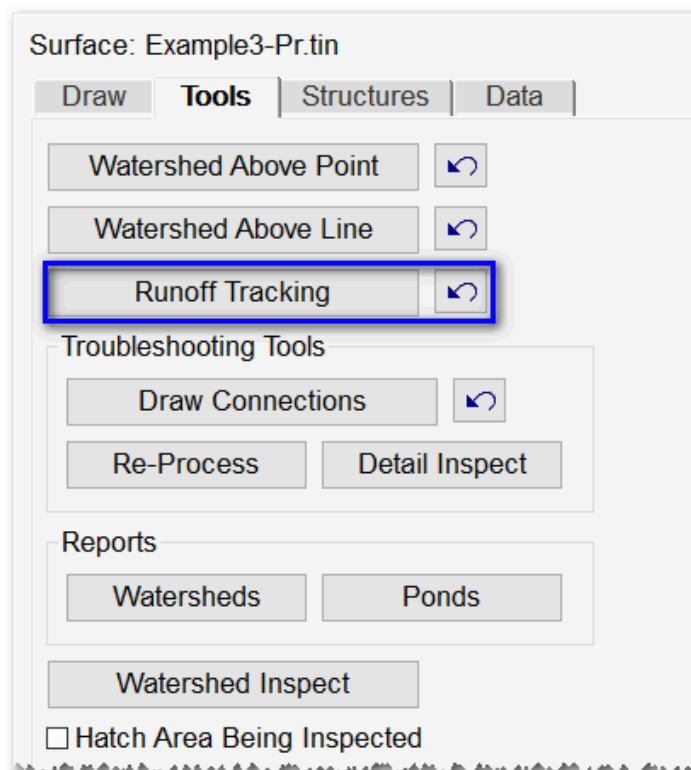
NOTE: The program uses the runoff volume calculated from the rainfall depth and drainage areas to figure when the runoff is enough for the flow to go through a low point. If the Rainfall Depth is set to zero, then the flow lines will stop at every low point or dimple in the surface. At this point, we have not defined our storm event to know the actual rainfall depth. If we did get the storm rainfall depth, we could enter it. For now we're just using *Watershed Analysis* to give a general idea of the watershed areas and runoff flow lines.

Click the **Draw** button. Each watershed area is drawn with a closed polyline and solid filled with different colors. Also, for each watershed the sink (lowest point) is drawn with a solid circle symbol. The areas covered by ponding are drawn as solid blue hatches. The depth and size of the pond areas is determined by the runoff volume. In many places, the pond areas are inside the detention pond structures. In a few places, the ponds are at low points in the road which indicate areas that we need to add storm sewer inlets:

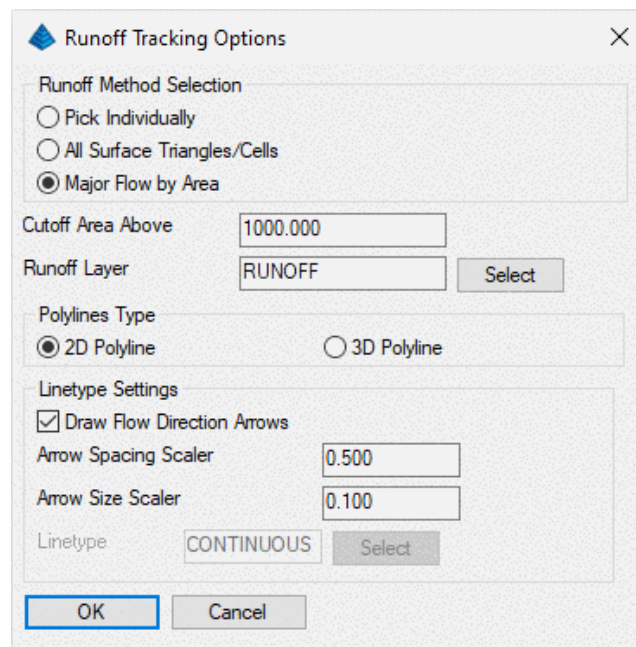


When you are done inspecting the watersheds, click the **Undo** arrow next to the *Draw* button (as illustrated earlier) to erase all the watershed entities.

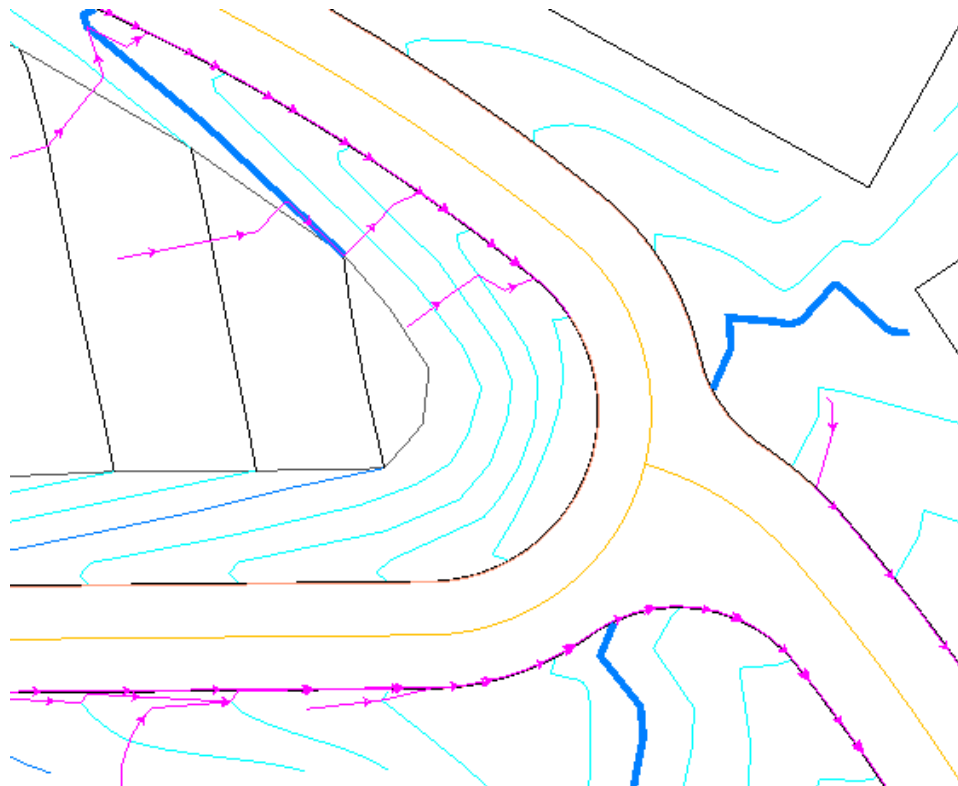
Next, activate the *Tools* tab as illustrated below:



Click the **Runoff Tracking** button to display a dialog box similar to that shown below:

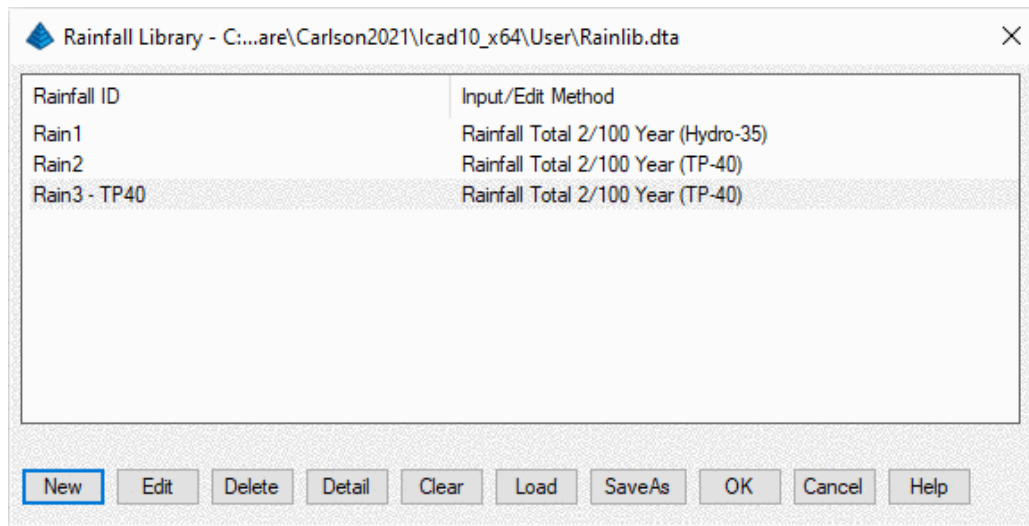


Set the values as shown above to draw flow lines only when the drainage area for the flow line is greater than the specified area and click **OK** when ready. The graphic below shows the flow lines coming off the road circle at the top of the site and following the curbs. We're going to leave the runoff flow lines on the drawing to help guide the placement of inlets.

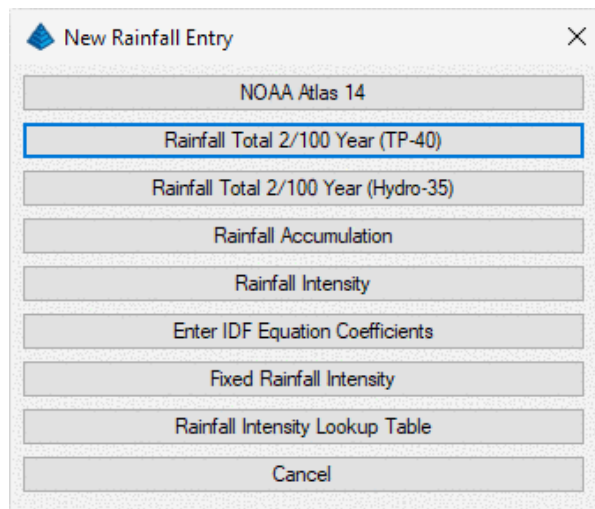


Click **Exit** to end Watershed Analysis.

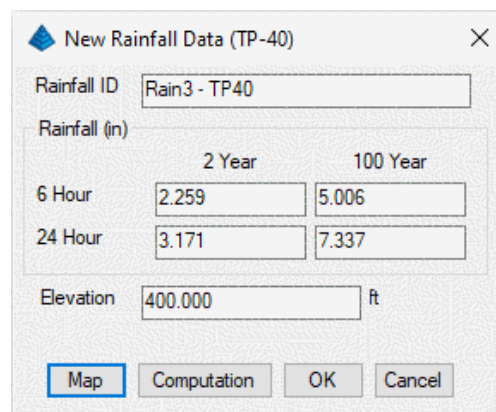
7. To setup the storm event to apply to this site, issue the Network – Sewer Network Libraries – Rational Rainfall Library command to display a dialog box similar to that shown below:



This command keeps a list of different storm events that you can use for different locations and requirements. Let's add a storm by clicking the **New** button. There are numerous types of rainfall definitions:



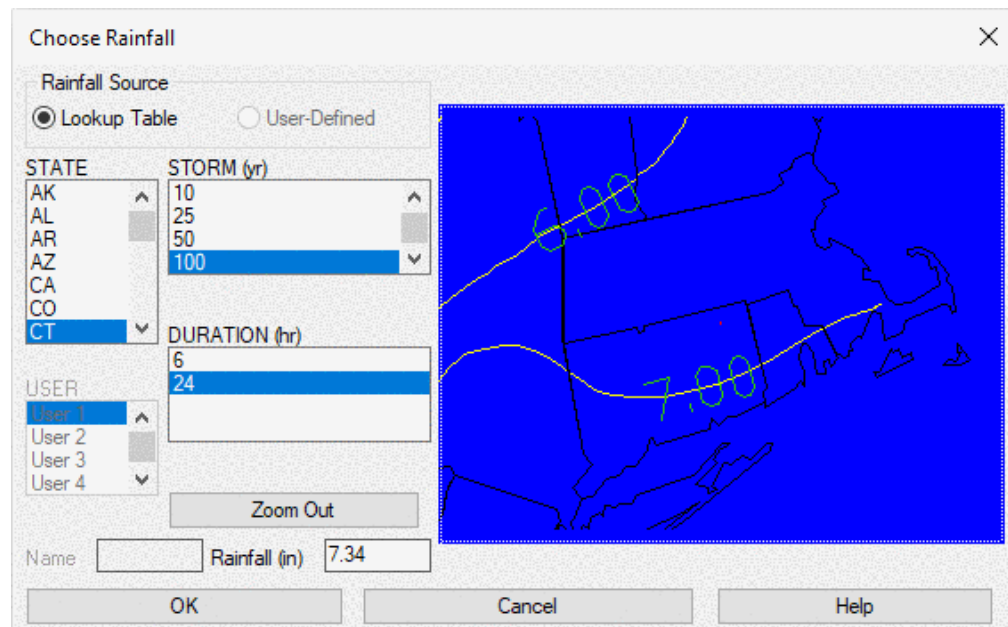
For this example, select the **Rainfall Total 2/100 Year (TP-40)** method to display a dialog box similar to that shown below:



Within the dialog box, supply the rainfall name as shown above and either:

- Use the **Map** button to show the TP-40 rainfall maps for the different storms. And if you pick on the

map display, the program will interpolate the rainfall from the maps. In this case, south central CT was picked:



- Supply the rainfall amounts for the 2 and 100 year storms for 6 and 24 hours, and the Average Elevation for the site.

Click **OK** on the *New Rainfall* dialog and click **OK** on the *Rainfall Library* dialog to save the changes.

8. *Optional*: In preparation to align the inlet symbols with the road centerlines, we prefer to create centerline files (.cl) for the roads. Activate the Civil menu via Settings – Carlson Menus – **Civil Menu**. Issue the Centerline – Polyline To Centerline File command and when prompted:

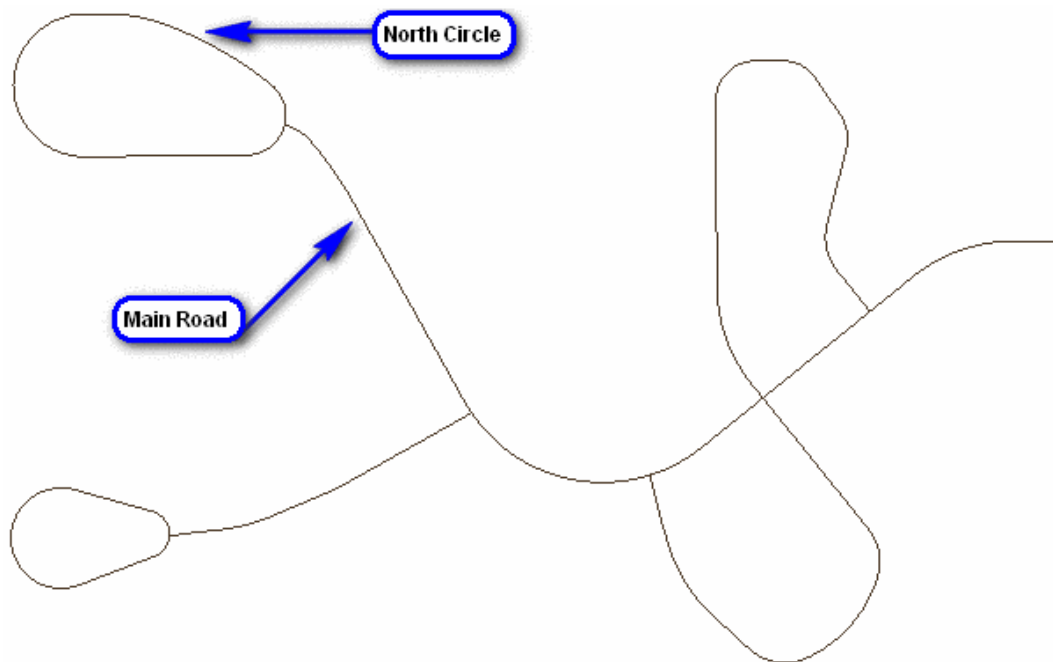
Centerline File to Write (dialog): supply a name of **North.cl** and click **Save**

Polyline should have been drawn in the direction of increasing stations.

Select polyline that represents centerline: pick the road centerline for the loop road at the top of the site as illustrated below

Centerline station [Reverse/Ending/<Beginning: 0+00>]: press Enter

Press ENTER to continue. press Enter



A centerline file is created. Immediately press Enter to re-run the **Polyline To Centerline File** command and when prompted:

Centerline File to Write (dialog): supply a name of **Main.cl** and click **Save**

Polyline should have been drawn in the direction of increasing stations.

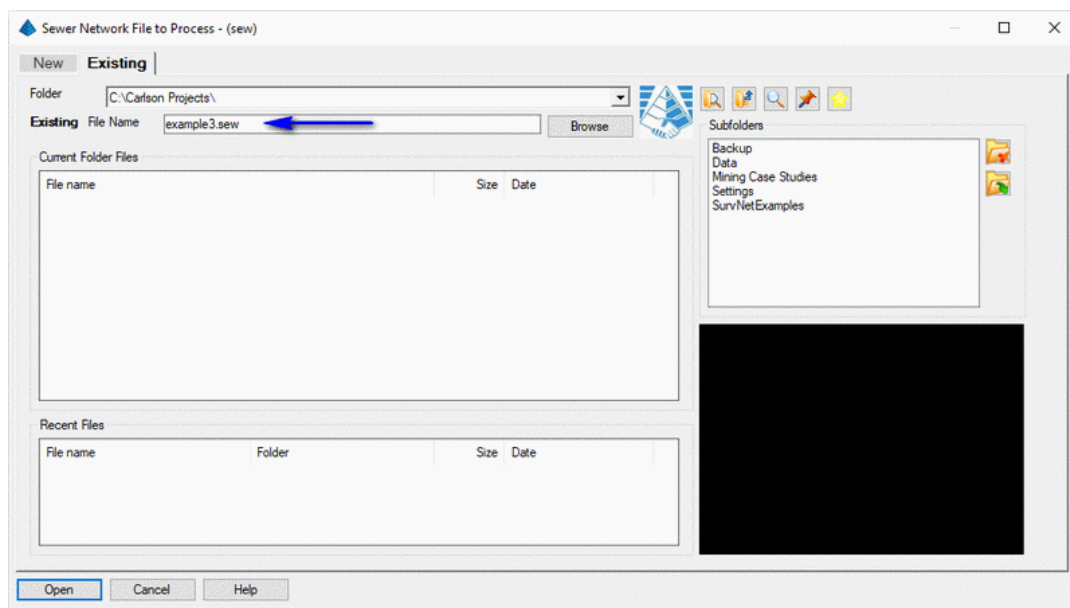
Select polyline that represents centerline: pick the road centerline for the main road as illustrated above

Centerline station [Reverse/Ending/<Beginning: 0+00>]: press Enter

Press ENTER to continue. press Enter

Activate the Hydrology menu via Settings – Carlson Menus – **Hydrology Menu**.

- Storm sewer network structures and pipes are stored in a sewer (.SEW) file. Once a .SEW file is set as current, the program will continue to automatically use that file. To start a new sewer network, issue the Network – Sewer Network Setup – Set Sewer File command to display a dialog similar to that shown below:

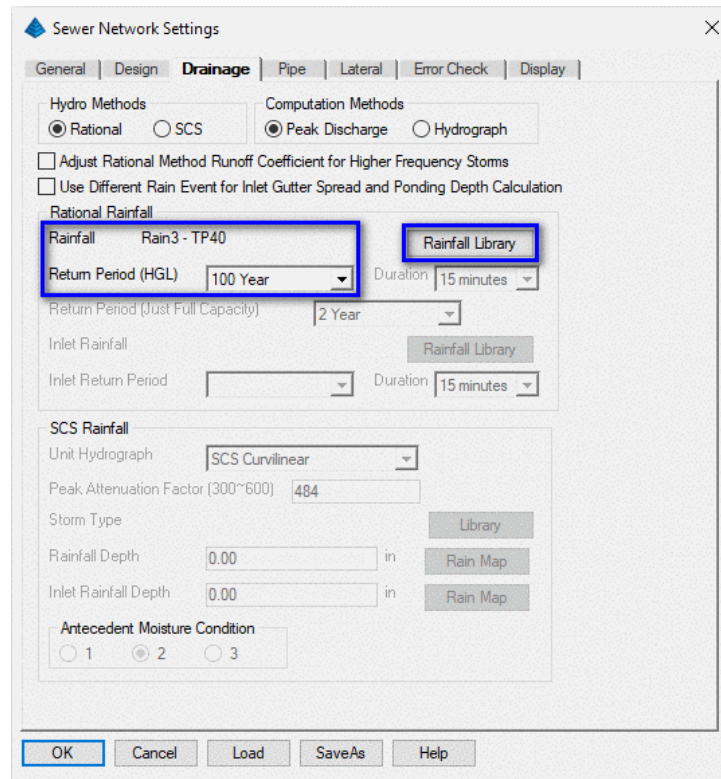


Set the filename as shown above and click **Open** when ready. The sewer network also works with a current

surface model that is used for the default rim elevations, reporting pipe cover and calculating inlet drainage areas. To set the current surface, issue the Network – Sewer Network Setup – Set Surface File and when prompted:

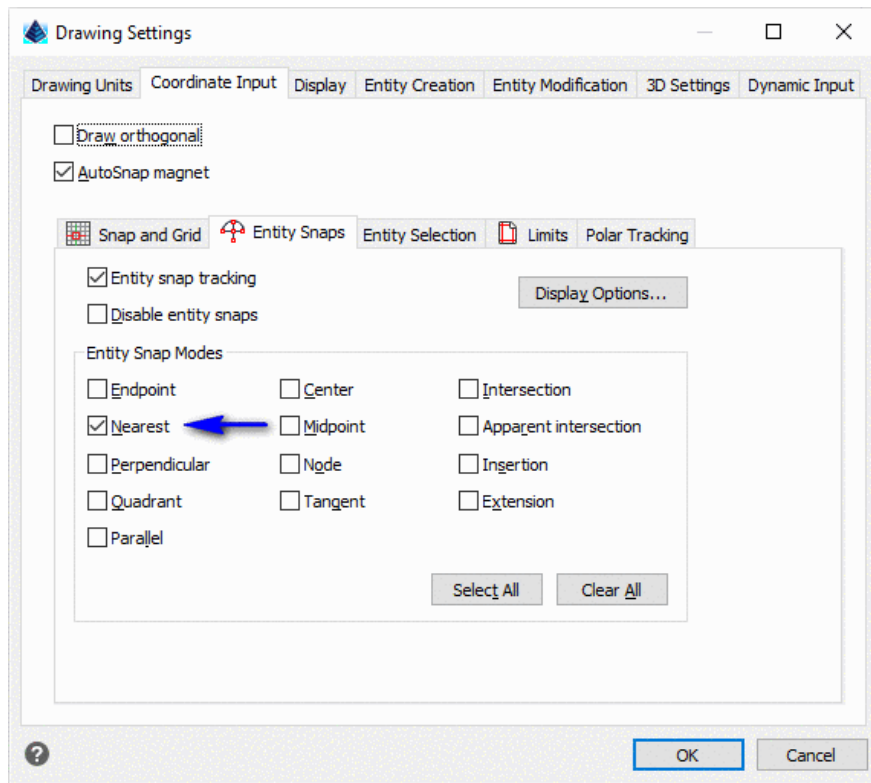
Surface Model to Reference (dialog): pick the TIN file created earlier and click **Open**

Additionally, issue the Network – Sewer Network Settings command to display a dialog box similar to that shown below. Click the *Drainage* and the click the **Library** to set the Rainfall to the data created earlier and specify a **Return Period** of 100 years as illustrated below:



Review the content of the other tabs and click the **OK** button when ready.

10. As we start into the creation of the storm sewer network, let's set a running entity/object snap (esnap/osnap) to nearest (nea) to use for locating the inlets along the curb polylines. Issue the Settings – **Object Snap** and turn on only the *Nearest* snap mode as illustrated below (click **OK** when ready):

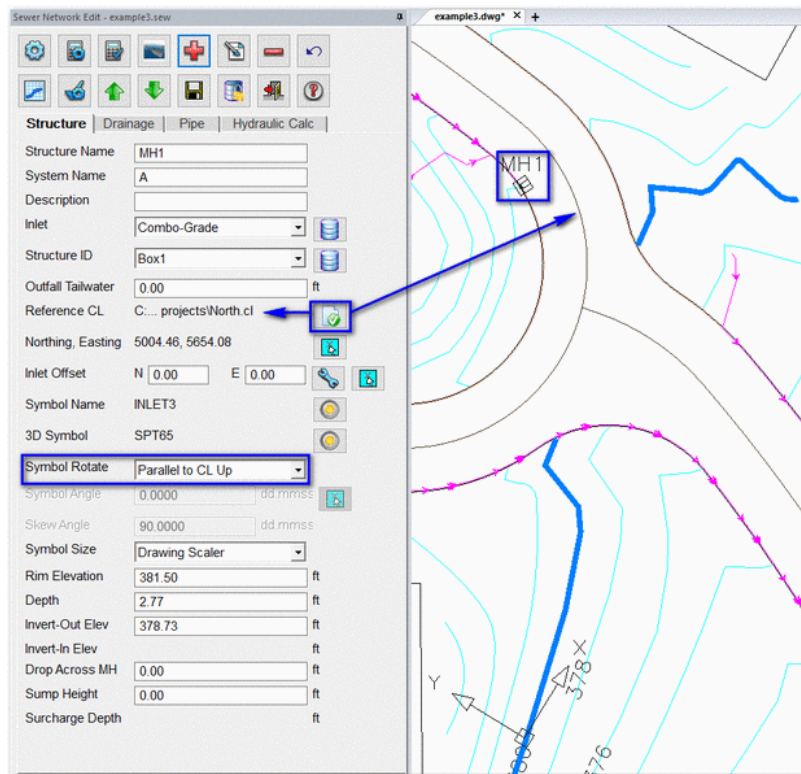


Now we're ready to lay out the inlets and pipes. Let's work on the drainage for the roads of the North loop and the Main road and run this flow to an outlet in the central pond. Issue the Network – Create Sewer Structure command and when prompted:

Locate by pick point, point number or station-offset [<Pick>/Number/CL/Curb**]?** press Enter

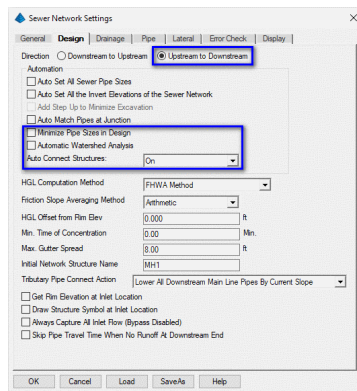
Pick structure location: zoom in very close so that you can see the curb line and pick a location on the inner curb line

NOTE: Be sure to pick the end of the flow line at the inside curb polyline and not the top of curb. Otherwise, the routine to find the drainage area from the surface model will not capture flow along the curb:

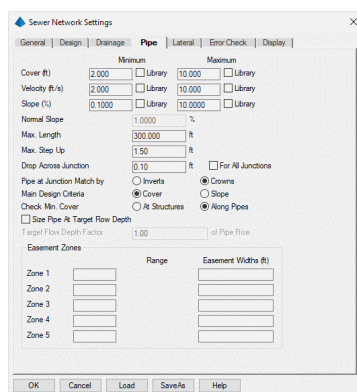


After this first inlet location is picked, the *Sewer Network* "docked" dialog is placed on the left side of the drawing as illustrated above. Click the **Select** (Paper) button next to the **Reference CL** label, select **By Centerline File** and select **North.cl** for the centerline. This centerline can/will be used to align the inlet symbol. In the **Symbol Rotate** field, select **Parallel To CL Up**. Let's establish some desired settings. Click the **Settings** (Gear) button and on the various tabs shown below, set the highlighted values:

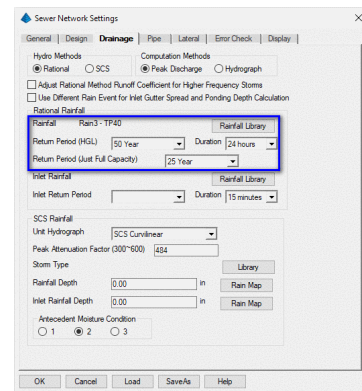
Design Tab



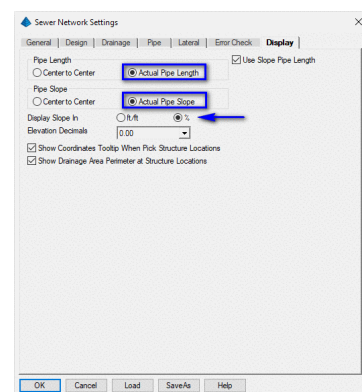
Pipe Tab



Drainage Tab



Display Tab



Storm Sewer Settings

Click the **OK** button when ready.

11. Now let's work on the *Structure* settings. Keeping the docked dialog box open, click the **Library** (Cylinder) button to the far right of the *Structure ID* field to display the list of structures as otherwise defined through the Network – Sewer Network Libraries – Sewer Structure Library command. For this example, use **MH3**. To check the dimensions for this structure, pick the **Edit** button to display the dialog box below:

Circular Structure

Structure ID:

Structure Desc:

Taper Format:
 Symmetric Taper Upstream Taper Downstream

Top Diameter: ft
Fixed Taper Height: ft
Taper Offset: ft
Bottom Diameter: ft

Custom Properties

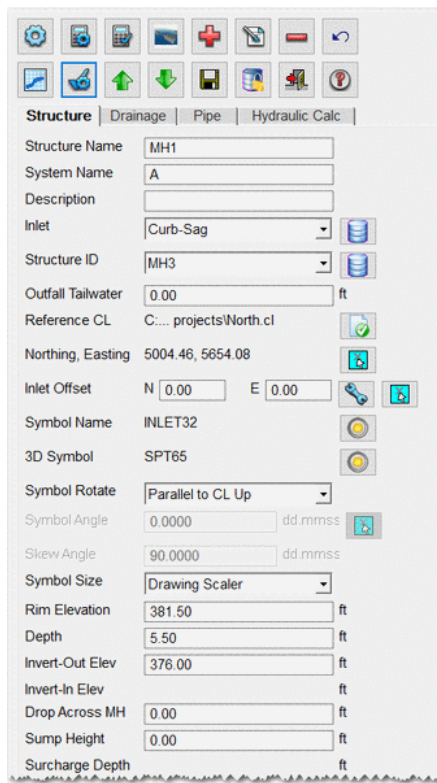
ID	Name	Value
----	------	-------

Buttons: Draw, **OK**, Cancel, Help

These dimensions are used for hydraulic calculations as well as drawing the structure in the profile and 3D views. Click **OK** from the *Edit* dialog, select/highlight **MH3** from the library list and pick **OK**.

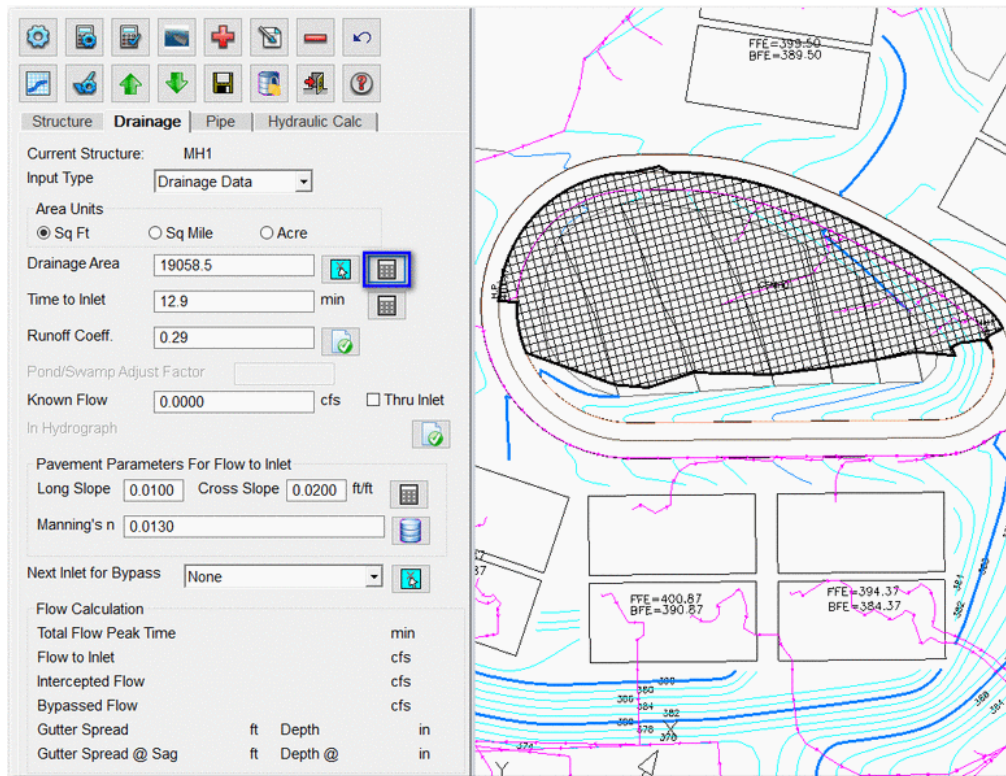
Next, pick the **Library** button next to the *Inlet* field. This function shows the inlets as otherwise defined through the Network – Sewer Network Libraries – Inlet Library. Like the **Structure Library**, you can add to and edit the Inlet Library. For this inlet, choose the **Curb-Sag** from the list and click **OK**.

The last change for the *Structure* tab is to set the **Depth** to 5.50. After making these changes, your dialog should match the settings as shown below:



Pick the **Apply Pipe Rules** (Checkmark with Gear) button to save the changes and you should see the plan view symbol for the inlet change to a grate symbol.

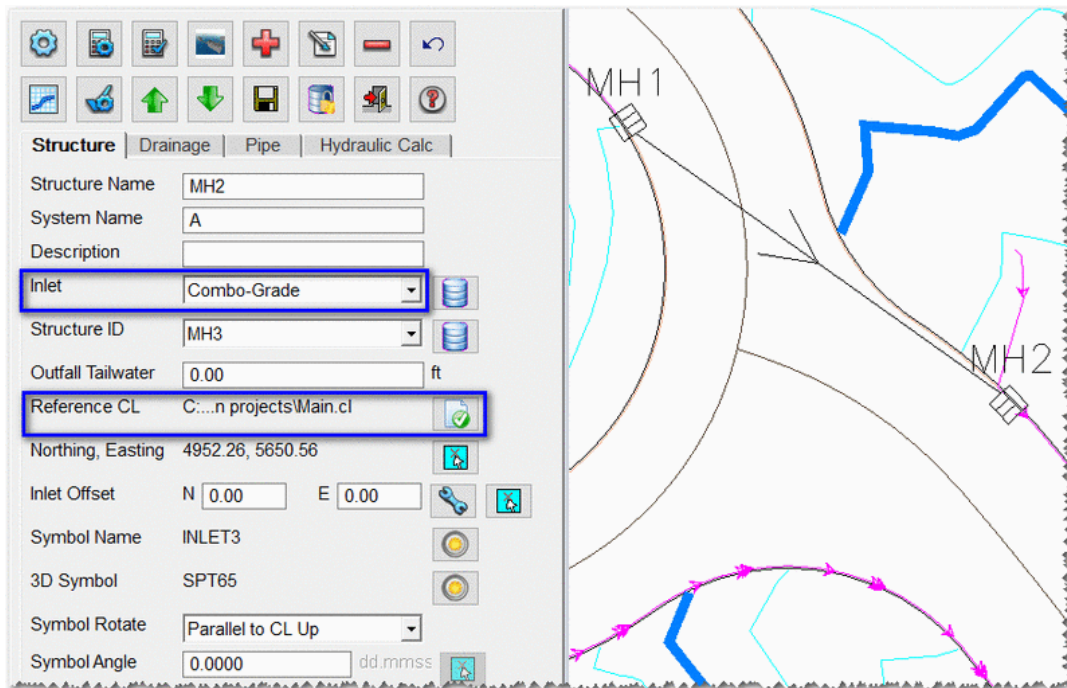
12. Now move onto the *Drainage* tab. Here the drainage area, time of concentration, runoff coefficient and pavement parameters are set (among other things) for the inlet. You can manually enter them in or have the program calculate these values. With the *Pick* button, you can select a drainage polyline perimeter and the program will calculate the area and the weighted average runoff coefficient from the runoff layers (if defined). In this example, use the **Calc** (Calculator) button to calculate all the parameters from the surface model. The first time that **Calc** is called, the program takes time to calculate the triangulation flows. Then the values are filled in and the drainage area is hatched in plan view as illustrated below:



- The Time To Inlet comes from the Max Flow Line within the drainage area and accounts for the surface slopes along the path.
- The Runoff Coefficient is calculated as the weighted average of the runoff sub-areas within the drainage using the runoff layers that we defined in the **Define Runoff Layers** command.
- In the Pavement Parameters section, the Calc button will calculate the Pavement slopes from the surface aligned by the Reference CL at the inlet location.

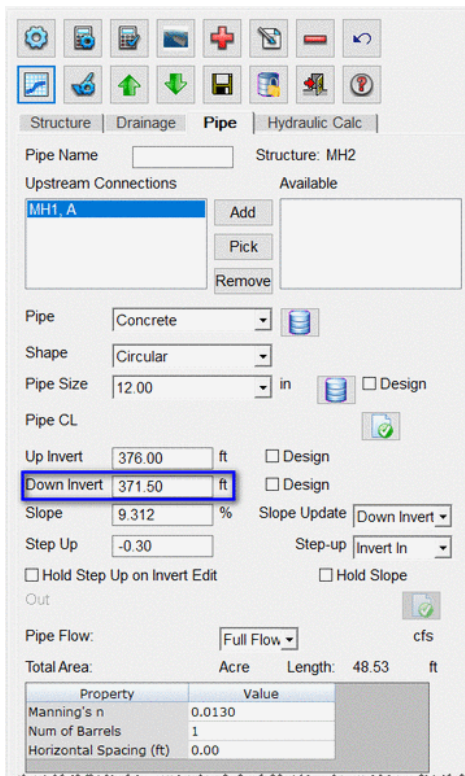
Notice how the drainage area for MH1 starts from the road high point and follows the crown of the road to the inlet. We're done for now with this first inlet.

13. To add the next inlet, pick the **Add** (Plus) button from the *Structure Actions* row. Pick a position along the right side curb polyline of the **Main** road near the intersection as shown below (MH2). Again, you may need to zoom in to be sure to snap onto the curb polyline:



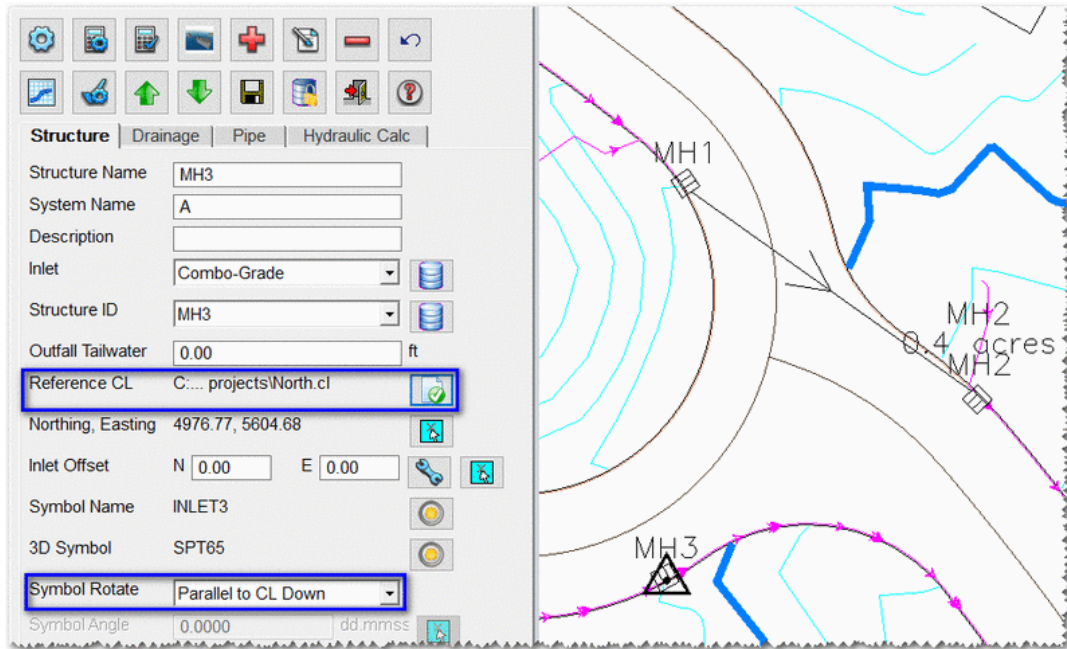
Go to the *Structure* tab for MH2 and change the **Inlet** type to **Combo-Grade** and the **Reference CL** to **Main.cl**. Then go to the *Drainage* tab and click the **Calc** to fill out all the drainage values.

Next, select the *Pipe* tab. The program lists all the used and available structures for a pipe connection to the current structure. By default, a connection is made to the nearest structure as long as it's within the *Maximum Pipe Length* as defined under **Settings**. Set the **Down Invert** to 371.5 (as illustrated below) and switch back to the *Structure* tab and set the **Invert-Out** as 371.5:



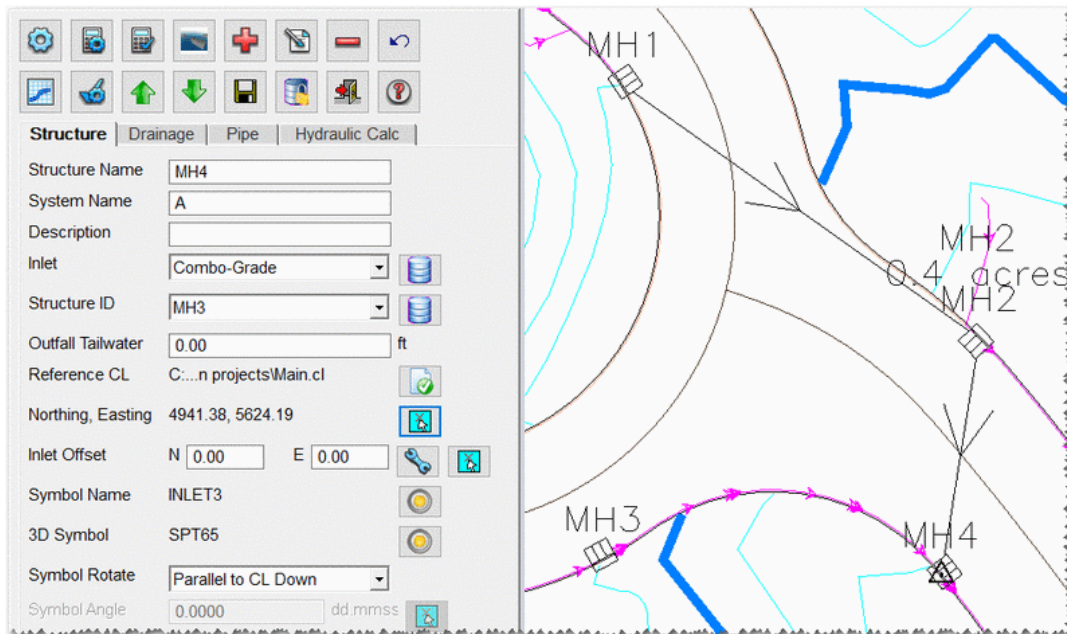
To add the next inlet, click the **Add** button again. Then pick a position along the inside North loop curb

polyline to the left of the intersection as shown here (MH3):



A pipe is automatically connected to the nearest structure (MH1). But for this network, we don't want MH3 to connect to MH1. Instead, we're going to start a new branch with MH3. So go to the *Pipe* tab, highlight the **Upstream Connection** of MH1 and click the **Remove** button. On the *Structure* tab for MH3, set the **Depth** to 7.0 and on the *Drainage* tab, click **Calc**.

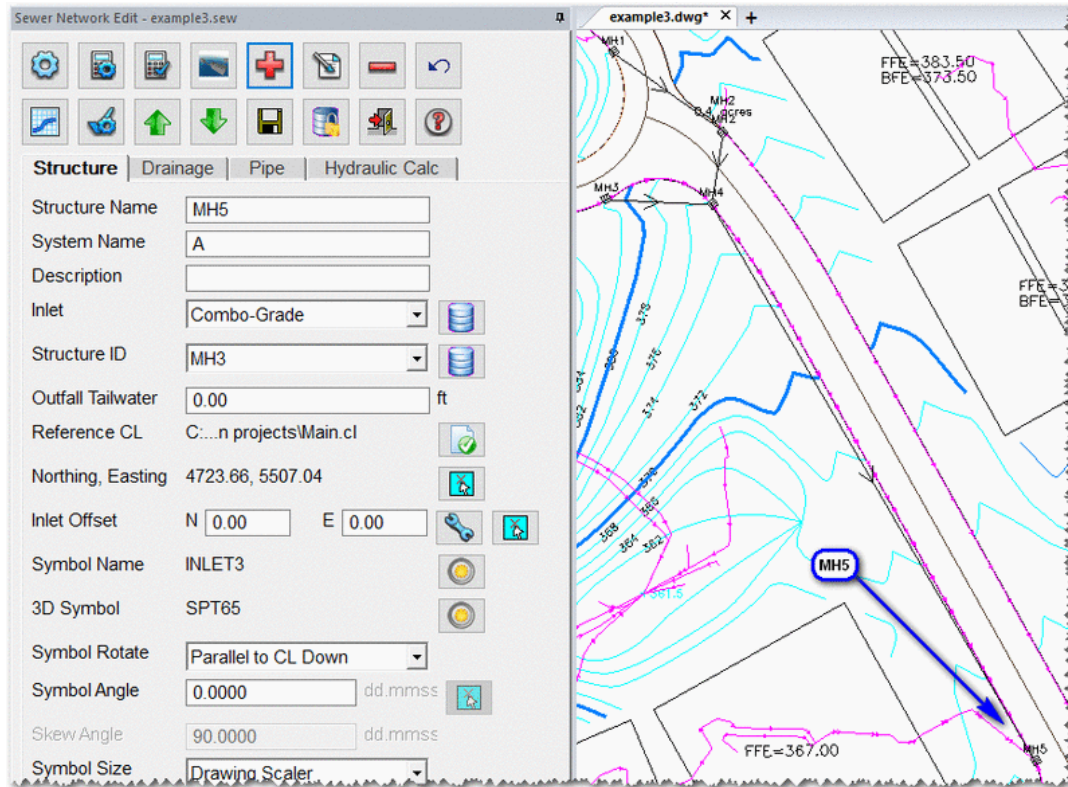
14. Now we're ready for the next inlet. Click the **Add** button from the *Structure Actions* row and screen pick the position along the main road across from MH2 as shown here (MH4):



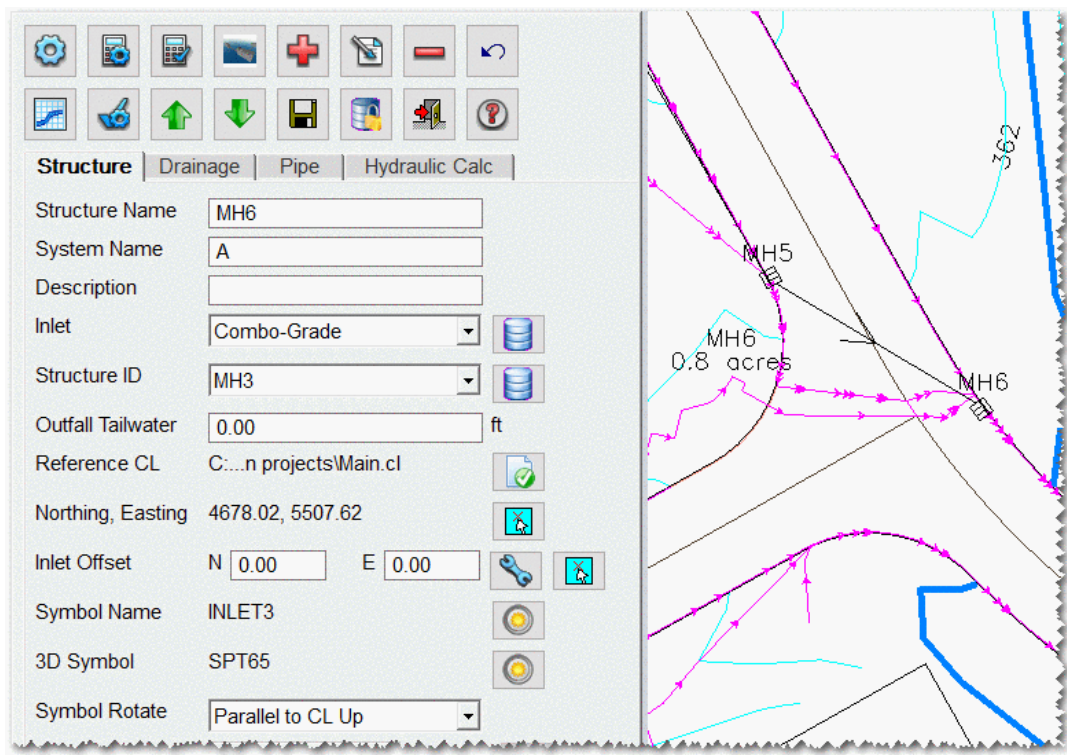
This inlet is on the other side of the road and the symbol is rotated the wrong way. To fix this, go to the *Structure* tab and change the **Symbol Rotate** to **Parallel To CL Down**. Also, change the **Invert-Out** to 371.0 and pick **Apply** to update the drawing. Next go to the *Drainage* tab and pick the **Calc** button. Again the pipe connection defaulted to the nearest structure of MH2. We want the connections from MH3 and MH2 to

go to MH4. Under the *Pipe* tab, highlight the **Available** connection for MH3 and click **Add**. For the pipe parameters, change the Down Invert to 371.1. Then go back to the Structure tab and set the Invert-Out to 371.0.

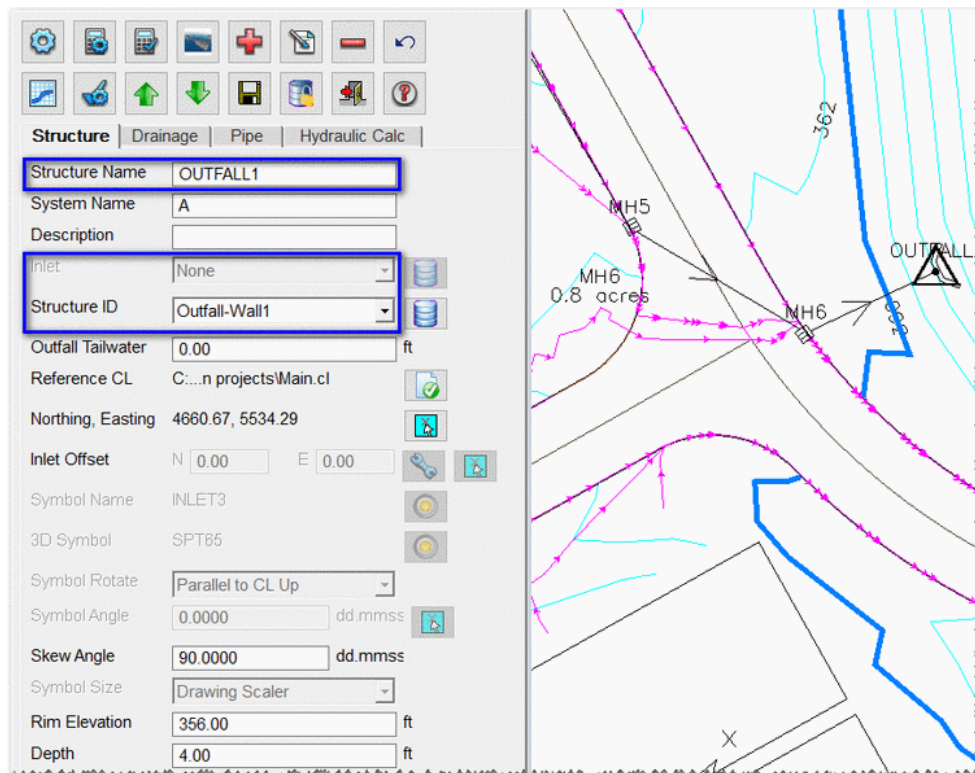
15. Now let's add the next inlet. Click the **Add** (Plus) button from the *Structure Actions* row and pick a position further down the Main road from MH4 as shown here (MH5):



Under the *Drainage* tab, pick **Calc**. We have one more inlet to add. Click the **Add** button from the *Structure Actions* row and screen pick along the curb on the other side of the Main road from MH5. See MH6 in the graphic below:



Under the *Structure* tab, set the **Symbol Rotate** to **Parallel To CL Up** to flip the symbol around. Under the *Drainage* tab, pick **Calc**. For the last structure, click the **Add** button from the *Structure Actions* row and pick on the 356 contour in the pond to the right of MH6:

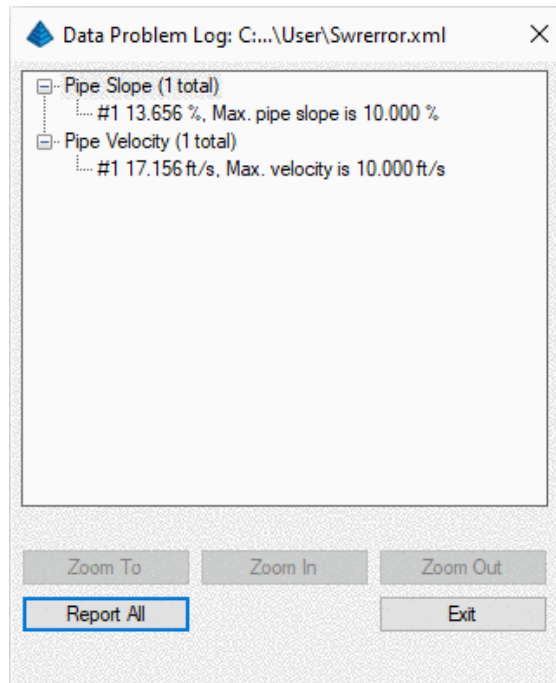


In the *Structure* tab, make the highlighted changes and click **Apply**.

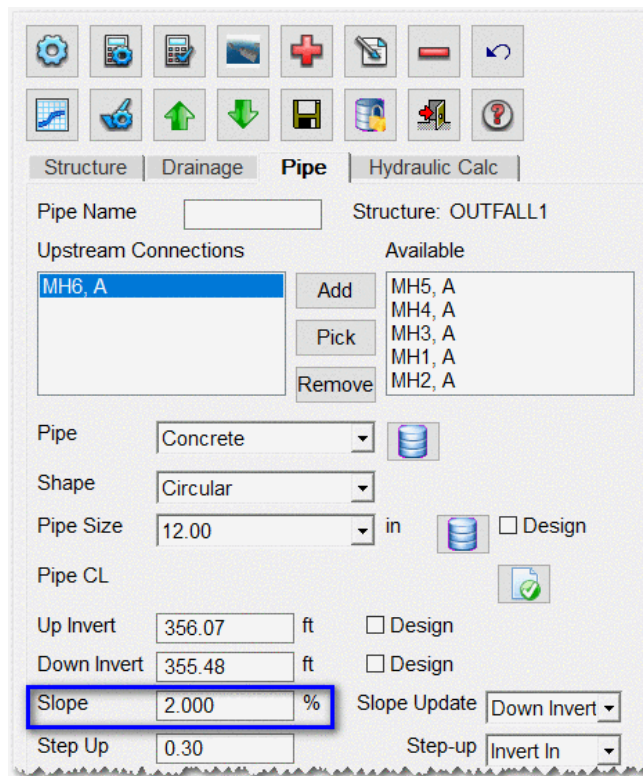
NOTE: You may get a *Pipe Rules* alert indicating potential problems with the network. If this is the case, click the **Update All** button.

The initial sewer network layout is done. Click the Save (Floppy Disc) button.

- The network flow can be analyzed. Click the **Analyze** (Calculator with Checkmark) button which runs the selected storm event through the system. If any of our design parameters are exceeded as specified under Settings, the program displays a report. Here's the report for our first analysis (your results may vary):

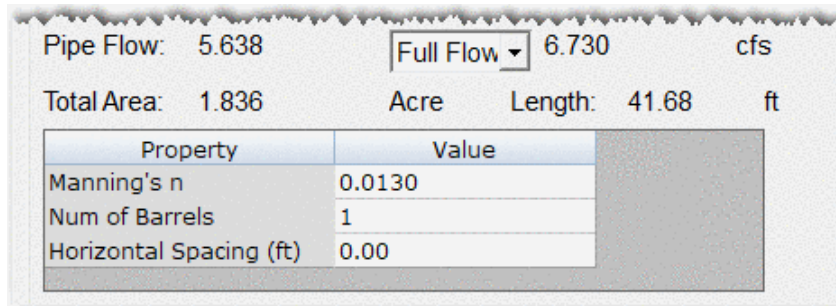


Let's take care of Warning #1 for the pipe slope. Use the **Up/Down Arrows** to navigate or click the **Edit** (Paper with Pencil) button on the *Structure Actions* row and pick the Outfall label or symbol to edit the **Outfall** structure. Set a target slope for the MH6 to Outfall as suggested below:

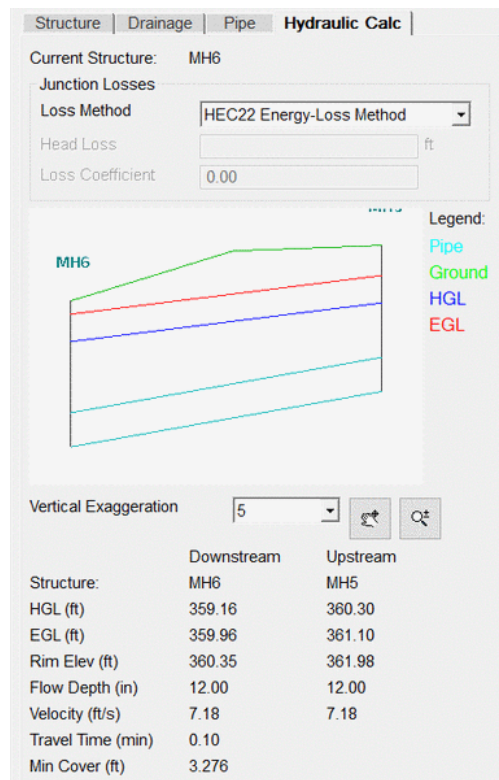


Click the **Analyze** button again. The warning report should only have the flow velocity warning(s), if any. Exit the report if it comes up. The flow velocity warnings can be resolved by resizing pipes and setting inverts which we will do later. Now let's review the flow results from the dialog. From the Outfall structure, click the **Up** button to move up to MH6. From the *Drainage* tab, the flow results are displayed in the **Flow Calculation** section. The **Flow To Inlet** is calculated by the Rational Method using the *Drainage Area*, *Time Of Concentration* and *Runoff Coefficient* for this inlet. The *Intercepted Flow*, *Bypassed Flow*, *Gutter Spread* and *Gutter Depth* are calculated from the inlet dimensions using formulas from HEC-22. These values can be used to determine whether you have the right inlet structure to capture the flow.

Switch to the *Pipe* tab. The *Flow*, *Area* and *Length* are displayed for the pipe connection currently highlighted from the **Upstream Connections** list. The **Total Flow** is the accumulated flow for the current pipe and the **Total Area** reports the accumulated drainage areas for all the inlets coming into this pipe:



Switch to the *Hydraulic Calc* tab which shows a graphic of the pipe structure, ground surface, hydraulic grade line (HGL) and energy grade line (EGL), along with the **HGL** and **EGL** elevations, **Flow Depth** and **Flow Velocity** at the pipe upstream and downstream connections. The **Min Cover** (calculated using the surface model to the top of the pipe) is also displayed:



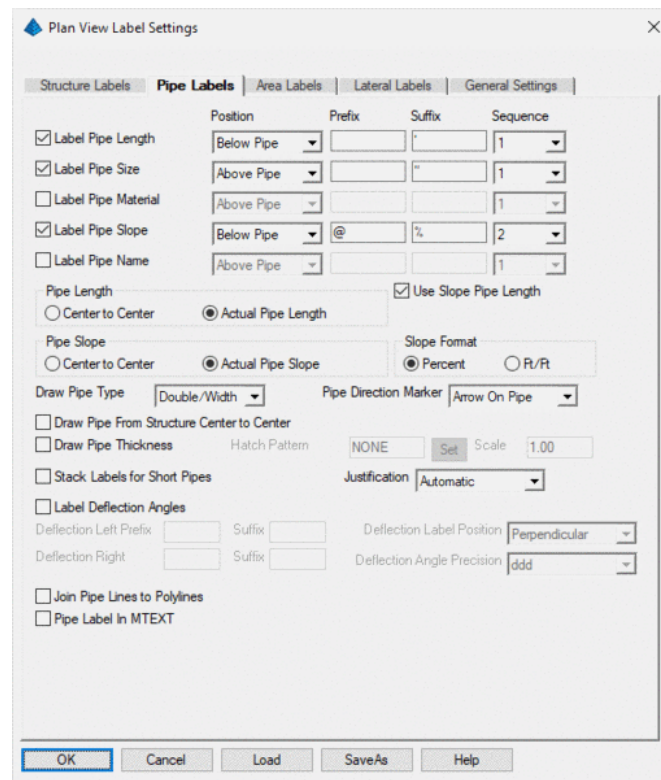
You can go to other structures to check the flow values for them or use the Network – Report Sewer Network

command to review the values in a report view. For our storm event, many of the pipes can be resized. To resize the pipes, you can go to the *Pipe* tab and change the **Pipe Size**. The program can also automatically size the pipes based on the flow:

- (a) To size specific pipes, go to the *Pipe* tab and pick on the **Design** toggle next to the **Pipe Size** field. Then click the **Design** (Calculator with Gear) button in the *Structure Actions* row and the program will run a flow analysis and set the pipe size for these pipes marked for *Design*.
- (b) To have the program size all the pipes, click the **Settings** button and select the *Design* tab and enable the **Auto Set All Pipe Sizes** option (click **OK**) and then click the **Design** button.

For this example, let's have the program assign all the pipe sizes. Issue the sequence described above. The pipes are resized to match the flow. Pick on the *Pipe* and *Hydraulic Calcs* tabs to see the changes. Pipes that can change would change and the different pipe sizes that the program uses are defined in the Network – Sewer Network Libraries – Pipe Size Library command. After the pipe sizing, there are likely a few lingering flow velocity warnings. Experiment with the flow velocity by adjusting invert elevations of the affected pipe(s) in your network. Click the **Edit** button in the *Structure Action* row and pick the structure whose invert elevations need to be modified. Now run **Analyze** function which should now complete without any warnings. Click the **Apply** button to save the results and then click the **Exit** (Doorway) button.

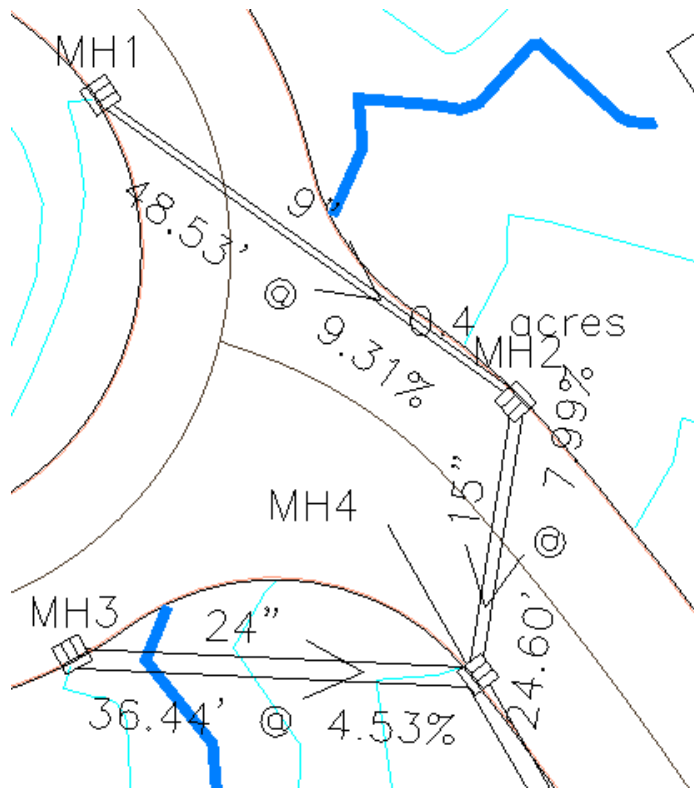
17. At this point, the sewer network labels are only showing the inlet name and pipe direction arrow. To change the label format, issue the Network – Sewer Network Setup – Plan View Label Settings command and click on the **Pipe Labels** tab to display a dialog box similar to that shown below:



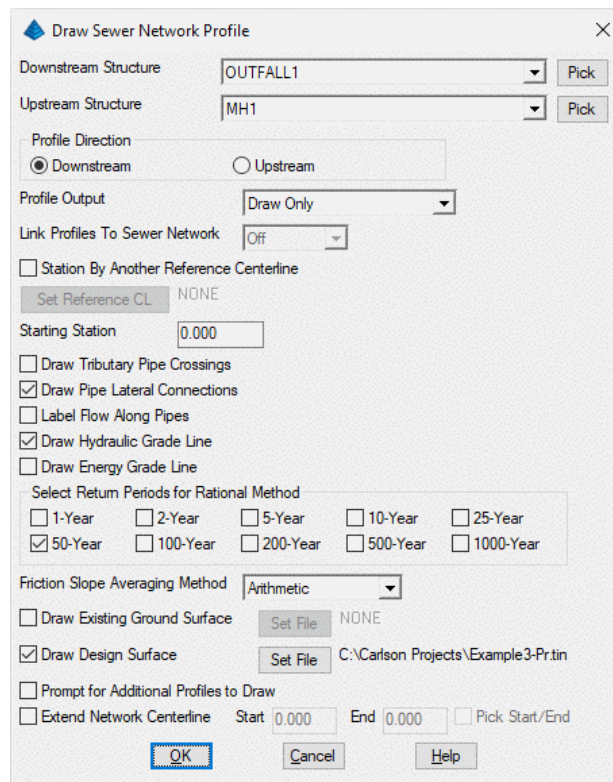
Set the values as illustrated above and click the **OK** button. The labels should update in the drawing.

NOTE: The sewer labels are linked to the sewer network definition so that any change to the sewer network updates the labels. If you want to explicitly update the sewer labels, issue the Network – Draw Sewer Network – Plan View command.

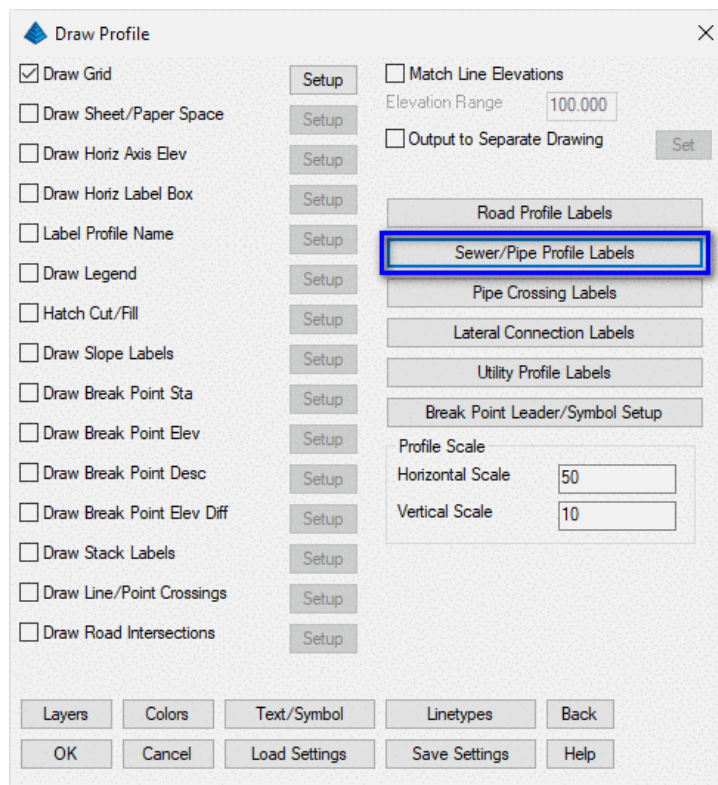
When sewer labels overlap other drawing entities, you can use the Network – Sewer Labels – Move Plan View Label command. Let's run this command and move the **MH4** label to the left of the inlet as illustrated in the example below:



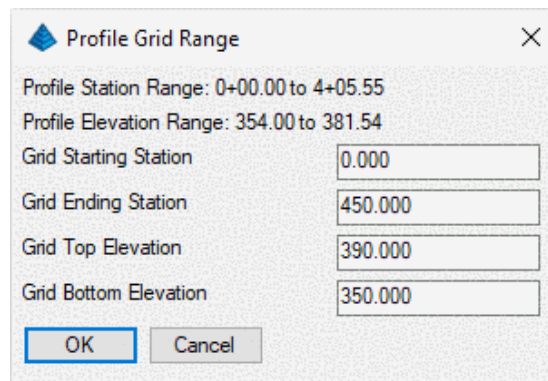
- To create a profile for the sewer network, issue the Network – Draw Sewer Network – Profile command to display a dialog box similar to that shown below:



Set the values as shown above and click **OK** when ready. A dialog box similar to that shown below appears:



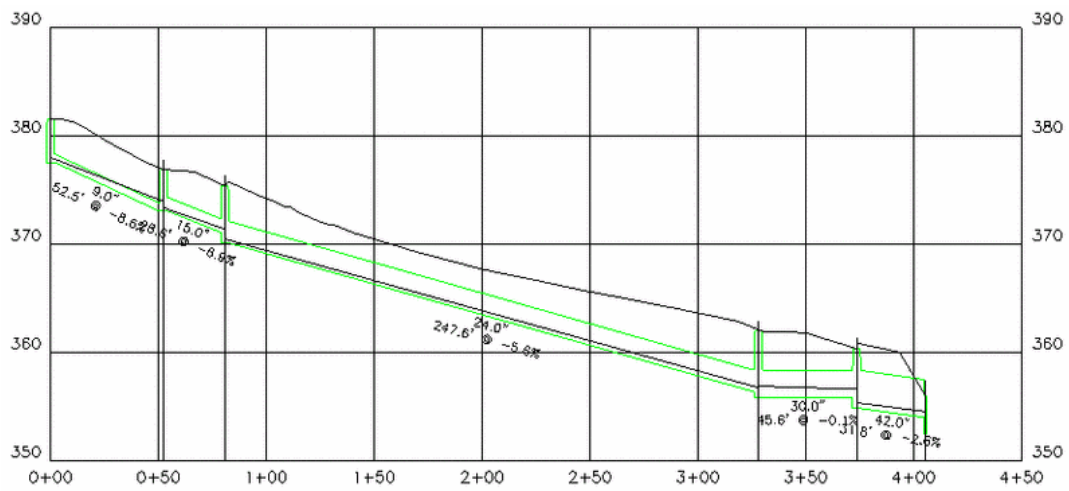
NOTE: Explore the **Sewer/Pipe Profile Labels** button to further detail the content to be shown. Set the values as shown above and click **OK** when ready. A dialog box similar to that shown below appears:



The elevations default to fit the profile, so click **OK**. When prompted:

Pick Starting Point for Grid <0.00,0.00>: pick the lower left profile grid point

Your profile may appear similar to that shown below:



19. Next, let's draw the sewer network in 3D. Issue the Network – Draw Sewer Network – 3D Faces command and when prompted:

Draw Sewer Network 3D Faces (Dialog): click OK

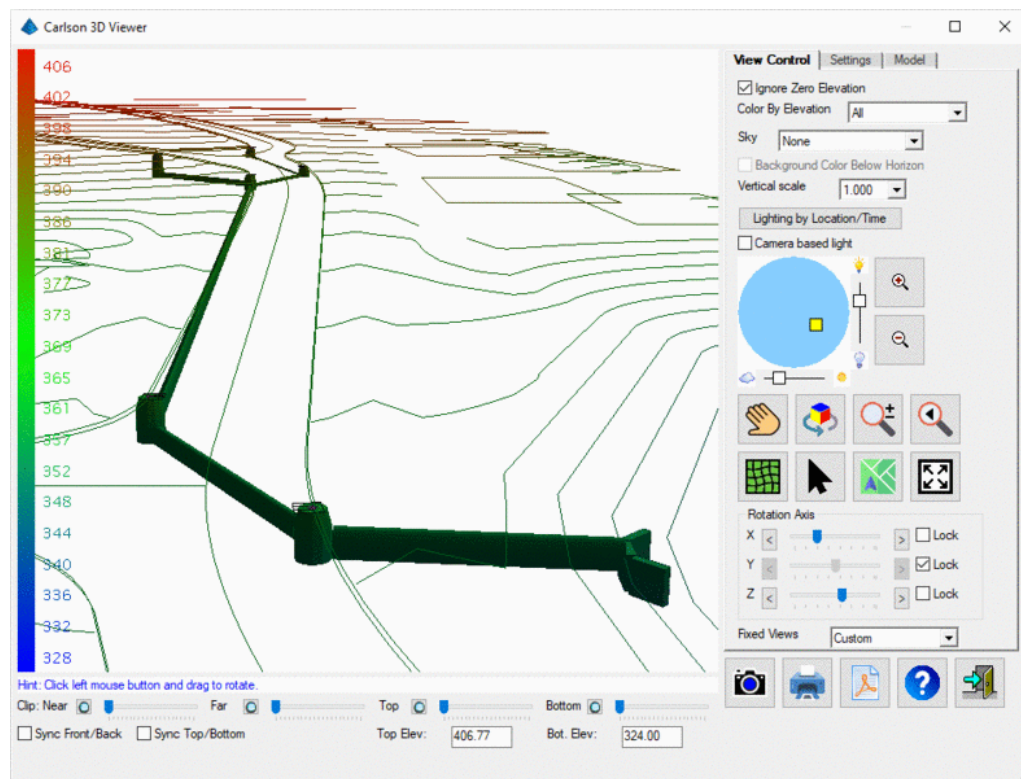
To view the 3D Faces, issue the View – 3D Viewer Window command and when prompted:

Select entities for the scene.

[FILter]/<Select entities>: type ALL and press Enter

[FILter]/<Select entities>: press Enter

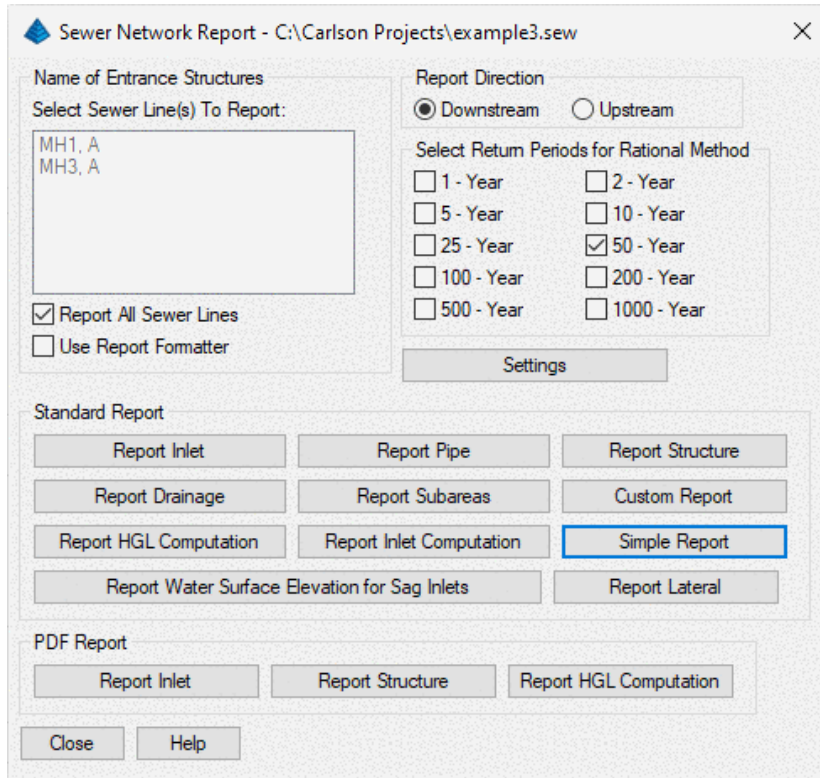
Click and drag the mouse to rotate the view to a good viewing angle as shown below:



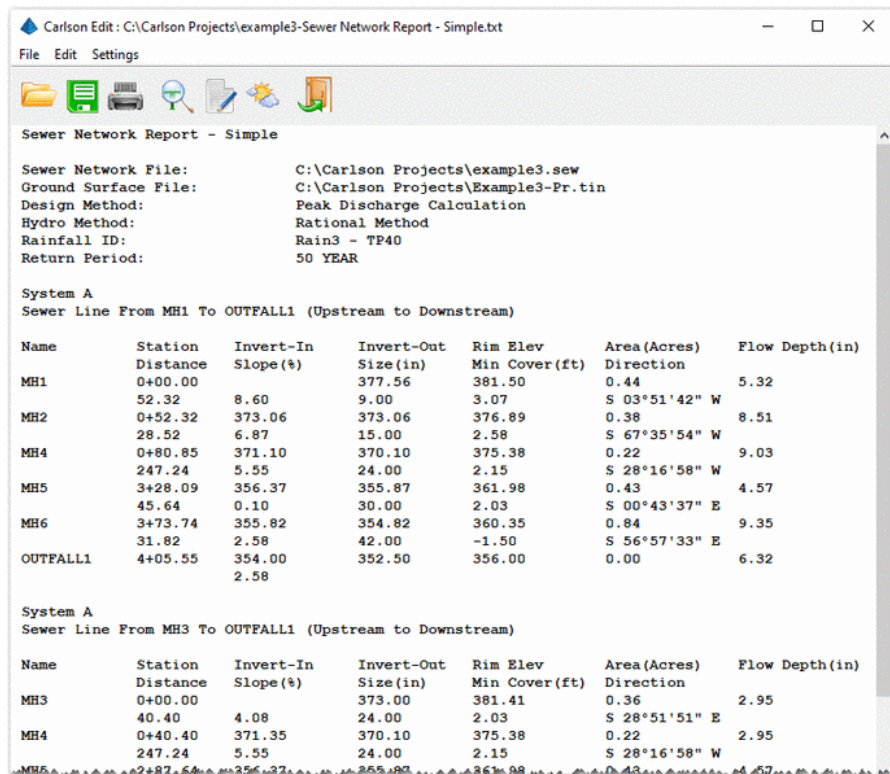
When done inspecting, dismiss the viewer by picking the **Exit (Doorway)** button.

20. Finally, let's check out the reports available and issue the Network – Report Sewer Network command to

display a dialog box similar to that shown below:



Let's run the **Simple Report** to get the summary of the system that displays in the Standard Report Viewer similar to that shown below:



Review the report and then click the **Exit** (Doorway) button and then click the **Close** button to dismiss the

Sewer Network Report dialog box.

This completes the tutorial: Stormwater Network Design.

Data Extraction for HydroCAD

This lesson covers extracting watershed data for HydroCAD stormwater modeling using Carlson Hydrology. The goal of this functionality within Carlson Hydrology is to automatically extract the sub-areas needed by HydroCAD to do its stormwater modeling. There are 3 main considerations to prepare a dataset for extraction:

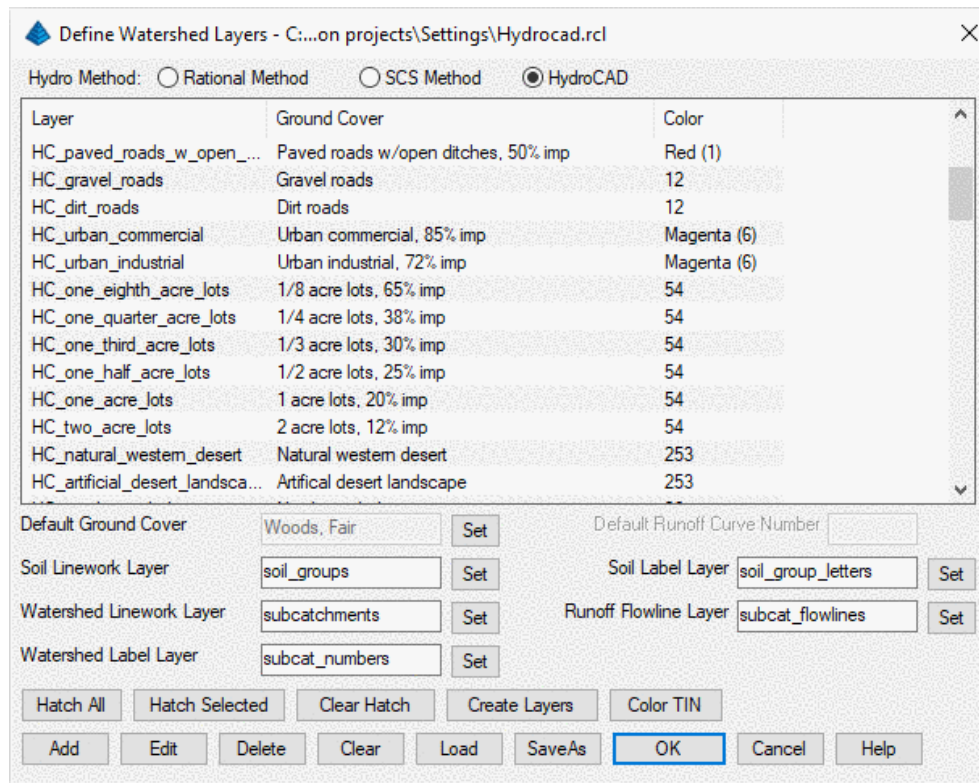
1. Watershed (subcatchment) delineation, with either:
 - (a) Longest flow path and average slope information for Curve Number/Lag method, or
 - (b) T_c with sheet, shallow, and channel flow information defined for TR55 method
2. Delineation of Hydrologic Soil Group (HSG) with:
 - (a) HSG boundaries defined on one layer, and
 - (b) HSG labels on another layer
3. Ground cover delineation, with:
 - (a) Each type of different ground cover area defined as closed polylines on specific layers, corresponding with entries in the hydrocad.rcl library

Let's get started.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

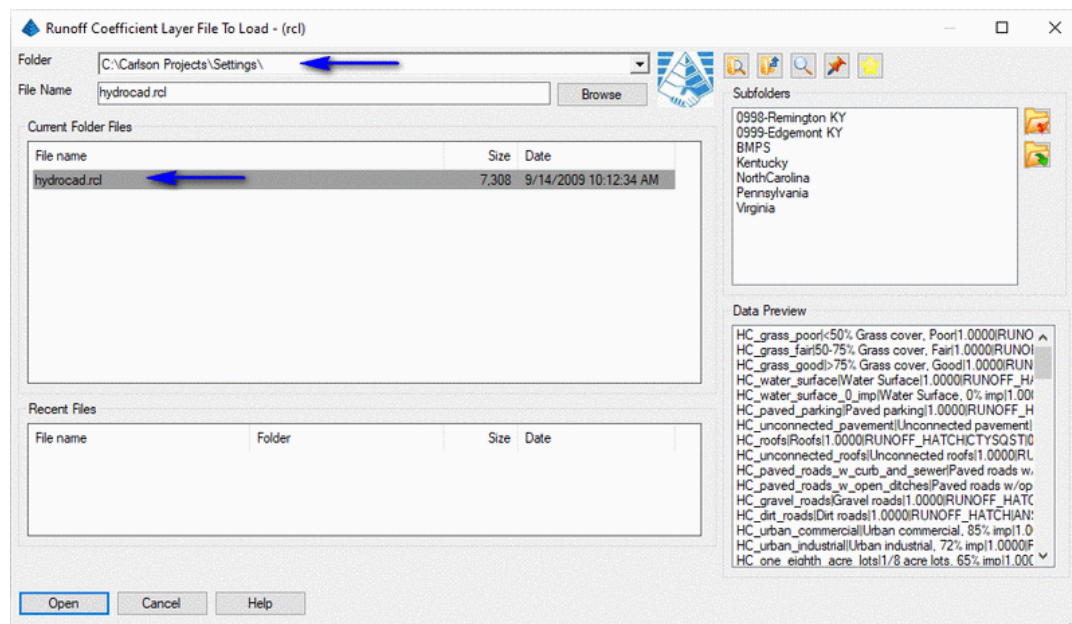
Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **HydroCAD.Tutorial.dwg** file. Use the File – **Save As** to save a copy named **HydroCAD_Tutorial-A.dwg**. Completing this tutorial will alter the drawing file and by renaming the file from the start, you'll keep the original file intact (allowing you to run through the tutorial a second time if desired). This is also a good practice to keep when working on drawings from 3rd parties.

2. Activate the Hydrology menu via Settings – Carlson Menus – **Hydrology Menu**. The entire process described herein is essentially a *layer-based* process, meaning that it is the drawing layers that everything is placed upon that makes the process work. So before going any further, take a look at the dialog box in which those layers are defined. Issue the HydroNet – Define Watershed Layers command to display a dialog box similar to that shown below:

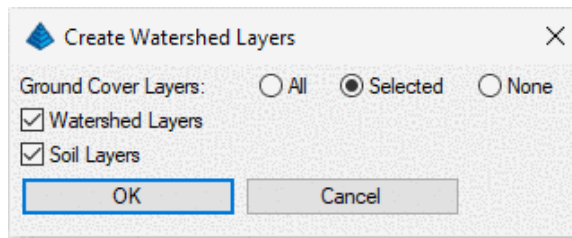


NOTE: If you have a predetermined grouping of watershed layers you wish to preserve, click the **Save** button to save your previous layers to a named *.RCL file. The step to follow will replace the existing layers with established *HydroCAD* default layers.

Click the **Load** button and load the **hydrocad.rcl** file as shown below and click **Open** when ready:



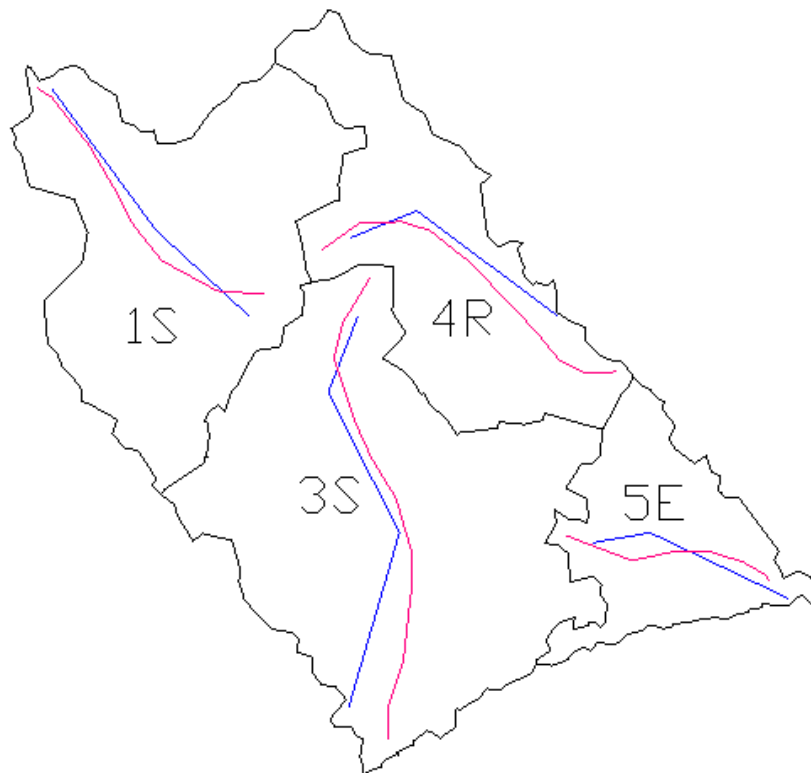
Once loaded, enable the **HydroCAD** radio button at the top of the dialog box. Any layer names in this dialog box can be changed to whatever you like, but the linework created must use the layers you decide on. A helpful trick is to highlight the layers in the list that you think you will need for the project at hand (using standard Windows *Shift+click* and/or *Ctrl+click* functionality) and then click the **Create Layers** button which would display a dialog box as shown below:



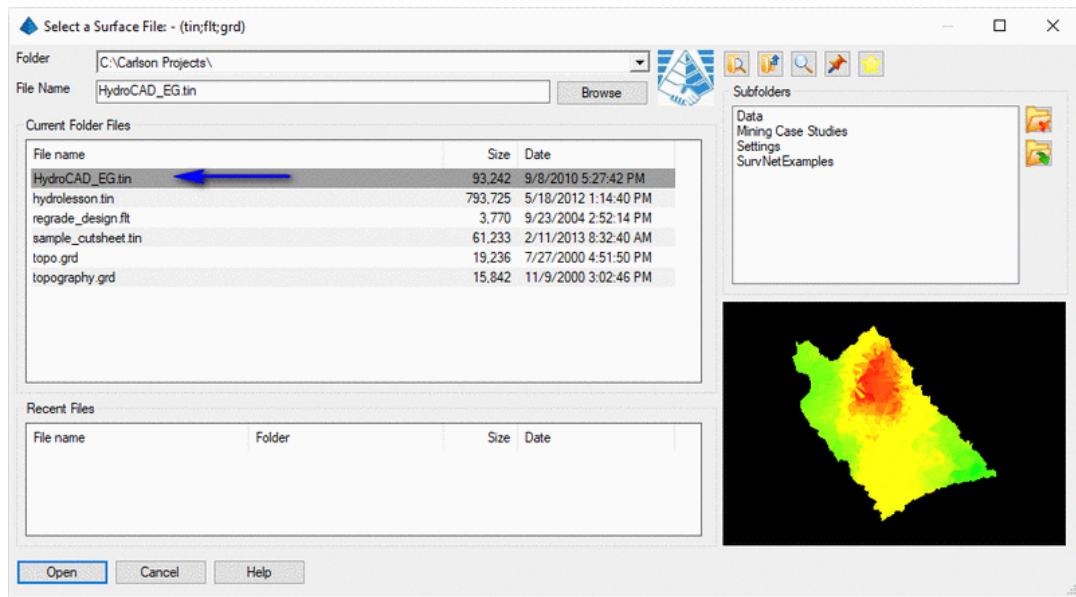
The layers will be created for you thereby eliminating the need to type them all in by hand, and also avoiding typos that will cause the data extraction to fail. Dismiss the *Create Watershed Layers* dialog box and click the **OK** button to dismiss the *Define Watershed Layers* dialog box. Now that the layers are established, you can focus on creating the geometry in the drawing.

3. In this example drawing, sample subcatchments (watersheds) for HydroCAD have been delineated for us with polyline geometry on the following layers:
 - subcatchments
 - subcat_numbers
 - subcat_flowlines
 - longest_flow_paths

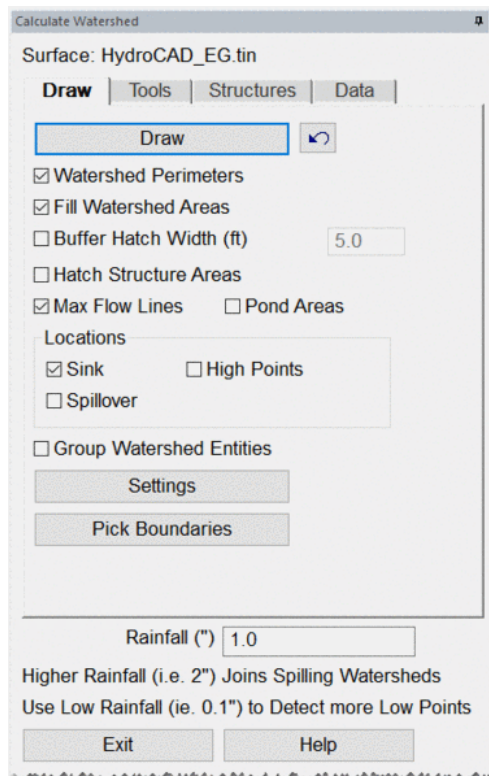
These subcatchment delineations can be done using any process desired... looking at the contours, looking at aerial photos, whatever method is most useful and comfortable is fine. It is the end result of linework that delineates the areas that are needed... the method to get them is inconsequential. The example data for this project is shown below (colors modified for clarity):



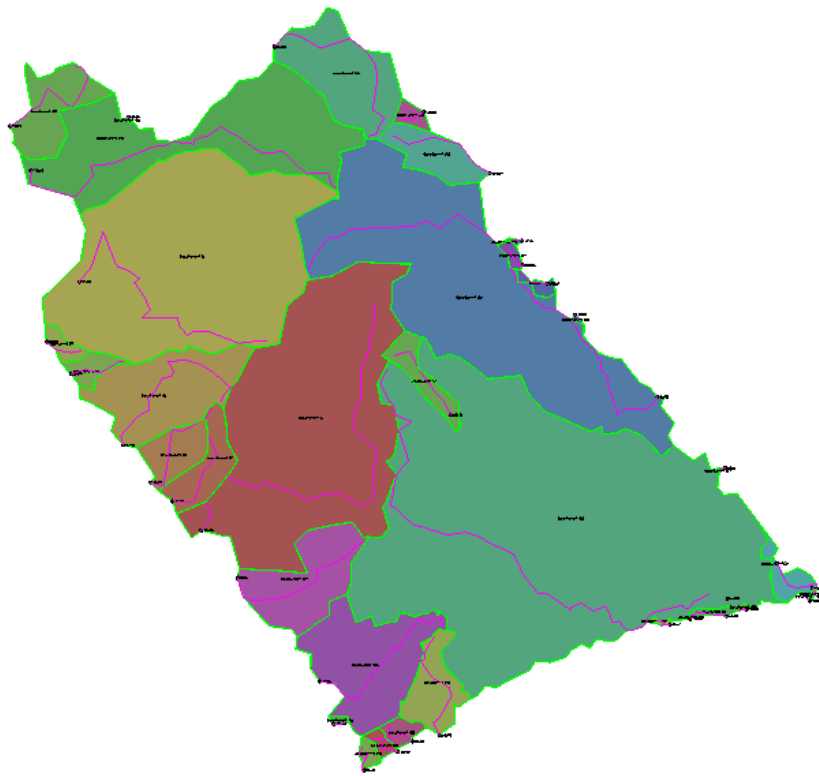
If there is a surface (TIN) for the project site (as there is for this example), **Carlson Hydrology** can greatly aid in the process of delineating watersheds (subcatchments) and determining the longest flow path for each subcatchment. These tasks are accomplished by issuing the HydroNet – (or Watershed –) Watershed Analysis command. When prompted:



Specify the surface model file as indicated above and click **Open** when ready. A "docked" dialog box similar to that shown below appears:



Set the various toggle options as cited above and click the **Draw** button when ready. The following should be displayed (layers frozen for clarity):



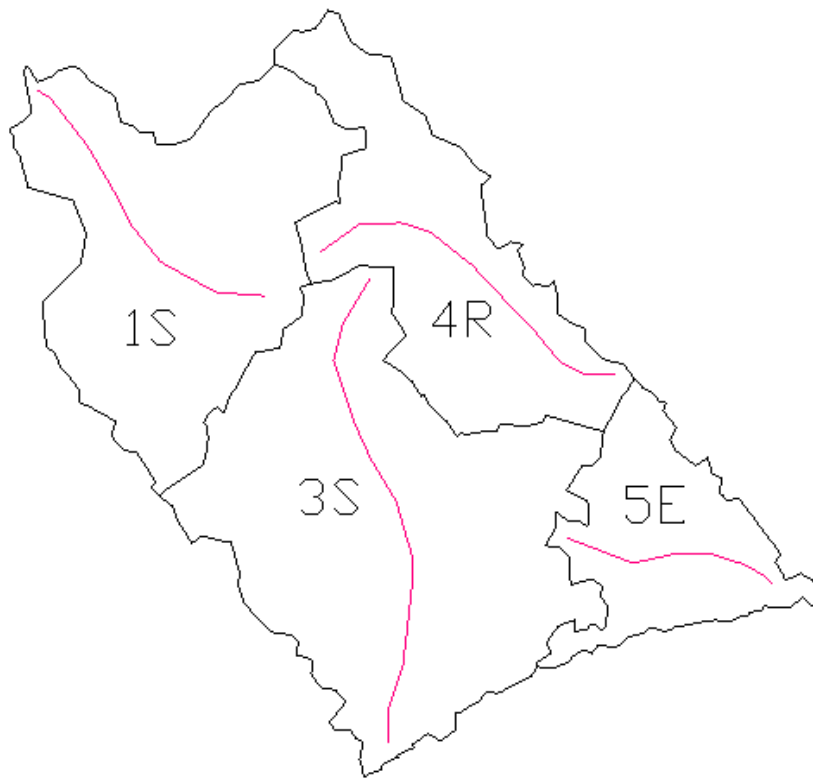
Once the watersheds and max flow lines are drawn, you can make decisions on them:

- Are there too many?
- Are there too few? *etc?*

Adjusting the **Rainfall** value and/or clicking the **Settings** button can assist with the desired delineation.

Tip: To combine several watershed perimeters into one, use the Draw – Shrink Wrap Entities command with the **Bound** method. You would pick the combination of watersheds, and a new boundary would be drawn around them.

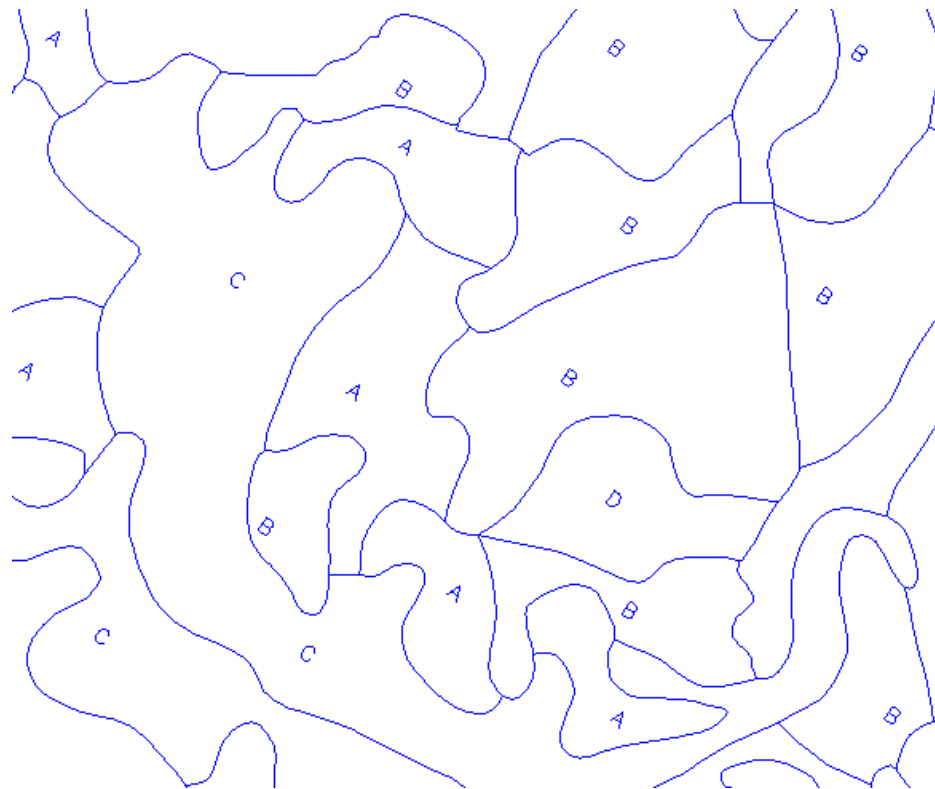
For the purposes of this tutorial, let's use the polyline geometry already established for us. Click the **Undo** button (next to the *Draw* button) to remove the **Watershed Analysis** results from the drawing. Ultimately, you should be left with a closed polyline for each watershed, labeled with a designator and containing a single 3D polyline within each watershed representing the longest flow path as shown below (certain layers frozen for clarity):



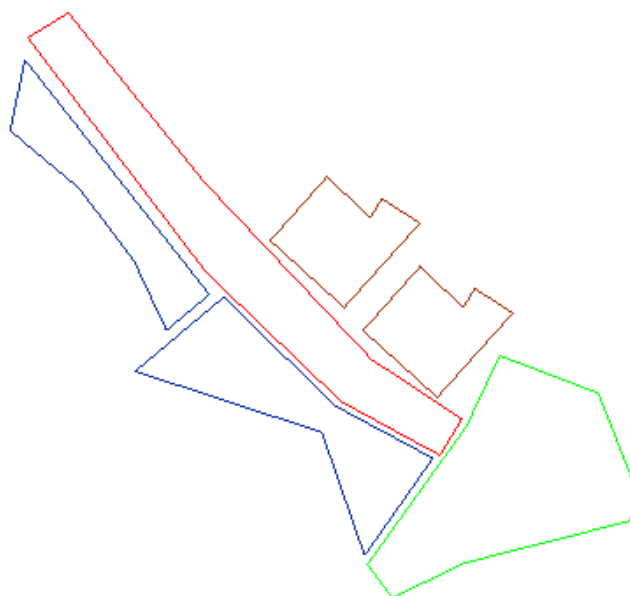
Click on the **Exit** button to dismiss the **Watershed Analysis** dialog box.

4. Soil conditions (Hydrologic Soil Groups or *HSG* for short) must also be properly delineated. What you'll need, ultimately, from whatever source(s) you have is a drawing with:
 - the soil groups drawn as polylines, all on one layer, and
 - the HSG types (A, B, C, or D) labeled within the areas on another single, specific layer.

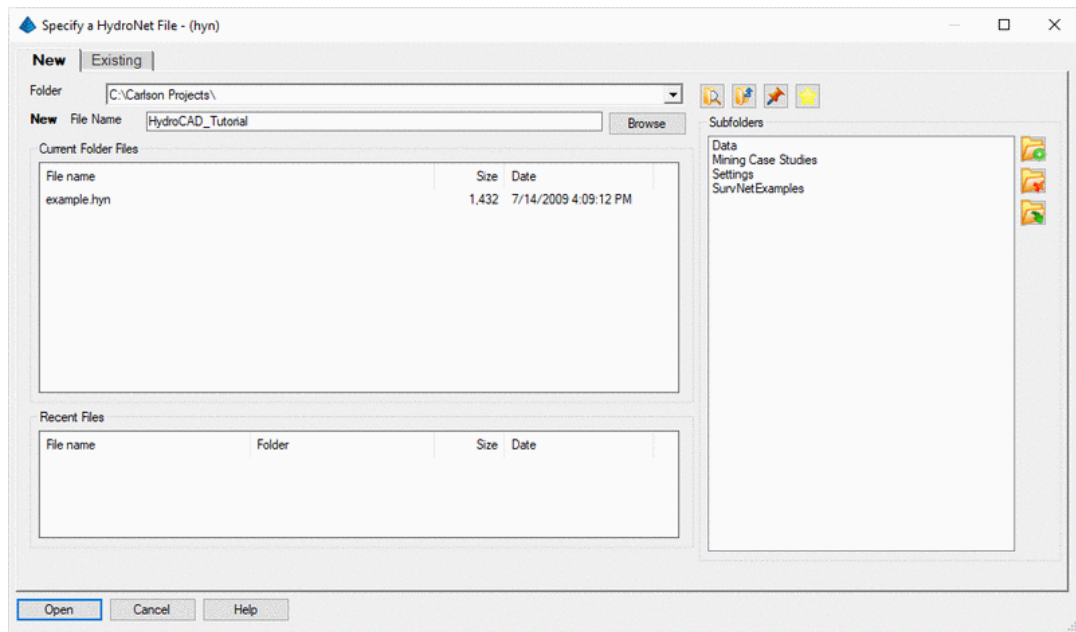
NOTE: The soil areas must be closed, but do not have to be individual specific closed polylines. The software will run a "boundary polyline" (*bpoly*) on each area to "close" it. The soils information can come from a shape (*SHP*) file such as that from USDA - NRCS or , from a ground soils survey, from digitized plans... it doesn't matter. The only thing necessary is the linework and labels such as those shown below (certain layers frozen for clarity):



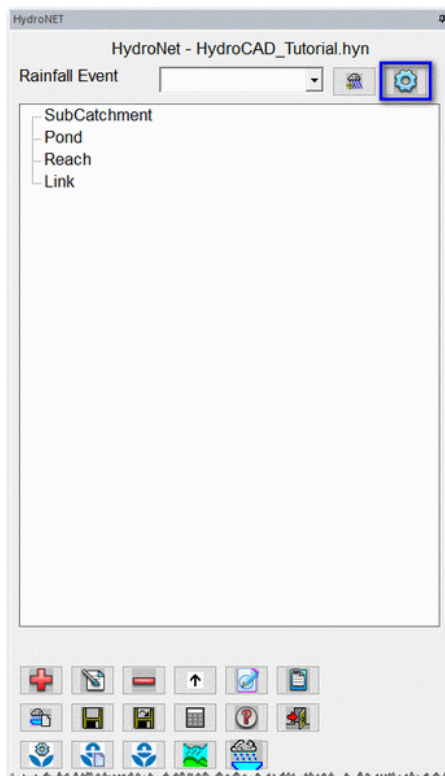
5. Finally, ground cover. In standard use, this term applies only to the ground cover which is independent of the underlying soil group(s). Neither layer equates to a specific Curve Number (CN) value which can only be determined after we analyze the intersection of these layers. The actual CN value(s) will be assigned by HydroCAD after this analysis is complete. Once again, with whatever method you like, create closed polylines on the various ground cover layers as illustrated below:



6. Now that you've drawn all the watershed and site data, we can start to extract the information we need for hydrologic analysis. Issue the HydroNet – HydroNet Explorer. When prompted, create the HydroNet file as cited below and click **Open** when ready:

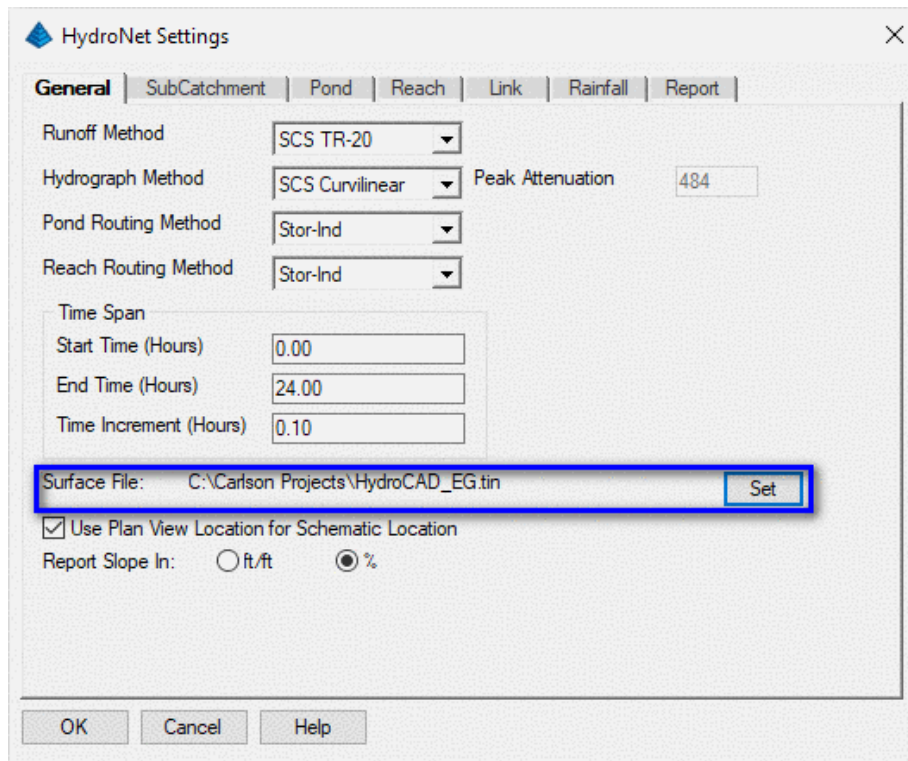


In the resulting "docked" dialog box, pick the **Settings** (Gear) button as highlighted below:



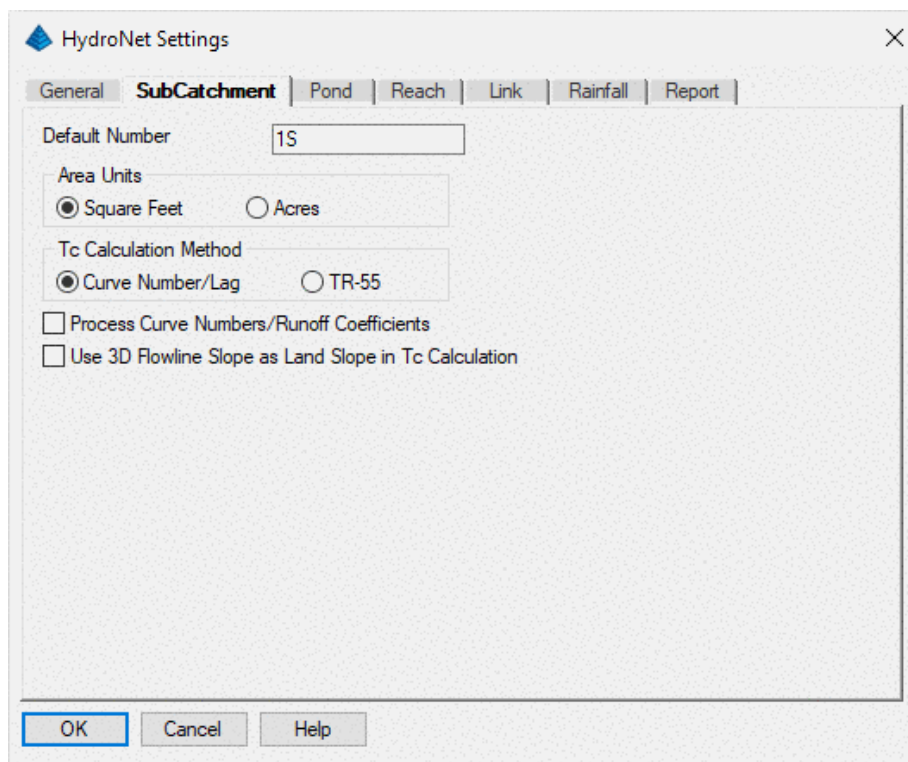
In the Settings dialog box:

- on the *General* tab, set the click the **Set** to set the **Surface File** we used earlier as shown below:



Set the other values as shown.

- on the *SubCatchment* tab, set the **T_c Calculation Method** to **Curve Number/Lag** as illustrated below:



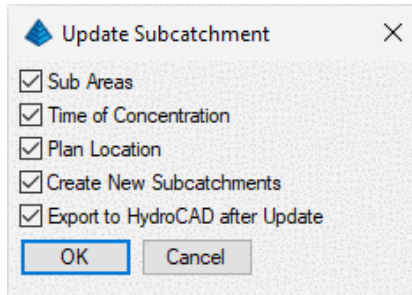
Also, clear the box to **Process Curve Numbers/Runoff Coefficients** as they will be determined in HydroCAD.

- Review the other tabs of settings as found in this dialog box and click **OK** when ready.
7. You are ready for the data extraction to be performed and the results exported to HydroCAD but we must make sure all pertinent layers are *On/Thawed*. Issue the View – Thaw/On All Layers command so the data

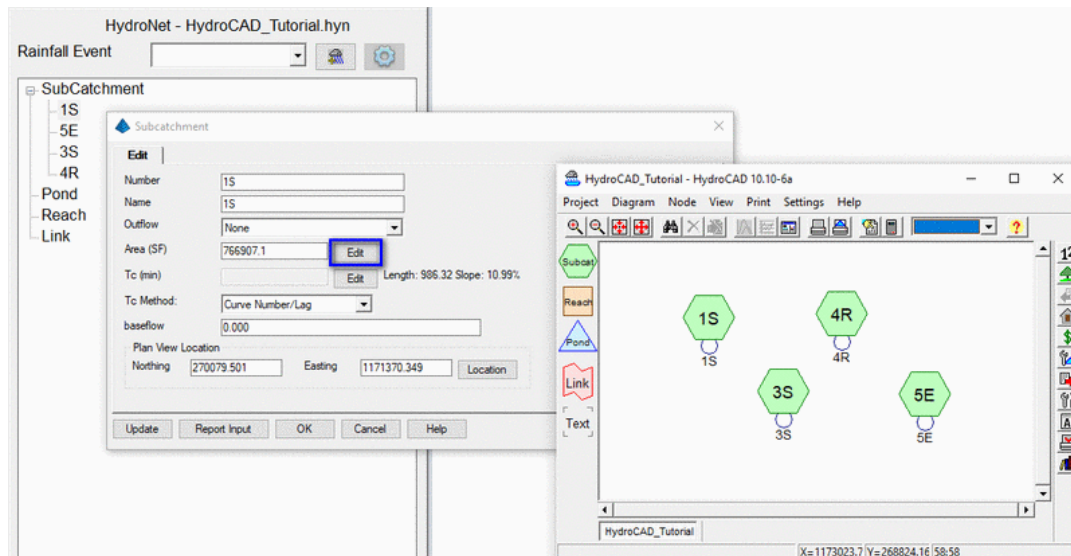
can be processed. From the main HydroNet docked dialog, click the **Update** (Up Arrow) button as illustrated below:



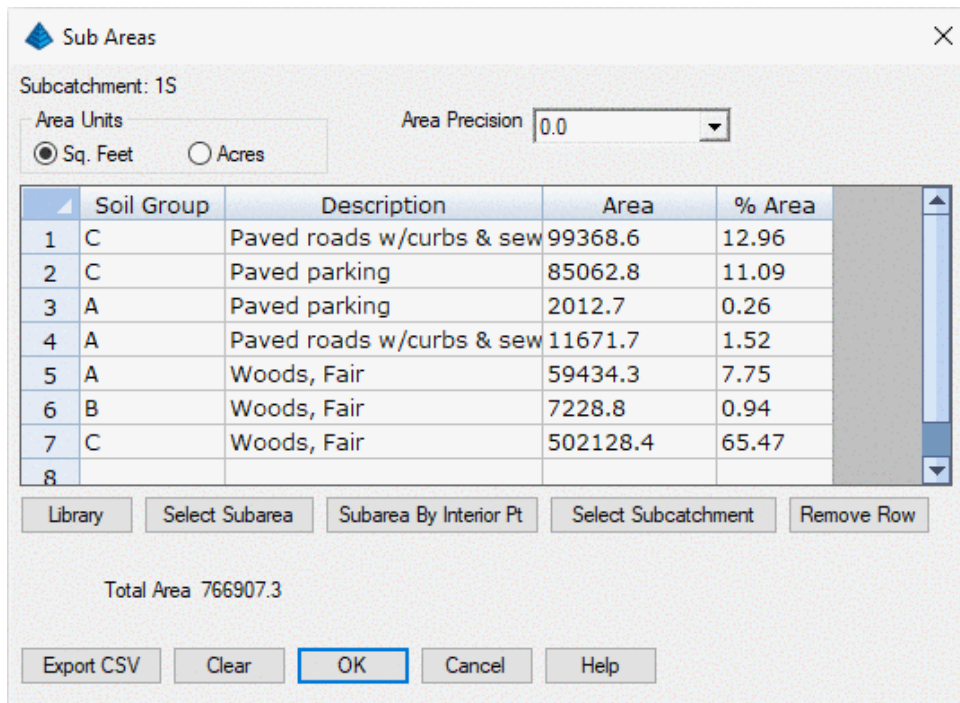
A secondary dialog box appears with the options for the update. Select the options as shown below and click **OK** when ready:



The subcatchments are listed in the **HydroNet Explorer** and *HydroCAD* (if installed), is launched. Double-clicking on any subcatchment name brings up a dialog with its data as shown below:

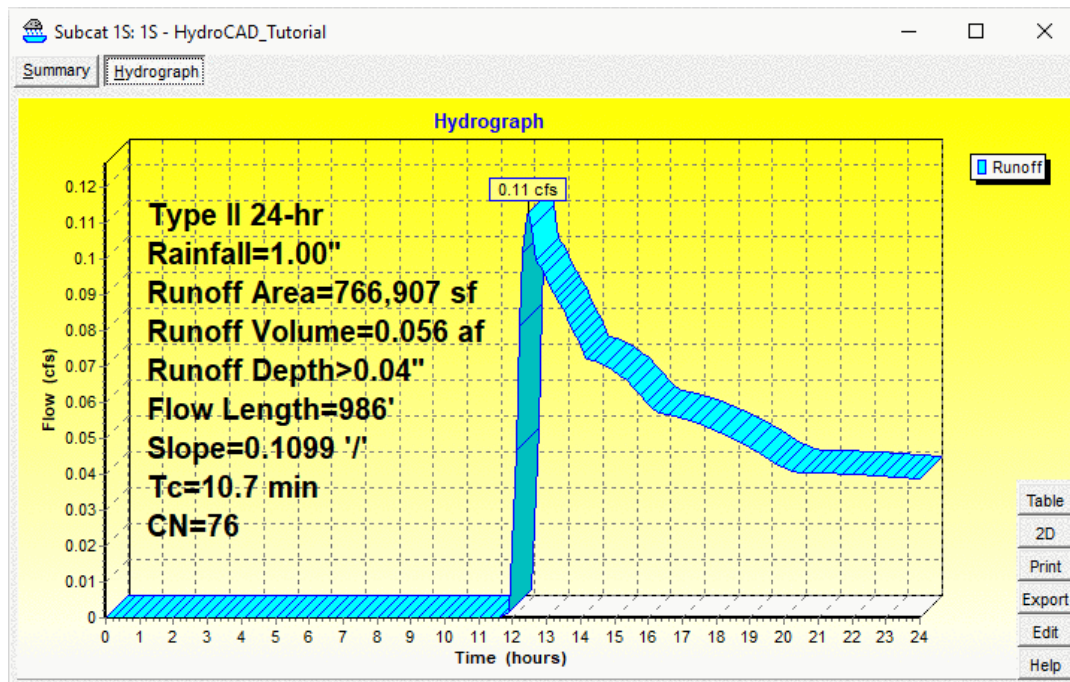


NOTE: The same subcatchments now exist in *HydroCAD*, ready for any further analysis and/or reporting. Click the **Edit** button next to the *Area* value shows all of the sub-area statistics as illustrated below:



Click **OK** to dismiss the **Sub Areas** dialog box and then click **OK** to dismiss the **Subcatchment** dialog box to return to the *HydroNet Explorer* docked dialog box. Click the **Exit** (Doorway) button to dismiss the **HydroNet Explorer** docked dialog box.

8. Double-clicking a subcatchment in *HydroCAD* opens the **Hydrograph** and **Summary** report windows as summarized below:



Subcat 1S: 1S - HydroCAD_Tutorial

Summary Hydrograph

CarlsonPlanXYPos[1171370.3492|270079.5010]

[49] Hint: Tc<2dt may require smaller dt

Runoff = 0.11 cfs @ 12.45 hrs, Volume= 0.056 af, Depth> 0.04"

Runoff by SCS TR-20 method, UH=SCS, Weighted-CN, Time Span= 0.00-24.00 hrs, dt= 0.10 hrs
Type II 24-hr Rainfall=1.00"

Area (sf)	CN	Description
99,369	98	Paved roads w/curbs & sewers HSG C
85,063	98	Paved parking HSG C
2,013	98	Paved parking HSG A
11,672	98	Paved roads w/curbs & sewers HSG A
59,434	36	Woods, Fair HSG A
7,229	60	Woods, Fair HSG B
502,128	73	Woods, Fair HSG C
766,907	76	Weighted Average
568,791		74.17% Pervious Area
198,116		25.83% Impervious Area

Tc (min)	Length (feet)	Slope (ft/ft)	Velocity (ft/sec)	Capacity (cfs)	Description
10.7	986	0.1099	1.54		Lag/CN Method,

Table
Shrink
Print
Export
Edit
Help

Project reports can also be generated for Nodes, Areas and Soils (among others):

Project Reports - HydroCAD_Tutorial

Node Listing Area Listing Soil Listing Ground Covers Notes

Time span=0.00-24.00 hrs, dt=0.10 hrs, 241 points
Runoff by SCS TR-20 method, UH=SCS, Weighted-CN
Reach routing by Stor-Ind method - Pond routing by Stor-Ind method

All Nodes
 Selected Nodes
 Show Totals

Subcatchment 1S: 1S
Runoff Area=766,907 sf 25.83% Impervious Runoff Depth>0.04"
Flow Length=986' Slope=0.1099 1/100 Tc=10.7 min CN=76 Runoff=0.11 cfs 0.056 af

Subcatchment 3S: 3S
Runoff Area=1,082,043 sf 37.75% Impervious Runoff Depth>0.01"
Flow Length=1,483' Slope=0.0934 1/100 Tc=18.0 min CN=72 Runoff=0.03 cfs 0.024 af

Subcatchment 4R: 4R
Runoff Area=550,294 sf 19.54% Impervious Runoff Depth=0.00"
Flow Length=1,099' Slope=0.1212 1/100 Tc=17.0 min CN=80 Runoff=0.00 cfs 0.000 af

Subcatchment 5E: 5E
Runoff Area=332,842 sf 0.00% Impervious Runoff Depth>0.01"
Flow Length=862' Slope=0.0811 1/100 Tc=10.1 min CN=72 Runoff=0.01 cfs 0.008 af

Total Runoff Area = 62.720 ac Runoff Volume = 0.088 af Average Runoff Depth = 0.02"
73.86% Pervious = 46.325 ac 26.14% Impervious = 16.395 ac

Table
Shrink
Print
Export
Edit
Help

Project Reports - HydroCAD_Tutorial

Node Listing | **Area Listing** | Soil Listing | Ground Covers | Notes

Area (acres)	CN	Description (subcatchment-numbers)
1.133	49	50-75% Grass cover, Fair HSG A (3S, 5E)
2.675	69	50-75% Grass cover, Fair HSG B (3S, 5E)
0.014	79	50-75% Grass cover, Fair HSG C (3S)
2.982	84	50-75% Grass cover, Fair HSG D (3S, 5E)
2.909	98	Paved parking HSG A (1S, 3S)
1.548	98	Paved parking HSG B (3S)
1.953	98	Paved parking HSG C (1S)
0.074	98	Paved parking HSG D (3S)
2.583	98	Paved roads w/curbs & sewers HSG A (1S, 3S)
0.716	98	Paved roads w/curbs & sewers HSG B (3S)
2.281	98	Paved roads w/curbs & sewers HSG C (1S)
0.460	98	Paved roads w/curbs & sewers HSG D (3S)
0.821	98	Roofs HSG A (3S, 4R)

All Nodes
 Selected Nodes
 Show Totals

Table
Shrink
Print
Export
Edit
Help

Project Reports - HydroCAD_Tutorial

Node Listing | Area Listing | **Soil Listing** | Ground Covers | Notes

Area (acres)	Soil Group	Subcatchment Numbers
18.932	HSG A	1S, 3S, 4R, 5E
17.168	HSG B	1S, 3S, 4R, 5E
20.325	HSG C	1S, 3S, 4R, 5E
6.295	HSG D	3S, 4R, 5E
0.000	Other	
62.720		TOTAL AREA

All Nodes
 Selected Nodes
 Show Totals

Table
Shrink
Print
Export
Edit
Help

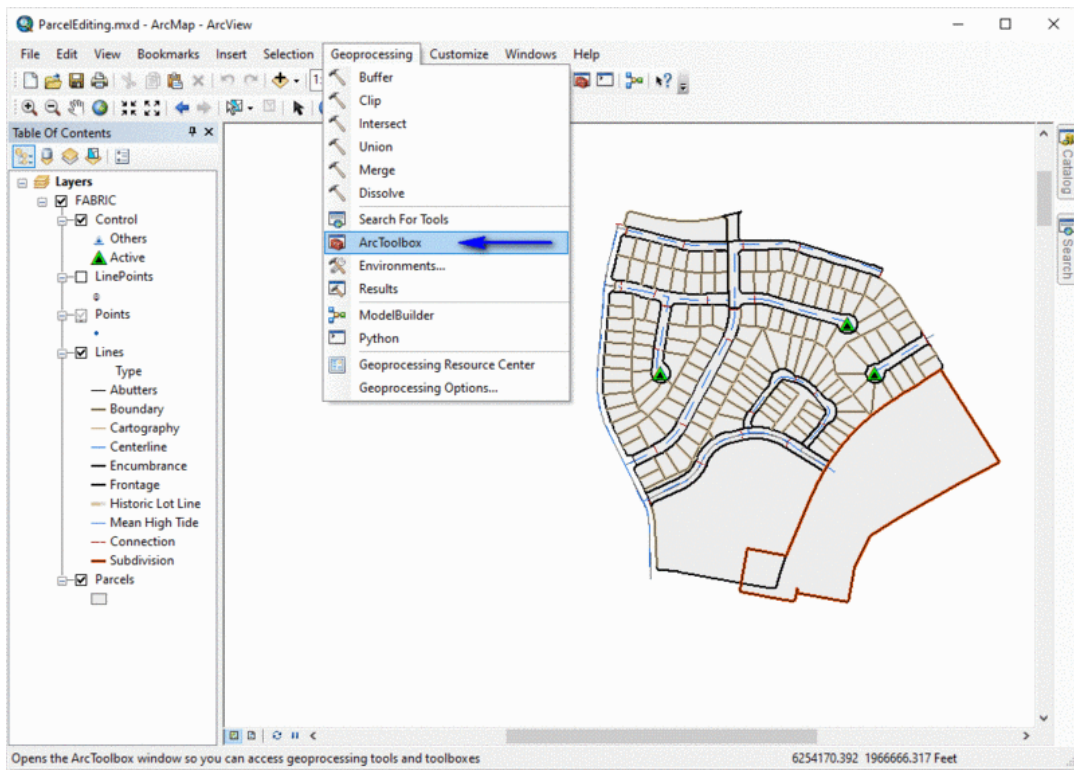
Any changes made in the drawing can instantly update the data in *HydroCAD* simply by picking the **Update** button.

This completes the tutorial: Data Extraction for HydroCAD.

ESRI to Office to Field and Back

This lesson takes an ESRI geodatabase from ArcView into Carlson GIS and then to Carlson SurvCE for data collection. Next the data is taken from SurvCE into Carlson Survey and processed by Carlson GIS back into ArcView.

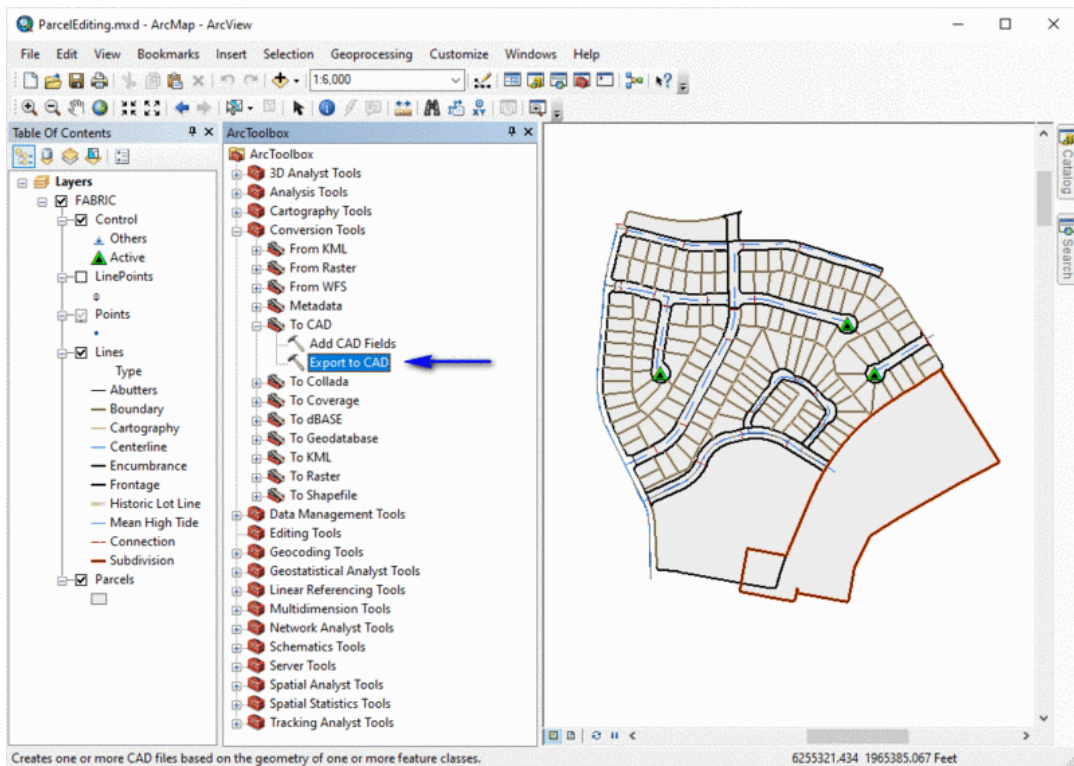
1. ArcGIS Desktop has a routine to output a geodatabase to a DWG file. The DWG file contains all the information in a single file. It has the graphic geometry, feature definitions and feature attribute data. The feature information is stored in a format defined by ESRI called Mapping Specification for DWG (MSD) using standard DWG entities and dictionary entries. In this tutorial, we're going to use ArcView 10.x to create the DWG with MSD. Launch ArcView (sold separately) and open a desired project. In this case, we are illustrating an Esri-supplied tutorial geodatabase:



Opens the ArcToolbox window so you can access geoprocessing tools and toolboxes

6254170.392 1966666.317 Feet

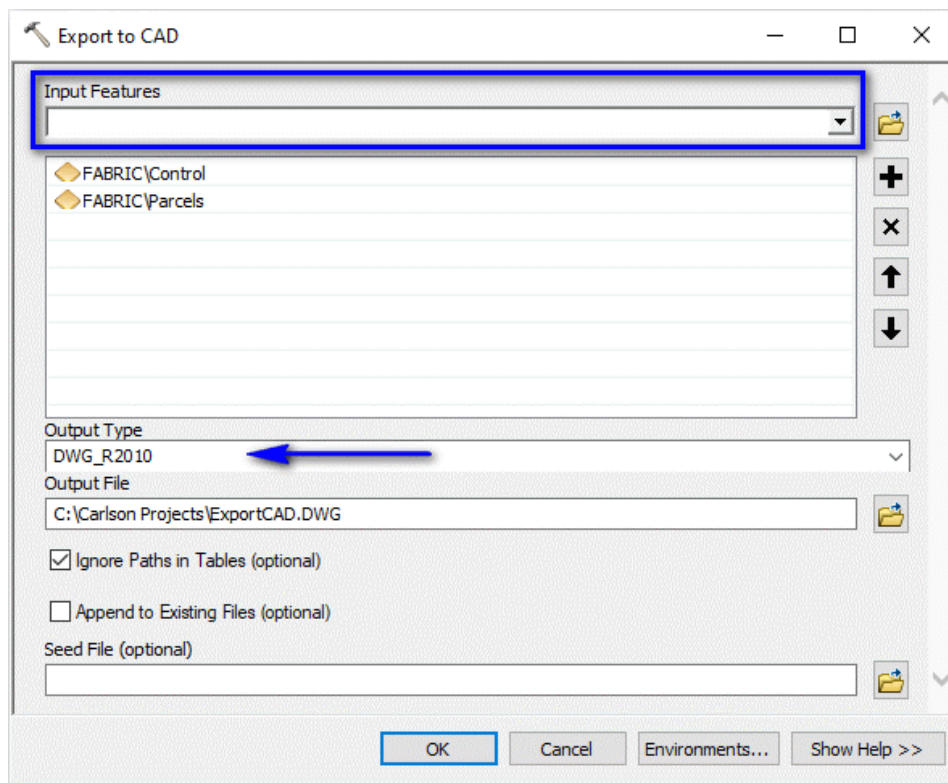
If not already open, launch the **ArcToolbox** as illustrated above. Issue the ArcToolbox – Conversion Tools – To CAD – Export To CAD as illustrated below:



Creates one or more CAD files based on the geometry of one or more feature classes.

6255321.434 1965385.067 Feet

The next task will be to select the features to export in the **Export To CAD** dialog box. To select a feature, click the **Input Features** drop-down list (as illustrated below) and pick a desired feature from the list:

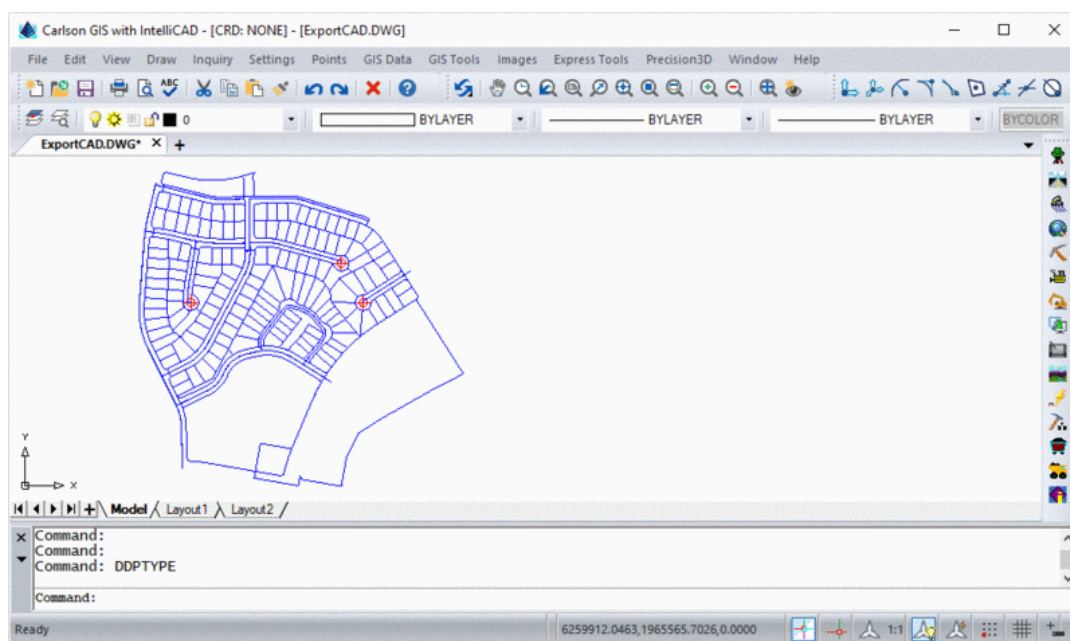


Set the remaining fields as desired including a desired DWG version (Carlson currently suggests the AutoCAD 2010 format for SurvCE users) and a path/filename for the DWG. Click **OK** when ready.

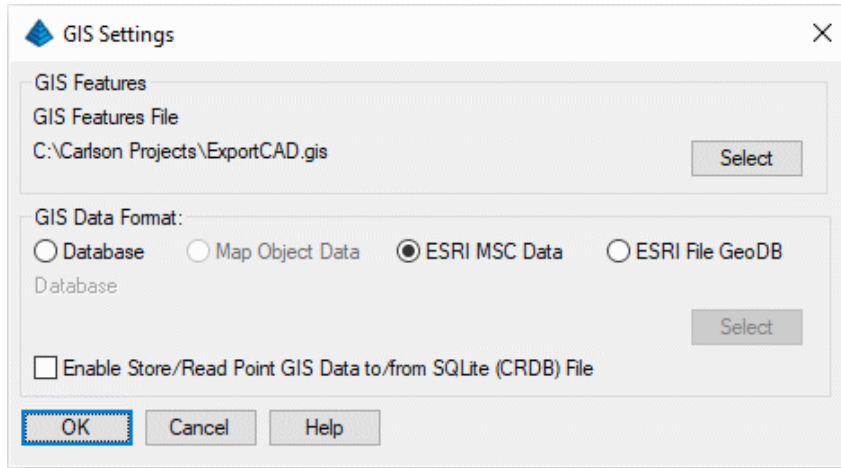
2. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **DWG** file you just exported from ArcView

3. Activate the GIS menu via Settings – Carlson Menus – **GIS Menu**. Your drawing should resemble that shown below (CAD formatting applied for clarity):

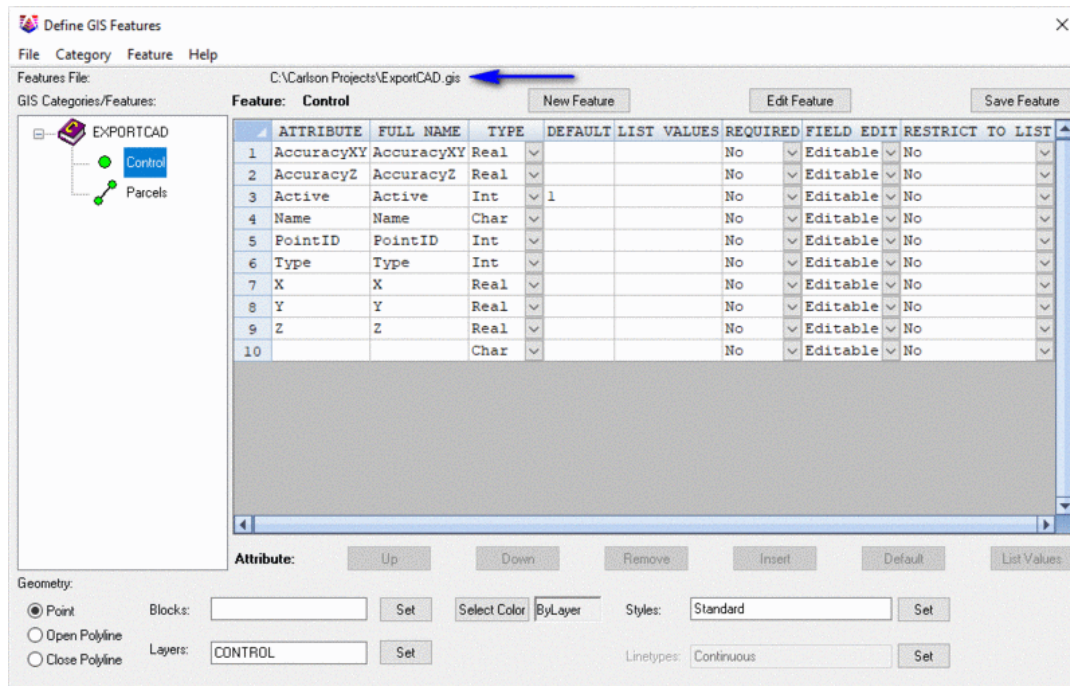


Next, run GIS Data – GIS Database Settings to display a dialog box similar to that shown below:



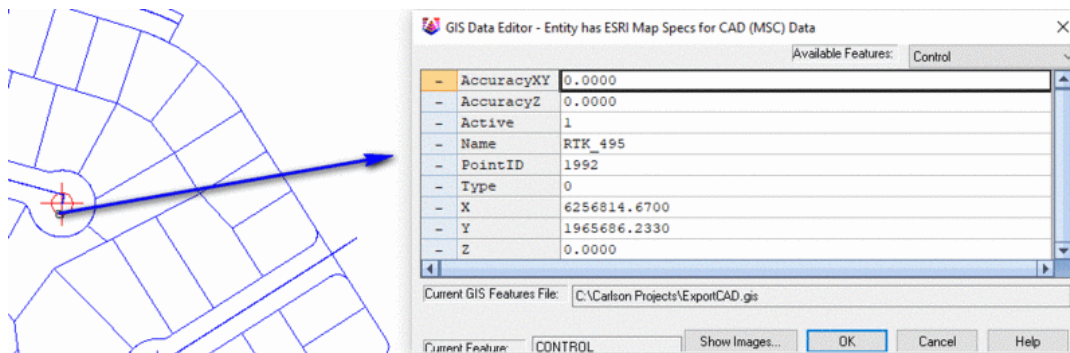
Set the values as shown above and click **OK** when ready.

- Now that we have a GIS "container" for the various GIS features, let's populate the container. Issue the GIS Data – Define GIS Features command to display a dialog box similar to that shown below:



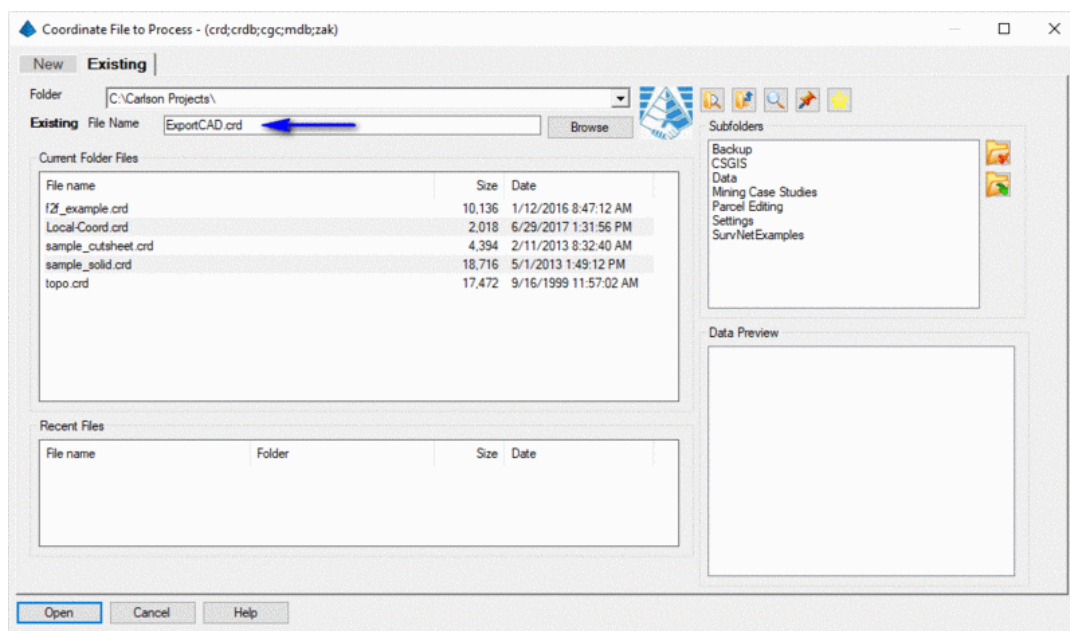
Issue its File – Import From – **ESRI Map Specs for CAD (MSC)** command which should populate the dialog box with the various fields of data as defined by the geodatabase. Dismiss the dialog box via its File – **Save** command followed by its File – **Exit** command.

- To verify that the GIS attribute data is in the drawing, issue the GIS – GIS Data Editor and pick on an entity in the drawing (in this case, a *Control* point). The attribute data is shown in a spreadsheet editor (click **OK** when ready):

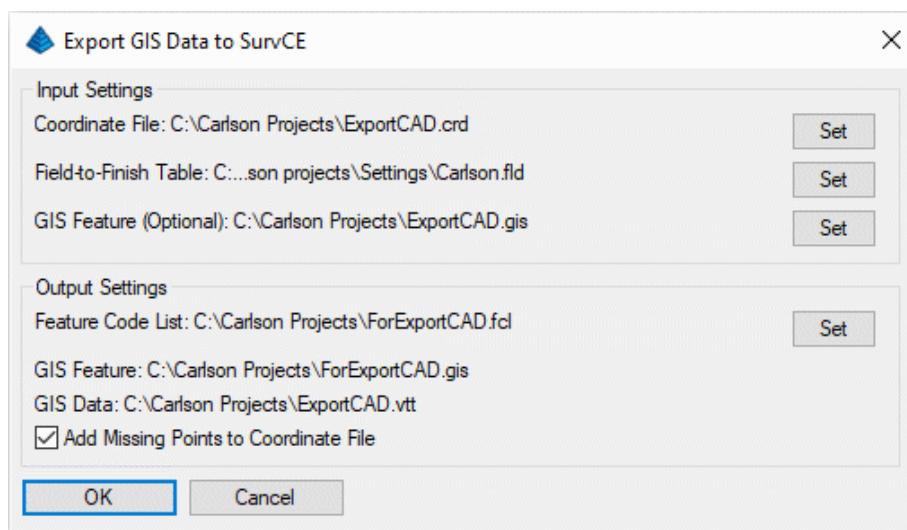


NOTE: Some graphical features may not have associated GIS data!

- Now that we have data available, let's package it up for Carlson SurvCE. Issue the GIS Data – Export GIS Data – Export GIS Data to SurvCE. You may be prompted:



Set the filename as shown above and click **Open** when ready to display a dialog box similar to that shown below:



This command takes selected data from the drawing and creates the GIS files that SurvCE uses. Set the values as shown above and click **OK** when ready. When prompted:

Select entities to export.

[FILter]/<Select entities>: type ALL and press Enter

[FILter]/<Select entities>: press Enter

For a brief summary on the files and content cited above:

For input files:

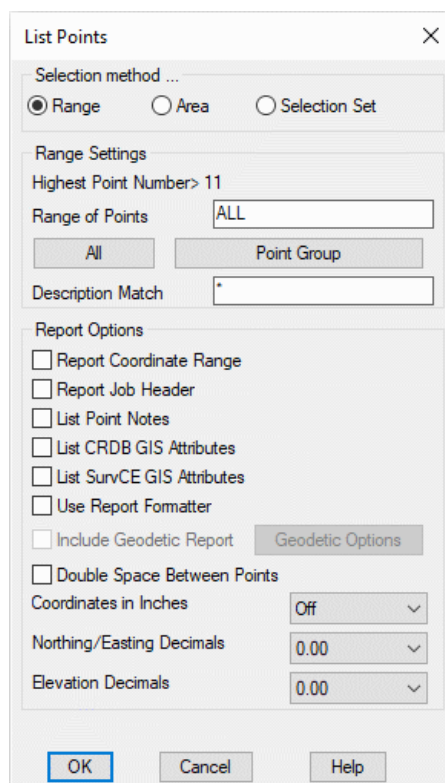
- **Coordinate File:** Contains the point database with Point#, Northing, Easting, Elevation and Description.
- **Field-to-Finish Table:** Defines the coding used by Carlson Survey for the description field that will be converted into a *Feature Code List* (FCL) for SurvCE.
- **GIS Feature:** Defines the feature and attribute names. Using this file is optional and applies in case the feature definitions contain more features or attributes than the data entities.

For Output files:

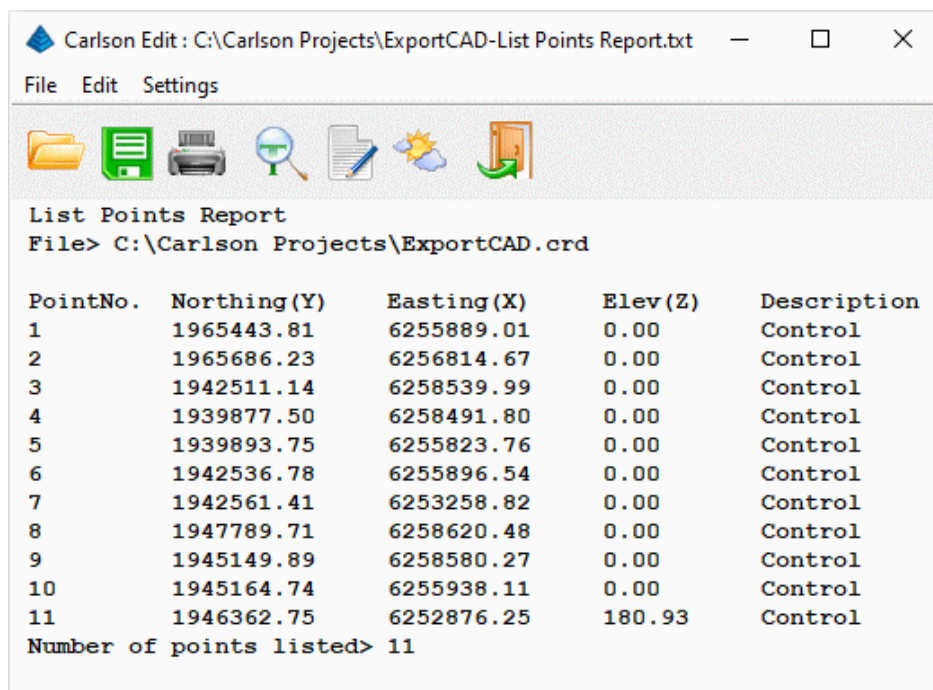
- **Feature Code File:** This is the SurvCE *Feature Code List* file for the description field coding definitions.
- **GIS Feature:** This defines the feature and attribute names. This file is automatically named after the *Feature Code File*.
- **GIS Data:** This file contains the attribute data. This file is automatically named after the *Coordinate File*.
- **Add Missing Points to Coordinate File:** This option creates points in the *Coordinate File* for any selected point entities that aren't already in the *Coordinate File*.

The program will read the GIS data from the selected entities to create the SurvCE GIS data file (.vtt) for SurvCE.

7. Let's confirm we have point coordinates for SurvCE. Issue the Points – List Points command to display a dialog box similar to that shown below:



Set the values as shown above and click **OK** when ready to display the results in the Standard Report Viewer as is illustrated below:



Review the list and click the **Exit** (Doorway) button to dismiss the report.

8. Now that the project data is converted to SurvCE format, upload/copy the following files on to the SurvCE collector:
 - Coordinate File (*.crd) as specified in the *Input Files* section.
 - Feature Code File (*.fcl) as specified in the *Output Files* section.
 - GIS Feature (*.gis) as specified in the *Output Files* section.
 - GIS Data (*.vtt) as specified in the *Output Files* section.

NOTE:If you have SurvCE 2.5 or later, then you can also use the DWG file format for the drawing. For earlier versions, use the DXF format.

9. With the data on the SurvCE collector, let's see what the process should look like. Launch SurvCE and open the *Coordinate File* generated earlier. You may get prompted for a coordinate system projection. It is often advisable to keep the projection systems consistent between the applications as illustrated below:

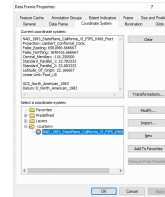
Application

Where to Find

Example

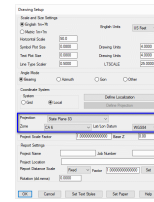
ArcView

View – Data Frame
Properties – Coordinate
System



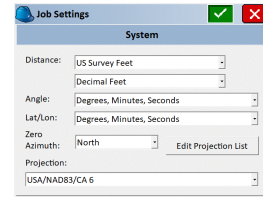
Carlson GIS/Survey

Settings – Drawing Setup



Carlson SurvCE/SurvPC

File – Job Settings – System
tab



Coordinate System Summary

Let's also explore the SurvCE **Feature Code List** (button #5 on the Main Menu – File tab) to display a dialog box similar to that shown below:

Code List: ForExportCAD

Category: ALL Set

Code	Linework	Line Type	Layer Name	Full Text	Co
2D	No	2D	TREE-SML	SMALL 2" TREE	2
2P	No	2D	PINE-SML	SMALL 2" PINE	2
4D	No	2D	TREE-SML	SMALL 4" TREE	4
4P	No	2D	PINE-SML	SMALL 4" PINE	4
6D	No	2D	TREE-SML	SMALL 6" TREE	6
6P	No	2D	PINE-SML	SMALL 6" PINE	6

Add Edit Remove
Load Save As Special Codes

NOTE: If not previously available in the source **Field-to-Finish Table** (*.FLD) file as specified in the Export GIS Data to SurvCE command above, it would be advisable to **Add** the code(s) to the SurvCE *Feature Code Library* as illustrated below:

When defined, the associated *attributes* will be available and presented to the user when the collect a shot using the cited code:

Attribute	Type	Req.	Editable	Select-Only
ACCURACYXY	Real	No	Allow	No
ACCURACYZ	Real	No	Allow	No
ACTIVE	Int	No	Allow	No
NAME	Char	No	Allow	No
POINTID	Int	No	Allow	No
TYPE	Int	No	Allow	No

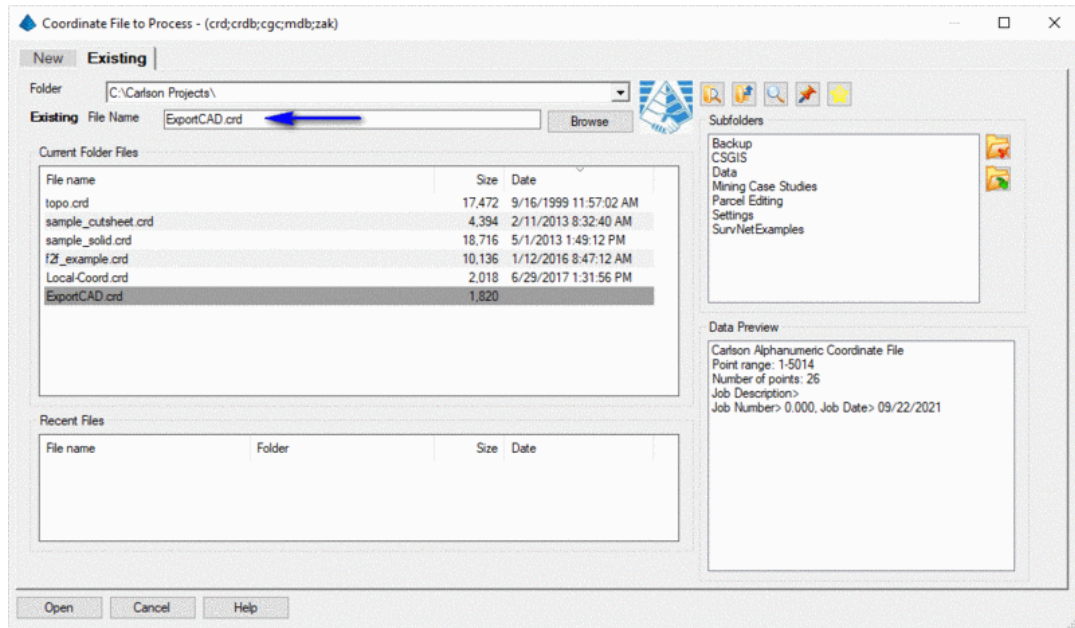
Collect/measure data in the field with SurvCE using conventional data gathering procedures. It is suggested that new features be coded at a point number range in excess of the existing point numbers to prevent confusion.

10. With data collected in the field, let's prepare it for its return trip. Migrate the following files from the data

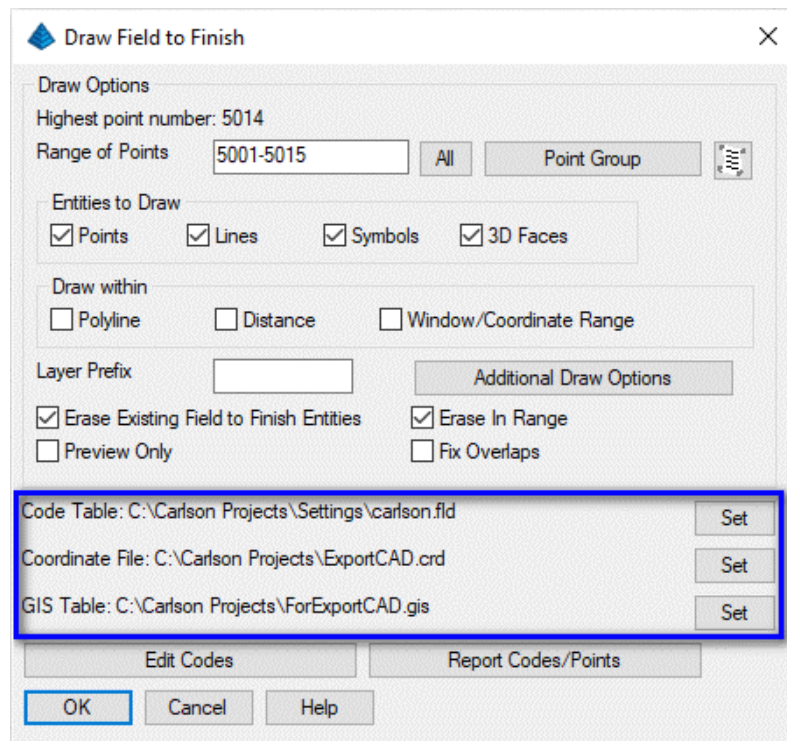
collector/controller of SurvCE back to the desktop computer:

- Updated Coordinate File (*.crd) as amended by *SurvCE*.
- GIS Data (*.vtt) as augmented by *SurvCE*.
- GIS Feature (*.gis) as used in the *SurvCE*.
- (Optional): DWG/DXF file as produced by *SurvCE*.

Re-launch the Carlson Survey desktop CAD application and then issue the Points – Set Coordinate File to set the coordinate file from SurvCE as current as illustrated below:

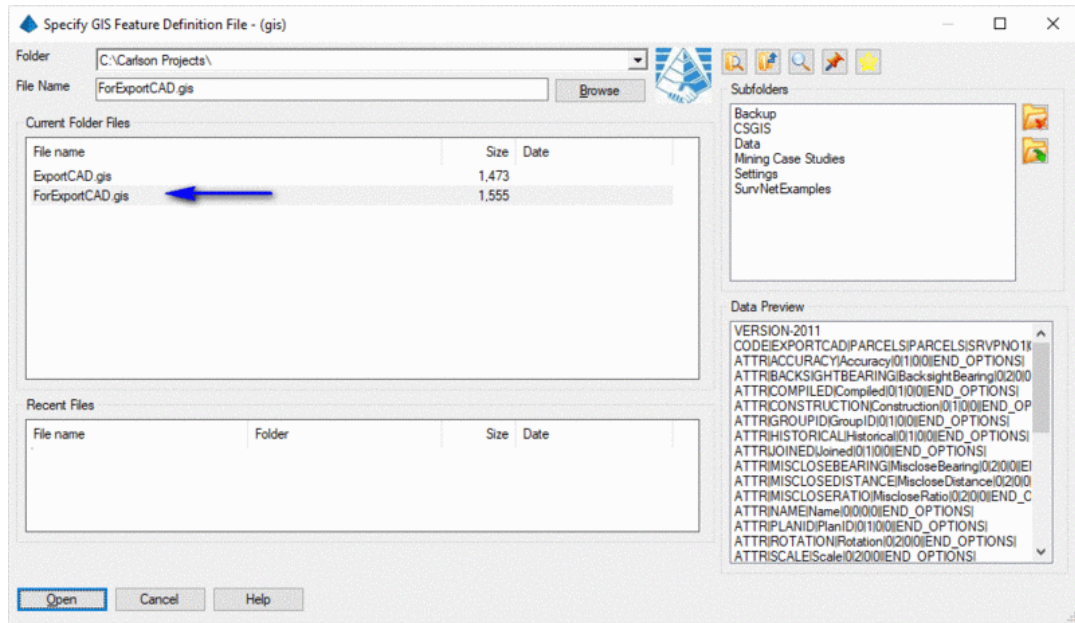


Activate the Survey menu via Settings – Carlson Menus – **Survey Menu**. Issue the Survey – Draw Field-to-Finish command and make note of the highlighted values illustrated below (as brought over from SurvCE):



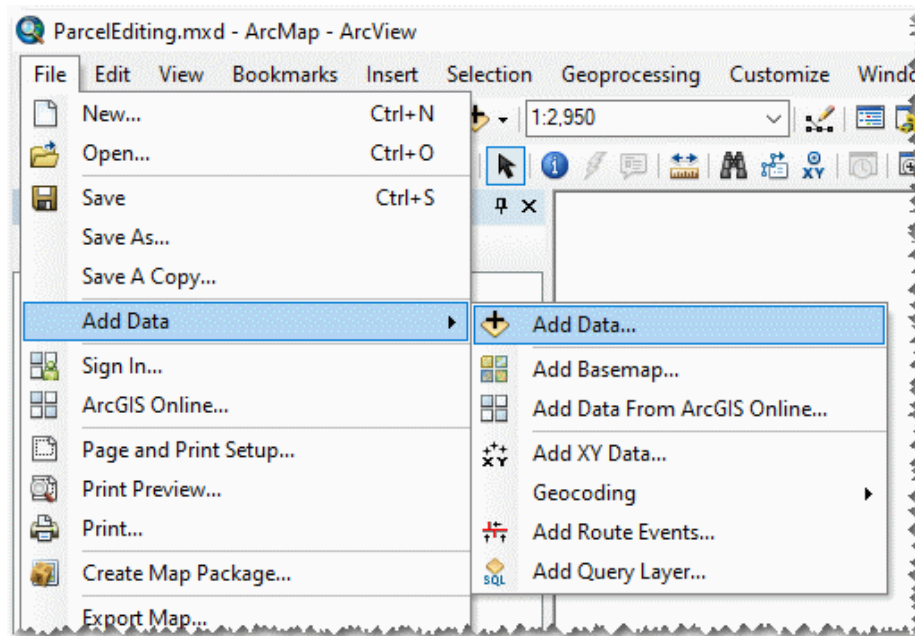
Click **OK** when ready to place the updated information into the drawing.

11. Activate the GIS menu via Settings – Carlson Menus – **GIS Menu**. Issue the the GIS Data – Import GIS Data from SurvCE command and when prompted:

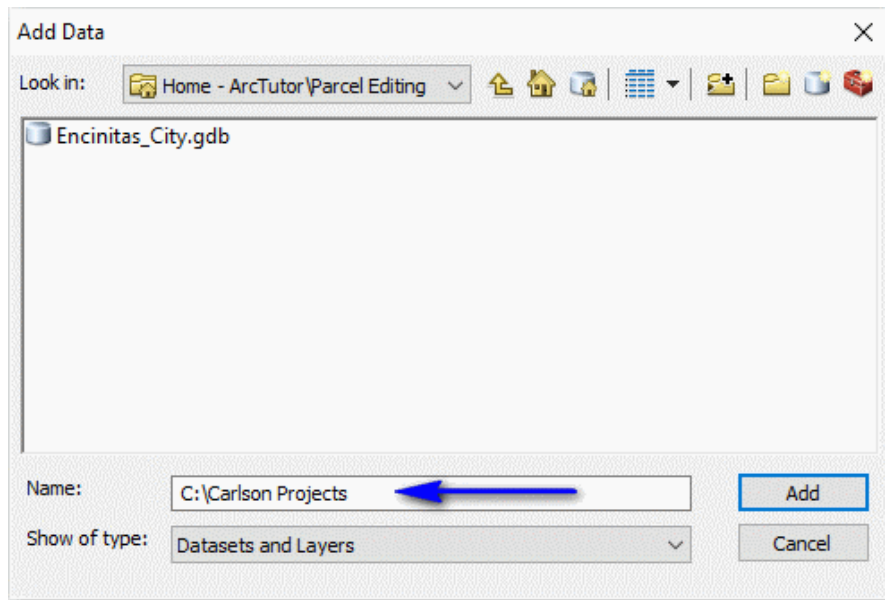


Select the *.GIS file that had been placed onto the collector (and possibly modified in the field) and click **Open** when ready. The DWG entities are packaged into an Esri MSC format. Save and exit from the DWG file.

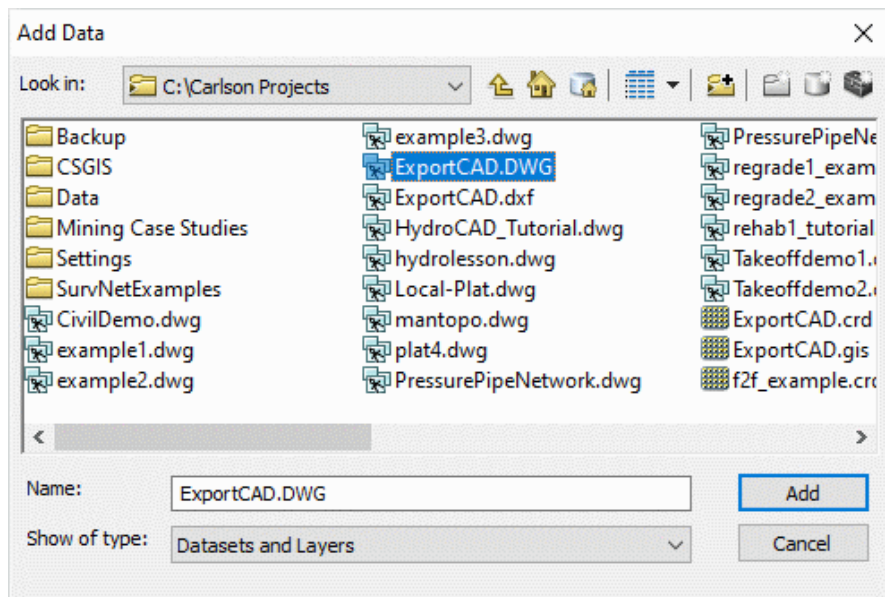
12. With the DWG saved, re-run ArcView and load the original project. Issue the File – Add Data – **Add Data** command as illustrated below:



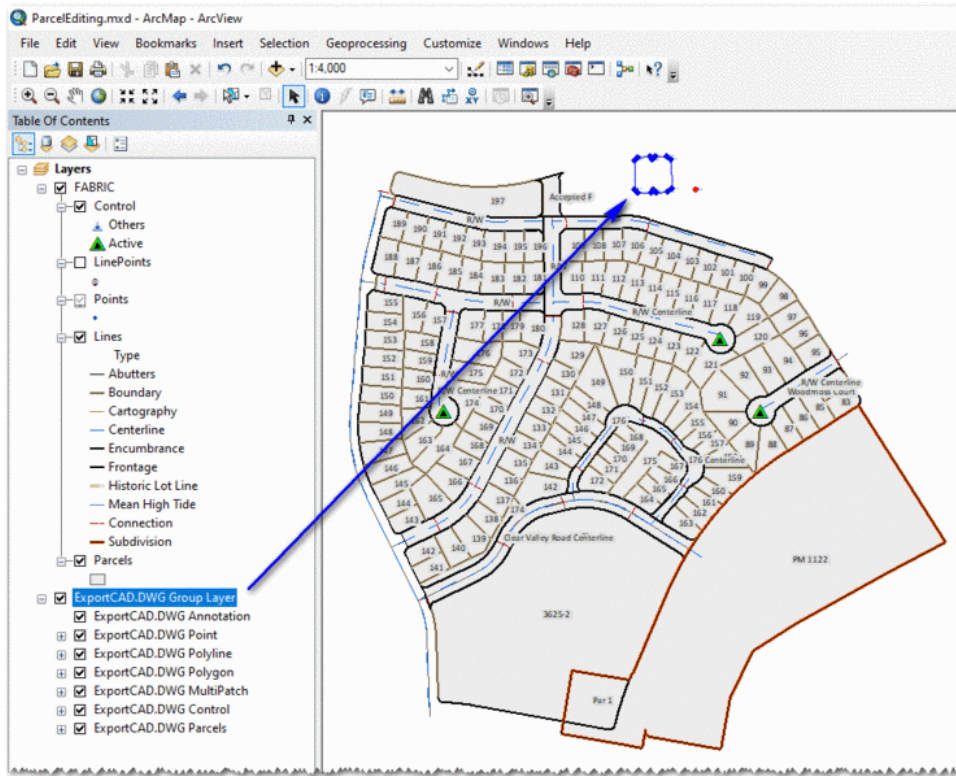
It might be generally easier to indicate the folder path where the DWG file is stored as illustrated below (click **Add** when ready):



Select the DWG file that was saved earlier and click on the **Add** button when ready:



The data is added to the ArcView project as illustrated below:



This completes the tutorial: ESRI to Office to Field and Back.

Takeoff/SiteNET CAD File From Start To Finish

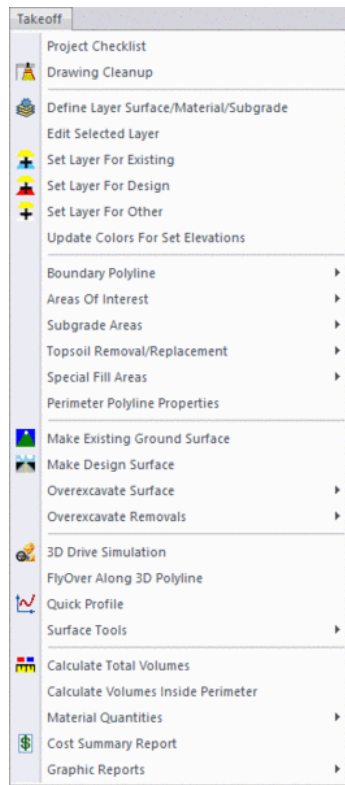
This lesson takes a drawing file from drawing cleanup to volume calculations and surface viewing.

NOTE: This tutorial is written for the Construction module. However, many of these steps can be completed using the SiteNET pull-down menu in the **Civil** module.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **Takeoffdemo1.dwg** file. Use the File – **Save As** to save a copy named **Takeoffdemo1-A.dwg**. Completing this tutorial will alter the drawing file and by renaming the file from the start, you'll keep the original file intact (allowing you to run through the tutorial a second time if desired). This is also a good practice to keep when working on drawings from 3rd parties.

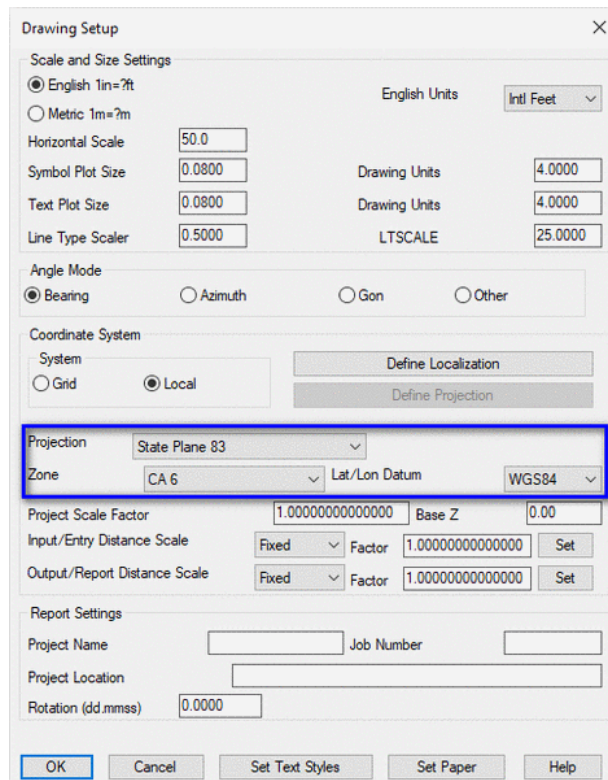
2. Activate the Construction menu via Settings – Carlson Menus – **Construction Menu**. Now we can begin to process this drawing. The main Takeoff commands are listed in processing sequence in the Takeoff (or SiteNET in the Civil module) pull-down menu:



Many of these commands are also grouped as icons in the **Takeoff General** toolbar shown here:

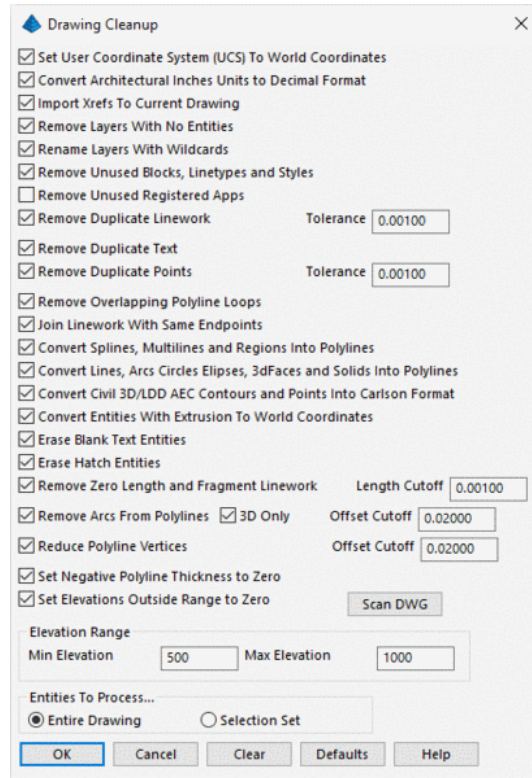


- As is often the case with construction-oriented projects, it is desirable to establish the drawing in its correct geo-location so the site can be constructed through the use of GPS-assisted machine control applications. Issue the Settings – Drawing Setup command to set the values shown below and click **OK** when ready:



- Typically, drawings have lots of drafting fixes that must be done before the surfaces can be modeled with any

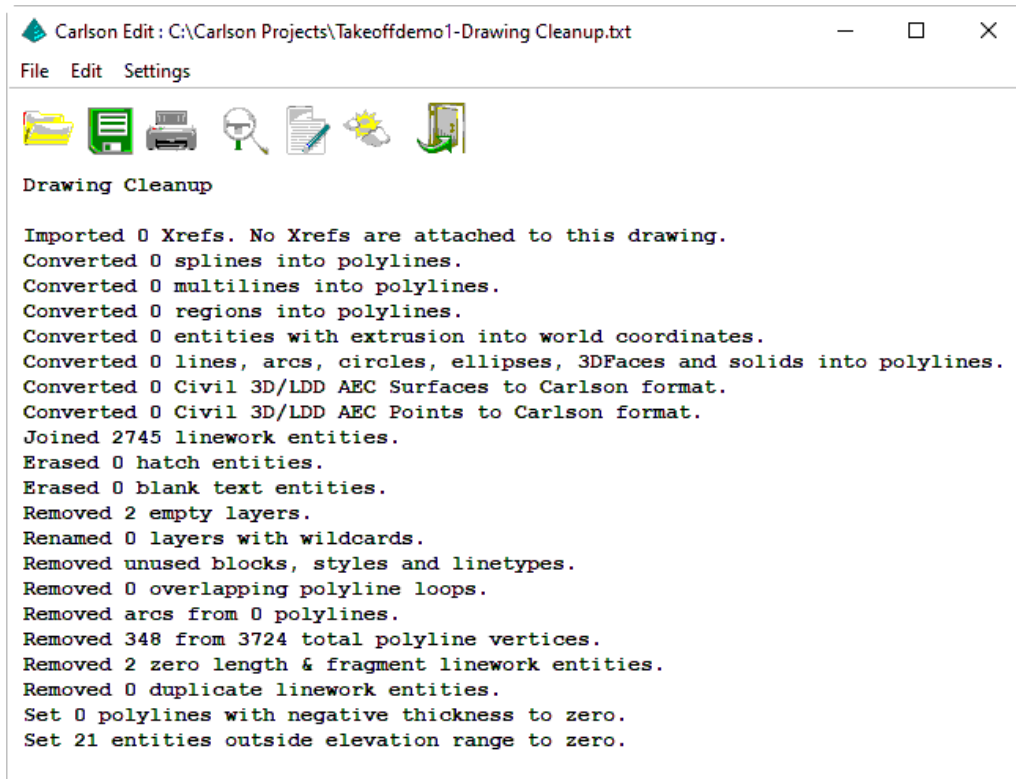
accuracy. Issue the Takeoff – Drawing Cleanup to display the dialog box shown below:



This command will apply the selected cleanup function(s) on the drawing to help automate the cleanup. Here's a brief description of the most important of these functions:

- **Remove Layers With No Entities:** Drawings often have lots of layers. This routine removes layers that have no entities in the drawing so that we don't have to deal with them.
- **Join Linework With Same Endpoints:** This routine will take linework that is broken into multiple segments and join them into a single linework entity. For example, it will join together broken segments of a contour polyline into a single polyline.
- **Reduce Polyline Vertices:** This routine removes extra vertices from polylines as long as the removing does not shift the polyline more than the specified *Offset Cutoff* distance. This will reduce the size and complexity of the drawing.
- **Set Elevation Outside Range To Zero:** In case the drawing contains entities that are outside the range of valid elevations for the site, this routine will set them to zero elevation. The program treats zero elevations as "no elevation" and modeling will filter out these zero elevation entities.

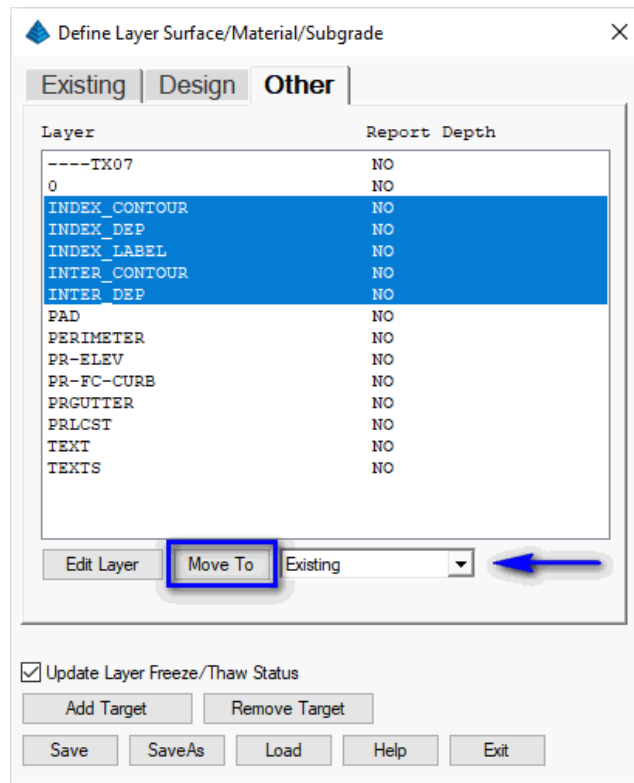
For this site, the elevations are around 800. So let's set the **Min Elevation** and **Max Elevation** fields to the values shown above. The cleanup will set any entities outside this elevation range to 0 (zero). With other Takeoff functions, we can later assign proper elevations to any of these zero elevation entities that need to be used in modeling. Set the remaining values as shown above and click **OK** when ready. When the cleanup is done, the program will show a report of the cleanup results. Click the **Exit** (doorway) button to exit the Standard Report Viewer:



5. Our next task will be to specify the layer(s) in the drawing which contain *existing* site conditions. Takeoff uses the layer name(s) to categorize data into one of three general categories for the purpose of building surface models:

- **Existing** - Used to model initial/pre-construction site conditions.
- **Design** - Used to model proposed/post-construction site conditions.
- **Other** - Used as a collection of unknown/irrelevant site conditions (text notes, hatches, *etc*).

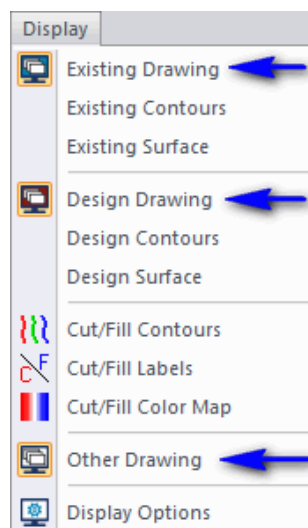
These surfaces are referred to as the "Target" surfaces. By default, everything starts in the *Other* classification. Use the Takeoff – Define Layer Surface/Material/Subgrade command to display a dialog box similar to that shown below:



To switch between lists, pick the tabs at the top of the dialog. In this drawing, all the contours are for the existing ground surface. In the layer list, all the layers that start with **INDEX** and **INTER** are for these existing ground contours. Select/highlight these layers and then click the **Move To** button with the **Existing** target as shown above (to highlight multiple layers at a time use standard Windows Ctrl+click and/or Shift+click functionality).

Next move the layer names that start with **PR** (for *proposed*) to the **Design** target by highlighting these layers and clicking **Move To** with the **Design** target selected. Also move the layer **PAD** to Design. Finish by clicking the **Save** button to save the changes and then pick **Exit**.

6. In the Display menu, you can turn on/off whether to display layer targets by using the Existing Drawing, Design Drawing and Other Drawing commands:

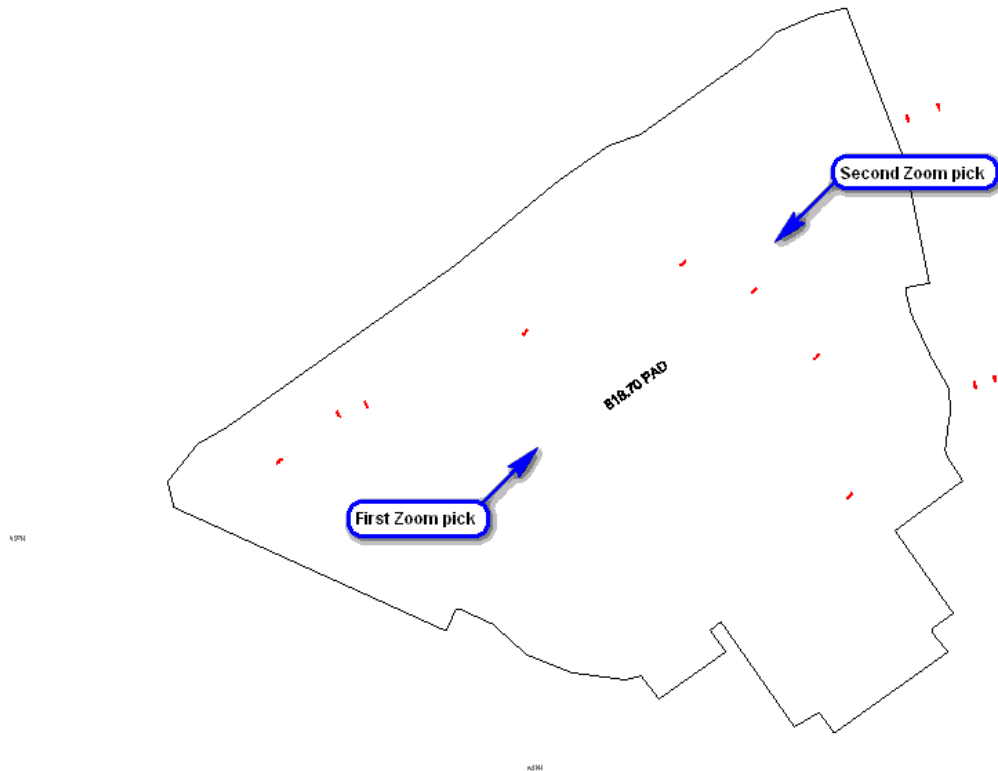


Practice turning on/off the **Existing Drawing**, **Design Drawing** and **Other Drawing**. When only *Existing Drawing* is on, you should see just the contours. When only *Design Drawing* is on, you should see just

the design polylines and leader labels. When only *Other Drawing* is on, you should see the entities that are assigned to neither Existing nor Design. Set this condition as active:

- Existing is **Off**
- Design is **Off**
- Other is **On**

to resemble the drawing as shown below:



Some of these layers we *do* want to assign to the Existing and Design targets. To better see the entities, use the View – **Window** command and pick two points that make a window around the entities as shown above. Once zoomed in, you can see a text label of **818.70 PAD** which is for the **Design** target. Other labels **817.00**, **818.00**, *etc.*, are contour labels for the Existing contours. There are a few commands in the Inquiry menu to find out the layer names for these entities and two quick ones are:

- **List**
- **Layer ID**

Let's run the Inquiry – Layer ID command and when prompted:

Pick entity to read layer: pick the **818.70 PAD** label (the entity is reported as being on the **—TX07** layer)

Pick entity to read layer: pick the **818.00** label (the entity is reported as being on the **TEXTS** layer)

Pick entity to read layer: press Enter

With this information known, we could return to the **Define Layer Surface/Material/Subgrade** but let's explore the use of organizing the data through a visual screen-pick. Issue the Takeoff – Set Layer for Design command and when prompted:

Select entities for design surface.

[FILter]/<Select entities>: pick the **818.70 PAD** label

[FILter]/<Select entities>: press Enter

The layer of the selected entity is migrated to the Design classification and its visibility set consistent

with that target. Issue the Takeoff – Set Layer for Existing command and when prompted:

Select entities for existing surface.

[FILTER]/<Select entities>: pick the **818.00** label

[FILTER]/<Select entities>: press Enter

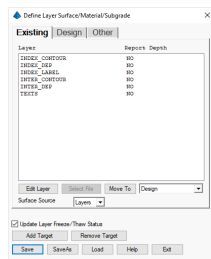
The layer of the selected entity is migrated to the Existing classification and its visibility set consistent with that target.

NOTE: To short-cut these commands, explore the use of the **Takeoff/SiteNET Display** toolbar as shown below:

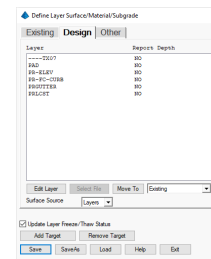


With this work complete, we can return the drawing view back by running View – **Ex-tents** command. Let's validate our layer classifications. Re-issue the Takeoff – **Define Layer Surface/Material/Subgrade** command and visit each tab to the values suggested below:

Existing Layer Targets



Design Layer Targets



Other Layer Targets



Takeoff1 Layer Targets

Check that your Layer Surfaces match the three lists shown above and then click **Save** and **Exit**.

7. Besides assigning target surfaces by layer, layers are also used to define material names and/or subgrades depths. By assigning material names and depths to layers, the volume, area, length and count for entities on these layers can be reported. Also the depth is used to vertically adjust the designated surface (subgrade depths in the Existing target would represent potential material to be demolished). The polylines used for subgrade depths must be closed polylines. Takeoff supports nested subgrade polylines for exclusion areas such as parking lot islands by counting how many subgrade polylines surround an area. If the number is odd, then the area is inside the subgrade. Otherwise the area is not part of the subgrade. Use the Takeoff – Display menu commands to set this condition as active:

- Existing is **Off**
- Design is **On**
- Other is **Off**

First, we need to know the layer names for our subgrades. Re-run Inquiry – **Layer ID** command and when prompted:

Pick entity to read layer: pick the large pad polyline (it reports the **PAD** layer)

Pick entity to read layer: pick the curb polyline (it reports the **PR-FC-CURB** layer)

Pick entity to read layer: press Enter

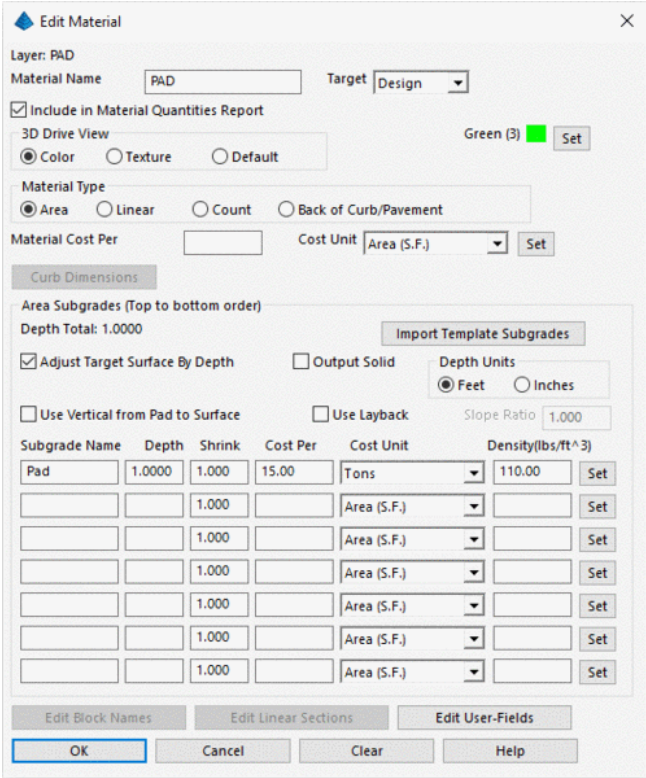
Next we need to make sure that these polylines are closed. In this example, the outside curb polyline is open at the top. To close the polylines, run Edit – Polyline Utilities – Edit Polyline – Close Polylines and when prompted:

Select Polylines to set closed.

[FILTER]/<Select entities>: pick each of the pad and curb polylines (six in all) and press Enter when done selecting

Re-issue the Takeoff – **Define Layer Target/Material/Subgrade** command and activate the **Design**

tab. Select/activate the **PAD** layer and click the **Edit Layer** button. A dialog appears for defining the *Pad* material properties. Set the values as shown below and click **OK** when ready:



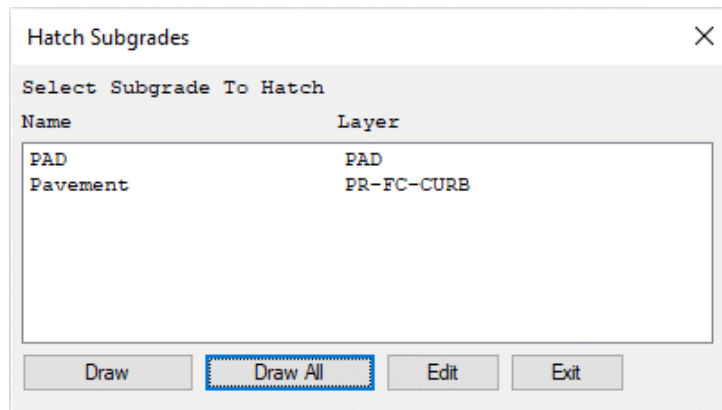
NOTE: Calculating cost is an optional feature. For this example, we cited a Cost of "\$15" per "Ton" with a density of "110."

Next pick layer **PR-FC-CURB** and choose **Edit Layer**. Set the values as shown below and click **OK** when ready:



Review the summary depths of the pay materials we just modified. Click the **Exit** button and indicate **Yes** to save changes.

- Now let's visually verify the subgrade areas. Issue the Takeoff – Subgrade Areas – Hatch Subgrade Areas to display the dialog box below:

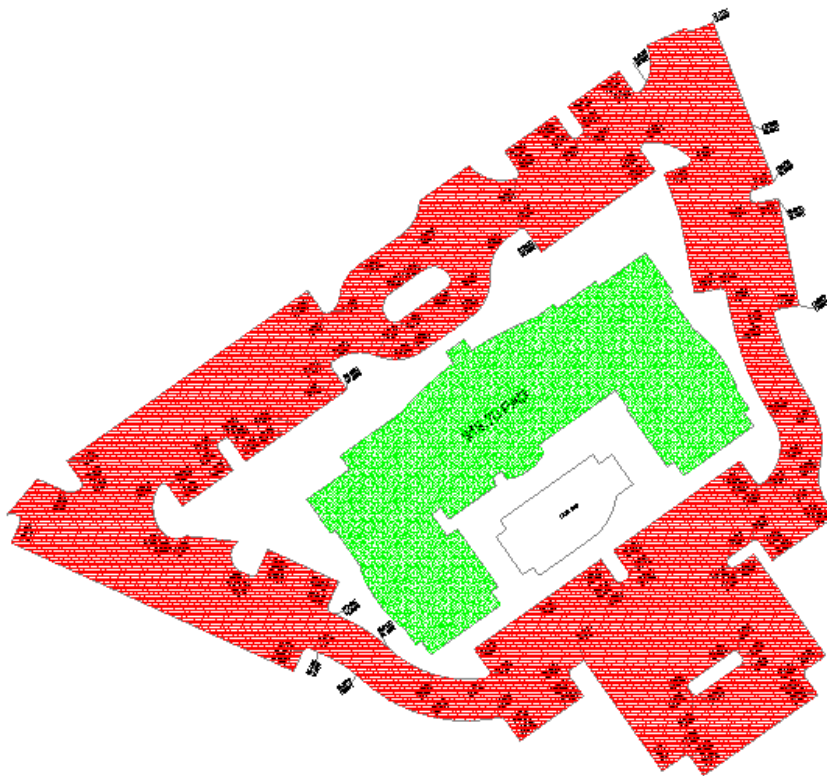


Click the **Draw All** button. The resulting hatch areas show where the subgrade is applied. Notice how the islands are not hatched because they are curb polylines that are already inside another curb polyline. Also note that the smaller pad area is not hatched because this polyline layer is different from the bigger pad polyline.

Optional: Re-run the **Hatch Subgrade Areas** command, select a desired subgrade layer and click the **Edit** and attempt to match the look of the image below through the following settings:

Subgrade	Hatch Pattern	Color
PAD	Conc	Green
Pavement	Asphalt	Red

Subgrade Hatch Settings



When finished viewing the subgrade areas, issue the Takeoff – Subgrade Areas – Erase Subgrade Hatches command.

9. Takeoff will model the existing ground and design surfaces based on points, lines and polylines with elevation. It is essential for these drawing entities to have correct elevations in order to get correct surface models. Often the provided drawings will have the drawing entities at elevation of zero with text labels indicating the true elevation. Takeoff has many tools for assigning elevations to these entities. To help visualize which entities need to be assigned elevation, Takeoff will color entities at zero elevation in grey (as established under Settings – Carlson Configure – Takeoff/SiteNET Settings). As entities get assigned elevation, they return to their original color. This elevation coloring is applied to layers that have been assigned to the Existing or Design surfaces.

Let's start by working on the Existing surface. To isolate the existing entities, use the Takeoff – Display menu commands to set this condition as active:

- Existing is **On**
- Design is **Off**
- Other is **Off**

Let's explore real-time elevations through the use of the Inquiry – Drawing Inspector command. Once activated, right+click and make sure **Display Elevation** is checked (you might also find it valuable to have the *Display Layer* and *Display Entity* options checked). Next, hover over a contour polyline and *Drawing Inspector* will report the elevation. Other info may be displayed as well depending on what is selected from the right+click options:

INTER_CONTOUR
816.000
LWPOLYLINE

In this example, the existing ground surface is defined by just contour polylines and these polylines already have elevation. So there are no changes needed for preparing the existing surface entities. If the contour polylines were at zero elevation, then you could use the Elevate – Assign Contour Elevation menu commands (such as Multiple in Series).

10. Next let's prepare the design surface. To isolate the existing entities, use the Takeoff – Display menu commands to set this condition as active:
 - Existing is **Off**

- Design is **On**
- Other is **Off**

Notice that all the design linework has a grey color because it is at zero elevation. Run the Inquiry – **List Elevation** command and when prompted:

List Elevation [Options/<Select Entity>]: pick the main pad polyline (its elevation of 0 is confirmed)

Select Entity (Enter to end): press Enter

To set the pad polyline elevation, run the Elevate – Set Polyline To Elevation command and when prompted:

Enter/<Select text or linework of elevation>: pick the **818.70 PAD** text (the non-numerical information is stripped and the numeric value is reported)

Enter elevation to add to text values <0.0>: press Enter

Select entities for elevation change.

[FILter]/<Select entities>: pick the larger pad polyline

[FILter]/<Select entities>: press Enter

Next, let's set the elevation of the smaller pad under the main pad. Use the View – **Window** command to zoom in around the smaller pad so that we can read the text label. The label of **17.56** is short for *817.56*. In this example, the *800* was dropped from many of the elevation labels to save on label clutter. Run the **Set Polyline To Elevation** again. This time when prompted:

Enter/<Select text or linework of elevation>: pick the **17.56** text (any non-numerical information is stripped and the numeric value is reported)

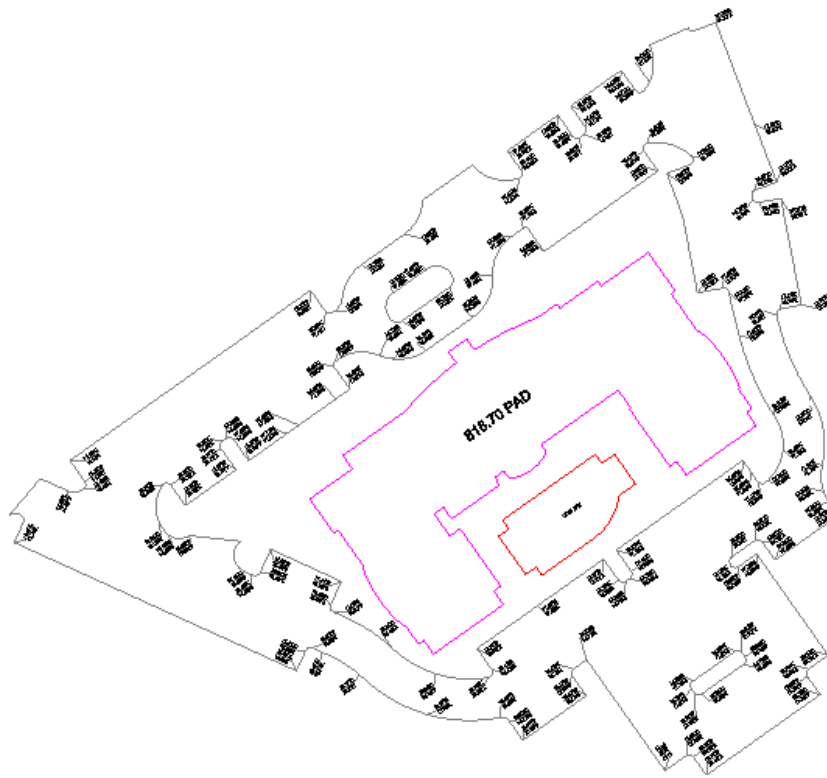
Enter elevation to add to text values <0.0>: type **800** and press Enter

Select entities for elevation change.

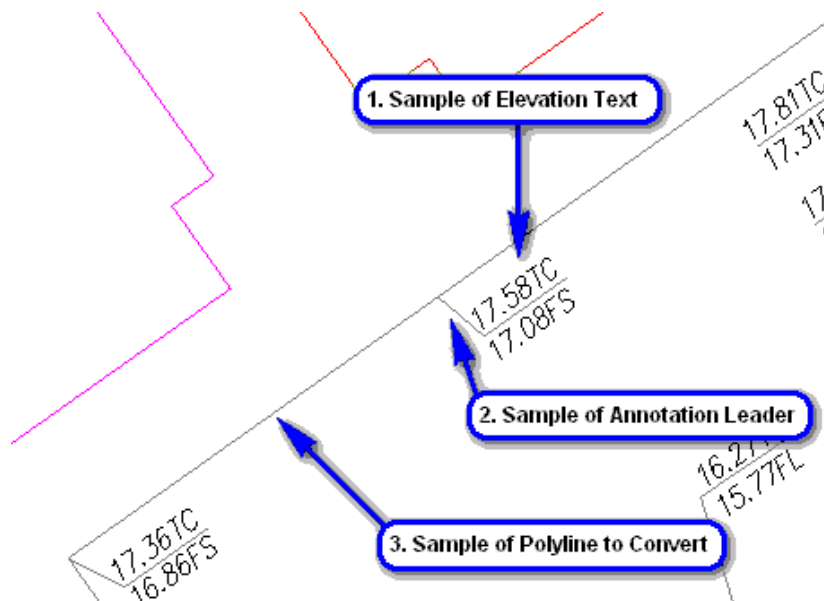
[FILter]/<Select entities>: pick the smaller pad polyline

[FILter]/<Select entities>: press Enter

Then run View – Zoom – **Previous** (keyin short-cut **ZP** to get back to the full view of the site (color changed for clarity):



Finally, we need to set the elevations for the curb polylines. First, use the View – **Window** to zoom in around some of the curb labels below the smaller pad as illustrated below:



Next, issue the Elevate – 2D to 3D Polyline – Text With Leader command. This command will assign the elevations from the labels to the polylines by following the label leader to find the position on the polyline. For polyline vertices without elevation labels, the elevations will be interpolated from the other labels. When prompted:

Options/Select sample of elevation text: pick one of the labels with a "TC" suffix as cited in **item 1** above and press Enter

Select sample of an annotation leader: pick the associated leader line as cited in **item 2** above and press Enter

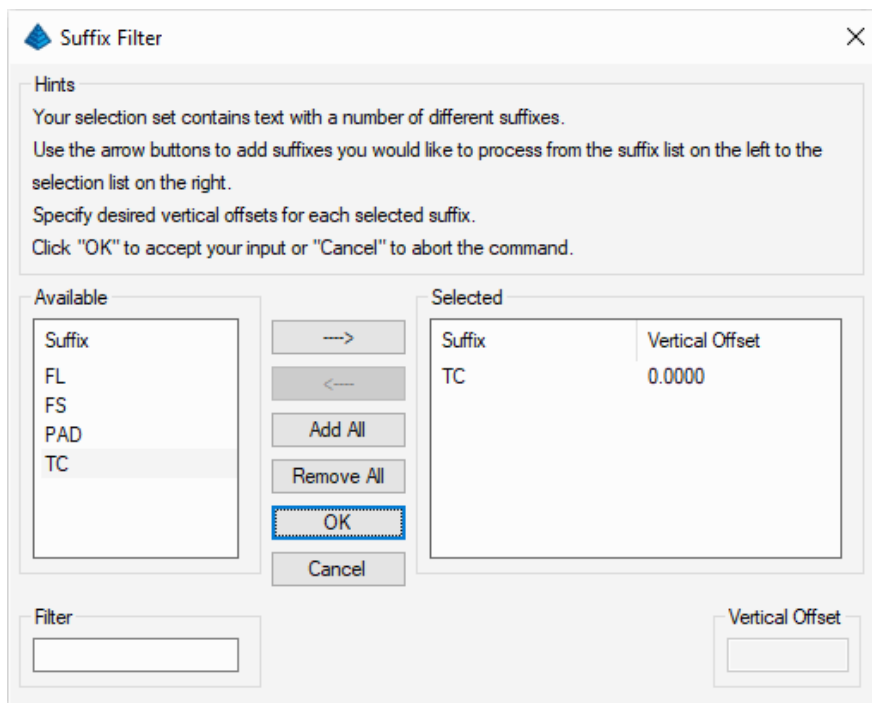
Select sample of a polyline to convert: select a sample of a polyline for conversion as cited in **item 3** above and press Enter

Select polylines to convert, leaders and elevation labels to process.

[FILter]/<Select entities>: type **all** and press Enter twice

Enter elevation to add to label values <0.00>: type **800** above and press Enter

Next, a dialog appears for selecting which label names to use:



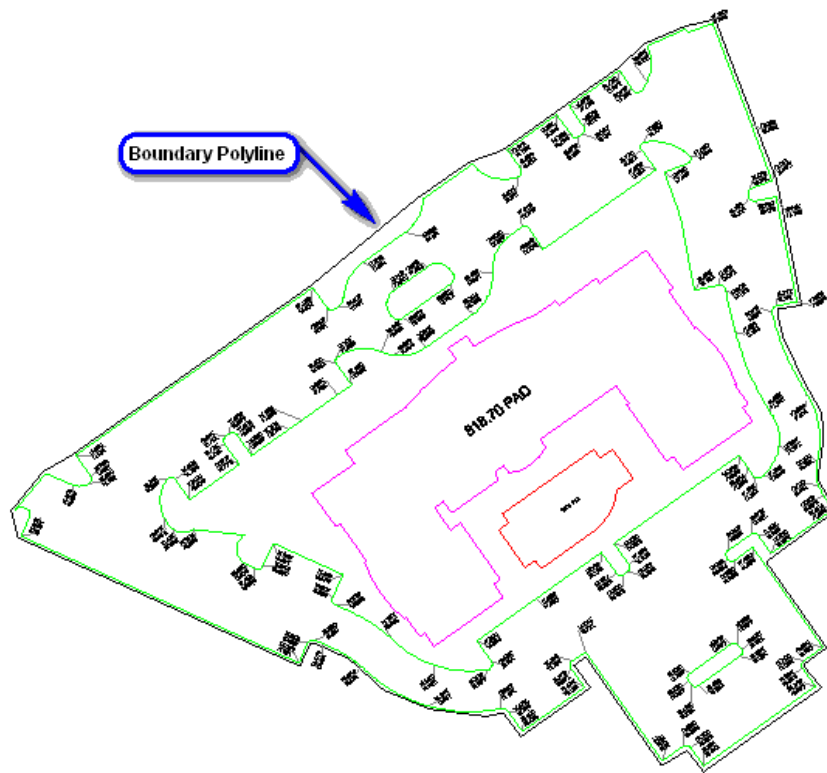
When Takeoff detects different text labels within the elevation labels, you need to choose which ones to process. In this case, we only want the labels with **TC** (Top of Curb), so highlight **TC** and click the **—>** (Add) button and then click **OK** when ready. All the curb polylines now have elevations and should also have colors because the elevations are assigned. Finally, issue the View – **Extents** to return to the full site view.

11. The limits of the site are defined by a closed polyline. This polyline is used as the boundary for the models and the volumes. In this example, there is a closed polyline on the **PERIMETER** which is currently part of the *Other* target. Use the Takeoff – Display menu commands to set this condition as active:
 - Existing is **Off**
 - Design is **On**
 - Other is **On**

Issue the run Takeoff – Boundary Polyline – Set Boundary Polyline command and when prompted:

Select boundary polylines.

[FILter]/<Select entities>: pick the perimeter polyline as shown below:



This selected polyline is now set as the boundary polyline for the rest of the Takeoff routines.

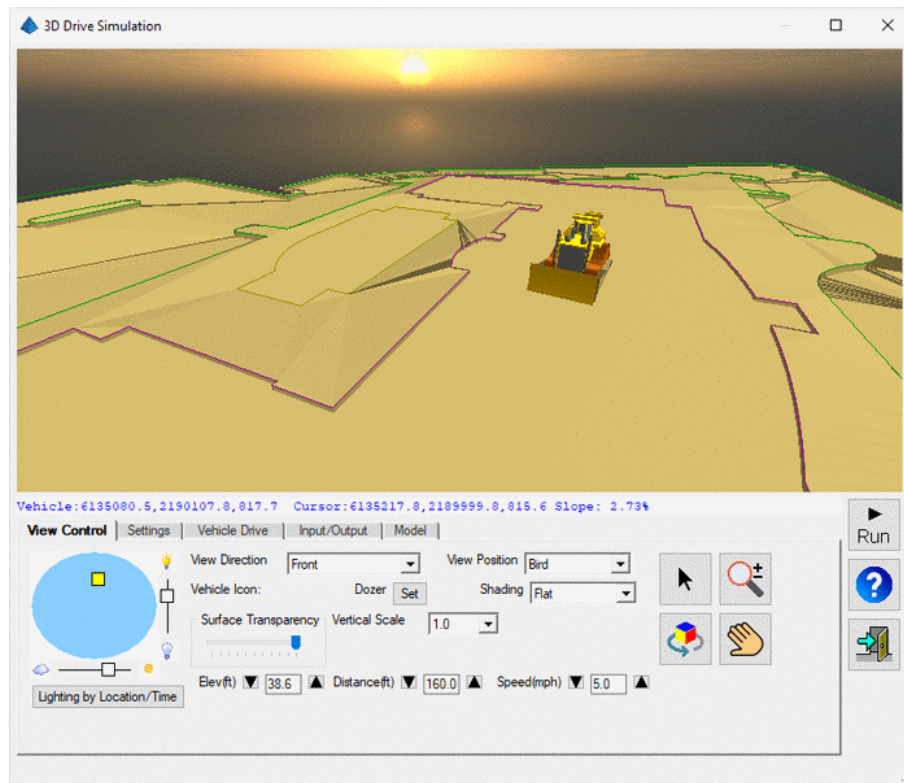
12. To calculate volumes, the program needs two surfaces:

- existing ground, and,
- design considerations

These surfaces are modeled by triangulation. With the preparation of the previous steps, we're now ready to make the models. The drawing entities have been cleaned up, assigned elevations and assigned target surfaces by layer. Making each model is now a one step process:

- To make the Existing Ground surface, issue the Takeoff – Make Existing Ground Surface. The program will process the entities (regardless of their screen visibility) and make the triangulation surface.
- To make the Design Ground surface, issue the Takeoff – Make Design Surface.

13. As a visual check that the Design Surface was formed correctly, let's explore the site in a 3D isometric view. Run the Takeoff – 3D Drive Simulation command to display a dialog box similar to that below:

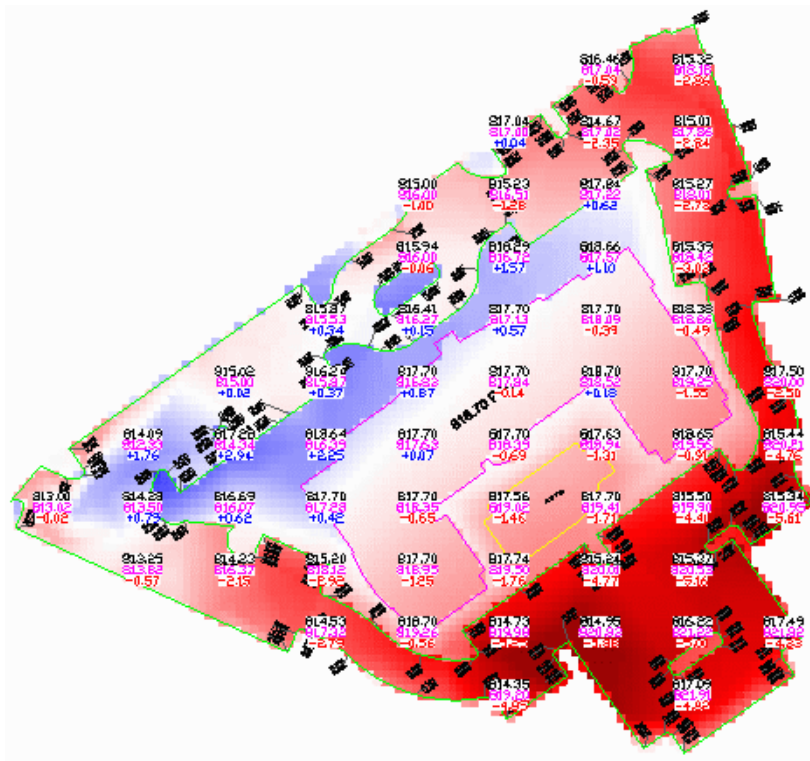


This routine shows a 3D view of the site and allows you to "drive" around through the use of the arrow keys on most standard keyboards. This is a good way to check that the surface triangulated correctly. We want to make sure that there are no elevation spikes and that the subgrade depths are modeled. To drive the site:

- choose a *View Direction*, and,
- choose a *View Position*, and,
- choose a *Vehicle*, and,
- choose a *Speed*

Then click the **Run** button and use the arrow keys to turn. Click the **Stop** button to pause the moving. Experiment with the various **Shading** options and "tabs" of other settings to achieve an appealing visualization style. Click the **Exit** (doorway) button when complete.

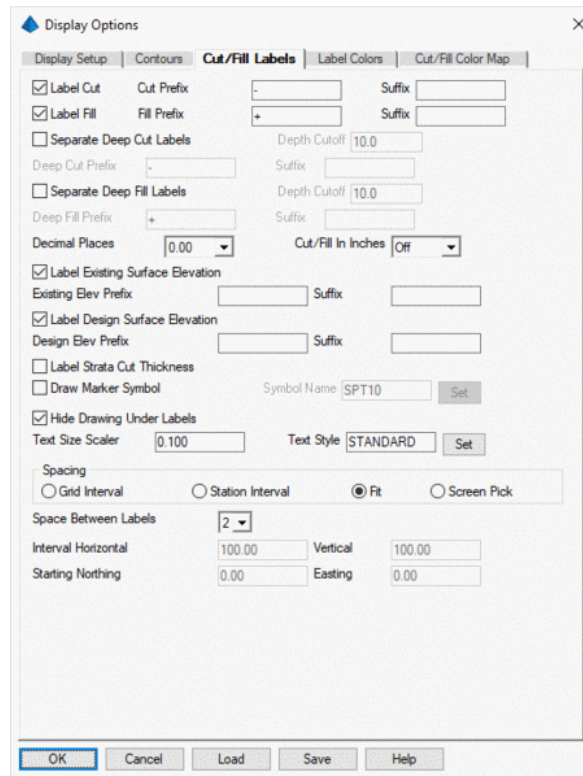
14. Cut/Fill color maps can be used for a visual output of the site cut/fill areas and also serves as another check that the models are correct. Issue the Display – Cut/Fill Color Map command to display an image similar to that shown below (also shown with *Cut/Fill Labels*):



Under default conditions:

- *Cut* areas are drawn in different shades of red (deeper colors represent deeper cut), and,
- *Fill* areas are drawn in blue, and,
- regions in Cut/Fill neutral areas tend to shade white

Continuing, issue the Display – Cut/Fill Labels to more closely resemble the image above. To change the size, color, spacing and other label options, run the Display – Display Options and go to **Cut/Fill Labels** tab shown below:



The resolution of the color blocks in the map can be changed through the **Spacing** group of controls. Turn off the *Cut/Fill Labels* by re-issuing the Display – **Cut/Fill Labels** command to uncheck it.

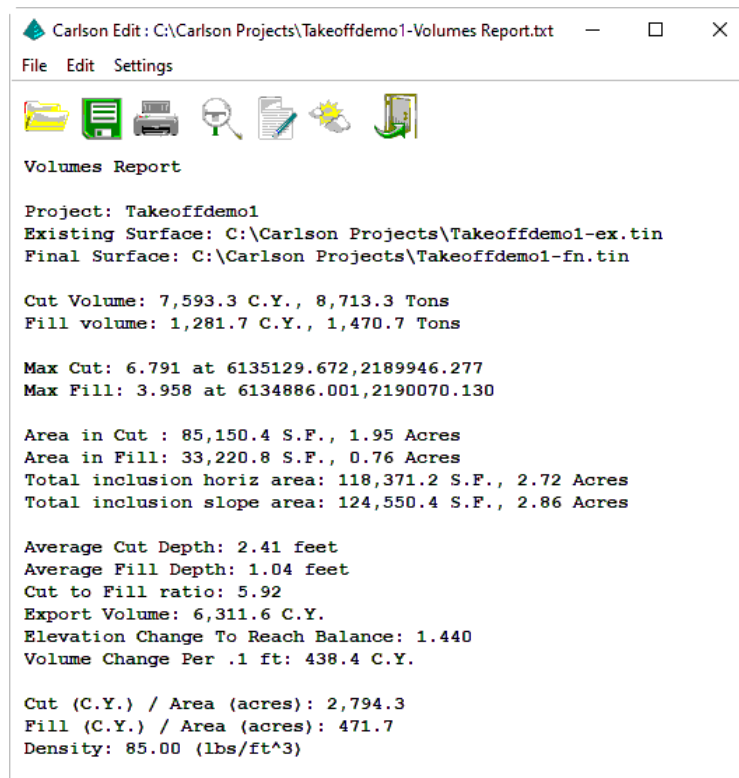
15. To calculate volumes, run the Takeoff – Calculate Total Volumes command to display a dialog box similar to that below:

The screenshot shows a dialog box titled "Volume Options" with a close button (X) in the top right corner. The dialog contains the following fields and controls:

- Cut Swell Factor: 1.000
- Fill Shrink Factor: 1.000
- Report Output: Standard Report Viewer (dropdown)
- Report Units: English (dropdown)
- Report Order: Total on top (dropdown)
- Report Tons: Density(lbs/ft^3) 85.000
- Report Cut Depth Zones: Interval 10.000
- Report Fill Depth Zones: Interval 10.000
- Report Cut/Fill in Color: Page Settings (button)
- Balance Cut/Fill
- Balance Tolerance (CY): 10.00
- Import Volume (CY): 0.00
- Export Volume (CY): 0.00
- Report Title: Volumes Report
- Project: Takeoffdemo1
- Comments: (empty text box)

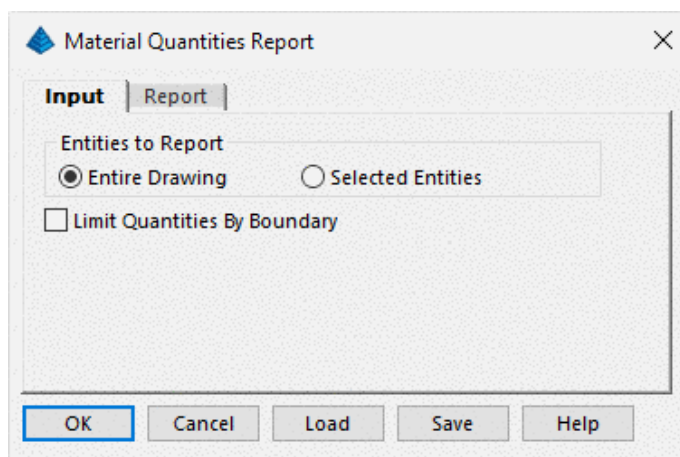
At the bottom of the dialog are three buttons: OK, Cancel, and Help.

The settings the *Cut Swell* and *Fill Shrink* factors get multiplied into the cut/fill volumes. Set these factors as desired and click **OK** when ready. The results are displayed similar to that shown below:



Click the **Exit** (doorway) icon when ready. Run the command again and change the **Report Output** mode to see the volumes in different formats like PDF or Excel.

16. To report the material (paving) quantities, issue the Takeoff – Material Quantities – Standard Report routine to display the dialog box below:



Specify the options as shown above (feel free to explore the options in the *Report* tab) and click **OK** when ready. The report includes (among other this) the:

- *Count*, and,
- *Length*, and,
- *Area*, and,
- *Volume*

for each type of material that was assigned for reporting in the previously discussed Takeoff – **Define Layer Target/Material/Subgrade** command and generates a report similar to that below:

Carlson Edit : C:\Carlson Projects\Takeoffdemo1-Material Quantities.txt

File Edit Settings

Material Quantities

Breakout Quantities

Material	Count	Length (Ft)	Area (S.F.)	Volume (C.Y.)	Shrink	Depth	Cost
PAD	1	927.6	22,331.6			1.00ft	
Pad				827.1	1.000	1.00ft	\$18,423.60
Pavement	4	3,284.1	63,106.3			1.50ft	
Pavement			63,106.3	3,505.9	1.000	1.50ft	\$674,448.96
				45.6		0.375sf	
Total							\$692,872.56

Sub-Total Quantities

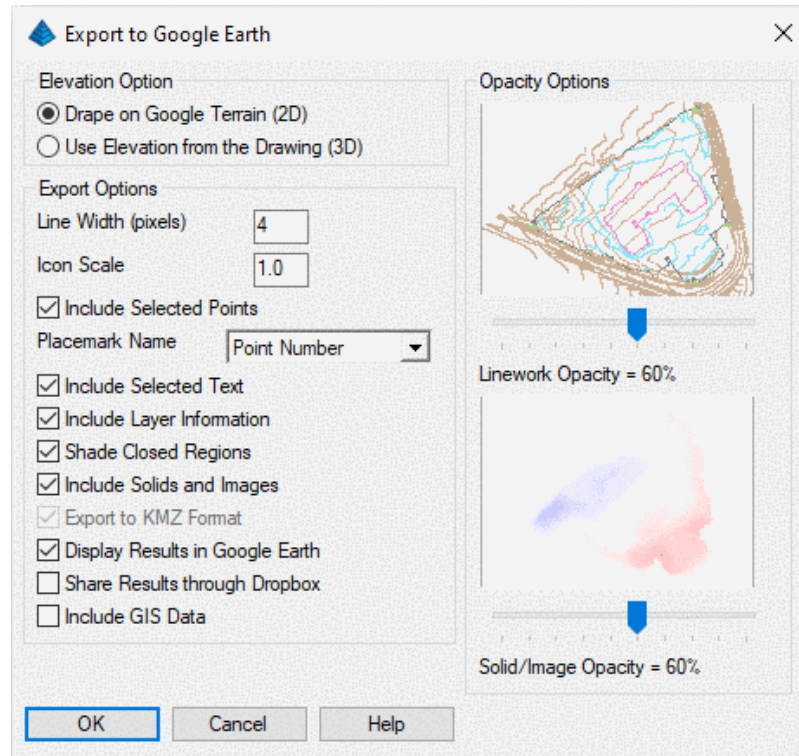
Material	Count	Length (Ft)	Area (S.F.)	Volume (C.Y.)	Tons	Cost
Pad	1	927.6	22,331.6	827.1	1,228.2	\$18,423.60
Pavement	4	3,284.1	63,106.3	3,505.9	4,496.3	\$674,448.96
Total	9	7,495.8	85,438.0	4,378.6	10,220.9	\$692,872.56

Material Parameters

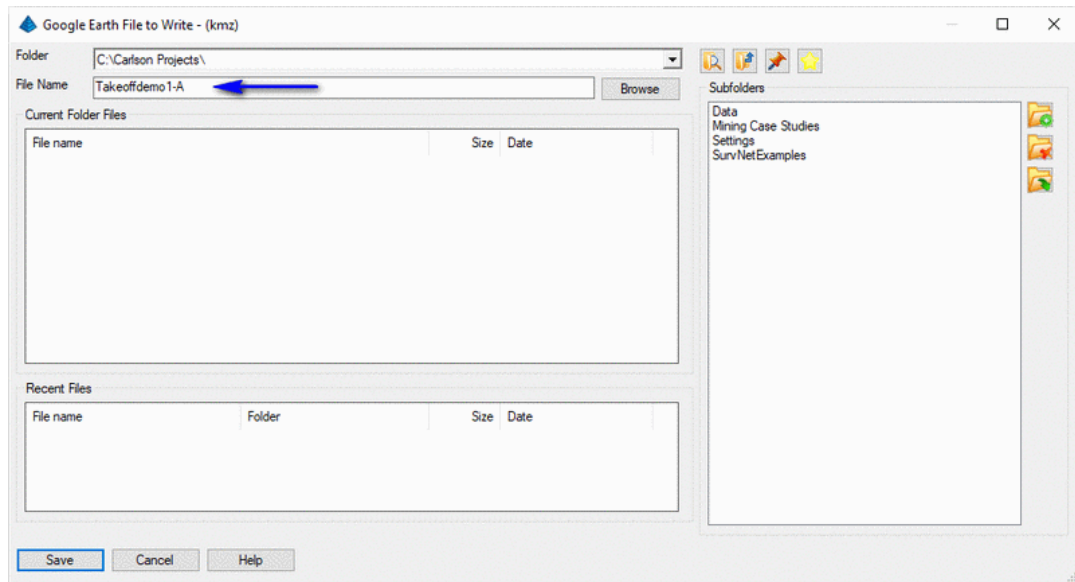
Material	Unit Cost	Shrink	Density (lbs/ft^3)
		1.000	0.00
Pad	\$15.00/Ton	1.000	110.00
Pavement	\$150.00/Ton	1.000	95.00

Click the **Exit** (doorway) icon when ready. The Takeoff – Material Quantities – **Custom Report** command can be used to report these values with control of the report format and the option to export to Excel.

17. *Optional:* For computers that have Google Earth™ Pro on desktop installed, let's see how the site looks when overlaid on this application (or any other application which can display Keyhole Markup Language (KML) files). Issue the File – Export – Google Earth File command to display the dialog box below:



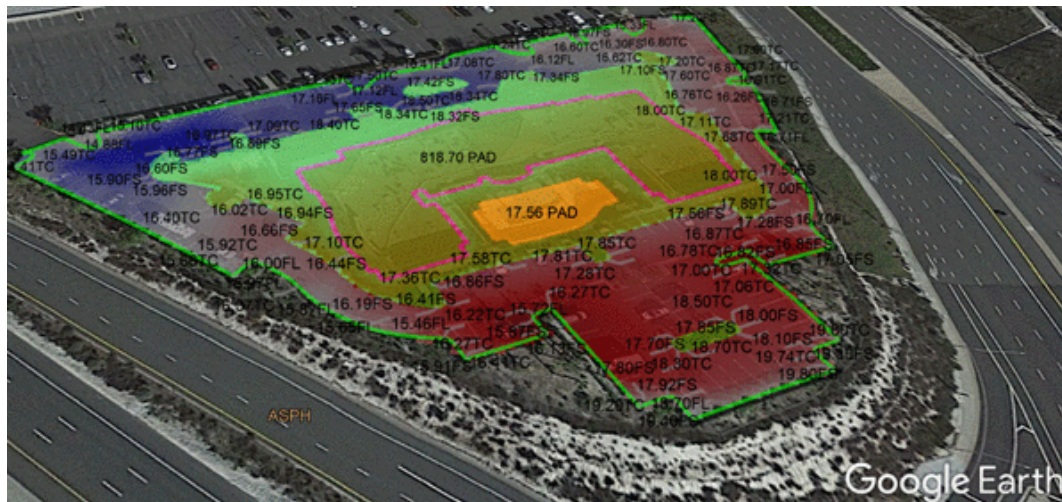
Review and set the values as suggested above and click **OK** when ready to display a dialog box similar to that shown below:



Provide the file name specified above and click the **Save** button when ready. When prompted:

Select points, polylines, text, solids, images, lines and arcs to write.
[FILter]/<Select entities>: type **all** and press Enter twice

The results are overlaid on Google Earth as illustrated below:



18. Lastly, let's remove the shaded hatch of the Cut/Fill color map through the use of the Display – **Cut/Fill Color Map** command.

This completes the tutorial: Takeoff/SiteNET CAD File From Start To Finish.

GeoTech Drillholes and Strata

This lesson creates and processes drillhole data in the GeoTech module.

1. This drillhole lesson builds on the results of the Takeoff/SiteNET Basics tutorial. Before continuing with this tutorial, complete that tutorial up through the Make Existing Ground Surface/Make Design Surface section of that tutorial. When the **Takeoff/SiteNET Basics** tutorial is completed, we're now ready to add drillholes for the drawing as shown below:



2. Activate the GeoTech menu via Settings – Carlson Menus – **GeoTech Menu**. Our initial task will be to establish a desired drillhole symbol and the default strata names. Issue the Drillhole – Define Drillhole and Strata Settings (*aka Define Drillhole*) command to display the dialog box similar to that shown below:

Drillhole and Strata Settings

Drillhole Settings

Select Symbol

Symbol Name: SPT12

Symbol Size: 10.0

Strata Definitions

Strata name	Model	Method	Density	Swell
DIRT	Elevation	InvDist-2	80.000	1.100
ROCK	Elevation	InvDist-2	165.000	1.150

Buttons: Add, Edit, Remove, Move Up, Move Down

Place Drillhole Prompts

Depth Thickness Dialog

Default Last Thickness Thickness: 1.00

Use Drillhole Surface Elevation for Existing Surface

Buttons: OK, Cancel, Load, SaveAs, Help

For this tutorial, we are interested in rock quantities and we need to define two strata:
 (a) **DIRT** (material above the rock), and,

(b) **ROCK**

Click the **Add** button to create a material strata as described below and click **OK** when ready:

Next, pick the **Add** button again. This time, fill out the dialog with a strata name of "ROCK" and use the information below to add the second strata material into the project and click **OK** when complete:

Values	Strata 1	Strata 2
Name	DIRT	ROCK
Density	80	165
Swell Factor	1.10	1.15
Model Values	By Elevation	By Elevation
Modeling Method	Inverse Distance - Power 2	Inverse Distance - Power 2

Strata Definitions

NOTE: There are three options of modeling methods:

- **Inverse Distance** will not carry trends and the calculated strata model will never be higher or lower than the original drillhole data and uses a weighted average of the drillhole data. In general, closer drillholes are weighted more than drillhole farther away:
 - **Inverse Distance - Power 2** will weigh drillholes more that are further away.
 - **Inverse Distance - Power 3** will weigh drillholes less that are further away.
- **Linear Least Squares** extrapolates trends and allows for a strata model to create new highs and lows that don't appear in the original drillhole data.

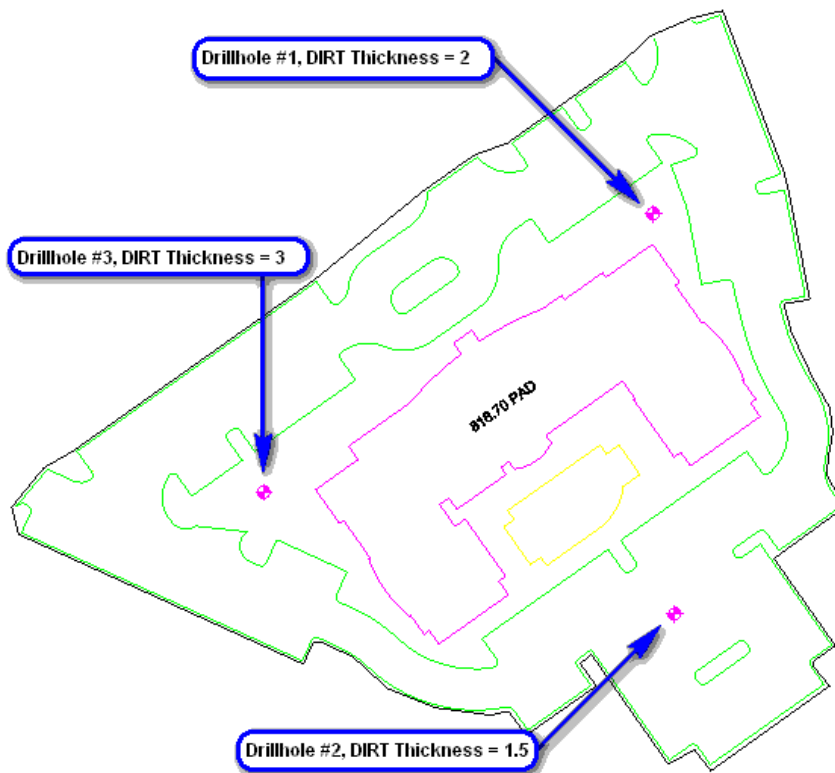
The *Strata Materials* in the main dialog need to be in top to bottom order. To change the order, highlight a strata name and use the **Move Up** or **Move Down** buttons. In this case, we want *DIRT* then *ROCK*. Click **OK** from the main dialog.

3. There are two different methods for entering drillhole data:

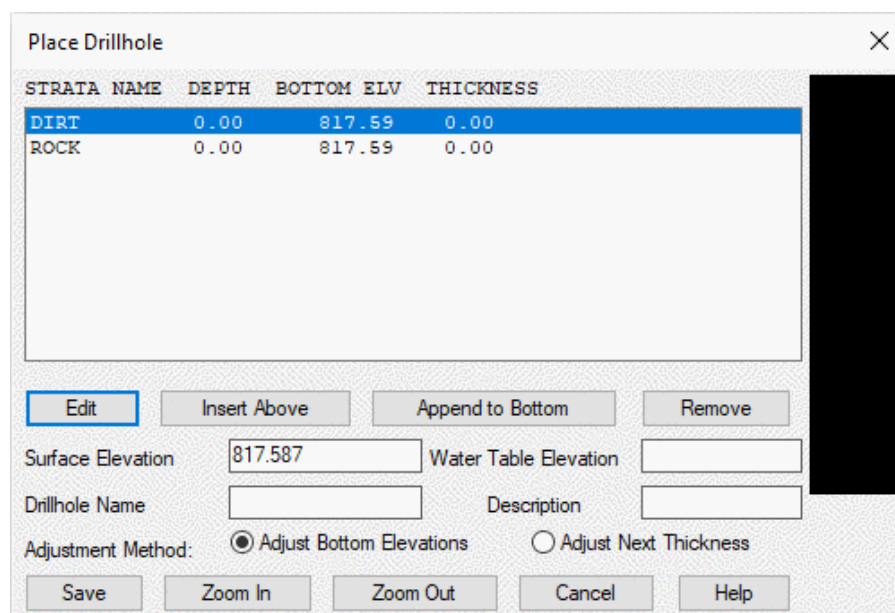
- **Drillhole Import** which reads the drillhole data from a text file. This command supports customizing the sequence of drillhole data fields to match the format of the text file. This process is discussed in the GeoTech Reports tutorial.
- **Place Drillhole** creates the drillholes at picked positions in the drawing and enters the data in a dialog.

For this tutorial, issue the Drillhole – Place Drillhole command. When prompted:

Station/<Pick Drillhole Location>: pick a location northeast of the building pad as shown below:



A dialog box similar to that shown below appears. The *Surface Elevation* is automatically filled in using the Existing Ground Surface. The *Drillhole Name*, *Water Table Elevation* and *Description* are optional. The list of strata defaults to the strata defined in **Drillhole/Strata Settings**. Each strata defaults to a thickness of zero.



Select/Highlight the **DIRT** entry and click the **Edit** button to display a secondary dialog box similar to that shown below:

Edit Strata ×
 Strata # 1
 Name:
 Thickness:
 Bottom Elev:
 Depth:
 Return Prev Strata Next Strata

For this location, indicate the DIRT strata has a *Thickness* of **2** (press the *Tab* key to see the updated values) and then click **Next Strata**.

For this example, we only know the depth to the top of rock depth and not the total rock thickness. We will treat all cut below the top of rock as ROCK strata. So we'll set the rock thickness deep enough to be lower than the Maximum Cut on site. In this case, indicate the ROCK strata has a *Thickness* of **15** (press the *Tab* key to see the updated values and then click **Return** to display a dialog box similar to that shown below:

Place Drillhole ×

STRATA NAME	DEPTH	BOTTOM ELV	THICKNESS
DIRT	2.00	815.59	2.00
ROCK	17.00	800.59	15.00

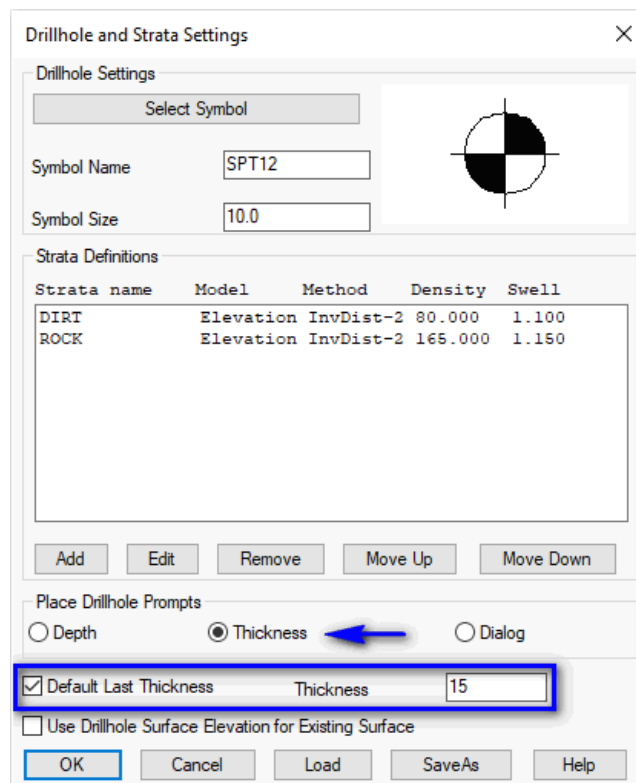
Edit Insert Above Append to Bottom Remove
 Surface Elevation: Water Table Elevation:
 Drillhole Name: Description:
 Adjustment Method: Adjust Bottom Elevations Adjust Next Thickness
 Save Zoom In Zoom Out Cancel Help

Complete the remaining values as desired and click **Save**. When prompted:

Pick Drillhole Location (Enter to end): press Enter

NOTE: If you had known the coordinates for the drillhole, you can type in the *Easting,Northing* format instead of picking on the screen.

Now let's locate two more drillholes using a different method. Return to the **Define Drillhole and Strata Settings** command and set the changes as illustrated below and click **OK** when ready:



Re-issue the **Place Drillhole** again and when prompted:

Station/<Pick Drillhole Location>: pick a position in the lower parking lot

Strata DIRT Thickness: type **1.5** and press Enter (complete any remaining items for DH2 and click **Save**)

Pick Drillhole Location (Enter to end): pick a position left of the main building

Strata DIRT Thickness: type **3** and press Enter (complete any remaining items for DH3 and click **Save**)

Pick Drillhole Location (Enter to end): press Enter

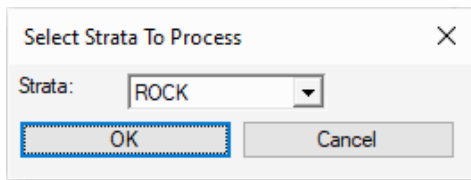
Now that the drillholes are in the drawing, we can explore the creation of sub-surface strata models.

4. To make the strata triangulation surfaces, run the StrataCalc – Make Strata Surfaces command. There are no prompts for this routine. The strata surfaces are modeled from the drillholes and saved with the project. The file names for the strata surfaces use the drawing name plus **-ch#** where the # is the strata sequence number. For this example, the file names will be **Takeoffdemo1-ch1** for bottom of DIRT and **Takeoffdemo1-ch2** for bottom of ROCK. Now that the strata surfaces are created, there are several Takeoff/Construction routines that will use these surfaces including (but not necessarily limited to):

- Calculate Total Volumes
- Calculate Volumes Inside Perimeter
- Define Topsoil Depths
- Cut/Fill Labels
- Surface Inspector
- Quick Profile
- Trench Network Quantities

5. Let's see where our ROCK cut locations are. Issue the StrataCalc – Draw Strata Cut Color Map command. When prompted:

Select Strata to Process: indicate the **ROCK** surface as illustrated below:



Select point for color legend (Enter for None): pick a location of your choosing (top center of the legend)

Legend size <10.0>: press Enter

Label all zones or summary [All/<Summary>]? press Enter

This command compares the design surface with the strata surface to make a cut color map of the cut depths for the strata. This command is one way to verify that the strata surfaces are modeled correctly. The result should be similar to that shown below:



Optional: Remove the Strata Cut Color Map through the StrataCalc – Erase Strata Cut Color Map command.

6. Let's explore the impact of the ROCK cut in terms of its contribution to the project volumes. Activate the Construction menu via Settings – Carlson Menus – **Construction Menu**. Next, issue the Takeoff – Calculate Total Volumes command to display a dialog box similar to that below:

Volume Options

Cut Swell Factor: 1.000

Fill Shrink Factor: 1.000

Report Output: Standard Report Viewer

Report Units: English

Report Order: Total on top

Report Tons Density(lbs/ft^3): 85.000

Report Cut Depth Zones Interval: 10.000

Report Fill Depth Zones Interval: 10.000

Report Cut/Fill in Color Page Settings

Report Strata Depth Zone Volumes: By Zone Area

Strata: ROCK

Depth Interval: 2.000

Balance Cut/Fill

Balance Tolerance [CY]: 10.00

Import Volume [CY]: 0.00

Export Volume [CY]: 0.00

Report Title: Volumes Report

Project: Takeoffdemo1

Comments:

OK Cancel Help

When strata surfaces are defined, the volume routine will separate the cut volume into the different strata. **Depth Zones** (2 ft zones in this case) can also be defined in this command. Set the values as shown above and click **OK** when ready. The resulting dirt and rock quantities are summarized in a report similar to that shown below:

```

Carlson Edit: C:\Carlson Projects\Takeoffdemo1-Volumes Report.txt
File Edit Settings
Volumes Report
Project: Takeoffdemo1
Existing Surface: C:\Carlson Projects\Takeoffdemo1-ex.tin
Final Surface: C:\Carlson Projects\Takeoffdemo1-fn.tin
DIRT cut volume: 5,407.9 C.Y., 5,309.6 Tons
DIRT Area: 85,150.4 S.F., 1.95 Acres
DIRT Max Cut: 3.517 at 6135251.597,2190031.097
DIRT Avg Cut: 1.884
ROCK cut volume: 3,078.6 C.Y., 5,963.1 Tons
ROCK Area: 44,731.4 S.F., 1.03 Acres
ROCK Max Cut: 3.710 at 6135171.185,2190019.178
ROCK Avg Cut: 1.258
Total Cut volume: 8,486.5 C.Y.
Fill volume: 1,281.7 C.Y., 1,470.7 Tons
Max Cut: 6.791 at 6135129.672,2189946.277
Max Fill: 3.958 at 6134886.001,2190070.130
Area in Cut : 85,150.4 S.F., 1.95 Acres
Area in Fill: 33,220.8 S.F., 0.76 Acres
Total inclusion horiz area: 118,371.2 S.F., 2.72 Acres
Total inclusion slope area: 124,550.4 S.F., 2.86 Acres
Average Cut Depth: 2.41 feet
Average Fill Depth: 1.04 feet
Cut to Fill ratio: 5.92
Export Volume: 7,204.8 C.Y.
Elevation Change To Reach Balance: 1.643
Volume Change Per .1 ft: 438.4 C.Y.
DIRT Swell Factor: 1.100
ROCK Swell Factor: 1.150
Cut (C.Y.) / Area (acres): 2,794.3
Fill (C.Y.) / Area (acres): 471.7
Density: 85.00 (lbs/ft^3)
DIRT Density: 80.00 (lbs/ft^3)
ROCK Density: 165.00 (lbs/ft^3)
Strata Depth Zone Volumes by Zone Area
Strata: ROCK
Zone Depth: 0.000 to 2.000
ROCK Cut volume: 24,080.21 C.F., 891.86 C.Y., 1,727.49 Tons
ROCK Area: 27,986.6 S.F., 0.64 Acres
Avg Depth: 0.860
Zone Depth: 2.000 to 4.000
ROCK Cut volume: 56,679.34 C.F., 2,099.23 C.Y., 4,066.13 Tons
ROCK Area: 16,247.3 S.F., 0.37 Acres
Avg Depth: 3.489
Zone Depth: 4.000 to 6.000
ROCK Cut volume: 2,362.18 C.F., 87.49 C.Y., 169.46 Tons
ROCK Area: 497.5 S.F., 0.01 Acres
Avg Depth: 4.748

```

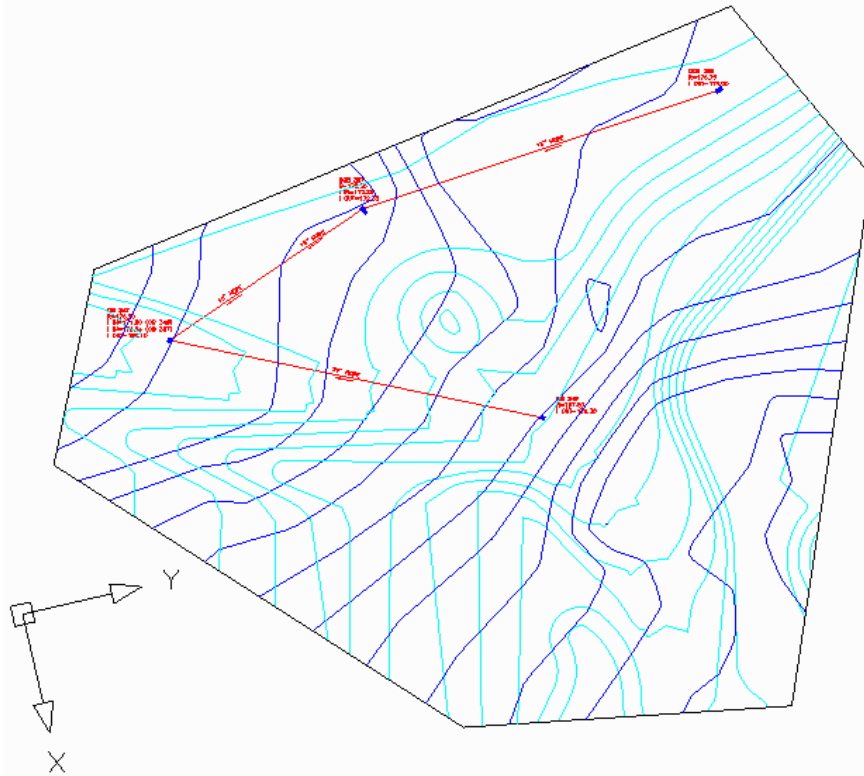
Review and click the **Exit** (Doorway) button when complete.

Takeoff Trench Network Quantities

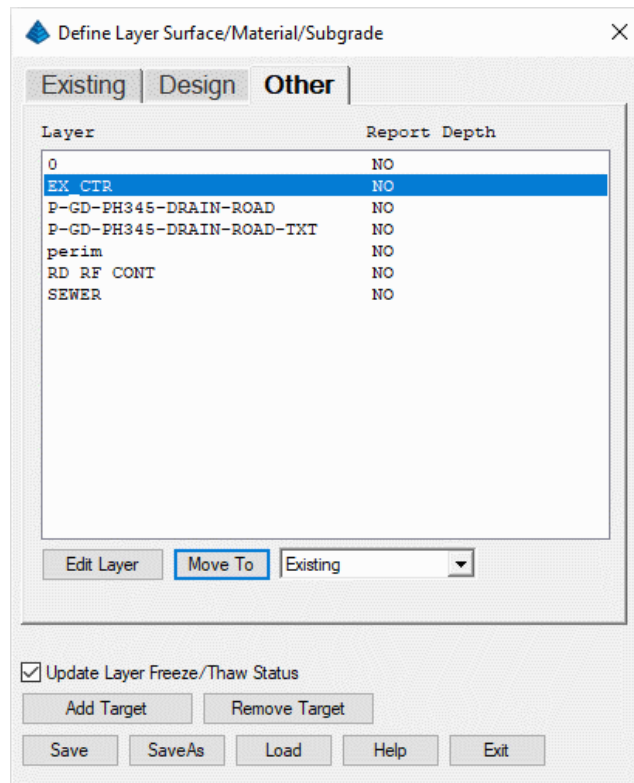
This lesson takes a drawing file through the steps of trench network quantities in the Trench module after some initial preparation through the use of the Construction module.

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **Takeoffdemo2.dwg** file. Use the File – **Save As** to save a copy named **Takeoffdemo2-A.dwg**. Completing this tutorial will alter the drawing file and by renaming the file from the start, you'll keep the original file intact (allowing you to run through the tutorial a second time if desired). This is also a good practice to keep when working on drawings from 3rd parties. The following drawing is displayed:



2. Activate the Construction menu via Settings – Carlson Menus – **Construction Menu** (or the Civil module). In order to calculate trench quantities and profiles in this drawing, we need surfaces for the Existing Ground and Design Surface. We'll follow the general process as introduced in the Takeoff/SiteNET CAD File From Start To Finish tutorial. Issue the Takeoff – Define Layer Surface/Material/Subgrade command to display a dialog box similar to that shown below:



Use the table below along with the **Move To** button to assign the layers cited below to their desired target:

Existing	Design	Other
EX_CTR	RD RF CONT	0
		P-GD-PH345-DRAIN-ROAD
		perim
		SEWER

Takeoff2.dwg Layer Targets

Click the **Save** and then the **Exit** buttons when ready. This assigned layer **EX_CTR** to the Existing Ground Surface and **RD RF CONT** to the Design Surface.

- Issue the run Takeoff – Boundary Polyline – Set Boundary Polyline command and when prompted:

Select boundary polylines.

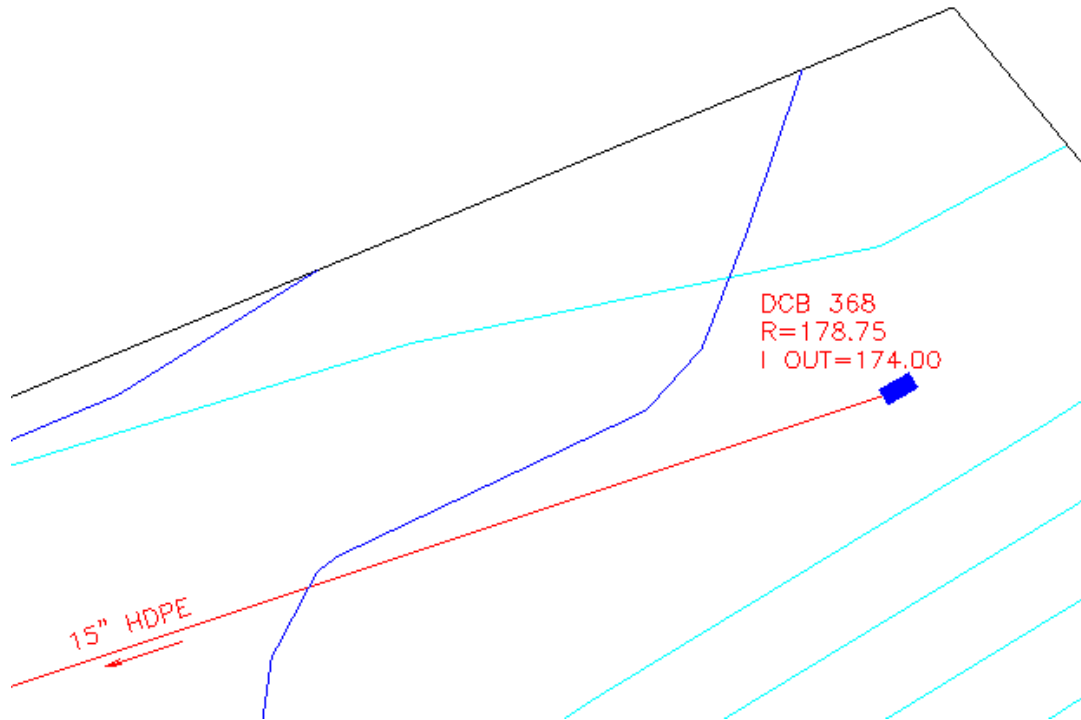
[FILter]/<Select entities>: pick the perimeter polyline

This selected polyline is now set as the boundary polyline for the rest of this tutorial.

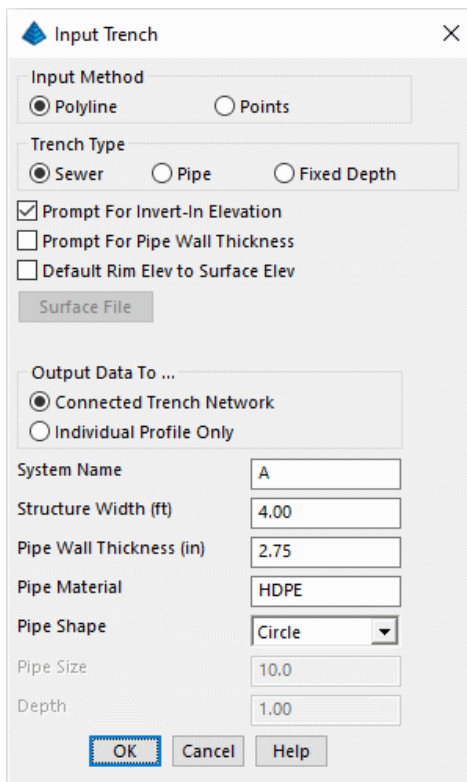
- Our next tasks will be to create the two surfaces:
 - To make the Existing Ground surface, issue the Takeoff – Make Existing Ground Surface.
 - To make the Design Ground surface, issue the Takeoff – Make Design Surface.
- Now that we have Existing and Design surfaces, we are ready to activate the Trench module via Settings – Carlson Menu – **Trench Menu**. The trench network data consists of linked structures where each structure has a:
 - name*
 - location (x,y)*
 - rim elevation*
 - invert-in*
 - invert-out*

Each "link" (pipe) between structures has a *pipe size*. There are two ways of entering the trench data (and we'll explore both methods):

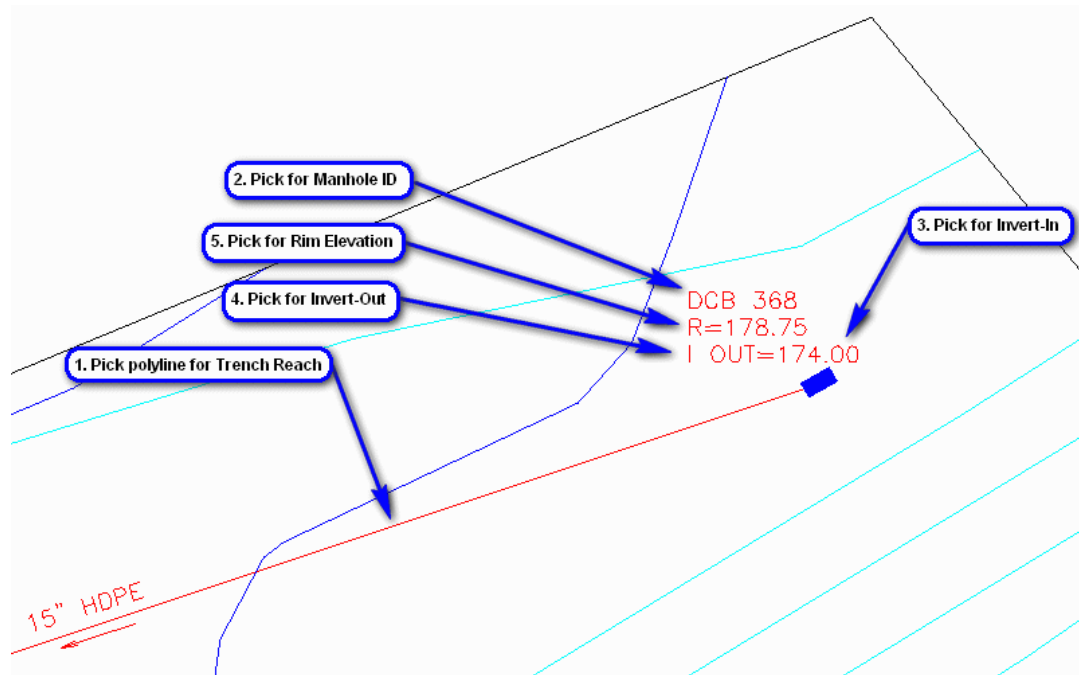
- (a) **Input Trench From Polyline** is used when the drawing contains polylines for the trench lines and labels with the trench data.
 - (b) **Create Trench Network Structure** lets you pick the structure locations and enter the data in a dialog.
6. In our first example, there is trench data already drawn in the drawing. Use the View – **Window** command to establish the view as shown below:



Issue the Trench – Input Trench Line command to display the dialog box below:



With the data available to us, set the values as shown in the dialog box and click **OK** when ready. For each point in the trench polyline, the program zooms the drawing to that point. The trench data can be picked from labels in the drawing:



When prompted:

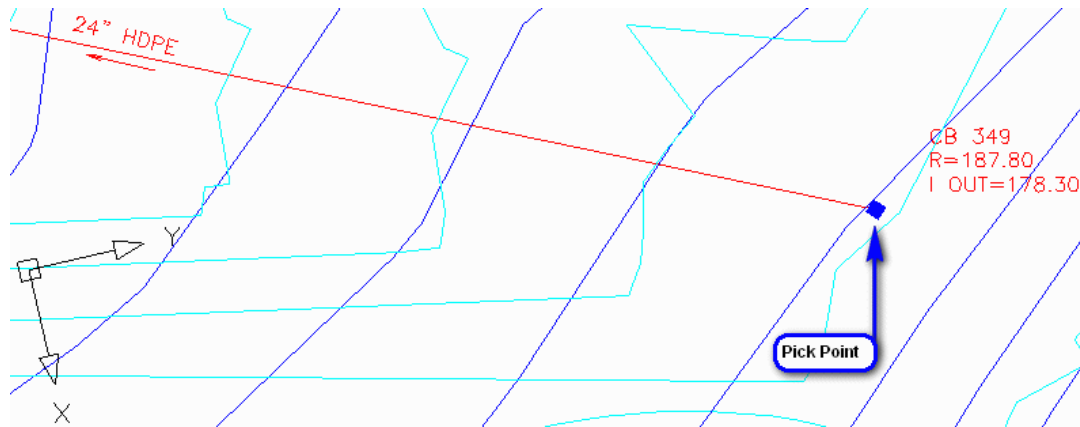
Pick a polyline that represents a trench reach: pick the pipe polyline
Starting Station of trench reach <0.0>: press Enter
Select/<Enter Manhole ID>: type s and press Enter
Enter/<Select text of Manhole ID>: pick the **DCB 368** label
Undo/Select/Interpolate/<Enter Invert-in Elevation <0.0000> >: type s and press Enter
Undo/Enter/Interpolate/<Select text of Invert-in elevation>: pick the **I Out=174.00** label (since this is the upstream starting manhole, there really isn't a separate invert-in so we are using the invert-out)
Undo/Select/<Enter Invert-out Elevation <0.0000> >: type s and press Enter
Undo/Enter/<Select text of Invert-out elevation>: pick the **I Out=174.00** label
Undo/None/Select/<Enter manhole rim elevation <0.0000> >: type s and press Enter
Undo/None/Enter/<Select text of manhole rim elevation>: pick the **R=178.75** label
Enter/<Select text of Manhole ID> [DCB 369]: pick the **DCB 367** label
Undo/Enter/Interpolate/<Select text of Invert-in elevation>: pick the **I In=172.85** label
Undo/Enter/<Select text of Invert-out elevation>: pick the **I Out=172.35** label
Undo/None/Enter/<Select text of manhole rim elevation>: pick the **R=178.50** label
Undo/Select/Group/<Enter Pipe Size in <0.0000> >: type s and press Enter
Undo/Enter/Group/<Select text of pipe size>: pick the **15" HDPE** label
Enter/<Select text of Manhole ID> [DCB 368]: pick the **CB 347** label
Undo/Enter/Interpolate/<Select text of Invert-in elevation>: pick the **I In=170.54 (CB 367)** label
Undo/Enter/<Select text of Invert-out elevation>: pick the **I Out=166.10** label
Undo/None/Enter/<Select text of manhole rim elevation>: pick the **R=176.50** label
Undo/Enter/Group/<Select text of pipe size>: pick the **15" HDPE** label
Another Trench Line [<Yes>/No]? type n and press Enter

That completes this trench run and Takeoff/Trench draws its own trench polyline and labels.

NOTE: If the drawing doesn't have labels for the data, then you can enter the values.

7. The drawing contains another trench polyline and we could use **Input Trench Line** again. However, for

practice, let's use another method for data entry. Issue the View – **Extents** command followed by View – **Window** command to zoom to the drawing location (CB 349) as illustrated below:



Next, run the Trench – Create Trench Network Structure command. When prompted:

Locate by pick point, point number or station-offset [<Pick>/Number/CL**]?** press Enter for the **Pick** option

Pick structure location: pick the pipe polyline near the **CB 349** end-point

A dialog box similar to that shown below appears:

Trench Structure Data ✕

Structure Data

N: 207062.8 E: 409437.6 Z: 187.8

Structure Name: Symbol Name:

System Name:

No Structure (Junction Only) Structure Type:

Structure Template:

Rim Elevation (ft): Structure Width (ft):

Invert Elev OUT (ft): Depth (ft):

Sump Depth (ft):

Pipe Data

Upstream Connections:

Available	
DCB 368	A
DCB 367	A
CB 347	A

Invert Elev IN (ft): Slope Percent: Pipe Material:

Use Pipe Group

Pipe Interior Size (in): Pipe Width (in): Pipe Shape:

Pipe Wall Thickness (in): Trench Type:

Set the values as shown and click **OK** when ready. Prompting resumes:

Pick structure location (Enter to end): press Enter

Since the downstream location for this reach has already been created from the **Input Trench Line**

exercise, we don't need to create it again. Use **Zoom** and/or **Pan** functionality to center the display in the *CB 347* area. Initiate the Trench – Edit Trench Network Structure command. When prompted:

Select sewer structure to edit: pick either the symbol for **CB 347** or its recently created label

A dialog box similar to that shown below appears:

Trench Structure Data

Structure Data

N: 206876.4 E: 409351.8 Z: 176.5 Set Location

Structure Name: CB 347 Symbol Name: SPT5

System Name: A Set Symbol

No Structure (Junction Only) Structure Type: Set Template

Structure Template:

Rim Elevation (ft): 176.500 Structure Width (ft): 4.000

Invert Elev OUT (ft): 166.100 Depth (ft): 10.400

Sump Depth (ft):

Pipe Data

Upstream Connections: DCB 367 A Add

Available: CB 349 A, DCB 368 A, DCB 367 A Remove

Invert Elev IN (ft): 166.100 Slope Percent: -4.974 Pipe Material: HDPE Set

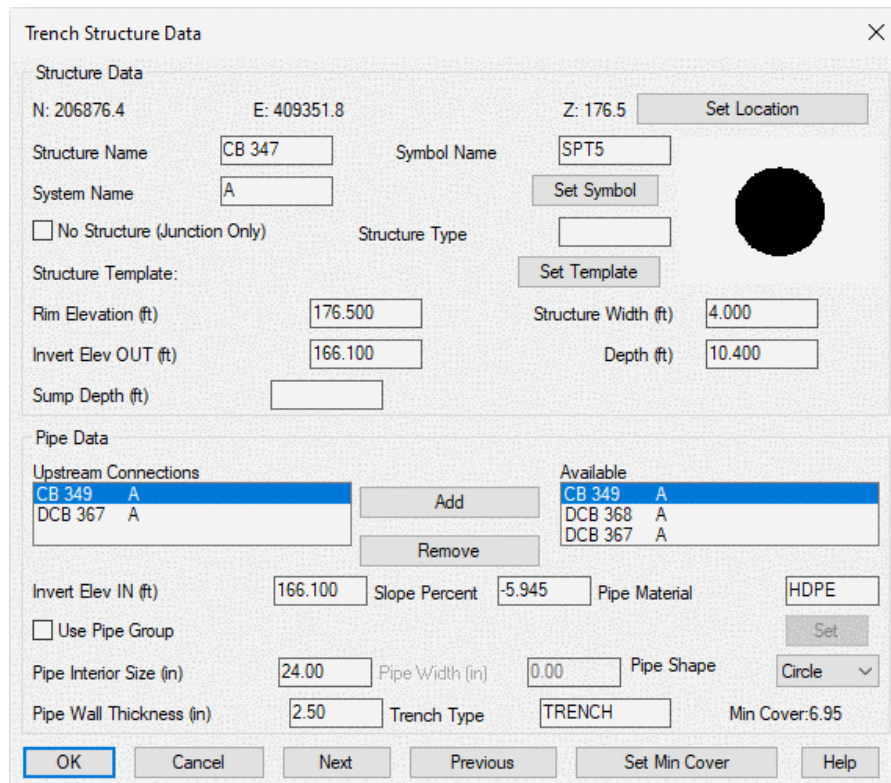
Use Pipe Group

Pipe Interior Size (in): 15.00 Pipe Width (in): 0.00 Pipe Shape: Circle

Pipe Wall Thickness (in): 2.50 Trench Type: TRENCH Min Cover: 5.19

OK Cancel Next Previous Set Min Cover Help

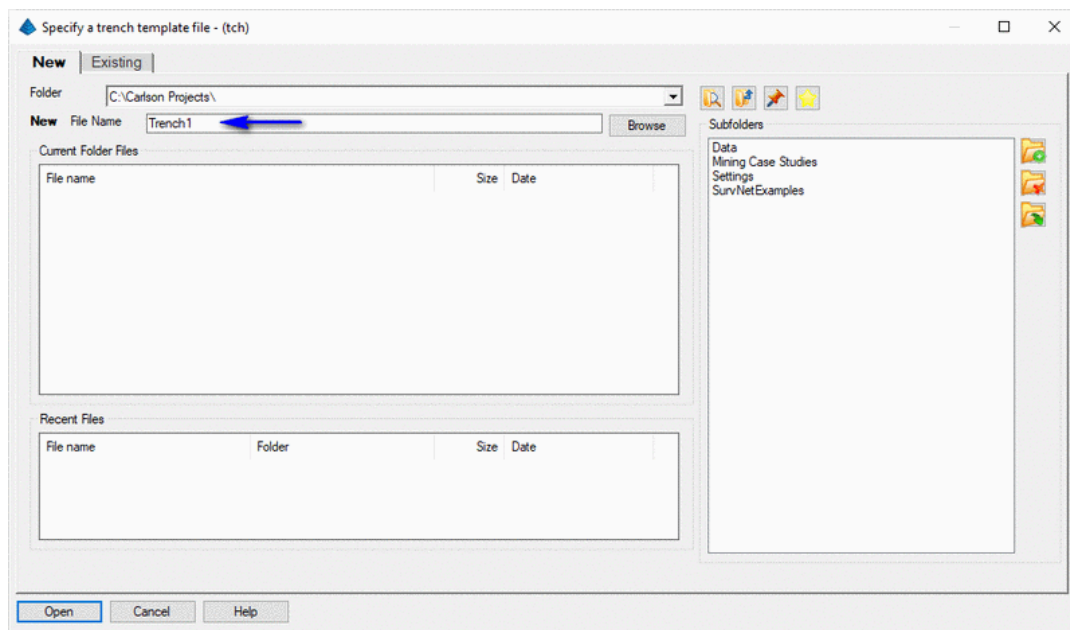
Since this is the downstream terminus for the network, highlight/select the **CB 349** structure and click the **Add** button. Fill the values as shown above. When complete and as needed, highlight/select the **DCB 367** structure and click the **Add** button. Fill the values as shown below:



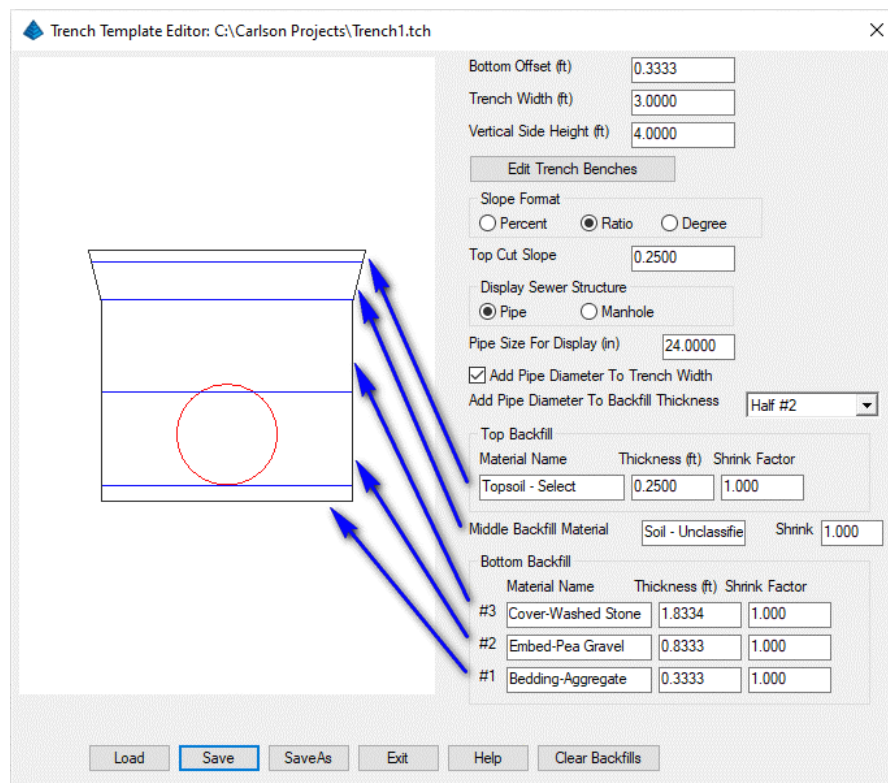
When complete, click the **OK** button. This creates a link from **CB 347** to its upstream location(s).

NOTE: It is also possible to directly edit the structure data by using double-click functionality on the *Trench*-placed structure or its label.

8. With the network structures defined, let's create the trench geometry that will ultimately yield the trench quantities. Run the Trench – Trench Template Editor command. Provide the name as shown below and click **Open** when ready:



The primary dialog box appears, similar to that shown below:

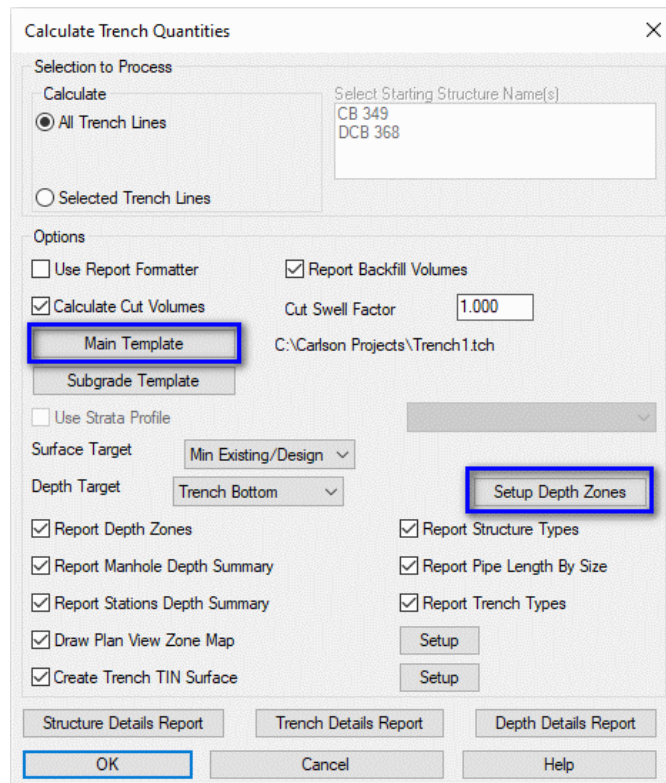


Supply the values as illustrated below (or design your own!). In the above example, we are:

- (a) assuming the in-situ material is of suitable composition (e.g. not "fatty clays", peat moss, *etc*),
- (b) using 4" of crushed aggregate, granite, *etc*,
- (c) placing the pipe directly on top of the bedding material,
- (d) using 10" of pea gravel, for the embedment material (this thickness will vary by $\approx \frac{1}{2}$ of the pipe diameter),
- (e) using 22" of washed stone, for the cover material,
- (f) using a variable thickness of unclassified, excavated material to fill in the remainder of the trench (except the final 3"), and,
- (g) using 3" of select topsoil to complete the backfill.

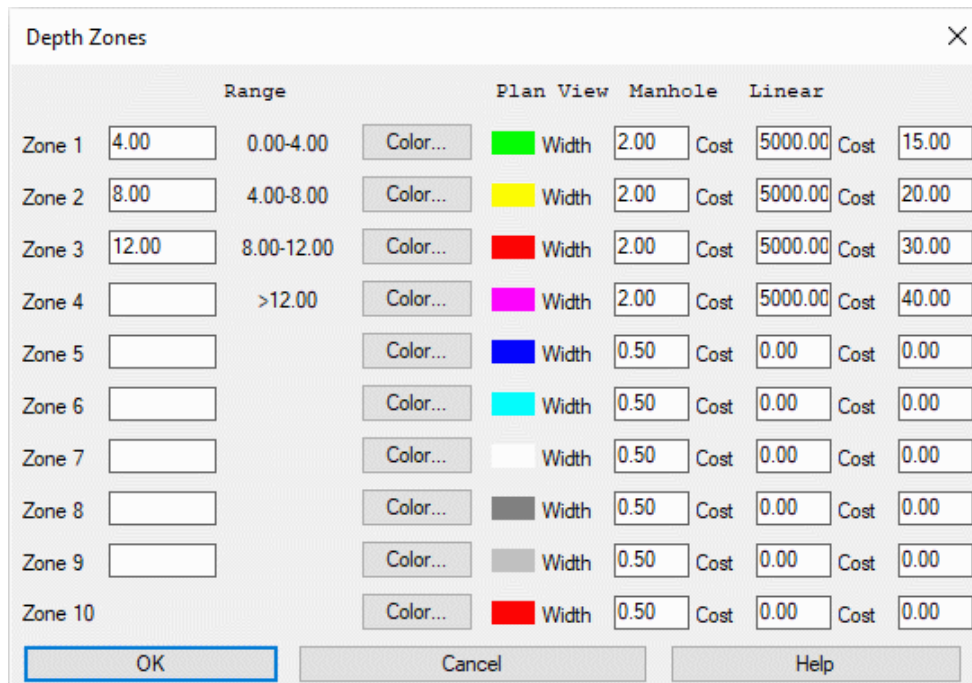
Click **Save** and **Exit**.

9. To calculate and report the trench quantities, run the Trench – Trench Network Quantities command. A dialog similar to that shown below appears:



Set the values shown above making particular note of:

- Specifying the **Main Template** trench configuration we just created, and,
- Setting the **Surface Target** to **Min Existing/Design** (permitting quick excavation of the site to the lower of *Existing* vs. *Design* elevation before trenching commences), and,
- Setting the **Setup Depth Zones** to the values as shown in the dialog box below (click **OK** when complete)



Finally, click **OK** in the main dialog and the report similar to that shown below is displayed:

Carlson Edit: C:\Carlson Projects\Takeoffdemo2-Trench Quantities Report.txt

Trench Quantities Report

Starting Structure	Length	Avg Depth
DCB 368	327.1	7.1
CB 349	205.2	7.5
Total	532.4	7.2

Trench Dimensions
 Bottom Offset: 0.33 ft
 Trench Width: 3.00 ft Plus Pipe Diameter
 Vertical Side Height: 4.00 ft
 Cut Slope: 40:04 Ratio: 0.25:1 Degree: 76.0
 Surface Target: Min Existing/Design
 Depth Target: Trench Bottom

Total Cut Volume: 811.48 C.Y.

Add Half Pipe Diameter to Backfill #2 Thickness	Backfill Material	Depth(ft)	Volume(CY)	Shrink Factor
	Bedding-Aggregate	0.333	32.74	1.000
	Embed-Pea Gravel	0.833	123.44	1.000
	Cover-Washed Stone	1.833	174.42	1.000
	Topsoil - Select	0.250	33.03	1.000
	Spill - Unclassified		385.21	1.000
	Total		748.84	

Depth	Manholes	Linear(ft)	% of Total	MS Cost	Cost/MS	Linear Cost	Cost/Linear	Total Cost
0-4.0	0	0.0	0.0	0	5000	0	15	
4.0-8.0	3	388.9	73.0	15000	5000	7777	20	22777
8.0-12.0	1	143.5	27.0	5000	5000	4306	30	9306
>12.0	0	0.0	0.0	0	5000	0	40	
Total	4	532.4		20000		12083		32083

Stations	Depth	Starting Structure
0+00.0 to 0+70.9	8.0-12.0	CB 349
0+70.9 to 2+05.2	4.0-8.0	CB 349
0+00.0 to 2+29.6	4.0-8.0	DCB 368
2+29.6 to 2+64.0	8.0-12.0	DCB 368
2+64.0 to 2+72.6	4.0-8.0	DCB 368
2+72.6 to 3+10.8	8.0-12.0	DCB 368
3+10.8 to 3+27.1	4.0-8.0	DCB 368

Pipe Size(in)	Linear(ft)	% of Total
15	327.1	61.5
24	205.2	38.5

Trench Type	Linear(ft)	% of Total
Blank	201.4	37.8
TRENCH	330.8	62.2

Structure Type	Count
Blank	4

Manhole	Depth
DCB 367	6.48
DCB 368	5.08
CB 347	10.73
CB 349	9.83
Total	32.13

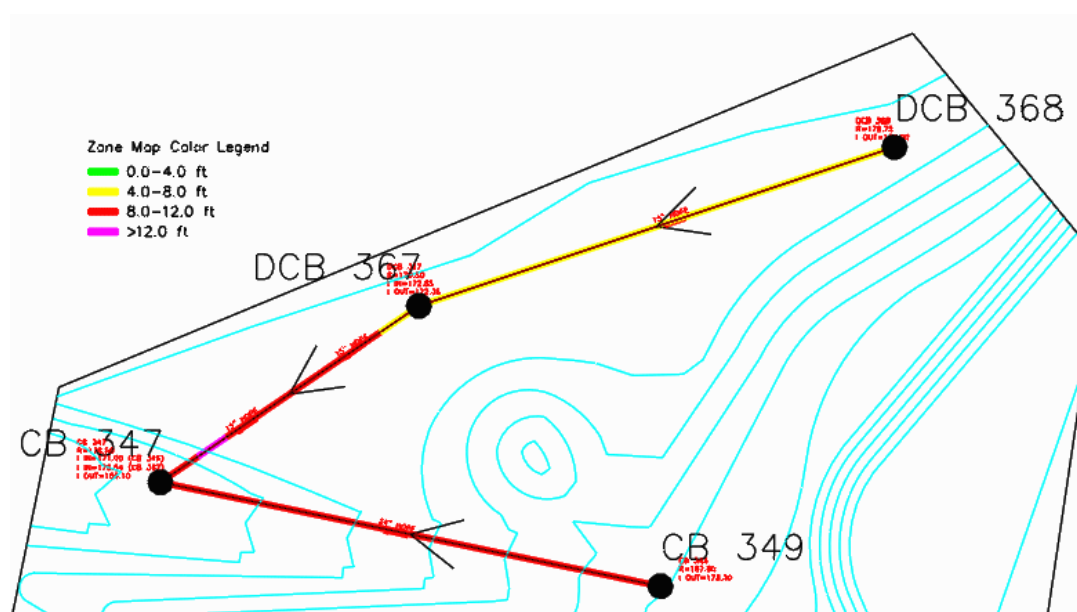
Review the report and click the **Exit** (doorway) button when ready. When prompted:

Draw zone map color legend on the screen [Yes/<No>]? type y and press Enter

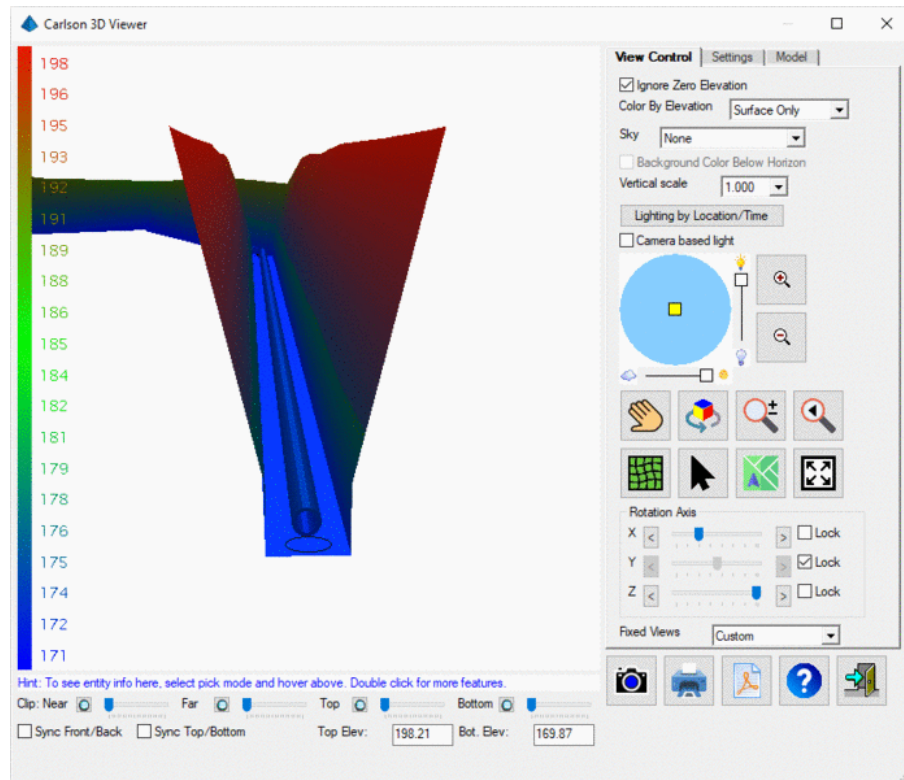
Pick a point for color legend: pick a location toward the upper left of the project site

Legend size <4.00>: press Enter

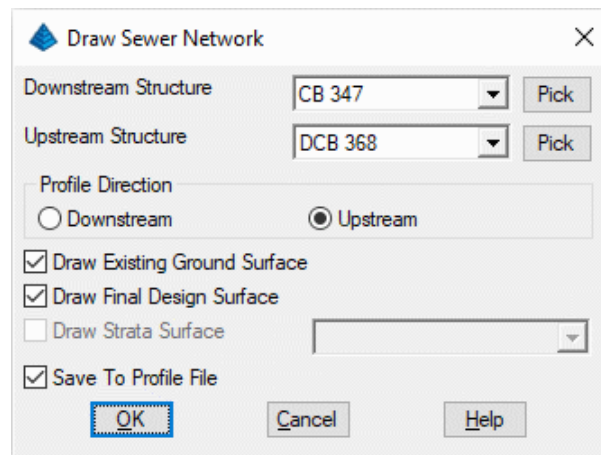
The depth zones in the plan view with Zone Map Color Legend are shown similar to that below (certain layers have been frozen for clarity):



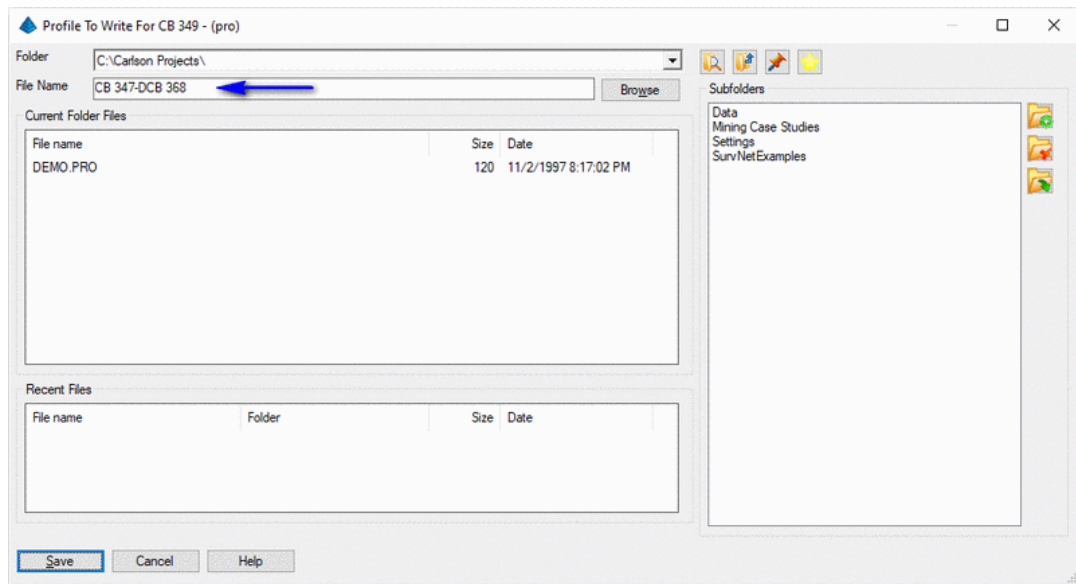
Finally, use the Display – 3D Viewer Window and window around the 3D faces to view the trench in 3D as illustrated below:



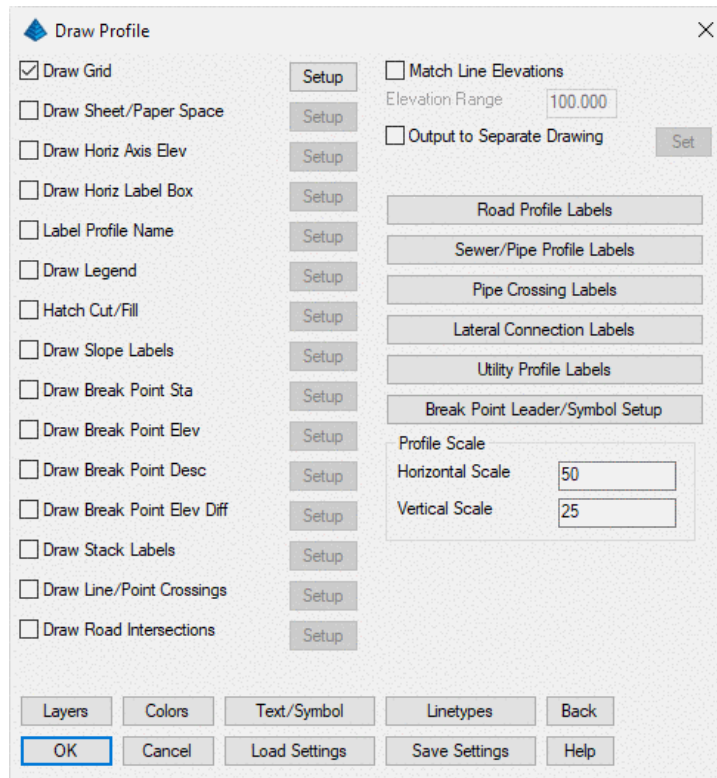
10. To draw a profile of the trench line, run the Trench – Draw Trench Network (Profile) command. Set the values as shown below and click **OK** when ready:



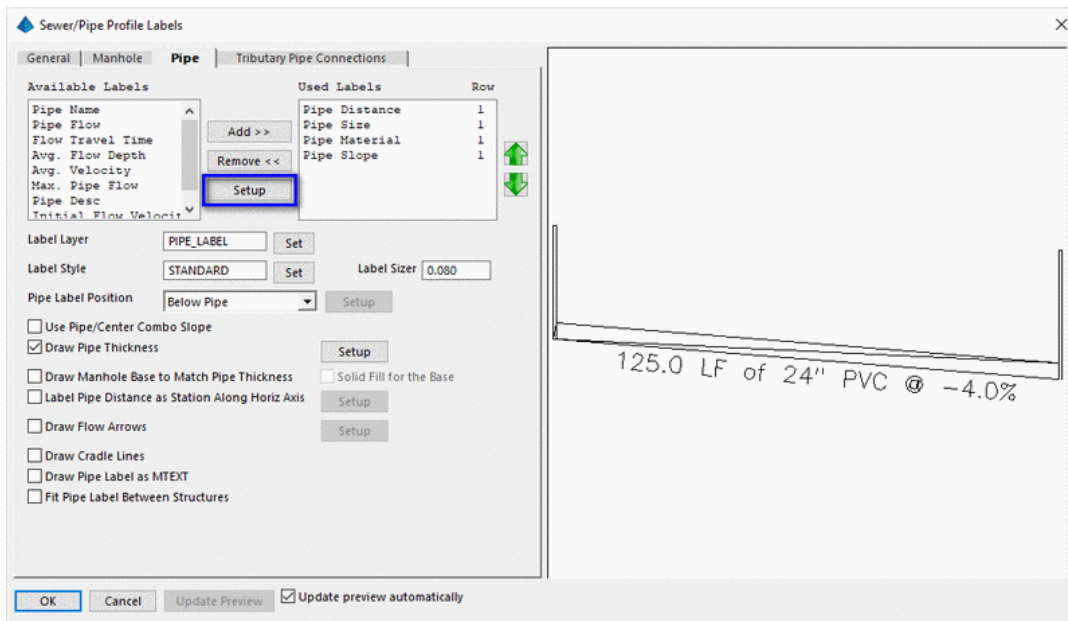
When prompted:



Supply the profile name as cited above and click **Save** when ready. Next, the Draw Profile dialog appears as illustrated below:



We'd like to fine-tune the content of the forthcoming annotation. Click the Sewer/Pipe Profile Labels button and navigate to the **Pipe** tab as illustrated below:

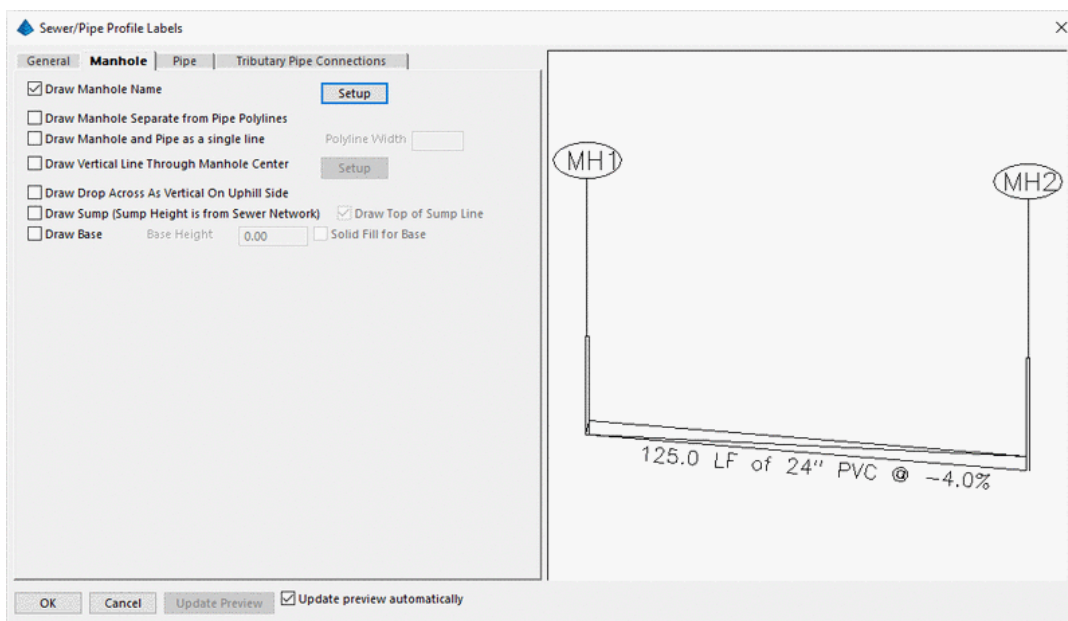


Use the **Add >>** button to add the following annotation items with the specified settings (use the **Setup** button to adjust for particular preferences):

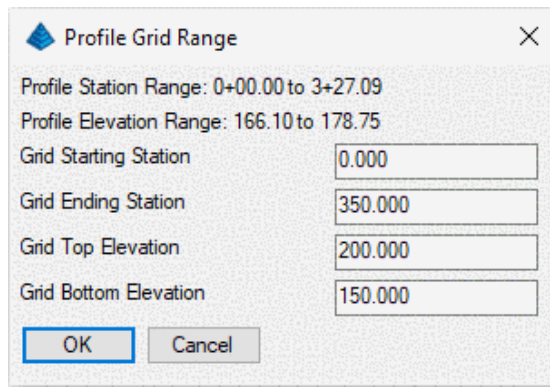
Item	Prefix	Suffix	Other
Pipe Distance		LF	
Pipe Size	of	"	Decimals: 0
Pipe Material			
Pipe Slope	@	%	Slope Format: %

Sewer/Pipe Profile Label Settings

Navigate to the **Manhole** tab and modify as desired to obtain the illustration below:



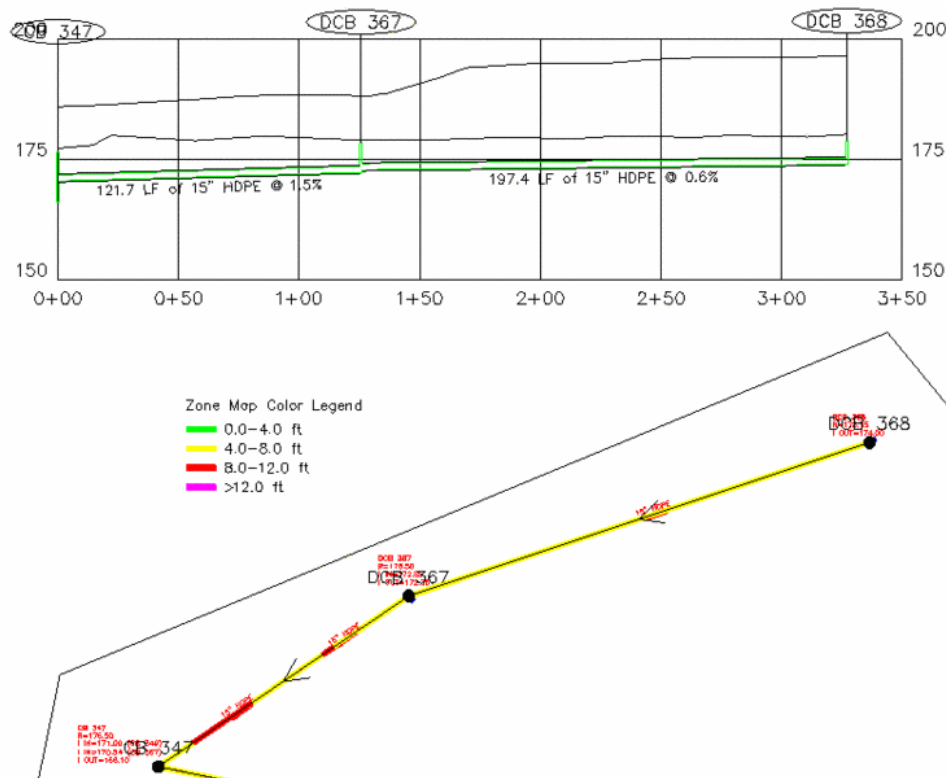
Click **OK** to dismiss the **Sewer/Pipe Profile Labels** dialog box. Set the **Horizontal Scale** and **Vertical Scale** as shown earlier and click **OK** when ready. When prompted:



Accept the default values and click **OK** when ready. When prompted:

Pick Starting Point for Grid <0.00,0.00>: pick a desired location for the profile as illustrated below

NOTE: Some layers frozen for clarity.



This completes the tutorial: Trench Network Quantities.

Digitizing

This lesson transfers a paper plan-set into a DWG drawing using Carlson CADnet and then uses Carlson Construction to calculate volumes from the digitized linework.

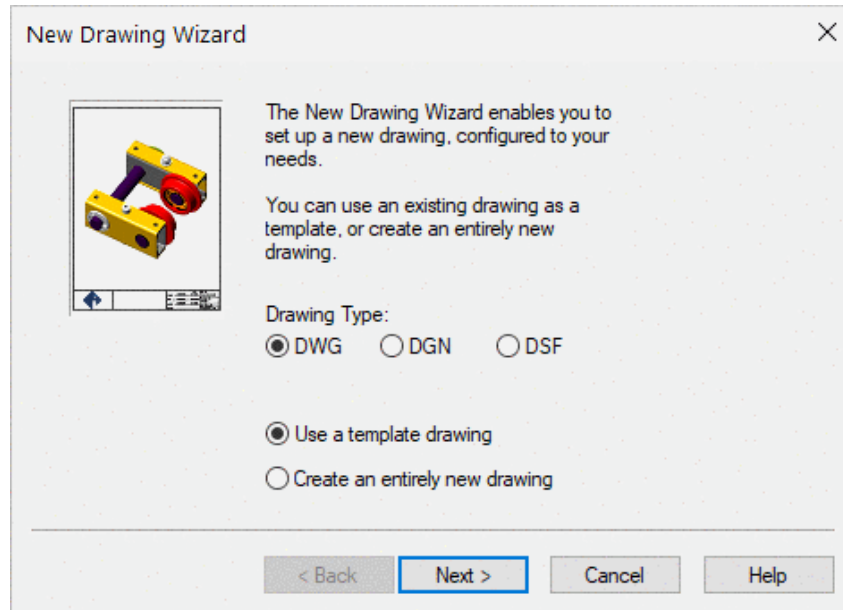
1. **Setup:** To digitize from a paper plan in CADnet, you need to install the Wintab digitizer driver. See Digitizer Setup in the manual if you have not installed or have problems with the Wintab driver. If Wintab is installed, then make sure your drawing board is on and take the paper plan provided with the manual and place it on your drawing board.
2. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **New Drawing**.

- If you get the Startup Wizard dialog box, click **New**.
- If you are taken directly into CAD, click the File – **New** command.

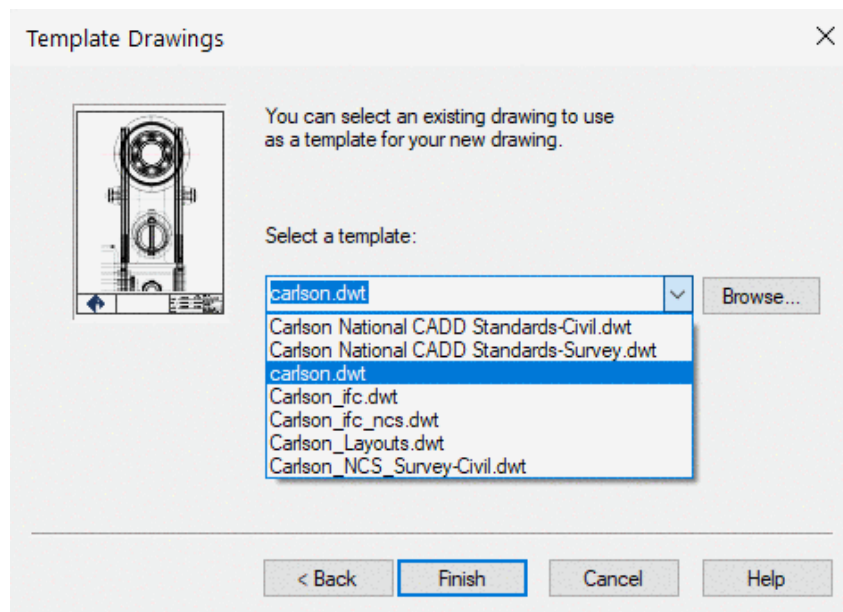
NOTE: If you want to consistently use the **Startup Wizard**, issue the Settings – Carlson Configure – General Settings and enable the **Use Startup Wizard** option. Click OK to dismiss the dialog boxes. The **Startup Wizard** will be used the next time the software is started.

The first of several Startup Wizard dialog boxes appears:

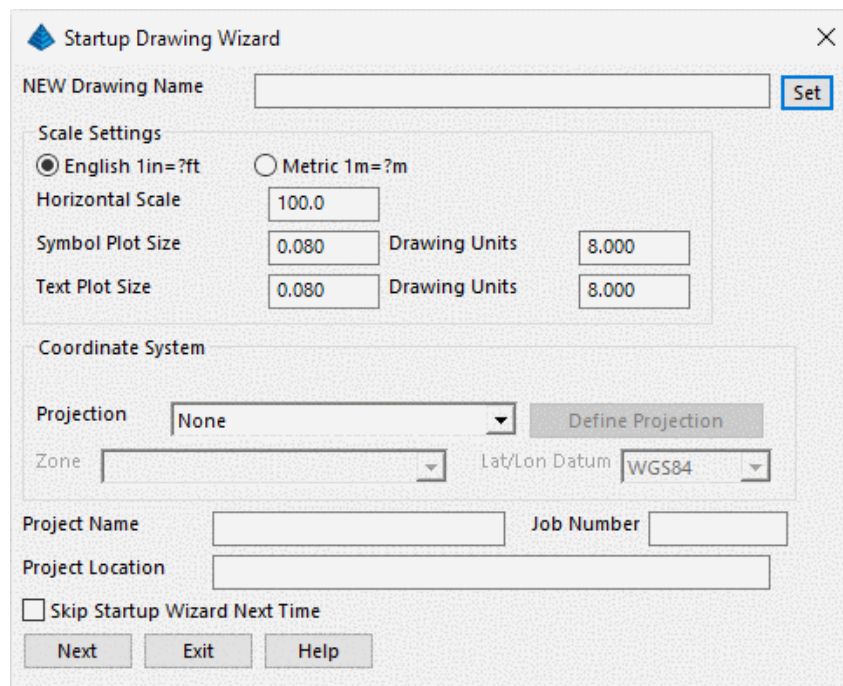
- Choose the DWG document type and the desire to base the document on a **Drawing Template** as illustrated below and then click **Next >**:



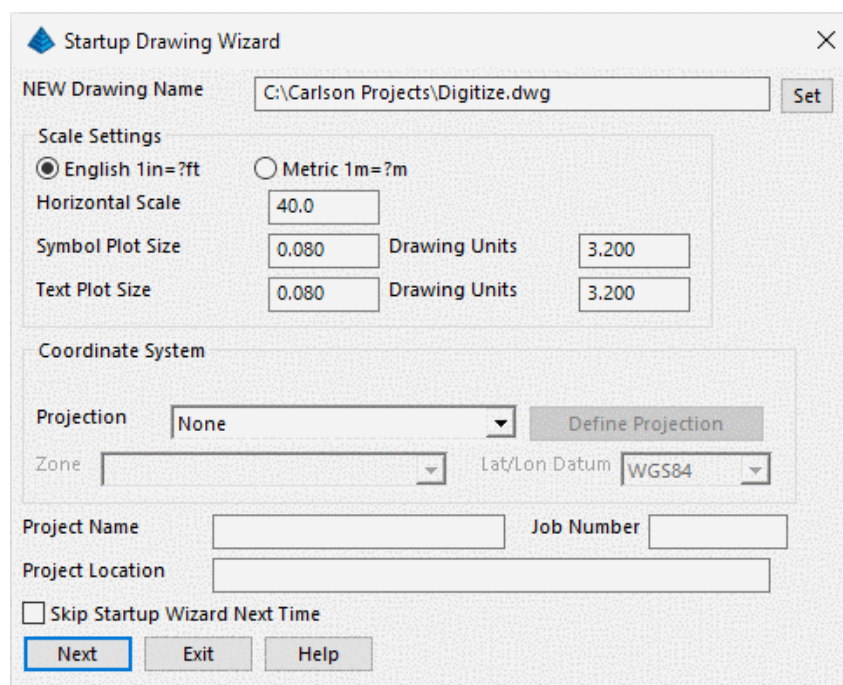
- Choose the **carlson.dwt** as illustrated below (or *site.dwt* if *carlson.dwt* is not available) and click **Finish**:



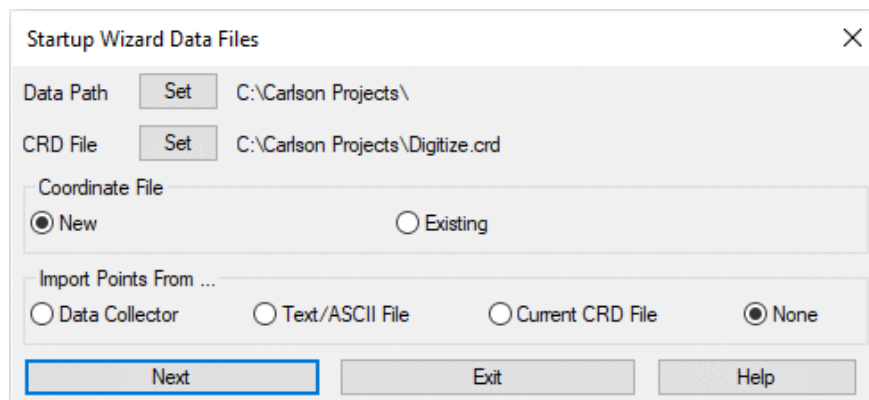
- We can now begin the more pertinent settings for the project to come based on some preliminary settings that should be similar to the default scenario shown below:



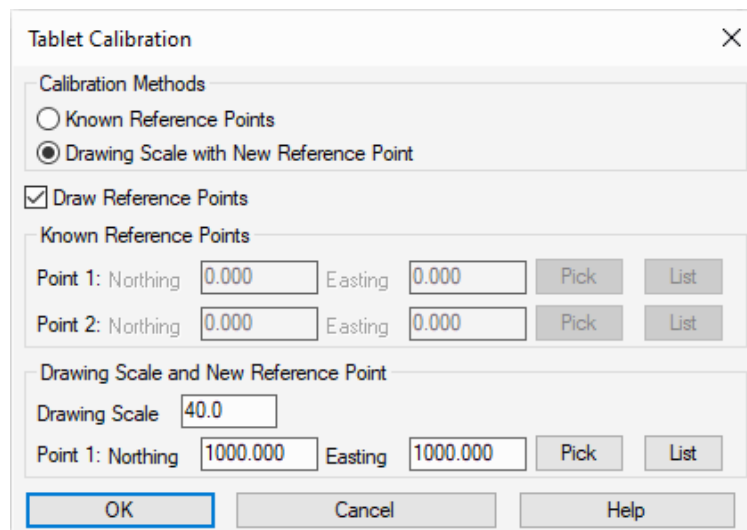
- (d) Click **Set** at the top of the dialog box, and enter in a NEW Drawing Name called **Digitize**. Verify that the other settings match the settings shown below, and click **Next**:



- (e) You will see the Startup Wizard Data Files dialog to set/confirm where to store data and indicate an information source for points/coordinates. Set/match the values as shown below and click **Next**:



3. Activate the CADnet menu via Settings – Carlson Menus – **CADnet Menu**. To start things off, you need to set the coordinate system for the paper plan by running the Digitize – Tablet – Calibrate command. Calibration is required to let the program know the orientation and scale of the paper plan.



There are two different Calibration methods:

- (a) **Known Reference Points:** allows you to enter in the coordinates of two marked points on the paper plan. This method applies when you know the coordinates of at least two points on the paper plans.
- (b) **Drawing Scale with New Reference Points:** allows you to set up a coordinate system for the plans by entering the plan scale and picking any two points from the paper plan with the digitizer puck.

In this case, we will use **Drawing Scale with New Reference Points**. For this drawing, the scale is 1:40, so enter in **40**. Use the default coordinates for Point 1 and click **OK**. Our target icon is shown as follows:



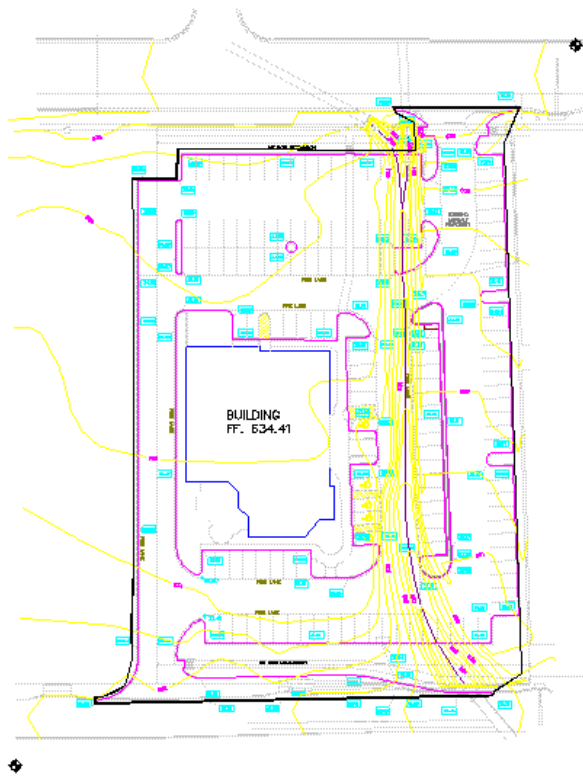
When prompted:

Digitize the First Reference Point: digitize on the target icon in the lower left

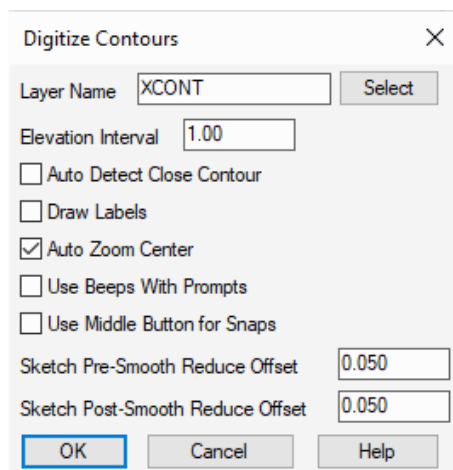
Digitize the Second Reference Point: digitize on the target icon in the upper right

The first point is assigned the coordinate specified earlier in the **Calibrate** dialog box and the second point is assigned coordinates to match with the plan scale. From now on, all of your points will be in relation to these two points.

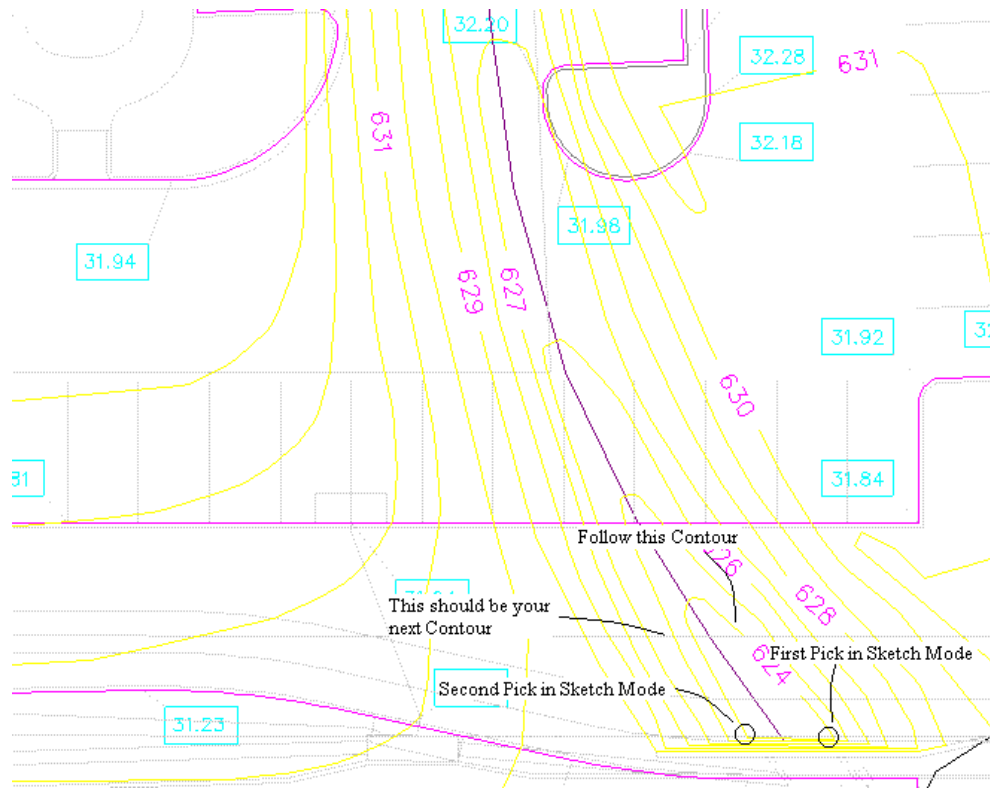
NOTE: Generally, you want to pick points on the drawing that you can find and use again in case you need to recalibrate. Also, the further away the points are from each other, the more accurate the coordinate system will be.



- We will now digitize the existing contours. The digitize routines in CADnet can be used to populate the *Existing*, *Design*, and *Other* targets used in Carlson Construction and the SiteNET portion of Carlson Civil. Issue the Digitize – Existing Surface command. Next, issue the Digitize – Contour Polyline command which will display the dialog box below:



Set the values as shown above (making note of your applicable contour interval) and click **OK** when ready. The rest of the prompting occurs at the command line and starts with the contour elevation. Find the lowest elevation for the Existing contours labeled in the bottom right corner of the paper plan zoomed in on below. In this example, the lowest elevation is 624 feet. You want to enter in the lowest contour so that as Carlson CADnet adds the Elevation Interval, it is from lowest to highest. The elevation can be entered either with the digitizer puck keys or with the computer keyboard. The layout of the digitizer keys is set in Digitize – Digitizer Settings – **Puck Layout**.



When prompted:

Increment(1.00)[A]/Direction(+)[B]/Elevation <0.00>: indicate 624 and press Enter

Sketch[0]/Exit[A]/Pick the first point: type in [0] and press Enter to get into Sketch Mode. In Sketch Mode, you will be prompted to "Pick and drag." The first point you pick is the starting point of a contour. Drag is asking you to follow that contour with the digitizer puck on the paper plan. Click a second time when you have traced the entire contour and have reached the end of the contour. You will then be prompted as follows:

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): indicate [A] to close the contour. Indicate [B] for Undo if you made a mistake and need to sketch part of the contour again. [0] will switch you into Pick Mode.

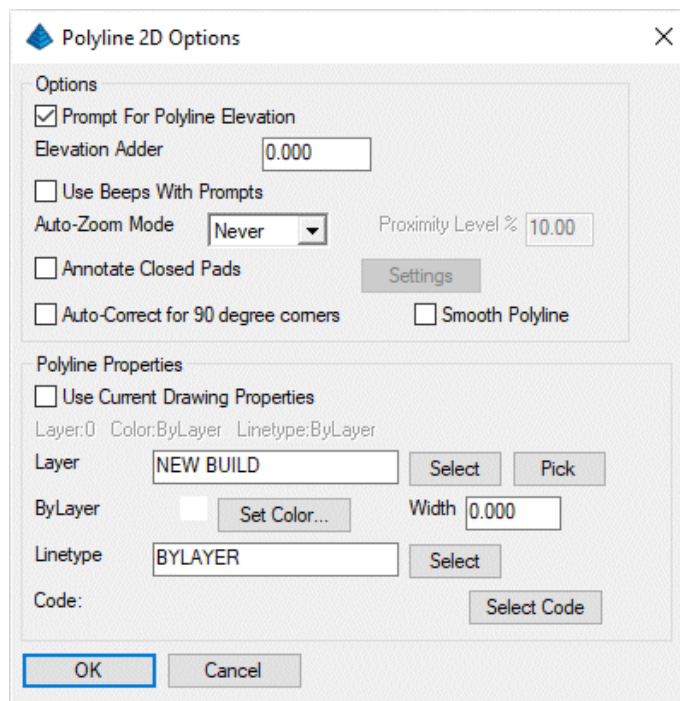
Increment(1.00)[A]/Direction(+)[B]/Elevation <625.00>: press Enter to accept the next elevation in the series and continue to digitize the next contour elevation. However, pick [0] to get into *Pick Mode*. In *Pick Mode*, you do not have to trace the contour. Rather, pick with the digitizer puck to create points that will make up the contour. When all contours are digitized, press **E** to exit the routine.

NOTE: When in *Pick Mode*, less picks are needed on fairly straight segments. Conversely, more picks will give you a more accurate contour. When completed, the Existing contour map should resemble the following:



5. Now we will digitize the building and curb line work of the *Design Surface*. In this example, there are no design contours, only the design building and curb line work and spot elevations. We will also assign layer names to the line work that we will use later to identify the types of linework. Issue the Digitize – Design Surface command.

Let's begin by digitizing the main building. Issue the Digitize – 2D Polyline command (**2D Polyline** is used to digitize line work entities with one elevation) to display the dialog box below:



Set the values as shown above and click **OK** when ready. When prompted:

Enter polyline elevation <0.00>: 634.41 and press Enter (this is the value annotated in the middle of the building)

First point: pick a building point

Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): pick the next building point

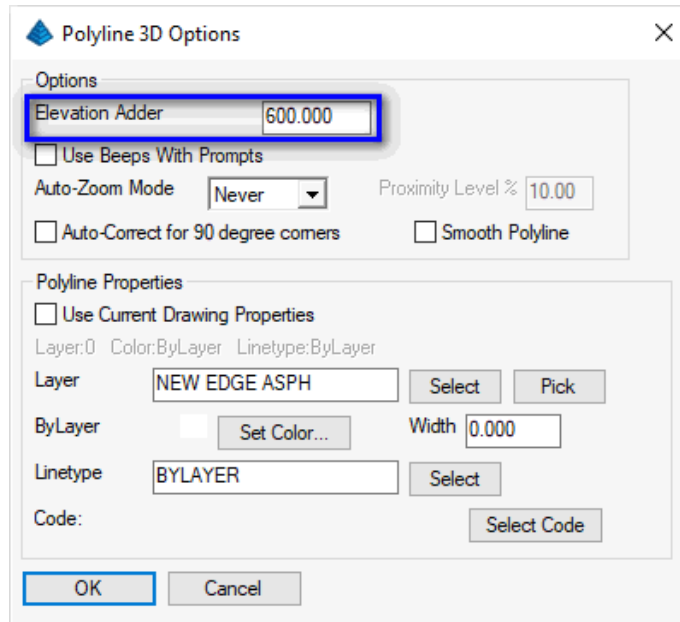
Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): repeat...

Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): pick the last building point

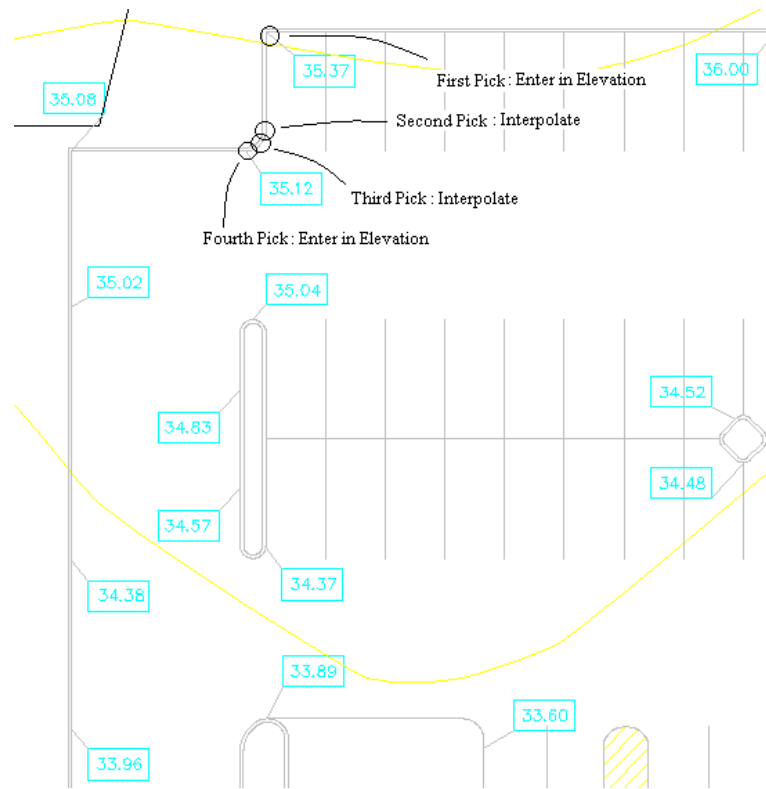
Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): A to close

Digitize Another NEW BUILD Polyline [Yes(A)/<No(B)>]? B for No

Notice that the parking lot line work consists of different elevation levels. For this scenario, we'll use the Digitize – 3D Polyline command which displays a dialog box similar to that shown below:



NOTE: Because the parking lot elevation labels have been shortened on the paper plan (*e.g.* they read 35.37 and 35.12), be sure to set the desired **Elevation Adder**. Set the remaining values as shown and click **OK** when ready. Let's start by digitizing the parking lot starting from the zoomed in section below:



The edge of asphalt is the inside line. When prompted:

First point: Click on the point with the digitizer puck where the 35.37 elevation label points to in the upper left corner of the parking lot

Elevation[B]/Object/Osnap[.]/Next point or elevation<Interpolate>: 35.37 and press Enter

Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): pick below the first point where the line work starts to curve. We do not have an elevation for this point, but we can interpolate the elevation from the two points around it using the Interpolate option.

Percent/Ratio/Interpolate[A]/Degree/Object/<Elevation[B]> <635.37>: A

This point elevation will be interpolated upon completion.

Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): A

Percent/Ratio/Elevation[B]/Degree/Object/Osnap[.]/Next point or elevation<Interpolate>: pick the middle point of the curve

This point elevation will be interpolated upon completion.

Percent/Ratio/Elevation[B]/Degree/Object/Osnap[.]/Next point or elevation<Interpolate>: pick the end of the curve at the 35.12

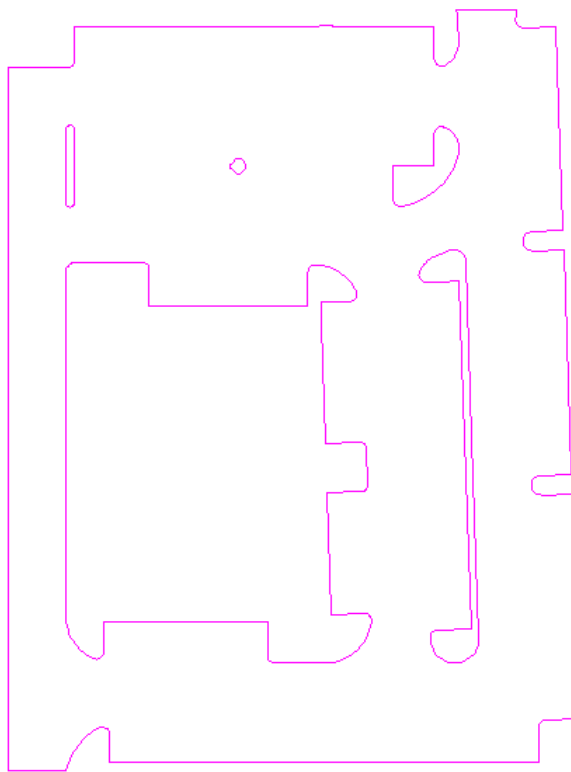
Percent/Ratio/Interpolate[A]/Degree/Object/<Elevation[B]> <635.37>: 35.12 and press Enter

Continue digitizing for the rest of the edge of asphalt linework. Digitize each point where there is an elevation label and each point where the curb line changes direction and press Enter when complete.

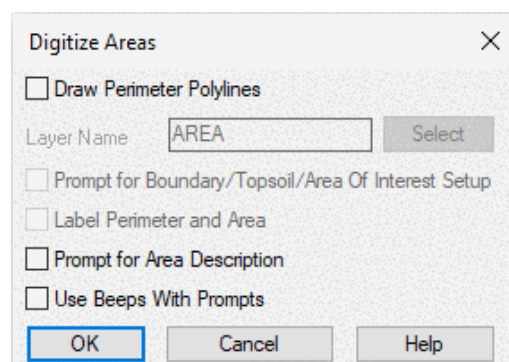
Digitize Another NEW EDGE ASPH Polyline [<Yes(A)>/No(B)]? indicate No

To check the elevations of the interpolated points, issue the Inquiry – Drawing Inspector command and hover over the polyline you just created. A window will appear showing you information about the polyline. If no elevations appear, right+click and enable the **Display Elevations** option.

Use the 3D Polyline command to digitize the rest of the parking lot as seen below.



6. Now that we have digitized the **Design Surface**, let's check the area of certain portions of the site. Run the Digitize – Areas command and match the below dialog and click **OK** when ready:



To approximate the area of the main building, pick the points of the building outline.

Pick starting point: Pick points as close to the building design line work as you can

Undo[B]/Pick next point (Enter to end): repeat... (press Enter when done)

Digitize Another Area [<Yes(A)>/No(B)]? B

An Area Report similar to the one shown below appears:

Carlson Edit : C:\Carlson Projects\Digitize-Area Report.txt

File Edit Settings

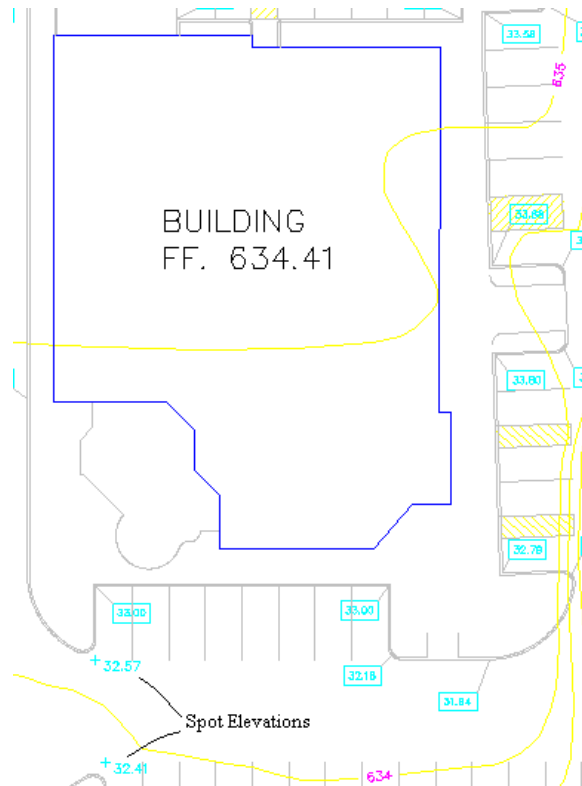
Area Report

Digitized Areas

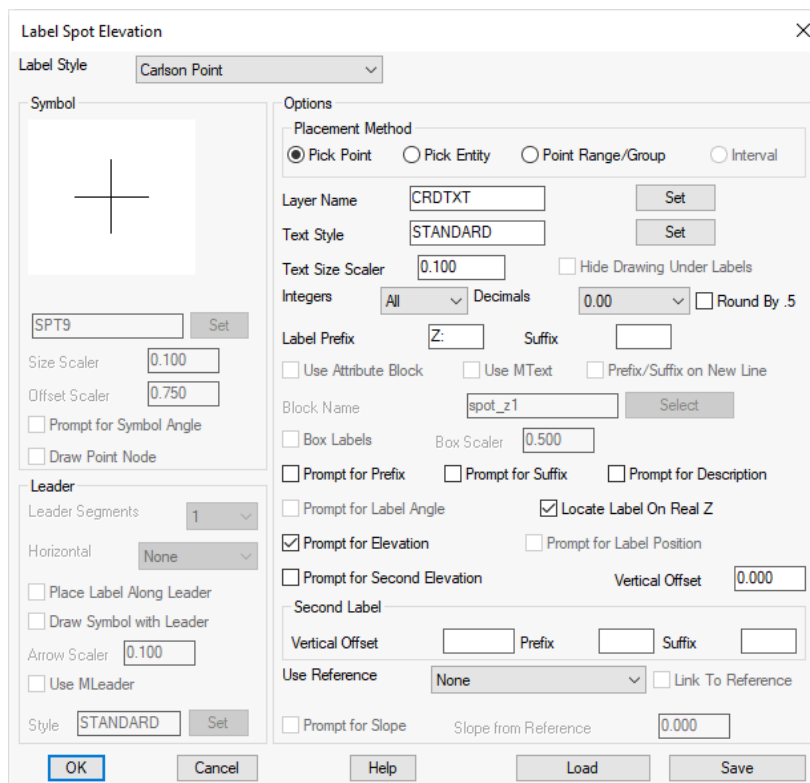
Area Index	Perimeter	Square Feet	Acres
1	405.14	9459.93	0.22
Total:	405.14	9459.93	0.22

Click the **Exit** (doorway) icon.

- In our paper drawing, we have two spot elevations labeled 32.57 and 32.41 shown in the bottom left below.



To digitize these elevations, we can use the Digitize – Spot Elevation command. Fill out the dialog as shown and pick **OK** when ready:



When prompted:

Point to Label?

Pick point or point number: find and click on the spot elevation labeled 632.57

Elevation <0.00>: 632.57

Point to Label (ENTER to End)?

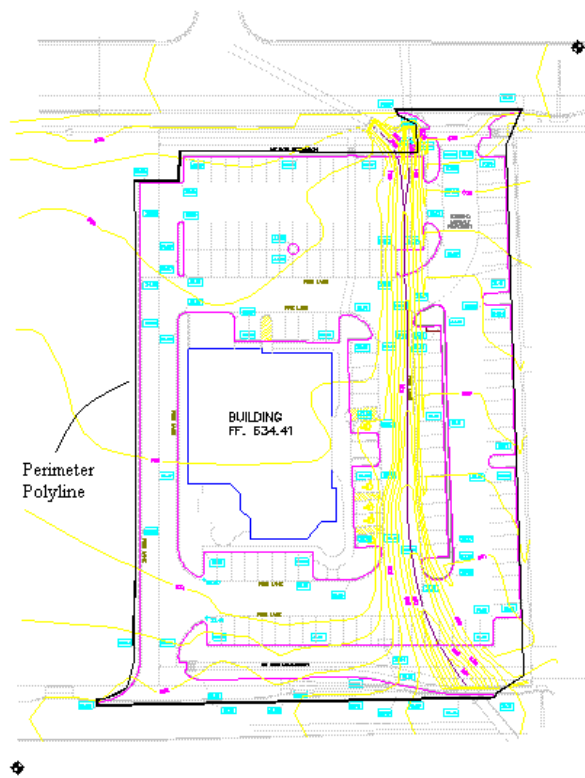
Pick point or point number: find and click on the spot elevation labeled 632.41

Elevation <0.00>: 632.41

Point to Label (ENTER to End)?

Pick point or point number: press Enter

8. The limits of the site are defined by a closed polyline. This polyline is used as the boundary for the models and the volumes. Set a new target via the Digitize – Other Drawing command. Follow this with the Digitize – Perimeter command. The goal will be to digitize around the bold, outside line shown below:



When prompted:

Layer name <PERIM>: PERIMETER and press Enter

First point: pick the first point on the boundary

Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): pick the next point on the boundary

Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): repeat...

Arc[0]/Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): C

Digitize Another PERIMETER Polyline [<Yes(A)>/No(B)]? indicate No

9. Now that we have digitized line work with CADnet, we can continue to calculate Cut/Fill volumes and material quantities. Activate the Construction (or Civil) menu via Settings – Carlson Menus – **Construction Menu** (or **Civil Menu**, respectively). Issue the Takeoff (SiteNET in Civil) – Boundary Polyline – Set Boundary Polyline command and when prompted:

Select boundary polylines.

[FILter]/<Select entities>: pick the perimeter polyline created in the previous step

10. Our next task will be to specify the layer(s) in the drawing which contain *existing* site conditions. Every entity (line, polyline, point, *etc*) in the drawing was assigned a layer name and pre-categorized for us based on our digitize target specification. Takeoff uses the layer name(s) to categorize data into one of three general categories for the purpose of building surface models:

- **Existing** - Used to model initial/pre-construction site conditions.
- **Design** - Used to model proposed/post-construction site conditions.
- **Other** - Used as a collection of unknown/irrelevant site conditions (text notes, hatches, *etc*).

Use the Takeoff – Define Layer Surface/Material/Subgrade command and review/set the dialog boxes similar

to that shown below:

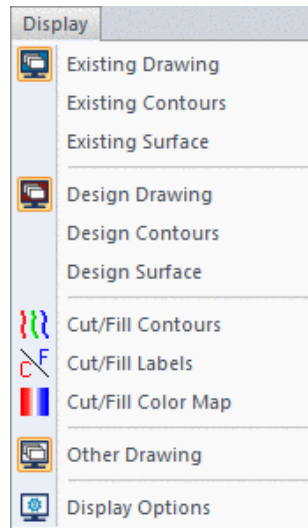


Example Digitizing Layer Targets

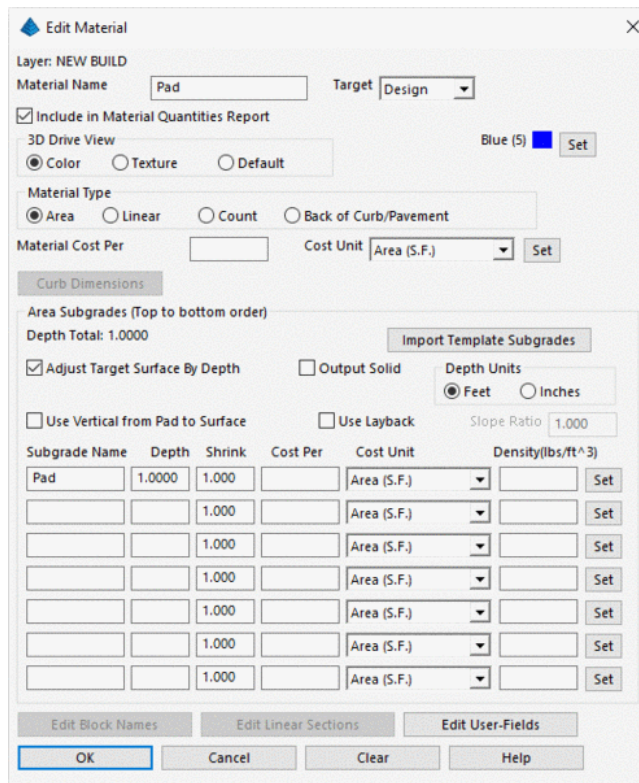
To switch between lists, pick the tabs at the top of the dialog. Check that your Layer Targets resemble the three lists shown here. If a layer is out of place, highlight it, and click the **Move To** button after selecting the correct target to send it to.

Now that the layer targets are defined, there are several commands that can be applied. In the Display menu (SiteNET in Civil), you can turn on/off whether to display layer targets by using the Existing Drawing, Design Drawing and Other Drawing commands:

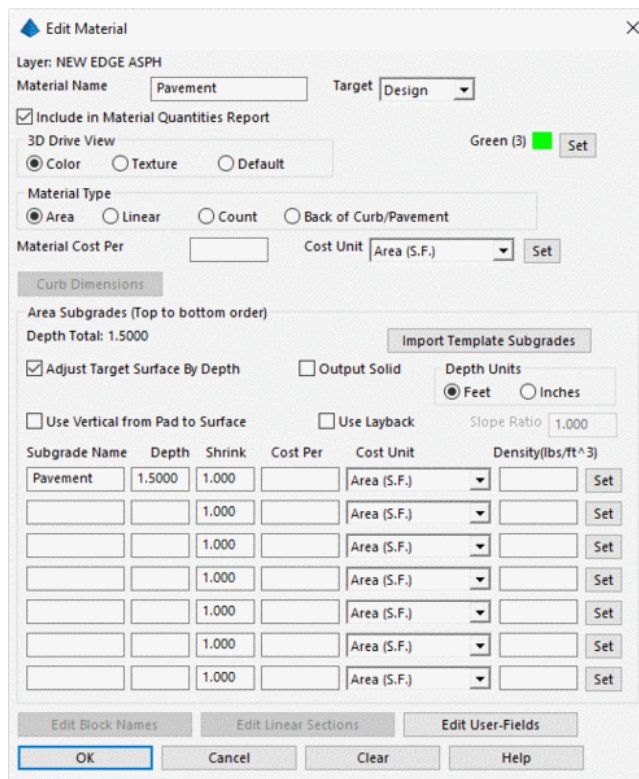
Practice turning on/off the **Existing Drawing**, **Design Drawing** and **Other Drawing** through the Display menu:



11. Besides assigning target surfaces by layer, layers are also used to define material names and subgrade depths. By assigning material names and depths to layers, the volume, area, length and count for entities on these layers can be reported. Also the depth is used to vertically adjust the design surface. The polylines used for subgrade depth must be closed polylines. Takeoff supports nested subgrade polylines for exclusion areas such as islands by counting how many subgrade polylines surround an area. If the number is odd, then the area is inside the subgrade. Otherwise the area is not part of the subgrade. Activate the **Design** tab. Select/activate the **NEW BUILD** layer and click the **Edit Layer** button. A dialog appears for defining what will be the *Pad* material properties. Set the values as shown below and click **OK** when ready:

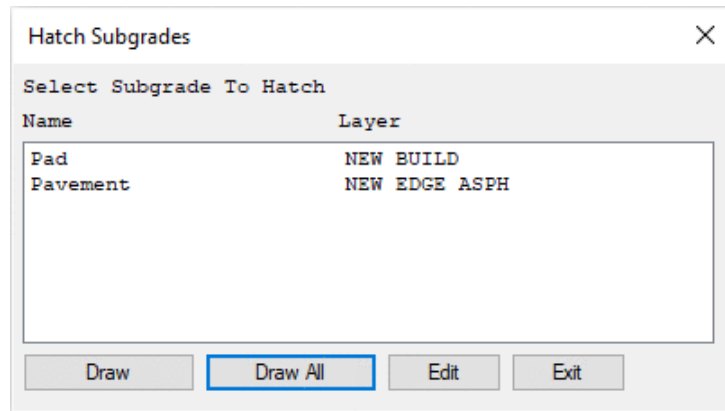


Next pick layer **NEW EDGE ASPH** and click the **Edit Layer** button. A dialog appears for defining what will be the *Pavement* material properties. Set the values as shown below and click **OK** when ready:



After the modifications have been made, pick **Save** and **Exit**.

Now let's visually verify the subgrade areas. Issue the Takeoff (or SiteNET) – Subgrade Areas – Hatch Subgrade Areas to display the dialog box below:

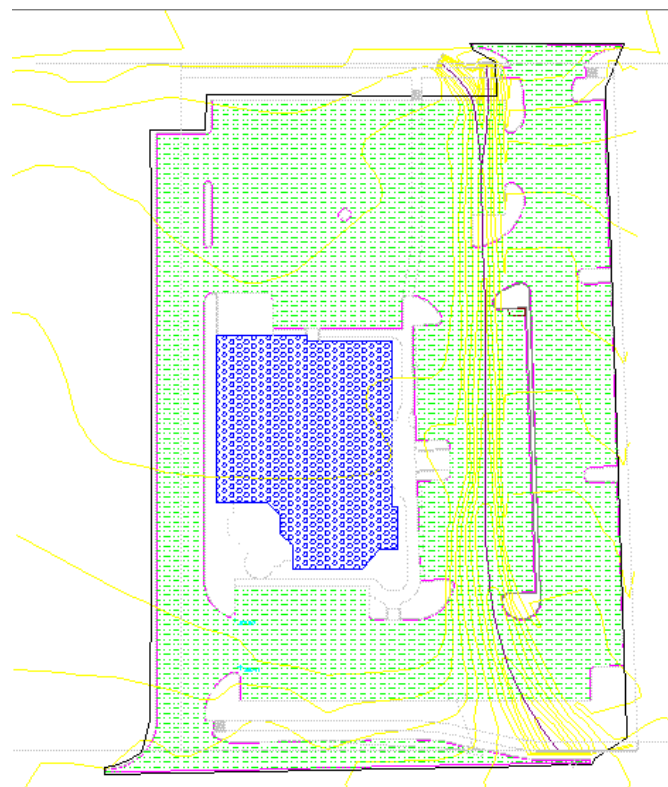


Select a desired subgrade layer and click the **Edit** and establish the following settings:

Subgrade	Hatch Pattern	Color
Pad	Hex	Blue
Pavement	Asphalt	Green

Subgrade Hatch Settings

Click **Draw All** on the dialog box above:



The resulting hatch areas show where the subgrade is applied. Notice how the islands are not hatched because they are curb polylines that are already inside another curb polyline. Takeoff – Subgrade Areas – Erase Subgrade Hatches command.

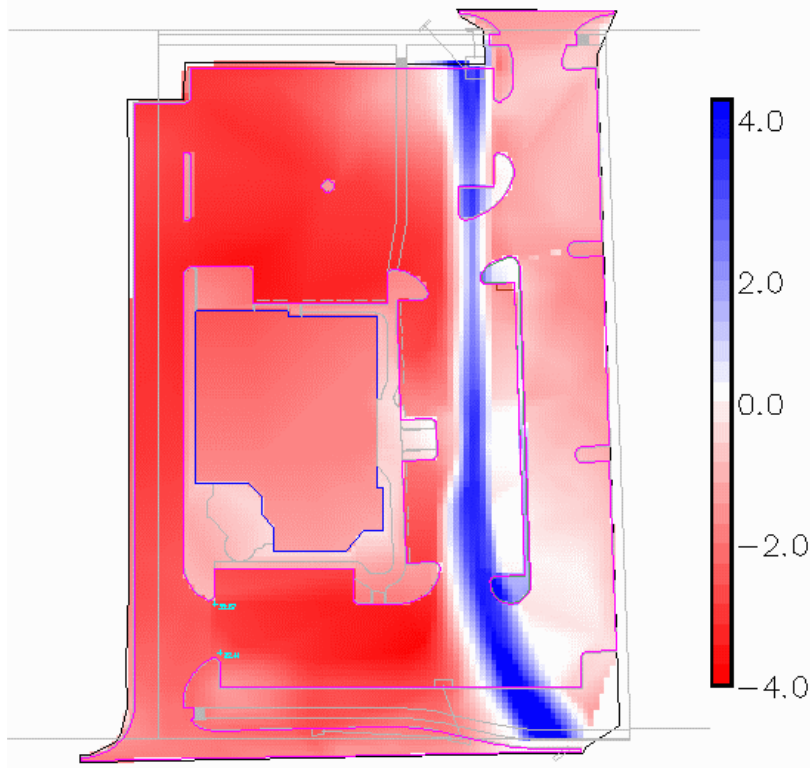
12. To calculate volumes, the program needs two surfaces:

- existing ground, and,
- design considerations

These surfaces are modeled by triangulation. With the preparation of the previous steps, we're now ready to

make the models. The drawing entities have been cleaned up, assigned elevations and assigned target surfaces by layer. Making each model is now a one step process:

- To make the Existing Ground surface, issue the Takeoff – Make Existing Ground Surface. The program will process the entities (regardless of their screen visibility) and make the triangulation surface.
 - To make the Design Ground surface, issue the Takeoff – Make Design Surface.
13. Cut/Fill color maps can be used for a visual output of the site cut/fill areas and also serves as another check that the models are correct. Issue the Display – Cut/Fill Color Map command to display an image similar to that shown below:



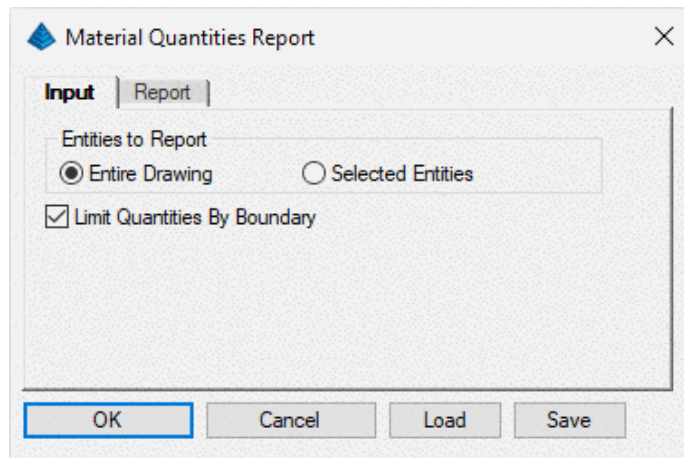
Let's remove the shaded hatch of the Cut/Fill color map through the use of the Display – **Cut/Fill Color Map** command.

14. To determine volumes, issue the Takeoff – Calculate Total Volumes command to display a dialog box similar to that below:

Set the as shown above and click **OK** when ready to display a volume report similar to that shown below:

Click the **Exit** (doorway) button to exit the report viewer.

- To report the material (paving) quantities, issue the Takeoff – Material Quantities – Standard Report routine to display the dialog box below:



Specify the options as shown above (feel free to explore the options in the *Report* tab) and click **OK** when ready. The report includes (among other this) the:

- *Count*, and,
- *Length*, and,
- *Area*, and,
- *Volume*

for each type of material that was assigned for reporting in the previously discussed Takeoff – **Define Layer Target/Material/Subgrade** command and generates a report similar to that below:

Material Quantities

Breakout Quantities

Material	Count	Length(Ft)	Area(S.F.)	Volume(C.Y.)	Shrink	Depth	Cost
Pad	1	409.8	9315.3			1.00ft	
Pad				345.0	1.000	1.00ft	
Pavement	10	3100.8	56318.8			1.50ft	
Pavement				3128.8	1.000	1.50ft	

Sub-Total Quantities

Material	Count	Length(Ft)	Area(S.F.)	Volume(C.Y.)	Tons	Cost
Pad	1	409.8	9315.3	345.0		
Pavement	10	3100.8	56318.8	3128.8		
Total	11	3510.6	65634.1	3473.8	0.0	

Material Parameters

Material	Unit Cost	Shrink Density (lbs/ft^3)
Pad		1.000
Pavement		1.000

Click the **Exit** (doorway) icon when ready. The Takeoff – Material Quantities – **Custom Report** command can be used to report these values with control of the report format and the option to export to Excel.

This completes the tutorial: CADnet Paper Plan Digitizing to Volumes.

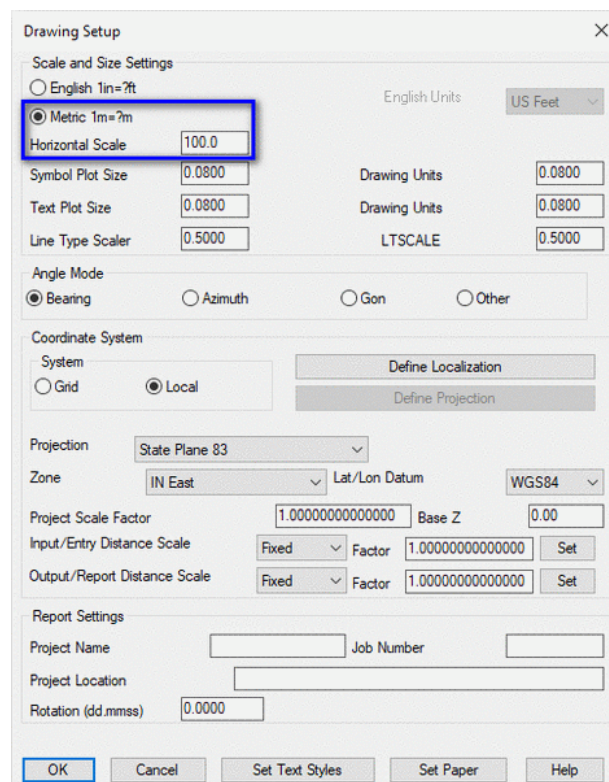
PDF Section Import

This lesson imports cross sections from a PDF into Carlson cross-section format. PDF files can contain different types of data. Some PDF's contain:

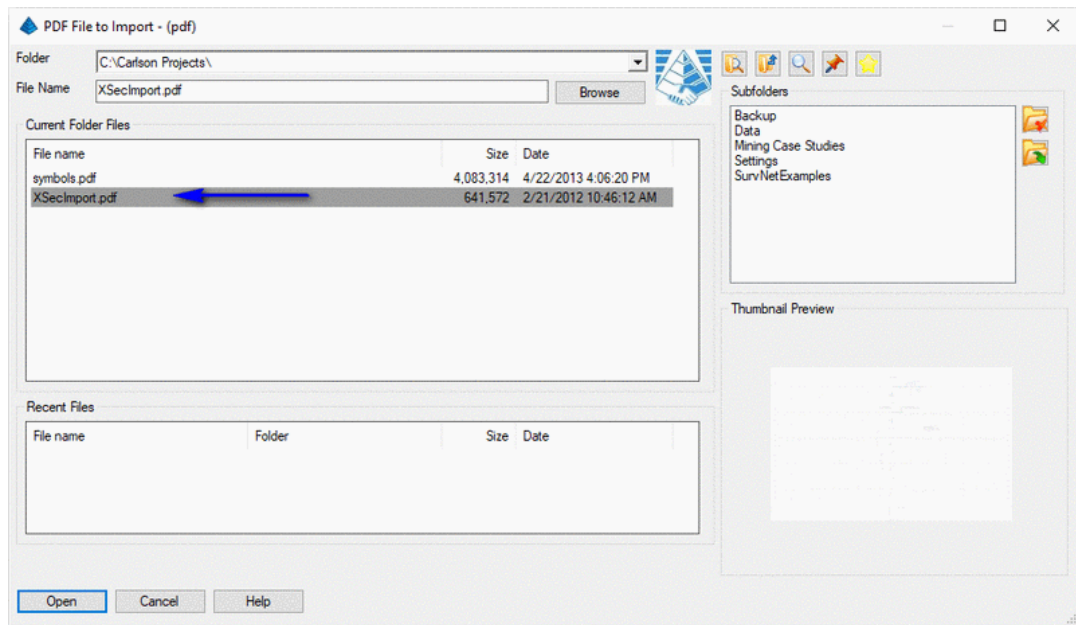
- linework vectors... this is the preferred format as the program can restore the CAD data (*i.e.* polylines) with minimal loss of accuracy and fidelity, and/or
- raster images... depending on the scan resolution used to represent the data, it may be possible to perform a raster-to-vector conversion process to create linework in the drawing. Many times, the scan resolution is not sufficient and the operator might be relegated to either raster-trace the data or perform on-screen digitizing.

In this example, the PDF file contains vector information.

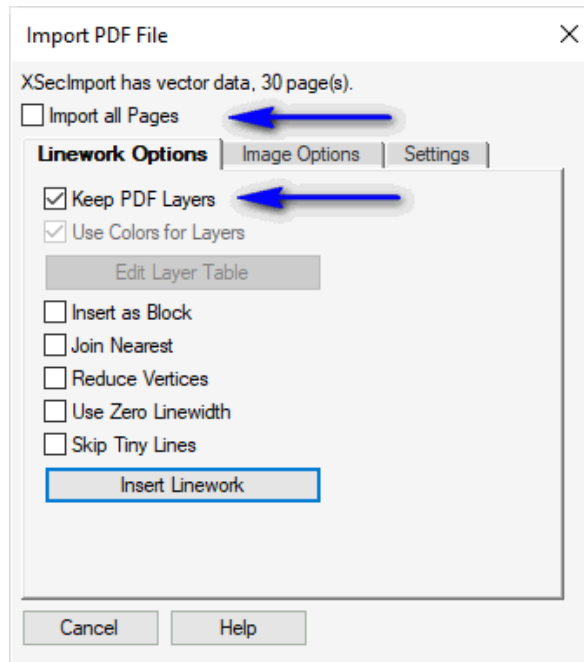
1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **New Drawing**.
 - If you get the Startup Wizard dialog box, click the **Exit** button.
 - If you are taken directly into CAD, continue as is.
2. Activate the CADnet menu via Settings – Carlson Menus – **CADnet Menu**. Additionally, issue the Settings – Drawing Setup command to set the desired values as shown below for this project and click **OK** when ready:



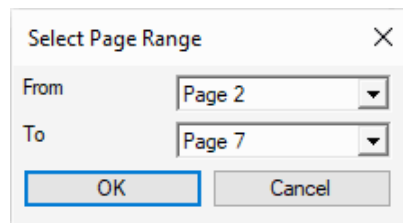
3. With an empty drawing, issue the PDF – Import PDF File to display the dialog box shown below:



Select the file as shown above and click **Open** when ready to display the following dialog box:



NOTE: For the purposes of this tutorial, we do not need to process all pages within the PDF. Set the values as shown above and click the **Insert Linework** button when ready. A secondary dialog box displays as shown below:



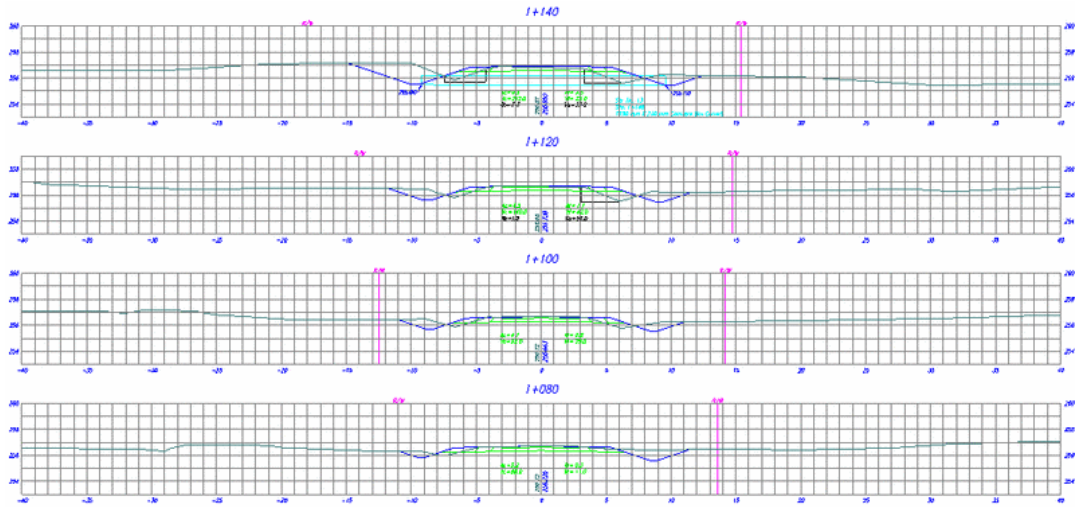
Set the values as shown above and click **OK** when ready. When prompted:

Pick point to insert PDF: pick a point

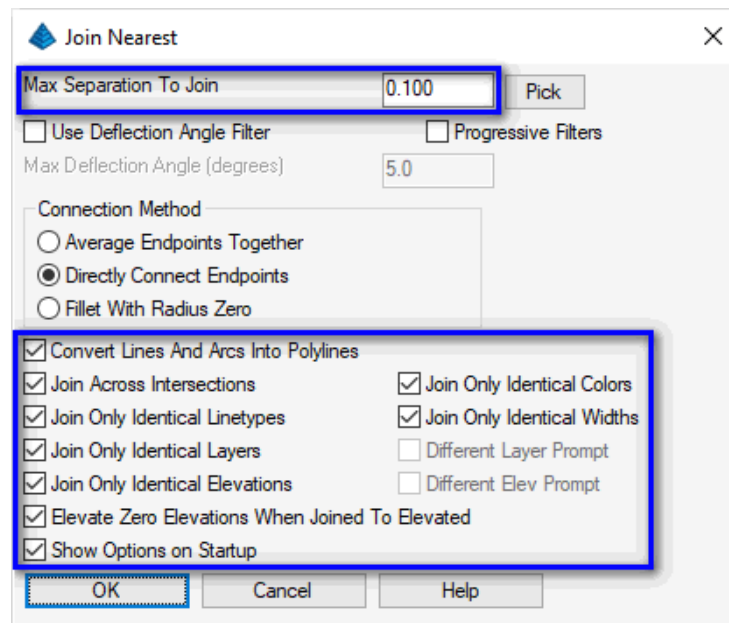
Specify rotation angle <0.0>: 0

Specify scale <1.0>: press Enter

Issue the View – **Extents** to display the imported data as shown in the example below:



As is often the case with PDF documents, there may be linework that needs to be joined for efficient processing of the PDF document. Issue the Edit – **Join Nearest** command to display the dialog box below:



Set the values as shown above and click **OK** when ready. When prompted:

Select lines, arcs and unclosed polylines to join [Settings].

Select entities: type ALL and press Enter

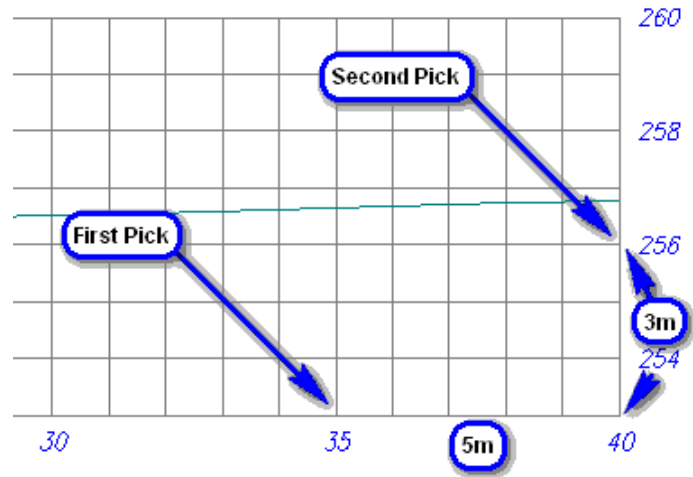
Select entities: press Enter

4. The next thing we'll want to do is validate the horizontal and/or vertical scale based on grid-spacing. Issue

the Inquiry – **Standard Distance** command and when prompted:

Starting point for distance: pick the ENDpoint of the grid line illustrated below

End point: pick the ENDpoint of the grid line illustrated below



With correct picks (and a proper PDF insertion scale from earlier), the corresponding "X" and "Y" distance units should display on the Command line similar to that shown below:

Delta X = 5.00, Delta Y = 3.00, Delta Z = 0.00

Because the distances picked in CAD match with the distances annotated on the grid sheet, no further action is needed.

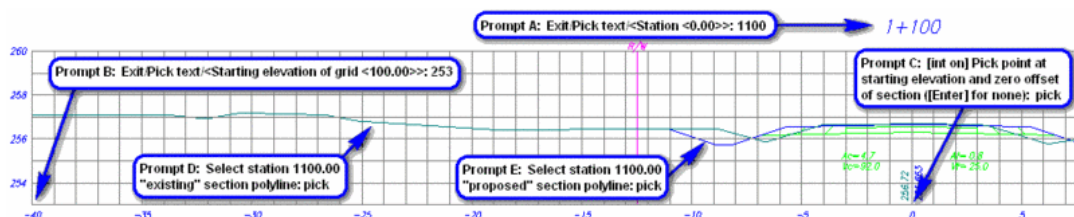
NOTE: In the event that further scaling action is needed, two commands that can help overcome this would generally be either:

- Edit – Scale – Scale Wizard command, or the,
 - Edit – Align – 2D Align command.
5. With proper polyline vector data and a correct scale, we can now make use of this data to create physical cross-section data sets. For the purposes of this tutorial, we'd like to process 400-meters of cross-section data from 1+100 - 1+500. Depending on your application, activate one of the two section-processing modules through the:
- Settings – Carlson Menu – **Construction Menu** (commands found in the *Roads* menu)
 - Settings – Carlson Menu – **Civil Menu** (commands found in the *Sections* menu)

For the purposes of the discussion below, we will make the assumption that *Carlson Construction/Takeoff* is being used.

6. Issue the Roads – Create Sections – Sections from Polylines on Section Grids – Individual Selection command to display a dialog box similar to that shown below:

Set the values as shown above paying particular attention to the highlighted items and click **OK** when ready. A series of prompts will be presented for each cross-section station as illustrated below:



When prompted for the first section:

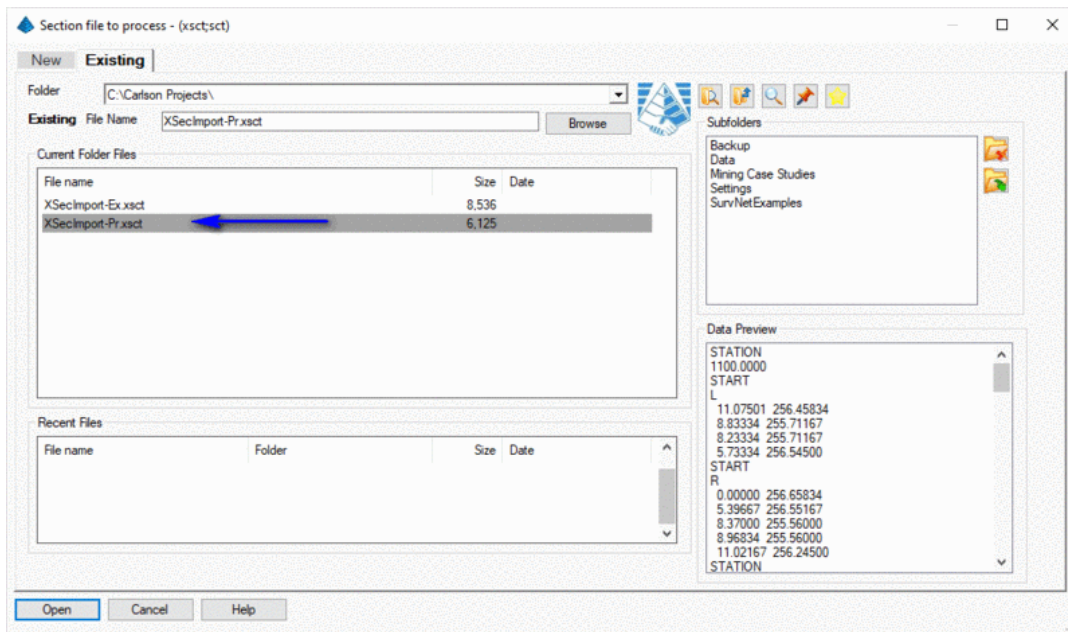
- Exit/Pick text/Station <0.00>>**: type 1100 and press Enter
- Exit/Pick text/Starting elevation of grid <100.00>>**: type 253 and press Enter
- [int on] Pick point at starting elevation and zero offset of section ([Enter] for none)**: pick the starting grid elevation at the 0 offset
- Select station 1100.00 [section file 1] section polyline**: pick the appropriate polyline
- Select station 1100.00 [section file 2] section polyline**: pick the appropriate polyline

The sequence will restart for the next expected cross-section station. Accept the default value or override with a desired value.

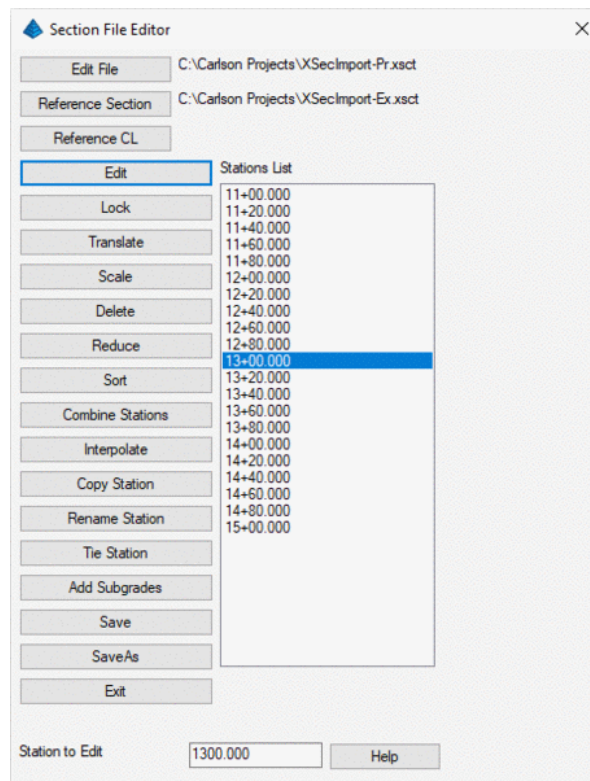
NOTE: There will be a starting grid elevation changes starting at **1+300!** Make the appropriate adjustment during the prompting!

Pan the drawing to see the next available cross-section station and note its starting grid elevation. Repeat the sequence until you reach station **1+500** and then issue the **Exit** to complete the conversion process.

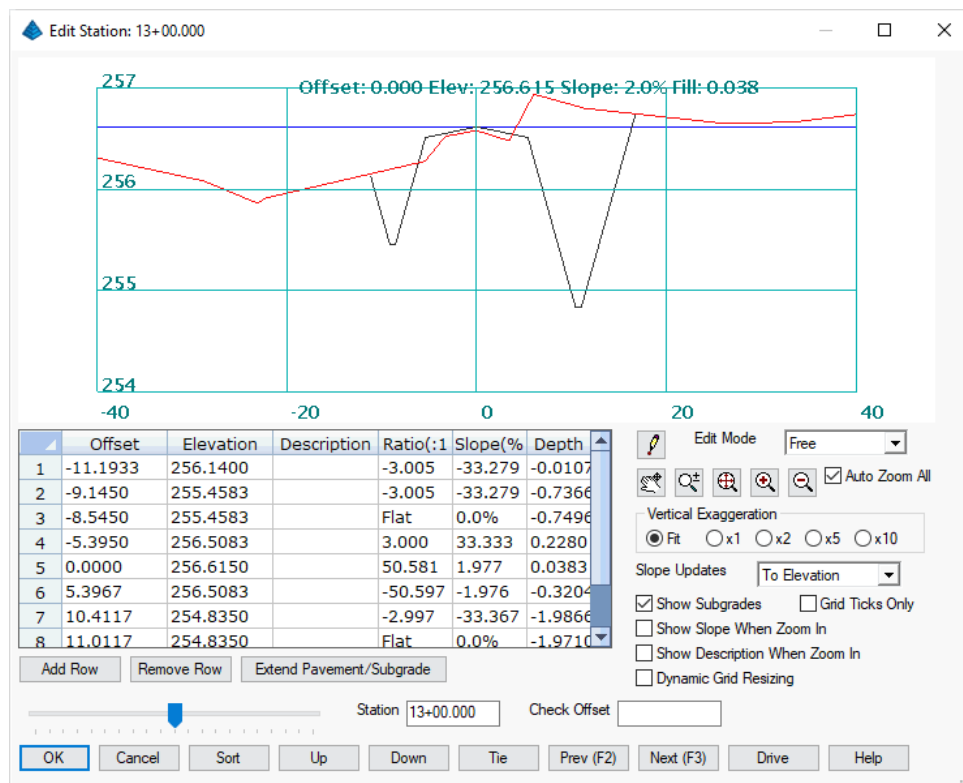
- When complete, let's review/check the section data. Issue the Roads – Section File Editor command to display a dialog box similar to that shown below:



Select the file as illustrated above and click **Open** when ready to display a dialog box similar to that shown below:



Set the **Reference Section** file as shown above, highlight a desired to inspect and click the **Edit** button to display a dialog box similar to that shown below:



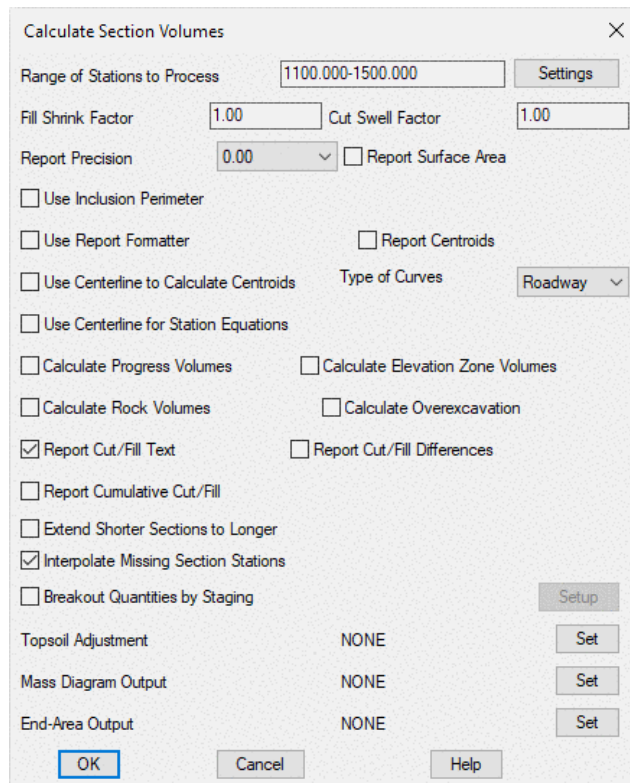
The command shows a graphic preview of the sections along with a spreadsheet of the *Edit File* values. Explore the remaining controls on this dialog box and click the **Cancel** button to dismiss it when ready. Click the **Exit** button to dismiss its parent dialog box.

- Now that we have two section files, we can derive the volume between them. Issue the Sections – Calculate Section Volumes command. When prompted:

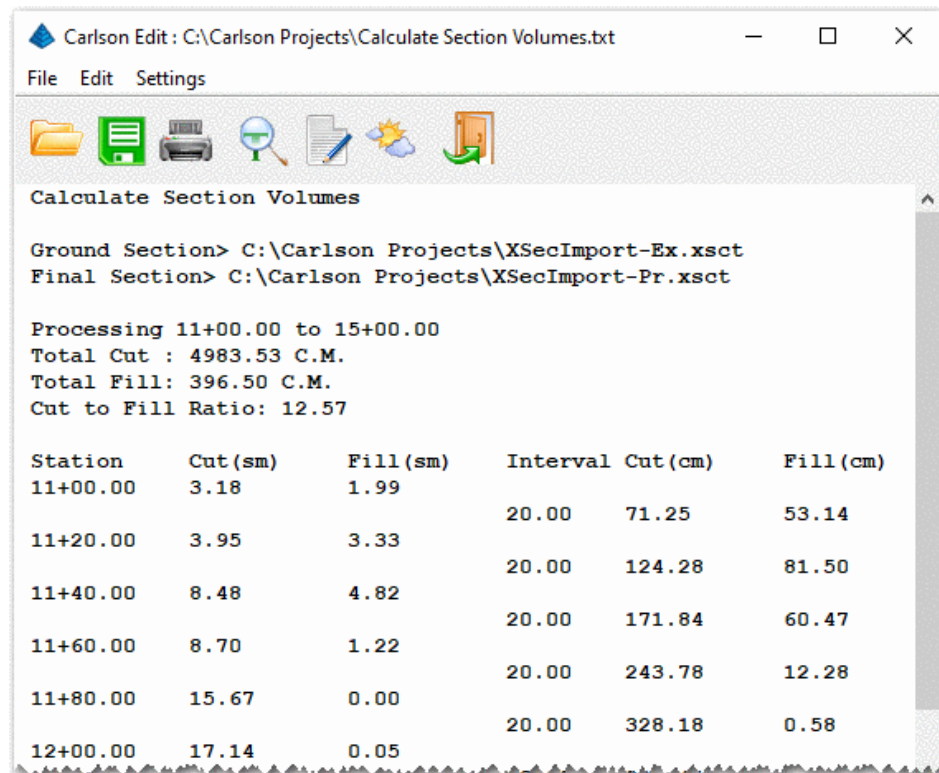
Section File (Existing Ground) to Read (dialog): select the *-Ex* section file and click **Open** when ready

Section File (Final Ground) to Read (dialog): select the *-PR* section file and click **Open** when ready

Calculate Section Volumes (dialog): set the values as shown below and click **OK** when ready



The volumes are calculated and reported, along with the cut and fill end-areas at each station as illustrated below:



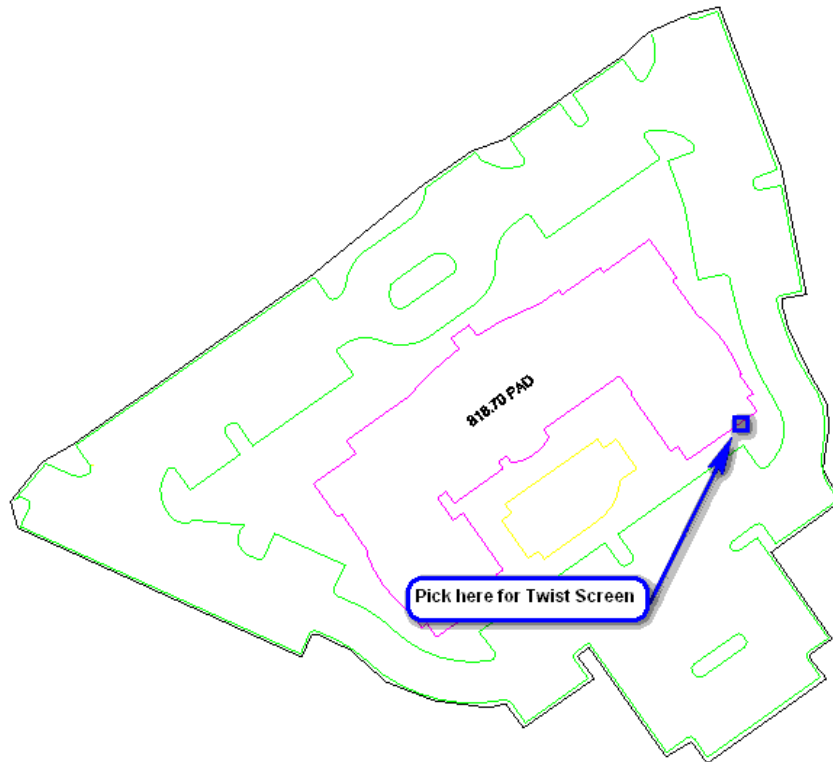
Review the results and click the **Exit** (Doorway) button to dismiss the report.

This completes the tutorial: CADnet PDF Section Import.

GeoTech Reports

This lesson creates and reports drillhole data.

1. This drillhole lesson builds on the results of the Takeoff/SiteNET Basics tutorial. Before continuing with this tutorial, complete that tutorial up through the Make Existing Ground Surface/Make Design Surface section of that tutorial. When the **Takeoff/SiteNET Basics** tutorial is completed, we're now ready to add drillholes for the project.
2. Activate the GeoTech menu via Settings – Carlson Menus – **GeoTech Menu**. For mapping the drillholes, we don't need to see all the drawing layers. Run the View – Freeze Layer By Pick command and pick on a contour polyline and a curb elevation label. In this case, the contours are on a few different layers. As a result, re-run the **Freeze Layer By Pick** command and keep picking contour polylines and labels until all contours are gone and the drawing resembles that shown below:

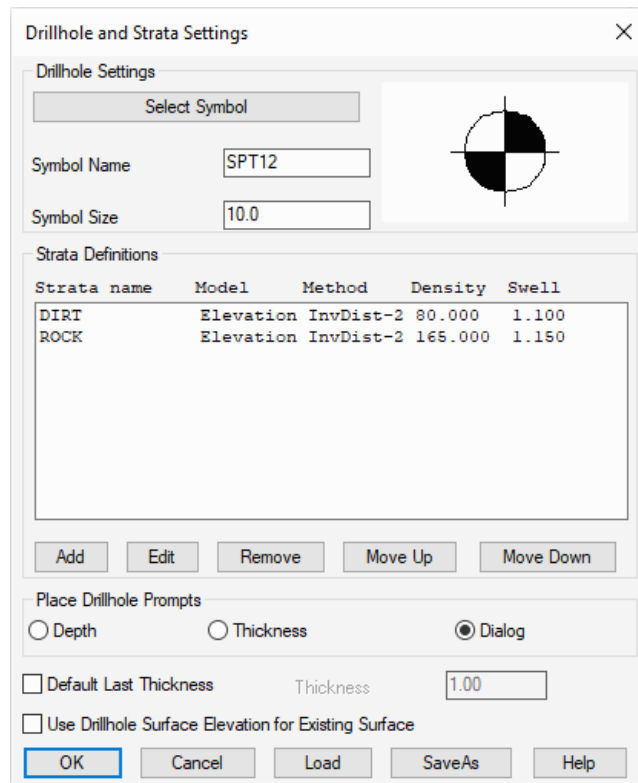


Next, let's orient the drawing so that the building better aligns with the screen. Issue the View – Twist Screen – Line, Polyline or Text command. When prompted:

Pick a line, polyline or text: pick on the northeast end of one of the building as shown above
Twist azimuth for selected LWPOLYLINE (Use 90 for Due East) <90.0>: press Enter

The drawing view is changed (and the coordinates remain as-is).

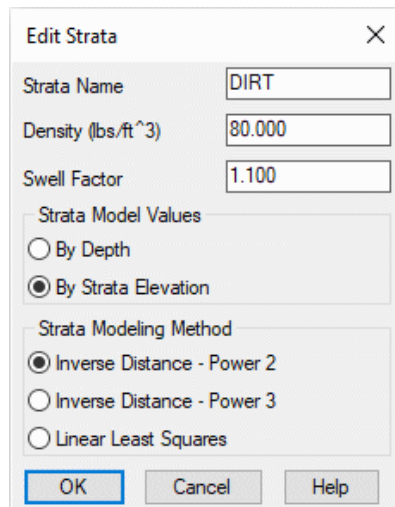
3. Issue the Drillhole – Define Drillhole and Strata Settings (*aka* **Define Drillhole**) command to display the dialog box similar to that shown below:



For this tutorial, we need to define two strata:

- (a) **DIRT** (material above the rock), and,
- (b) **ROCK**

Click the **Add** button to create a material strata as described below and click **OK** when ready:



Next, pick the **Add** button again. This time, fill out the dialog with a strata name of "ROCK" and use the information below to add the second strata material into the project and click **OK** when complete:

Values	Strata 1	Strata 2
Name	CLAY	LIMESTONE
Density	80	165
Swell Factor	1.10	1.15
Model Values	By Elevation	By Elevation

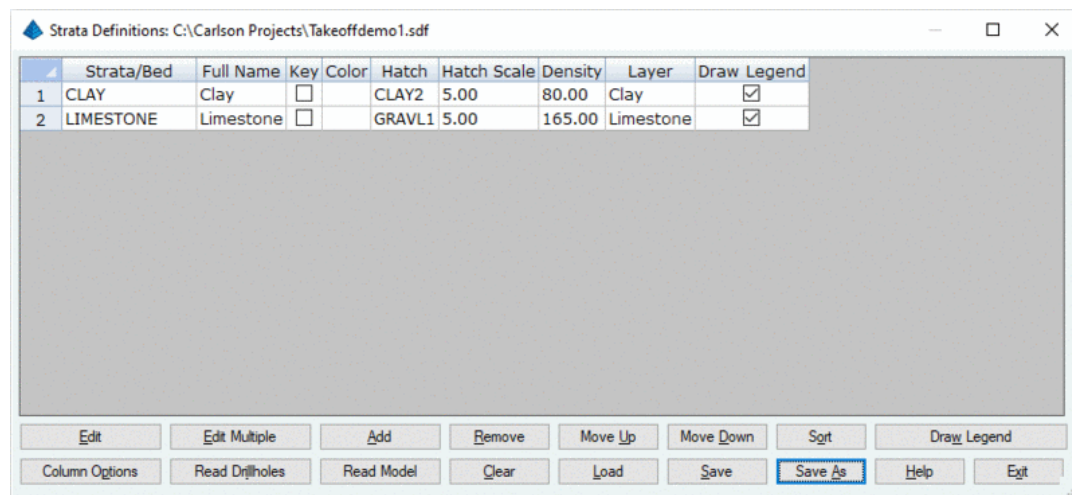
Strata Definitions

NOTE: There are three options of modeling methods:

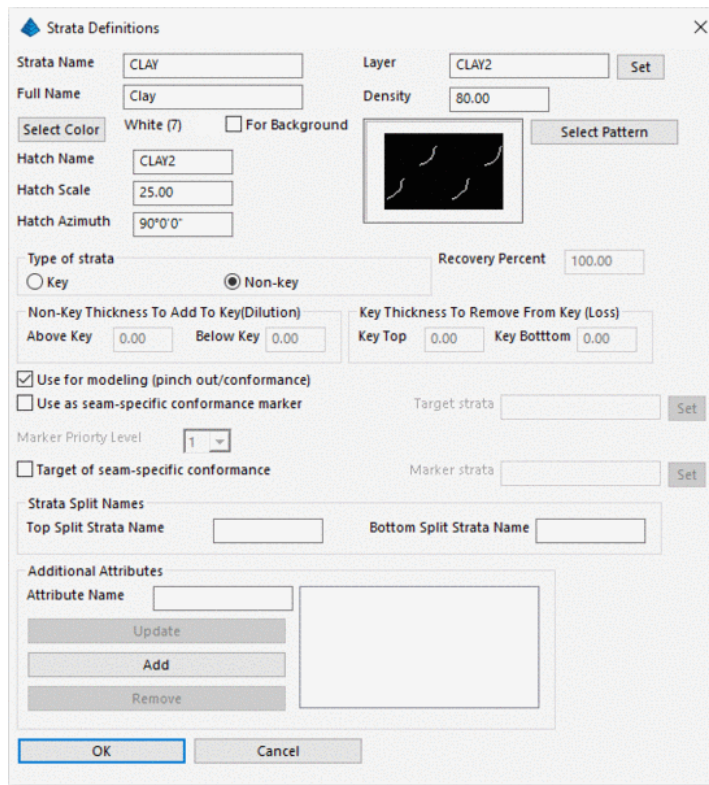
- **Inverse Distance** will not carry trends and the calculated strata model will never be higher or lower than the original drillhole data and uses a weighted average of the drillhole data. In general, closer drillholes are weighted more than drillhole farther away:
 - **Inverse Distance - Power 2** will weigh drillholes more that are further away.
 - **Inverse Distance - Power 3** will weigh drillholes less that are further away.
- **Linear Least Squares** extrapolates trends and allows for a strata model to create new highs and lows that don't appear in the original drillhole data.

The *Strata Materials* in the main dialog need to be in top to bottom order. To change the order, highlight a strata name and use the **Move Up** or **Move Down** buttons. In this case, we want *DIRT* then *ROCK*. Click **OK** from the main dialog.

4. Next, we'd like to define the label name, hatch pattern and color for the strata. Run the Drillhole – Define Strata/Bed command to display a dialog box similar to that shown below:



Click **Add** and fill out the data as shown below and click **OK** when ready:

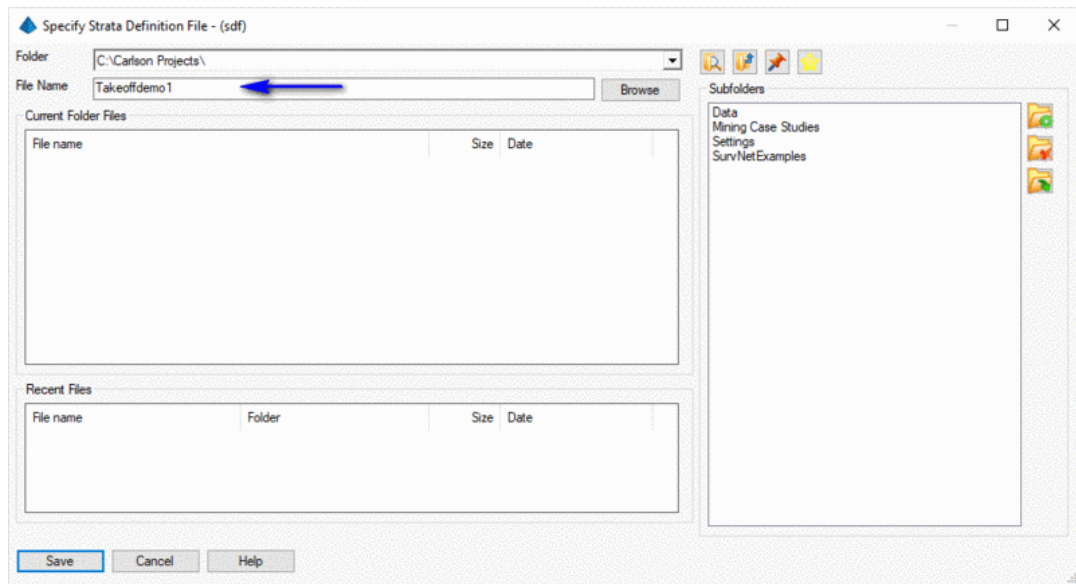


Use the table below for the specific values:

Values	Strata 1	Strata 2
Strata Name	CLAY	LIMESTONE
Full Name	Clay	Limestone
Layer	Clay	Limestone
Density	80	165
Hatch Name	CLAY2	GRAVL1
Hatch Scale	5.00	5.00
Hatch Azimuth	90°0'00"	90°0'00"

Strata Definitions

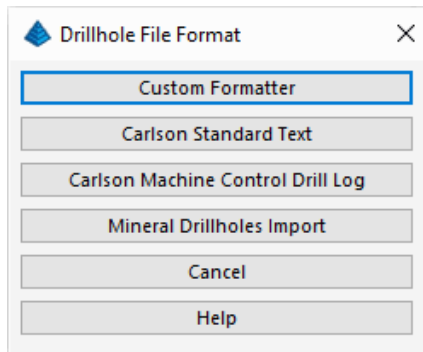
Once created, pick **SaveAs** and when prompted:



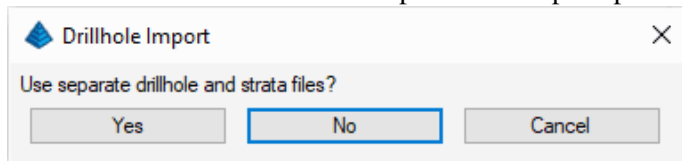
Supply the name as shown above and click **Save** when ready. Finally, click the **Exit** button to dismiss the **Define Strata/Bed** dialog box.

5. There are two different methods for entering drillhole data:
 - **Drillhole Import** which reads the drillhole data from a text file. This command supports customizing the sequence of drillhole data fields to match the format of the text file.
 - **Place Drillhole** creates the drillholes at picked positions in the drawing and enters the data in a dialog. This process is discussed in the Takeoff Drillholes and Strata tutorial.

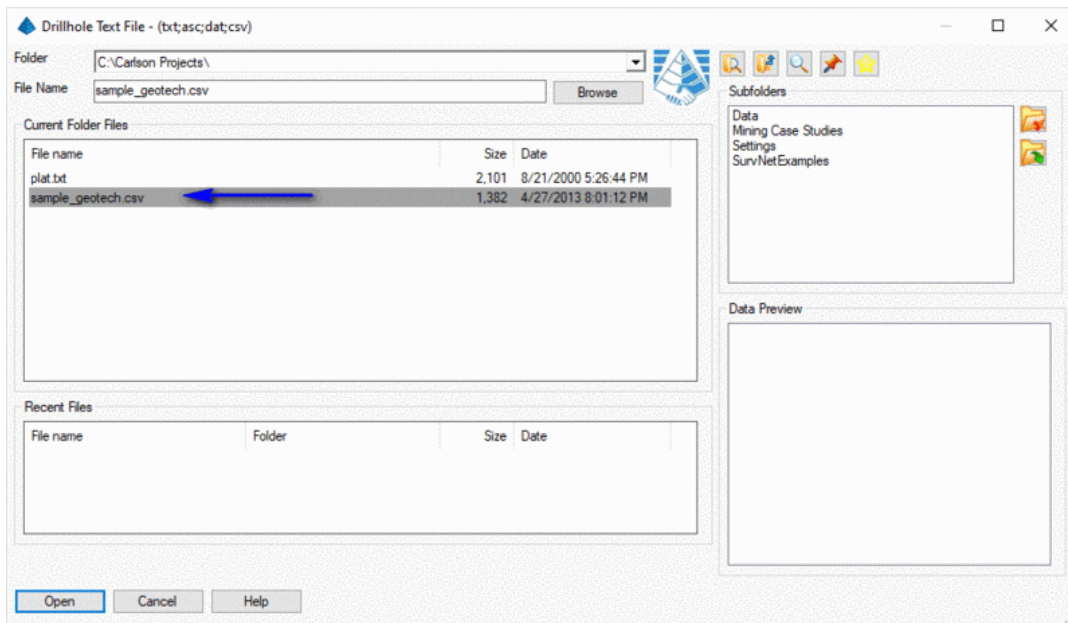
For this tutorial, issue the Drillhole – Import/Export Drillholes – Drillhole Import command. When prompted:



Choose the **Custom Formatter** option. When prompted:



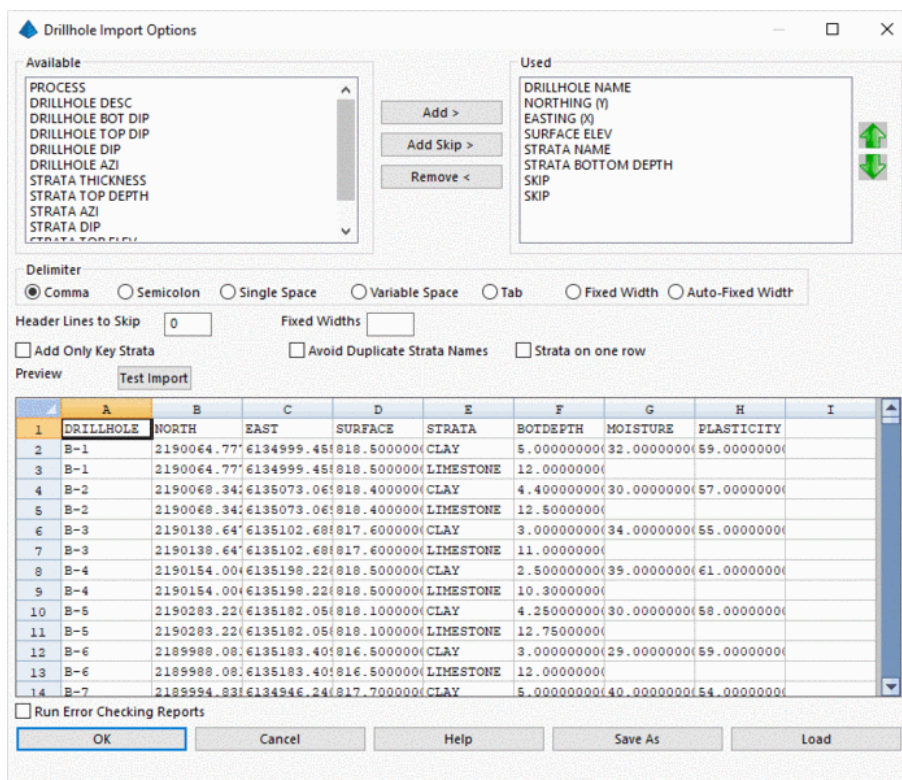
Indicate **No** as both the drillhole and strata data are stored in the same file. For the **Drillhole Text File**, specify the file shown below and click **Open** when ready:



NOTE: The file contains fields of data in the following order:

- (a) DRILLHOLE
- (b) NORTH
- (c) EAST
- (d) SURFACE
- (e) STRATA
- (f) BOTDEPTH
- (g) MOISTURE
- (h) PLASTICITY

Use the **Add >**, **Add Skip >** and/or **Remove <** buttons to fill out the *Used* list at shown and click **OK** when ready:



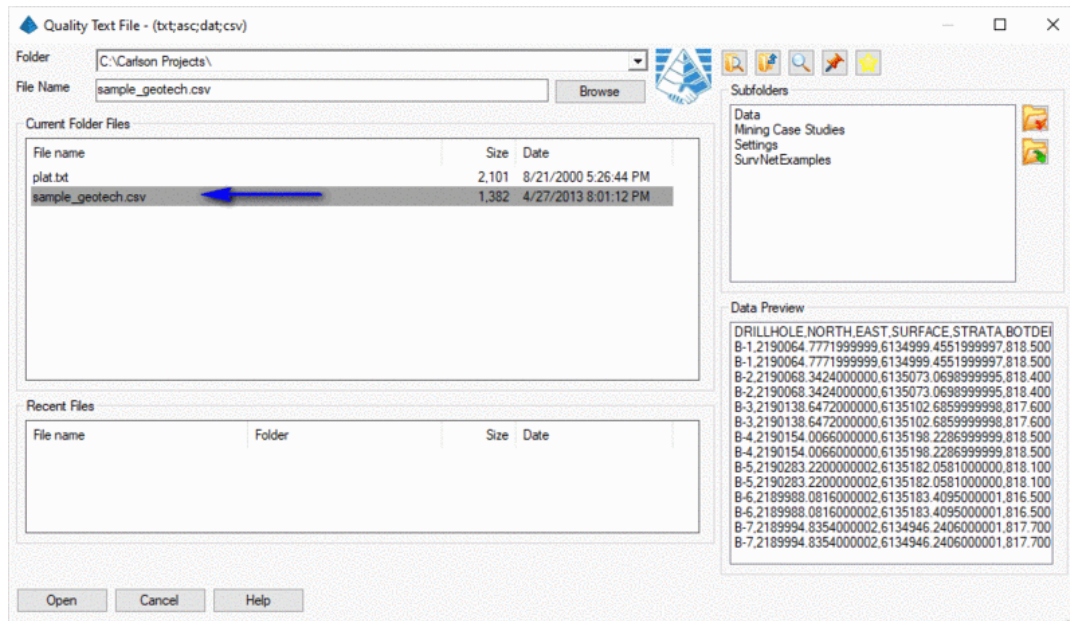
The drillhole data is imported into the project.

- The **Drillhole Import** has brought in the structure for the drillhole and strata but not the attributes (*i.e.* the strata attributes for Moisture and Plasticity). Issue the Drillhole – Import/Export Drillholes – Import Qualities command and when prompted:

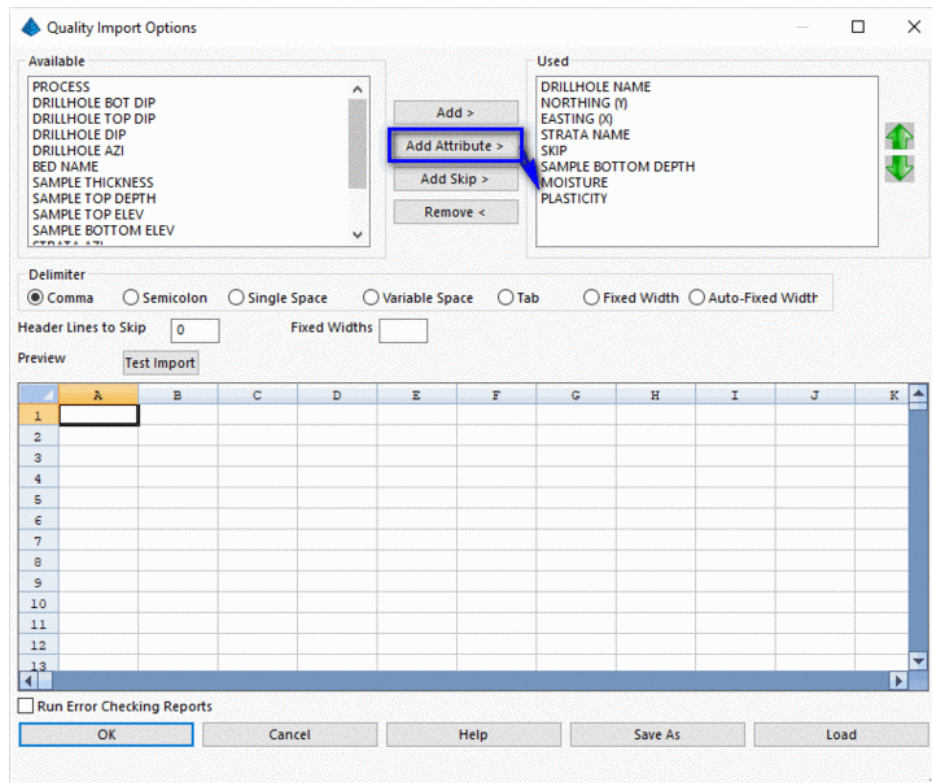
Select Drillholes to update.

[FILter]/<Select entities>: type ALL and press Enter twice

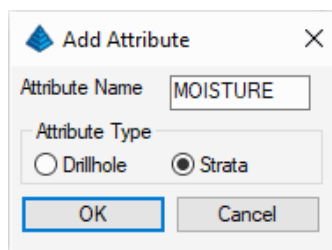
Re-select the drillhole file as illustrated earlier and click **Open** when ready:



A **Quality Import Options** dialog box similar to that shown below appears:

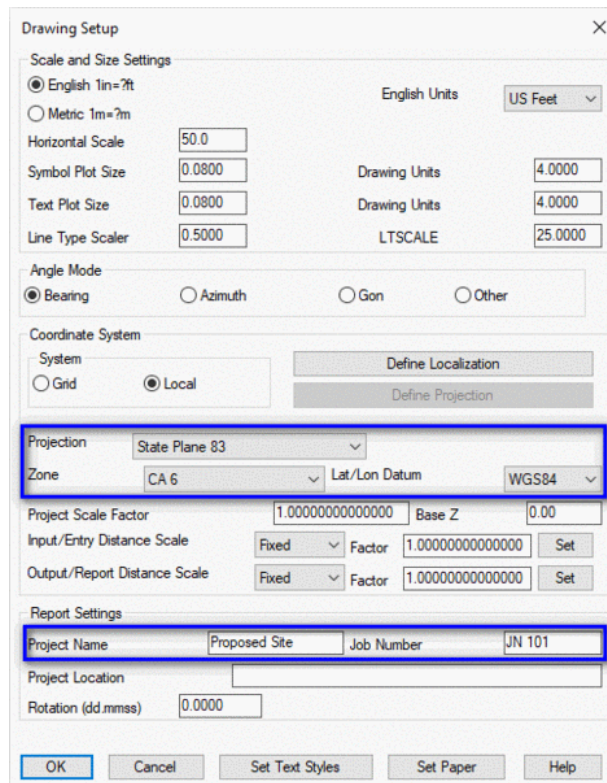


Use the **Add >**, **Add Skip >** and/or **Remove <** buttons to fill out the *Used* list at shown. Use the **Add Attribute >** button to create fields for the *Moisture* and *Plasticity* fields. When creating these fields, indicate they are **Strata** properties as illustrated below (click **OK** when ready):



Lastly, click the **OK** button on the *Quality Import Options* dialog box when ready to import the drillhole qualities.

7. To setup the project name for the reports, issue the Settings – Drawing Setup command to set the values shown below and click **OK** when ready:



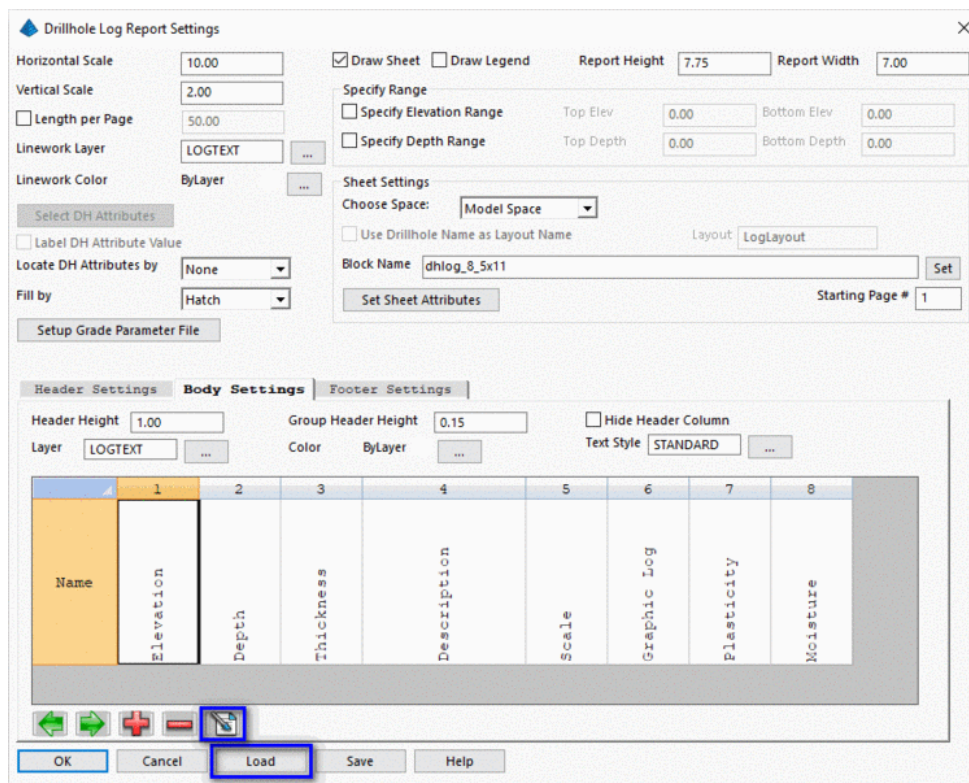
8. To make log reports, issue the Drillhole – Graphic Drillhole Report command. When prompted:

Select Drillhole [**<Select>/Name/Dialog**]: press Enter

Select drillholes

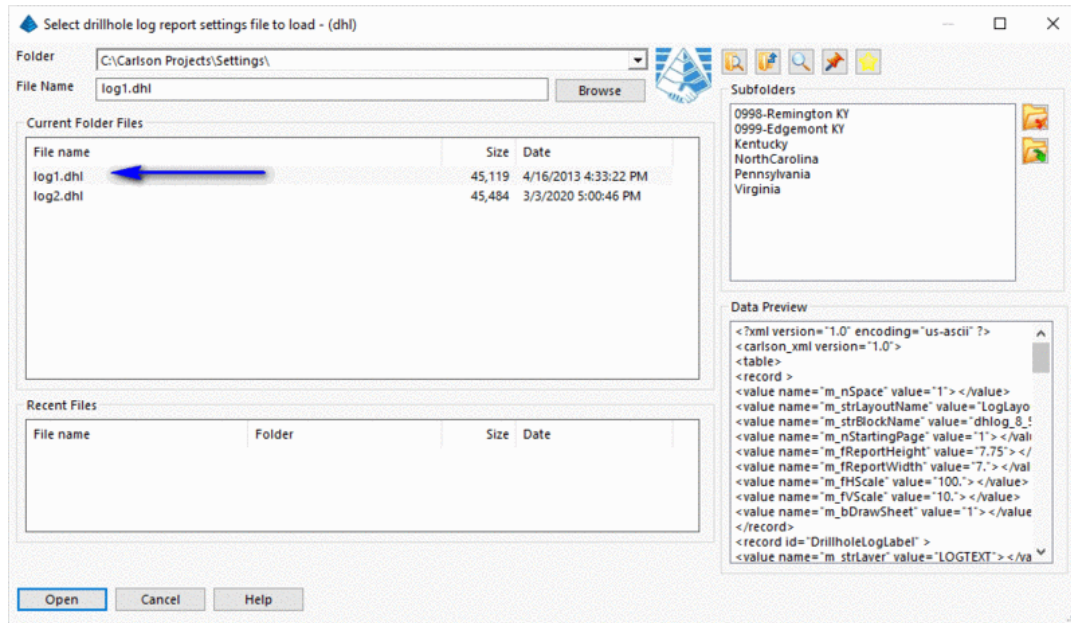
[FILter]/<Select entities>: type ALL and press Enter twice

A dialog box similar to that shown below appears:



NOTE: The contents of the various Drillhole Log columns can be customized through the **Edit Column** button highlighted above.

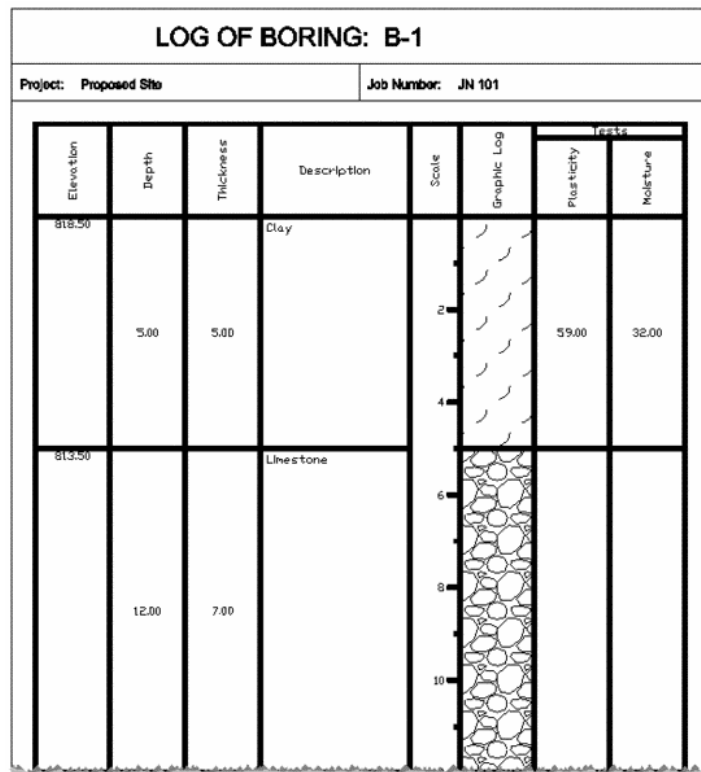
Use the **Load** button to load a predefined Drillhole Log as shown below (click **Open** when ready):



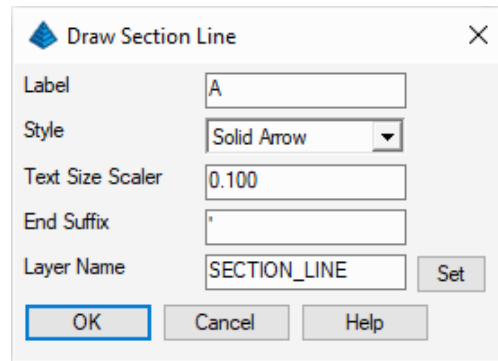
Click **OK** on the *Drillhole Log Report Settings* dialog box. When prompted:

Pick a point to insert log report: pick an empty location within the drawing

An example of the report is shown below:



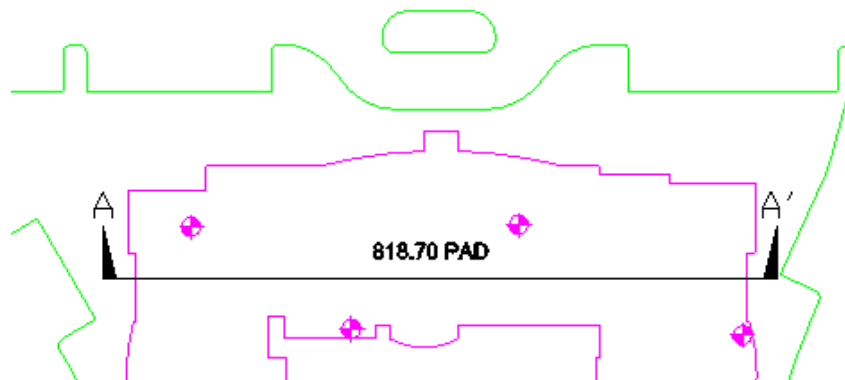
9. Now, let's create a cross section view of the drillholes across the building pad. Issue the Draw – Section Line command to display a dialog box similar to that shown below:



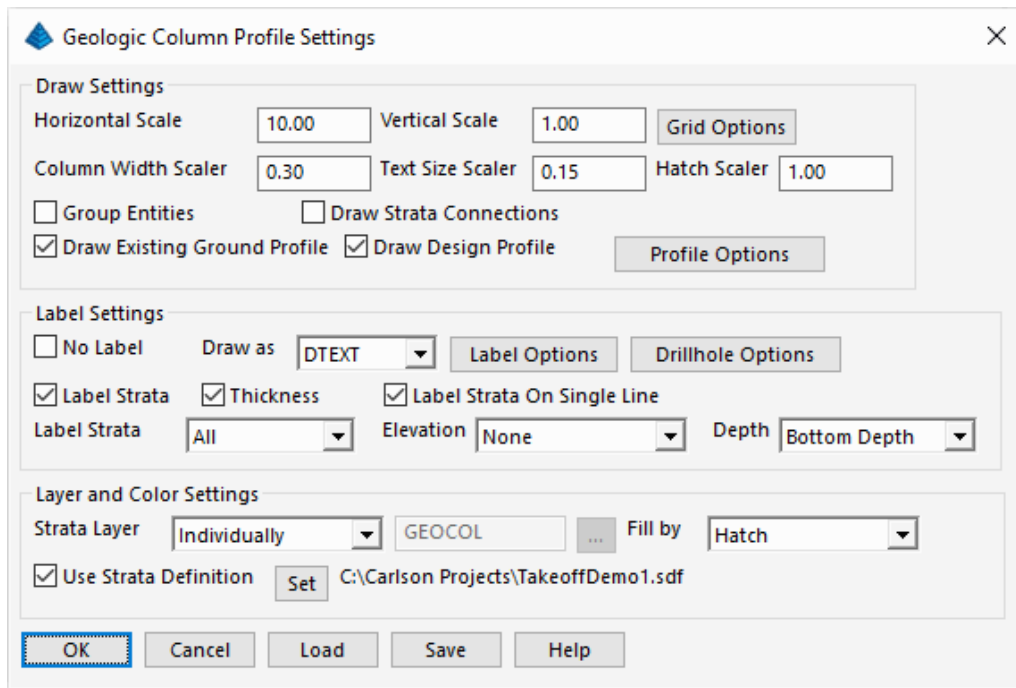
Fill out the dialog as shown above and click **OK** when ready. When prompted:

Pick 1st point: pick a location left of the building

Pick 2nd point: pick a location right of the building as illustrated below

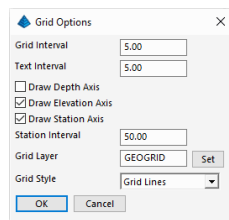


Next, run the StrataCalc – Geologic Column Profile command which displays a dialog box similar to that shown below:

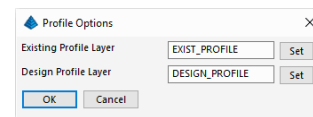


Set the values as shown above. Click the **Set** button to load the file saved in the Define Strata/Bed discussion section above. Use the table of sub-options outlined below within this command to further control the eventual report:

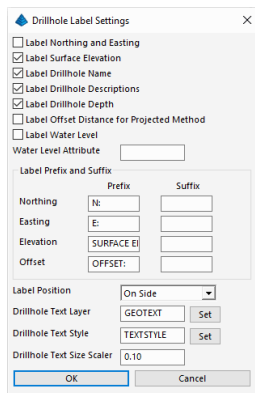
Grid Options



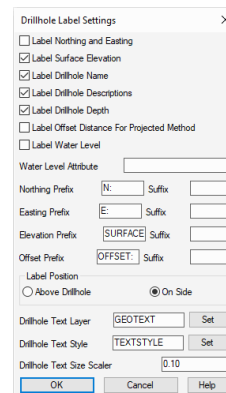
Profile Options



Label Options



Drillhole Options



Select Attributes (Prompt)

Geologic Column Profile Sub-options

Click **OK** when ready. When prompted:

Select Drillholes for geologic column.

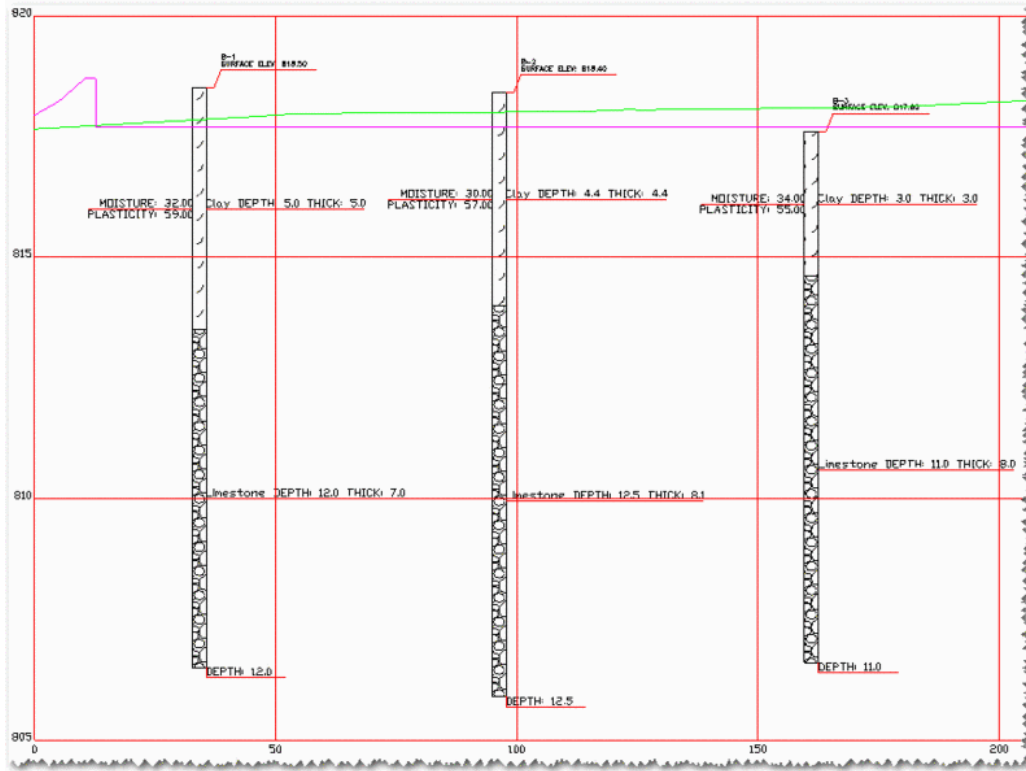
[FILter]/<Select entities>: type ALL and press Enter

[FILter]/<Select entities>: press Enter

Select Fence alignment polyline: pick the section "A" line created earlier

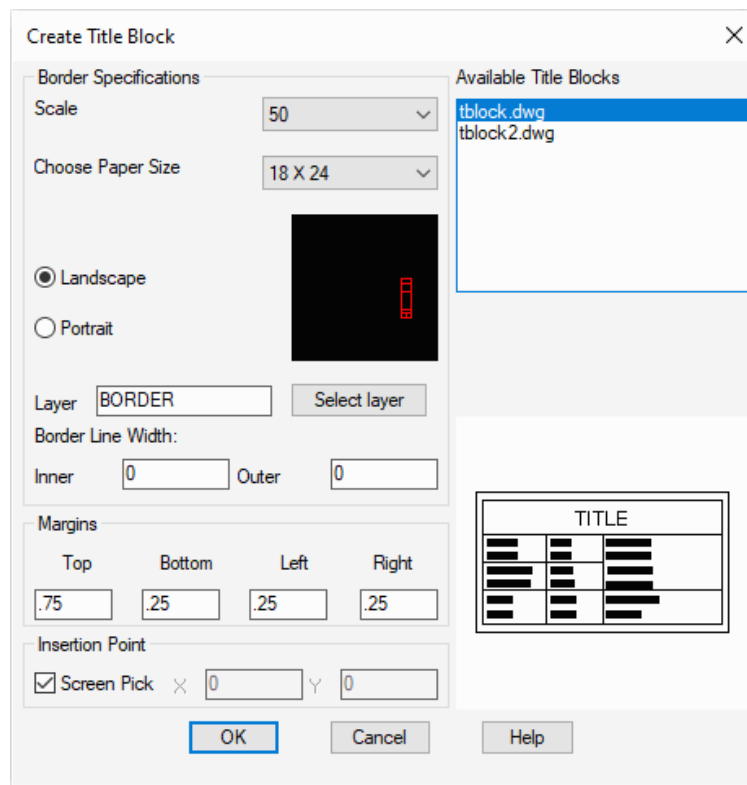
Maximum drillhole distance from alignment polyline <1000.0>: type 100 press Enter
Select Attributes (dialog): select the Attributes cited in the table above and click **OK**
Pick location for geologic column(s): pick a lower left location for the report

The report similar to that shown below is placed into the drawing:



NOTE: As a reminder, complete the **Takeoff/SiteNET Basics** tutorial to see the *Existing Ground* and/or *Design* profiles.

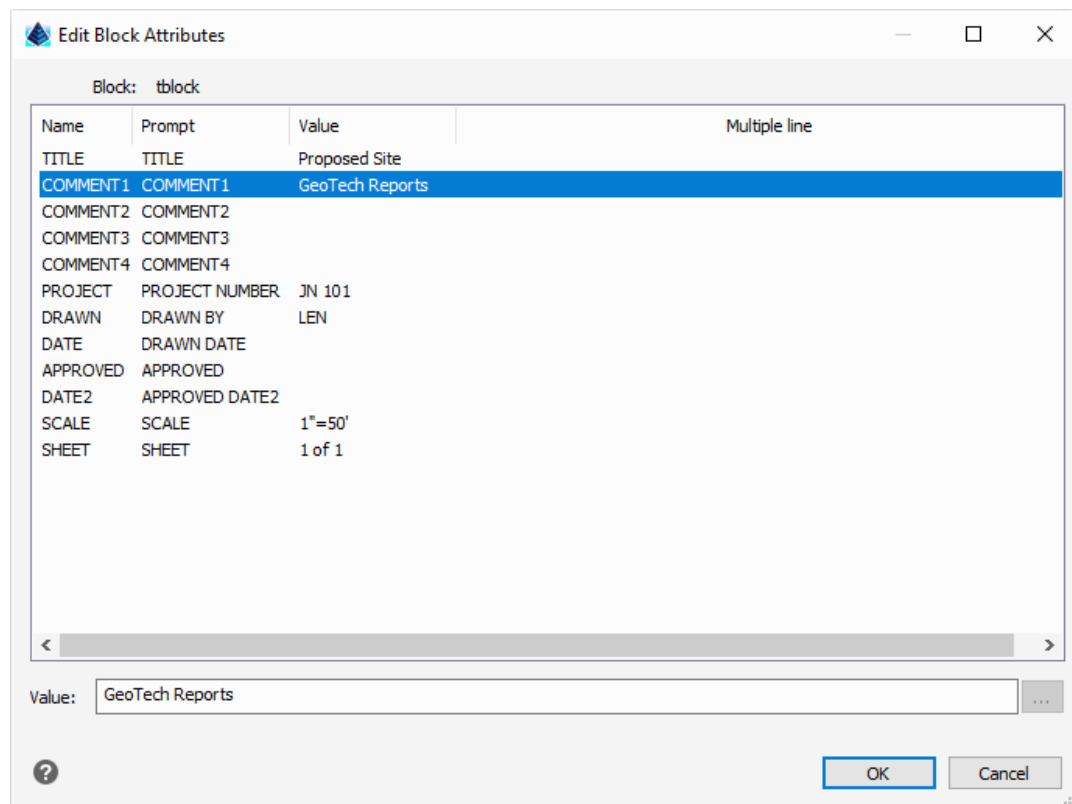
- To prepare for plotting, run the Map – Title Block command (the same as that found under Settings – Title Block) to display a dialog box similar to that shown below:



Set the values as shown above and click **OK** when ready. When prompted:

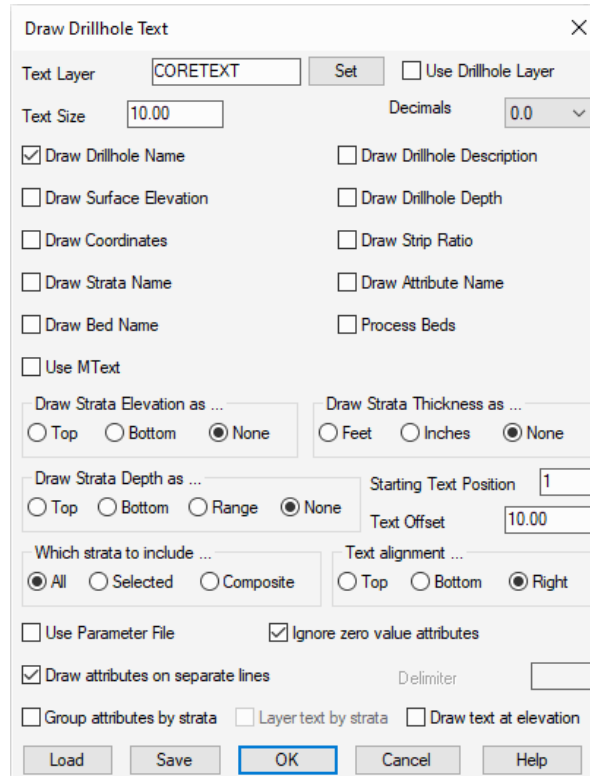
Pick location: pick a desired location for the Title Block

Enter attribute values: supply values as suggested below and click **OK** when ready



11. Let's add some annotation to the drawing for the plan view. Run the Drillhole – Label Drillholes – Standard

Drillhole Text command to display a dialog box similar to that shown below:



Set the values as shown above (a very minimal drillhole label) and click **OK** when ready. When prompted:

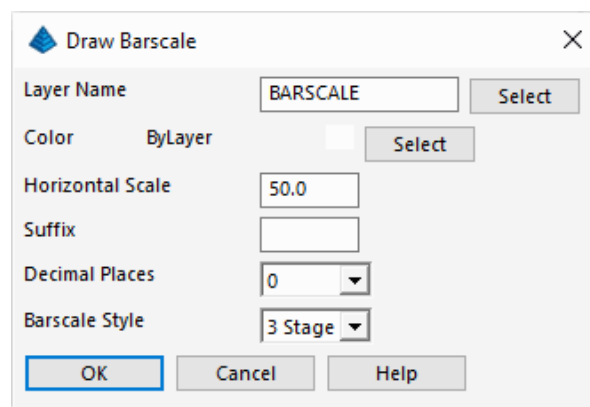
Select the Drillholes to label.

[FILter]/<Select entities>: type ALL and press Enter

[FILter]/<Select entities>: press Enter

Select Attributes (dialog): select the Attributes cited in the table above and click **OK**

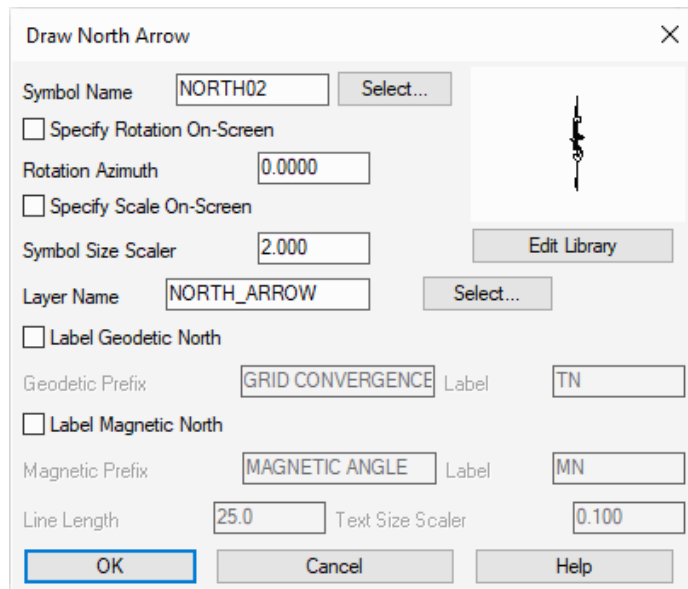
Next, run the Map – Bar Scale command to display the dialog box shown below:



Click **OK** when ready. When prompted:

Pick location for barscale: pick a desired location for the barscale

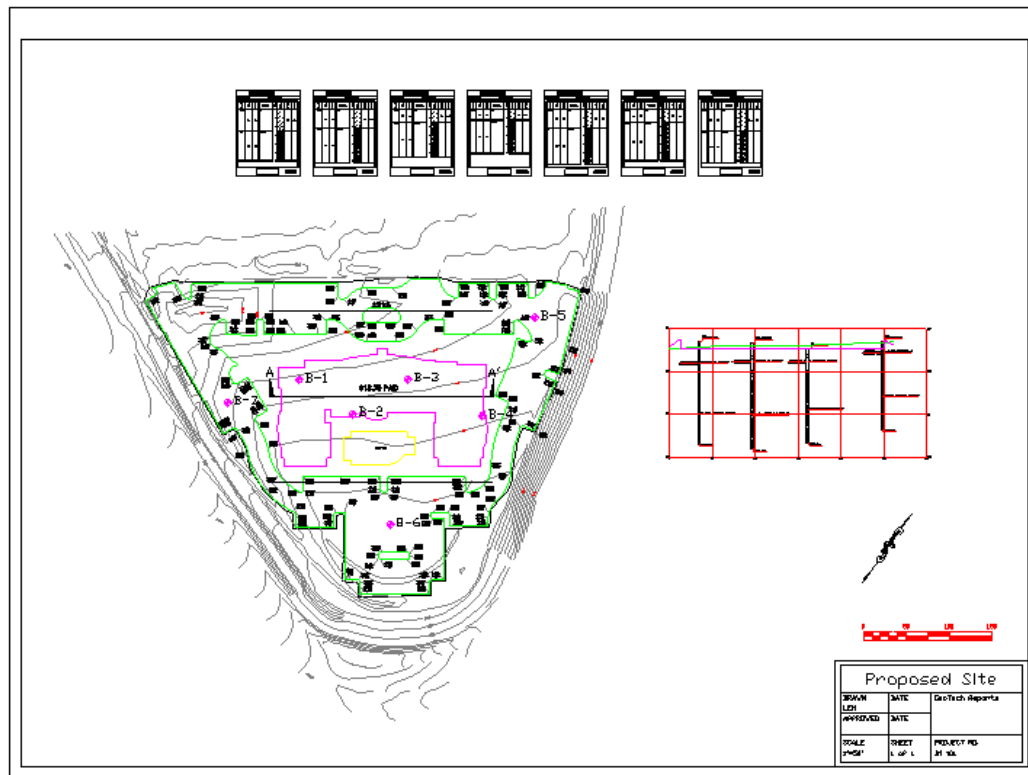
Next, run the Map – North Arrow command to display the dialog box shown below:



Click **OK** when ready. When prompted:

Pick insertion point: pick a desired location for the North arrow

When complete, your drawing could resemble that shown below:



This completes the tutorial: GeoTech Reports.

Cut Sheet

In this lesson you will create a Cut Sheet report for staking grade for a proposed parking lot. This tutorial utilizes the data limits of the Contouring, Break Lines and Stockpiles tutorial. Once opened, you will:

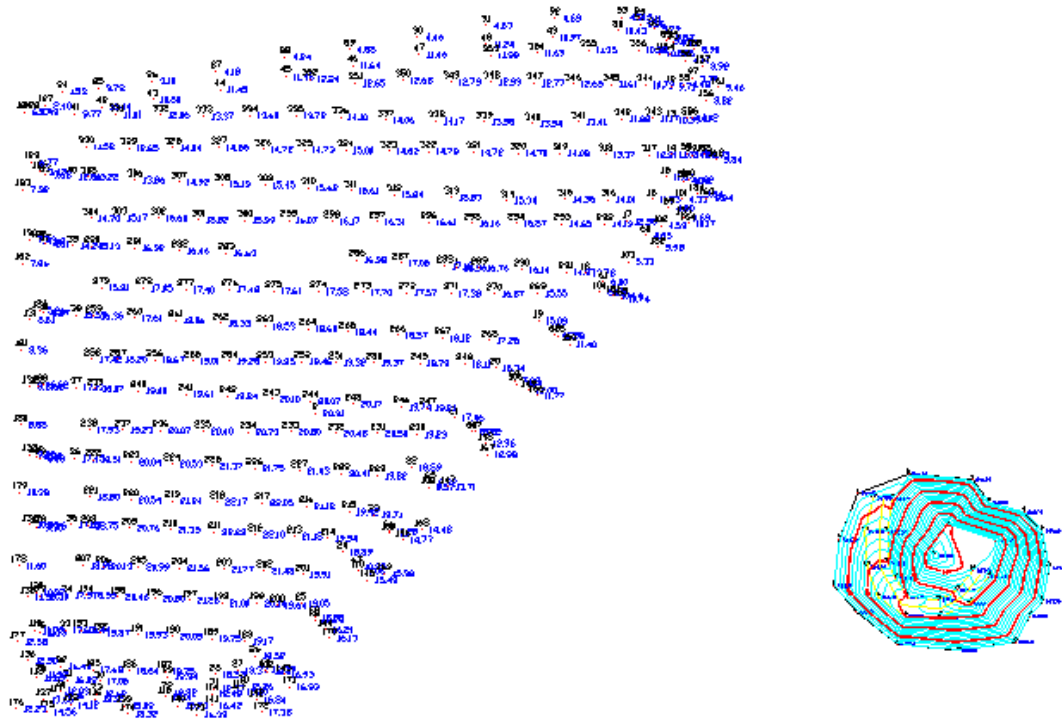
1. import a *LandXML* comprised of digital data content
2. draw a *centerline file* for spatial reference
3. draw a *polyline file* for additional spatial reference
4. offset the centerline "alignment" to establish a Back of Curb (BOC)
5. create stake-out points along the BOC but at existing ground elevations
6. create a cutsheet to compare the points against a design surface model

Let's get started.

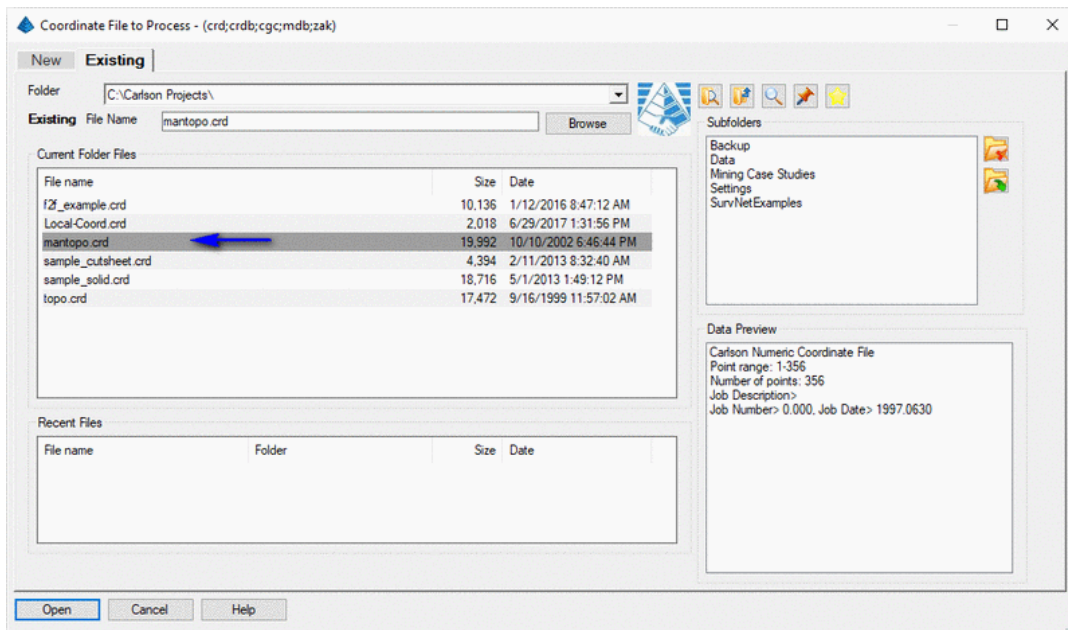
1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **Mantopo.dwg** file. Use the File – **Save As** to save a copy named **Mantopo-C.dwg**. Completing this tutorial will alter the drawing file and by renaming the file from the start, you'll keep the original file intact (allowing you to run through the tutorial a second time if desired). This is also a good practice to keep when working on drawings from 3rd parties.

2. Activate the Survey menu via Settings – Carlson Menus – **Survey Menu**. Your drawing may (should) resemble that shown below:

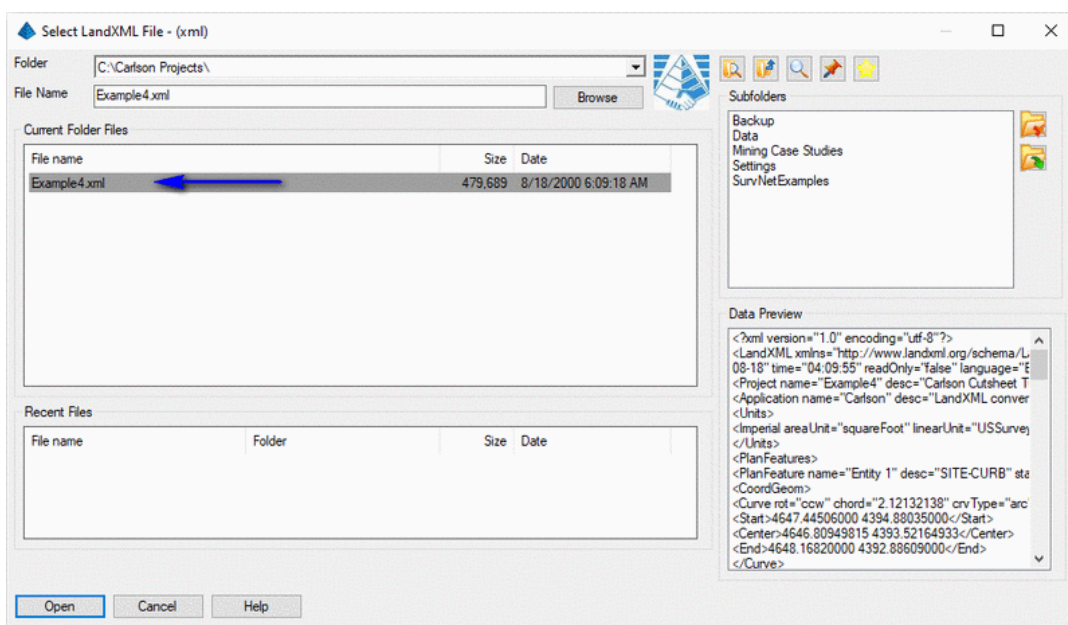


Issue the Points – Set Coordinate File command to display the following dialog box:

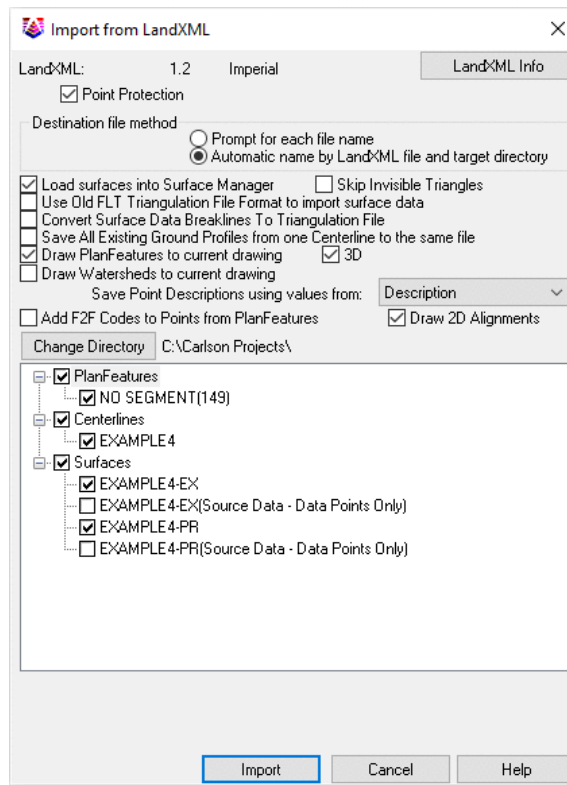


Select the file as shown above and click **Open** when ready.

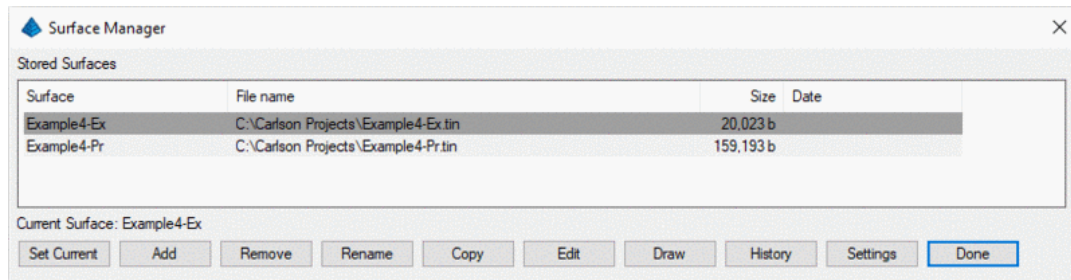
- For the content to follow, a LandXML file containing various pieces of design data will be used to complete this tutorial. Issue the File – Import – LandXML File to display the following dialog box:



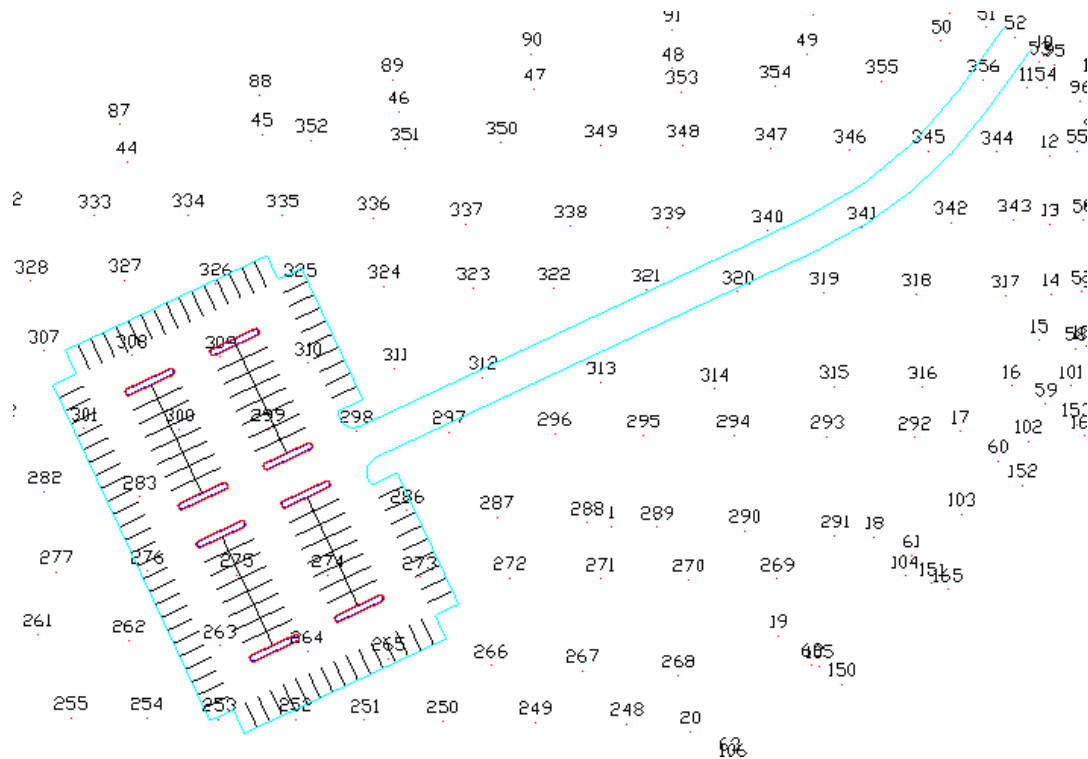
Select the file as shown above and click **Open** when ready. A secondary box similar to that shown below will appear:



Set the various options as shown above and click **Import** when ready. A summary of the surface model *TIN* files will appear in the Surface Manager similar to that shown below:

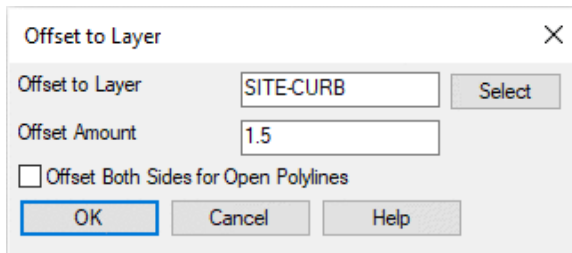


Click the **Done** button when ready. The drawing should resemble that shown below (layer colors and visibility altered for clarity):



The drawing contains a proposed parking lot with striping and curb islands. A closer examination will reveal that the entire length of the roadway and parking lot perimeter has been defined as a *centerline* (aka alignment, **Existing4.cl**) file as created by the Polyline to Centerline File command (e.g. double-clicking the perimeter should display the Centerline Editor command).

- Our next task will be to offset this alignment by 1.5' to simulate *back-of-curb* geometry. Issue the Edit – Offset – Offset to Layer command to display the dialog box below:

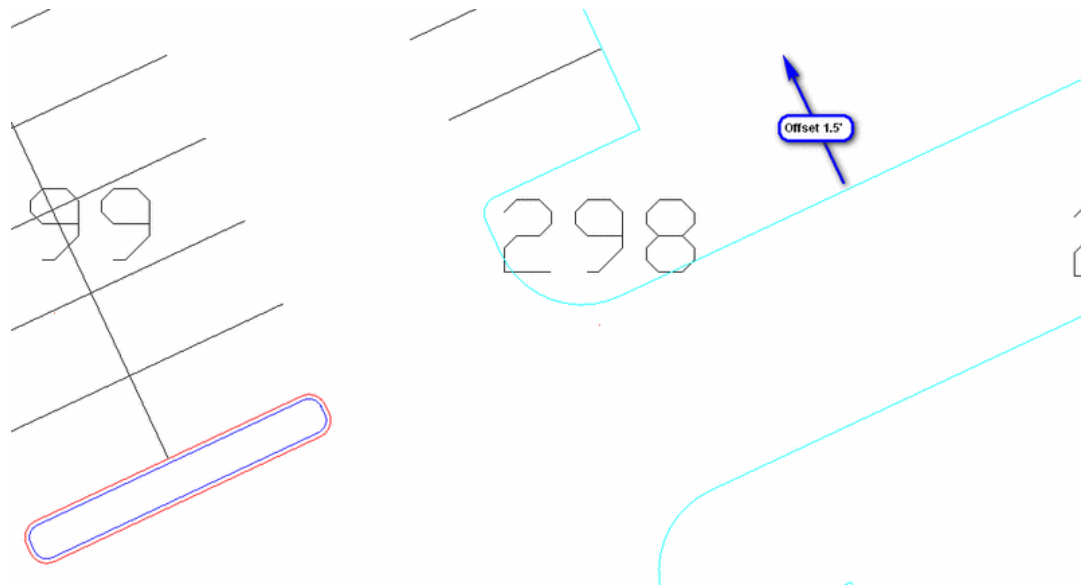


Set the values as shown above and click **OK** when ready. When prompted:

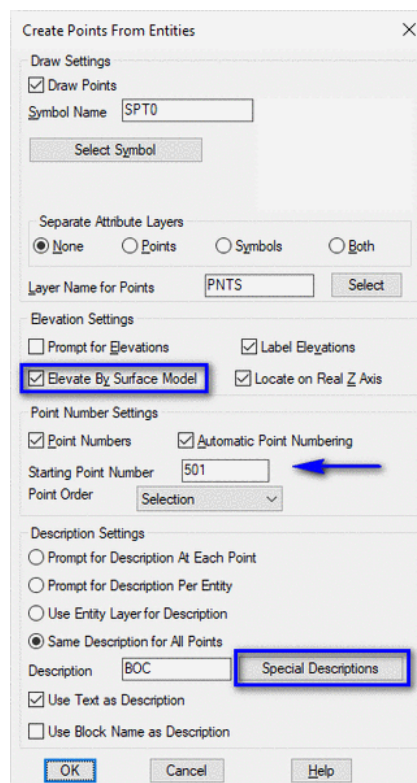
Select a polyline to offset: pick the perimeter polyline as shown below

Select side to offset: pick to the outside as shown below

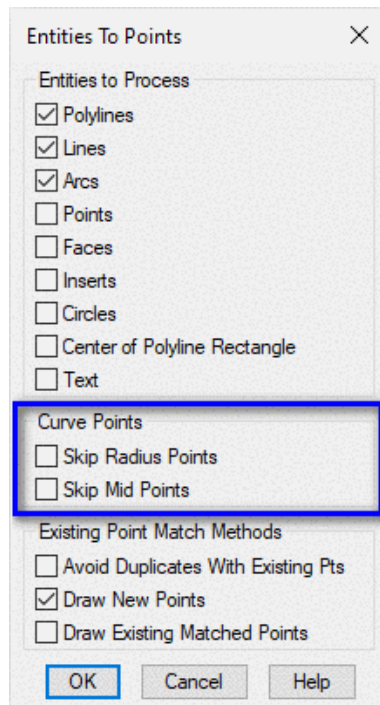
Select a polyline to offset (Enter to end): press Enter



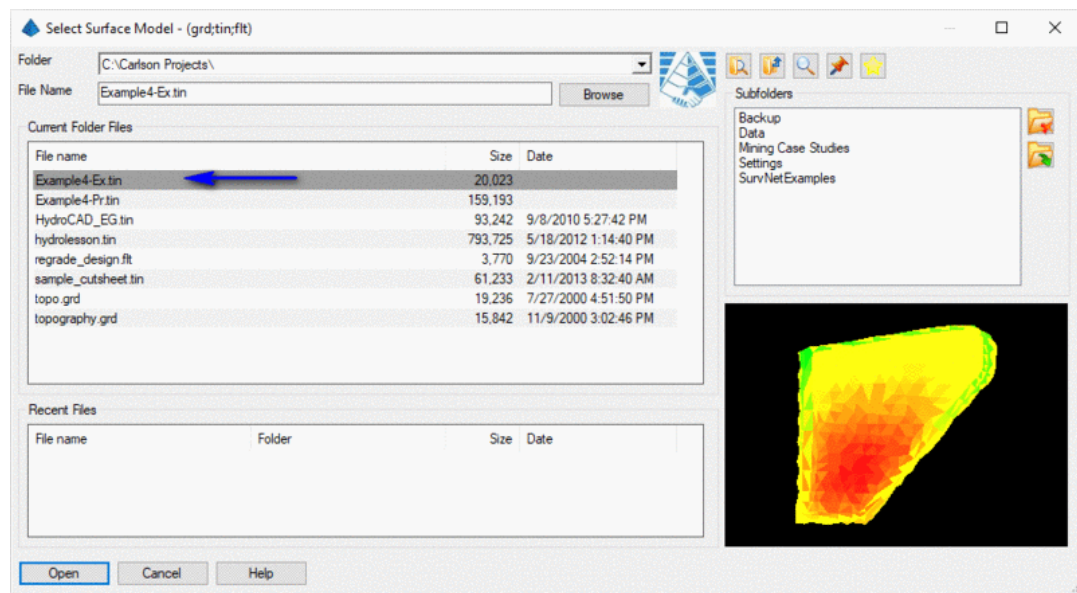
- The next thing we'd like to do is locate stake-out points along the *Back-of-Curb* at important inflection points. Issue the COGO – Create Points from Entities command to display the dialog box below:



Set the values as shown above (making particular note of the highlighted items above) and click **OK** when ready. A secondary dialog box as shown below appears:



Set the values as shown above (making particular note of the highlighted items above) and click **OK** when ready. A dialog box prompting for the surface model use for the point elevation appears (similar to that shown below):



Select the surface model as indicated above and click **Open** when ready. When prompted:

Select arcs, circles, faces, points, text, lines and polylines.

[FILter]/<Select entities>: pick the **back-of-curb** polyline created in the previous step

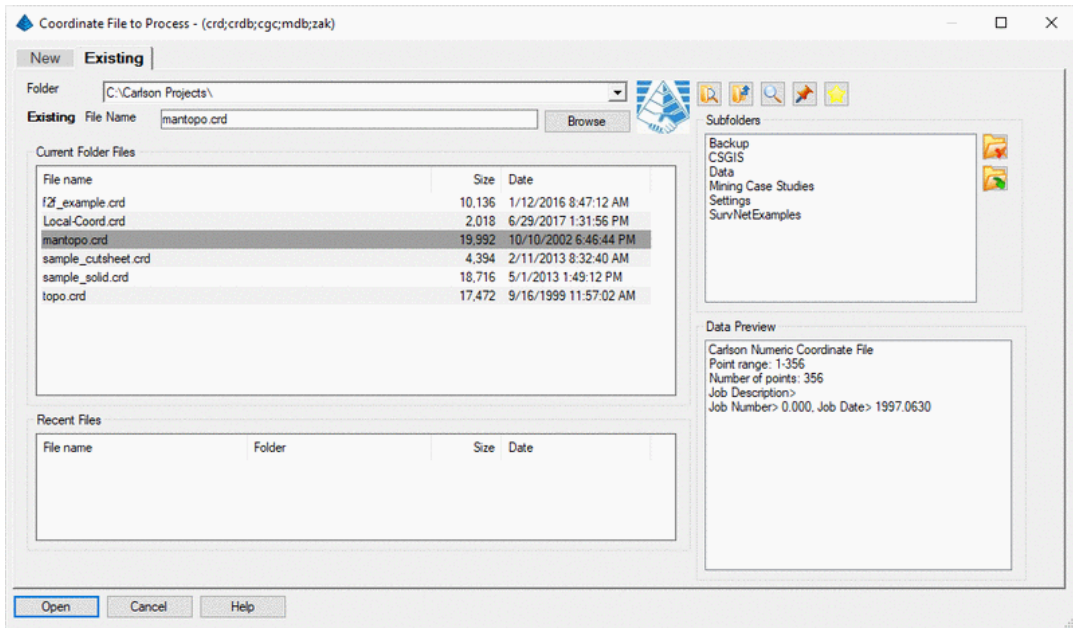
[FILter]/<Select entities>: press Enter

A collection of points is inserted into the drawing and at an elevation governed by the previously cited surface model.

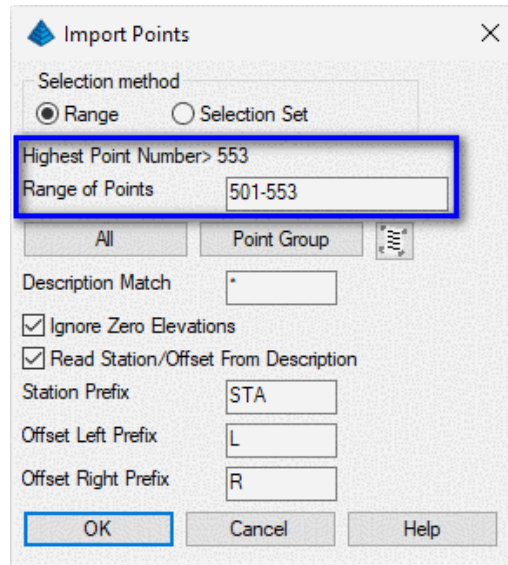
6. Issue the Survey – Cut Sheet command to display the dialog below:



- Click the Import – **Coordinate File** as illustrated above and when prompted for a coordinate file, select the file shown below and click **Open** when ready:



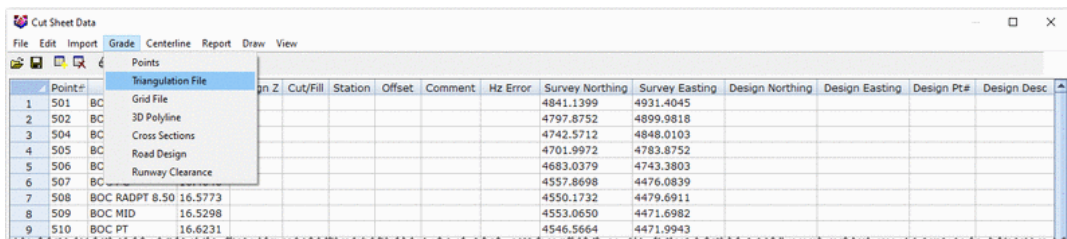
- A secondary dialog box appears similar to that shown below:



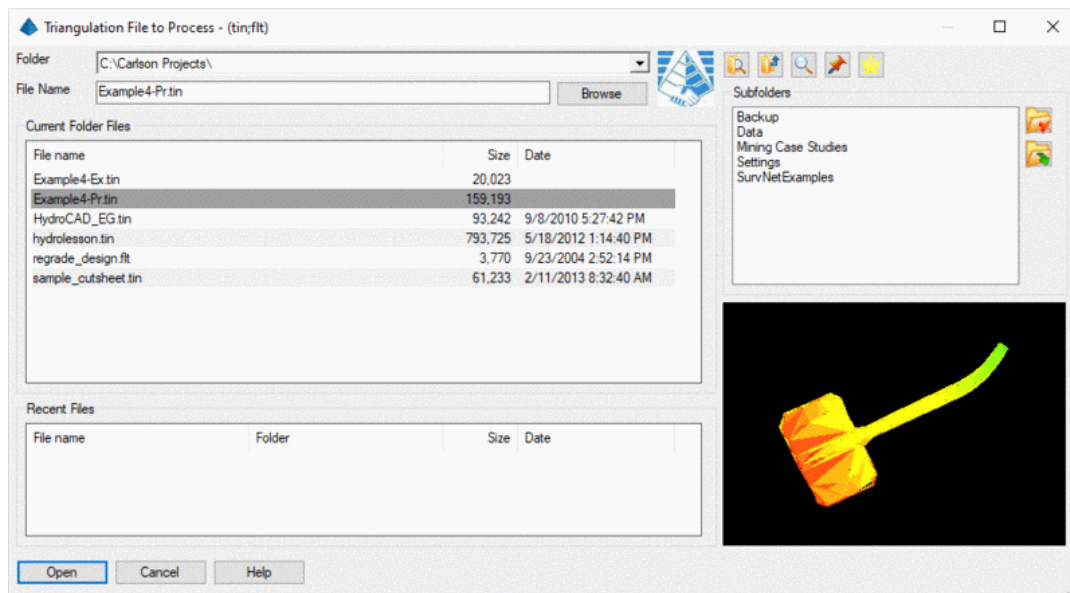
The coordinate file may have more data than we need (*i.e.* original ground points from an earlier lesson). Set the values as shown above, paying particular attention to the **Range** of points in the range of **501-553**. Click **OK** when ready. The **Cut Sheet Data** dialog box is populated with data similar to that shown below:

Point#	Description	Survey Z	Design Z	Cut/Fill	Station	Offset	Comment	Hz Error	Survey Northing	Survey Easting	Design Northing	Design Easting	Design Pt#	Design Desc
1	501 BOC	7.6089							4841.1399	4931.4045				
2	502 BOC PC	10.5818							4797.8752	4899.9818				
3	504 BOC MID	12.2363							4742.5712	4848.0103				
4	505 BOC PT	13.8053							4701.9972	4783.8752				
5	506 BOC	14.4081							4683.0379	4743.3803				
6	507 BOC PC	16.4646							4557.8698	4476.0839				
7	508 BOC RADPT 8.50	16.5773							4550.1732	4479.6911				
8	509 BOC MID	16.5298							4553.0650	4471.6982				
9	510 BOC PT	16.6231							4546.5664	4471.9943				

9. The next thing we'd like to do is bring in the target elevations for each of these points. Issue the Grade – **Triangulation File** command (but note the additional methods of determining the proposed grades) as illustrated below:



When prompted, select the triangulation (TIN) file cited below and click **Open** when ready:



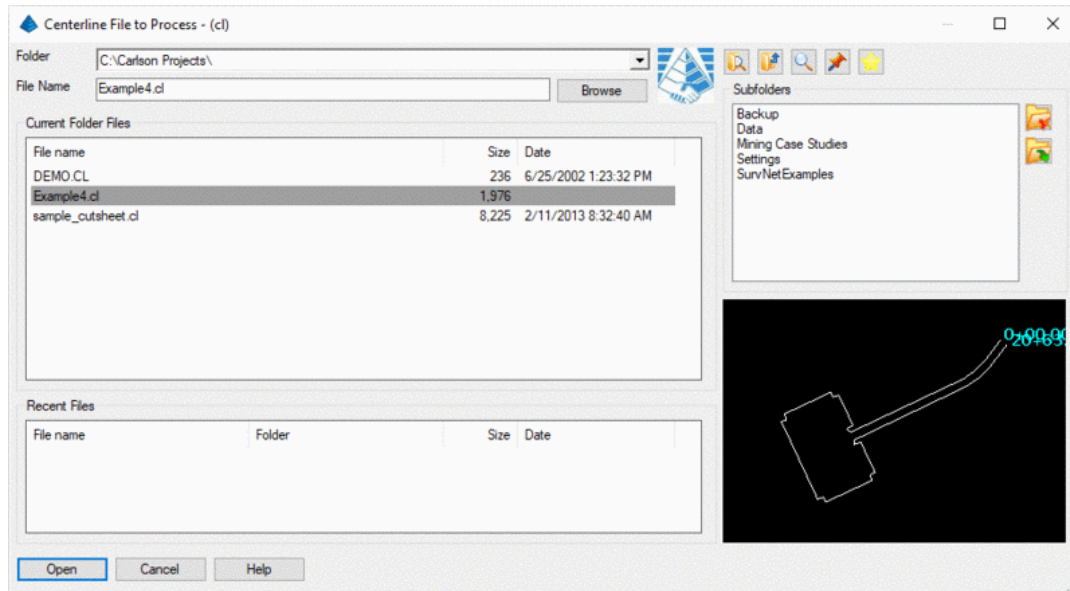
This process will select a design surface model for the proposed grades. The **Design Z** elevation field is populated from the proposed TIN and the **Cut/Fill** amount per point is calculated as illustrated below:

Point#	Description	Survey Z	Design Z	Cut/Fill	Station	Offset	Comment	Hz Error	Survey Northing	Survey Easting	Design Northing	Design Easting	Design Pt#	Design Desc
1	501 BOC	7.6089							4841.1399	4931.4045				
2	502 BOC PC	10.5818	8.9270	C1.655*					4797.8752	4899.9818				
3	504 BOC MID	12.2363	9.7978	C2.438*					4742.5712	4848.0103				
4	505 BOC PT	13.8053	10.6408	C3.165*					4701.9972	4783.8752				
5	506 BOC	14.4081	11.1468	C3.261*					4683.0379	4743.3803				
6	507 BOC PC	16.4646	14.4870	C1.978*					4557.8698	4476.0839				
7	508 BOC RADPT 8.50	16.5773	16.3361	C0.241*					4550.1732	4479.6911				
8	509 BOC MID	16.5298	14.5756	C1.954*					4553.0650	4471.6982				
9	510 BOC PT	16.6231	14.6647	C1.958*					4546.5664	4471.9943				

10. The next piece of information we would like to obtain is an alignment from which Station/Offset information can be derived. Issue the Centerline – **Centerline File** command as illustrated below:

Point#	Descrip	Polyline	Cut/Fill	Station	Offset	Comment	Hx Error	Survey Northing	Survey Easting	Design Northing	Design Easting	Design Pt#	Design Desc
1	501	BOC						4841.1399	4931.4045				
2	502	BOC PC	10.5818 8.9270	C1.655*				4797.8752	4899.9818				
3	504	BOC MID	12.2363 9.7978	C2.438*				4742.5712	4848.0103				
4	505	BOC PT	13.8053 10.6408	C3.165*				4701.9972	4783.8752				
5	506	BOC	14.4081 11.1468	C3.261*				4683.0379	4743.3803				
6	507	BOC PC	16.4646 14.4870	C1.978*				4557.8698	4476.0839				
7	508	BOC RADPT 8.50	16.5773 16.3361	C0.241*				4550.1732	4479.6911				
8	509	BOC MID	16.5298 14.5756	C1.954*				4553.0650	4471.6982				
9	510	BOC PT	16.6231 14.6647	C1.958*				4546.5664	4471.9943				

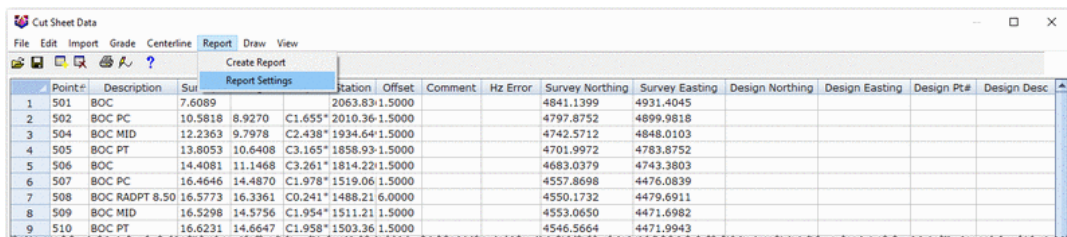
When prompted, select the centerline (*.cl) file shown below and click **Open** when ready:



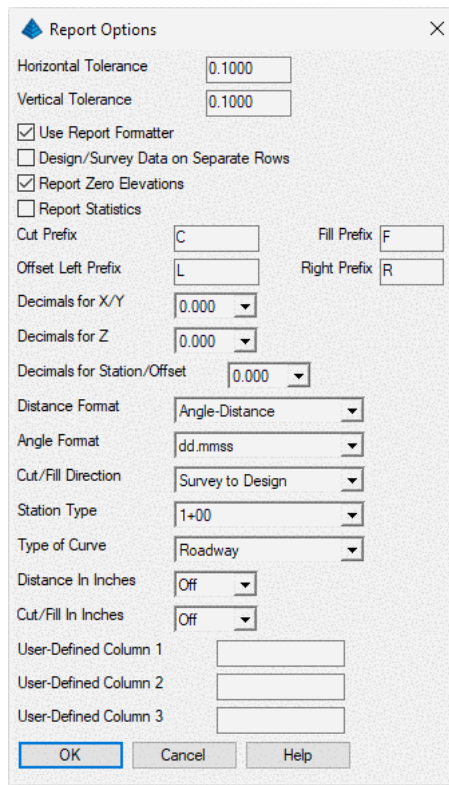
The Cut Sheet now contains the station and offset information for the curb line as illustrated below:

Point#	Description	Survey Z	Design Z	Cut/Fill	Station	Offset	Comment	Hx Error	Survey Northing	Survey Easting	Design Northing	Design Easting	Design Pt#	Design Desc
1	501	BOC	7.6089		2063.831	1.5000			4841.1399	4931.4045				
2	502	BOC PC	10.5818 8.9270	C1.655*	2010.36	1.5000			4797.8752	4899.9818				
3	504	BOC MID	12.2363 9.7978	C2.438*	1934.64	1.5000			4742.5712	4848.0103				
4	505	BOC PT	13.8053 10.6408	C3.165*	1858.93	1.5000			4701.9972	4783.8752				
5	506	BOC	14.4081 11.1468	C3.261*	1814.22	1.5000			4683.0379	4743.3803				
6	507	BOC PC	16.4646 14.4870	C1.978*	1519.06	1.5000			4557.8698	4476.0839				
7	508	BOC RADPT 8.50	16.5773 16.3361	C0.241*	1488.21	6.0000			4550.1732	4479.6911				
8	509	BOC MID	16.5298 14.5756	C1.954*	1511.21	1.5000			4553.0650	4471.6982				
9	510	BOC PT	16.6231 14.6647	C1.958*	1503.36	1.5000			4546.5664	4471.9943				

- Our next task will be to configure the content of the *Cut Sheet Report* we'd like to generate. Issue the Report – **Report Settings** command as illustrated below:

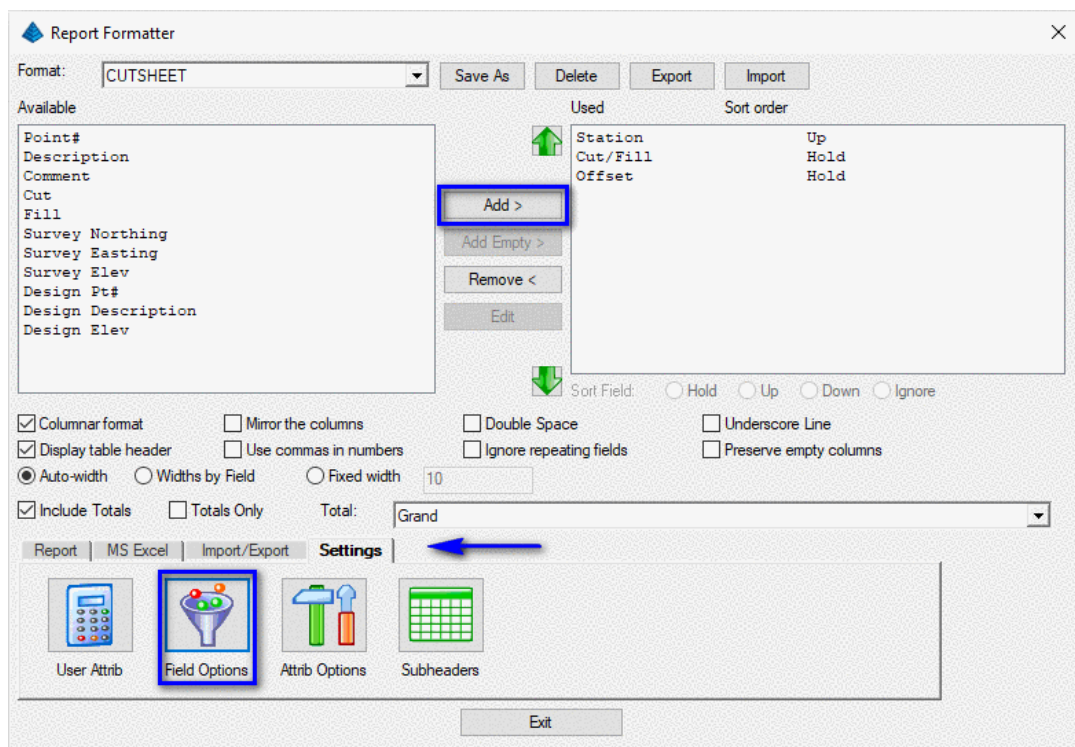


The **Report Options** dialog box appears permitting the eventual report to be customized:



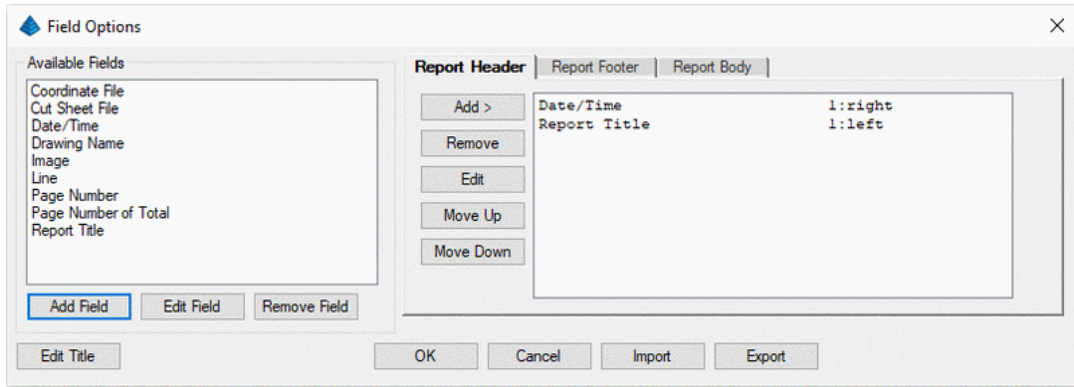
Ensure the **Report Options** are set as shown above and click **OK** when ready.

- Let's create the report. Issue the Report – **Create Report** command to display the Report Formatter dialog box similar to that shown below:

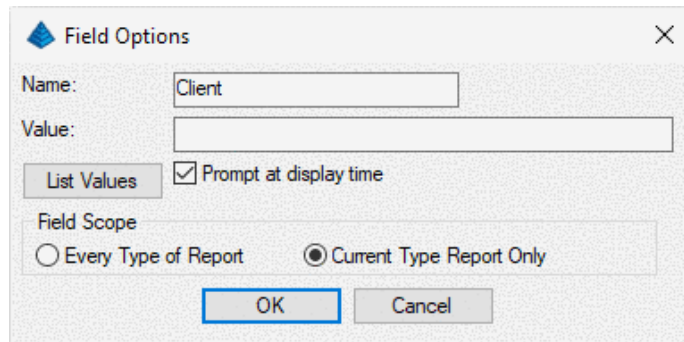


Select the desired **Available** fields on the left and click **Add >** to move them to the **Used** fields on the right. For this lesson, just select **Station** (sorted *Up*), **Offset** and **Cut/Fill** to create a simple cut/fill report.

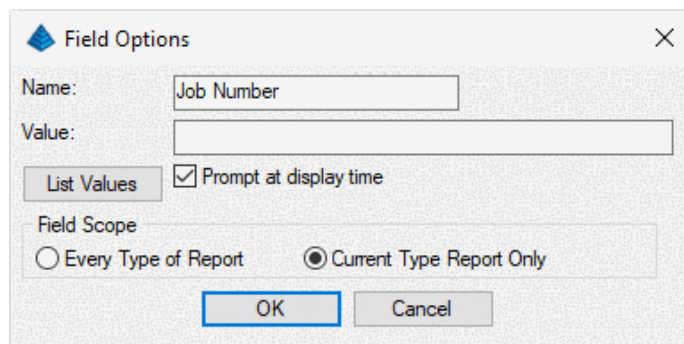
- Click the **Settings** tab as seen above and click **Field Options** button to display the **Field Options** dialog box as shown below:



14. To create *user-defined* additional fields to include in the report, click the **Add Field** button and set the values as shown below. Click **OK** when ready:

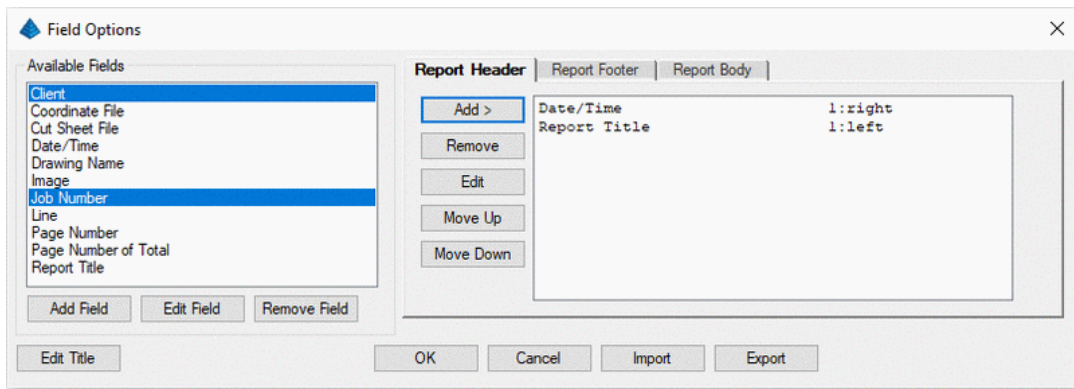


This new *user-defined* field is added to the list of **Available Fields**. Click the **Add Field** button again to set the values as shown below and click **OK** when ready:

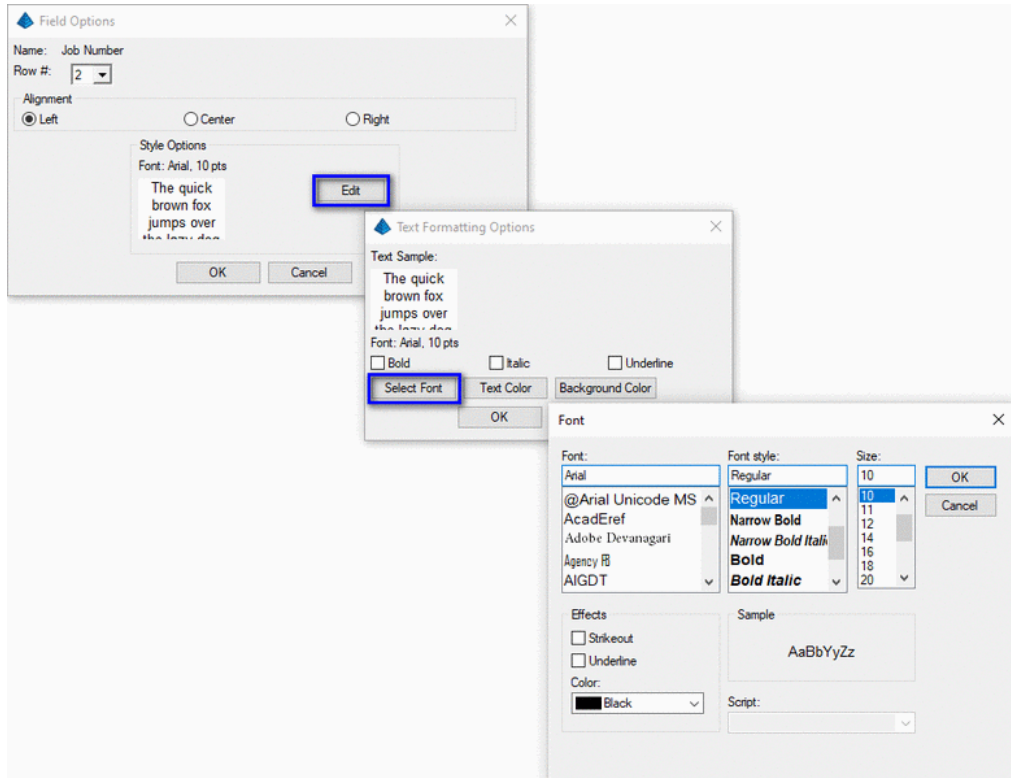


NOTE: Make sure to enable the **Prompt at display time** toggle. This will cause the program to pause for user input at the time of creating the report.

15. Select the two new fields and click the **Add >** button to add them to the **Report Header** tab as illustrated below:



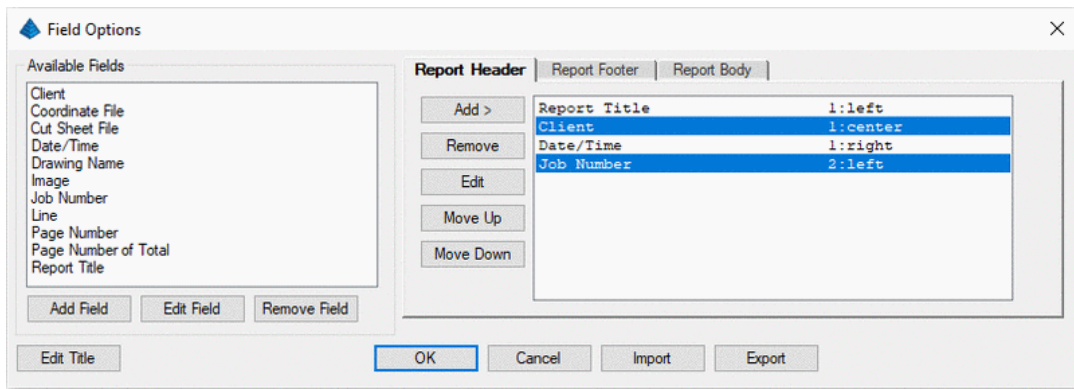
When the fields are moved, you will be prompted to select various format options as illustrated below:



16. Set the following fields:

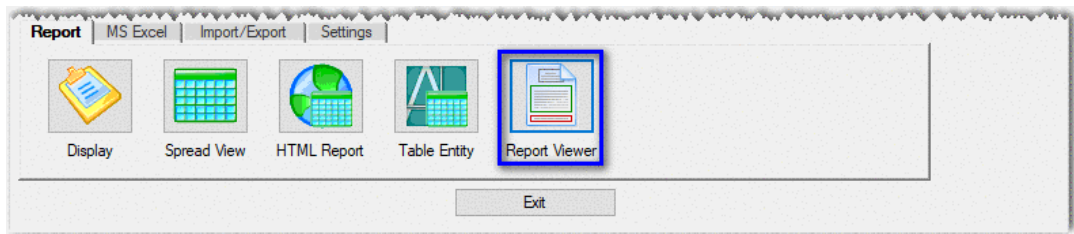
- **Client** to:
 - Row #1
 - Center justified
 - Bold Font
 - 14 pt text size
- **Job Number** to:
 - Row #2
 - Left justified
 - Regular Font
 - 10 pt text size

The result should look similar to that shown below:

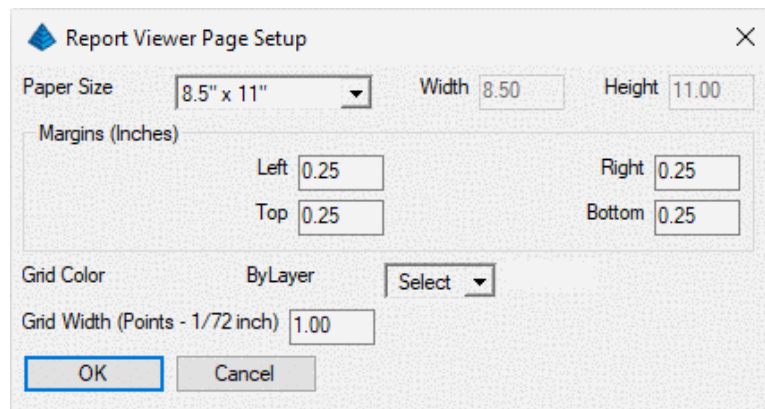


Click **OK** when ready.

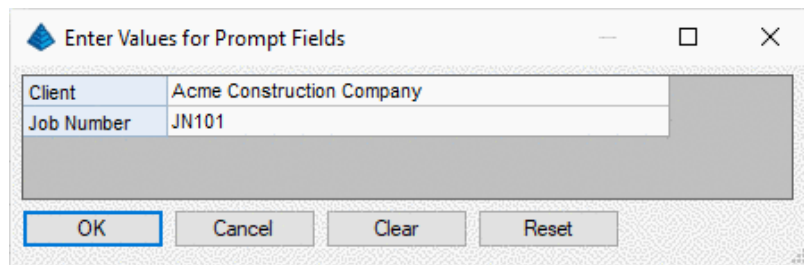
17. Select the **Report** tab and click on the **Report Viewer** button as shown below:



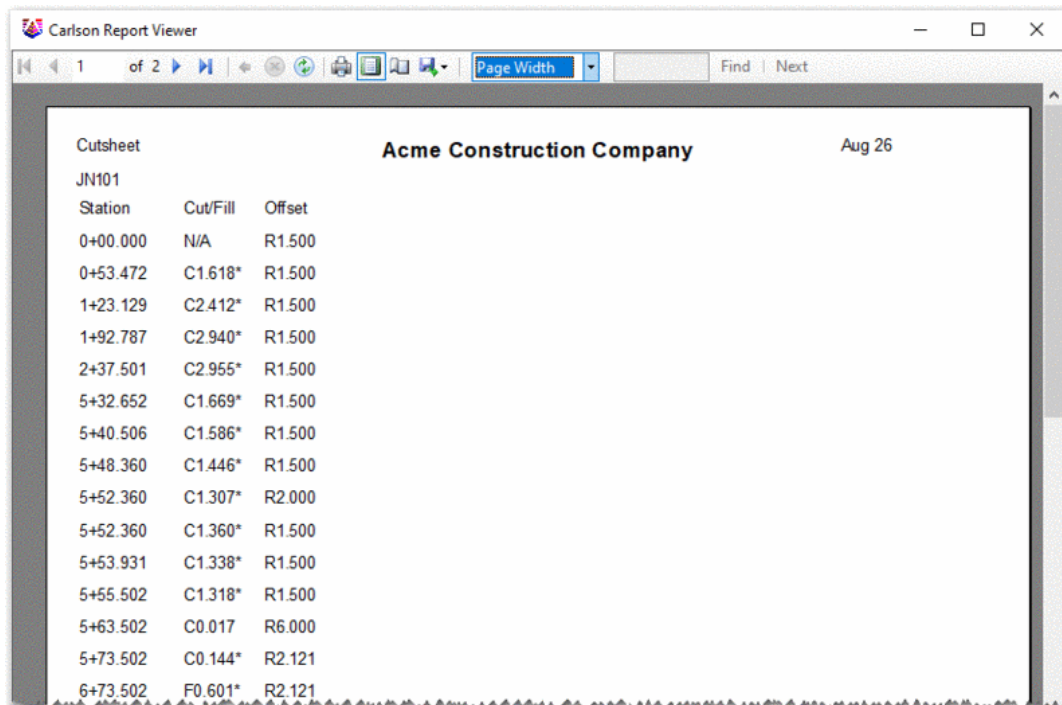
Select an **8.5" x 11"** paper size with **0.25"** margins all around as shown below and click **OK** when ready:



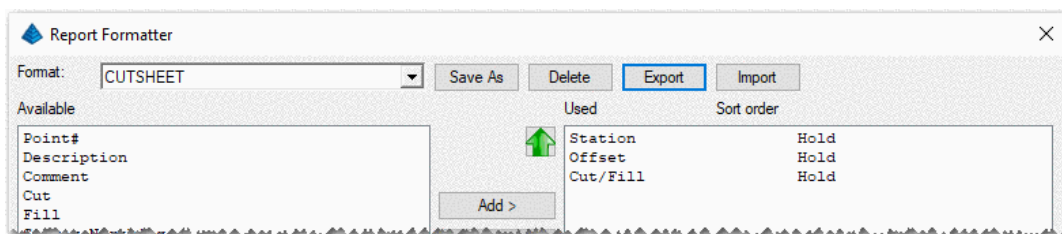
18. When prompted for the values of the *user-defined* fields, set the values as shown below and click **OK** when ready:



The resulting report is displayed similar to that shown below which can be printed or exported to various formats:

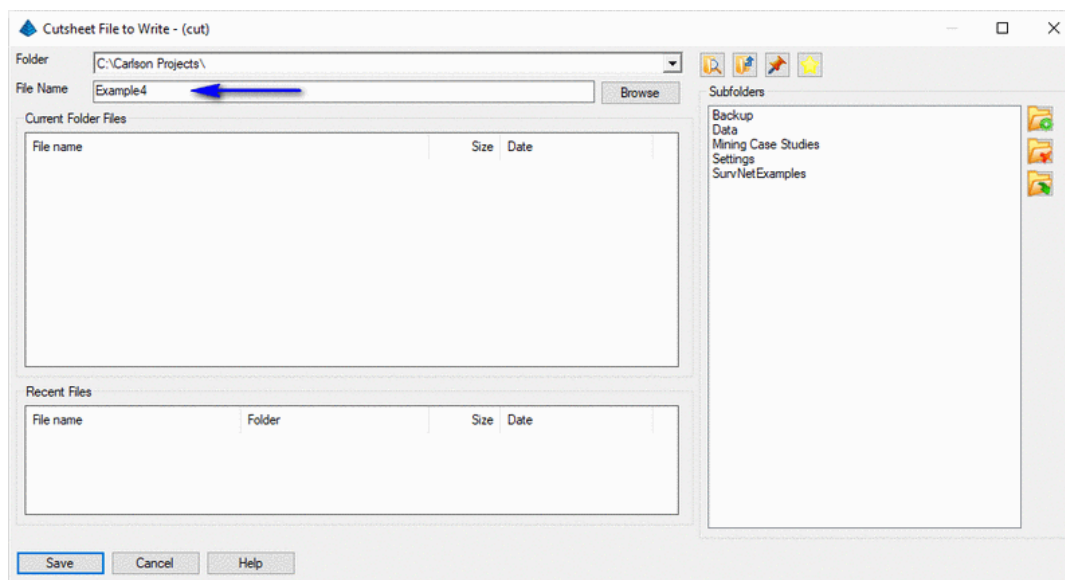


19. Close the report to return to the **Report Formatter** dialog box as shown below:



NOTE: The above process can be repeated for any number of fields and exported as a *.FMS file for future use or to develop company or client report standards. Click the **Exit** button to dismiss the *Report Formatter* dialog box.

20. Back in the main *Cut Sheet Data* spreadsheet editor, click File – **Save** and save the file as shown below and click the **Save** button when ready:



Click File – **Exit** to close the command.

This completes the tutorial: Cut Sheet.

Geodetic Reports

In this lesson you will create a geodetic report containing both local and Grid coordinates from a survey based on an assumed local datum. The plan geometry will be annotated with both ground and grid distances. This is useful when a survey has been performed with assumed coordinates and elevations but must be reported in Grid coordinates or in instances where both ground and grid coordinates need to be reported.

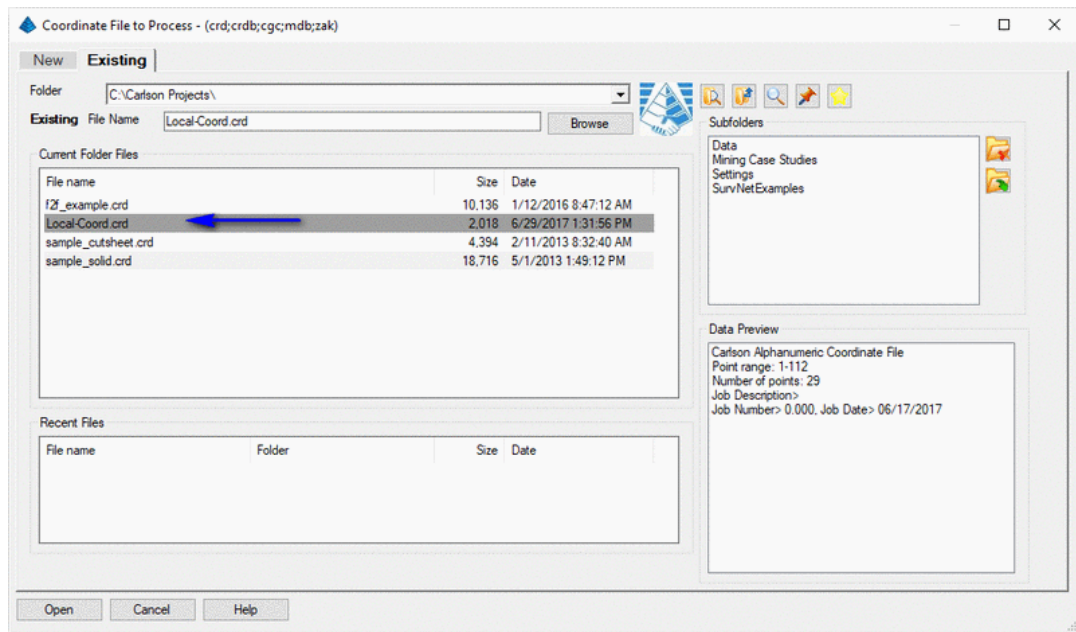
1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **Local-Plat.DWG** file.

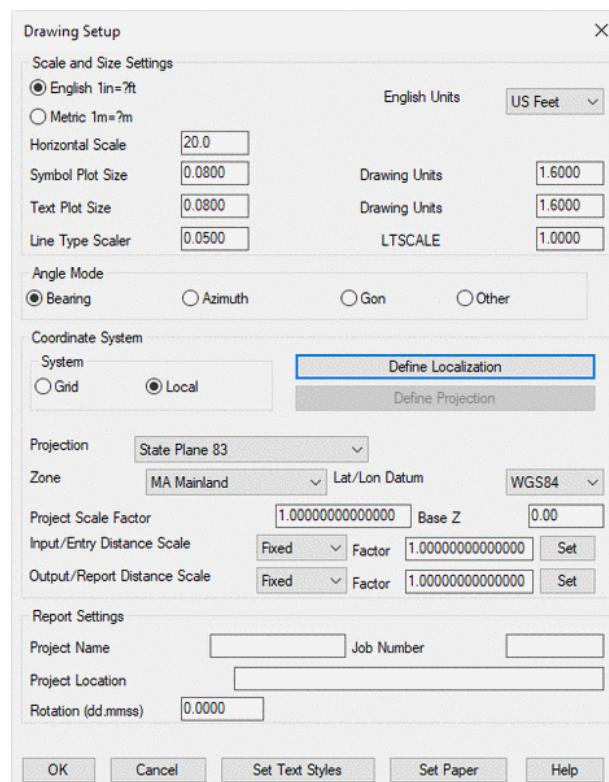
2. Activate the Survey menu via Settings – Carlson Menus – **Survey Menu**. Your drawing should resemble that shown below:



3. Issue the Points – Set Coordinate File command and select the supplied file **Local-Coord.CRD** as shown below:



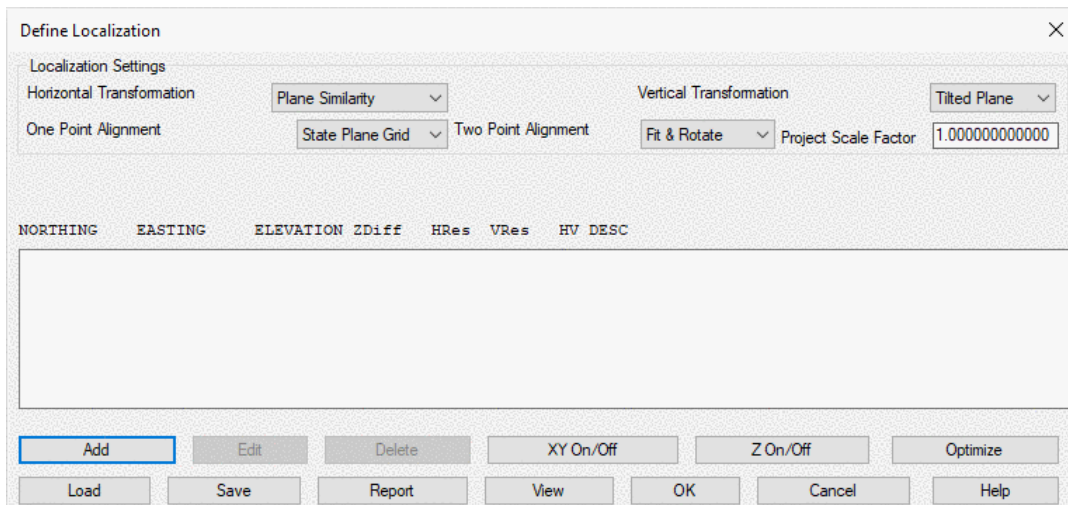
4. Issue the Settings – Drawing Setup command to display a dialog box similar to that shown below:



Set the:

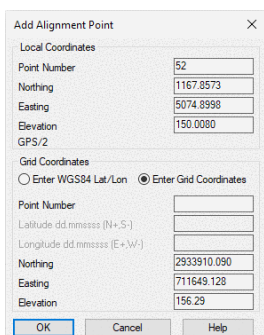
- (a) **Projection** to **State Plane 83**, and
- (b) **Zone** to **MA Mainland**.

5. Click the **Define Localization** button to display a dialog box similar to that shown below:

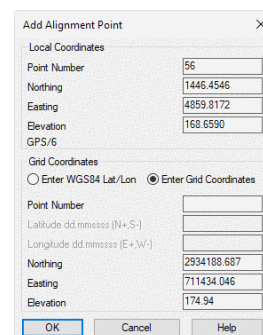


6. We'd like to localize the drawing based on two RTK GPS points that were located to establish grid coordinates. We'll be using points **52** and **56**. Click the **Add** button to display a dialog box similar to those shown below:

Point 52



Point 56



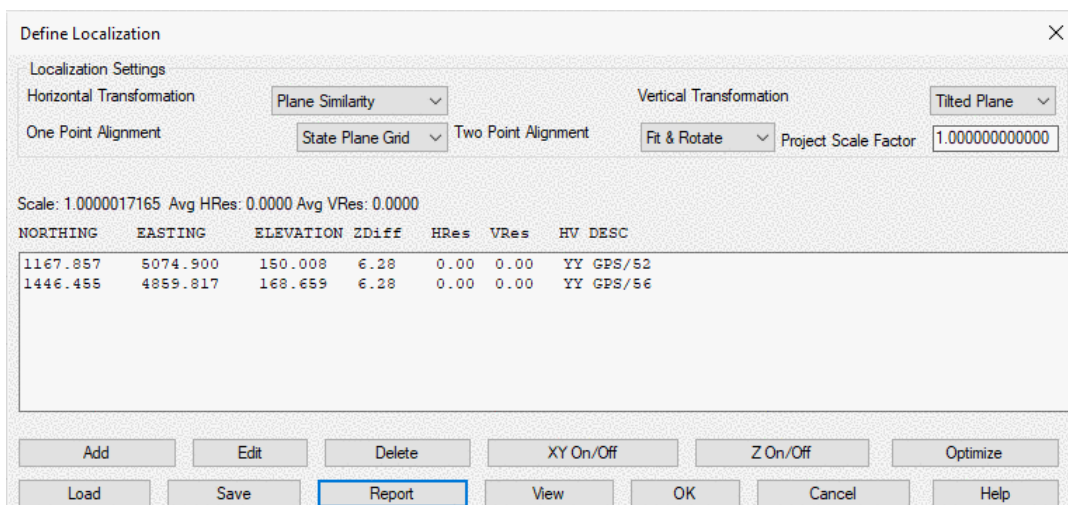
Localized Points

Add the point and coordinate information for each point by entering the **Local Point Number** (press the *Tab* key to complete the Local data) and their respective **Grid Coordinates** as outlined in the table below (click **OK** when ready and subsequently click **OK** to accept the default point descriptions):

Point Number	Local Northing	Local Easting	Local Elevation	Grid Northing	Grid Easting	Grid Elevation
52	1167.8573	5074.8998	150.0080	2933910.090	711649.128	156.29
56	1446.4546	4859.8172	168.6590	2934188.687	711434.046	174.94

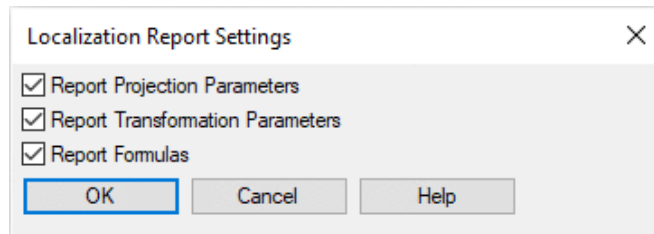
Localized Point Data

7. The results should resemble that shown below:

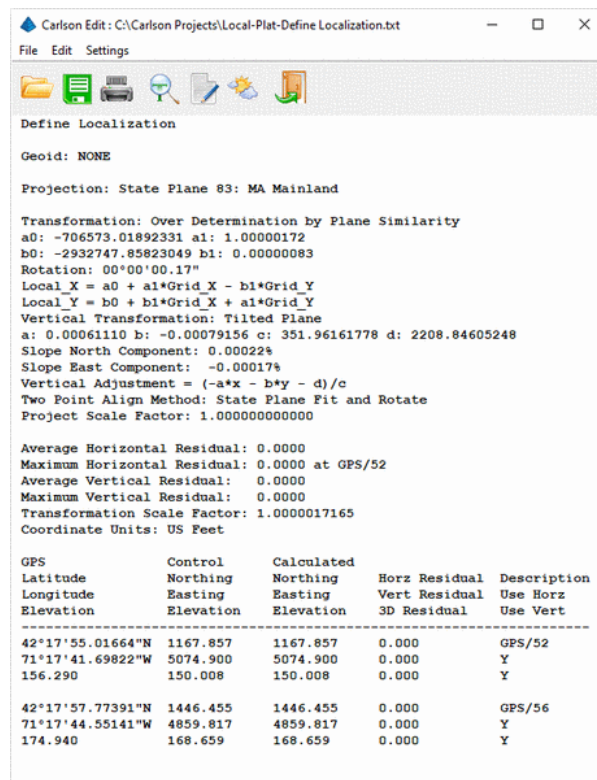


NOTE: For each point, make sure the **HV** (Horizontal/Vertical) fields are set to **YY** (indicating *Yes*, both the Horizontal and Vertical components should be used in the solution). Use the **XY On/Off** button and/or the **Z On/Off** button as necessary.

- Click the **Report** button to display the dialog box below:

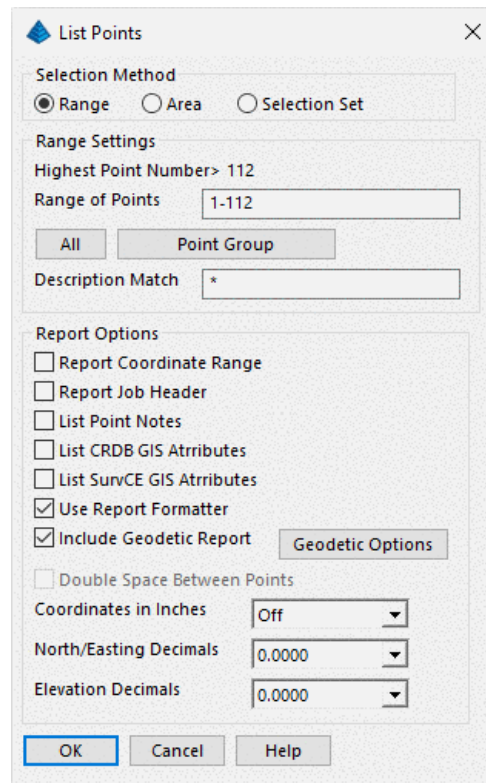


Ensure all of the report options are enabled and click **OK**. The localization report similar to that shown below is displayed in the Standard Report Viewer:

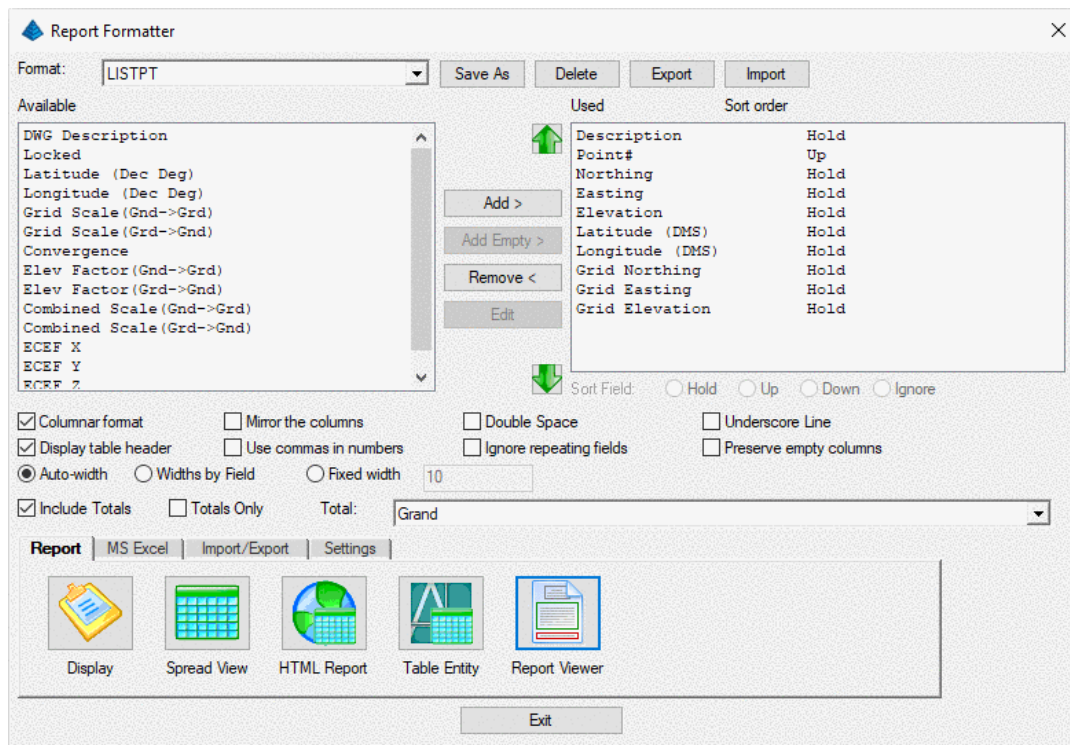


Click the **Exit** (Doorway) button to dismiss the report. Further, click the **OK** button on the *Define Localization* and *Drawing Setup* dialog boxes, respectively.

- Issue the Points – List Points command to display the dialog box similar to that shown below:

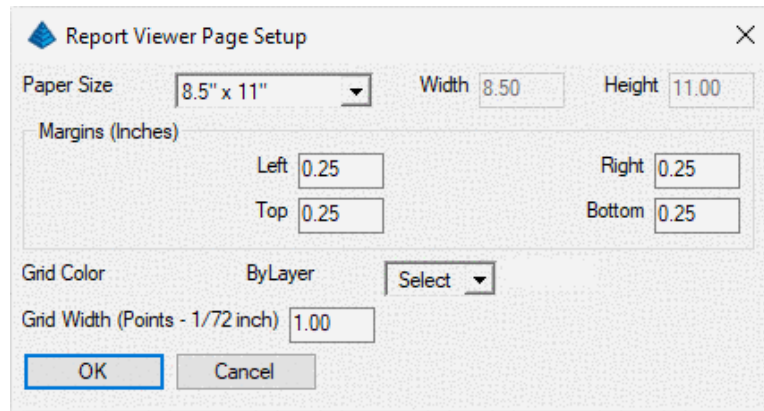


Enable the **Use Report Formatter** and **Include Geodetic Report** options and click **OK** when ready.
 10. The Report Formatter dialog box appears similar to that shown below:



Select the desired **Available** fields on the left and click **Add >** to move them to the **Used** fields on the right. Set the fields to that shown above.

11. From the **Report** tab, click on the **Report Viewer** button. Select an **8.5" x 11"** paper size with **0.25"** margins all around as shown below and click **OK** when ready:



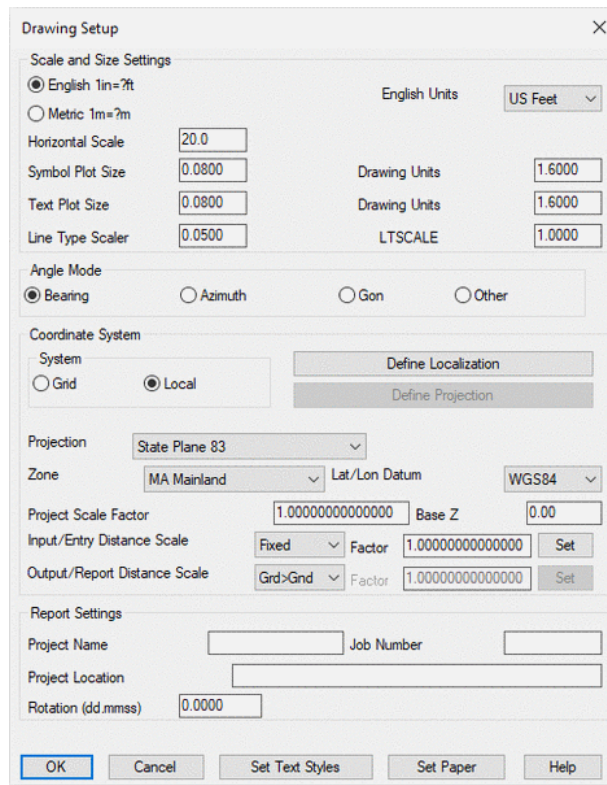
The resulting report is displayed similar to that shown below which can be printed or exported to various formats:

Description	Point#	Northing	Easting	Elevation	Latitude (DMS)	Longitude (DMS)	Grid Northing	Grid Easting	Grid Elevation
MNS	1	1000.0000	5000.0000	154.2940	N42°17'53.36"	W71°17'42.70"	2933742.2331	711574.2282	160.5762
MNS	2	1167.8269	5074.8863	150.0289	N42°17'55.02"	W71°17'41.70"	2933910.0596	711649.1145	156.3109
MNS	3	1333.8035	4915.3038	164.0529	N42°17'55.66"	W71°17'43.82"	2934076.0360	711489.5324	170.3343
SPKS	4	1207.9257	4765.4906	183.2189	N42°17'55.42"	W71°17'45.81"	2933950.1586	711339.7194	189.5003
MNS	5	1161.3941	4926.7535	162.2223	N42°17'54.96"	W71°17'43.67"	2933903.6269	711500.9820	168.5041
MNS	6	1446.5046	4859.8772	168.5987	N42°17'57.77"	W71°17'44.55"	2934188.7370	711434.1060	174.8797
CHK1	11	1000.0037	4999.9885	154.2903	N42°17'53.36"	W71°17'42.70"	2933742.2367	711574.2167	160.5725
CHK5	15	1161.3935	4926.7549	162.2113	N42°17'54.96"	W71°17'43.67"	2933903.6263	711500.9833	168.4931

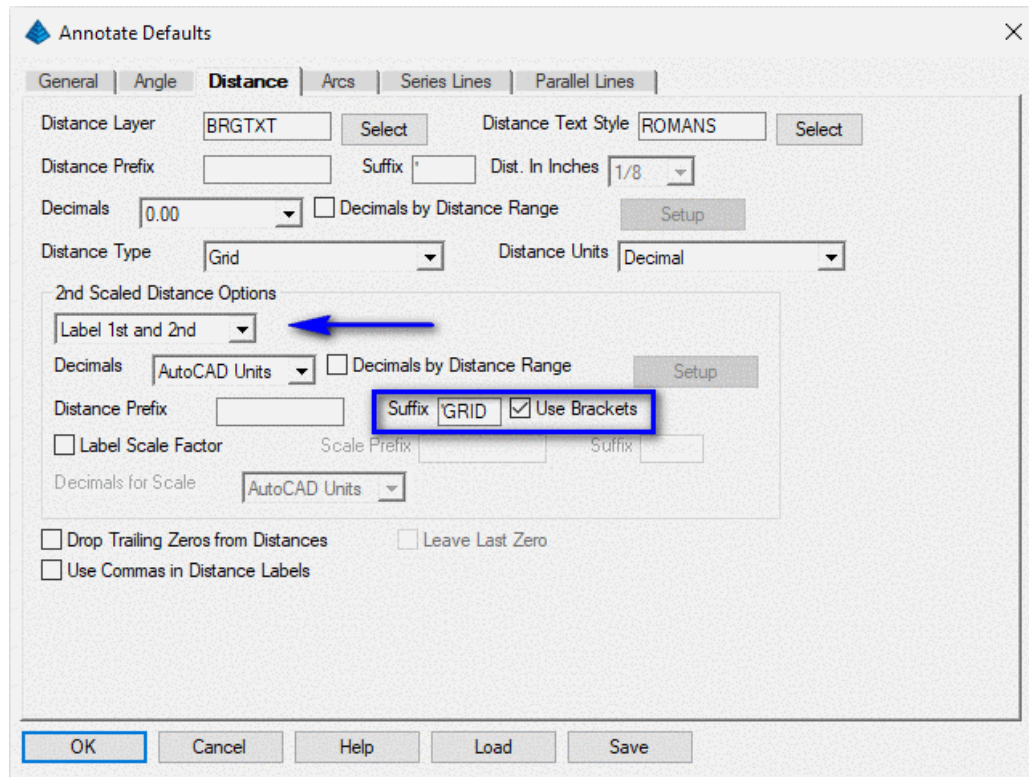
Both local and grid coordinates are displayed.

NOTE: To further control the report by adding user-defined fields, customized Headers and Subheaders, Fonts *etc.*, refer to the Cut Sheet tutorial.

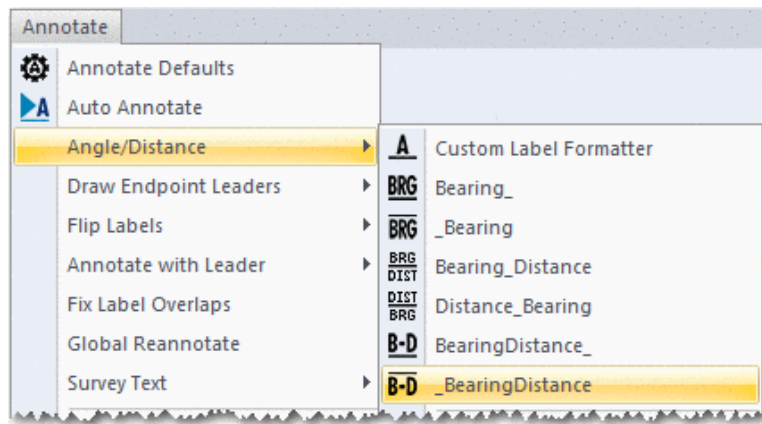
12. Close the report to return to the **Report Formatter** dialog box. Click the **Exit** button to dismiss the *Report Formatter* dialog box.
13. Re-run the Settings – Drawing Setup as discussed earlier:



- Set the **Report Distance Scale** option to **Gnd>Grd** for ground to grid distances and click **OK** when ready.
- Issue the Annotate – Annotate Defaults command to display the dialog box similar to that shown below:



- In the *Distance* tab, set the **2nd Scaled Distance Options** to **Label 1st and 2nd**. Also, set the **Suffix** to **'GRID** and enable the **Use Brackets** option. Click **OK** when ready.
- Issue the Annotate – Annotate Defaults – **_BearingDistance** command:

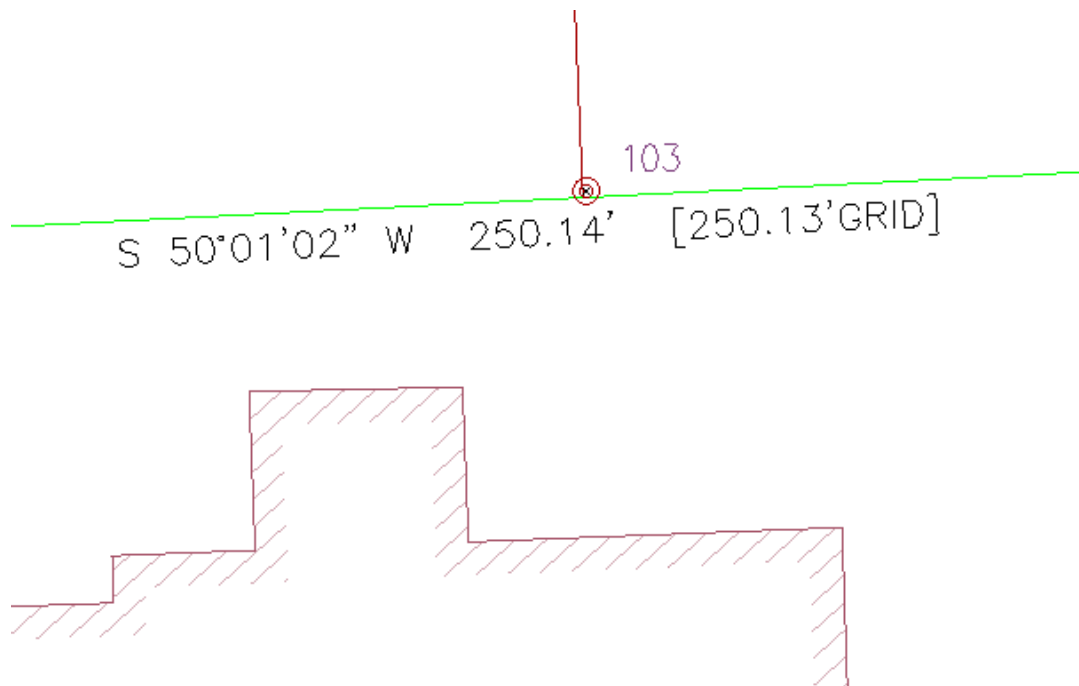


When prompted:

Select line or polyline [Size/Points]: select the most northerly property line of the parcel

Select line or polyline [Size/Points] (Enter to end): press Enter

Both the ground distance and the projected scaled distance are labeled as shown below (layer visibility altered for clarity):



This completes the tutorial: Geodetic Reports.

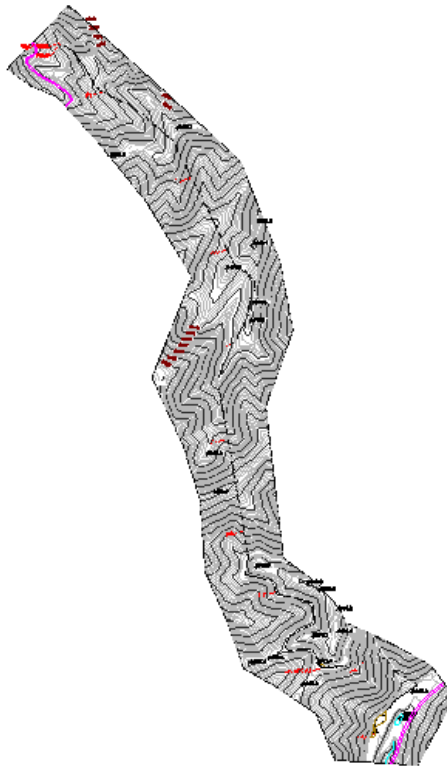
Pressure Pipe Network Design

In this tutorial, we'll design the features and pipes for a pressure pipe network and analyze the flow for the site and ultimately prepare an input file for EPANet (Application for Modeling Drinking Water Distribution Systems).

1. Click the Windows desktop icon for Carlson to start the program.
 - If you get the Start Page, pick **Open Files**.
 - If you get the Startup Wizard dialog box, click the **Browse** button.
 - If you are taken directly into CAD, click File – **Open**.

Browse/navigate to the default folder location of **C:\Carlson Projects** and open the **PressurePipeExample.dwg** file.

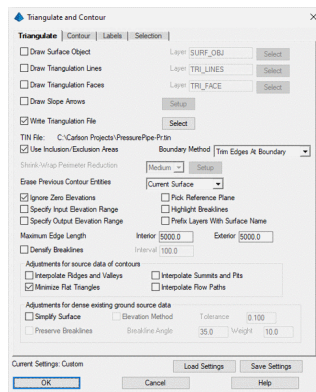
2. Activate the Hydrology menu via Settings – Carlson Menus – **Hydrology Menu**. Your drawing should resemble that shown below:



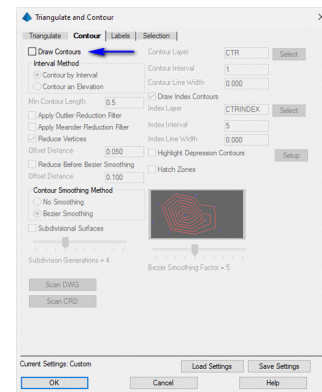
This project contains a series of polylines that represents the pressure pipe network we are designing.

3. The drawing entities for the design surface that we will use to model drainage have already been prepared. These entities consist of design contours. Issue the Surface – Triangulate & Contour command to display a dialog box similar to that shown below:

Triangulate Tab



Contour Tab



Triangulate & Contour

NOTE: Use the **Select** button on the *Triangulate* to create an output TIN file as illustrated above. Once set, click the **OK** button and when prompted:

Select Inclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: pick the boundary polyline

[FILter]/<Select entities>: press Enter

Select Exclusion perimeter polylines or ENTER for none.

[FILter]/<Select entities>: press Enter

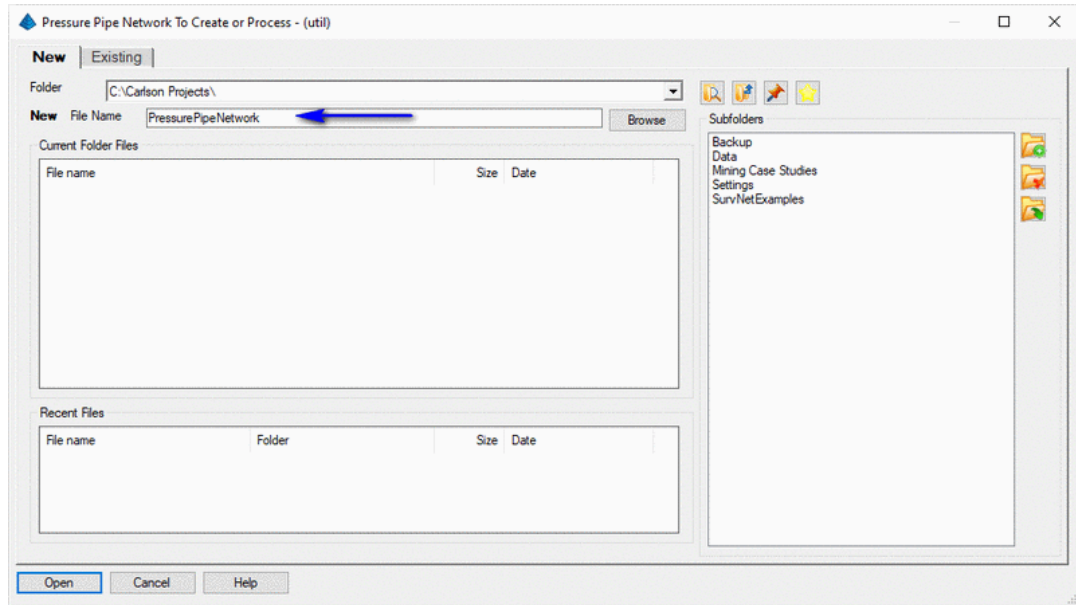
Select points and breaklines to Triangulate.

[FILter]/<Select entities>: type ALL and press Enter

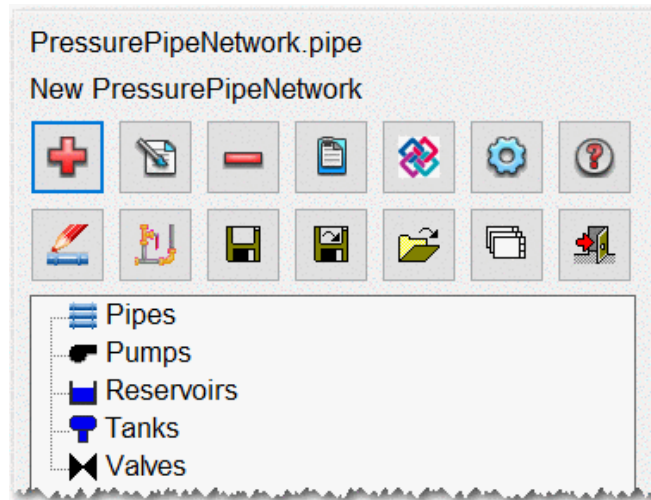
[FILter]/<Select entities>: press Enter

The surface representing proposed conditions is written.

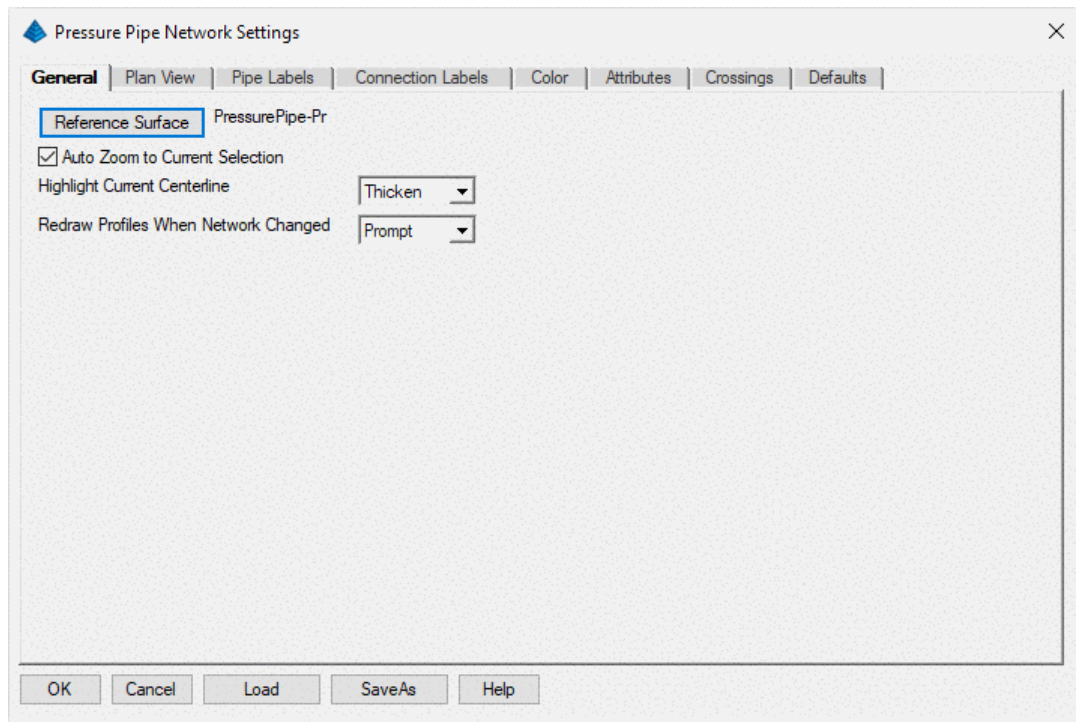
- Let's create a pressure pipe network file from the geometry presented. Issue the Network – Pressure Pipe Network command to display a dialog box similar to that shown below:



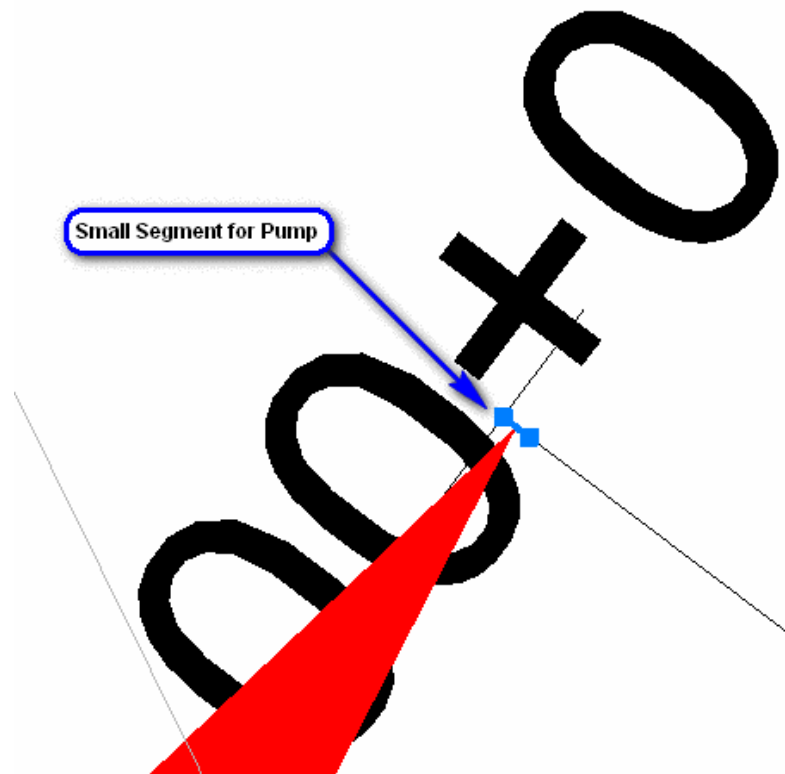
Set the file name as illustrated above and click the **Open** button to display a "docked" dialog box as shown below:



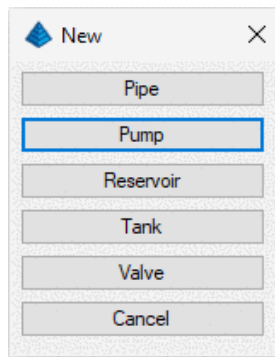
- We'd like to establish some initial, default settings for the project. Click the **Settings** (Gear) button to display a dialog box similar to that shown below:



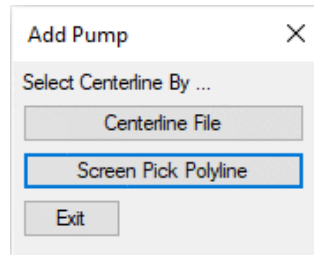
- Click the **Reference Surface** button to select the TIN file created earlier and then click the **OK** button.
6. For this example, zoom in to the upper left start of the pipe run so that we can select a small segment for our pump. The display should look like the following:



Click the **Add** (Plus) button from the *Network Actions* row to display a dialog box similar to that shown below:



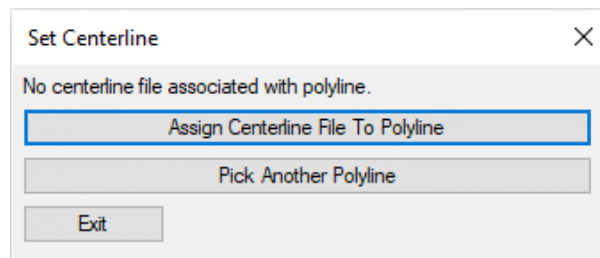
To create a new pump, click the **Pump** button and the following dialog will appear:



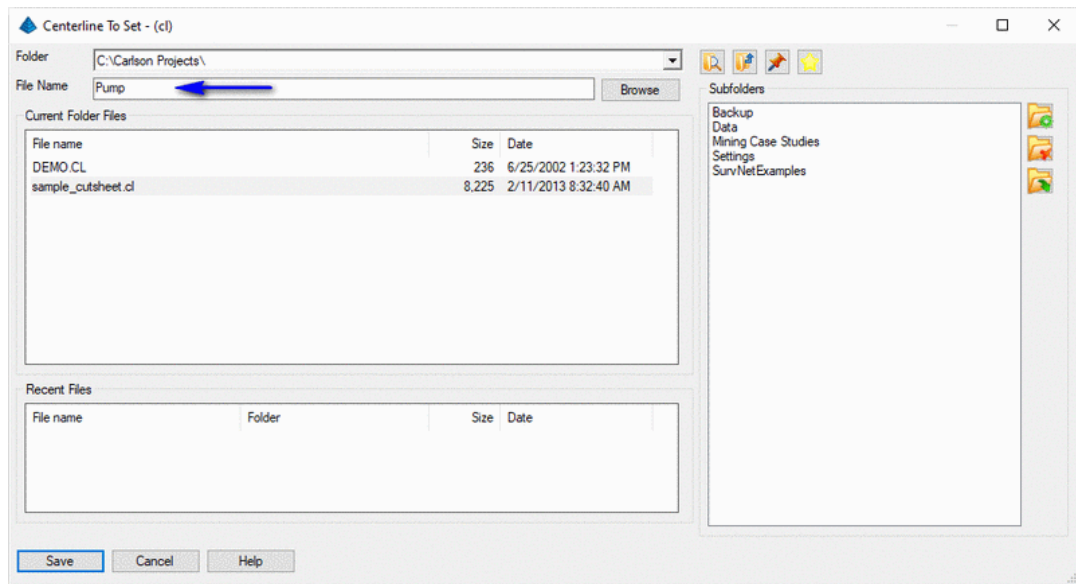
For this example, click the **Screen Pick Polyline** button and when prompted:

Select centerline polyline: pick the small polyline segment

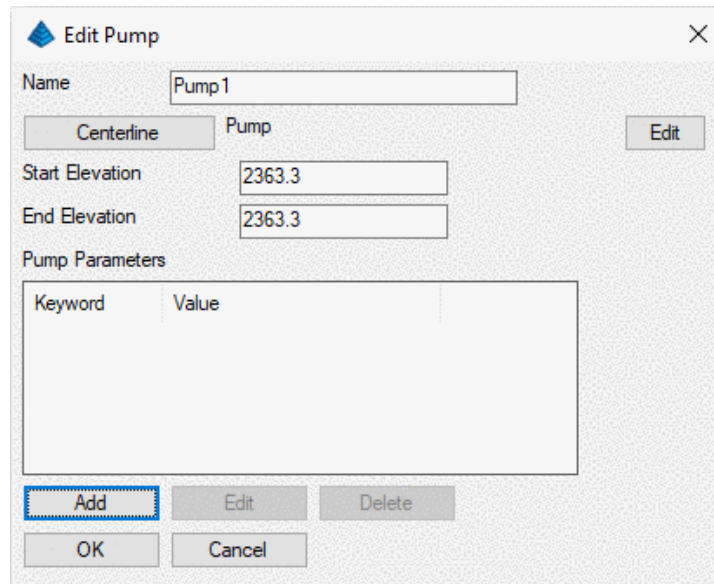
NOTE: Make sure the small segment is selected and not the longer pipe polyline.
After selecting the small segment, the following dialog will appear:



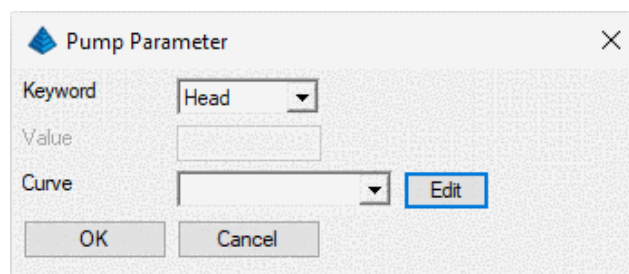
Click the **Assign Centerline File to Polyline** to display a dialog box similar to that shown below:



Set the file name as shown above and click **Save** when ready. Upon creation of the centerline file, a dialog box similar to that shown below appears:

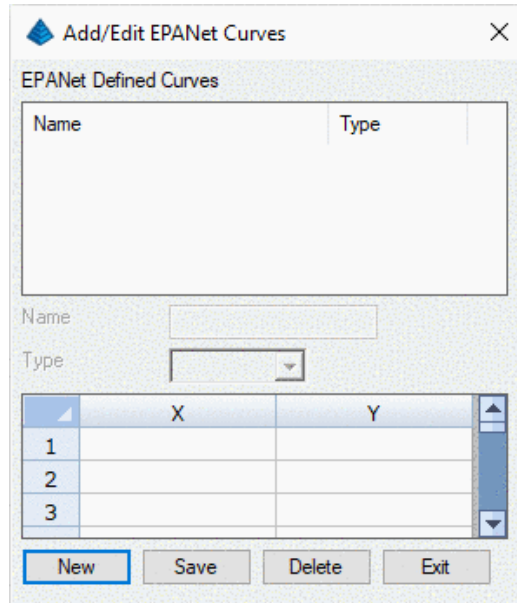


Set a name for the pump as suggested above and also set the **Start Elevation** and **End Elevation** to **2363.3** (the surface model elevation at the pump location). The next step is to add the required pump parameters. Click the **Add** button and the following dialog box will appear:



Select the **Head** keyword and the **Value** field will be disabled and the **Curve** field will be enabled. To create

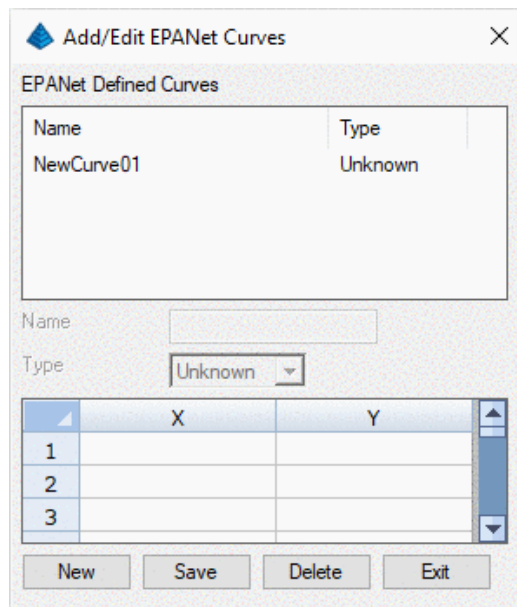
a new pump curve, click the **Edit** button and the following dialog will appear:



This dialog allows for the entry of any one of the four EPANet Curves:

- (a) Pump
- (b) Efficiency
- (c) Volume
- (d) Headloss

Click **New** and a new table will be created with a default name of *NewCurve01*:



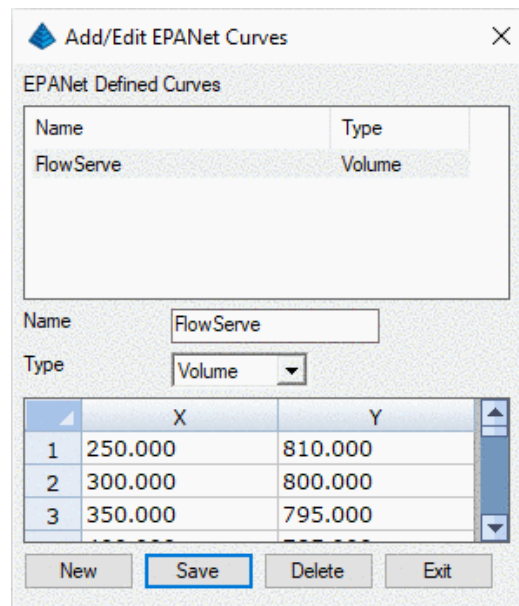
Select the *NewCurve01* entry and enter a new name of **FlowServe** and enter the following X, Y values:

Entry #	X (Head) Value	Y (Flow) Value
1	250	810
2	300	800
3	350	795
4	400	785

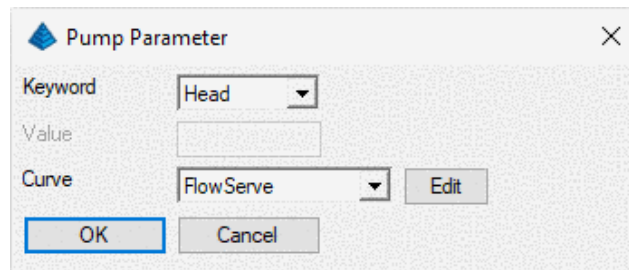
5	500	750
6	600	700
7	700	640
8	800	575

FlowServe Pump Curve Parameters

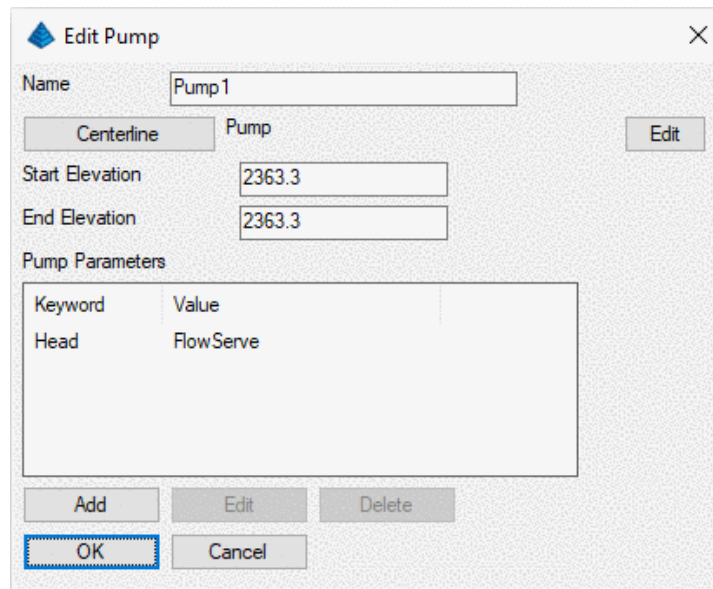
The result should look similar to that shown below:



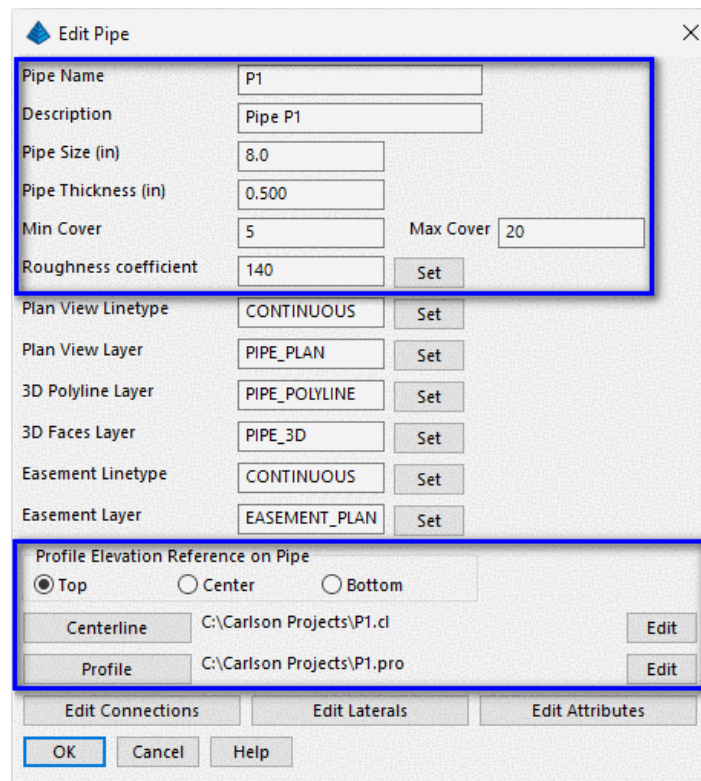
Click the **Save** button and then click **Exit** to return to the **Pump Parameter** dialog box. Select the pump curve you created in the **Curve** field as illustrated below and click **OK** when ready:



The **Edit Pump** dialog box should look as follows:



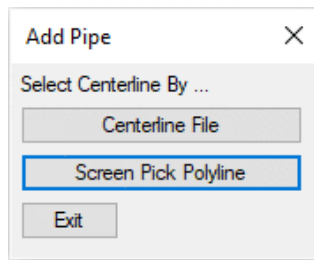
Click **OK** to return to the docked dialog box which should look as follows:



7. To add a new pipe, either:

- Click the **Add** (Plus) button on the *Network Actions* row and choose the **Pipe** option, or
- Right-click on the **Pipes** category in the tree-view list and choose the **Add Pipe** option

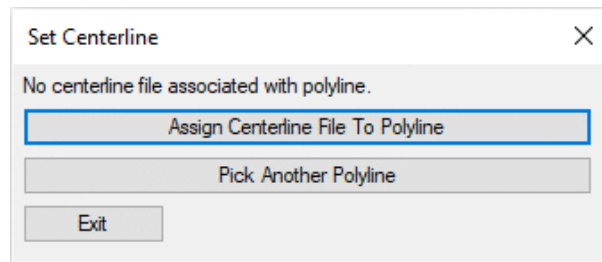
Do either option now to display the dialog box similar to that shown below:



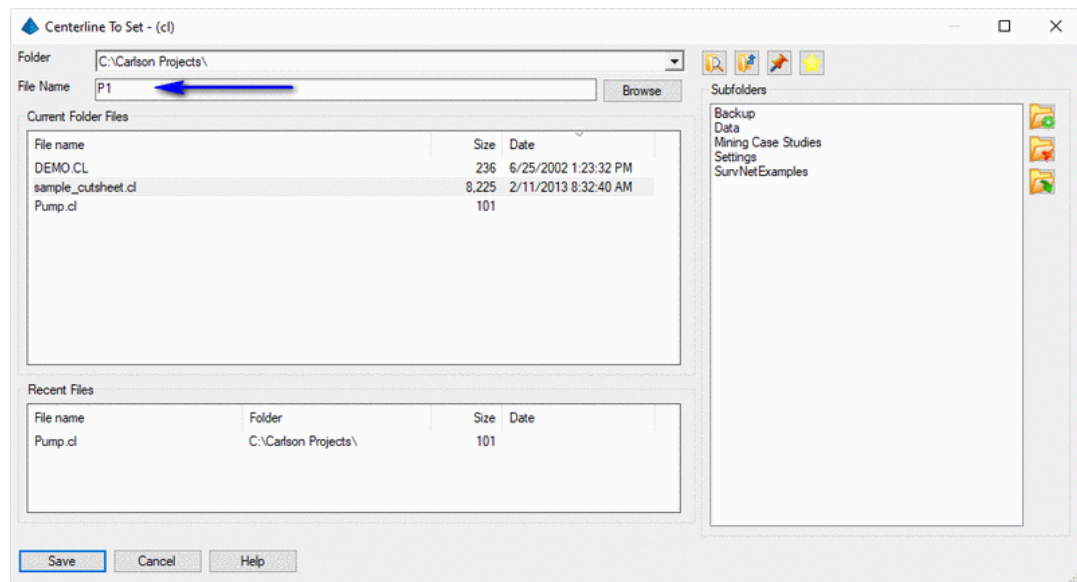
For this example, click the **Screen Pick Polyline** button and when prompted:

Select centerline polyline: pick the **Pipe 1** polyline segment

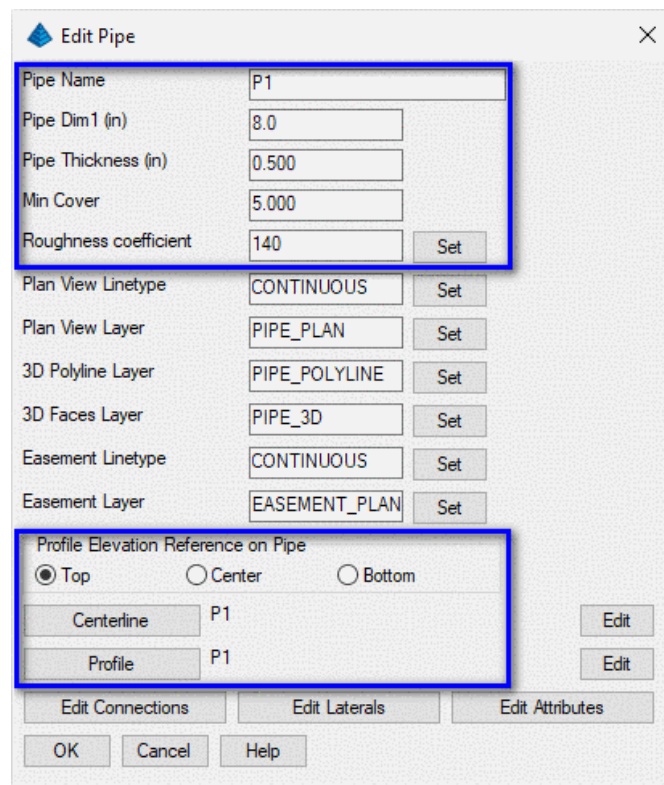
A dialog box similar to that shown below appears:



Press the **Assign Centerline File to Polyline** to display a dialog box similar to that shown below:



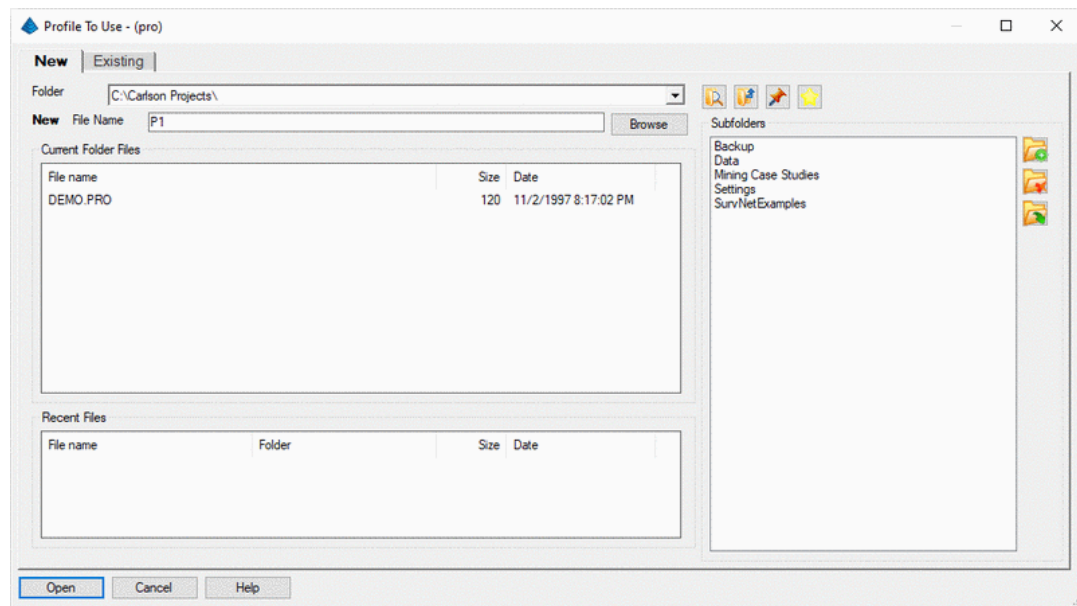
Set the filename as shown above and click **Save** when ready to display a dialog box similar to that shown below:



Update the following values:

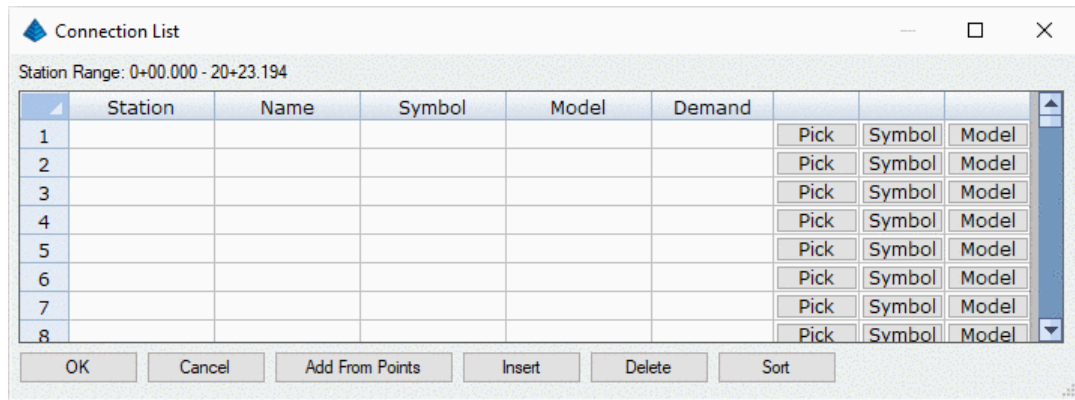
- **Pipe Dim 1** to 8.0
- **Pipe Thickness** to 0.500
- **Min Cover** to 5.0
- **Roughness Coefficient** to 140
- **Profile Elevation Reference on Pipe** to Top

Additionally, click the **Profile** button to open a dialog box similar to that shown below:

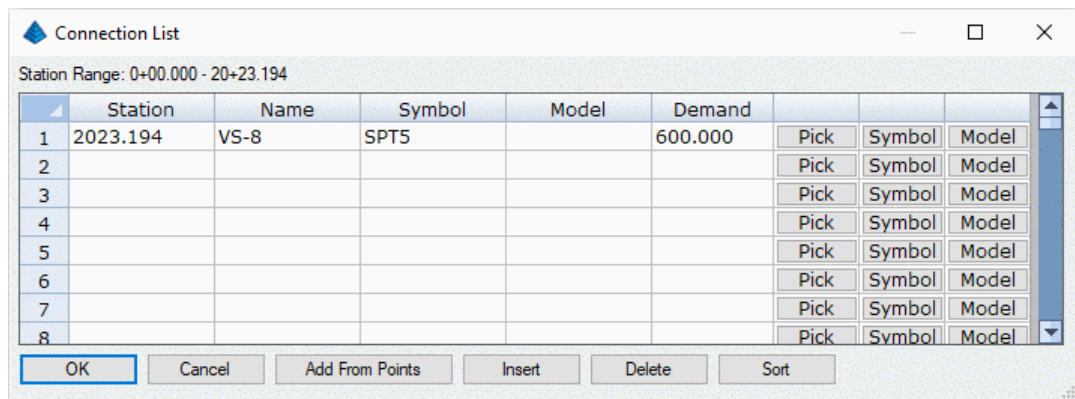


Set the profile file name to be the same as the centerline name cited earlier and click **Open** when ready. Click **OK** to dismiss the *Edit Pipe* dialog box and add the pipe to the tree-view collection.

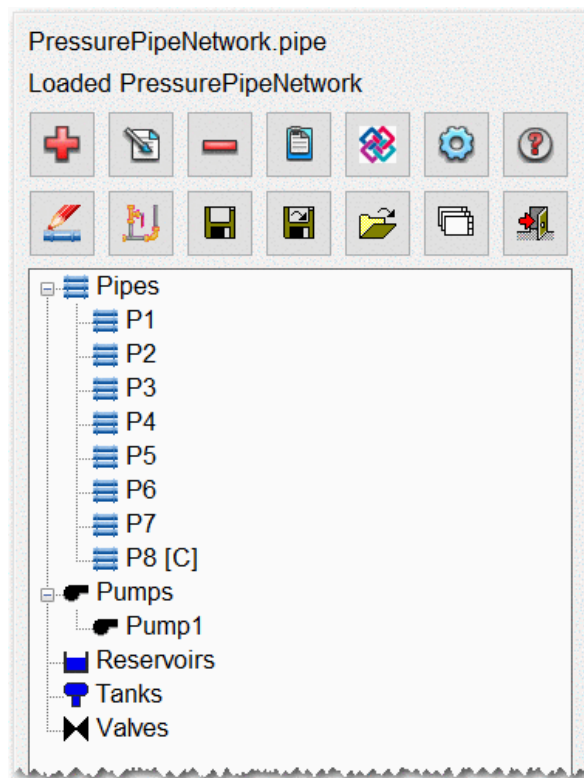
- Repeat the process of adding pipes for all eight pipes and use the same parameters for each pipe.
- Pipe **P8** has an required outflow of **600 cfs**. To enter the demand, right-click on pipe **P8** and click its **Edit Pipe** option. On the **Edit Pipe** dialog box, click the **Edit Connections** button at the bottom of the dialog to display a dialog box similar to that shown below:



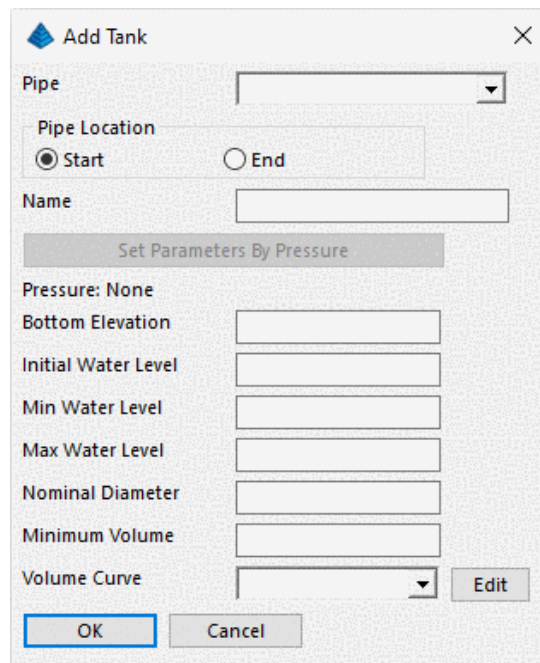
Enter the **End Station** value 2023.194 which is displayed at the top of the dialog. The remaining values should be as shown in the following:



Click **OK** to dismiss the *Connection List* dialog box and click **OK** to dismiss the *Edit Pipe* dialog box. The docked dialog should now look like the following:



9. In this example, we have two tanks. The **Upper Tank** and the **Lower Tank** as indicated on the drawing. To add a new tank, right-click the **Tanks** category in the tree-view and click the **Add Tank** button to display a dialog box similar to that shown below:



The **Pipefield** contains a list of *pipes*, *pumps*, and *valves* that are currently defined in the project. For the **Upper Tank**, select the **pump** created earlier and indicate the tank is at the **Start** of the pump. Enter the remaining tank parameters as follows and click **OK** when ready:

The 'Add Tank' dialog box is shown with the following settings:

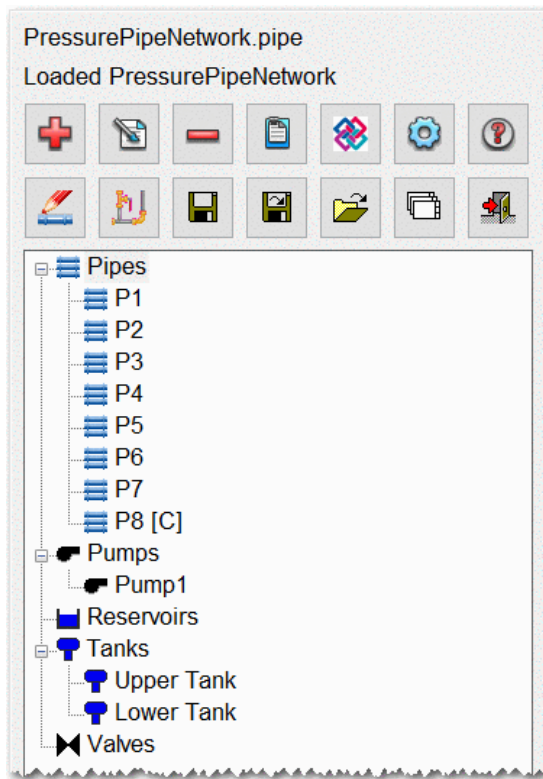
- Pipe:** Pump1
- Pipe Location:** Start (selected)
- Name:** Upper Tank
- Set Parameters By Pressure:** (button)
- Pressure:** None
- Bottom Elevation:** 2363.5
- Initial Water Level:** 35
- Min Water Level:** 10
- Max Water Level:** 38
- Nominal Diameter:** 30
- Minimum Volume:** 0
- Volume Curve:** (dropdown menu)
- Buttons:** OK, Cancel, Edit

Repeat the process and add the second **Lower Tank** with the following parameters and click **OK** when ready:

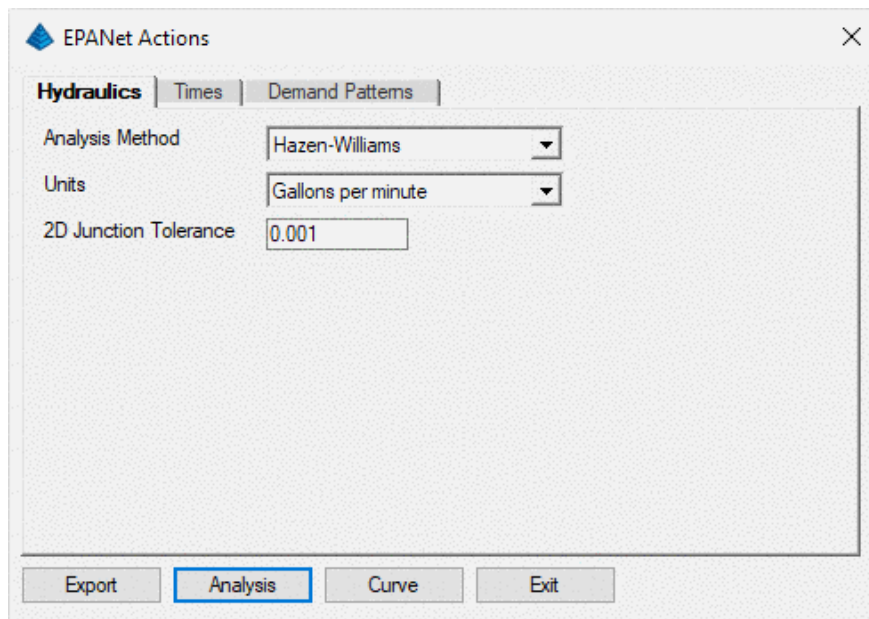
The 'Add Tank' dialog box is shown with the following settings:

- Pipe:** P7
- Pipe Location:** End (selected)
- Name:** Lower Tank
- Set Parameters By Pressure:** (button)
- Pressure:** None
- Bottom Elevation:** 2322.500
- Initial Water Level:** 35
- Min Water Level:** 15
- Max Water Level:** 38
- Nominal Diameter:** 30
- Minimum Volume:** 0
- Volume Curve:** (dropdown menu)
- Buttons:** OK, Cancel, Edit

NOTE: The *Lower Tank* is located to be at the **End** of P7.
The docked dialog should now look like the following:



10. Now that the model has been developed, an analysis can be performed. Click the **EPANet (Pipes with Valve)** button on the *Network Actions* row and the following dialog box will be displayed:



Set the values as shown above and click the **Analysis** button to run the analysis which displays the results in the Standard Report Viewer similar to that shown below:



LDT Migration Guide

19

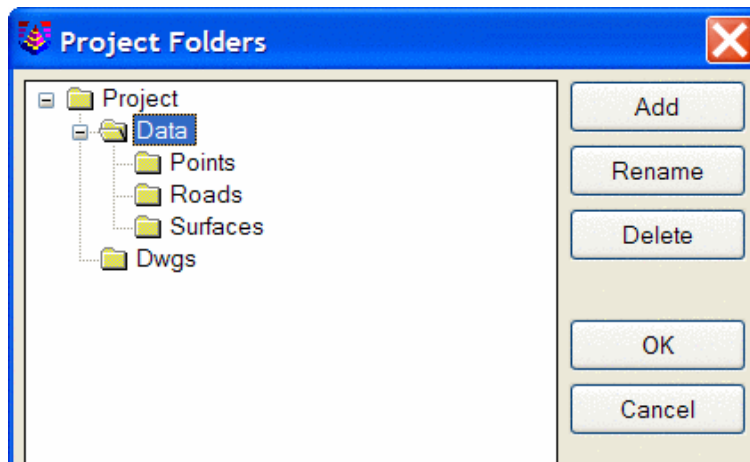
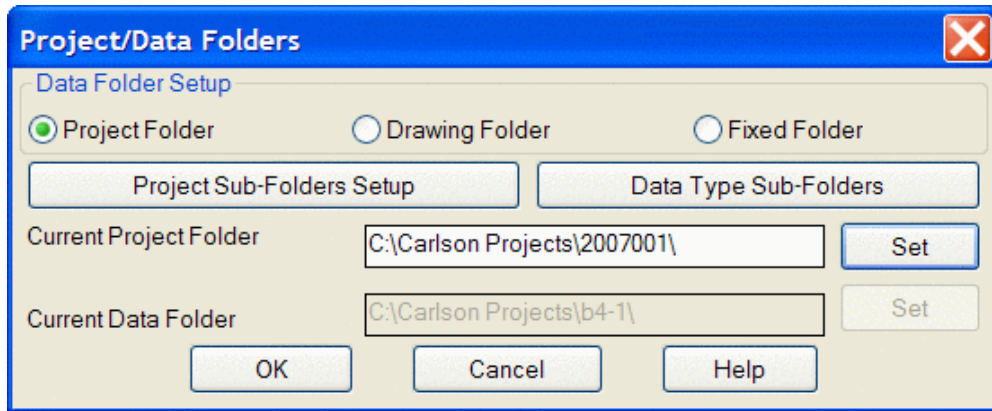
Carlson Civil Suite encompasses all of the functionality of Survey, Site Grading, Road Design, Hydrology and GIS, along with a comprehensive set of drafting and annotation tools. Carlson Civil Suite runs atop any installed AutoCAD executable, running as plain AutoCAD, or found within Map, Land Desktop or Civil 3D as well as IntelliCAD.

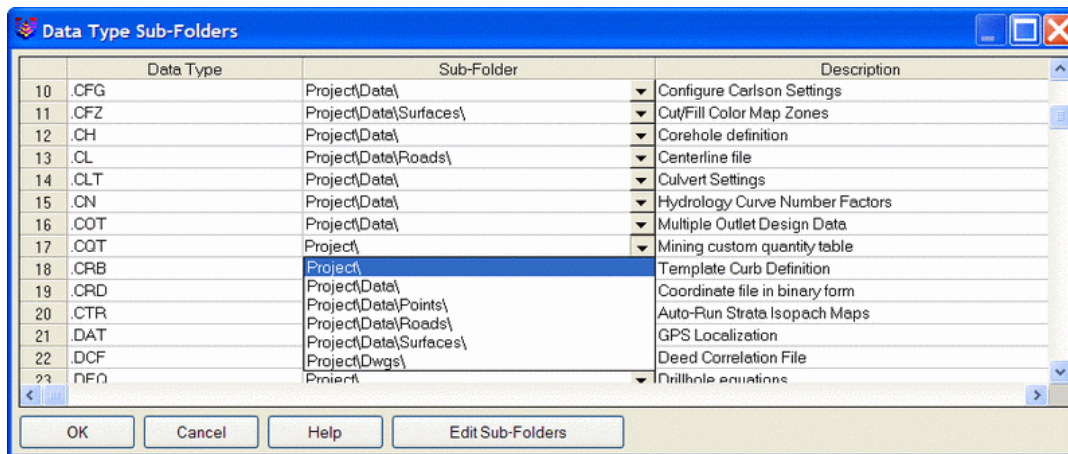
Evolved from the Carlson SurvCADD product line, first introduced in 1989, Carlson Civil Suite introduces several brand new state of the art tools, including a dynamic road design program called Road Network, and a dynamic storm drain design program called HydroNet. While these functions automatically respond to design changes, adjusting sections, profiles, and grading, Carlson Civil Suite accomplishes this automation without introducing custom objects, so sharing drawings with others is not an issue.

Carlson Civil Suite continues the Carlson Software tradition of unlimited free tech support, so when needed, help is always a free phone call away, and also introduces the newest support technology in the form of an online, web-based reference manual, complete with demonstration and training movies.

Data File Types and Storage

Carlson Civil has a similarity to Land Desktop in the use of external files to store design data, but differs significantly in that in Carlson Civil the naming and placement of these files is determined by the user, not the software. Carlson Civil offers three distinct methods of file storage, the choice is up to the end user. Carlson Civil data files can either be placed in a single location, known as a data folder, placed with the drawing they are associated with, or placed in a user-defined folder structure. The placement of files within that structure is also totally user-defined, based on the assignment of file types (extensions) to folders.





File types used by Carlson Civil include:

.crd - Point data, coordinate file

.rw5 - raw survey data, contains all observations

.cl - centerline, describes a 2D alignment

.tin - Surface, newer format, more efficient than .flt in most cases, especially for machine control

.flt - Surface, original format

.grd - Surface grid file, used for volumes

.cfg - Stores configuration settings

.fld - Field to Finish file, stores rules for inserting symbols for points (LDT Description Key functionality) and automated linework functionality (Autodesk Survey Figures equivalent)

.lot - Lot file, stores parcel geometry

.adf - Annotation Default file

.pro - Profile file

.mxs - Section Alignment file

.grp - Point Group definitions

.rdn - Roadway Networks

.sct - Road Sections

.tpl - Road Templates

.rdf - Road Design Files

Settings

Carlson Civil uses several techniques to store settings. There are three main categories of settings; Drawing Setup settings, such as drawing scale and units, Command-specific settings, such as the layer to draw contours on, and Generic control settings, such as whether to link drawing points to the external coordinate file (.CRD).

Drawing Setup settings are stored directly within the drawing files (.DWG). Carlson Civil also creates a file for each drawing using the drawing name with a (.INI) file extension. This file stores a list of all of the design files that are used or created from within the drawing, such as centerline files (.CL), profile files, (.PRO), etc.

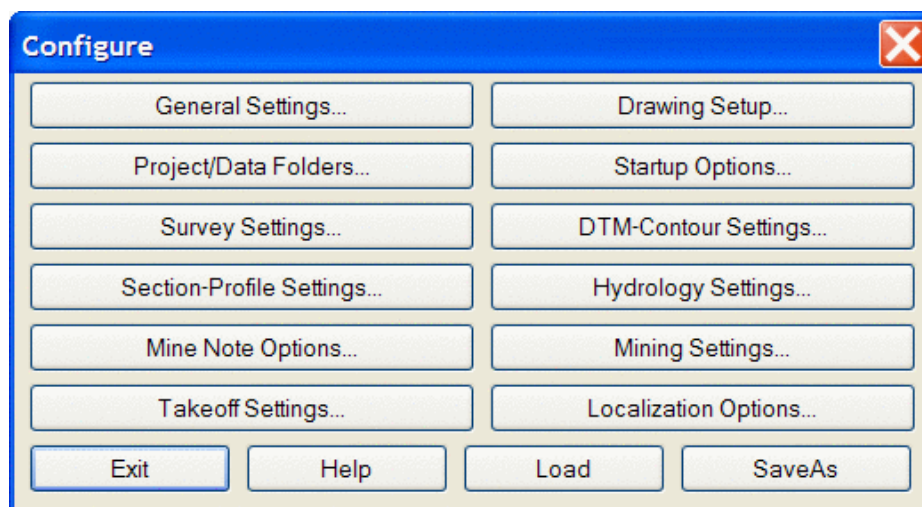
Command-specific settings are stored within a set of files with (.INI) file extensions, with the command name as a filename, such as roadnet.ini, or mapcheck.ini. These are typically stored in the \USER folder, and are created as the commands are first accessed.

Generic control settings are stored within a special (.INI) file named Carlson.ini. When new drawings are created, this file is read to set these type of generic controls.

Drawing Setup settings can be accessed directly from the Settings menu, or through the Configure command. If the Drawing Setup dialog is accessed through the Configure command, when exiting the main Configure dialog, the user is prompted whether to save changes to Current and Future drawings, or Future drawings only. Current and Future saves the Drawing Setup settings to the current drawing internally, and updates the Carlson.ini file, while the choice of Future Only does not affect the current drawing, only the Carlson.ini file, and therefore any new drawings created. To change Drawing Setup settings for only the current drawing, do not go through Configure, but use the Drawing Setup command directly off of the Settings menu.

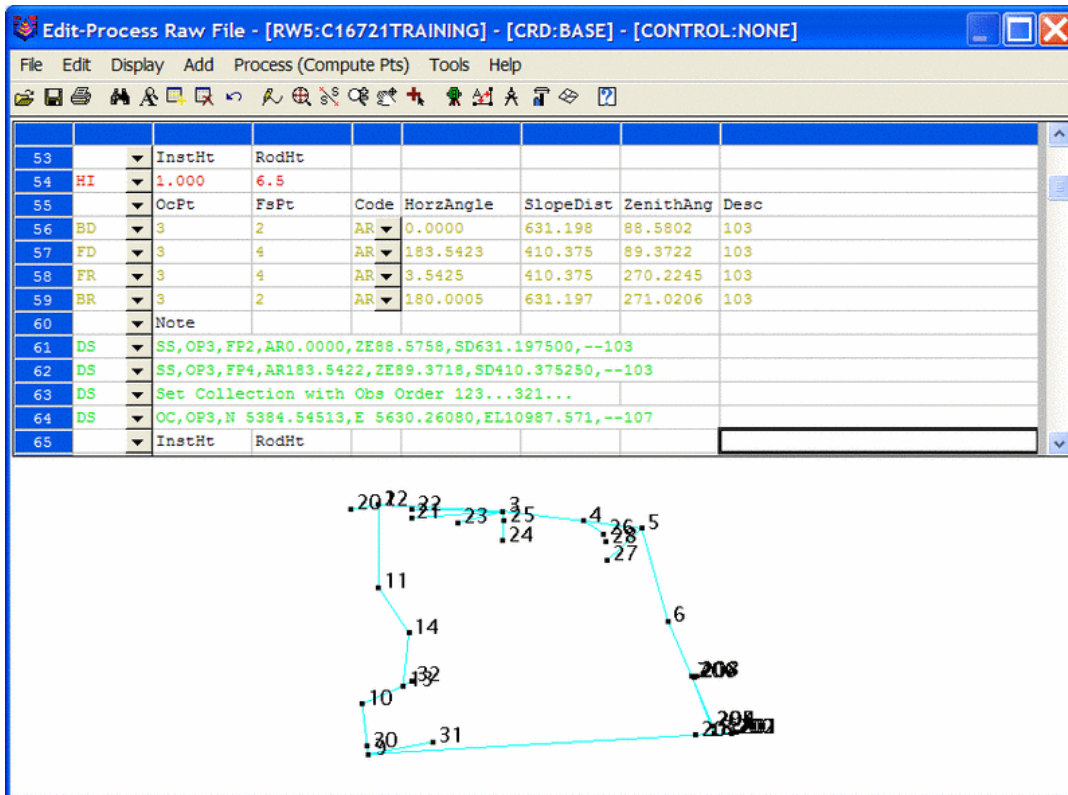
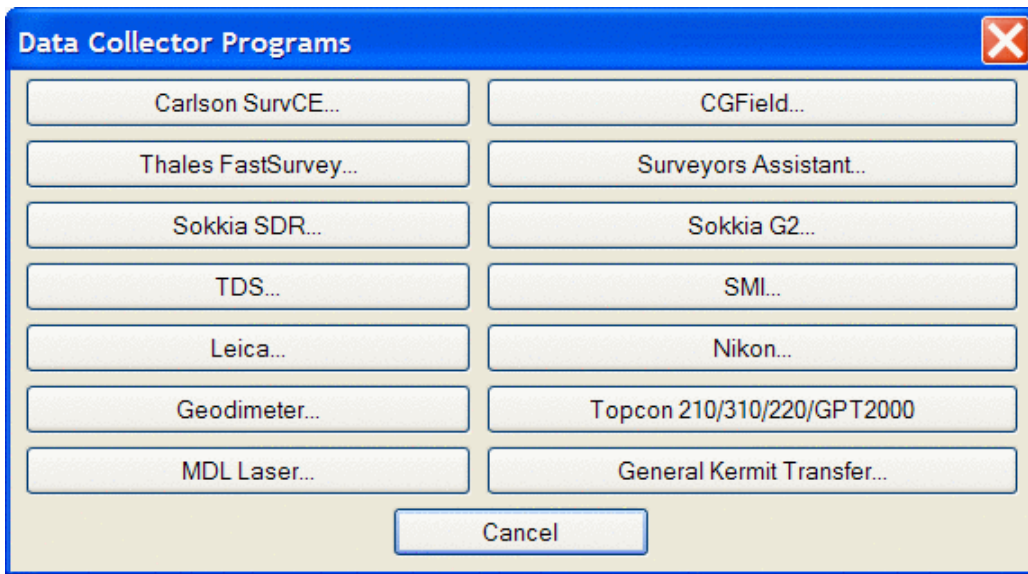
Command-specific settings are accessed when the commands are actually run. The settings displayed are being read from the command's own specific (.INI) file, and any changes made are written to the same files. This way the commands automatically recall the settings that were used for the previous run. Alternatively, the Configure command can be used to access command-specific settings.

The Configure command provides access to a dialog box with 12 buttons, each leading to settings for specific aspects of the software. Changes made within any of these are also automatically saved to the corresponding command (.INI) file, or to the Carlson.ini file in the case of generic settings. The Configure command also provides a Save and Load functionality, utilizing configuration files (.CFG). Saving a configuration file (.CFG) saves all settings currently stored in all of the command-specific (.INI) files, and the Carlson.ini file. Loading a configuration file (.CFG) sets all settings within all these files.



Survey

The complete range of Survey functionality is contained within the Carlson Civil, and is also available in the Carlson Survey program. This includes communication with data collectors, editing and processing of raw survey data, including traverse adjustment, and Field to Finish, which controls the generation of point symbols and linework.



Land Desktop uses Description Keys for point-based symbol insertion, and Autodesk Survey uses Figure notation for the generation of linework. Autodesk requires field coding to produce linework, and the processing of that information takes place when the raw file (.fbk) is Imported. Carlson inserts symbols and linework with one function known as Field to Finish (F2F), and performs this task using the point descriptions from the coordinate file, not the raw observations file. So linework can actually be generated from any set of points, even if no field coding has taken place. However, the addition of field coding can certainly make the generation of linework more precise. Carlson Civil can use LDT Description Key file to start a Field to Finish Code File.

Code Table Settings

Code File:

Coding Method

Process Carlson Coding

Process Eagle Point Coding

Process CAICE Coding

Draw Field Codes Without a Suffix as Points Only

Use Multiple Codes for Linework Only

Split Multiple Codes

All None Prompt

Max Delta-Height for Linework

Max Length for Linework

Field to Finish

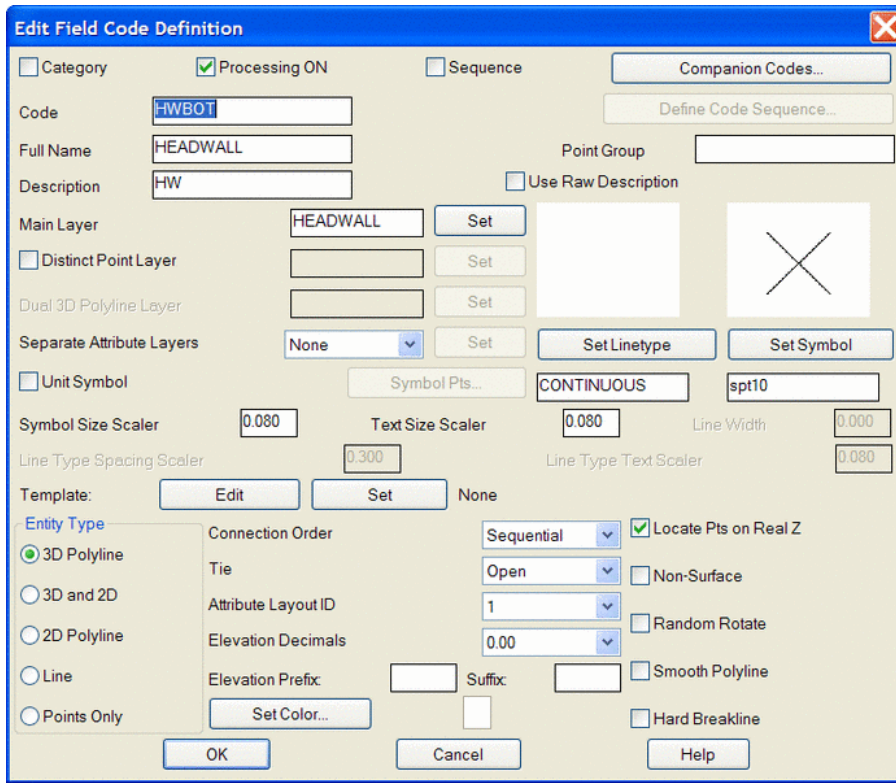
DATA:C:\Carlson projects\B4-1\Deed-reader.crd, CODE:C:\Carlson2007b5\Data\Carlson.fld

CODE	FULL NAME	DESC	SYMBOL	LINETYPE	ENTITY	TIE	LAYER	ON/OFF
SIGN	SIGN	SIGN	spt17	BYLAYER	Point	Open	SIGN	On
SHD	TOP OF CUR	SHD	spt10	BYLAYER	3DPline	Open	CURB-TOP	On
WLK	SIDEWALK	SW	spt10	BYLAYER	2DPline	Open	SIDEWALK	On
---> UTILITIES <---								
WG	WG	WG	spt39	CONTINUOUS	Line	Open	UTILITY	On
MH	MANHOLE	MANHOLE	SPT34	sewer	2DPline	Open	SEWER	On
SMH	SMH	SMH	spt50	CONTINUOUS	Line	Open	SEWER	On
TELBOX	TELBOX	TELE	spt30	CONTINUOUS	Line	Open	UTILITY	On
BOX	BOX JUNCTI	BOX	spt30	BYLAYER	Point	Open	UTILITY	On
CATU	CABLE TV	CATU BO	spt29	BYLAYER	Line	Open	UTILTY	On
CB	CATCH BASI	CB	spt66	BYLAYER	Line	Open	DRAINAGE	On

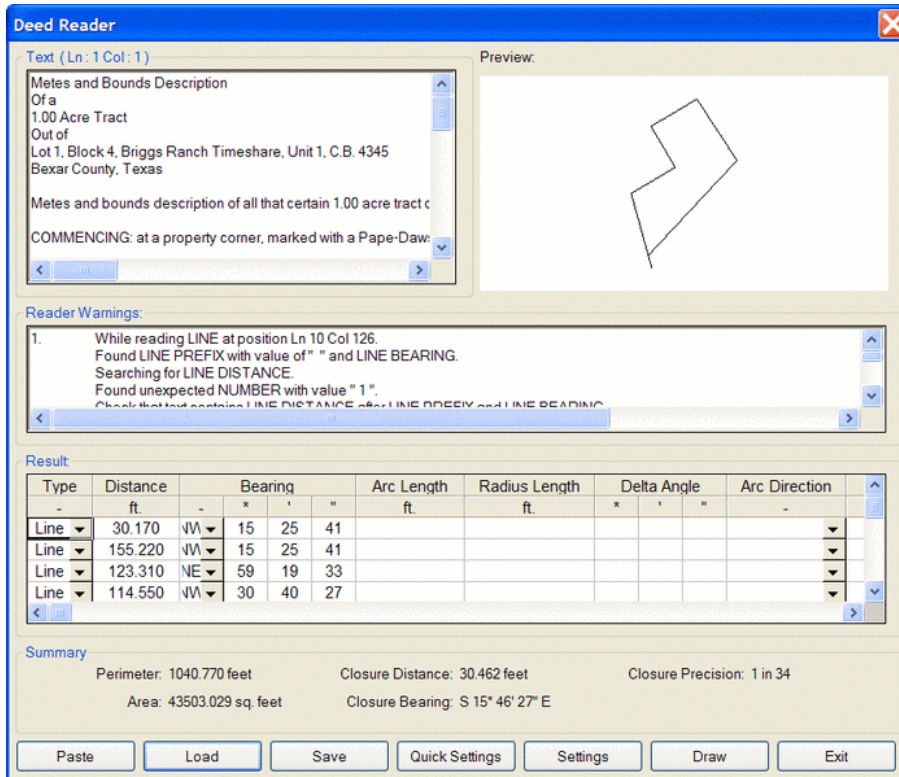
Code Table

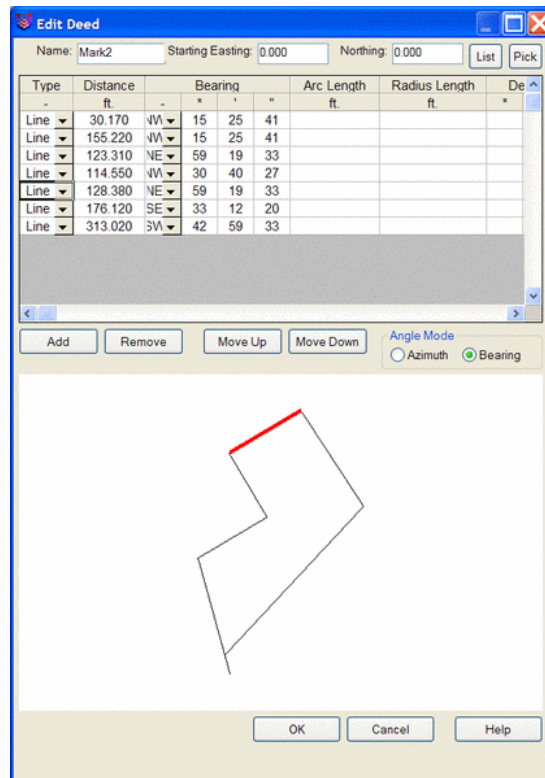
Code Definitions

Coordinate File

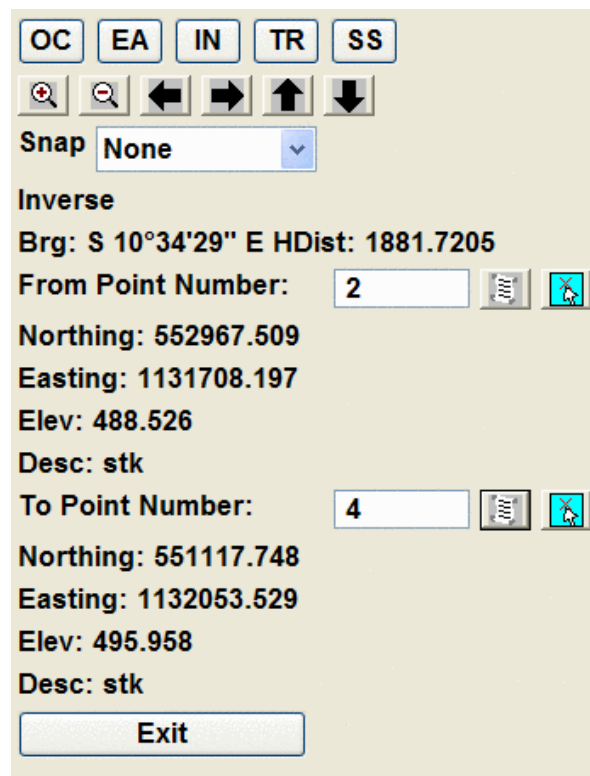


Carlson Civil also has powerful functions to Enter Deeds to draw linework, and a Deed Reader which reads a text file of a deed and extracts the data needed to define and draw the deed in the drawing.





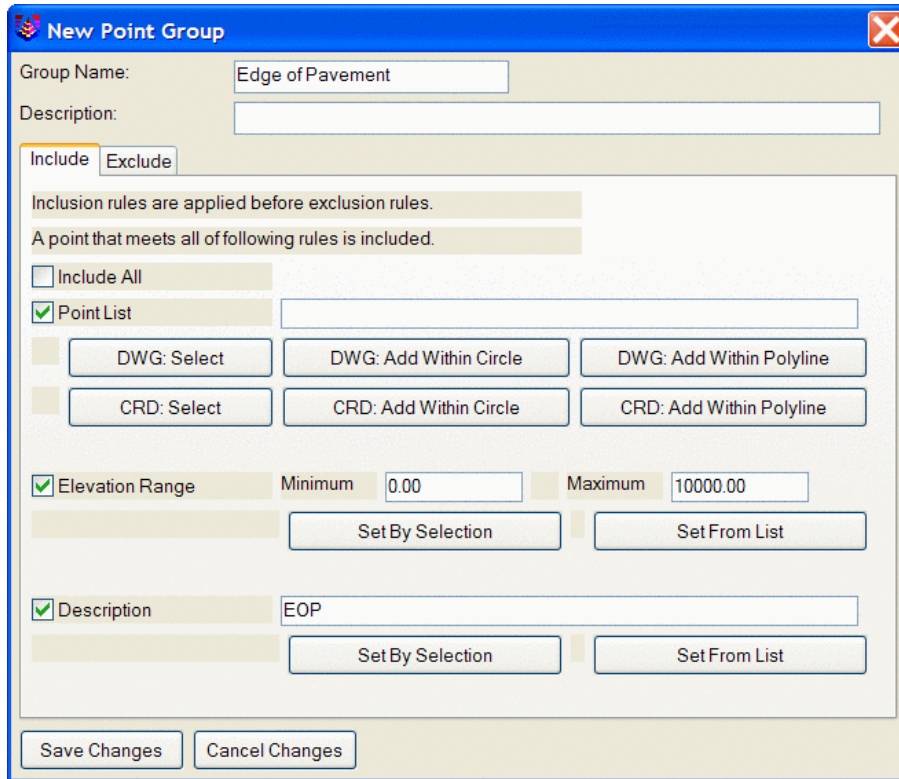
Carlson Civil also has a sidebar COGO function called Visual COGO to enter Traverse, Sideshot and other survey data.



Points and Point Groups

This use of point data in Carlson Civil revolves around the use of .crd files, also known as coordinate files.

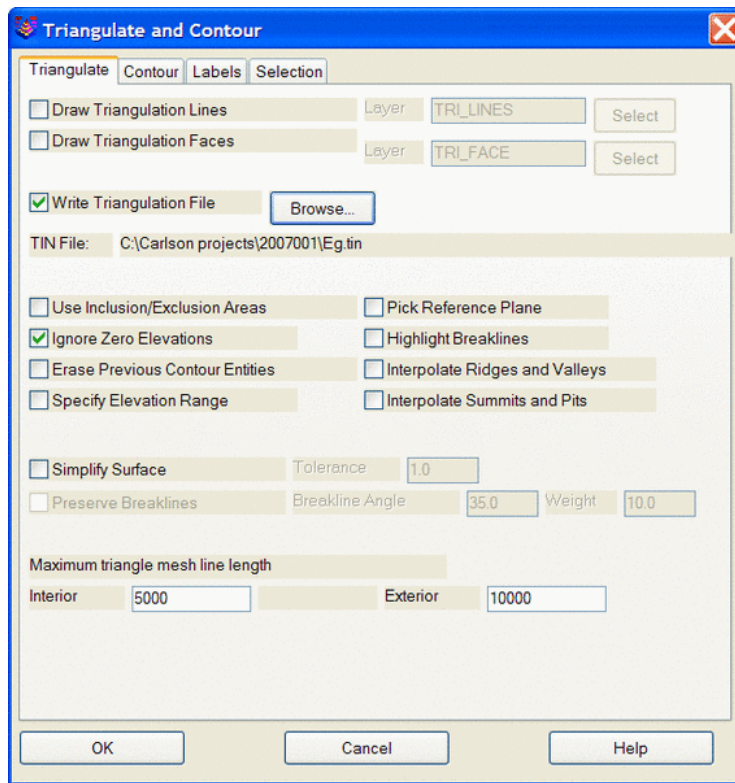
Carlson Civil supports the creation of Point Groups similar to LDT and applies them in many applications, such as using them to create surfaces, editing and listing. Carlson Civil stores Point Group definitions are associated individually to each .crd file.



Surfaces and Contours

In Carlson Civil, Surfaces can be written out as external files, but a lot of design and computations involving surfaces can also be accomplished directly within the drawing without writing out external files. Triangulation surfaces can be written out as .tin or .flt and grid surfaces as .grd files. When an external file is generated, a named Surface is also stored in the drawing. This named Surface is accessed through the Surface Manager, where it can be edited.

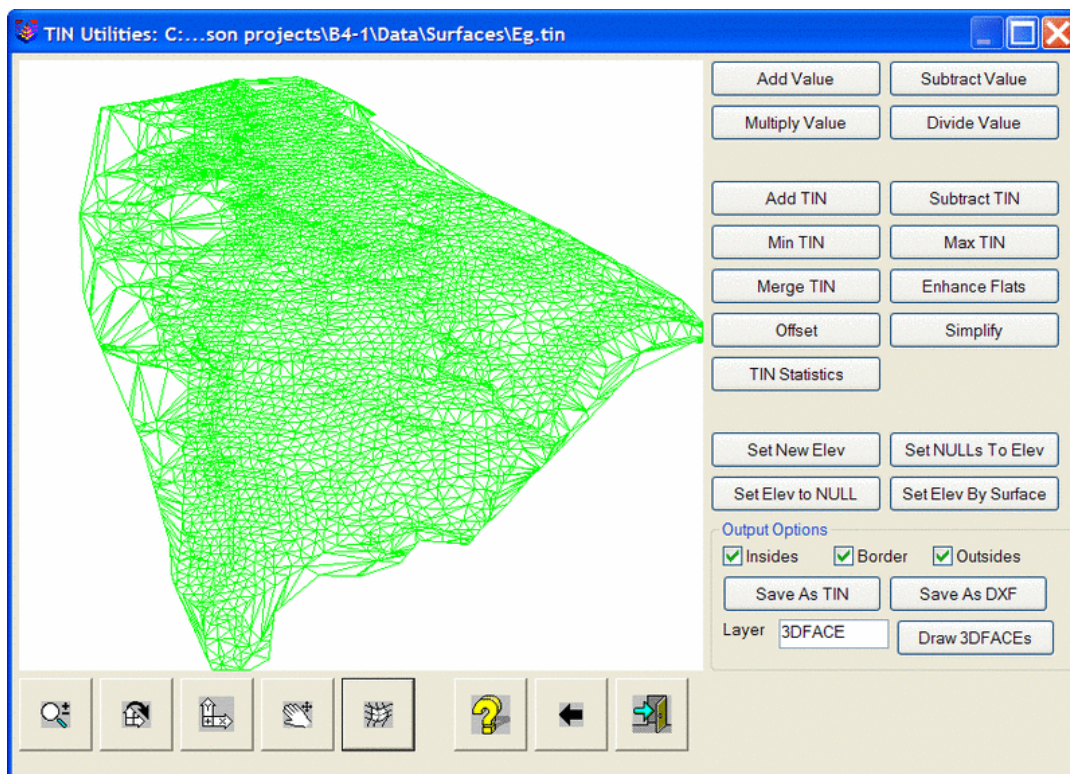
The main Carlson Civil command for working with Surfaces is called Triangulate and Contour. It is a single dialog box with four tabs, and covers the entire process of specifying the general settings to generate the Surface, creating Contours, generating Labels, and specifying the data source(s) for the Surface.



In Carlson Civil, Contours are generated as regular AutoCAD Polylines. Contours can be generated and automatically labeled simultaneously, or labeled after they are generated. Labels can be generated with wipeouts to hide the contour beneath them, and can also be slid along the contour to easily change their location.

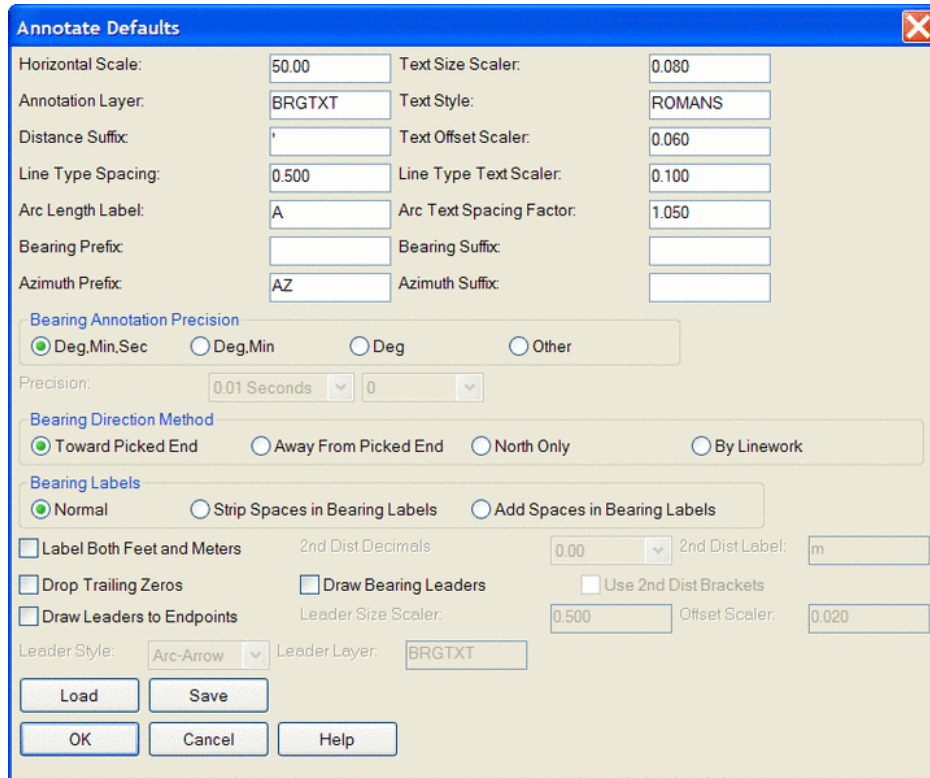
The Triangulation Surface Manager has tools to edit, add and remove data points and breaklines and update the triangulation dynamically. It also allows you to change the display properties for the triangulation, contours and labels.

The TIN Utilities function provides a powerful set of editing tools.

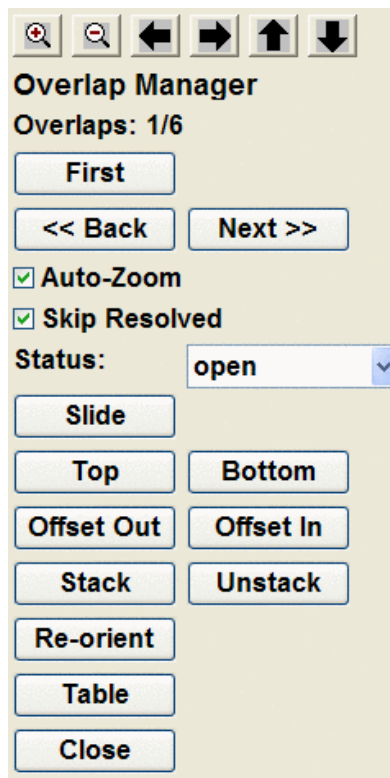


Line and Curve Labeling

Carlson Civil refers to the process of labeling lines and curves as Annotation. Lines and curves can be labeled in a dynamic or static mode, depending on the label settings.



There is also a powerful set of tools to check for and correct overlapping labels.



Volumes

There are several ways to generate volumetric calculations within Carlson Civil. Volumes By Layers, Volumes by Triangulation, Calculate Section Volumes and Two Surface Volumes (Grid volumes).

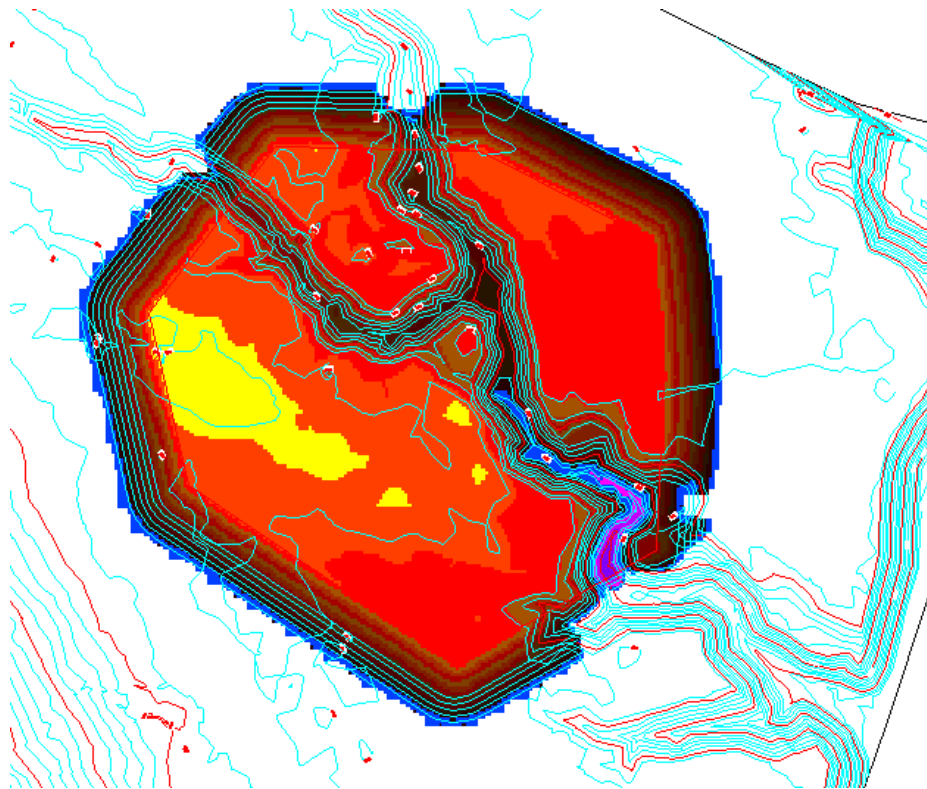
Volumes by Layers is potentially the quickest method. It needs no existing data files and creates no files in the process. You simply specify which layers to use for each of the 2 surfaces.

Volumes by Triangulation uses two triangulated surface files as the source of the data. These files are created through the Triangulate and Contour command. You can choose to create contours in the drawing when you create the TIN files, or just create the files without generating contours.

Calculate Section Volumes calculates volumes by end areas from two cross section files.

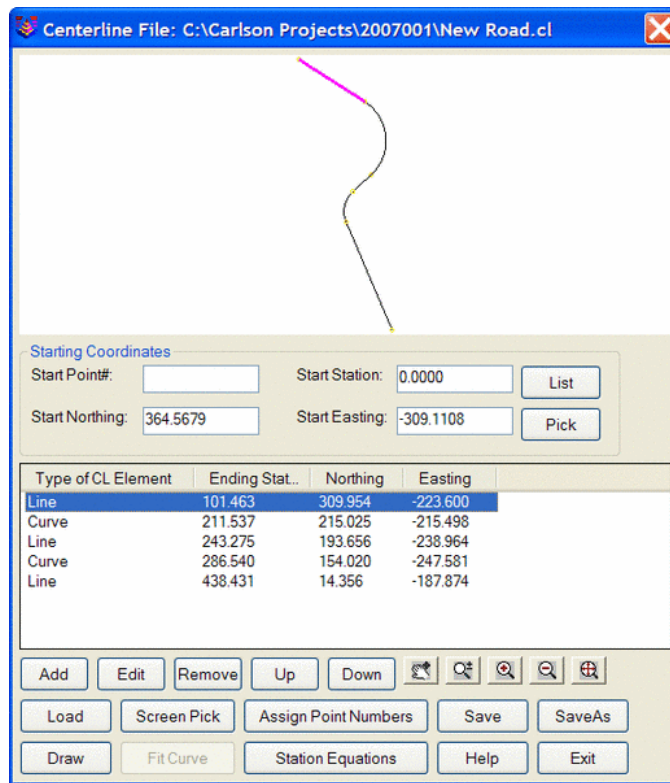
The Two Surface Volumes method uses two predefined Surface files as the data source for the calculations.

Once calculated, you can generate Cut and Fill Color Maps, Cut and Fill Centroids, and Cut and Fill Labels to illustrate the volumes.



Alignments

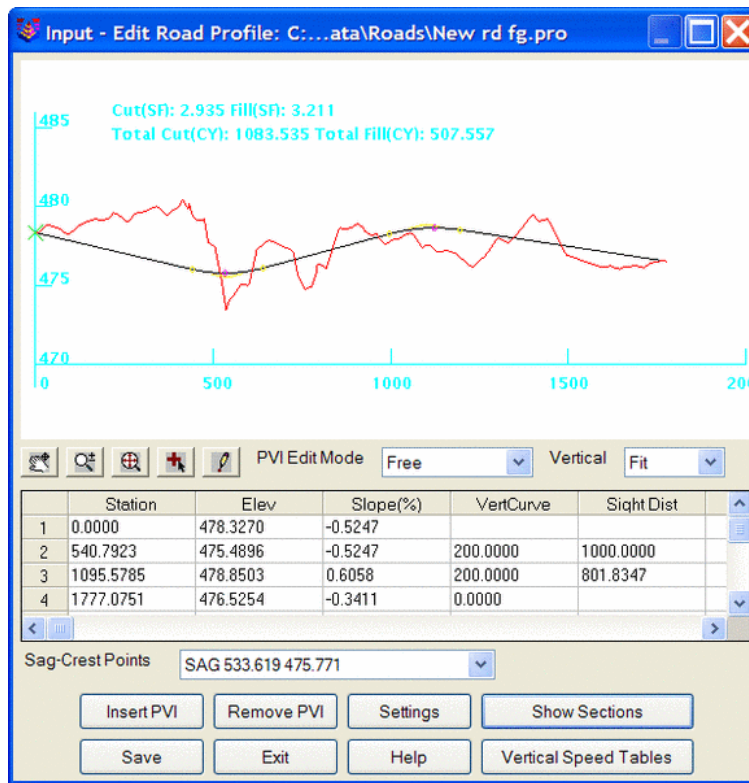
Horizontal Alignments in Carlson Civil are known as Centerlines. They are stored in .cl files. They can be created and edited through the Centerline Editor command. Polyline can be drawn first and then used to define Centerlines. Once defined as a centerline, double-clicking on the polyline invokes the editor dialog box.



Profiles

Profiles are stored within .pro files, with user-defined names. Existing Ground/Surface Profiles and Proposed Finished Grade/Design Profiles both use this filetype. Multiple .pro files can be drawn on the same Profile Grid.

There are several different routines for creating profiles including Profiles From Triangulation Files, Profile From Surface Entities and Profile From Points On Centerline. Before using these profile creation routines, the horizontal alignment needs to be created as a centerline file or polyline. The Quick Profile routine can be used to create profiles in one step.

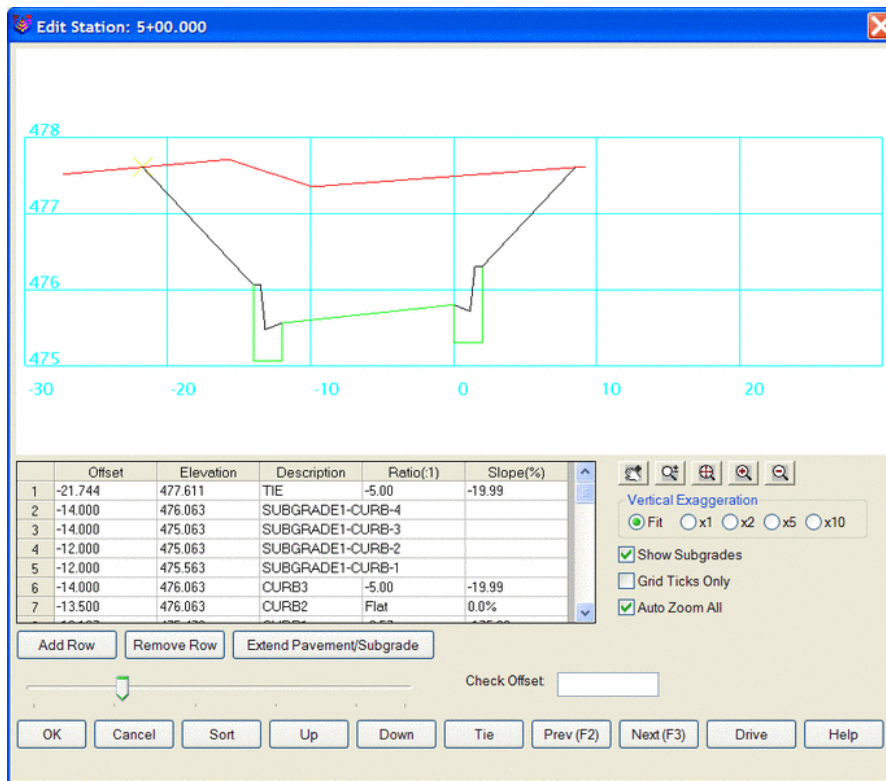
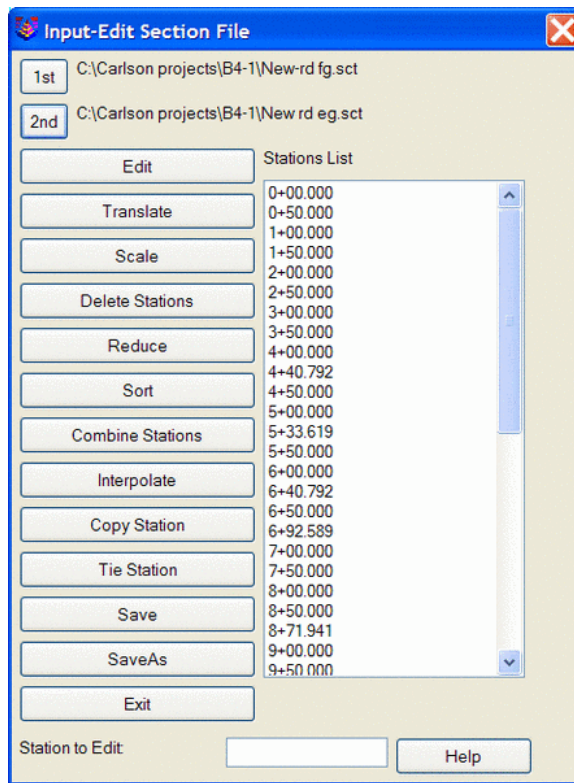


When using Process Road Design or RoadNet, the existing ground Profile can be generated automatically as part of the process, simply by specifying the Surface to use, and so is not a separate prerequisite. The Proposed Finish Grade Profile can then be added in the editor, and the Roadway processed, all without ever drawing anything in the drawing itself. The more traditional LDT approach of generating an existing ground Profile in the drawing and then adding a proposed finish grade Profile by drawing on it in the drawing is also an option.

Roadway Cross Sections

Roadway Cross Sections are based on Cross Section Alignments (.mxs files) that are defined by the Section Alignment Editor command to set the station interval and max offsets left and right. Similar to profile creation, there are several routines to create sections including Sections From Triangulation, Sections From Surface Entities and Sections From Points. The Process Road Design and RoadNet commands can create final sections.

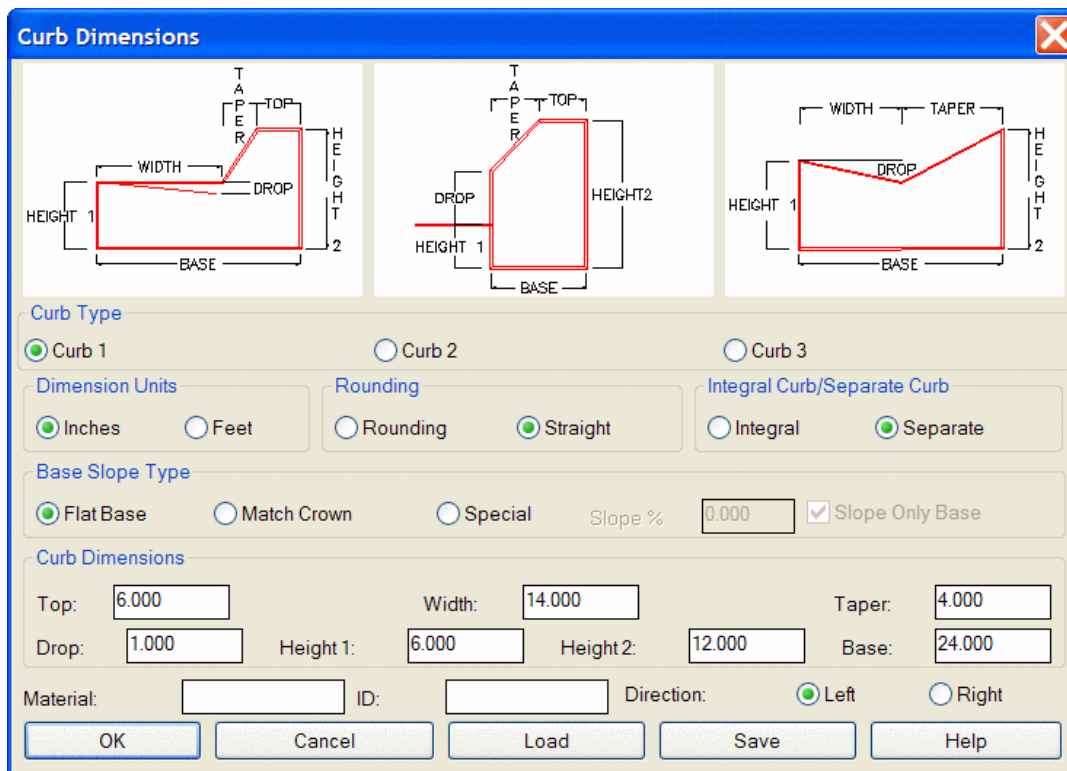
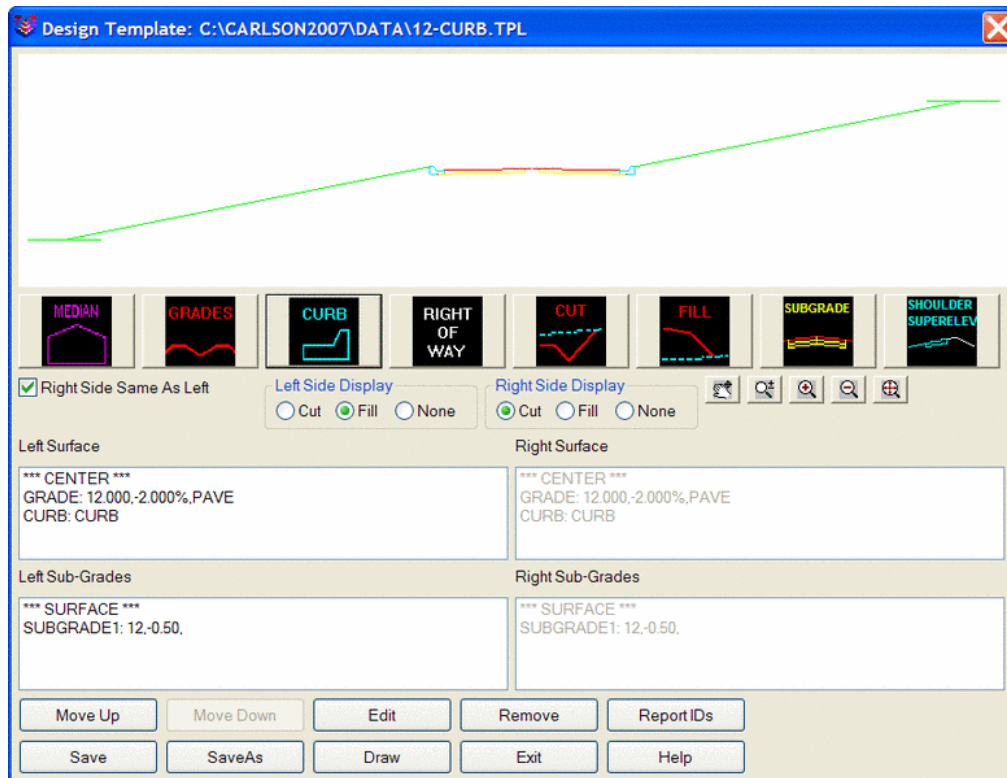
Once section files (.sct) are created, the Section File Editor command allows you to review and edit the section data. Also the Draw Section File and Section Report commands can be used.



Roadway Templates

Roadway Templates are created within the Design Template dialog box. They are stored as .tpl files, and can be applied to any road design. Templates are used in Process Road Design, Road Network and within the Road Profile Editor dialog.

The Design Template defines the road grades, subgrades, curb, superelevation break points and cut/fill slope treatments.



Design Control

The idea of Design Control in LDT exists in Carlson Civil in a number of places.

- Template Control - Templates are assigned to centerlines either in the Process Road Design dialog box, or in the Road Profile Editor dialog.
- Slope Control - In Carlson Civil, side slopes are actually part of the Template definition.
- Ditches - Ditches are defined within the Design Template as part of the cut/fill treatment.

Transitions - There are four methods to work with Transitions in Carlson Civil.

The first is called a Template Series, in which multiple Templates are assigned to a single Centerline at different stations. Next is a Template transition, in which a single Template is assigned to the entire Centerline and the user edits it at different stations. Next is a Template Point Centerline, in which a secondary Centerline is defined and attached to a point on the Template for specific horizontal control, such as a lane widening. Last is a Template Point Profile, in which an additional Profile is defined and attached to a point on the Template for specific vertical control, like the flow line of a ditch.

Superelevations - The template breakpoints for superelevation pivots are defined within Design Template. The stations for the superelevation transitions are set in the Superelevation Editor command.

Roadway Intersections

One of the very exciting features of Carlson Civil is the use of Roadway Networks. These are sets of centerlines that are aware of each other and clean up at intersections, horizontally and vertically.

Road Network: Salem

Road Name

- Main
- NEW RD

Add Edit Remove

Intersection

- Main and NEW RD
- Main and MULT2:NEW RD

Edit Intersection

Cul-de-Sac

- NEW RD at End

Add Edit Remove

Process Report Settings Help

Save SaveAs Load/New Exit

Cul-de-sacs

Cul-de-sacs are handled elegantly with the Road Network. Cul-de-sacs can be easily added to any roadway, and designed at a very detailed level, including a profile for the circumference.

Input Data

Cul-de-Sac Centerline Position

Start End

Center Station: 283.315 Delta: 0.000

Cul-de-Sac Radius: 45.000

Fillet Radius: 25.000

Offset: 0.000

Tear Drop Mode Setback: 0.000

Template ID: PAVE

Profile Transition VC: 0.000

Output Files

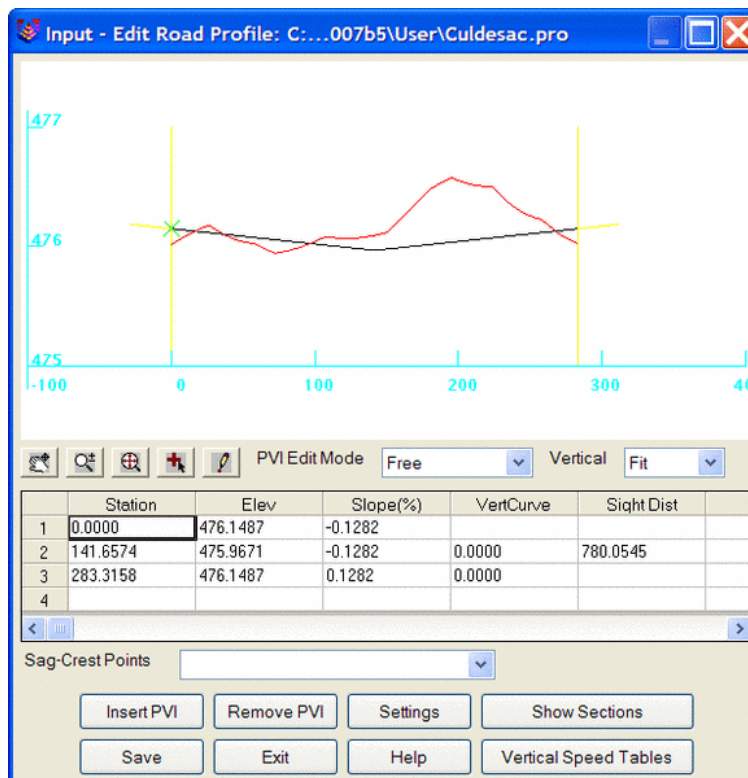
Centerline: NONE

Profile: NONE

Existing Section File: NONE

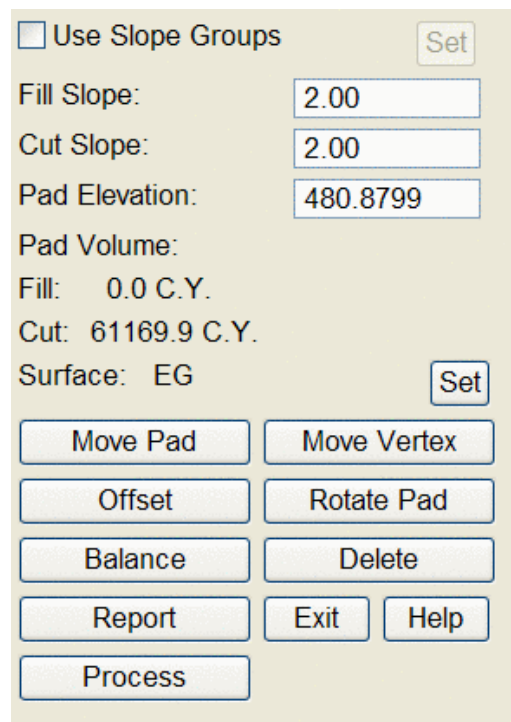
Final Section File: NONE

OK Cancel Help



Grading

Carlson Civil includes a variety of tools to accomplish the task of grading, including point commands, 3D polylines commands, contour commands and a grading mechanism known as a Pad Template, which is essentially the equivalent of an LDT Grading Object. Pad Templates provide some significant improvements, however, such as the ability to have a separate surface for the area inside the "pad", which moves horizontally and vertically with edits to the pad, and the ability to use a template for the side slopes, so they can project complex grading designs.



The screenshot shows a software interface for grading. At the top, there is a checkbox labeled "Use Slope Groups" with a "Set" button to its right. Below this are several input fields: "Fill Slope:" with a value of "2.00", "Cut Slope:" with a value of "2.00", and "Pad Elevation:" with a value of "480.8799". Underneath these are labels for "Pad Volume:", "Fill: 0.0 C.Y.", and "Cut: 61169.9 C.Y.". The "Surface:" is set to "EG" with a "Set" button. At the bottom, there is a grid of buttons: "Move Pad", "Move Vertex", "Offset", "Rotate Pad", "Balance", "Delete", "Report", "Exit", "Help", and "Process".

LandXML Data Transfer

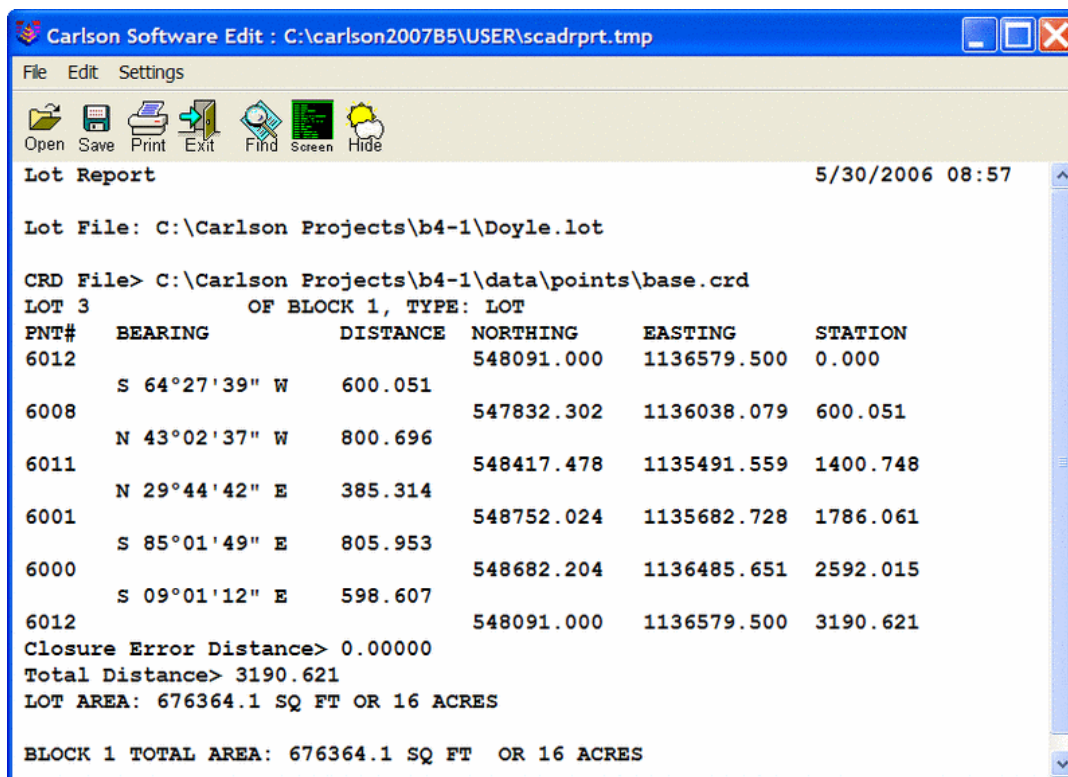
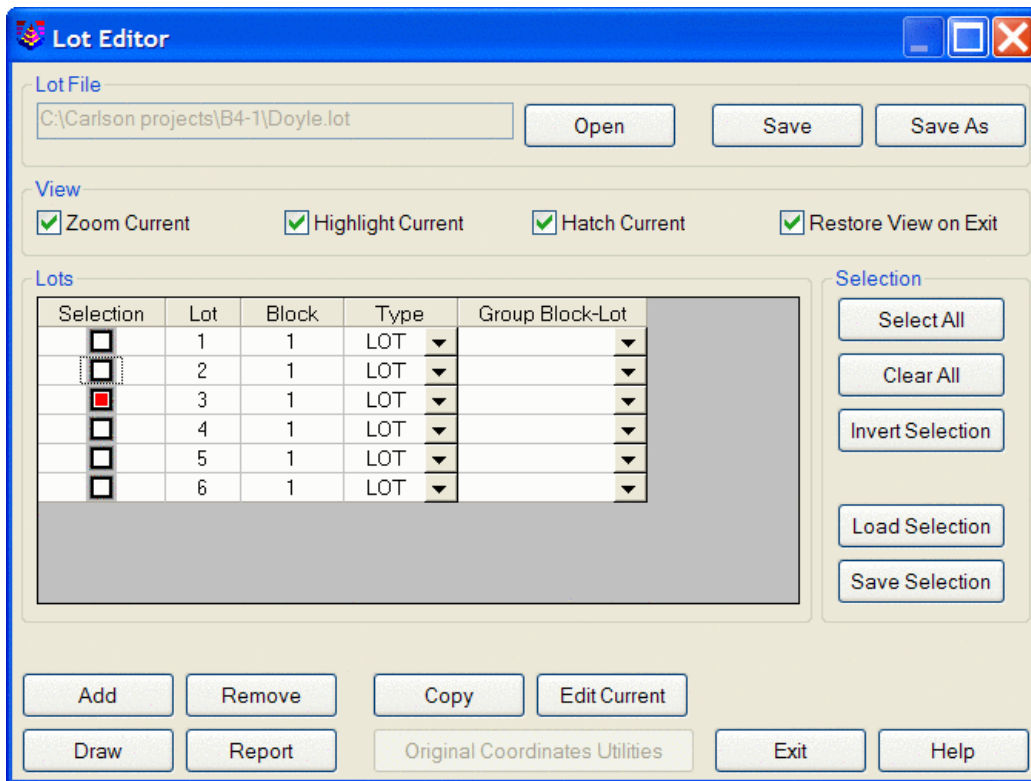
Transfer of data between LDT and Carlson Civil can be accomplished in several ways, depending on the type of data in question, but the best overall method is the use of LandXML files.

Pipes

Pipe Profiles can be designed and edited in the Profile Editor dialog, and then drafted in the drawing. In Hydrology, Pipe Networks can be created, which are dynamic and "intelligent", and so automatically respond to design changes.

Lots

Carlson Civil includes a set of tools for lot layout and lot design. Defined lots can be stored in a lot file (.lot). Another way to define a set of Lots automatically is with the Lot by Enclosed Text command, which searches for closed areas with enclosed text, and creates lots out of them, using the enclosed text for the lot name/number. Defined Lots can be accessed and edited through the Lot Editor.



Text Height

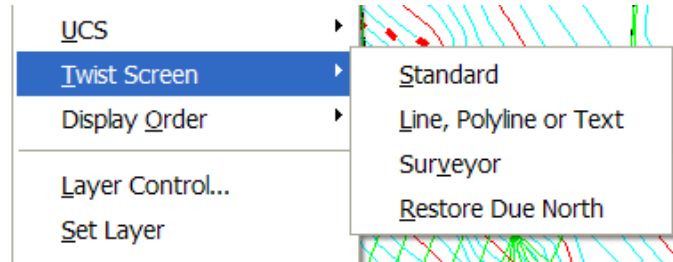
In LDT, as part of drawing setup, a set of text styles is created based on information contained in a file with a .STP file extension, most commonly used is the LEROY.STP. These text styles all have fixed heights assigned to them,

based on the current horizontal scale set in the drawing. If the horizontal scale is changed, the heights of the text styles are all changed.

In Carlson Civil, each of the various commands that involve annotation set the text style to be used and the desired height for the text, using a "scaler", which is multiplied by the current drawing scale. The AutoCAD text style should be set with a height of zero.

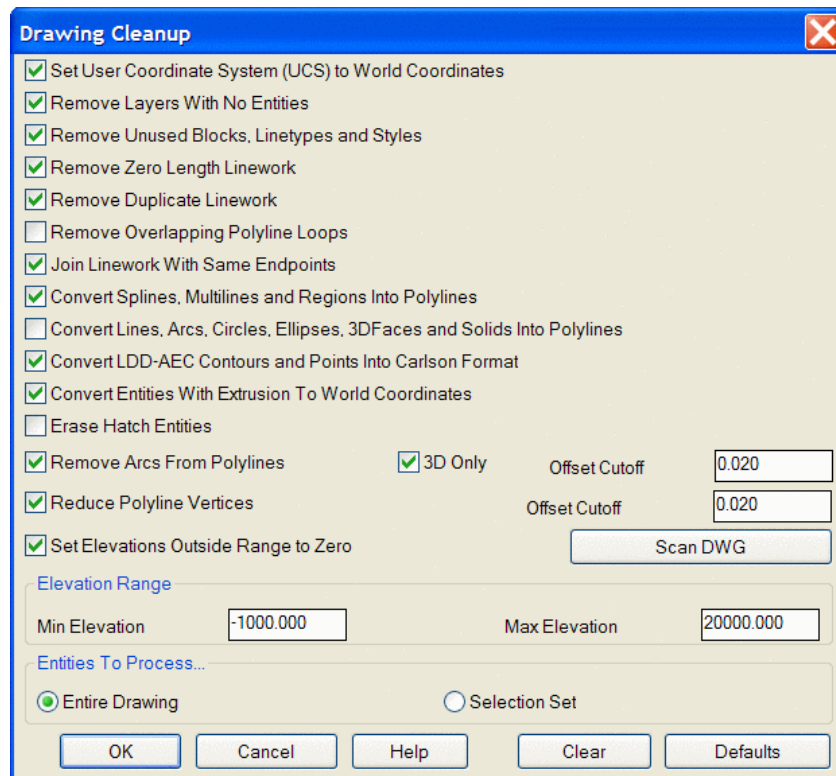
North Rotation

Carlson Civil does not support the concept of a secondary UCS to define and store North, as is done with LDT North Rotation. Instead, Carlson Civil relies on the use of DVIEW Twist to reorient North, and contains a thorough set of tools to work with that command. Any LDT drawings that are going to be brought into Carlson Civil should first have their LDT Base Point and North Rotation checked. If they are not all zeros, they should all be set to 0, have all points inserted to the drawing, and all linework moved and rotated to the location of the points.



Drawing Cleanup

If you're not running Carlson Civil on top of Map, or even if you are, Carlson Civil includes an awesome Drawing Cleanup function to find and resolve a wide range of common drawing problems.



Programming Interfaces 20

LSP Functions

Overview

Many of the Carlson functions and program environment settings are defined in LSP. These functions and variables are available for you to use in your own LSP routines.

The following is a list of useful functions:

scad_getfiled - file selection dialog

The following is a list of useful variables:

lspdir\$ - Carlson LSP folder where program files are located (string)

tmpdir\$ - current project data folder (string)

usrdir\$ - folder for program settings and temporary files (string)

psname - Carlson Support folder (string)

sv:sm - horizontal scale (real)

sv:vs - vertical scale (real)

sv:ts - text size scaler (real)

sv:ps - symbol size scaler (real)

is_metric - english/metric mode (0=english, 1=metric)

crdfile - current coordinate file (string)

scad_getfiled

Runs a file selection dialog. The function returns a string of the selected file name.

Usage: **(scad_getfiled title folder extension mode)**

title - A string for the dialog title

folder - A string for the initial default folder

extension - A string for the file type extension

mode - An integer for the selection mode (0 = existing file, 1 = new file)

Example: **(setq file_name (scad_getfiled 'Pick Surface To Process' tmpdir\$ 'tin' 0))**

centroid

Calculates the centroid point of a polyline. The function returns a point for the centroid. This function is within the poly3d program which must be loaded using scload.

Usage: **(cf:poly3d usrdir\$ 112 ename)**

ename - entity name of closed polyline

Example: **(scload (strcat lspdir\$ 'poly3d'))**
(setq pnt (cf:poly3d usrdir\$ 112 ename))

Coordinate API

Overview

We have created a Lisp file, CrdAPI.FAS, that contains some simple and easy-to-use functions for accessing, modifying, drawing and selecting any type of coordinate file that Carlson Software supports. The CrdAPI.FAS file is located in the Carlson LSP folder. This collection of functions can help you write your own tools. We call this toolkit the Coordinate File Application Programming Interface, or CRD API.

This document describes the Lisp interface of the CRD API.

Supported Types of Coordinate Files

The following enumerated variables are available for specifying the different types of coordinate files that are supported by the CRD API.

```
;;; Coordinate File format number
```

```
(setq
enCrdTypeUnrecognized -2 ;file exists but is not a recognized type.
enCrdInvalid -1 ;not a valid type - used in creating new files.
enCrdSurvCADDNumeric 0 ;Carlson original; point name is numeric. Extension
'crd'.
enCrdSurvCADDAlpha 1 ;Carlson sequel; point names can be alphanumeric.
Extension 'crd'.
enCrdMemory 2 ;NOT YET IMPLEMENTED.
enCrdCgAlpha 3 ;C&G alphanumeric. Extension 'cgc'.
enCrdCgNumeric 4 ;C&G numeric. Extension 'crd'.
enCrdSimplicity 5 ;Simplicity 'Sight Survey'. Extension is 'zak'.
enCrdLDD 6 ;Land Desktop Development. Filename is 'points.mdb'.
enCrdCivil3D 7 ;Civil 3D; point name is numeric.
enCrdSQLite 8 ;Carlson SQLite. Extension 'crdb'.
)
```

Working with Points in Coordinate Files

The following functions are for working with Crd Point structures.

Internally, the Crd Point looks like the following list.

```
(name desc x y z)
```

To allow for expansions and changes of the Crd Point in the future, please use only the following Crd Point functions for setting and accessing Crd Points.

CrdPointInit

A Crd Point structure will be returned that can be passed into CrdAddPoint, etc. *name* and *desc* must both be strings.

```
(defun CrdPointInit( x y z name desc)
(list name desc x y z)
)
```

CrdPointSetName

Sets the name of the Crd Point structure. The Crd Point is returned.

```
(defun CrdPointSetName( point name)
  (cons name (cdr point))
)
```

CrdPointGetName

Gets the name of the Crd Point structure.

```
(defun CrdPointGetName( point)
  (car point)
)
```

CrdPointSetDesc

Sets the description of the Crd Point structure. The Crd Point is returned.

```
(defun CrdPointSetDesc( point desc)
  (cons (car point) (cons desc (caddr point))))
)
```

CrdPointGetDesc

Gets the description of the Crd Point structure.

```
(defun CrdPointGetDesc( point)
  (cadr point)
)
```

CrdPointSetXYZ

Sets the x, y, z values of the Crd Point structure given a list of (x y z). The Crd Point is returned.

```
(defun CrdPointSetXYZ( point xyz)
  (append (list (car point) (cadr point)) xyz)
)
```

CrdPointGetXYZ

Gets the x, y, z values of the Crd Point structure as a list of (x y z).

```
(defun CrdPointGetXYZ( point)
  (caddr point)
)
```

Working with the Coordinate File

The following functions are for working with coordinate files.

CrdOpen

Opens a coordinate file with the given filename. Only one coordinate file can be open at a time through the Lisp interface. If CrdOpen is called twice, the first crd file is closed automatically.

filename – The file name of the coordinate file with required extension (.crd, .cgc, .mdb).

bOverwrite – If 1 (integer), then any existing file of the same name will be destroyed. Use nil to select default value which is 0.

crdType – See enCrdType above. This specifies the type of CRD file to create if the given name does not already exist. Use nil to select the default value which is enCrdSurvCADDAlpha.

Returns 1 (integer) if successful and 0 if there was some kind of failure. Call CrdGetLastError() to get the text message of the error.

```
(defun CrdOpen( filename bOverwrite crdType)
```

```
(cf:crdapi 'CrdOpen' filename bOverwrite crdType)
)
```

CrdClose

Closes the current Crd file, if it is open.

```
(defun CrdClose()
(cf:crdapi 'CrdClose')
)
```

CrdAddPoint

Adds the given CrdPoint to the current open Crd File, overwriting any point with the same name that may already exist. The CrdPoint is a (list name desc x y z).

Returns 1 (integer) if successful and 0 otherwise. Call CrdGetLastError() to get text message of error.

```
(defun CrdAddPoint(CrdPoint)
(cf:crdapi 'CrdAddPoint' point)
)
```

CrdBegin

Starts a traversal of all the points of the Crd File by resetting internal state so that CrdNext will start from beginning of file.

range – if not nil, then it must be a Carlson-style string of point names suggested by the following examples: "1-5,6,12-19" or "all". If nil, that is the same as all. The points will be returned on successive calls to CrdNext() in the order listed in the range.

```
(defun CrdBegin( range)
(cf:crdapi 'CrdBegin' range)
)
```

CrdNext

Gets the next point in the Crd File. Returns a Crd Point if successful and nil if there are no more points or some other failure.

```
(defun CrdNext()
(cf:crdapi 'CrdNext')
)
```

CrdGetPoint

Gets the named point from the current open Crd File. A Crd Point is returned if successful and nil is returned if the point could not be found. Call CrdGetLastError() to get text message of error.

```
(defun CrdGetPoint( name)
(cf:crdapi 'CrdGetPoint' name)
)
```

CrdRemovePoint

Removes the named point from the current open Crd File. Returns 1 (integer) if successful and 0 otherwise. Call CrdGetLastError() to get text message of error.

```
(defun CrdRemovePoint( name)
(cf:crdapi 'CrdRemovePoint' name)
)
```

CrdRemovePoints

Removes the given range of points (a string) from the current open Crd File. For example, the range could be "1-5,7,10-12".

```
(defun CrdRemovePoints( range)
(cf:crdapi 'CrdRemovePoints' range)
)
```

CrdTestPointName

Returns nil if the point name is valid for the current Crd type. Otherwise, returns an error string describing the error.

```
(defun CrdTestPointName( name)
(cf:crdapi 'CrdTestPointName' name)
)
```

CrdTestPoint

Returns nil if the Crd Point is valid for the current Crd type. Otherwise, returns an error string describing the error.

```
(defun CrdTestPoint( point)
(cf:crdapi 'CrdTestPoint' point)
)
```

CrdGetLastError

Returns the error string of the result of the last operation.

```
(defun CrdGetLastError()
(cf:crdapi 'CrdGetLastError')
)
```

CrdDeleteFile

Deletes the data file(s) underlying the Crd File and closes the Crd File. May not work if the original filename was not a full path including drive letter. Returns 1 (integer) if successful and 0 otherwise. Call CrdGetLastError() to get text message of error.

```
(defun CrdDeleteFile()
(cf:crdapi 'CrdDeleteFile')
)
```

GetCrdType

Returns the crd type of the current open Crd File.

```
(defun GetCrdType()
(cf:crdapi 'GetCrdType')
)
```

GetCrdTypeName

Returns the official name of the given crd type.

```
(defun GetCrdTypeName( enCRDType)
(cf:crdapi 'GetCrdTypeName' enCRDType)
)
```

CrdDraw

Accepts an argument list to indicate a set of points to be drawn, which at a minimum takes the form ' ('Points' '<points designator>')'.

```
(defun CrdDraw(arglist)
(eval (append '(cf:crdapi 'CrdDraw') arglist)))
)
```

The syntax for additional optional parameters which may be included in arglist (indicated by "[]") is as follows:

```
(CrdDraw ('Points' '<points designator>'
['SymbolName' '<symbol block name>']
['SymbolXScale' <X scale>]
['SymbolYScale' <Y scale>]
['SymbolRotation' '<symbol rotation in AUNITS>']
['SymbolLayer' '<symbol layer name>']
```

```

['BlockName' ' '<attribute block name>']
['BlockXScale' <X scale>]
['BlockYScale' <Y scale>]
['BlockRotation' ' '<attribute block rotation in AUNITS>']
['BlockNameLayer' ' '<layer for point name>']
['BlockElevationLayer' ' '<layer for point elevation>']
['BlockDescriptionLayer' ' '<layer for point description>']
['PointLayer' ' '<layer for POINT>'])
)
)

```

CrdSelect

Accepts an argument list to indicate a set of points to be selected, which at a minimum takes the form `nil` to select all points.

```

(defun CrdSelect (arglist)
  (eval (append '(cf:crdapi 'CrdSelect') arglist))
)

```

The syntax for additional optional parameters which may be included in `arglist` (indicated by "[]") is as follows:

```

(CrdSelect '( ; Select all points in drawing
['Points' ' '<points designator>'] ; Select all points in <point
designator>
['Prompt' ' '<prompt for points>'] ; Interactive (spatial) select points
in drawing
['P0' point 'P1' point] ; Select all points in window having corners
P0,P1
)
)

```

Note that all optional parameters within "[]" are mutually exclusive. The function returns the selected points as a list of points, each of which is itself a sublist indicating the point identifier and its associated drawing entity name.

Example Code

```

;;; Examples on how to use.
(CrdOpen 'junk2.crd' nil nil)
(princ (strcat 'Created a file of type: ' (GetCrdTypeName (GetCrdType))
'\n'))

; create a couple of points in memory.
; This first point is at coordinate 100,101,10.5 and has the point name
; of 1012 and the description of IP.
(setq point1 (CrdPointInit 100 101 10.5 '1012' 'IP'))
(setq point2 point1)
(setq point2 (CrdPointSetXYZ point2 '(105 101 10.5)))
(setq point2 (CrdPointSetName point2 '3'))
(setq point3 (CrdPointSetName point2 '7'))

; adding a point and checking for errors.
(setq bSuccess (CrdAddPoint point1))
(if (= bSuccess 0)
  (princ (strcat 'Error on point ' (CrdPointGetName point1) ': '

```

```

(CrdGetLastError))) '.\n')

; adding points without checking for errors.
(CrdAddPoint point2)
(CrdAddPoint point3)

; now print out the points 1 through 5
(princ 'Listing the point names.\n')
(CrdBegin '1-5') ; nil or 'all' would select all the points.
(while (setq point (CrdNext))
(princ (strcat (CrdPointGetName point) ' ' (CrdPointGetDesc point) '\n'))
)
;
; Drawing points.
; Draw all points in the drawing using defaults. Argument list ('Points'
'<points designator>') is required as a minimum.
(CrdDraw ('Points' 'ALL'))
;
; Syntax for additional optional parameters indicated by '[' is as
follows:
; (CrdDraw ('Points' '<points designator>'
; ['SymbolName' '<symbol block name>']
; ['SymbolXScale' <X scale>]
; ['SymbolYScale' <Y scale>]
; ['SymbolRotation' '<symbol rotation in AUNITS>']
; ['SymbolLayer' '<symbol layer name>']
; ['BlockName' '<attribute block name>']
; ['BlockXScale' <X scale>]
; ['BlockYScale' <Y scale>]
; ['BlockRotation' '<attribute block rotation in AUNITS>']
; ['BlockNameLayer' '<layer for point name>']
; ['BlockElevationLayer' '<layer for point elevation>']
; ['BlockDescriptionLayer' '<layer for point description>']
; ['PointLayer' '<layer for POINT>']))
;
; Selecting points.
; Select all points in the drawing
(CrdSelect nil)
;
; Select all points in window
(CrdSelect ('P0' (setq p0 (getpoint 'First Corner: ')) 'P1'
(getcorner p0 'Opposite corner: ')))
;
; Syntax for additional optional parameters indicated by '[' is as
follows.
; (CrdSelect (['Points' '<points designator>'] ; Select all points in
<point designator>
['Prompt' '<prompt for points>'] ; Interactive (spatial) select points
in drawing
['P0' point 'P1' point] ; Select all points in window
; All parameters within [] are optional and are mutually exclusive.
;
;

```



```
(CrdDeleteFile) ;the only proper way to delete the file(s). C&G uses two files, for example.
```

```
(CrdClose) ;not necessary because of CrdDeleteFile, but doesn't hurt.
```

```
(princ '--the end--\n')
```

DTM API

Overview

A lot of functionality of Carlson triangulation and TIN file manipulation is now available from LISP for the advanced users to use in their routines. This part of the Carlson interface is called DTM API.

API supports the older FLT file format, which stores only edge information, and the new TIN format, which is a binary format containing all the structure of triangulation and therefore is faster to load and takes less space. The file extension controls which file type is being created.

The functionality of DTM API is implemented in TRI4.ARX. It would be the responsibility of the caller to make sure that this file gets loaded by adding this line at the beginning:

```
(scload (strcat lspdir$ 'tri4'))
```

The ARX should never be unloaded by the caller.

The following is a list of currently supported functions (it can be obtained at any time by calling **(cf:dtm.api)**):

create_tin - create a TIN from selection set

draw_tin - draws a TIN as 3d faces;

tri_volume - produce volume report from 1 or 2 TINs

tri_change - modify tin using inc/exc and operators

tri_contour - contour tin using inc/exc and ini file with settings

tri_profile - creates a profile of the TIN along a centerline

tri_diff - produce a difference TIN from two TINs

tin_combine - produce a combined TIN performing a specified operation to elevation

tri_store_regions - create and store TIN difference

tri_apply_region - apply a specific region revision

tri_forget_region - forget all region revisions

surface_util - routines for tin surface manager

load_tin - loads TIN into memory for tin.z function

unload_tin - unloads the current TIN from memory

tin_z - returns the z at the specified x,y for the current TIN set by load_tin

create_tin

Creates TIN from a given selection set, optionally using inside/outside logic.

Usage: **(cf:dtm.api 'create_tin' entities_ss inclusion_ss exclusion_ss regions_ss file_name options)**

entities_ss - Selection set of all entities to be used for triangulation. Currently supported are point, line, arc, polyline, insert, circle, 3dface, solids, text, and mtext.

inclusion_ss - Selection set containing inclusion polylines if needed. Pass nil if inclusion polylines are not being used.

exclusion_ss - Selection set containing exclusion polylines if needed. Pass nil if exclusion polylines are not being

used.

regions_ss - Selection set containing region polylines if needed. Pass nil if regions are not being used. Regions logic is on/off - crossing region line reverses the inside/outside state. Note: Only one of inside/outside methods can be used for one call (either inclusion/exclusion or regions).

file_name - Full path of file to be created. Extension controls a type of TIN file created.

options - Optional. String containing one or more of the following keywords, space separated:

ignore_zero - Ignore zero elevation data points.

view_error_log - In case of warnings during triangulation, bring up the report at the end.

densify - Perform ridge/valley detection and improve triangulation as needed.

draw_tin

Draws TIN file at a given layer as 3DFACES.

Usage: `(cf:dtm_api 'draw_tin' file_name layer_name is_road)`

file_name - Full path of file to be loaded.

layer_name - Layer name to use. If does not yet exist, new layer will be created. Color of entities is set to BYLAYER.

is_road - Optional. If set to 1 and route follows completion of Process Road Design feature it will turn on road coloring.

tri_volume

Calculates volume of the TIN within a given polyline, optionally with report generated.

Usage: `(cf:dtm_api 'tri_volume' inclusion_ss exclusion_ss regions_ss filename filename2|elevation is_report is_silent)`

Return values: If successful, the function returns a list with 4 real values: cut, fill, cut area, and fill area. The volume is in cubic ft or meters and area is in square ft or meters, depending on the configuration.

inclusion_ss - Selection set containing inclusion polylines if needed. Pass nil if inclusion polylines are not being used.

exclusion_ss - Selection set containing exclusion polylines if needed. Pass nil if exclusion polylines are not being used.

regions_ss - Selection set containing region polylines if needed. Pass nil if regions are not being used. Regions logic is on/off - crossing region line reverses the inside/outside state. Note: Only one of inside/outside methods can be used for one call (either inclusion/exclusion or regions).

filename - Full path of main TIN file to be used.

filename2 - Full path of secondary TIN file to be used. If used, two surface volumes will be calculated between the first and second TINs.

elevation - Reference elevation passed as real value. If this option is used, one surface volume will be calculated between the first TIN and this reference elevation.

is_report - 0/1. Specifies whether to bring up the report with calculation results.

is_silent - 0/1. If 0 value is supplied the routine will provide no output at the command line and no progress indicators.

tri_change

Change TIN file using specified optional inclusion polylines and using one of the possible math operations. For all

operations but "embed" the inclusion/exclusion polylines are offset a 0.1 so that a reasonably sharp wall could be produced by modifications.

Usage: **(cf:dtm_api 'tri_change' inclusion_ss exclusion_ss regions_ss filename operation value)**

inclusion_ss - Selection set containing inclusion polylines if needed. Pass nil if inclusion polylines are not being used.

exclusion_ss - Selection set containing exclusion polylines if needed. Pass nil if exclusion polylines are not being used.

regions_ss - Selection set containing region polylines if needed. Pass nil if regions are not being used. Regions logic is on/off - crossing region line reverses the inside/outside state. Note: Only one of inside/outside methods can be used for one call (either inclusion/exclusion or regions).

filename - Full path of main TIN file to be modified.

operation - keywords describing what operation is to be performed on the TIN:

add - Add a value to elevation of inside nodes.

scale - Scale elevation of inside nodes by the value.

set - Set elevation of inside nodes to the value specified.

perp - Sets elevation of nodes as if TIN is offset by value along normal at the point. The node is not moved horizontally, just elevation is adjusted.

nil - Removes inside nodes. No value is needed.

embed - No change of elevations, but the inclusion polyline is still added to TIN.

tri_contour

Contours TIN file as defined by INI file and inclusion polylines

Usage: **(cf:dtm_api 'tri_contour' inclusion_ss exclusion_ss regions_ss tin_filename ini_filename)**

inclusion_ss - Selection set containing inclusion polylines if needed. Pass nil if inclusion polylines are not being used.

exclusion_ss - Selection set containing exclusion polylines if needed. Pass nil if exclusion polylines are not being used.

regions_ss - Selection set containing region polylines if needed. Pass nil if regions are not being used. Regions logic is on/off - crossing region line reverses the inside/outside state. Note: Only one of inside/outside methods can be used for one call (either inclusion/exclusion or regions).

tin_filename - Full path of main TIN file to be used.

ini_filename - Full path of INI file defining how contouring is performed. For list of values, check tri4.ini in USER folder which stores values used in Triangulate and Contour function.

tri_profile

Creates a profile of the TIN along a centerline

Usage: **(cf:dtm_api 'tri_profile' tin_filename cl_filename pro_filename)**

Return values: If successful, the function returns 1, otherwise nil.

tin_filename - Full path of main TIN file to be used.

cl_filename - Full path of centerline (.cl) file to be used.

pro_filename - Full path of the profile file to be created.

tri_diff

Calculates difference between two TIN files and creates combined TIN with elevation being the difference of elevations. TINs do not have to match perfectly - face intersections will be performed. Inclusion logic will be applied if needed.

Usage: `(cf:dtm_api 'tri_diff' inclusion_ss exclusion_ss regions_ss filename1 filename2 diff_filename)`

inclusion_ss - Selection set containing inclusion polylines if needed. Pass nil if inclusion polylines are not being used.

exclusion_ss - Selection set containing exclusion polylines if needed. Pass nil if exclusion polylines are not being used.

regions_ss - Selection set containing region polylines if needed. Pass nil if regions are not being used. Regions logic is on/off - crossing region line reverses the inside/outside state. Note: Only one of inside/outside methods can be used for one call (either inclusion/exclusion or regions).

filename1, filename2 - Full path to TIN files to be used. Second TIN is subtracted from first one.

diff_filename - Full path of differential TIN file to be created.

tin_combine

Combines two TINs into one while applying one of the specified operations. First two TINs are combined into one intersecting all faces which need to be intersected and then elevations are assigned as specified.

Usage: `(cf:dtm_api 'tin_combine' inclusion_ss exclusion_ss regions_ss filename1 filename2 merged_filename operation)`

inclusion_ss - Selection set containing inclusion polylines if needed. Pass nil if inclusion polylines are not being used.

exclusion_ss - Selection set containing exclusion polylines if needed. Pass nil if exclusion polylines are not being used.

regions_ss - Selection set containing region polylines if needed. Pass nil if regions are not being used. Regions logic is on/off - crossing region line reverses the inside/outside state. Note: Only one of inside/outside methods can be used for one call (either inclusion/exclusion or regions).

filename1, filename2 - Full path to TIN files to be used.

merged_filename - Full path to the file where results are stored.

operation - Keywords describing the operation to perform. The following operations are supported:

keep - The elevations of first TIN are used.

copy - Elevations of second TIN are used.

min - Lesser of two elevations is used.

max - Greater of two elevations is used.

join - Points inside are given elevations of first TIN, outside - second one.

tri_store_regions

Evaluates "before" and "after" TIN and stores "affected" area as a region with two revisions "0" and "1". The regions then can be applied as needed using other commands. This functionality only works with binary .TIN files.

Usage: `(cf:dtm_api 'tri_store_regions' filename1 filename2 mark_name filename3)`

filename1, filename2 - Full path to TIN files to be used.

mark_name - String label for region to be used.

filename3 - Full path to TIN files to be Save As. If omitted, the file specified by filename1 will be replaced with resulting file.

tri_apply_region

Applies a specific pre-stored region revision, so that when TIN file is used it has that revision of the region in use.

Usage: **(cf:dtm_api 'tri_apply_region' filename mark_name revision)**

filename1 - Full path to TIN file to be modified.

mark_name - String label for region to be used.

revision - Integer. Optional ID of revision to apply. If omitted, the latest revision of the region is used.

tri_forget_region

Removes any knowledge of a specific pre-stored region revision from TIN file. Optionally remove triangles found in the region from the TIN.

Usage: **(cf:dtm_api 'tri_forget_region' filename mark_name remove_first)**

filename - Full path to TIN file to be modified.

mark_name - String label for region to be used.

remove_first - 0/1. Optional flag indicating whether region triangles should be removed, leaving a hole in the TIN.

surface_util

Utilities for dealing with surfaces stored in the drawing.

Usage: **(cf:dtm_api 'surface_util' filename operation)**

filename - Full path to TIN file to be modified.

operation - Keywords describing the operation to perform. The following operations are supported:

check - Check if TIN is used in one of the surfaces. Returns 0 or 1 depending on outcome.

process - Apply current surface parameters to TIN file. This function should be used to update surface in the drawing after TIN has been modified.

load_tin

Loads a TIN into memory for use with the tin_z function.

Usage: **(cf:dtm_api 'load_tin' file_name)**

file_name - Full path of file to be loaded.

unload_tin

Unloads the current TIN from memory.

Usage: `(cf:dtm_api 'unload_tin')`

tin_z

Calculates the elevation of the TIN at the specified x,y location. The TIN to process must be already loaded by the `load_tin` command. The calculated elevation is returned to LSP on success. A nil is returned to LSP if the command arguments are invalid, a TIN is not loaded or the point is off the surface.

Usage: `(setq elev (cf:dtm_api 'tin_z' point))`

point - List of doubles in x,y format

Examples:

Calculate TIN Elevation at Point

```
(scload (strcat lspdir$ "tri4"))
(setq file_name (scad_getfiled "Select TIN to Process" "" "tin" 0))
(cf:dtm_api "load_tin" file_name)
(setq pnt (getpoint "\nPick point:"))
(setq elev (cf:dtm_api "tin_z" pnt))
(cf:dtm_api "unload_tin")
```

Road API

Overview

The Road API is a collection of functions for centerline location and road design calculations. These functions are available from LISP for advanced users to use in their own routines.

The functionality of Road API is implemented in the Carlson EWORKS.ARX program file. This ARX program file must be loaded before calling the Road API functions. Use the following LSP code to load the ARX:

```
(scload (strcat lspdir$ 'eworks'))
```

The following is a list of currently supported functions (it can be obtained at any time by calling `(cf:road_api)`):

cl_location_at_pt - calculates the station-offset and centerline position from a specified centerline and point

cl_location_at_sta - calculates the point and vectors for a specified station on a centerline

cl_sta_range - returns the start and end stations for the specified centerline

profile_z - calculates the profile elevation for the specified station on the profile

profile_sta_range - returns the start and end station for the specified profile

road_z - calculates the road design elevation for the specified point and design files

cl_location_at_pt

Given a centerline file name and a point, this function calculates the station-offset and the projected location of the

point along the centerline. On success, the function returns a list of the station, offset and projected centerline point. On failure, the function returns nil.

Usage: `(cf:road_api 'cl_location_at_pt' file_name point is_rr)`

file_name - Full path of the centerline file to process (string).

point - Coordinates (x,y) of the point to process (list of reals).

is_rr - Optional integer parameter for station type (0 = roadway stations (default), 1 = railroad stations)

cl_location_at_sta

Given a centerline file name and a station, this function calculates the point on the centerline at the specified station along with the vectors for the tangent and normal of the centerline at that station. On success, the function returns a list of the point on centerline, the tangent vector and the normal vector. On failure, the function returns nil.

Usage: `(cf:road_api 'cl_location_at_sta' file_name station is_rr)`

file_name - Full path of the centerline file to process (string).

station - Target station to process (real).

is_rr - Optional integer parameter for station type (0 = roadway stations (default), 1 = railroad stations)

cl_sta_range

This function simply return the starting and ending stations for the specified centerline. On success, the function returns a list of the starting station and the ending station as reals. On failure, the function returns nil.

Usage: `(cf:road_api 'cl_sta_range' file_name is_rr)`

file_name - Full path of the centerline file to process (string).

is_rr - Optional integer parameter for station type (0 = roadway stations (default), 1 = railroad stations)

profile_z

Given a profile file name and a station, this function calculates the elevation on the profile at the specified station. On success, the function returns the elevation. On failure, the function returns nil.

Usage: `(cf:road_api 'profile_z' file_name station)`

file_name - Full path of the profile file to process (string).

station - Target station on profile (real).

profile_sta_range

This function simply returns the starting and ending stations for the specified profile. On success, the function returns a list of the starting and ending stations as reals. On failure, the function returns nil.

Usage: `(cf:road_api 'profile_sta_range' file_name)`

file_name - Full path of the profile file to process (string).

road_z

This function calculates elevation of a road design at the specified point. The road design is defined by the design files names. The centerline, profile and template files are required. The other design files are optional. On success, the function returns the elevation. On failure, the function returns nil.

Usage: **(cf:road_api 'road_z' point cl_file pro_file tpl_file sup_file tpt_file tpc_file tpp_file tgt_file)**

point - Coordinates (x,y) of the point to process (list of reals).

cl_file - Full path of the centerline file (.cl) (string).

pro_file - Full path of the profile file (.pro) (string).

tpl_file - Full path of the template file or template series file (.tpl or .tsf) (string).

sup_file - Full path of the superelevation file (.sup) (string).

tpt_file - Full path of the template transition file (.tpt) (string).

tpc_file - Full path of the template point centerline file (.tpc) (string).

tpp_file - Full path of the template point profile file (.tpp) (string).

tgt_file - Full path of the template grade table file (.tgt) (string).

3D Viewer API

Overview

It is now possible to call 3d viewer from LISP for the advanced users to use in their routines.

The functionality of 3D Viewer API is implemented in CUBE.ARX. It would be the responsibility of the caller to make sure that this file gets loaded by adding this line at the beginning:

```
(scload (strcat lspdir$ 'cube'))
```

Displaying a selection set of entities

Prompts the user for a selection set and displays it in a 3d viewer window

Usage: **(cf:sc3dview)**

Loading a surface file into 3D viewer

Loads the surface given as grid (.GRD) or triangulation (.FLT or .TIN) file.

Usage: **(cf:sc3dview file_name)**

Lat/Lon and Grid Projections API

Overview

To use these functions, the Coordinate System must be set under Settings >Drawing Setup.

Drawing X/Y to Lat/Lon

Converts a drawing coordinate to latitude and longitude.

Usage: (setq result (cf:lablat 8 x_val y_val is_dms))

x_val - Easting

y_val - Northing

is_dms - Optional parameter to control format of result (0-decimal degrees (default), 1=dd.mmss)

result - Function output containing list of latitude and longitude

Lat/Lon to Drawing X/Y

Converts latitude and longitude to a drawing coordinate.

Usage: (setq result (cf:lablat 9 lon_val latitude_val is_dms))

lon_val - Longitude

lat_val - Latitude

is_dms - Optional parameter to define format of input (0-decimal degrees (default), 1=dd.mmss)

result - Function output containing list of easting and northing

plus-0.5in
2 Point - 2 Point Intersect, 874
2 Tangents, Arc Length, 232
2 Tangents, Chord Length, 232
2 Tangents, Degree of Curve, 234
2 Tangents, External, 233
2 Tangents, Mid-Ordinate, 233
2 Tangents, Radius, 232, 854
2 Tangents, Tangent Length, 234
2 Tangents, Through Point, 234
2-Radius Curve Series, 238
2D Align, 168, 2857, 4670
2D Line, 219, 220, 399, 694, 772
2D Linework Library, 355
2D Polyline, 221, 223, 224, 226, 270, 352, 657, 691,
718, 725, 735, 769, 796, 803, 942, 943,
1707, 1710, 1716, 2079, 2454, 2843, 4290–
4292, 4294–4298, 4326, 4344, 4352, 4402,
4432, 4434, 4454, 4488, 4654
2D Scale, 157
2D to 3D By Surface Model, 2644
2D to 3D By Text With Leader, 2647
2D to 3D Polyline by Points, 1429, 2645
2D to 3D Polyline by Profile, 1434
2D to 3D Polyline by Screen Entities, 1428
2D to 3D Polyline by Slope From Reference, 1430
2D to 3D Polyline by Start-End Elevations, 2649
2D to 3D Polyline by Start/End Elevations, 1434
2D to 3D Polyline by Surface Model, 1427
2D to 3D Polyline by Text, 1431
2D to 3D Polyline by Text With Leader, 1432
2D to 3D Polyline-By Text, 2646
2D-1D Local Coordinate System, 623
2D-1D State Plane Coordinate System, 630
3 D Points - Interpolate Entity, 1458
3 Point, 230, 240, 241, 674, 675, 752, 753, 904, 3113,
4446
3-Radius Curve Series, 238
3D Arc, 237, 661, 739
3D Data Menu, 1418
3D Dragline, 4111
3D Drive Simulation, 2690, 2691, 2709, 4621
3D Entity By Surface Model, 1437
3D Entity to 2D, 193, 1420
3D GeoFluv Contour Viewer, 3185
3D GeoFluv Surface Viewer, 3187
3D Grade Simulation, 4174
3D Line, 220

3D Polyline, 27, 189–191, 225–229, 270, 657, 675, 687, 692, 718, 722, 725, 735, 753, 765, 770, 796, 800, 803, 836, 907, 1420, 1426, 1428–1430, 1432–1435, 1437, 1462–1464, 1466, 1469, 1472, 1473, 1475, 1477, 1482, 1483, 1486–1488, 1491, 1496, 1499, 1500, 1519, 1562, 1624, 1706, 1707, 1710, 1711, 1714, 1716, 1726, 1824, 1864, 1872, 1873, 1998, 2051, 2337, 2413, 2455, 2477, 2510, 2638, 2644, 2734, 2947, 2951, 2952, 2955, 3505, 3765, 3785, 3793, 3835, 3850, 4336, 4406, 4407, 4419, 4442, 4450, 4452, 4466, 4619, 4655, 4656

3D Polyline Flow Values, 2164

3D Polyline by Slope on Surface, 1465

3D Polyline, 190, 191

3D Viewer 3DX Model File, 206

3D Viewer API, 4766

3D Viewer Window, 196, 205, 206, 210, 307, 308, 1645, 1646, 1648, 1662, 1663, 2068, 2117, 2118, 2122, 2127, 2334, 2728, 2779, 2788, 2857, 2867, 3203, 3257, 3472, 3480, 3513, 3617, 3790, 4582, 4644

3D Viewer to 3D PDF, 2867

3D Viewer/Model Library, 356, 1653

3DSolid and Polymesh Converter, 129

4 Sided Building, 852, 4296, 4327

AASHTO, 1686, 1723, 1741, 1979

About Carlson, 450

Action Tab, 3073, 3078, 3085–3087, 3101, 3107, 3111, 3113, 3114, 3116, 3118, 3120, 3121, 3123–3125, 3134

Add/Replace Layer Prefix, 211

Add Core Hole, 3631

Add Culvert to Polyline, 1118

Add Grid Lines, 1814

Add Grid Ticks and Dots, 1813

Add Hydrographs, 2215

Add Intersection Points, 174

Add Link, 2579

Add Pin Point, 3735, 3928

Add Pinpoint To Solid, 3797

Add Point by Two Slopes, 190

Add Points At Elevation, 190, 1473

Add Points To Solid, 3831

Add Polyline Arcs, 176

Add Polyline Vertex, 175

Add Prefix/Suffix To Text, 164

Add Strata, 3294, 4122
Add Zig to Polyline, 1118
Adjoiner Text, 1076, 1078
Adjust Color, 3026, 3042
Adjust Draw Profile Settings, 1815, 1816
Adjust Elevation Labels, 1456
Adjust Overexcavate Surface, 2681
Adjust Plan/Profile Sheet, 1816
Adjust Profile Grid, 1814
A Dozen Tools for Surface Design, 4428, 4464
Advanced Projections, 3644, 3713, 3740, 3917, 3920,
3937
Advanced Weir Design, 2262, 2264, 2265
Align by Two Pairs of Points, 413
Align GPS To Local Coordinates, 2911
Alignments, 4740, 4742
Align Paper With Model Space, 195
Align Points, 385, 388, 408
All topo items, 1345
Alphanumeric, 308, 369, 565, 610, 1190, 3225, 3452
Angle/Distance, 405, 828, 1034, 1043, 1063, 4306,
4711
Angle Balance, 492, 510, 516, 518, 519, 827
Angle Info, 285, 286
Angle Mode, 288, 307, 311, 811, 862, 1108, 1163
Angles Right, 1265
Annotate Menu, 1043
Annotation, 34, 42, 263, 264, 288, 307, 809, 1003,
1060, 1067, 1105, 1126, 1353, 1484, 1751,
2332, 2588, 4731, 4739
Annotation Defaults, 1045, 1053, 1055, 1076, 1100,
1108
Annotative Text, 42
Apache Lightbar, 2956
Append Another Raw File, 530
Append File, 3629, 3939
Apply Faults to Grid, 3383
Apply New Definition, 3285, 3286
Apply Wash, 3398, 3402
Arc Dimensions, 1114
Arc From Last Point, 236
ArcGIS Feature/Image/Map/Services, 2568
ArcInfo, 3693, 4004
Arc Length, 224, 229, 232, 235, 1051, 1402, 1403,
1686, 4331
Arcview, 2572
Area/Layout Menu, 913, 1010
Area by Closed Polylines, 921

Area by Interior Point, 921, 4352
Area by Lines & Arcs, 920, 921
Area Defaults, 914, 917, 919–921, 924, 927, 1003,
1033, 1034, 4254–4256, 4266–4268, 4278–
4280, 4316, 4332, 4358, 4365
Area Descriptions By Interior Text, 932
Area Label Defaults, 856, 923
Area Radial from Curve, 934
Areas Of Interest, 2667, 2719
Area Summary, 1258
Area Table Defaults, 924, 1003
Area To Section Report, 4138
Arrowhead, 26, 155, 156, 166, 257, 424, 429, 1479
Arrowhead With Line, 257
ASCII File, 379, 381, 383, 1136, 1169, 1170, 1174,
2578, 3626–3628, 3939
ASE, 56, 64, 132, 1306, 1571, 1583, 1618, 1619,
1621, 2707, 2708, 2715, 2902, 2965–2969,
3237, 3537, 3550
Assign Bed Names, 3297, 3558
Assign Building Type, 1026
Assign Contour Elevation - From Contour Labels,
2656
Assign Contour Elevation - Multiple in Series, 2655
Assign Contour Elevation - Single Elevation Group,
2657
Assign Contour Elevations: From Contour Labels,
1436
Assign Contour Elevations: Multiple in Series, 1435
Assign Contour Elevations: Single Elevation Group,
1437
Assign Directions, 29, 3970, 3971, 3976, 4052, 4182,
4183, 4186, 4216
Assign Panel Attributes, 3711
Assign Pipe Width to Polyline, 1850, 1939
Assign Pit Attributes, 3982
Assign Pit Names By Layer, 3951, 4051
Assign Pit Precedence, 3977, 3979
Assign Property Names, 3417, 3685, 3996, 4011
Assign Solid Data To Polyline, 3796
Assign Spoil Names, 4176
Assign Strata Correlation, 3287
Assign Template Point Centerline, 1994, 2035, 2048
Assign Template Point Profile, 1992, 2035
Assign Timing Grids, 3417, 3983, 3988, 3989, 4011,
4051
Attach Image to Entity, 2611
Attribute Definitions, 1039, 3206, 3306, 3335, 3694,

Attribute Grid from Block Model, 3487
Attribute Layers, 687, 691, 765, 769
Attribute Layout ID, 371, 420, 435, 691, 769, 4419
Attribute Validation Report, 3319
Audit Database File, 3235
Audit Image Links, 2613
Audit Links, 2545
Auto-Connect Pillars, 3664, 3665
Auto Annotate, 174, 732, 1003, 1051–1053, 1055,
1057, 1059, 1060, 1062, 1064, 1094, 4330
AutoCAD Map LPN links, 2575, 2576
Auto Create Points, 1321, 1322
Auto Lines, 1369
Auto Longitudinal Profile, 3195, 3196
Auto Map, 1300, 1310
Automatic Point Numbers, 371, 3933
AutoMine Connections, 3627, 3628, 3661, 3941
Auto Panel Layout, 3717, 3739
Auto Place Panel by Text, 3719, 3759, 3924
Auto Point Number, 1368
Auto Point Plot, 1369
Auto Points at Interval, 2926, 2959
AutoRun Residuals, 3433
AutoRun Strata Quantities, 3367
Auto Tablet On, 309, 1557
Average Profiles, 1849
Average Section Files, 1947
Average Slope, 1675, 2508
Azimuth-Distance with Leader, 1073

Backfill, 2785, 2789
Backsight, 311, 492, 493, 507, 511, 526, 527, 529,
532–534, 543, 546, 548, 551, 654, 655, 858,
859, 862, 863, 866, 912, 913, 1152, 1184,
1195, 1197, 1355, 1381, 1392, 1397, 2885,
2886, 2921, 2922, 4366
Bare Earth, 3076–3078, 3113
Bar Graph, 3260
Barscale, 1085, 4252, 4264, 4276
Basic Projections, 3642, 3935
Basic Road Design with Volumes, 4488, 4514
Bearing-Bearing Intersect, 869, 870, 874, 1234
Bearing-Distance Intersect, 871, 1235
Bearing-Distance with Leader, 1072
Bearing & 3D Distance, 279
Bearing Annotation, 1047, 4330
Bearing Area Cutoff, 936
Bearing with Leader, 1071

Bed Composite Report, 3315
Bed Names, 3215, 3224, 3240, 3259, 3290, 3292,
3460, 3561, 3598
Beltline, 3655
Benchmark, 480, 542, 548, 654, 655, 876, 877, 1183,
2930, 2931, 3632
Bench Marks, 3632, 3912
Bench Mining, 4229
Bench Pond, 1638, 1640, 1642, 4428, 4442, 4444,
4446
Berm Grades, 1970
Best Fit, 1266, 1267, 1353, 1369, 3123, 3290
Best Fit Centerline, 896
Best Fit Circle, 894, 896
Best Fit Curve, 239
Best Fit Line by Average, 898
Best Fit Line by Least Squares, 899
Best Fit Point, 893
Best Fit Profile, 1847
Best Fit Transformation, 1331
BIM Menu, 2856
Black/White By Threshold, 2848
Blast Pattern Layout, 4166, 4168
Blast Point Report, 4168, 4169
Blending Weighted Average, 3430
Blips, 172, 277, 279
Block Attributes, 690, 768
Block Data Report, 283, 284
Block Diagram, 3446
Block Explode, 139
Block Model 3D Viewer, 3479, 3608, 3614
Block Model Editor, 3466, 3472, 3474
Block Model Inspector, 3477
Block Model Menu, 3451
Block Model Statistics, 3483
Bold Curve Leader, 261
Border, 1276, 1277, 1425, 1521, 2636, 2643, 3004,
3912, 3914
Boundaries, 1413, 1498, 1668, 2701, 3189
Boundary, 243, 269, 818, 852–854, 881, 921, 1007,
1010, 1012, 1026, 1159, 1310, 1497, 1498,
1502, 1515, 1525, 1529, 2152, 2220, 2224,
2667, 2688, 2766, 2824, 3151, 3152, 3161,
3166–3172, 3174, 3176, 3178, 3182, 3188,
3199–3202, 3428, 3429, 3674, 3685, 3686,
3689–3693, 3780, 3837, 3949–3951, 3954–
3957, 3962, 3964, 3965, 3968–3970, 3975–
3977, 3979, 3980, 3982–3985, 3995–3997,

4000–4004, 4051, 4052, 4179
 Boundary Conditions, 2224
 Boundary Enclosure, 3667
 Boundary Menu, 3950
 Boundary Polyline, 26, 269, 270, 818, 1500, 1519,
 2666, 2667, 2676, 2677, 2679, 2727, 2776,
 3577, 3598, 3689, 3835, 3955–3957, 4000,
 4177, 4352, 4620, 4636, 4660
 Break, 140, 143, 144, 163, 178–180, 854, 903, 1034,
 1344, 1441, 1445, 1519, 1778, 1783, 1784,
 1787, 1788, 1791, 1953, 2488, 3416, 3600,
 3617, 3654–3656, 4009, 4089, 4235, 4351,
 4353
 Break 3D Polyline by Surface, 1471
 Break at Intersection, 144
 Break at Selected Point, 144
 Break by Crossing Polyline, 143
 Breakline, 198, 703, 781, 1505, 1519, 1520, 1524,
 1526, 1527, 1662, 3071, 3098, 3100, 3840
 Breaklines, 30, 324, 1285, 1486, 1495, 1496, 1499,
 1500, 1502, 1503, 1505, 1519, 1867, 2626,
 3076, 3098, 3101, 3835, 3838, 3840, 3841
 Break Polyline at Specified Distances, 143
 Break Polyline by Property, 3692, 4003
 Brighten/Darken Image, 2849
 Bubble Cul-de-Sac, 1262
 Buffer Offset, 187, 915
 Building Dimensions, 1075, 1077, 4302, 4303, 4330
 Building Face Surface, 2934
 Building Offset Extensions, 910, 911
 Building Pad, 945, 948, 949, 1004, 1005

 C-factor, 2170, 2171
 CAD Basics, 4236
 CADnet Module, 2792
 Calculate C-Factor, 2170, 2172
 Calculate End Area, 1954, 1955
 Calculate Fault Shift, 3376
 Calculate GeoFluv Volume, 3188
 Calculate Haul Factors, 1934
 Calculate Horizontal, 1248
 Calculate Intersection Point, 1857
 Calculate Offsets, 241, 1718, 1842
 Calculate Pond/Pit Volume, 1621, 2706, 2708, 2714,
 2715, 4468
 Calculate Removals Volumes, 2689
 Calculate Residuals, 3431, 3433, 3587
 Calculate Section Volumes, 1866, 1951, 1955, 3852,
 4481, 4486, 4487, 4673

Calculate Solid Volumes, 3795
Calculate Spoil Volumes, 4187
Calculate Stage-Storage, 2276
Calculate Stockpile Volume, 1619, 2706, 4428, 4467,
4470
Calculate Total Volumes, 2710, 2714, 2760, 4624,
4632, 4633, 4664
Calculate Variogram, 3403, 3455, 3462
Calculate Volumes by Five Methods, 4464
Calculate Volumes Inside Perimeter, 2714
Calculator, 280, 281, 1725, 3991, 3993, 4571, 4577,
4579
Calendar, 3697, 3700–3702, 3744, 3757, 3799, 3801,
3815, 3927, 4024, 4027–4029, 4055, 4079,
4224, 4226
Callout Leader, 260, 261
Calls, 1293–1295, 1310, 1313, 1347, 1364, 1365
Call Table, 1297, 1298
Camera Tab, 3002, 3004, 3013, 3016
CAPD Command Reference, 853
CAP files, 2277
Carlson Academy, 447, 448
Carlson Cloud, 455–461
Carlson Community, 86–89, 697, 775
Carlson Directory Structure, 2
Carlson Field Icon Menu, 2937
Carlson File Types, 34
Carlson GIS and Esri, 2516
Carlson Grade Project, 4173, 4174
Carlson Migration, 9
Carlson On-line Manual with Movies, 447
Carlson Points, 42, 151, 402, 435, 443
Carlson Registration, 14
Carlson Settings Explorer, 347
Carlson.PhotoCapture, 89
Case Studies, 3558, 3585, 3917, 4199, 4220
Case Study #1: Techniques of Geological Composi-
ting, 3558
Case Study #2: Outcrop and Subcrop Modeling, 3570
Case Study #3: Techniques Of Gridding, 3579
Case Study #4: Limestone Block Modeling, 3589
Case Study #5: Block Modeling by Quality At-
tributes, 3611
Case Study #6: Underground Mine Layout and Tim-
ing, 3917
Case Study #7: Dragline Range Diagrams, 4199
Case Study #8, Part 1: Underground Mine Mapping
Procedure, 3931

Case Study #8, Part 2: Computing Tonnage and Acreage, 3942
Case Study #9: Surface Timing With Benches, 4216
Cast Blast Profile, 4136, 4139, 4140, 4199
Cast Blast Report, 4169, 4170
Cast to Final, 4137, 4138, 4170, 4201
Centerline, 25, 28, 34, 94, 101, 104, 106, 221, 224, 241, 250, 295–297, 308, 322, 832, 837, 839, 896, 922, 943, 948, 949, 957, 995, 1420, 1423, 1652, 1685, 1686, 1688, 1689, 1698–1702, 1704, 1706, 1707, 1709–1712, 1714–1716, 1718–1723, 1728–1731, 1733, 1739, 1765, 1790, 1791, 1811, 1812, 1826, 1828, 1836, 1842, 1855, 1865, 1878, 1915, 1941, 1946, 1952, 1955, 1962, 1967, 1970, 1977, 1983, 1984, 1986, 1996, 1997, 2002, 2014–2017, 2028, 2029, 2032, 2033, 2039, 2055, 2065, 2066, 2070, 2071, 2076–2080, 2082, 2087, 2088, 2091, 2094–2098, 2101, 2103–2105, 2107, 2109, 2110, 2133, 2220, 2231, 2295, 2324, 2334, 2413, 2436, 2454, 2455, 2471, 2473, 2478, 2479, 2483, 2487, 2488, 2490, 2497, 2501, 2638, 2641, 2698, 2700, 2701, 2780, 2882, 2949, 3115, 3119, 3120, 3437, 3445, 3626–3628, 3662, 3663, 3672, 3682, 3788, 3851, 3883, 3889, 3890, 3899, 3900, 3905, 3944, 4138, 4142, 4144, 4161, 4214, 4481, 4490, 4497, 4516, 4517, 4524, 4565, 4566, 4698, 4740, 4745
Centerline Conversion, 1720, 1721
Centerline Conversions, 1720
Centerline Editor, 1688, 1700, 1827, 2002, 2082, 3785, 3889, 3900, 4482, 4516, 4694, 4740
Centerline File, 27, 819, 821, 837, 840, 1043, 1701, 1702, 1711, 1714, 1941, 2002, 2004–2007, 2009, 2011, 2014, 2043, 2079, 2080, 2082, 2109, 2381, 2386, 3120, 3772, 4214, 4481, 4516, 4565, 4566, 4569, 4698, 4716, 4721
Centerline ID, 1510, 1701, 3846
Centerline Menu, 1685, 4490
Centerline Position, 2945, 2947
Centerline Report, 1700, 1701
Centroid Point, 272
Centroids, 1926, 1952, 2039, 3189, 3191, 4740
CGCogo, 1230
CGDos Drawings, 1146–1148
CGDraw, 1273, 1285, 1288, 1294
CGEditor, 1371–1376, 1379, 1386, 1388, 1389, 1392,

1393, 1401, 1403, 1407, 1408, 1411, 1413,
1416
CGFile, 1130, 1133, 1134, 1139, 1141, 1149, 1150,
1232
CGMngmt, 1313
CG Options, 1347, 2302
CG Snap, 1369
CGSurvey Module, 1129
CGTools, 1347
CGTopo, 1344, 1362
CGTrav, 1151, 1154, 1223, 1225, 1227, 1230
Chamfer By Chord Length, 144, 145
Chamfer Pillars, 3660, 3661
Change Attribute Style, 145
Change Block/Inserts Resize, 150
Change Block/Inserts Rotate, 149
Change Block/Inserts Substitute, 149
Change Colors, 147
Change Contour-Plines Width, 1551
Change Drillhole Surface Z, 3286
Change Drillhole Symbol Number, 3285
Change Drillhole Symbol Size, 3286
Change Elevations, 145, 1419, 2637, 2857
Change History Report, 3312
Change Layer, 189, 207, 1419, 1662, 2638, 4299
Change Layer Color, 210
Change Lineweight, 148
Change Point LayerColor, 433
Change Polyline Linetype, 703, 781, 1115, 1117,
1120, 4292, 4298
Change Polyline Width, 187, 4299
Change Process On/Off Status, 3286
Change Projection, 3026, 3046
Change Style, 146
Change Text Font, 158
Change Text Oblique Angle, 160
Change Text Size, 159
Change Text Width, 159
Channel Add, 3170
Channel Delete, 3171
Channel Design - Erodible Mannings Equation, 2293
Channel Design - NonErodible Mannings Equation,
2291
Channel Geometry, 2220
Channel Lining Analysis, 2287, 2291
Channel Name, 3171
Channel Routing Hydrograph - Convex Method,
2210, 2211

Channel Routing Hydrograph - Modified Att-Kin
 Method, 2212, 2240
 Channel Sample, 34
 Channel Samples, 3244, 3245, 3247, 3335
 Channels Tab, 3169
 Channel Transition, 3171
 Check/Label Clearance, 2411, 2446
 Check 3D Polylines Match Vertices, 1472
 Check Elevation Range, 183, 226
 Check Lot Network Parameters, 1022
 Check Pit Polyline Overlaps, 3955
 Check Property Polyline Overlaps, 3692, 4003
 Check Reference Centerlines and Surface, 2411
 Check Sewer Network Parameters, 2411
 Chord Bearing, 282, 733, 1299
 Chord Length, 145, 231, 232, 236
 Civil 3D All, 98
 Civil 3D Labels, 98
 Civil 3D Linework, 98
 Civil 3D Pipes, 98
 Civil Module, 1417, 1709
 CLAYER, 34, 374, 2587, 3598
 Clean Cloud, 3043, 3054, 3074
 Clean Solid, 3827
 Cleanup Grid Area, 3552
 Cleanup Lot Linework, 939, 940
 Clear Directions, 3971, 3976, 4183, 4185, 4187
 Clear Image Set, 2623
 Clear Overexcavate Surface, 2681
 Clear Pit Bench Quantities, 3979
 Clear Pit Precedence, 3979
 Clear Solid Timing, 3820
 Clear Spoil Volumes, 4181
 Clear Strata Surface, 2766
 Clear Timing Map, 3760, 4087
 CL File, 25, 169, 909, 1466, 1707, 1711, 1729, 1730,
 1733, 1735, 1812, 1823, 1826, 1852, 1866,
 1936, 2030, 2094, 2701, 3445, 3852, 4456,
 4457, 4483, 4491, 4497, 4517
 Close, 1133, 1146, 1209–1211, 1213, 1214, 1221–
 1223, 1290, 1301, 1339, 1373, 1376, 1377,
 1380, 1384, 1385, 1451, 1473, 1478, 1497,
 1557, 1558, 1618, 1624, 1640, 1709, 1803,
 1849, 1939, 2151, 2167, 2202, 2390, 2526,
 2528, 2540, 2541, 2596, 2715, 2750, 2805–
 2809, 2811, 2812, 2821, 2822, 2824, 2826,
 2836, 2841, 2842, 2899, 2916, 3006, 3015,
 3083, 3091, 3093, 3204, 3343, 3373, 3409,

3659, 3660, 3664, 3776, 3777, 3950, 4176,
4189, 4190, 4291, 4292, 4296–4299, 4326,
4344, 4349, 4352, 4403, 4433, 4443, 4466,
4489, 4539, 4551, 4583, 4614, 4653, 4655,
4656, 4660, 4704, 4710, 4755

Close/Open Polylines, 186

Close Current, 1339

Closed Polylines, 924, 931, 932, 1033, 1054

Close Map Check File, 1146

Close Raw File, 1146

Closure, 513, 515, 537, 550, 566, 567, 569, 570,
585, 588, 603, 618, 619, 623, 626, 627, 631,
637, 645, 646, 734, 810, 826–830, 848–851,
1034, 1035, 1151, 1152, 1154, 1215, 1218,
1219, 1225, 1355, 4249, 4261, 4273

Closure by Point Numbers, 826

Closure Error, 848, 920, 1035

Cloud Objects, 3041

Coal Sections, 322, 3672, 3912

Coal Section to Drillhole, 3234

Code Definitions, 663, 664, 684, 706, 741, 742, 762,
784, 3207, 3694, 4005, 4419

COGO Menu, 855, 2870

Collision Conflicts Check, 2413

Color 3D Lines By Slope, 1945

Color by Elevation, 1647, 3513

Color Contours by Elevation, 1547

Color Contours by Interval, 1548

Color Elev Grid by Block Model, 3485

Color Elev Grid by Strata, 3450

Color Pits by Grade Parameters, 3484

Colors By Elevation, 148, 325

Colors From Layer, 147

Color Solid, 3834

Color Surface By Layer, 2703

Color TIN By Image, 2607

Combined Transformations, 1327

Combine Solids, 3830

Command Entry, 23

Command Search, 448

Common Format List, 379

Common Utilities, 3129

Compare Drillholes to Block Model, 3479

Compare Points, 385, 390, 391

Compare Section Files, 1948

Compare Sewer Networks, 2448

Compare Strata To Model, 3314

Compass, 510, 514, 517, 518, 536–538, 540, 852,

853, 918, 1208, 1354
 Compass Polyline Adjustment, 850, 851, 853
 Compliance Areas from Drillholes, 3326
 Compliance Areas from Grids, 3325
 Compliance Drillhole Report, 3323
 Composite Bed Qualities by Density, 321
 Composite Duplicate Strata, 3288
 Composite Qualities, 3567
 Composite Qualities Analysis, 3557
 Composite Strata By Beds, 3289
 Composite Strata by Elevation, 3291
 Composite Strata By Interval, 3289
 Composite Zones, 3560
 Compositing, 3289, 3291, 3410, 3558
 Compound or Reverse, 237
 Configuration, 34, 37, 309, 325, 349, 363, 949, 1191,
 1278, 1357, 1359, 1361, 1684, 2963, 2966,
 2969, 2976, 2977, 3209, 3231, 3234, 3238,
 3244, 3246, 3672, 3673, 3676, 3681, 3682,
 3942, 3943, 4161, 4162
 Configure Carlson, 305, 341, 819, 924
 Configure Field, 2872, 2890, 2893, 2899, 2901, 2902,
 2907, 2911, 2914, 2916, 2919, 2923, 2924,
 2928, 2937–2939, 2941, 2945, 2958–2963,
 2970, 2972–2974
 Configure Section Info, 3670, 3671, 3942, 3943
 Conformance, 319, 3589
 Connect Driveways, 1023, 1024
 Connecting Pillars and Perimeter, 3941
 Connect Mine Plans, 3720, 3730
 Consolidate Area Table, 931
 Consolidate Table, 167, 930, 931
 Construction Module, 2624
 Contour Elevation Label, 1510, 1552, 3360, 3845,
 4317
 Contour From Grid File, 3515
 Contour From Section File, 1541
 Contour from TIN File, 1540
 Contour from Triangular Mesh, 1532
 Contour ID, 1546
 Contouring, Break Lines and Stockpiles, 4411, 4428
 Contouring, DTM and Design, 4396, 4411
 Contours, 187, 312, 1344, 1346, 1354, 1437, 1461,
 1496, 1506, 1510, 1524, 1525, 1536, 1540,
 1544, 1547–1553, 1555, 1557, 1558, 1635,
 1642, 1657, 1674, 1675, 1680, 1681, 1878,
 2044, 2126, 2274, 2630, 2631, 2633, 2660,
 2745, 2825, 3007, 3014, 3025, 3059, 3076,

3107–3110, 3180, 3336, 3354, 3358, 3361,
3362, 3518, 3522, 3841, 3845, 3846, 3910,
4399, 4413, 4738, 4761
Contour SDPS Output File, 3909
Contours from Grid File, 1533
Control Point Editor, 3027, 3028, 3030
Convert AcadMap to GIS Links, 2576
Convert As Determined Qualities, 3556
Convert C&G to Carlson Points, 441
Convert CAICE KCM to Carlson CRD, 443
Convert CAL Images, 2605
Convert Carlson Points to Ashtech GIS, 443
Convert Carlson Points to C&G, 441
Convert Carlson Points to Eagle Point, 444
Convert Carlson Points to Land Desktop, 440
Convert Carlson Points to Simplicity, 441
Convert Carlson Points to Softdesk, 443
Convert Civil 3D to Carlson Points, 439
Convert CRD to Land Desktop MDB, 438
Convert CRD to TDS CR5/Convert TDS CR5 to
CRD, 438
Convert Dashes To Polylines, 2730, 2865
Convert Drillholes to External Database/Convert
Drillholes to Drawing Data, 3235
Convert Eagle Point to Carlson Points, 444
Convert Geodimeter to Carlson Points, 443
Convert GIS Links to AcadMap, 2575
Convert Land Desktop MDB to Carlson Points, 439
Convert Land Desktop to Carlson Points, 440
Convert LDD Contours, 1680, 2868
Convert LDT/Civil3D Surface Drawing, 1681
Convert Leica to Carlson Points, 442
Convert Old CG Dos Cross Section File to New For-
mat, 1149
Convert Old CG Dos Level File to New Format, 1149
Convert Old CG Dos Raw File to New format, 1149
Convert Old CG Dos Template File to New Format,
1150
Convert PacSoft CRD to Carlson CRD, 443
Convert Polylines To Text, 2864
Convert Simplicity to Carlson Points, 442
Convert Softdesk to Carlson Points, 440
Convert Spot Elev To Points, 1450, 2652
Convert Surveyor1 to CRD, 438
Convert Text To MText, 160
Convert to SurvCE FCL file, 1339
Coordinate API, 4753
Coordinate Cloud, 430, 2980

Coordinate File, 34, 240, 290, 308, 311, 373, 377,
383, 387, 397, 399, 403, 406, 460, 468,
470, 494, 503, 508, 529, 532, 545, 548, 577,
579, 582, 660, 695, 738, 773, 834, 835, 844,
877, 884, 900, 965, 1130, 1133, 1158, 1168,
1169, 1286, 1313, 1314, 1316, 1348, 1373,
1383, 1407, 1408, 1410, 1412, 1567, 1716,
1826, 1873, 1885, 1887, 1937, 2033, 2519,
2879, 3534, 4238, 4340, 4356, 4601, 4602,
4605, 4697, 4753, 4754

Coordinate Files, 1131–1134, 1145, 1146, 1155,
1327, 1343, 1410, 1414, 4753

CoorDinate File Utilities, 384

Coordinate File Utilities, 145, 311, 369, 374, 377,
385, 386, 406, 437, 440, 442–444, 859, 4465

Coordinate Grid, 1277, 1278

Coordinate Leader, 1307

Coordinate Points, 3031, 3033–3037, 3050, 3057,
3059, 3060, 3076, 3082, 3134, 3136–3139

Coordinates, 1295, 1332, 1383, 1718, 2071, 2332,
2572, 2875, 2876, 2878, 2911, 2930, 2951,
3027, 3045, 3250, 3616, 3913, 4393, 4707,
4765, 4766

Coordinate Transformation, 36, 290, 385, 389, 412,
1331, 1333, 2837

Coords From Station Offset, 1241

Copy Coordinates, 1333

Copy Entity to Layer, 1365

Copy Polyline Section, 135

Copy To Layer, 135

Corner point, 3627, 3628, 3661

Cost Summary Report, 2726

Cover Factor, 2297–2299

Crandall, 510, 518, 850, 1214, 1218, 1354

Crandall Polyline Adjustment, 849

CRD File, 33, 169, 276, 308, 369, 385, 386, 388,
389, 392, 403, 406, 411, 416, 432, 440–444,
460, 473, 475, 481, 482, 494, 498, 650, 920,
1035, 1842, 2582

Create, 1278, 1279, 1292, 1321, 1322, 1335, 1339,
1341, 1423, 1425–1428, 1443, 1446, 1523,
1541, 1544, 1562, 1604, 1609, 1653, 1672,
1713, 1718, 1726, 1730–1734, 1746, 1748,
1761, 1790, 1800, 1801, 1825, 1826, 1865,
1866, 1874, 1896, 1925, 1936, 1937, 1990,
2003, 2105, 2148, 2197, 2220, 2223, 2228,
2259, 2260, 2324, 2333, 2338, 2374, 2451,
2465, 2529–2535, 2539, 2543, 2561, 2569,

2578, 2583, 2603, 2608, 2614, 2641, 2643,
2644, 2701, 2727, 2733, 2774, 2775, 2844,
2868, 2885, 2892, 2893, 2895, 2932, 2987,
2990, 3015, 3016, 3018–3023, 3025, 3042,
3045, 3047–3049, 3051, 3055, 3059, 3060,
3063, 3064, 3067, 3068, 3073–3075, 3078,
3085–3087, 3093, 3096, 3097, 3102–3104,
3107, 3109, 3110, 3112, 3120, 3121, 3126,
3129, 3130, 3169, 3181, 3202, 3236, 3245,
3247, 3282, 3284, 3341, 3353, 3354, 3358,
3367, 3383, 3434, 3476, 3505, 3560, 3624,
3626, 3634, 3694, 3770, 3787, 3851, 3852,
3864, 3903, 3906, 3945, 3970, 3975, 4050,
4096, 4114, 4150, 4157, 4179, 4182, 4200,
4242, 4336, 4349, 4358, 4446, 4464, 4471,
4472, 4484, 4485, 4498, 4517, 4568, 4585,
4586, 4757, 4759, 4761

Create/Edit Lateral Structure, 2404
Create AutoCAD Icon, 364
Create Breaklines From Triangulation, 1461
Create Cloud, 3042, 3043, 3059, 3075
Create Curb Profiles, 1745
Create Drillhole From eLog, 3281
Create Gap Points, 3822, 3823
Create GIS Polylines by Interior Text, 2546
Create Grid, 3086
Create Image from Drawing, 2608
Create Image World File, 2599
Create Links, 2544, 2563, 2577
Create Overshoot Section File, 1884
Create Point, 3078, 3079, 3081–3083, 4166, 4354,
4358, 4695
Create Point Group From Station Offset, 1243
Create Points from Entities, 904, 906
Create Point Table, 1086, 1088
Create Polyline, 1539, 3087, 3089, 3123, 3521
Create Polyline ID Labels, 181
Create Ridge Polylines From Contours, 1460
Create Solid, 3040, 3075, 3476, 3770
Create StarNet File, 1227
Create Strata Polylines at Faults, 3382
Create Table, 1085, 1088, 1095
Create Text, 3096, 3097
Create Trench Network Structure, 2773, 4637, 4639
Create Unique Polylines, 2587
Create Vegetation Scene, 3202
Create World File by Image in Drawing, 2599
Create World File from Image Alignment, 2598

Crop Block Model, 3487
 Crop Image, 2845
 Crop loss, 3576
 Crosshairs, 363
 Crossing Breaklines, 1512, 2628, 3184, 3847
 Crossing Profile, 1806
 Cross Sections, 836, 1864, 2132, 2133, 2233, 2698,
 3120, 3850
 CSI GBX Pro, 2957
 CSTYLE, 34
 Cul-de-Sacs, 943, 1259, 1353, 1369, 2102, 4347
 Cul-de-sacs, 2102, 4746
 Current Channel, 3171–3174, 3176
 Current Channel Settings, 3172
 Current Information, 1130
 Curve, 1299–1301, 1304, 1305, 1310, 1341, 1342,
 1350, 1353, 1354, 1356, 1369, 1401–1404,
 1414, 1415, 1542, 1543, 1686–1690, 1693–
 1696, 1702, 1704, 1707, 1714, 1717, 1719,
 1725, 1741, 1744, 1746, 1777, 1791, 1794,
 1796, 1826, 1829, 1847, 1859, 1865, 1935,
 1979, 1981, 1986–1988, 2016, 2017, 2092,
 2094, 2095, 2104, 2147, 2149, 2166, 2167,
 2186, 2189, 2190, 2220, 2225, 2246, 2252,
 2275, 2283, 2286, 2295, 2297, 2356, 2443,
 2484, 2488, 2499, 2500, 2502, 2503, 2826,
 3083, 3084, 3103, 3105, 3280, 3356, 3357,
 3640, 3851, 3886, 3890, 3891, 3894–3897,
 4102, 4160, 4247–4249, 4256, 4260, 4261,
 4268, 4272, 4273, 4280, 4489, 4495, 4537–
 4544, 4548, 4584, 4590, 4592, 4717–4719
 Curve - Arrow, 257, 258
 Curve Between Tangents, 1249
 Curve Calc, 240, 241, 1248
 Curve Info, 282, 306, 855
 Curve Number Library, 2369, 2370
 Curve Numbers & Runoff, 2167, 2173, 2186, 2190,
 2252
 Curves, 1248–1251, 1253–1258
 Curve Table, 27, 1055, 1057, 1089–1091, 1093–1097,
 1099–1105, 1285
 Custom, 11, 21, 22, 35, 64, 84, 157, 198, 368, 371,
 409, 411, 512, 513, 690, 697, 703, 768, 775,
 781, 1043, 1046, 1063, 1110, 1337, 1601,
 1899, 2345, 2355, 2362, 2365, 2366, 2373,
 2418, 2431, 2538, 2711, 2718, 2719, 2722,
 2752, 2758, 2759, 2996, 2997, 3056, 3102,
 3104, 3122, 3126, 3220, 3221, 3223, 3225,

3229, 3231, 3235, 3267, 3591, 3634–3637,
3749–3753, 3810–3813, 3867, 4042, 4072,
4074, 4075, 4679

Custom Drillhole Report, 3303
Custom Label Formatter, 1109
Custom Label Formatter AD, 1062
Cut/Fill Centroids, 1616, 2701, 3188
Cut/Fill Color Map, 1504, 1594, 1602, 1604, 2709,
2740, 2747, 3526, 3840, 4664
Cut/Fill Contours, 1607, 1608, 2709, 2738
Cut/Fill Grid Map, 1605, 1606
Cut/Fill Labels, 1600, 2735, 2737, 2739, 4622–4624,
4632
Cut/Fill Map Legend, 2703
Cut/Fill Movement, 1612
Cut/Fill Report, 1599
Cut/Fill Slope Color Map, 1604
Cut/Fill Slope Lines, 1609, 1611
Cut/Fill Width Analysis, 1929
Cut and Place (Spoil Removal), 4130, 4205
Cut Image, 2845
Cut Only, 1476, 4134
Cut Only (Coal Removal), 4134, 4207, 4210, 4212
Cut Sheet, 832, 835, 837, 840, 845, 912, 1930, 4235,
4690, 4696, 4697, 4699, 4704, 4705, 4710
Cutsheet, 2871, 2879
Cutsheet Spreadsheet Editor, 2871

Database File Utilities, 2577
Data Capture, 2529, 2531, 2533, 2535, 2537
Data Capture Add Point Data to Linework, 2535
Data Capture Block Attributes, 2533
Data Capture Enclosed Text, 2531
Data Capture From CAD Properties, 2528
Data Capture Text By Sample, 2529
Data Collectors, 461, 462, 495, 1164, 1168
Data Collector Transfer, 618, 1155, 1161, 1184, 1208
Data Depot, 326, 341, 346
Data Entry, 3158
Data Extraction for HydroCAD, 4584, 4596
Data File Types and Storage, 4730
Data for Current Channel, 3174
Data for GeoFluv Work Area, 3178
Data for Main Channel, 3167
Data Links, 3709, 4036
Data Objects, 3020, 3057, 3059
Data Source Name, 2553, 2567, 2585
Data Tab, 3004, 3017
Decompress Image, 2847

Deed Correlation, 814
Deed Description, 26, 36, 293, 4249, 4261, 4273
Deed File By Interior Text, 813
Deed Linework ID, 814
Deed Reader, 807, 809, 4735
Define Area Layers, 2593, 2594
Define Attributes, 3206, 3208, 3216, 3218, 3224,
3250, 3296, 3335, 3694, 4005
Define Block Database Links, 2578, 2579
Define Block Model Auto-Run, 3458
Define Dragline Equipment, 4110, 4129, 4199, 4203
Define Drillhole, 2766, 3206, 3207, 3209, 3214, 3231,
3237, 3238, 3242, 3285, 3310, 3312, 3363,
3365, 3590, 3694, 4004, 4628, 4631, 4675
Define Equations, 3207, 3217, 3490, 3491, 3694,
4005, 4060
Define Equipment, 3716, 3926, 4225
Define Ferm Codes, 3207, 3218, 3694, 4005
Define Geologic Model, 3368, 3413, 3563, 3586,
3795, 4006
Define Geologic Order, 3207, 3216, 3694, 4005
Define Grade Parameters, 3469, 3472
Define Horizon Codes, 3207, 3219, 3694, 4005
Define Layer Target/Material/Subgrade, 2661, 2665,
2669, 2677, 2679, 2706, 2709, 2720, 2721,
2735, 2737, 2741, 2818, 4614
Define Lookup Database, 3206, 3214, 3237, 3305,
3310, 3694, 4005
Define Lot Attributes, 1039
Define Lot Edge Grade Rules, 1440, 1449
Define Materials, 2039, 2665, 2720
Define Median Adjustments, 2014
Define Mining Model, 3371, 3413, 4006
Define Note File Prompts, 2580, 2581, 2882
Define Parameters, 3251, 3320, 3328, 3330, 3331,
3340, 3341, 3350
Define Pipe Groups, 2783
Define Pit Attributes, 4232
Define Road Design Parameters, 2016, 2035, 2089
Define Strata, 319, 3209–3211, 3214, 3217, 3218,
3239, 3272, 3352, 3355, 3438, 3446, 3451,
3567, 3570, 3589, 4038
Define Strata/Bed, 3206, 3209, 3694, 4004, 4677,
4679
Define Strata Grids AutoRun, 3347, 3355
Define Strata Isopach AutoRun, 3354
Define Surface Mine Auto-Run, 4020
Define Surface Mine Auto Run, 3426

Define Template Database, 2519, 2523, 2882
Define Washing Curve, 3396, 3398, 3399
Define Watershed Layers, 2143, 2148, 2159, 2243,
4558, 4584, 4586
Delete, 1280, 1316–1318, 1324, 1338, 1344, 1374,
1423, 1426, 1634, 1642, 1724, 1725, 1742,
1746, 1864, 1880, 1893, 1978, 1980, 1993,
2203, 2283, 2341, 2344, 2345, 2369, 2370,
2372, 2373, 2472, 2474, 2480, 2482–2484,
2521, 2524, 2529, 2532, 2534, 2548, 2554,
2559, 2568, 2578, 2591, 2604, 2641, 2644,
2913, 3015, 3028–3032, 3036, 3038, 3040,
3051, 3056, 3058, 3074, 3091, 3128, 3130,
3131, 3135, 3139, 3171, 3242, 3387, 3397,
3400, 3635, 3824, 3850, 3861, 3956, 4159,
4387, 4418, 4756, 4759
Delete Attribute, 3297, 3397
Delete Labels, 1346
Delete Layer, 353
Delete Points, 385, 387
Delete Strata, 3296
Delete Table Elements, 1105
Delta Angle, 236, 1051, 1056, 1058, 1108, 1110,
1627, 1703, 1705, 3063, 4331
Densify Polyline Vertices, 170
Depth Contours, 1597, 2766, 2767
Depth Sounder, 2958
Description for Points, 385, 389
Description of Software, 3148
Descriptions, 1368, 1382, 1391, 1402, 1406, 1411,
1587, 1729, 1869, 1877, 1905, 1910, 1919,
2686, 2803, 2819, 2828, 2878, 3873, 3878,
4245, 4258, 4270, 4287, 4325, 4342, 4422
Description Tables, 1334, 1335, 1339, 1340, 1349
Design Bench Pit, 3415, 3418, 3497, 3581, 4008,
4012, 4081, 4145, 4148, 4152, 4216, 4217,
4222
Design Bench Pond, 1634, 1636, 1640, 1642, 2255,
4438
Design Centerline, 1685, 1686, 1688, 1700, 2033,
3889, 3900, 4481, 4482
Design Contours, 2737, 2738
Design Control, 4744
Design Detention Pond, 2196, 2242, 2252
Design Dragline Pit, 4145, 4146
Design Drawing, 2737, 4612, 4661
Design Fill Surface, 4156, 4159, 4160
Design GeoFluv Regrade, 3160–3162, 3185, 3187,

3189, 3191
Design Lot, 1027
Design Pad Template, 37, 1623, 1626–1628, 1630–
1634, 1824, 1957, 4409, 4410, 4428, 4435,
4452, 4461
Design Parking Inside Perimeter, 970, 973
Design Profile, 994, 995, 2033, 2132, 2762
Design Ramp Profile, 4160, 4161
Design Regrade, 1931, 1933, 1934
Design Road Profile, 1736, 1743, 1749, 1824, 2033
Design Section Staging, 1937
Design Sewer/Pipe Profile, 1749, 1803, 2319
Design Spillway, 2257
Design Spoil Pile, 4154
Design Surface, 2738, 2746, 2773, 2775, 2776, 2780,
2783, 2789, 2809, 2825, 3154, 3156, 3158,
3175, 3180, 4173, 4621, 4636, 4654, 4657,
4664
Design Surface 3D Viewer, 2716
Design Surface Vertical Offset, 2710
Design Template, 1624, 1740, 1914, 1956, 1976,
1989–1992, 1994–1996, 2003, 2004, 2007,
2009, 2014, 2029, 2032–2034, 2039, 2046,
2050, 2051, 2055, 2058, 2060, 2084, 2136,
3882, 4428, 4458, 4498, 4506, 4518, 4743,
4745
Design Tunnel Template, 3887, 3888
Design Valley Pond, 1642, 2241, 4403, 4447
Deskew Image, 2847
Detach Image From Entities, 2612
Detention Pond Sizing, 2252–2254, 4545
Detention Pond Sizing - Linear Storage Estimate
Method, 2253
Difficulty factor, 3707, 3714, 3736, 4035
Digitize, 2796–2799, 2801–2806, 2808–2818, 2821,
2823–2826, 2828–2831, 4648, 4650–4652,
4654–4660
Digitize 2D Polyline, 2820
Digitize 3D Polyline, 2822
Digitize Areas, 923, 2824
Digitize Contour Polyline, 2825
Digitize Contours (Polyline), 1557
Digitize Design, 2818
Digitize End Areas, 1882, 2830
Digitize Existing, 2818
Digitize Menu, 2793
Digitize Other, 2818
Digitize Perimeter, 2824

Digitize Point, 2819
Digitize Profiles, 2826
Digitize Rectangle, 2823, 2824
Digitizer Settings, 2816, 2818, 2828, 2830
Digitizer Setup, 2795
Digitize Sections, 1894, 1896, 2828, 3862, 3864
Digitize Sections Plan, 1879
Digitize Sections XSec, 1880
Digitize Spot Elevation, 2819
Digitizing, 4648, 4661, 4666
Direct-Reverse Report, 530
Direct-Reverse Settings, 530
Disconnect Mineplans, 3733
Discussion Groups, 8, 22
Display-Edit File, 286
Display Centerline Stations, 1243
Display Directions, 3975–3977, 4185–4187
Display Last Report, 286
Display Last TIN Error Log, 1558
Display Menu, 2734
Display Options, 2741, 3787, 4623
Display Order by Layer, 214
Display Precision, 576
Distance-Bearing with Leader, 1073
Distance-Distance Intersect, 870, 1235
Distance Between Two Entities, 1719, 1720
Distance Table, 1085
Distance with Leader, 1071
Disturbed Area, 2141
Ditch, 1859, 1860, 1968–1970, 1976, 1989, 2355,
2359
Ditch Grades, 1968–1970
Divide Along Entity, 902
Divide Between Points, 901
Documentation References, 3160
Dozer, 36, 105, 114, 2691, 4114, 4117, 4118, 4140
Dozer Push, 35, 4124, 4127, 4133, 4202, 4203
Dragline, 35, 37, 4110, 4112, 4114, 4116, 4119, 4120,
4122, 4129, 4130, 4132, 4138, 4139, 4142,
4146, 4205
Dragline Pit, 322
Dragline Pits, 4163
Dragline Section Report, 4139
Dragline Sequence, 4210
Drainage Runoff Library, 2370, 2371
Drape 3D Polyline On Surface, 2657
Drape Image on Surface, 2606
Draw, 1300–1303, 1306, 1308, 1334, 1336, 1338,

1346, 1347, 1352, 1353, 1358, 1359, 1418,
1425, 1429, 1440, 1445, 1453, 1454, 1457,
1460, 1468, 1471, 1478, 1479, 1499–1501,
1505, 1506, 1510–1513, 1518, 1521, 1523,
1530, 1535–1537, 1545, 1550, 1552, 1553,
1555–1557, 1567, 1584, 1594, 1595, 1597,
1600, 1602, 1607, 1612, 1613, 1618, 1619,
1621, 1626, 1634, 1635, 1638, 1645–1649,
1653, 1661, 1662, 1667, 1672, 1674, 1675,
1677, 1678, 1681–1683, 1685, 1689, 1694,
1700, 1701, 1704–1706, 1712, 1713, 1716,
1722, 1723, 1727, 1734–1736, 1748, 1749,
1751, 1755, 1756, 1760, 1764, 1766, 1767,
1770–1778, 1780–1782, 1784, 1786, 1787,
1789, 1791, 1793, 1794, 1796, 1798–1807,
1809–1811, 1813, 1815, 1820, 1824, 1834,
1836, 1846, 1849, 1859, 1867, 1875–1877,
1879, 1884, 1887, 1888, 1890, 1896, 1898,
1900, 1902, 1903, 1910–1913, 1915, 1916,
1928, 1935, 1936, 1939, 1942, 1945, 1949,
1957, 1960, 1964, 1971, 1972, 1984, 1987,
2017, 2018, 2020–2023, 2025, 2028, 2041,
2042, 2061, 2064, 2068–2070, 2093, 2105,
2114, 2120, 2153, 2154, 2157, 2158, 2160,
2217, 2254–2256, 2259–2263, 2266, 2275–
2277, 2280, 2281, 2283, 2284, 2286, 2292,
2293, 2302, 2303, 2312, 2317, 2318, 2327,
2334, 2335, 2337–2339, 2362, 2382, 2383,
2389, 2411, 2413, 2422, 2425, 2426, 2431–
2433, 2435–2439, 2441, 2442, 2446, 2453,
2455, 2457, 2460, 2472–2474, 2477, 2479,
2491, 2496, 2497, 2512, 2516, 2519, 2527,
2539, 2541, 2542, 2548, 2549, 2558, 2560–
2562, 2564, 2569, 2570, 2572–2574, 2579,
2582, 2586, 2588, 2593, 2598, 2599, 2601,
2604–2606, 2611, 2625, 2629–2631, 2636,
2643, 2650, 2678, 2684, 2686, 2694, 2695,
2700, 2707, 2708, 2710, 2723, 2738, 2743,
2745, 2761–2763, 2780, 2787–2789, 2795,
2799, 2801, 2810, 2816, 2824, 2825, 2827,
2828, 2831, 2836–2838, 2842, 2843, 2857,
2878, 2927, 2937, 2970, 2973, 2988, 3012,
3023, 3024, 3028, 3031, 3032, 3038, 3081,
3089, 3091, 3093, 3102–3105, 3109, 3115,
3119, 3122, 3130, 3135, 3143, 3153, 3166,
3180, 3184, 3185, 3202, 3210, 3218, 3241,
3250, 3252, 3253, 3256–3262, 3264, 3267,
3268, 3272, 3276, 3285, 3305, 3306, 3313,

3335, 3355, 3359, 3361, 3362, 3372, 3383,
3422, 3438, 3441–3443, 3445, 3448, 3451,
3470, 3471, 3473, 3485, 3513, 3514, 3517–
3519, 3526, 3527, 3529, 3534, 3554, 3572,
3584, 3593, 3594, 3613, 3634, 3643, 3644,
3648, 3649, 3655, 3656, 3664, 3669, 3671,
3696, 3719, 3744, 3753, 3754, 3757, 3758,
3762, 3765, 3766, 3769, 3774, 3778, 3785,
3792, 3793, 3802, 3808, 3813, 3816, 3825,
3828, 3834–3837, 3841, 3846–3849, 3858,
3864, 3866, 3868, 3870, 3871, 3878–3881,
3883, 3884, 3886, 3887, 3890, 3895, 3900,
3901, 3931, 3935, 3936, 3938, 3942, 3946,
3956, 3975, 3978, 4016, 4023, 4040, 4055,
4063, 4075, 4076, 4079, 4080, 4111, 4116,
4123, 4128, 4142, 4144, 4151, 4157, 4173,
4186, 4215, 4236, 4239, 4245, 4251, 4252,
4257, 4264, 4269, 4276, 4281, 4283, 4284,
4288, 4290, 4294–4298, 4300, 4307, 4311,
4312, 4314, 4318, 4320, 4322–4327, 4334,
4336, 4338, 4344, 4345, 4347, 4352, 4356,
4363, 4367, 4368, 4379, 4391, 4393, 4396,
4397, 4399, 4402, 4405, 4411, 4413–4415,
4420–4424, 4428, 4430, 4432, 4433, 4442,
4446, 4454, 4465, 4466, 4477, 4488, 4506,
4507, 4511, 4512, 4530, 4553, 4554, 4561,
4562, 4579, 4580, 4582, 4587, 4588, 4605,
4609, 4610, 4612, 4616, 4617, 4644, 4648–
4650, 4656, 4661, 4663, 4667, 4682, 4684,
4706, 4708, 4710, 4731, 4732, 4756, 4758,
4760, 4767

Draw-Locate Points, 372, 388, 402, 423, 460, 657,
733, 735, 857–861

Draw 3D Grid File, 1594, 1646, 1678, 2139, 2606,
3512, 3526

Draw 3D Mine Model, 3669

Draw 3D Model File, 256

Draw 3DPoly Base Breakline, 1618, 1619, 1621,
2707, 2708, 2715

Draw 3DPoly Perimeter, 1617, 1618, 2714, 4466

Draw 3D Solid, 3792

Draw Barscale, 1085

Draw Block Model, 3472, 3614

Draw Box Around Text, 249

Draw Building Envelope Polyline, 1439, 2649

Draw By Example, 267

Draw Centerline File, 1510, 1699, 3846, 3899

Draw Contour Gradient Marks, 1550

Draw Contour Label Marker Line, 1554
Draw Curb Ramp, 1476, 1998, 2000
Draw Depth Contours, 3338
Draw Design Surface, 3178–3180, 3183, 3185, 3187–
3189, 3199
Draw Dragline Limits, 4129, 4130, 4204, 4205, 4209
Draw eLogs, 3278
Draw End Point Leaders, 1064
Draw Entity Images, 2613
Draw Fault Labels, 3379
Draw Fault Line, 3372, 3376–3379, 3383, 3385
Draw Fault Surface, 3378
Draw Fault Symbols, 3379, 3380
Draw Field to Finish, 656–660, 662, 663, 667, 669,
672, 684, 685, 692, 714, 734–738, 740, 741,
745, 747, 750, 762, 763, 770, 792
Draw Flow Polylines TR20, 2215
Draw GeoFluv Contours, 3183, 3185, 3194, 3196
Draw Geologic Column, 3255, 3278, 3287, 3435,
3441, 3558, 3595, 3608, 3617
Draw Grade Boundary, 3474
Draw Grid, 1078, 1082
Draw Heave Zones, 3380
Draw Hec-Ras Watermark, 2234
Draw Hydrograph, 2217, 4554
Draw IDF Curve, 2441–2443
Draw Image Boundaries, 2621
Drawing Cleanup, 115, 118, 4749
Drawing Explorer, 69, 103, 119–121, 304, 306
Drawing Inspector, 26, 277–279, 370
Drawing Save Log, 131, 132, 306
Drawing Settings, 1273, 1285–1287, 1297, 1299
Drawing Setup, 31, 41, 42, 94, 96, 98, 104, 111,
122, 208, 213, 261, 263, 267, 278, 287, 289,
290, 292, 294, 305, 307, 309, 361, 362, 375,
377, 399, 434, 485, 490, 697, 719, 775, 797,
817, 822, 825, 851, 852, 856, 857, 869–871,
880, 882, 914, 915, 1038, 1045, 1048, 1050,
1082, 1093, 1096, 1099, 1110, 1120, 1122,
1124, 1125
Drawing Template, 31, 309
Drawing to 3D PDF, 2867, 2868
Draw Label For Companion Linework, 264
Draw Legend, 1080, 1082, 1083
Draw Lot Setback, 957
Draw Lots from File, 921
Draw Mapcheck, 1280, 1282
Draw Mass Diagram, 1923, 1953

Draw Menu, 218
Draw Model View, 206
Draw North Arrow, 1083, 1084
Draw Outcrops, 3335, 3570, 3571
Draw Outline, 3659
Draw Overexcavate Cut Color Map, 2681
Draw Overexcavate Surface 3D Faces, 2681
Draw Perimeter, 918, 3659
Draw Pillars, 3660, 3674, 3941
Draw Pipe, 718, 719, 796, 797
Draw Pipe 3D Polyline, 1849, 1850, 1876, 1915,
1939, 3883
Draw Pit/Channel Sample Text, 3247
Draw Plan View Sheets, 1818
Draw Polyline Blips, 172, 180
Draw Polyline File, 127
Draw Polyline Start/End, 173
Draw Profile, 28, 29, 36, 294, 300, 313, 1764–1766,
1769, 1770, 1775, 1780, 1791, 1803, 1812,
1815–1817, 1820, 1839, 1846, 1848, 1849,
1855, 1923, 1926, 1939, 1979, 2064, 2228,
2261, 2432, 2435, 2470, 2471, 2473, 2497,
2778, 2780
Draw Profile Grid, 1812
Draw Raster Image, 2832, 2834
Draw Reame File, 4172
Draw Removal Breakline, 2686
Draw Removal Contours, 2689
Draw Removal Field to Finish, 2686
Draw Removal Surface, 2688
Draw Roadside Ditch, 1860
Draw Road Stripes, 2029–2031, 2035, 2089
Draw SB-Slope File, 4172
Draw Section Alignment, 1865, 1866, 3850, 3852
Draw Section File, 1870, 1896, 1897, 1916, 1949,
2004, 2032, 2035, 2036, 2064, 3793, 3853,
3864, 3865, 3884, 4215, 4526, 4742
Draw SectionTemplate DWG, 1884
Draw Sewer Network-3DFaces, 2437
Draw Sewer Network Centerlines, 2435
Draw Sewer Network Data Table, 2432
Draw Sewer Network Plan View, 2432
Draw Sewer Network Profile, 2435–2437
Draw Single Manhole, 1844
Draw Solid 3D Faces, 3792
Draw Solid Centerline, 3793
Draw Solid Perimeter, 3793
Draw Spot Elevations, 1452, 1455

Draw Stage-Discharge Graph, 2284
Draw Stage-Storage Curve, 2277, 4548
Draw Standard Item, 262, 265
Draw Standard Item by Quick Key, 265
Draw Strata Cut Color Map, 2767, 4632
Draw Strata Cut Depth Contours, 2766
Draw Strata Surface, 2767, 2768, 2780
Draw Strike-Dip Symbol, 3388
Draw Subgrade Hatch Legend, 2671
Draw Subsidence Profile, 3910
Draw Super Elevation Color Centerline, 1982
Draw Super Elevation Diagram, 1983, 2036
Draw Surface As Grid, 2704
Draw Surface Boundary, 1649, 3514
Draw Surface Intersection, 1649, 3515
Draw Table, 256, 257
Draw Template Polylines, 1473, 1475, 2042
Draw Text On Arc, 1110
Draw Text on Tangent, 1112, 1113
Draw Top Surface 3D Faces, 2684
Draw Traverse-Sideshot Lines, 530
Draw Trench Network - Plan, 2779
Draw Trench Network - Profile, 2780
Draw Trench Network Centerline, 2779
Draw Triangular Mesh, 1645, 1646, 1648, 1674, 2140
Draw Typical Template, 1972, 2084, 4506
Draw Typical Trench Template, 1841, 2785
Draw Typical Tunnel Template, 3888
Drift Offset, 138
Drillhole Core Images, 3276
Drillhole Data Sheet, 3206, 3243, 3312, 3694, 4004
Drillhole Import, 2751, 3209, 3236, 3591, 4629,
4679, 4681
Drillhole Inspector, 3274
Drillhole Menu, 2749, 3206
Drillhole Reports, 2758
Drillholes to Points, 3236
Drillhole Strata Settings, 2749
Drillholes Without Key Strata, 3313
Drillhole Text Formatter, 3252
Drillhole Top to Surface Model, 3315
Drone Mission Planning, 485
Drop C&G Attributes, 1310
Drop Pipe Spillway Design, 2258, 2287, 4550
DTM API, 4759
Duplicate Drillhole Report, 3312
Duplicate Points, 117, 385, 390
Duplicate Strata, 2752, 3224

DWG Tab, 3158, 3183
DXFOUT Mine Plan to SDPS, 3906
Dynamic Annotation Note, 1064
Dynamic Blocks and Tool Palettes, 3633
Dynamic Point Attributes, 422

Eagle Point, 444, 665, 666, 668, 743, 744, 746

Eagle Point Coding, 665, 666, 743, 744

Earth Curvature, 512, 534, 537, 2880

Earthworks File, 1955

Edit, 1279–1281, 1316–1318, 1320, 1332, 1334–
1336, 1339, 1341, 1343, 1344, 1354, 1374,
1379, 1381, 1391–1393, 1401, 1403–1406,
1408, 1413, 1414, 1420, 1426, 1427, 1445,
1464, 1470, 1473, 1474, 1523, 1524, 1529,
1530, 1546, 1573, 1574, 1632, 1634, 1658,
1662, 1663, 1689, 1693, 1699, 1713, 1724,
1725, 1729, 1738, 1739, 1742, 1760, 1761,
1769, 1778, 1795, 1815, 1834, 1838–1840,
1855, 1864, 1867, 1888, 1889, 1891, 1892,
1923, 1924, 1953, 1954, 1961, 1965, 1968,
1971, 1974, 1975, 1977, 1980–1982, 1993–
1996, 2030, 2032–2034, 2038, 2055, 2061–
2063, 2066, 2074, 2078, 2079, 2081–2090,
2092, 2093, 2095, 2096, 2099–2105, 2110,
2111, 2114–2116, 2124–2127, 2129, 2130,
2140, 2141, 2144, 2147, 2166, 2167, 2188,
2189, 2231–2234, 2246, 2247, 2269, 2272,
2302, 2321, 2324–2326, 2341, 2349, 2355,
2360, 2366, 2369–2376, 2379, 2380, 2383,
2386, 2389–2392, 2395, 2396, 2399, 2402,
2404, 2405, 2408, 2427–2429, 2452, 2457,
2470–2472, 2476, 2478, 2479, 2481–2488,
2499, 2508, 2516, 2520, 2521, 2536–2539,
2548, 2549, 2578, 2581, 2594, 2595, 2614,
2615, 2617, 2619, 2660, 2662–2665, 2672,
2720–2724, 2726, 2731, 2749, 2753, 2755,
2756, 2783, 2784, 2833, 2834, 2839, 2863,
2875, 2883, 2936, 2966, 2967, 2969, 2988,
3015, 3024, 3025, 3028–3032, 3036, 3038,
3040, 3043, 3051, 3055–3058, 3060, 3062,
3073, 3074, 3091, 3094, 3095, 3104, 3107,
3118, 3119, 3124, 3125, 3128–3131, 3134,
3135, 3139, 3144, 3155, 3156, 3179, 3185,
3190, 3195, 3202, 3210, 3214, 3239–3241,
3244, 3267, 3268, 3308, 3309, 3340, 3348,
3354, 3360, 3387, 3396, 3426, 3434, 3459,
3466, 3469, 3490, 3540, 3541, 3585, 3594,
3623, 3634, 3635, 3637, 3639, 3683, 3698,

3700, 3711, 3713–3715, 3742, 3753, 3765,
3778, 3787, 3788, 3790, 3791, 3800, 3805,
3813, 3824, 3850, 3856, 3857, 3859, 3860,
3887, 3890, 3894, 3902, 3921, 3922, 3927,
3952, 3963, 3973, 3977, 3979, 3982, 3984,
3989, 3992, 3995, 4020, 4025, 4027, 4054,
4060, 4067, 4075, 4081, 4091, 4110, 4111,
4121, 4184, 4187, 4193, 4225, 4232, 4251–
4255, 4263–4267, 4275–4279, 4295, 4298,
4299, 4309, 4311, 4316, 4324, 4325, 4327–
4329, 4331, 4332, 4334, 4336, 4338, 4345,
4346, 4350, 4351, 4353, 4359, 4361, 4362,
4374, 4375, 4379, 4401, 4407, 4408, 4415,
4419, 4421, 4423–4425, 4434, 4443, 4444,
4447, 4456, 4485, 4486, 4507, 4509, 4570,
4577, 4579, 4593, 4614–4616, 4630, 4641,
4661–4663, 4669, 4670, 4672, 4673, 4684,
4718, 4719, 4722, 4723, 4745, 4747

Edit-Process Level Data, 542

Edit-Process Raw Data File, 461, 490, 491, 733, 826–
828

Edit/Create Sanitary/Utility Structure, 2389

Edit/Create Sewer Structure - Plan View, 2395

Edit/Create Sewer Structure - Profile View, 2395

Edit/Create Structure With Inverts, 2451

Edit 3D Grid, 1584

Edit Area Table Properties, 927

Edit Bench Pond, 1640, 1641

Edit Centerline On-Screen, 1698

Edit Coal Sections, 3673

Edit Contours, 1545, 2659, 3359, 4401

Edit Coordinates (CGEditor), 1320, 1321

Edit Curb Ramps, 1998, 1999

Edit Dragline Sequence, 4140

Edit Drillhole, 2754, 3208, 3213–3215, 3235, 3238,
3241, 3243, 3294

Edit Fault Line, 3374

Edit Field to Finish Codes, 731

Edit File, 131

Edit Layout Element, 2215

Edit Longitudinal Profile, 3194–3196

Edit Map Check File, 1223

Edit Menu, 132, 499, 547

Edit Mining Symbols Library, 3624

Edit Multiple Pt Attributes, 418

Edit Pad Template, 1632–1634

Edit Panel, 3720, 3735, 3760, 3925

Edit Panel Attributes, 3710, 3712, 3716

Edit Parking Inside Perimeter, 973
Edit Pit, 3988, 3989, 3992–3994, 4021, 4037, 4050,
4052, 4223, 4231
Edit Pit/Channel Sample, 3249
Edit Point, 370, 378, 398
Edit Point Attributes, 308, 370, 411, 415
Edit Points, 401, 726, 804, 815
Edit Polyline Section, 177
Edit Polyline Vertex, 176, 177, 2658
Edit Process End Area File, 1955
Edit Process SDMS File, 549
Edit Raw File, 1154, 1155
Edit Selected Layer, 2665
Edit Sewer Structure, 2375, 2397, 2443
Edit Solid, 3824
Edit Spoil Source, 4190
Edit Spoil Volumes, 4189
Edit Symbol Library, 351
Edit Table, 1091, 1096, 1104
Edit Table Properties, 1096
Edit Table Values, 1101
Edit Text on Arc or Tangent, 1113
Edit Trench Network Structure, 2775, 4640
Edit World File, 2599
Elevate 2D Polylines, 953
Elevate Intersection Curb Return, 1466
Elevate Lot Edges by Grade Rules, 1442
Elevate Menu, 2637
Elevate Pads by Grade Rules, 1444
Elevate Text, 1457
Elevation-Thickness Points, 3630
Elevation Along Entity, 902
Elevation Between Points, 1457
Elevation Difference, 856, 863, 1518, 1519, 1594,
1777, 1787, 1913, 1919, 1961, 2881, 2882,
2939, 2940, 3071, 3315, 3342, 3526, 3881,
3934, 4424
Elevations, 1368, 1421, 1428, 1436, 1437, 1455,
1457, 1458, 1461, 1478, 1479, 1502, 1532,
1535, 1560, 1619, 1622, 1701, 1707, 1728–
1730, 1767, 1772, 1776, 1777, 1811, 1812,
1821, 1822, 1871, 1877, 1904, 1910, 1911,
2254, 2323, 2325, 2386, 2400, 2406, 2465,
2626, 2639, 2707, 2708, 2754, 2755, 2773,
2787, 2803, 2819, 2878, 3158, 3240, 3254,
3293, 3340, 3348, 3418, 3436, 3454, 3462,
3503, 3517, 3608, 3613, 3838, 3855, 3872,
3878, 3879, 3905, 3912, 4012, 4149, 4157,

4245, 4258, 4270, 4287, 4342, 4424, 4434,
4656, 4762
Elevation Zone Analysis, 27, 1647, 1667–1669, 2140,
3513
Elevation Zone Section Volumes, 1922, 1923
Empty Print File, 1150
End Areas, 35, 1907, 1909, 2076, 3875, 3877
Enter-Assign Point, 860
Enter and Assign, 375, 4343
Enter Deed Description, 731, 810, 4344
Entering a Deed, 4245, 4257, 4269, 4281, 4344
Enter Profile On-Screen, 1736
Enter Right of Way, 1692, 1721, 1722, 3893
Enter Roadside Ditch, 1859, 1860
Entities to Polylines, 110, 169, 1698, 3899, 4447
Entity Insertion Point Rotate, 151
Entry Width, 3680, 3714, 3721
EPA_SWMM, 2464
Equipment Calendar, 3701, 3716, 4028
Equipment Menu, 2956
Equipment Setup, 2885, 2890, 2892, 2901, 2904,
2907, 2911, 2935, 2950, 2959–2961, 2963–
2965, 2969, 2971, 2972, 2974, 2977
Erase, 1306, 1468, 1471, 1478, 1482, 1502, 1626,
1645, 1662, 1704, 1824, 2021, 2028, 2042,
2425, 2525, 2527, 2539, 2542, 2543, 2626,
2666, 2667, 2669, 2672, 2674, 2675, 2786,
2836–2838, 2842, 2852, 3837, 4130, 4304,
4328, 4330, 4350, 4405, 4423, 4425, 4458
Erase by Closed Polyline, 133
Erase by Layer, 132
Erase Isolated Points, 3822
Erase Links, 2545, 2546
Erase Outside, 134
Erase Overexcavate Cut Color Map, 2681
Erase Overexcavate Surface 3D Faces, 2681
Erase Plan View, 2425
Erase Point Attributes, 424
Erase Points, 388, 402
Erase Strata Cut Color Map, 2767, 4633
Erase Strata Cut Depth Contours, 2767
Erase Strata Surface, 2768
Erase Sub-Areas Hatch, 1009
Erase Subgrade Hatches, 2671, 4617, 4663
Erase Subgrade Labels, 2672
Erase Surface from DWG, 1344, 1345
Escapeways, 3632, 3656
ESRI ArcGIS Services, 2560

ESRI to Office to Field and Back, 4596, 4608
EW File, 1954, 1955
Example Projects, 618
Existing Contours, 2735, 2736
Existing Drawing, 2735, 4612, 4661
Existing File, 39, 76
Existing Section, 1938, 1967, 1969, 2034, 2038,
2042, 2101, 2105
Existing Surface, 2736, 2787, 4164, 4448, 4453,
4524, 4546, 4652
Existing Surface 3D Viewer, 2717
Existing Surface Vertical Offset, 2710
Explode Carlson Points, 437
Export 12D File, 108
Export ADAC Network, 2461
Export Block Model, 3468
Export Carlson Sewer Network, 2461
Export Civil 3D Alignments, 107
Export Civil 3D Drawing, 106
Export Civil 3D Points, 107
Export Civil 3D Surface, 107
Export Cloud, 3047
Export Coal Sections To Text File, 3673
Export Coordinates to ASCII, 1134
Export DEM TIF File, 1684
Export Drawing to AutoCAD 14, 111
Export Drillholes, 3231, 3233, 3235, 3236, 4679,
4681
Export DWG File with Esri MSC, 2572
Export ePlan File, 111
Export ESRI File GeoDatabase, 2573, 2574
Export ESRI Projection File, 2573
Export GIS Data to SurvCE, 2574, 4600, 4603
Export Google Earth File, 108, 112, 114, 1683, 2603,
3278
Export LandXML File, 100, 1883
Export Lot File to MDB Database, 1041
Export Lot File To Old SurvCADD, 1041
Export OpenGIS GML, 2575
Export Pit/Channel Samples, 3247
Export Polyline File, 105
Export Pre-Carlson 2020 Sewer Network, 2463
Export RoadXML File, 103
Export SHP File, 2571, 2572
Export Solid, 3821
Export Solid Model IFC File, 2861
Export SQLite File, 2569
Export Surface Architectural IFC File, 2860

Export Text, 388
Export Text/ASCII File, 381, 385, 388, 400
Export To Esri shapefiles, 2463
Export Topcon TIN File, 1685
Export To Points, 2462
Export To Profiles, 2462
Export Trench Network Data, 2777
Extend, 140–143, 219–221, 223, 226–228, 452, 566,
676, 754, 949, 974, 980, 1446, 1834, 1868,
1890, 1891, 1926, 1942, 1953, 1954, 1970,
2095, 2338, 2436, 2499, 2715, 3373, 3746,
3807, 3858, 3859, 3970, 4068, 4295–4298,
4344, 4352, 4403, 4433, 4443, 4466, 4489
Extend 3D Polyline to Surface, 1496
Extend Arc, 140
Extend Bench, 4154
Extend by Distance, 141, 4327
Extended Bench, 4114, 4117
Extended Entity Data, 129, 1015, 3976, 4187
Extend Progress Sections, 1942, 1943
Extend Sections to Offset Limits, 1942
Extend To Elevation, 1470
Extend to Intersection, 140
Extract Breaklines, 3098
Extract Centroid Data, 3693, 4004
Extract Color Lines, 3101
Extract Contours, 3107
Extract Curb, 3109
Extract Intersection, 3110
Extract Plane, 3111
Extract Polyline, 3116
Extract Power Lines, 3113, 3114
Extract Profile, 3114
Extract Project Archive, 304
Extract Sections, 3118
Extract Slices, 3120, 3121
Extract Slope Edge, 3121, 3123
Extract Volume, 3123, 3124
Extrapolate, 324, 1428, 1465, 1540, 1561, 1564,
1565, 1578, 1584, 1585, 1591, 1596, 1603,
1732, 1868, 2160, 2892, 3301, 3340, 3370,
3413, 3504, 3522, 3523, 3528, 3530, 3532,
3545, 3551, 3560, 3585, 3782, 4006
Extrapolate Grid, 1648, 1651, 3509, 3514
Extrapolation, 1561, 1584, 1603, 3349, 3418, 3427,
3504, 3551, 3966, 4012

Fault Line, 3443

Fence Diagram, 35, 3369, 3370, 3372, 3392, 3393,
3435, 3438, 3441, 3442, 3444–3446, 3466,
3498, 3562, 3573, 3578, 3579, 3583, 3594,
3609, 3616, 3617, 4124, 4129, 4130, 4134,
4200
Fence diagram, 3392, 3435
Fence Polylines, 3445, 3446
Ferm Codes, 35, 3218
Field Menu, 2872
Field Module, 2869
Field Text, 245
Field to Finish, 26, 33, 35, 220, 222, 227, 253, 254,
372, 416, 455, 529, 656–658, 660, 662, 663,
667–669, 671, 672, 674, 685, 688, 694, 696,
718, 731, 734, 735, 738, 740, 741, 745–
747, 749, 750, 752, 763, 766, 772, 774, 796,
1524, 2519, 2878, 2980, 3032, 3081, 3135,
4318, 4321, 4322, 4324, 4325, 4415, 4417,
4419, 4731, 4733
Field to Finish for Faster Drafting, 4318, 4341
Field to Finish Inspector, 729
File Editor, 543, 544, 546
File Menu, 67, 493, 547, 558, 1373, 1403, 2614, 3015,
4045
File Selector, 37, 38, 40, 96, 99, 103, 4236, 4237,
4317
Fill-in Bed Names, 3298
Fill-in Missing Key as Zero, 3294
Fillet, 171, 189, 192, 942, 1463, 1464, 2103, 2116,
2124, 4347, 4433
Fillet 3D Polyline, 189, 1463
Final Sections, 1938
Find And Replace Data Values, 2443
Find Bad Angle, 530, 1218, 1354
Find Drillhole, 3273
Find Lot Name, 1023
Find Panel, 3729
Find Pit, 3951
Find Point, 280
Find Property Lines, 3691, 4002
Find Sewer Structure, 2417
Find Spoil Name, 4179
Find Trench Network Structure, 2777
Fit Image To Aspect Ratio, 2608
Fit Polylines, 1291
Fit Structure, 958, 959, 962, 963, 965–967, 969, 1290
Fit Text Inside Arc, 1113
Fit Text Outside Arc, 1114

Fix Coords, 1325
 Fix Label Overlaps, 1065
 Fix Point Attribute Overlaps, 427, 430
 Fix Text Overlaps, 161
 Flatten Spoil Top, 4135, 4136, 4141, 4199
 Flip Contour Labels-Text, 1555, 3363
 Flip Last Label, 1068
 Flip ON/OFF, 1068
 Flip Selected Labels, 1074
 Flip Solid Order, 3833
 Flip Text, 162, 661, 705, 739, 783, 915, 1055, 1056,
 1058, 1110
 Flip Text By Twist Screen, 162
 Flood Note, 362
 Flow Leader, 261
 FlyOver Along 3D Polyline, 2691
 Follow TIN Edges, 1469
 Footprint Creator, 943, 949
 Four Way Control, 888
 Freeze ALL Layers, 1366
 Freeze Layer, 209, 211, 402, 4301, 4317, 4357, 4402,
 4422, 4516, 4532, 4535, 4544
 Freeze Layer By Name, 216
 Freeze Layer By Pick, 211, 4675
 Freeze Layer By Selection, 208
 Freeze Point Attributes, 427
 Freeze Points, 402, 403

Gantt Charts, 4104
 General Commands, 66, 857–861, 3186, 3188
 Geodetic Double Break, 882
 Geodetic Double Proportion Line Division, 879
 Geodetic Middle Break, 882
 Geodetic Reports, 4705, 4712
 Geodetic Single Proportion Line Division, 882
 Geodetic Traverse, 879
 Geodimeter, 443, 462, 481–483, 496, 541, 1204,
 2886, 2887, 2959, 2960, 4466
 GeoFluv Channel Cross-Section Report, 3191
 GeoFluv Channel Inspector, 3192, 3193
 Geoid Utilities, 436
 Geolocate DWG Files, 122
 Geologic Column, 29, 35, 3259, 3470
 Geologic Column Profile, 2760
 geologic model, 316, 322, 3342, 3347, 3368, 3370,
 3398, 3401, 3436, 3442, 3444, 3488, 3593,
 3625, 4038
 Geologic Model Viewer, 3450
 Geologic Order, 3207, 3210, 3694, 4005

Geologic To Mining Model, 3371
Geology Module, 3205, 3284, 3418, 3430, 3458,
3462, 3465, 3496, 3589, 4011
Geometric Data, 2233, 2236, 2238, 2240
Geometry Specifications, 3654
GeoTech Drillholes and Strata, 4627, 4635
GeoTech Module, 2748
GeoTech Reports, 4675, 4690
Get Project from Data Depot, 78
GIS Database Settings, 2517, 2544
GIS Data Editor, 2521, 2539, 2543, 4599
GIS Data Menu, 2516
GIS File, 372, 2882
GIS Image Menu, 2598
GIS Inspector, 109, 2522
GIS Inspector Settings, 2521
GIS Links, 46, 2563
GIS Menu, 2956, 4598, 4606
GIS Module, 2515, 3424, 4018
GIS Query/Report, 2523
GIS Tools Menu, 2585
Global Edit, 1274–1276
Global Reannotate, 1074
GLO Corner Proportioning, 883, 887, 889
GPS Network, 635
GPS Vectors and Total Station, 639
Grade Parameters By Attribute Range, 3472
Grade Rules Editor, 1448
Grading, 4747
Grant Boundary Adjustment, 852
Graphical Method, 4544
Graphic Drillhole Report, 3304, 3310, 4683
Graphic Reports, 2695, 2698, 2699, 2728–2730
Graphic Scale, 1287, 1288, 3309
Grass Channel Design, 2294, 2295, 2304
GRD file, 1869, 2162–2164, 2606, 3025, 3370, 3853,
4142
Grid File, 121, 178, 307, 373, 512, 836, 1465, 1471,
1540, 1562, 1574, 1584, 1585, 1591, 1594,
1596, 1597, 1603, 1620, 1622, 1628, 1636,
1667, 1674, 1675, 1869, 2707, 2709, 3238,
3300, 3317, 3340, 3342, 3347, 3349, 3352,
3353, 3355, 3368, 3370, 3386, 3422, 3426,
3429, 3434, 3465, 3505, 3522, 3523, 3526,
3528, 3529, 3541, 3551, 3552, 3572, 3709,
3710, 3853, 3966, 4016, 4020, 4037, 4146,
4149, 4155, 4157, 4161, 4163, 4164, 4170,
4473, 4475

Grid File Utilities, 27, 35, 105, 115, 320, 1561, 1563,
 2140, 2162, 3338, 3349, 3352, 3369, 3370,
 3413, 3423, 3504, 3529, 3581, 3584–3586,
 3983, 4006, 4017
 Grid History, 4217
 Grid History Review, 3510
 Grid Inspector, 3371, 3586
 Grid Sequence File, 4216
 Grid Size, 1540, 3522
 Grids Menu, 3502
 Guard Rail, 1120

 Handicap Marker, 997
 Hatch GIS Polylines, 2525
 Hatch Pits, 3953
 Hatch Sub-Areas, 1008, 1009
 Hatch Subgrade Areas, 2669, 2671, 4616, 4662
 Hatch Wizard, 243
 Haul Cycle Analysis, 4093, 4094, 4098, 4102, 4195
 Haul Fleet Manager, 4091, 4093, 4094, 4100–4103
 Haul Road Manager, 4088, 4093, 4100, 4102, 4103,
 4195
 Heal Solid, 3826
 HEC-2, 35, 2139, 2230, 2355–2358, 2361, 2376,
 2388, 4578
 HEC-RAS, 1866, 2139, 2219–2221, 2223–2226,
 2228, 2230–2241, 3851
 HEC-RAS Manager, 2219
 Help Menu, 447, 557, 726, 804, 2139
 Highlight Crossing Breaklines, 1486
 Highlight Crossing Plines, 185
 Highlight Depression Contours, 1507, 1549, 3843
 Highlight Fault Lines, 3377
 Highlight Hard Breakline Polylines, 1495
 Highlight Index Contours, 1549, 1551
 Highlight Limit Polylines, 3394
 Highlight Non-Perpendicular Intersections, 184
 Highlight Non-Tangent Polylines, 184
 Highlight Segments by Slope, 1485
 Highlight Strata Polylines, 3390
 Highlight Tractive Force Zones, 3143
 Highlight Unclosed Plines, 3660
 Highlight Unclosed Polylines, 186, 3660, 3941
 Highwall Slopes, 3418, 4012
 Hinged Area, 933
 Horizontal Alignment, 2764, 2765, 3000, 3001, 3258,
 3260–3262, 3265, 3266
 Horizontal Axis Crossings, 1822, 1823
 Horizontal Axis Elevations, 1821

Horizontal Speed Table, 1690, 1723–1725, 3891
How to Setup ODBC Data Sources, 2583
HydroCAD, 2502, 2503, 2505, 2507, 2512, 2514,
4584–4586, 4590, 4592–4594, 4596
Hydrograph, 35, 2139, 2193–2196, 2198–2205,
2207–2217, 2243, 2249, 2328, 2383, 2384,
2424, 2425, 2465, 2466, 2505, 2514, 4594
Hydrograph Development, 2215, 2216, 4554, 4556
Hydrograph Editor, 2198
Hydrology and Watershed Analysis, 4530, 4556
Hydrology Module, 2138–2140, 2263, 2265, 2272,
2276, 2319, 2341, 2371, 2372, 2411, 2418,
2443, 2446
HydroNet Explorer, 2502, 2503, 2505, 2507, 2512,
4593, 4594
Hypothetical, 3553

Identify Area Descriptions, 932
Identify Fault Polylines, 3378
Identify Hard Breakline Polylines, 1495
Identify Limit Polylines, 3394
Identify Pipe Polylines, 1850
Identify Pit Polylines, 3954, 4052, 4231
Identify Predefined Boundaries, 1498
Identify Property Lines, 3689, 4000
Identify Sewer Network Entity, 2426
Identify Strata Polylines, 3391
Identify Sub-Area, 1008
ID Linework, 1015
ID Solid, 3834
Image 3D Viewer, 2605, 2606
Image Drape, 3124, 3125
Image Inspector, 679, 757, 2609, 2611
Image Pan, 2622
Image Redraw, 2622
Image Set Manager, 2614, 2618
Image Zoom Extents, 2622
Image Zoom Previous, 2622
Image Zoom Window, 2622
Import/Export Carlson Triangulation Files, 2705
Import/Export Trimble TTM File, 1683
Import 12D File, 96
Import 3D Viewer File, 93
Import ADAC, 2458
Import Angle/Distance File, 809
Import Architectural IFC File to Drawing, 2857
Import Architectural IFC File to Model, 2857, 2860
Import ASCII File into Coordinates, 1145
Import Bed Names, 3232

Import Block Model, 3467
Import Carlson Sewer Network, 2453
Import Cloud, 3045, 3047
Import Coal Sections From Text File, 3673
Import Dip/Azimuth, 3233, 3243
Import Drillhole, 3220
Import ePlan File, 100
Import ESRI GeoDatabase, 2564
Import ESRI SHP File, 2562
Import Fault Line, 3373
Import File To Solid, 3820
Import Flow Depth Points, 2238–2241
Import Flow Velocity Points, 2236–2238
Import GIS Data from SurvCE, 2566, 4606
Import Google Earth File, 97, 1683, 2603
Import Google Earth Surface, 1682, 1683
Import Haestad Network, 2339
Import InRoads Settings, 99
Import LandXML File, 93
Import Laterals From Text File, 2410
Import Lot File From MDB Database, 1041
Import MicroStation DGN, 2718
Import Mine Plan, 3733, 3734
Import MrSID Images, 2605
Import MxRoad GENIO, 98
Import NASA World Elevation Dataset, 1681
Import National Elevation Dataset, 1682
Import Network From 3D Polylines, 2454
Import Network From Bentley, 2458
Import Network from Centerline/Profile, 2457
Import Network From Text File, 2458
Import ODBC data source, 2566
Import OpenGIS GML, 2565
Import PDF File, 2862
Import Pit Points, 3968, 3969, 4051
Import Pit Timing Data, 3984, 3986, 3989, 4051
Import Qualities, 3231, 4681
Import Raster To Vector, 2852
Import Removal Text ASCII File, 2685
Import RoadXML File, 95
Import Sewer Network From 2D Polylines, 2455
Import Sewer Network From Esri Shapefiles, 2460
Import SketchUp File to Drawing, 2856
Import SQLite, 2561
Import StormCAD Network, 2460
Import Text/ASCII File, 33, 378, 380, 385, 388, 657,
734, 854
Import Trench Network Data - Utility Network, 2777

Inclusion Polylines, 3181
Indicated, 1772, 3553, 3555, 3556
Inferred, 3553, 3555, 3556
INI Files, 31, 306
Inlet Library, 2355, 2381, 2382, 2454–2456, 4570
InnerSpace Tech depth sounder, 2960
Input-Edit Point, 377, 392
Input Sewer Line, 2374, 2375
Input Trench Line, 2771, 4638, 4639
Inquiry Menu, 275
InRoads, 99, 100, 682, 760, 1720, 1883
Insert Block with GIS Data, 2582
Insert Mining Symbols, 3622
Insert Multi-Point Symbols, 253
Insert Symbols, 250, 253, 307, 351, 4300
Insert with Join/Align, 2597
Inspect eLog, 3285
Installation Guide, 3, 11, 12
Instruction Manual and Program Conventions, 54
Instrument Data Project Items, 3057
Interior Point, 193, 243, 917, 1525, 2167, 4359,
4538–4540
Interior Points Filter, 3823, 3824
Intermediate Contours, 1345
International Roughness Index, 1855
Interpolate Points, 901–904, 1457
Intersection Points, 26
Intersections, 868, 942, 943, 1016, 1236, 1265, 1278,
1369, 1512, 1513, 2061, 2062, 2067, 2077,
2078, 2095, 2096, 2124, 2677, 3571, 3847,
3849, 4235
Intersections/Cul-de-sacs, 1259–1262
Intersections and Subdivisions, 4341, 4366
Intersects, 1233, 1234, 1239, 1240, 1728
Intersect Solids, 3833
Interval Along Entity, 903, 4168
Interval Between Points, 903
Introduction, 1, 514, 550, 3164, 3344, 4245
Invalid Strata Report, 3317
Inverse, 855, 856, 858–860, 862, 878, 920, 1231,
1425, 2038, 2643, 2750, 2837, 2850, 3343,
3456, 3457, 3463–3465, 3572, 3585, 3586,
3588, 3613, 3745, 3767, 3933, 4039, 4056,
4255, 4256, 4267, 4268, 4279, 4280, 4343,
4345, 4346, 4629, 4677
Inverse with Area, 918, 920
Irregular Boundary Adjustment, 880
Isolate Layer, 209, 212, 373, 1551

Isolate Layer By Name, 216
Isolate Layers, 2596, 3765, 3766, 3768, 3770, 4293–
4295, 4301, 4537, 4541
Item Properties, 3140
Item Standards Manager, 265

Join, 117, 163, 166, 182, 191, 192, 676, 719, 754, 797,
1433, 1437, 2043, 2217, 2636, 2657, 3650,
3652, 3918, 4408, 4445, 4553, 4610
Join 3D Polyline, 190, 1464
Join Nearest, 169, 171, 182, 191, 1464, 1510, 1698,
1824, 2862, 3100, 3846, 3899, 4345, 4408,
4428, 4444, 4452, 4669
Join Polyline By Trace, 182
Join Text Entities, 165
Join Two Polylines, 171

Key Strata Missing Bed, 3314
Kriging, 3344, 3345, 3403, 3409–3411, 3431, 3432,
3455, 3462, 3586, 3607
K Value, 1859

Label/Draw Right of Way, 1722
Label Angle, 27, 1090, 1091, 1098, 1120, 1124, 1454
Label Arc, 1105–1107, 4306, 4331
Label Block Model, 3476
Label Contours, 1345–1347, 1510, 2630, 2744, 3845
Label Coordinates/Elevation, 1121, 1123
Label Curb Flow Elevations, 1125
Label Drillhole, 2756, 2764, 3265, 3272, 4688
Label Elevations Along Polyline, 1479, 1481, 1482
Label GIS Point Data, 2537
Label GIS Polyline: Closed Polyline Data, 2541
Label GIS Polyline: Closed Polyline Image, 2539
Label GIS Polyline: Label Along Polylines, 2542
Label Last Area, 923
Label Lat/Long, 854
Label LatLong, 1124
Label Object Data Areas, 2588
Label Offset Distances, 1128
Label Pad Elevation, 1477
Label Pipe Polylines, 1851
Label Pit/Site Names, 3951
Label Point Comparison, 1123
Label Polyline High/Low Points, 1482, 1483
Label Polyline Segment, 1483
Label Profile Differentials, 1853
Label Profile On Centerline, 1835, 1839, 2042
Label Projection Distances, 3658

Label Property Lines, 3690, 4001
Label Sewer Laterals, 1854
Label Solid, 3828
Label Spoil Names, 4179
Label Station-Offset, 1711
Label Sub-Areas, 1009
Label Subgrade Areas, 2672
Label Zeros, 374, 660, 738
Landuse, 2141
LandXML Data Transfer, 4747
Language Localization, 55
Laser Atlanta, 2960
Lat/Lon and Grid Projections API, 4767
Lateral Design Overview, 2398
Latitude/Longitude, 197, 381, 460, 856, 1655, 3027,
3029, 3035, 3137
Latitude and Longitude, 2906
Layer and Style Defaults, 34
Layer by Description, 373
Layer Control, 1366–1368, 3657, 3768, 3770, 4290,
4299, 4325, 4333, 4348, 4546
Layer ID, 276, 4325, 4613, 4614
Layer Inspector, 277
Layer Library, 352–355
Layer Properties Manager, 3129
Layer Report, 277, 285
Layout Manager, 81–83, 245
LDD, 117, 438, 439, 1359, 4753
LDT Migration Guide, 4729
Leaders, 1306, 1484, 3264, 3975, 4186
Leader With Text, 258, 259
Least squares, 550, 552, 555, 625, 2751, 3344
Legal Description Writer, 817–819, 1035
Legend Definition, 1081, 1083, 4312
Leica, 442, 461, 476–480, 496, 497, 547, 564, 601,
609, 665, 743, 1195, 1861, 1863, 2887,
2893, 2895, 2935, 2961–2963, 2981, 3045
Leica Disto, 2961
Leica Format, 663, 741
Leica GPS System 500, 2961
Leica TC Series, 2962
Level Manager, 207, 1767
License Agreement, 61
License Models, 12
Lift Station Design, 2320
Limitations of, 3145, 3146
Limit Polylines, 321, 3380, 3392–3394, 3578, 3584
Line and Curve Labeling, 4739

Linear, 317, 1791, 1958, 2247, 2253, 2254, 2309,
2560, 2663, 2750, 3344, 3346, 3427, 3699,
3943, 3944, 4026, 4629, 4677
Linear 3D Object Polylines, 1665, 1666
Line by Angle-Distance, 878
Line of Sight Viewshed, 1666
Line On/Off, 861, 865
Lines, 1370, 1402, 1420, 1441, 1442, 1500, 1512,
1513, 1519, 1523, 1532, 1533, 1536, 1559,
1645, 1700, 1704, 1707, 1710, 1729, 1739,
1771, 1778, 1795, 1796, 1802, 1805, 1867–
1869, 1911, 1946, 1985, 2030, 2095, 2153,
2255, 2256, 2418, 2428, 2590, 2625, 2629,
2632, 2650, 2686, 2744, 2752, 2761, 2776,
2862, 3076, 3084, 3103, 3104, 3224, 3251,
3262, 3342, 3392, 3437, 3441, 3443, 3468,
3502, 3518, 3655, 3752, 3774, 3836, 3847,
3849, 3879, 3900, 3911–3913, 3915, 3916,
3933, 4075, 4166, 4171, 4235, 4256, 4268,
4280, 4290, 4316, 4330, 4332, 4491, 4498,
4690, 4739
Lines and Polylines, 1288–1290, 1292
Lines by Codes, 1290
Lines by Description, 1289
Lines by Point Number, 1288
Line Table, 1055, 1090, 1091, 1093, 1094, 1098, 1099
Line Type Scaler, 288
Line Up Text, 165, 853
Linework Intersection Points, 868
Linework Length Report, 2725
Link Linework With Points, 434
Link Points with Coordinate File, 369
Links Manager, 391, 2543
Links with Other Software, 3157
LISCAD, 479, 480
LisCad, 442
List, 1325, 1330, 1338, 1349, 1564, 1571, 1681, 1888,
1890, 1893, 1993, 2023, 2025, 2398, 2410,
2414, 2415, 2480, 2520, 2686, 2752, 2776,
2868, 2885, 2932, 2934, 2949, 3050, 3082,
3209, 3250, 3405, 3492, 3530, 3538, 3605,
3703, 3733, 3757, 3856, 3858, 3861, 4030,
4062, 4079, 4122, 4345, 4425, 4601, 4603,
4613, 4618, 4723, 4758, 4764
List Points, 288, 370, 376, 378, 385, 386, 661, 739
Load Image Set, 2621
Loading Carlson Menus, 21
Load Last Report Formatter, 286

Load Saved Report, 287
Load Tablet Calibration, 2796
Local Elevation Label, 1554
Locate by Azimuth, 866, 3625, 3932
Locate by Bearing, 866, 867, 871, 872, 3625
Locate by Delta, 867
Locate by Line Bearing, 865
Locate by Turned Angle, 866
Locate on Real Z Axis, 371, 374, 905, 956, 2878,
4342
Locate Point, 33, 131, 371, 372, 375, 427, 490, 660,
738, 854, 867, 3939
Locate Reach, 2215
Locate Structures, 4555
Locate Structures TR20, 2215
Locate Template Points, 2133
Lock Layers, 217
Lock Points, 403
Lot File, 817, 1002, 1020, 1026, 1028–1032, 1035,
1039, 1041–1043, 4357, 4358, 4360, 4362,
4363, 4366
Lot File by Closed Linework, 1030
Lot File by Interior Text, 1030
Lot File by Pick Interior, 1029
Lot File to Centerline, 1042
Lot Inspector, 1035
Lot Layout, 938–941, 4349, 4350, 4353
Lot Manager, 1031, 1032, 4361–4363, 4366
Lot Network - Assign Lot Type, 1025
Lot Network Boundary, 1007
Lot Network Hinged Area, 1019
Lot Network Inspector, 1022, 1023
Lot Network Labels, 1020
Lot Network Linework, 1014–1016
Lot Network Output To Lot File, 1026
Lot Network Renumber Lots, 1023
Lot Network Report, 1020, 1021
Lot Network Road Network, 1010
Lot Network Settings, 1002, 1019, 1020, 1022, 1023
Lot Network Sliding Side Area, 1019
Lot Network Subdivide Area, 1017
Lot Number Report, 1021
Lots, 4747
LSP Functions, 4752
LWPolyline to 3D Polyline, 1435

Main Contours, 1345
Make 3D Grid File, 1514, 1559, 1571, 1584, 1593,
1619, 1621, 1627, 1628, 1643, 1671, 2139,

2143, 2160, 2276, 3502, 3525, 3538, 3551,
 3767, 3768, 3849, 4164, 4169, 4471, 4472
 Make Arcs Tangential, 183
 Make Block Model, 3425, 3452, 3458, 3459, 3466,
 3607, 4019
 Make Combined Removal Surface, 2689
 Make Design Surface, 2678, 2692, 2709
 Make Existing Ground Surface, 2676, 2677, 2709,
 2780, 4621, 4627, 4636, 4664, 4675
 Make Grade Solid Model, 3474
 Make Layout, 2727
 Make Nearest Data Point Grid, 3507
 Make Overexcavate Surface From Existing/Design Sur-
 faces, 2680
 Make Overexcavate Surface From Screen Entities,
 2680
 Make Overexcavate Surface From Strata, 2680
 Make Overexcavate Surface From Surface File, 2680
 Make Removal Surface, 2688–2690
 Make Solid From Design, 3785, 3786, 3887, 3888
 Make Solid From Entities, 3786
 Make Solid From Points By 3D Triangulation (Ad-
 vancing Front), 3782
 Make Solid From Points By 3D Triangulation (Pro-
 jection), 3778
 Make Solid From Points By 3D Triangulation (Tun-
 nel), 3780
 Make Solid From Points By Poisson, 3773
 Make Solid From Points By Sections, 3776
 Make Solid From Polylines, 3784
 Make Solid From Sections, 3770, 3772
 Make Solid From Surfaces, 3762, 3770
 Make Solid From Survey Points, 3772
 Make Strata Grid Files, 3339, 3353, 3563, 4146
 Make Strata Surface, 2760, 2766–2768, 4632
 Make Top Of Key Grid, 3506
 Make Top Surface, 2682, 2684
 Make Unfaulted Strata Grid, 3386
 Make User Defined Surface, 2705
 Making a Plat, 4281, 4318
 Mandoors, 3656
 Manhole, 1197, 1198, 1324, 1750, 1751, 1756, 1802,
 1803, 1834, 1846, 2331, 2337, 2362, 2422,
 2458, 2460, 2547, 2772, 2773, 2781, 2787,
 2790, 4311, 4324, 4638, 4647
 Manual Storage, 1323
 Manual Total Station, 2964
 Map Check by Pnts, 828

Mapcheck by Screen Entities, 830
Map Menu, 2768
Map Points from, 385, 388
Maps, 108, 1562, 2140, 2187, 2252, 3356, 3505, 4740
Maptech, 496
Mark Compliance Drillholes, 3328
Mark GIS Polylines, 2527
Mark Insertion Points, 2596
Mask Point Attributes, 426
Mass Diagram, 28, 1923, 1924, 1926, 2036
Mass Diagram Output File, 1953
Mass Diagram Report, 1924, 1955
Mass Haul Analysis, 1925, 1926, 1928
Mass Haul Report, 1924, 1925
Material Quantities, 2718, 2720, 2725–2727, 4625,
4626, 4665, 4666
Max Samples, 312, 317, 3343, 3350
Measured, 480, 589, 2935, 3241, 3280, 3552, 3553,
3555, 3556
Median Offset, 137, 854
Menu System Overview, 555
Merge Closed Polylines, 171, 172
Merge Clouds, 3049
Merge Crossing 3D Polylines, 1464, 1465
Merge Elevation for Zero Thickness, 3552
Merge Existing With Design, 2710
Merge Grid Files, 1584, 3551
Merge Grids, 1568, 3535
Merge PDF Files, 85
Merge Pits, 3969
Merge Profiles, 1848
Merge Raster Files, 2843, 2845
Merge Sections, 1948
Merge Small Benches, 3980, 3982
Merge Solids, 3829
Merge Spoils, 4181
Merge Stage-Discharge Files, 2286
Merge Tables, 1103
Merge Watersheds, 2159
MicroStation, 105, 114, 127, 128, 1883, 2718
Microstation DGN, 113
Middle Ordinate Solution, 1250, 1353, 1369
Mikrofyn Lightbar, 2964
Min/Max Slopes 3D Polyline, 1469
Mine Map Insert Commands, 3631
Mine Name, 3632, 3671, 3914, 3943
Mine Note Auto Left/Right, 3628, 3629, 3664, 3939
Mine Note Defaults, 3625

Mine Note From CoorD File, 3628
Mine Note Left/Right/Face, 3627, 3628
Mine Note on Centerline, 3629
Minimum Frontage, 1013, 1018, 1022
Mining Project Settings, 3206, 3694, 4004
Min Quadrants, 3343
Mirror, 44, 963, 965, 969, 970, 986, 1067, 1068, 1941,
2846
Mirror and Flip Selected Labels, 1068
Mirror Image, 2846
Mirror Selected Labels, 1067
Missing Strata Report, 3318
Model 3D Viewer, 2860
Modeling Options, 3588
Monitor GPS Position, 2930, 2958, 2959
Mortgage Block, 362
Motion 3D Viewer, 1663
Move, 1279, 1295, 1296, 1307–1309, 1317, 1395,
1423, 1479, 1481, 1524–1526, 1531, 1555,
1633, 1641, 1689, 1801, 1817, 1949, 1971,
1978, 2339, 2380, 2390, 2450, 2523, 2573,
2581, 2609, 2641, 2662, 2683, 2691, 2701,
2703, 2716, 2717, 2751, 2752, 2754, 2873,
3077, 3095, 3275, 3278, 3350, 3354, 3363,
3370, 3434, 3438, 3459, 3471, 3602, 3635,
3890, 3952, 4122, 4240, 4243, 4251, 4252,
4263–4265, 4275–4277, 4309, 4311, 4313,
4329–4331, 4334, 4579, 4612, 4629, 4636,
4661, 4677
Move Attributes with Leader, 156, 166
Move Calls, 1295
Move Elevation Labels, 1481
Move Label Along Contour, 1555, 3362
Move Label with Leader, 1069
Move Point Attributes, 430
Move Point Attributes Single, 422
Move Point Attributes with Leader, 423, 430
Move Points, 219, 411
Move Section Leader Labels, 1949
Move Sewer Label, 2438, 2439
Move Sewer Profile Labels, 1817
Move Sewer Profile Pipe Labels, 2449
Move Spot Elevations, 1455
Move Text, 154, 155
Move Text with Leader, 155
Multi-Draw, 1282, 1283
Multi-Point Symbols, 253, 256
Multileader with Text, 244

Multiple in Series, 4617
Multiple Offsets, 137, 674, 752
Multiple Outlet Design, 2269–2271
MXS File, 1869, 1871, 1872, 1874–1876, 3119, 3793,
3853, 3855, 4498

N-E Line, 3632
NAD83, 1271, 1350
Name Limit Polylines, 3391
Name Pit Polylines, 3950
National Geodetic Survey, 412, 601, 2876
Natural Regrade File, 3161
Natural Regrade Global Settings, 3162, 3163, 3168,
3174, 3178, 3182
Natural Regrade Menu, 3160
Natural Regrade Module, 3142
Navcom Configuration Guide, 2964
Navcom GPS Setup, 2966
Nearest Found, 275, 673, 751, 1871, 3765, 3855
Negative Image, 2849
Network License Monitoring, 17
Network License Setup, 15
Network Menu, 2322
Network Processing Reports, 620
New, 67–69, 82, 146, 150, 166, 174, 187, 192, 207,
211, 223, 224, 228, 229, 341–344, 362, 368,
372, 375, 384, 385, 390, 394, 405–407, 413,
460, 480, 494, 501, 503, 523, 547, 558, 571,
607, 684, 713, 762, 791, 833, 849, 851, 861,
880, 881, 905, 927, 961, 1017, 1126, 1131,
1158, 1195, 1303, 1306, 1326–1329, 1334,
1348, 1357, 1372, 1373, 1376, 1379, 1401,
1407, 1413, 1445, 1454, 1470, 1518, 1520,
1575, 1724, 1738, 1739, 1742, 1746, 1750,
1790, 1869, 1875, 1876, 1888, 1900, 1982,
2012, 2033, 2064, 2073, 2092, 2105, 2112,
2113, 2179, 2181, 2192, 2269, 2270, 2341,
2345, 2360, 2362, 2370–2373, 2378, 2395,
2473, 2477, 2501, 2521, 2529, 2532, 2545,
2546, 2573, 2574, 2577, 2578, 2611, 2635,
2845, 2917, 2938, 2955, 2966, 2982, 2994,
3006, 3020, 3028, 3030, 3033, 3036, 3039,
3042, 3049, 3054, 3057, 3060, 3064, 3091,
3094, 3125, 3131, 3132, 3136, 3138, 3154,
3159, 3160, 3166, 3181, 3209, 3506, 3542,
3552, 3624, 3634, 3637, 3700, 3701, 3753,
3787, 3814, 3856, 3868, 3932, 3957, 3990,
4027, 4028, 4076, 4102, 4130, 4210, 4211,
4216, 4220, 4237, 4246, 4259, 4271, 4284,

4299, 4311, 4315, 4318, 4325, 4343, 4349,
4363, 4367, 4386, 4422, 4496, 4498, 4509,
4518, 4564, 4565, 4649, 4718
New/Startup Wizard, 31
New Area Table, 927
New File, 40
Nikon, 461, 480, 481, 496, 541, 1207, 1208, 2969,
2970
Nikon Total Stations, 2969
Nodes, 372, 373, 3053, 3664, 4595
Non-Surface Entities, 1494
North Arrow, 1776, 1820, 2868, 3914, 4309, 4689
North Rotation, 4749
Note File, 835, 1883, 2577, 2578, 2879, 2882
Notes Menu, 3622
Numeric, 308, 855, 1038, 1130, 1158, 1189, 1310,
1407, 2853, 2854, 3291, 3293, 3671, 3672,
3943, 3944, 4282, 4753
Numeric Pad COGO, 877

Object Data, 323, 2518, 2519, 2521, 2588, 2589
Object Height, 313
Object Linking, 130, 308, 1064
Obtaining Technical Support, 22
Occupy Point, 857, 862
ODBC, 46, 2518, 2519, 2522, 2553, 2566, 2567,
2583, 3209, 3686, 3997
Off, 1370, 1385, 1386, 1391, 1396, 1402, 1405, 1406,
1410, 1422, 1423, 1445, 1446, 1453, 1454,
1456, 1479, 1502, 1506, 1507, 1510, 1518,
1536–1538, 1553, 1555, 1564, 1633, 1638,
1641, 1652, 1706, 1708, 1713–1719, 1729,
1732, 1739, 1747, 1748, 1755, 1761, 1763,
1767, 1770–1773, 1775, 1777, 1778, 1789,
1790, 1794, 1796, 1803, 1804, 1810–1813,
1820, 1826, 1827, 1830, 1845, 1852, 1855,
1865, 1868, 1870–1872, 1877, 1879, 1880,
1887, 1889–1891, 1899–1901, 1903, 1907–
1912, 1914, 1915, 1918, 1919, 1921, 1930,
1935–1938, 1941, 1946, 1948, 1950, 1960–
1964, 1967, 1969, 1970, 1984, 1993, 2018,
2020, 2023, 2028, 2029, 2038–2041, 2043,
2065, 2066, 2073, 2074, 2088, 2095, 2103,
2134–2136, 2173, 2221, 2324, 2326, 2337,
2364, 2379, 2381, 2382, 2398, 2408, 2413,
2456, 2519, 2539, 2572, 2629, 2630, 2636,
2640, 2641, 2650, 2673, 2734, 2743, 2745,
2753, 2764, 2765, 2775, 2783, 2813, 2814,
2816, 2828, 2830, 2877, 2879, 2880, 2886,

2887, 2892, 2913, 2918–2921, 2927, 2933,
 2939, 2943, 2946, 2949, 2954, 2967, 2975,
 2981, 3051, 3108–3110, 3119, 3121, 3128,
 3160, 3251, 3253, 3256, 3265, 3266, 3268,
 3350, 3354, 3362, 3416, 3421, 3434, 3436,
 3437, 3441, 3518–3520, 3531, 3626–3629,
 3643, 3645, 3646, 3652, 3656, 3665, 3785,
 3838, 3842, 3846, 3851, 3854, 3855, 3857–
 3859, 3867–3869, 3871, 3875–3880, 3882,
 3883, 3886, 3887, 3902, 3904, 3935, 3938,
 3939, 4010, 4015, 4061–4063, 4119–4122,
 4137, 4150, 4152–4154, 4173, 4235, 4285,
 4295, 4298, 4302, 4304, 4305, 4334, 4336,
 4344, 4346, 4349, 4350, 4354, 4357, 4407,
 4443, 4520, 4522, 4610, 4613, 4614, 4617,
 4618, 4620, 4694, 4698, 4700, 4708
 Offset 3D Polyline, 187, 1462, 1471, 1824, 2051,
 4336, 4407, 4428, 4444, 4451
 Offset 3D Polyline By Grade Table, 2733
 Offset & Elevation Report/Plot, 1920
 Offset Cutoff, 115, 117, 182
 Offset Dimensions, 27, 1075–1077
 Offset Point Entry, 27, 241, 1715
 Offset Points By Interval, 907
 Offsets & Intersections, 941, 943
 Offsets from ASCII File, 3629
 Offset Stakeout, 2947
 Offset Strata Polylines, 3389
 Offset to Area, 136
 Offset To Layer, 136
 OmniStar Otto, 2970
 One Grid Surface Volumes, 1591
 One Surface Volumes, 3522
 One Triangulation Surface Volumes, 1585
 One Way Control, 884
 Open, 1132, 1133, 1135, 1146, 1154, 1155, 1212–
 1214, 1217, 1222, 1223, 1225, 1230, 1258,
 1282, 1332, 1344, 1372, 1373, 1376, 1378–
 1380, 1384, 1385, 1401, 1405–1408, 1413,
 1508, 1541, 1553, 1575, 1628, 1656, 1681,
 1727, 1761, 1762, 1765, 1897, 1902–1904,
 1906, 2032, 2168, 2263, 2265, 2287, 2292,
 2356, 2357, 2497, 2522, 2556, 2565, 2566,
 2575, 2578, 2604, 2609, 2614, 2619, 2620,
 2795, 2797, 2857, 2858, 2860, 2861, 2868,
 2891, 2972, 2977, 2987, 3004, 3006, 3012,
 3029, 3033, 3045, 3059, 3061, 3078, 3091,
 3096, 3097, 3101, 3107, 3109, 3111, 3113,

3114, 3116, 3118, 3120, 3121, 3123–3125,
 3130, 3132, 3136, 3161, 3169, 3180, 3181,
 3197, 3354, 3362, 3542, 3637, 3667, 3844,
 3865, 3870–3872, 3874, 3903, 4107, 4236,
 4237, 4239, 4284, 4285, 4317, 4320, 4321,
 4340, 4341, 4343, 4358, 4363, 4370, 4386,
 4396, 4404, 4411, 4414, 4416, 4429, 4435,
 4439, 4446, 4448, 4456, 4458, 4464, 4465,
 4473, 4475, 4479–4481, 4483–4485, 4487,
 4488, 4496, 4498, 4506, 4507, 4511, 4514,
 4517, 4518, 4522, 4530, 4532, 4533, 4542,
 4546, 4556, 4557, 4560, 4566, 4567, 4584,
 4585, 4587, 4590, 4598, 4600, 4606, 4608,
 4635, 4641, 4668, 4672, 4673, 4679, 4681,
 4684, 4691, 4692, 4696–4699, 4705, 4712,
 4714, 4722, 4754, 4755, 4757
 Open Dos Drawing, 1147
 Open Drawing, 122, 301
 Open Polylines, 136
 Optimized Pit Design, 3496, 3611, 3618, 3619
 Order Table, 1105
 Ore Body, 3558
 Orifice Design, 2265–2267, 2269
 Other Drawing, 2740–2742, 4612, 4613, 4659, 4661
 Outcrops, 3570, 3577
 Output Coordinate File, 2036, 2041
 Output Layouts to PDF, 84
 Output Tab, 3176
 Output To Reame, 4170
 Output to SB-Slope, 4172
 Overburden, 3225, 3349, 3453, 3460, 4231
 Overland Sheet Flow Surface Mannings N Library,
 2371
 Overlay Section File, 1946
 Ownership, 36, 3681, 4162

 Pad Polyline By Interior Text, 1438, 2650
 Panel & Room Label Block, 3658
 Panel Attributes, 3706, 3710, 3712, 3714, 3716, 3717,
 3917, 3923, 3925, 4033
 Panel Precedence By Polylines, 3727
 Panels Report, 3739
 Parking, 954, 955, 971–975, 977, 983, 986–988, 990–
 1002, 2186
 Parking Buffer Zone, 983
 Parking Design, 985, 996
 Parking Island, 996, 997
 Parking Network, 998
 Parking Perimeter Tools, 973, 983

Patch Management, 59
Pattern Point Survey, 2934
Pavement Mannings N Library, 2372
PC, PT, Radius Length, 230
PC, PT, Radius Point, 230
PC, PT, Tangent, 231
PC, Radius, Arc Length, 231
PC, Radius, Chord, 231
PCMCIA card, 480
PC Point, 232
PDF Menu, 2861
PDF Section Import, 4667, 4674
PDMODE, 440
PDSIZE, 307, 369
Peak Flow - Graphical Method, 2186
Peak Flow - Modified Rational Method (LA County),
2192
Peak Flow - Rational Method (General), 2187
Peak Flow - Rational Method (KYDOT), 2189
Peak Flow - Rational Method(Riverside S. Califor-
nia), 2188
Peak Flow - Tabular Hydrograph Method, 2190
Perimeter Polylines Properties, 2676
Perpendicular Intersect, 872, 941, 1236
Pick & Place Panel, 3715, 3923
Pick Current Layer, 1367
Pick Intersection Points, 867
Pick Layers to Freeze, 1366
Pick Layers to Thaw, 1366
Pick Layers to turn Off, 1367
Pick Layers to turn On, 1367
Pillar Cut, 3666, 3667, 3946, 3947
Pinch Out, 3589
Pipe Connector, 2858, 2859
Pipe Culvert Design, 28, 2304–2306
Pipe Depth Summary, 1833
Pipe Elevation Label, 2440
Pipe Manning's n Library, 2373, 2386
Pipe Network, 2935, 4235, 4714, 4747
Pipe Network Drawing Links, 2450
Pipe Network ID, 2451
Pipe Polylines, 1876
Pipe Routing Hydrograph, 2194, 2207
Pipes, 4747
Pipe Size, 36, 717, 795, 1811, 1828, 2311, 2325,
2329, 2386, 2392, 2400, 2406, 2408, 2410,
2411, 2455, 2457, 2470, 2772, 2773, 2776,
4579, 4638, 4647

Pipe Size Library, 1763, 2366, 2369, 2379, 2386,
2392, 3905, 4579
Pipe Width, 1735, 1849, 1876, 1939
Pit/Channel Samples - Import from Coal Section,
3246
Pit/Channel Samples - Import from CRD File, 3245
Pit/Channel Samples - Import from Text File, 3244
Pit Areas, 3417, 3429, 4010
Pit by Interior Point, 3417, 3955, 4010, 4051
Pit By Interior Text, 3956
Pit Directions, 4216
Pit Inspector, 3994, 4052
Pit Label Formatter, 3951
Pit Layout, 3577, 3962, 3966, 4216
Pit Layout by Advance, 3963, 4051, 4216
Pit Layout by Rate, 3417, 3966, 4010, 4051
Pit Layout by Width, 3965, 4051
Pit Matrix Layout, 3957, 4051
Pit Plines from Mineplan, 3957
Pit Points Report, 3987, 4052
Pit Polylines, 3577, 3951, 3953, 3964, 3979, 4075,
4216
Pit polylines, 3954, 3982, 4145
Pit Precedence by Polylines, 3977
Pit Quantities Report, 3986, 4052
Pit Scheduler, 4045, 4046, 4048
Pivot Point Rotate by Bearing, 150
Place Calls, 1293–1295, 1364
Place Camera Symbol/Image, 2611
Place Coal Sections, 3672, 3674, 3677, 3944, 4163
Place Drillhole, 2038, 2751, 2753, 2757, 2760, 3206,
3213, 3694, 4004, 4629, 4632, 4679
Place Drillholes, 3236, 3318
Place Google Earth Image, 2601, 2603
Place Image By Circle, 2622
Place Image By Point, 2621
Place Image by World File, 2599, 2600, 2608
Place Image On Layout, 2728
Place Labels, 1345, 1783–1785, 1788, 1913, 3881,
4177
Place Panel, 3626, 3693, 3712, 3713, 3715, 3719,
3720, 3726, 3739, 3740, 3919, 3920, 3922,
4022
Place Pit/Channel Sample, 3248
Place Web Map Service (WMS) Image, 2603
Plain View Label Settings, 2778
Plan View Label Settings, 2333, 2451, 2779, 2935
Plan View To PDF, 2768

Plot Points and Symbols, 1286, 1287, 1289
Plot Points on Screen, 1286
Plot VC from Tangents, 1859
Point Attribute Layout Manager, 435
Point Check, 1016
Point Check By Robotics, 2935
Point Cloud Recommended Specifications, 2980
Point Cloud Setup, 2981
Point Clouds Getting Started, 2980
Point Clouds Module, 2979
Point Clouds Project Manager, 2992
Point Clouds Step-by-Step Tutorial, 2981
PointCloud Viewer, 3012
Point Code, 1368, 1411, 3084
Point Defaults, 42, 220, 222, 227, 251, 307, 309, 370,
372, 420, 433, 691, 769, 859, 860, 867, 872,
879, 884–886, 888, 903, 1003, 4245, 4258,
4270, 4287, 4288
Point Description, 94, 131, 280, 365, 375, 393, 437,
438, 662, 701, 713, 740, 779, 791, 860, 866,
867, 901, 902, 910, 1034, 1126, 1278, 1286,
1289, 1488, 1714, 1786, 1873, 1942, 1992,
1996, 2803, 2819, 2882, 2941, 3934, 4247–
4249, 4260, 4261, 4272, 4273
Point Editor, 3134
Point Entity, 385, 419, 420, 727, 805, 1732
Point Entity Grouping, 130
Point GIS Editor, 2547
Point Group Manager, 394, 398, 401, 669, 747, 818,
1733
Point Groups, 1340, 1342, 1512, 3847, 4737
Point ID, 276, 385, 389, 709, 787, 1166, 1167, 1189,
1227, 1241, 1257, 1278, 1302–1305, 1326,
1337, 1347, 1371, 1382, 1396, 1397, 1406,
1407, 1409, 1410, 1993, 1994, 1996, 2547,
2548, 2550, 2562, 2564–2567, 2891
Point Label, 1308, 1337, 1837
Point Label Settings, 364, 660, 738
Point Layer, 669, 687, 701, 747, 765, 779, 910, 2018,
3822
Point Manager, 1313, 1316, 1317, 1320
Point Notes, 377, 383, 512, 525, 661, 695, 739, 773,
1718, 2877, 2878, 2941
Point Number Report, 385, 390
Point Object Snap, 311, 364
Point Offset Report/Plot, 1887
Point on Arc, 900
Point Placement on Profile, 1842

Point Placement on Section, 1885

Point Protect, 94, 97, 102, 380, 393, 469, 471, 472,
474, 475, 478, 481, 482, 511, 525, 578, 859,
2563, 2686

Point Range, 36, 387, 467, 663, 741, 1454

Points, 1370, 1376, 1382, 1388, 1408–1410, 1412–
1414, 1419, 1424, 1426, 1429, 1457–1460,
1488, 1489, 1492–1494, 1499, 1521, 1532,
1555, 1556, 1658, 1671, 1672, 1674, 1686,
1689, 1693, 1705, 1707, 1708, 1710, 1716,
1732, 1739, 1761, 1826, 1827, 1830, 1835,
1842, 1843, 1847, 1850, 1869, 1875–1879,
1885–1888, 1902, 1914, 1915, 1929, 1930,
1936, 1940, 1957, 1959, 1965, 1973, 1986–
1988, 2033, 2041, 2071, 2093, 2095, 2134,
2135, 2140, 2228, 2229, 2232, 2237–2241,
2279, 2280, 2445, 2456, 2462, 2472, 2480,
2535, 2537, 2539, 2547, 2553, 2554, 2559,
2562–2565, 2567, 2569, 2571, 2572, 2577,
2611, 2617, 2642, 2644, 2685, 2694, 2695,
2772, 2777, 2793, 2794, 2799, 2800, 2860,
2878, 2879, 2926, 2932, 2934, 2939, 2941,
2947, 2949, 2984, 2988, 2989, 2994, 2996,
3000, 3007, 3011, 3012, 3024, 3027–3035,
3038, 3044, 3045, 3050, 3051, 3055, 3057–
3059, 3062, 3063, 3065, 3068, 3073, 3074,
3076, 3079, 3081, 3082, 3101, 3103, 3106,
3110, 3113, 3117, 3122, 3126–3128, 3135–
3138, 3140, 3236, 3246, 3249, 3283, 3341,
3405, 3409, 3455, 3508, 3609, 3630–3632,
3720, 3735, 3739, 3740, 3764, 3774, 3776,
3778, 3779, 3783, 3793, 3821, 3825, 3827,
3834, 3835, 3870, 3882, 3883, 3890, 3894,
3902, 3912, 3928, 3932–3935, 4089, 4127,
4128, 4130, 4137, 4163, 4171, 4172, 4245,
4258, 4270, 4284–4289, 4291, 4296, 4297,
4300, 4306, 4307, 4320, 4321, 4326–4329,
4331, 4334, 4340, 4342–4344, 4346, 4351,
4362, 4375, 4376, 4385, 4396, 4397, 4401,
4414, 4418, 4422, 4465, 4601, 4605, 4651,
4691, 4705, 4707, 4708, 4712, 4753, 4755–
4758, 4762

Points and Lines, 1370

Points and Point Groups, 4737

Points by Slope Ratio, 1459

Points Menu, 368

Points on Arc, 1251

Points on Line, 1245

Point Store, 2915, 2918–2920, 2926–2928, 2938,
2941, 2947
Point To Point Table, 1088
Polygon Export to Map, 2591
Polygon Inspector, 2590
Polygon Processor, 36, 2586, 2590, 2591
Polygon Query, 2590
Polyline by Nearest Found, 275
Polyline by Slope Ratio, 274, 1870, 2035, 3853
Polyline Editor, 3131, 3134
Polyline Elevation Editor, 1420, 1421, 1423–1426,
2638, 2639, 2641–2644, 2646, 2648, 4444
Polyline File, 26, 94, 98, 105, 114, 115, 128, 1575,
3542
Polyline Info, 283
Polyline Report, 847, 848
Polylines by Point, 1290
Polyline Slope, 1863
Polyline Slope Report, 1829
Polyline to Centerline File, 1698, 1701, 1702, 2048,
2052, 3899, 4138, 4214, 4694
Polyline to Deed File, 849
Polyline to Lot File, 1028
Polyline to Right of Way, 1692, 1722, 3893
Polyline to RW5 File, 849
Polyline to Special Line, 1117, 1120, 4299
Polyline to Tree Line, 1117, 1118
Polyline Vertices, 3132
Pond Design, 1639, 2242, 2243, 2246, 2250, 2274
Pond Exfiltration Design, 2267–2269
Pond Sizing, 2186, 2187, 2190, 4545
Post-Subsidence Contours, 3908
Pre-disturbed Surface, 3169
Predefined Boundaries, 1497, 1498
Prediction Point Output, 3907
Prepare HEC-RAS Input File, 2230, 2232, 2236,
2238, 2240
Prepare Story Stake, 2941, 2942
Prepare Value Block Model, 3488, 3500
Pressure Pipe Network, 2475, 2478, 2484, 2485,
2487, 2489, 2501
Pressure Pipe Network Design, 4712, 4727
Preview, 1577, 1647, 1900, 3177, 3223, 3467, 3513,
3544, 3592, 3868, 4022, 4088, 4093, 4159,
4166, 4285
Print View Print File, 1150
PRO, 31, 33, 36, 61–63, 65, 77, 94, 108, 294, 296,
301, 837, 839, 854, 855, 1181–1183, 1467,

1468, 1579, 1580, 1727, 1729–1735, 1737,
 1738, 1743, 1745, 1748–1750, 1759–1761,
 1764, 1765, 1767, 1812, 1813, 1821, 1822,
 1825–1827, 1829, 1830, 1832, 1833, 1839,
 1855, 1861–1863, 1868, 1874, 1885, 1925,
 1992–1994, 1998, 2003, 2033, 2072, 2080–
 2083, 2087, 2094–2096, 2101, 2105, 2110,
 2114, 2231, 2319, 2397, 2457, 2587, 2590,
 2591, 2701, 2947, 2965, 2981, 3060, 3116,
 3225, 3228, 3280, 3445, 3546, 3547, 3642,
 3644, 3653, 3740, 3901, 3902, 3906, 3915,
 3916, 3936, 4171, 4516, 4731
 Problems Addressed by Natural Regrade with Ge-
 oFluv, 3145
 Process Deed File, 27, 810, 849
 Process Design Sections, 2131, 2132
 Process Dragline Sequence, 4127, 4130, 4135, 4141,
 4142, 4144, 4210, 4213
 Process Data Project Items, 3059
 Process Layerized Text, 2586
 Process Menu, 582
 Process Only Strata with Beds, 321
 Process Road Design, 1896, 2031, 2032, 2034, 2039–
 2042, 2045, 2047, 2048, 2050–2054, 2057,
 2060, 2131, 2132, 2136, 3864, 4509, 4524,
 4528, 4760
 Production Timing, 322, 4038, 4229
 Profile, 1730, 1731, 1736, 1760, 1762–1764, 1865,
 1935, 3175, 3180, 3193, 3445, 3609, 3785,
 3788, 3795, 3851, 3886, 3901–3906, 3910,
 3911, 4124, 4137, 4148, 4161, 4200, 4428,
 4455–4458, 4491–4493, 4509, 4523, 4524,
 4527, 4535, 4580, 4581, 4632, 4645–4647,
 4685, 4686, 4722, 4731, 4741, 4742, 4745
 Profile Conversions, 1861–1863
 Profile Editor, 1759, 1761, 1765, 1766, 1827, 2061,
 2083, 2084, 2090, 2091, 2094, 2099, 2104,
 2111–2114, 2436, 3901, 3902, 4428, 4457,
 4524, 4743, 4745
 Profile from 3D Points, 1731
 Profile from 3D Polyline, 1730, 2052, 3905
 Profile from 3D Solids, 1736
 Profile from Grid or Triangulation Surface, 1729
 Profile from Layers, 1734
 Profile from Pipe Polyline, 1735, 1850
 Profile from Points on Centerline, 1733
 Profile from Polyline on Profile Grid, 1734
 Profile from Section File, 1732

Profile from Surface Entities, 1728, 1734
Profile ID, 1839, 2229
Profile Menu, 1725
Profile Offset Text, 1851–1853
Profile Report, 1827, 1829, 2320
Profiles, 4741, 4747
Profile Solid, 1736, 3795
Profile to 3D Polyline, 1824
Profile To Points, 1825
Programming Interfaces, 4751
Project Explorer, 69, 70, 72, 76, 78, 80, 120, 121, 301,
304, 341, 343
Projections, 3642, 3648, 3655, 3713, 3915, 3917,
3935
Projections & Ventilation, 3653
Project Manager, 2225, 2981, 2982, 2987, 2992,
3002, 3005, 3012, 3013, 3015, 3017, 3019,
3022, 3025, 3206, 3231, 3693, 3694, 3742,
3798, 3917, 3926, 3982, 4004, 4049, 4054,
4064
Project Settings, 3007, 3008, 3017, 3018, 3029, 3039,
3082
Project Setup, 25, 300, 349, 2982
Project Tab, 3005, 3012, 3017, 3031, 3037, 3040,
3047, 3067, 3073, 3111, 3120, 3129, 3130,
3140, 3141
Project Tree, 2983, 2985, 2989, 2990, 3006, 3060,
3076, 3115, 3121, 3126
Project_Checklist, 2661
Properties Converter, 2594
Property Boundaries, 3417, 3428, 3554, 4011
Property Data Report, 3686, 3996
Property Menu, 3684
Property Names, 323
Property Names By Text, 3689, 4000

Quantities by Average Method, 3667, 3674, 3677,
3945, 3947
Quantities by Avg Method, 3674
Quantities by Centerlines, 3671, 3680
Quantities by Grid Method, 3674, 3677, 3683, 3944,
3946
Query, 1309, 1310, 1353, 2523, 2524, 3275
Quick Contours, 1649
Quick Draw Surface, 1648
Quick Fence, 3444
Quick Geologic Column, 3272
Quick Keys, 23, 300, 358, 359
Quick Profile, 1726, 1727, 2700, 2709

Quick Section, 1876–1878
Quick Traverse, 1151, 1152, 1353, 1355, 1356, 1369

Radial Stake Out, 1265
Radial Stakeout, 911, 913, 1265
Radius Point, 224, 229, 235, 236, 282, 733, 848, 907,
1113, 1301
Railroad, 241, 1108, 1826, 1865, 3851
Railroad Curve Table, 1096
Rainfall Frequency and Amount, 2164
Rainfall Frequency and Amount - Lookup Table, 2166
Rainfall Frequency and Amount - NOAA Atlas 14,
2165
Range Diagram, 3442, 4114, 4215
Raster Edit Options, 2835, 2837
Raster EndPoint Snap, 2843
Raster Menu, 2831
Raster Nearest Snap, 2843
Raster To Text, 2852, 2855
Rational Method, 2143–2145, 2187–2189, 2192,
2204, 2206, 2340, 2502, 3174, 3201, 4558,
4578
Rational Rainfall Library, 2340, 2348, 4563
Raw File On/Off, 861
Raw Traverse Data, 607
re-Associate Coord. file, 1310, 1311
Reactors, 130
Reame, 36, 4170–4172
Reannotate, 1064
Reassign Database File, 3235
Reassign Pit Attr Grids Folder, 3983
Reassign Timing Grids Folder, 3984
Rebuild Inside Perimeter, 1531
Recalculate Extraction, 3726, 3739, 3740
Record Script, 131
Record Script, End Script, Run Script, 131
Recovery Percent, 3213, 3419, 3428, 4013
Rectangular Pond Design, 2254, 2255, 2274, 2277
Rectangular Weir Design, 2261, 4548
Reduce Contour Vertices, 1544, 3358
Reduce Direct-Reverse, 530
Reduce Map Check File, 1223
Reduce Points, 3821
Reduce Polyline Vertices, 105, 115, 117, 169, 170,
309, 1013, 4610
Reduce Traverse, 1208, 1209, 1223
Reduce Vertices, 189, 1423, 1506, 1537, 1538, 1894,
2629, 2630, 2641, 2734, 2743, 2744, 2862,
3109, 3375, 3519, 3520, 3842, 3862

Refine Solid, 3827
Refresh Screen, 1312, 1313
Register Cloud, 3050
Registration, 3, 7, 8, 12, 14, 18, 449, 451, 3050
Regrade Backfill, 4164
Regrade Fill Slope, 1946
Rehandle, 3708, 4128
Removal Entities, 2687, 2688
Removal Manager, 2685, 2688, 2689
Removal Settings, 2684, 2689
Remove All Non-Key Bed Names, 3298
Remove Area Table Rows, 930
Remove Bed Names After Last Key, 3298
Remove Civil 3D Custom Objects, 129
Remove Duplicate Polylines, 178
Remove Empty Benches, 3980
Remove Faults From Grid, 3386
Remove Groups, 130
Remove Lateral Structure, 2410
Remove Panel, 3729, 3730
Remove Pin Point, 3737
Remove Pit Names, 3955
Remove Points from Screen, 1287
Remove Polyline Arcs, 178, 3963, 3973, 4184
Remove Polyline Blips, 180
Remove Polyline Segment, 179, 4325
Remove Polyline Vertex, 180, 854, 4350
Remove Property Data, 3685, 3996
Remove Reactors, 130, 434, 1065
Remove Sewer Structure, 2397
Remove Solid Dangles, 3826
Remove Spaces From Text, 165
Remove Speckles, 2846, 2851
Remove Spoil Names, 4180
Remove Trench Network Structure, 2776
Remove XData, 129
Rename Attribute, 3297
Rename Bed, 3299
Rename Strata, 3296
Render, Shade, and Hide Commands, 1666
Rendering, 205, 307
Renumber Points, 385, 391, 392, 437, 531, 1318,
1326
Renumber Structure Names, 2444
repackaged, lxiv
Replace Color, 2848
Replace Text, 42, 166
Replot Descriptions, 1126

Report, 1513, 1561, 1591, 1594–1596, 1651, 1761,
1890, 3175, 3176, 3183–3185, 3190–3192,
3207, 3216, 3235, 3276, 3302–3304, 3306,
3311–3320, 3334, 3340, 3341, 3351, 3387,
3390, 3402, 3415, 3417, 3418, 3421, 3423,
3426, 3428, 3429, 3431, 3434, 3455, 3479,
3483, 3504, 3509, 3523, 3526–3528, 3554,
3556, 3576, 3601, 3603, 3606, 3618, 3634,
3674–3676, 3679, 3680, 3682–3684, 3694,
3696, 3703, 3707–3709, 3716, 3747, 3748,
3752, 3755, 3758, 3761, 3774, 3796, 3798,
3804, 3807–3809, 3813, 3814, 3829, 3848,
3858, 3902, 3903, 3924, 3945, 3946, 3983,
3985, 3987, 3988, 4005, 4009, 4011, 4015,
4017, 4020–4022, 4030, 4035, 4036, 4040,
4043, 4045, 4050, 4059, 4066, 4067, 4069–
4071, 4075, 4077, 4078, 4080, 4081, 4094–
4096, 4123, 4127, 4128, 4130–4132, 4138,
4139, 4149, 4169, 4171, 4188, 4191, 4196,
4199, 4220, 4228, 4233, 4235, 4256, 4268,
4280, 4286, 4360, 4361, 4379, 4383, 4384,
4391, 4392, 4427, 4428, 4436, 4440, 4453,
4462, 4463, 4468, 4469, 4475, 4480, 4532,
4535, 4545, 4554, 4578, 4582–4584, 4602,
4610, 4625, 4626, 4629, 4657, 4665, 4666,
4699–4701, 4703, 4704, 4708–4711, 4726

Report 3D Polyline Station/Elevation, 1487

Report Areas By Layers, 2594

Report Fault Lines, 3376

Report Formatter Dialog, 43

Report Grade Elevation, 1718, 1719

Report Hydrograph, 2216

Report Limit Polylines, 3393

Report Locked Points, 404

Report Log, 306

Report Non-Surface Points, 1493

Report Offset Point To Solid, 3828

Report Sewer Network, 2418, 2421

Report Stage-Discharge, 2285

Report Strata Polylines, 3390

Report Sub-Area, 1008

Report Subgrade Areas, 2671

Report Tons & Acres, 3682, 4162

Report Trench Network, 2790

Reread Valley Bottoms, 3202

Resample Cloud, 3052

Rescale Drawing, 362

Resection, 874–876, 2931, 2933, 2934

Reserve Classification, 3417, 3552, 3555, 3556, 3685,
3996, 4011
Reserves Timing Menu, 4004
Reservoir Routing Hydrograph, 2209
Reset Invalid Attribute, 3296
Reset Panel Attributes, 3712
Residuals, 36, 240, 526, 629, 635, 651, 656, 894, 895,
900, 3431, 3434, 3587, 4382
Resize, 505, 2588, 2837, 4334
Resize Point Attributes, 425, 4289
Resource-Reserve Classification, 3552
Restore Due North, 208, 854, 1091
Restore Layer, 209, 216, 218, 3766, 3768, 3769, 4294,
4301, 4541
Restore Layout Layers, 2730
Restore World View, 214
Restricted Grade Design, 1844, 1857
Retaining Wall, 1970
Reverse Calls, 1295
Reverse Directions, 3971, 3976, 4183, 4186
Reverse Panel, 3729
Reverse Polyline, 169, 938, 1423, 1426, 1698, 1872,
2231, 2641, 3131, 3343, 3899, 4350
Review Profile Links, 1840
Review Section Links, 1949
Review Sewer Network Links, 2452
Review Sewer Profile Links, 2452
Right-of-Way Crossing Table, 1036
Right-of-Way Easements, 1259
Right of Way, 1018, 1353, 1685, 1689, 1723, 1902,
1970, 3870, 3890
Rigid Body Transformation, 414, 3050
RIVERMorph tab, 3197, 3198
Road API, 4764
RoadCalc, 1883, 1884
Road Design, 36, 37, 295, 313, 837, 1011, 1624,
1721, 1722, 1740, 1741, 1867, 1923, 1924,
1957, 1959, 1960, 1967, 1969, 1973, 1983,
1990–1992, 1994, 1996, 1998–2001, 2003–
2007, 2009, 2011, 2013, 2015, 2016, 2032,
2033, 2035, 2036, 2043, 2077, 2089, 2093,
2095, 2469, 4525, 4729, 4731
Road Design Inspector, 2132
Road Network, 1896, 2061–2066, 2068, 2069, 2071,
2072, 2074–2079, 2081–2084, 2089, 2090,
2094–2096, 2098, 2099, 2102, 2104–2108,
2111, 2112, 2114–2117, 2119–2126, 2131,
2469, 2731, 3864

Road Profile Editor, 1738, 1743
Road Rehabilitation, 4514, 4527
Road Rehabilitation Profile, 2001, 2003, 2005–2008,
2011, 2013
Roadside Ditch, 36, 1860, 1861
Roads Menu, 1956, 2731, 2944
Road Stripe Library, 2030, 2031, 2035
Roadway Cross Sections, 4742
Roadway Intersections, 4745
Roadways, 1259–1262, 1265
Roadway Templates, 4743
Rock Section, 1884
Rock Surface, 2065
Room Label Block, 3658, 3659, 3937
Rooms, 3649, 3650, 3652, 3657, 3658, 3935–3937
Rotate Points, 212, 369, 385, 388, 391, 406, 1328
Rotate Solid, 3832
Rotate Text, 154, 1078
ROW Offsets Editor, 1009
Rubber Sheet, 2849–2851
Run Isopach Maps AutoRun, 3356
Runoff, 36, 1988, 2141, 2149, 2152, 2155, 2156,
2159, 2176, 2179, 2186, 2189, 2192, 2200,
2201, 2203, 2204, 2206, 2220, 2252, 2292,
2327, 2328, 2370, 2384, 2418, 2465, 2503,
3150, 3172, 3174, 3199, 3201, 4533, 4537,
4542, 4545, 4558, 4559, 4562, 4572, 4578
Runoff Hydrograph - Rational Method, 2204, 2207
Runoff Hydrograph - SCS Method, 2199
Runoff Hydrograph - TR-55 Tabular Method, 2203,
2204
Run Off Tracking, 2159, 2163, 2164
Run Off Tracking - Grid Surface, 2163
Run Script, 131
Run Strata Grids AutoRun, 3352

Sag & Crest Report, 1833
Satellite SkyPlot, 2929
Save/Load Tablet Calibration, 363
Save/Restore Layer State, 218
Save As, 1133, 1374, 1390–1393, 1412, 1579, 1971,
2474, 2852, 2865, 3006, 3161, 3210, 3546,
3585, 4429, 4465, 4488, 4514, 4584, 4608,
4635, 4691, 4763
Save As Raster Image, 2835
Save Design Surface, 3180
Save Drawing Named by Current Time, 119
Save Layer State, 218
Save Layout Layers, 2729, 2730

Save Sewer Network File, 2339
Save Tablet Calibration, 2796
SB-Slope, 37, 4172
Scale Cloud, 3056
Scale Point Attributes, 424, 425
Scale Points, 293, 385, 388, 409
Scale Solid, 3832
Scans, 3061, 3063
Scenes, 3017, 3025, 3060, 3061
Scene Tab, 2993, 3074
SCJSTART.EXE, 19, 20
Screen Pick Point, 385, 388
SCS Rainfall Library, 2348, 2349, 2354, 2355
SCT File, 1874, 1938
SDPS, 323, 3906–3911
Seam Stacking, 318, 3349
Search Point Files, 435
Search Published Control, 444
Search Radius, 317, 1567, 3070, 3109, 3110, 3113,
3343, 3350, 3455, 3457, 3465, 3534, 3821
Section Alignment, 1778, 1869, 1915, 1929, 1942,
2221, 3119, 3777, 3825, 3853, 3883, 4484,
4485, 4498, 4518, 4731
Section Alignment Editor, 1864, 1872, 2003, 3850,
4482, 4496, 4516
Section Area Report, 1944
Section Conversion, 1884
Section Conversions, 1883
Section Defaults, 1869
Section File, 28, 836, 1149, 1150, 1155, 1343, 1371,
1373, 1405, 1865, 1869, 1870, 1873–1876,
1880, 1884, 1888, 1896–1898, 1919, 1923,
1929–1931, 1936, 1940–1945, 1948, 1951,
1954, 2032, 2034–2036, 2043, 2089, 2090,
2101, 2105, 2220, 2813, 2829, 2947, 2951,
3772, 3795, 3851, 3854, 3856, 3864–3866,
3887, 4484, 4485, 4487, 4498, 4507, 4518,
4522, 4673, 4742
Section File Editor, 1888, 1896, 1951, 1996, 2034,
2035, 2038, 2040, 2049, 2131, 3856, 3864,
4485, 4508, 4509, 4671
Section ID, 1950
Section Information, 3942
Section Line, 273, 883, 4684
Section Menu, 1863
Section Name, 3632, 3916
Section Report, 28, 1883, 1918, 1919, 1929, 2043
Sections from 3D Polylines, 1872

Sections from Coordinate File, 1873
Sections from Grid or Triangulation Surface, 1869,
3853, 4485
Sections from Layers, 1874
Sections from Points, 1871, 2034, 3855
Sections from Polylines, 1869, 3853, 4670
Sections From Polylines On Section Grids, 2732
Sections from Profile, 1874
Sections from Surface Entities, 1867, 1874, 1875,
2034
Section Solid, 3793
Sections Points from Pipes, 1876
Sections Points from Right of Way, 1876
Sections Points from Surface Entities, 1875, 1876
Sections to 3D Polylines, 1935, 1936, 3886, 3887
Sections to Points, 1936, 1937
Section Subdivision, 883
SEDCAD, 35, 313, 2143, 2216–2219
SEDCAD Draw Flow Polylines, 2217
SEDCAD Label Structure Layout, 2218
SEDCAD Locate Structures, 2217
Select by Area, 152, 153
Select by Block, 152
Select by Elevation, 167
Select by Filter, 153
Select by Length, 151
Select CAD Default Settings, 368
Select Drillholes By Filter, 3275
Selected Strata Quantities, 3365, 3367, 3558, 3559,
3563–3565, 3568
Select GeoFluv Boundary, 3166
Select Main Channel, 3166, 3168
Select Similar, 153
Sequential Numbers, 267, 939, 3956, 3957, 4356
Serial Number Report, 449
Set/Reset X-Hairs, 363
Set Active Area Table, 927
Set Active Surfaces, 2709
Set Attribute By Grid File, 4037
Set Attribute by Grid File, 3709
Set Background Color, 2855
Set Back Measure-Move, 956
Set Coordinate File, 369, 383, 386, 491, 509, 1686,
2597, 4465, 4605, 4691, 4705
Set CRD File for Lot Files, 1042
Set Current, 1339, 1367, 1523, 2692, 3704, 4031,
4244
Set Current Position, 3737

Set Data Directory, 31, 33
Set Default, 1339, 1390–1393, 1412, 1900, 3109,
3868, 4526
Set Descriptions By Offset, 1950
Set Descriptions By Polyline, 1950
Set Google Earth Drillhole Tags, 3277
Set Google Tags, 112
Set Layer, 129, 209, 1431, 1539, 1673, 2069, 2070,
2075, 3473, 3521, 3750, 4042, 4072, 4613,
4614
Set Layer For Design, 2666
Set Layer For Existing, 2665
Set Layer For Other, 2666
Set Level, 207, 661, 739, 1767
Set Line Type, 1273, 1274
Set Linework Angles To Nearest Second, 941
Set Linework Intersections To Perpendicular, 941
Set Lot Edge Angles To Nearest Second, 1016
Set Lot File, 1027
Set Parameter For Entire Network, 2427
Set Point Elevations by 3D Polyline, 846
Set Point Elevations by Surface Model, 847
Set Polyline Origin, 174
Set Polyline To Elevation, 1420, 4618
Set Polyline to Elevation, 2638
Set Project/Data Folders, 294
Set Raster Image, 2834, 2835, 2839, 2842, 2847–
2849
Set Removal Boundary, 2688
Set Rim Elevation to Match Surface, 2445
Set Sewer File, 2333, 2444, 2453, 2457, 2461, 4566
Set Strata by Parameters, 3329, 3330
Set Strata Key Status, 3294
Set Surface File, 2333
Set Table Position, 1094
Set Text Color, 2855
Set Text Layer By Text String, 2597
Settings, 1501, 1513, 1534, 1567, 1700, 1761, 1864,
1888, 1900, 1911, 1916, 3199, 3200, 3202,
3203, 3207, 3226, 3241, 3242, 3250, 3255,
3256, 3260, 3263, 3265, 3269, 3272, 3284,
3304–3307, 3314, 3324, 3335, 3337, 3349,
3355, 3398, 3402, 3410, 3439, 3441, 3442,
3462, 3516, 3534, 3575, 3582, 3583, 3588,
3609, 3645, 3647, 3669, 3694, 3696, 3764,
3786, 3787, 3819, 3837, 3848, 3850, 3857,
3868, 3879, 3884, 3900, 3903, 3917, 3942,
3954, 4005, 4021, 4022, 4047, 4048, 4064,

4066, 4086, 4107, 4115, 4117, 4118, 4121,
 4133, 4171, 4173, 4239, 4245, 4249, 4256,
 4257, 4262, 4268, 4269, 4274, 4280, 4281,
 4309, 4314, 4318, 4321, 4323, 4325, 4331,
 4333, 4338, 4342, 4345, 4364, 4365, 4369–
 4371, 4375, 4376, 4380–4382, 4385, 4387,
 4388, 4390, 4394, 4396, 4397, 4408, 4412,
 4415–4417, 4420, 4428, 4429, 4445, 4465,
 4483, 4488, 4515, 4517, 4522, 4530, 4556,
 4565–4567, 4569, 4570, 4573, 4577, 4579,
 4584, 4588, 4591, 4598, 4599, 4603, 4605,
 4606, 4608, 4609, 4616, 4617, 4628, 4630,
 4631, 4633, 4635, 4636, 4647, 4649, 4651,
 4652, 4660, 4663, 4667, 4669, 4670, 4675,
 4682, 4684, 4687, 4691, 4699, 4700, 4705,
 4706, 4710, 4713, 4714, 4731, 4732, 4767
 Settings File Manager, 349, 350
 Settings Menu, 287, 558, 593, 1375, 1380, 1389,
 1401, 1403–1405, 1408, 1410, 1416
 Setting Up a Project, 31
 Set UCS to World, 206, 307
 Setup DOS Dwg, 1148
 Setup Property Link, 3686, 3997
 Setup Tab, 3165
 Sewer, 29, 37, 203, 301, 696, 705, 717, 774, 783, 795,
 1732, 1749–1751, 1760, 1766, 1767, 1798,
 1817, 1818, 1854, 1855, 1915, 2318–2320,
 2323, 2325, 2328, 2339, 2375, 2386, 2398,
 2411, 2418, 2432, 2435, 2437, 2438, 2441,
 2449–2452, 2454, 2455, 2457, 2462, 2463,
 2465, 2516, 2772, 3883, 3902, 4298, 4301,
 4325, 4567, 4579
 Sewer/Pipe Profile Editor, 1756
 Sewer Annotation, 1751
 Sewer Network, 28, 29, 314, 717, 795, 1809, 2145,
 2319, 2323, 2324, 2331, 2333, 2340, 2361,
 2362, 2366, 2372–2374, 2386, 2390, 2398,
 2404, 2411, 2413, 2414, 2417, 2419, 2425–
 2427, 2433, 2435–2437, 2444–2446, 2448,
 2452, 2453, 2455, 2457–2463, 2469, 2517,
 2776, 4563, 4566, 4567, 4569, 4570, 4579
 Sewer Network Hydrographs, 2424, 2425
 Sewer Network Inspector, 2423, 2424
 Sewer Network Model, 2323
 Sewer Network Quantities Report, 2422
 Sewer Network Settings, 2323, 2327, 2374–2378,
 2383, 2384, 2386, 2387, 2389, 2390, 2393,
 2394, 2398, 2404, 2453, 2461

Sewer Pipe Design: Individual, 2318
Sewer Pipe Design: Read Profile, 2319
Sewer Pipe Design: Sewer Network Segment, 2319
Sewer Structure, 37, 2395, 2433, 2434, 2437, 2438,
2457, 2773–2775, 2785
Sewer Structure Library, 2361, 2366, 2391, 2454,
2456, 4570
Shade, 109, 110, 201, 681, 759, 1666, 2679
Show/Hide By Selection, 217
Show All Property Lines, 3691, 4002
SHP File, 109, 2548, 2572
Shrink-Wrap Entities, 26, 270, 272, 4466
Shrink/Swell Table, 2136
Shrink Resolution, 2847
Side Shots, 860, 862, 864, 1216, 1218
Sight Distance, 313, 1739, 1741, 1744, 1762, 1859,
2016, 2092, 2095, 3903
Sight Distance Design, 1858
Simplify Solid, 3825
Simulation GPS, 2970
Single Runoff Hydrograph, 2215, 2216
SiteNet Menu, 1418
Size Lot by Frontage, 1019
Sketch, 274, 356, 1119, 1557, 1558, 2809–2812,
2825, 2826, 2856, 2857, 4653
Sketch Tree Line, 1119
Sliding Side Area, 933, 4350
Slope/Length/Elevation Calculator, 1858
Slope At Point, 27
Slope At Points, 1672, 1673
Slope Direction Analysis, 1678
Slope Distance, 483, 533, 535, 543, 548, 640, 863,
875, 912, 1197, 1830, 1857, 1859
Slope Inspector, 2955
Slope Ratio, 274, 1424, 1459, 1830, 1857, 1859,
2274, 2295, 2642, 2662
Slope Report, 1670, 1675, 1830, 2140
Slope Stake Report, 1939, 1941
Slope Staking, 2951, 2952, 2954
Slope Zone Analysis, 1674, 1945, 2140
Slope Zone Section Analysis, 1944
Smart Prompting, 2521
SmarTrace, 274
SmartTrace, 274
SMI, 35, 461, 475, 496, 541, 1204–1207, 1686, 1884,
2040
SMI Chain File, 2040
Smooth Cloud, 3054, 3074

Smooth Contours, 1537, 1542, 1675, 3356, 3519
Smooth Grid, 1567, 3534
Smoothing, 182, 1507, 1537, 1544, 1567, 1681, 2629,
2744, 2868, 3070, 3092, 3099, 3100, 3108,
3115, 3119, 3345, 3358, 3519, 3534, 3842,
3843
Smooth Points, 3821
Smooth Polyline, 182, 222, 226, 2836, 2841, 2917,
4295
Smooth Solid, 3826
Snap Contours to 3D Polylines, 2660
Snap Linework To Points, 2597
Snap Tolerance, 176, 269, 271, 896, 1438, 1847,
2597, 2651, 3110
Softdesk, 324, 440, 443, 495, 1720, 1861–1863, 1884
Software Compatibility, 3158
Sokkia, 2971–2973
Sokkia/SDR, 472, 473, 540, 1862
Sokkia G, 461, 474, 2897–2899
Solar Observations, 889
Solid Fill Polyline, 192
Solid Information, 3834
Solid Menu, 3762
Solid Timing, 3797–3799, 3806, 3820
Spad Number, 3626–3629, 3939, 3940
Special Fill Areas, 2675, 2676
Special Leader, 260, 4307
Special Line, 703, 781, 1115
Special Line/Ent, 1115
Special Line/Entity, 1120
Spillway, 28, 2255–2258, 2278–2280, 2284, 4550
Spiral, 241, 242, 821, 1253–1255, 1687, 1691, 1693,
1694, 1696, 1704, 1988, 3892, 3894, 3895,
3897
Spiral Curve, 241
Spiral Curve Design, 1253
Spiral Curve Stakeout, 1254
Spiral In, 1694, 3895
Spirals, 1686
Spline, 117, 1292, 1354, 1542, 1543, 3356, 3357
Split Bed, 3299, 3560, 3561, 3563
Split Bed by Parameters, 321, 3330, 3331
Split Table, 1102
Split Text into Two Lines, 163
Spoil by Interior Point, 4176
Spoil By Interior Text, 4178
Spoil Layout by Width, 4178
Spoil Menu, 4175

Spoil Pile, 4155
Spoil Placement Timing, 4191, 4195
Spoil Side, 4142, 4208, 4214
Spot Drillholes, 3238
Spot Elevation Links Manager, 1455
Spreadsheet Sewer Editor, 2427
SQL, 46, 301, 308, 368, 384, 388, 403, 550, 552,
2518, 2519, 2558, 2561, 2562, 2564, 2567,
2569–2571, 2583–2585, 3208, 3228, 3755,
4076, 4753
SRVPNO, 433, 435, 691, 769, 1512, 3847
Stack Label Arc, 27
Stack Label Arc Auto Place, 1107
Stack Label Arc Pick Side, 1107, 4306
Stage-Discharge, 2259–2261, 2263–2269, 2271,
2283–2286, 2312, 2313, 4550–4552
Stage-Discharge Editor, 2283
Stage-Storage, 313, 1634, 1642, 2139, 2255, 2272–
2274, 2276, 2278, 2281, 4548
Stage-Storage Editor, 2272
Stage Storage, 2277, 2278, 4548
Stake-Out, 1265, 1266
StakeOut, 2923
Stakeout Horizontal, 1255
Stakeout Point Comparison, 835
Standard Cul-de-Sac, 1262, 1263
Standard deviation, 2814, 2815, 2828, 2829, 2831,
3112, 3301
Standard Drillhole Report, 3302, 3303
Standard Drillhole Text, 3249, 4688
Standard Report Viewer, 41, 43, 45, 282, 286, 378,
512, 513, 515, 819, 848, 1101, 1487, 2318,
2319, 2690, 2710, 2711
Starting AutoCAD with Carlson, 11
Starting Survnet, 552
Start Point Number, 371
Startup Wizard, 31–33, 306, 4236, 4237, 4239, 4245,
4257, 4269, 4281, 4282, 4318, 4320, 4341,
4363, 4367, 4369, 4396, 4411, 4429, 4464,
4488, 4514, 4530, 4556, 4584, 4598, 4608,
4635, 4649, 4650, 4667, 4691, 4705, 4712
State Plane, 289, 292, 410, 412, 512, 522–524, 560,
579, 587, 619, 630, 635, 639, 1684, 2570,
2599, 2875, 2877, 2880, 2904, 2907, 2914,
2962, 2972, 2977, 3042, 3050, 4388, 4706
Station-Elevation-Slope Report, 1830, 1831
Station-Offset, 1243, 1308, 1309
Station/Elevation, 1806, 1827

Station Offset, 1240, 1242–1245
Station Offset From Coords, 1244
Station Polyline/Centerline, 28, 1700, 1702, 3900
Statistical Analysis, 3300
Stockpile, 1587, 1618, 1620, 1621, 2707, 2708, 2714,
2715, 4428, 4690
Stoppings, 3632, 3656, 3916
Store Project Archive, 304
Stormwater Network Design, 4556, 4584
Story Stake Along Polyline, 2943
Story Stake By Points, 2942
Story Stake By Points/Polyline, 1489
Story Stake from Surface Entities, 1488
Strata Attributes, 3208, 3209, 3215, 3470
StrataCalc, 29, 35, 322, 2766, 3205, 3209, 3214,
3216, 3289, 3301, 3332, 3338, 3339, 3347,
3352, 3354, 3356, 3365, 3367–3374, 3376–
3383, 3385, 3386, 3392–3396, 3401, 3402,
3412, 3427, 3433, 3435, 3444–3446, 3448,
3450, 3451, 3506, 3508, 3512, 3552, 3556–
3558, 4021, 4220, 4632, 4633, 4685
StrataCalc Data Sheet, 3332
StrataCalc Menu, 3332
Strata density, 4164
Strata Grid Files, 3241
Strata Isopach Maps, 35, 3217, 3220, 3245, 3352,
3391
Strata Order and Count Report, 3318
Strata Polylines, 2757, 2758, 3390, 3441
Strata Quantities, 37, 3363
Strata Quantities in Series, 3363, 3365, 3366
Stream Network, 2231, 2232
Strike-Dip Symbol Editor, 3386
Strip Ratio, 322, 3250, 3422, 3424, 4016, 4018, 4049
Strip ratio, 4164
Strip ratios, 3566
Structure Menu, 2241
Sub-Watersheds By Land Use, 2166
Subcrop, 3577
Subcrops, 3571, 3578
Subdivide Panel, 3712
Subgrades, 1870, 1890, 1895, 1908, 1909, 1914,
1957, 1994, 2665, 2786, 2828, 3854, 3858,
3863, 3876, 3877, 3882
Subsidence Menu, 3906
Subtract Cloud, 3054
Subtract Hydrographs, 2216
Suggested Standards, 3911

Summary, 3159, 3291, 3292, 3400, 3461, 3491, 4464,
4488, 4559, 4594, 4603, 4633
Summary Report, 3182, 3183, 3434
Superelevation, 37, 1958, 1964–1966, 1982, 2058,
2951
Super Elevation Editor, 1979, 2011, 2035, 2085
Surface, 1423, 1426, 1499–1503, 1505, 1507, 1514,
1540, 1544–1546, 1554, 1555, 1559, 1562,
1563, 1574, 1581, 1584, 1585, 1591, 1593,
1597, 1646–1651, 1864, 1894, 1901, 2625–
2627, 2629, 2633, 2634, 2636, 2637, 2641,
2644, 2661, 2662, 2664, 2673, 2676, 2677,
2679, 2680, 2682, 2684, 2687–2689, 2691–
2697, 2700, 2703–2706, 2709, 2710, 2714,
2715, 2725, 2734, 2742, 2744, 2754, 2755,
2760, 2764–2766, 2772, 2780, 2787, 2788,
2809, 2825, 2828, 2856, 2860, 2867, 2939,
2940, 3045, 3073, 3152, 3153, 3158, 3160,
3165, 3167–3169, 3171, 3180, 3181, 3185,
3188, 3189, 3202, 3203, 3207, 3213–3216,
3234, 3241, 3258, 3265, 3266, 3280, 3358–
3360, 3362, 3363, 3369–3371, 3381, 3384,
3414, 3415, 3417, 3418, 3425, 3429, 3441–
3444, 3470, 3473, 3476, 3477, 3479, 3484,
3487, 3491, 3502, 3505, 3506, 3509, 3510,
3512, 3514, 3515, 3522, 3525, 3529, 3541,
3548, 3551, 3552, 3554, 3566, 3567, 3569,
3571, 3575, 3576, 3578–3581, 3588, 3612,
3618, 3619, 3632, 3685, 3694–3696, 3708–
3710, 3716, 3745, 3746, 3756, 3763, 3768,
3770, 3792, 3803, 3806, 3816, 3818, 3835,
3836, 3838–3842, 3849, 3850, 3862, 3869,
3949, 3953, 3956, 3965, 3980, 3983, 3988,
3996, 4005, 4007–4010, 4012, 4019, 4021,
4022, 4037, 4038, 4044–4046, 4048, 4050,
4051, 4064, 4081–4083, 4085, 4088, 4091,
4093, 4096, 4104, 4110–4112, 4114, 4124,
4129, 4130, 4133–4136, 4138–4141, 4144–
4146, 4148, 4149, 4151, 4152, 4156–4158,
4161–4164, 4166, 4168–4172, 4174–4177,
4187, 4189–4191, 4197, 4199, 4200, 4203,
4205, 4210, 4214, 4216, 4217, 4220, 4221,
4223, 4224, 4231, 4232, 4235, 4316, 4317,
4338, 4398, 4401–4403, 4406, 4409–4412,
4422, 4425, 4428, 4429, 4435, 4437, 4438,
4445–4447, 4450, 4452–4455, 4461–4464,
4466–4475, 4477–4480, 4484, 4485, 4488,
4498, 4503, 4515, 4518, 4520, 4522, 4532,

4557, 4558, 4560, 4567, 4591, 4611, 4613,
 4614, 4627, 4630, 4635, 4636, 4643, 4660,
 4675, 4713, 4715, 4731, 4737, 4740, 4742,
 4752
 Surface 3D FlyOver, 1652, 1663–1665
 Surface 3D Viewer, 210, 1662, 2118, 2123
 Surface Area, 2268, 2467
 Surface Entities, 199
 Surface Equipment Timing, 29, 3816, 4050, 4052,
 4053, 4056, 4062, 4065, 4068, 4083, 4105,
 4150, 4190, 4191, 4195–4197, 4199, 4216,
 4226, 4229, 4232
 Surface Inspector, 37, 1650, 1651, 2693, 2709, 3509,
 3510, 4474, 4632
 Surface Manager, 1503, 1522, 1532, 1559, 1628,
 2692, 3838, 4693, 4737, 4738
 Surface Menu, 1496, 1497, 2139, 2939, 4110
 Surface Mine Reserves, 3412, 3417, 3425–3427,
 3429, 3455, 3552, 3554, 3564, 3565, 3567,
 3570, 3575, 3576, 3598, 3603, 3617, 3685,
 3950, 3955, 3966, 3983, 3989, 3991, 3996,
 4004, 4005, 4011, 4019, 4020, 4051, 4081,
 4082, 4149, 4190, 4192, 4220, 4221, 4229–
 4231
 Surface Mining Module, 3417, 3426, 3949, 3994,
 4010, 4011, 4020, 4120, 4124, 4154, 4160
 Surface Model, 169, 226, 847, 907, 1623, 1628,
 1635–1637, 1673, 1729, 1735, 1824, 1869,
 1874, 1877, 1878, 2159, 2162, 2163, 2222,
 2234, 2236, 2323, 2324
 Surface Object View Properties, 206
 Surface Production Timing, 4038, 4039, 4043, 4050
 Surface Report, 2694
 Surfaces and Contours, 4737
 SurvCE, 26, 35–37, 58, 88, 97, 289–291, 377, 381,
 455, 459–463, 466, 468, 470, 474, 492, 493,
 497, 524, 542, 552, 563, 609, 667, 668, 679,
 688, 689, 695, 717, 745, 746, 757, 766, 767,
 773, 795, 832, 835, 858, 1176–1179, 1339,
 1340, 1842, 1942, 2069, 2519, 2548, 2566,
 2574, 2575, 2577, 2578, 2614, 2619, 2883,
 2913, 4235, 4238, 4598, 4600–4605
 SurvCOGO, 497, 542
 Survey, 4732, 4733, 4753
 Survey Menu, 455, 610, 4245, 4257, 4269, 4281,
 4323, 4342, 4412, 4605, 4691, 4705
 Survey Module, 454, 491, 1039, 2139, 3933
 Surveyors Assistant, 461

Survey Text Defaults, 1075–1078
SurvNET, 549–555, 560, 561, 566, 571, 577, 578,
580–583, 593–595, 601, 602, 606–608, 612,
616–619, 653, 654, 1354, 1372, 1376, 1384–
1386, 1388, 1400, 4235, 4366, 4367, 4369,
4370, 4372, 4374, 4376, 4385, 4386, 4390,
4394, 4396
SurvNET Editor, 618
SurvNET Window Menu, 606
SurvStar, 461, 462, 467, 2580, 2912
Swing Table, 1095
Switch Bearing/Azimuth Quadrant, 1067
Symbol Library, 37, 250, 300, 369, 371, 416, 697,
726, 775, 804, 2455, 2456, 3202, 4145
Symbol Plot Size, 260, 288
Symbol Size, 307, 375, 445, 679, 697, 701, 708, 713,
757, 775, 779, 786, 791, 1082, 1115, 1117,
1275, 1338, 1359, 1513, 1796, 1830, 1836,
1901, 2022, 2382, 2391, 2474, 2490, 2538,
2678, 2749, 3379, 3748, 3849, 3869, 4070,
4334
System Requirements, 3
System Variable Editor, 365

Table Defaults, 830, 1089, 1094
Table Header, 44, 1038, 1055, 1057, 1093–1095
Tables, 1295, 1297, 1298, 1741, 1761, 2017, 2094,
2237, 2239, 2240, 2433, 2434, 2546, 3215,
3902
Tablet Calibrate, 1555, 1558, 1878, 1880, 1883, 2793
Tablet Calibration, 363, 1555, 1556, 1878, 1879,
2794, 2795, 2800, 2801
Tablet Off, 2793, 2823
Tablet On, 2793
Tablet Template, 18
Tabular Hydrograph Method, 2190
Tag Area Descriptions, 931
Tag Crub Ramps, 2000
Tag Drillholes for Processing, 3331
Tag Hard Breakline Polylines, 1495
Tag Heave Zones, 3381
Tag Inner Pads, 1448
Tag No-Building Area, 1025
Tag Non-Surface Points, 1491
Tag Pad Reference Point, 1447
Tag Predefined Boundaries, 1497, 1587
Tag ROW Segments to Skip Frontage, 1024
Tag Segments To Skip Linear, 2725
Tag Slope Groups, 4145

Tag Strata Polylines, 3389
 Tag Sub-Area, 1007–1009
 Takeoff/SiteNET CAD File From Start To Finish,
 4608, 4627
 Takeoff Menu, 2661
 Takeoff Trench Network Quantities, 4635
 Takeup Date, 3631, 3632
 Tang, PC, Radius, Chord Length, 236
 Tang, PC, Radius, Delta Angle, 236
 Tangent, PC, Radius, Arc Length, 235
 Tangent, PC, Radius, Tangent Length, 235
 Tangent Between Curves, 1256
 Tangent Intersect, 873, 1237
 Tangent Line from Circles, 878
 Tangent Out Bearing, 732
 Tape Baseline, 2870
 Target Points, 3037–3040, 3057, 3060, 3062, 3140
 TDS, 461, 474, 475, 491, 497, 498, 544, 564, 601,
 609, 668, 746, 835, 1180, 1184, 1720, 1884,
 2519, 2566
 TDS CR, 438
 Technical Support, 9, 23, 2818, 2966, 2969, 4239
 Tech Support, 21
 Template Grade By 3D Polyline, 1998
 Template Grade Table, 1977, 1978, 1983, 1989,
 2008–2010, 2012, 2035, 2087, 2101, 4521,
 4522, 4524, 4528, 4529
 Template ID Library, 2029
 Template Point Centerline, 995, 1698, 1721, 1740,
 1961, 1962, 1993–1995, 2014, 2015, 2035,
 2047, 2048, 2050, 2054, 2055, 2088, 3899,
 4745
 Template Point Profile, 28, 837, 1740, 1961, 1962,
 1974, 1993, 1994, 1998, 2032, 2035, 2045,
 2050, 2052, 2054, 2087
 Template Series Editor, 1989, 1990, 2033, 2050, 2085
 Template Transition, 1973–1977, 1989, 2034, 2045–
 2048, 2084, 2086, 2087, 2100
 Temporary Erase, 134
 Terramodel, 1720, 1863, 1884
 Text, 1306, 1307, 1346, 1351, 1353, 1358, 1388,
 1410, 1450, 1451, 1454, 1479, 1482, 1484,
 1510, 1511, 1521, 1535–1537, 1553, 1555,
 1560, 1568, 1569, 1599, 1600, 1646, 1647,
 1650, 1657, 1658, 1703, 1705, 1706, 1712–
 1714, 1719, 1744, 1754, 1766, 1768–1772,
 1775, 1777–1779, 1790, 1806, 1809, 1810,
 1812, 1822, 1830, 1836, 1852, 1853, 1861,

1883, 1911, 1915, 1931, 1934, 1953, 1954,
1985, 2022, 2039, 2217, 2261, 2284, 2334,
2338, 2432, 2439, 2474, 2490, 2491, 2519,
2531, 2538, 2539, 2546, 2547, 2551, 2595,
2630–2633, 2653, 2654, 2668, 2686, 2718,
2733, 2745, 2746, 2761, 2763, 2765, 2848,
2852–2855, 2864, 3001, 3007, 3029, 3034,
3039, 3059, 3075, 3096, 3097, 3133, 3137,
3189, 3216, 3220, 3226, 3228, 3229, 3232,
3245, 3247, 3248, 3250–3253, 3255, 3262,
3264, 3265, 3272, 3291, 3293, 3308, 3341,
3361–3363, 3374, 3436, 3437, 3443, 3446,
3467, 3503, 3509, 3512, 3513, 3517–3519,
3535, 3627–3630, 3632, 3671–3673, 3676,
3679, 3681, 3693, 3714, 3726, 3739, 3740,
3753, 3813, 3814, 3845, 3846, 3879, 3883,
3907, 3911–3916, 3919, 3920, 3923, 3943,
3944, 3950–3952, 3956, 3957, 3969–3975,
3995, 4040, 4047, 4051, 4075, 4076, 4107,
4128, 4171, 4172, 4176, 4177, 4182–4185,
4217, 4252–4255, 4265–4267, 4277–4279,
4283–4285, 4293, 4301–4305, 4308, 4314–
4316, 4320, 4328–4330, 4332, 4334, 4336–
4338, 4357, 4358, 4553, 4619, 4675, 4679

Textbox, 1127

Text Capitalization, 163

Text Enlarge/Reduce, 154, 159

Text Explode To Polylines, 105, 114, 163

Text Export, 248

Text File To Report Formatter, 286

Text Height, 4748

Text Import, 248

Text Mask, 248

Text Math, 164

Text on Arc, 1278

Text on Line, 246

Text on Polyline, 247

Text Plot Size, 288

Text Size, 154, 258, 267, 293, 375, 505, 688, 705,
713, 719, 766, 783, 791, 797, 915, 925,
926, 928, 929, 956, 1009, 1038, 1045, 1051,
1052, 1075, 1076, 1079, 1082, 1090, 1098,
1275, 1279, 1296, 1299

Text Style, 293, 703, 781, 855, 915, 925, 926, 928,
929, 1038, 1047, 1049, 1052, 1075, 1076,
1090, 1097, 1098, 1110, 1115

Text UnMask, 249

Text Wizard, 242

Thaw/On All Layers, 216, 4424, 4592
 Thaw ALL Layers, 1366
 Thaw Layer, 211, 403, 3482, 4301
 Thaw Layer By Name, 216
 Thaw Point Attributes, 427
 Thaw Points, 403
 The Fluvial Geomorphic Solution, 3146
 The Transformation Panel, 3125
 Three Triangulation Surface Volumes, 1589
 Three Way Control, 886, 887
 TIF file, 2600
 Time of Concentration, 28, 2164, 2173, 2176–2179,
 2182–2187, 2192, 2204, 2246, 2326, 2383,
 2384, 4541, 4542, 4544
 Time of Concentration - Kerby-Kirpich Method, 2185
 Time of Concentration - Kinematic Wave Method,
 2177
 Time of Concentration - Kirpich Method, 2184
 Time of Concentration - Maricopa County, AZ, 2182
 Time of Concentration - Rational Method, 2176
 Time of Concentration - Regression Equation, LA
 County, 2178
 Time of Concentration - SCS Method, 2172
 Time of Concentration - TR-55 Method, 2173
 Timing Data, 3953, 3988, 4081
 Timing Grids, 3979, 4197, 4198
 Timing Project Manager, 3695, 3709, 3711, 3721,
 3734, 3741, 3742, 3744, 3746, 3757–3759,
 3800, 3802, 3807–3809, 3917, 3991–3994,
 4021, 4036, 4050, 4052, 4054, 4055, 4057,
 4061, 4065, 4067, 4077, 4079, 4080, 4193,
 4194, 4224
 Tin, 1344, 1362, 1603, 1605, 1685, 2140, 2220, 2636
 TINs, 1583, 3067, 3068, 3073–3075, 3123, 3140,
 3550, 4759, 4760, 4762
 T Intersections, 1259
 Title Block, 205, 360, 1079, 2695, 4250, 4252, 4253,
 4262, 4264, 4265, 4274, 4276, 4277, 4284,
 4309, 4333, 4687, 4688
 Toolbar Manager, 350
 Toolbars, 606, 2982, 4240, 4243
 Tools Menu, 594, 1150, 1375, 1412, 1416, 2614, 3016
 Topcon Total Stations, 2973
 Topographic Settings, 1344, 1345
 Topology Area Check, 2591, 2592
 Topology Point Check, 2593
 Topsoil Removal, 30, 2032, 2035, 2036, 2085, 2674,
 2694, 2700

Topsoil Removal/Replacement, 1926, 1953, 1991,
2035, 2085, 2086, 2676, 2677, 2679
Topsoil Removal and Replacement, 2672
Trace Circle, 2837, 2838
Trace Line, 2839, 2840, 2855
Trace Polyline, 2839–2842
Trace Rectangle, 2838
Track Position, 2928, 2929, 2970
Transformations, 1327, 1330, 1331, 1333, 1334,
3030, 3037, 3040, 3139
Transform Cloud, 3055, 3056, 3125, 3126
Transform Coordinates, 2445
Transform Elevations, 2426
Transit, 510, 517, 518, 528, 733, 828, 1154, 1354,
1356, 1465, 1602, 1664, 1740, 1746, 1912,
1965, 1966, 1969, 1974–1976, 1979, 1980,
1986, 1990, 1995, 2046, 2047, 2058, 2064,
2078, 2086, 2096–2098, 2100, 2101, 2104,
2107, 2108, 2470, 2660, 3171, 3419, 3880,
4013, 4745
Translate Layers, 128
Translate Points, 212, 385, 388, 404, 411, 1327
Translate Solid, 3831
Transparent Commands, 23, 4241
Traverse, 858–863, 963, 1151–1154, 1184, 1188,
1194, 1208, 1209, 1212, 1213, 1215–1218,
1220–1224, 1227, 1347, 1355, 1356, 1376,
1380, 1383–1385, 1390, 3932, 3934, 4249,
4261, 4273, 4343, 4344, 4346
Trench Menu, 2771, 4636
Trench Module, 2770
Trench Network File Backup, 2778
Trench Network Quantities, 2787, 4632, 4642, 4648
Trench Quantities By Profiles, 2790
Trench Spreadsheet Editor, 2775, 2776
Trench Subgrade Areas, 2786
Trench Template Editor, 1840, 2775, 2784, 2785,
2787
Triangle Solutions, 892
Triangular Mesh, 1667, 2606
Triangulate & Contour, 1499, 1502, 1513, 1514,
1519, 1546, 1587, 1635, 1639, 1645, 1674,
1677, 1678, 1824, 1847, 1867, 2003, 2033,
2034, 2041, 2043, 2068, 2140, 2159, 2276,
2454, 2606, 2625, 2628, 2632, 2706, 3762,
3769, 3770, 3835, 3837, 3849, 3909, 3910,
4316, 4317, 4338, 4398, 4413, 4422, 4429,
4477, 4478, 4515, 4557, 4713

Triangulation, 27, 30, 35, 37, 94, 312, 324, 836, 840,
1270, 1462, 1501, 1502, 1505, 1514, 1532,
1541, 1559–1561, 1567–1569, 1585–1589,
1591, 1593, 1603, 1605, 1606, 1627, 1628,
1645, 1648, 1649, 1682, 1729, 1730, 1869,
2068, 2106, 2121, 2122, 2127, 2139, 2159,
2626, 2633, 2634, 2705, 2706, 2788, 2861,
3180, 3238, 3337, 3340, 3343, 3349, 3364,
3366, 3382, 3412, 3418, 3427, 3432, 3434,
3444, 3502, 3503, 3525, 3534, 3535, 3560,
3568, 3571, 3573, 3586, 3677, 3762, 3769,
3820, 3837, 3838, 3841, 3853, 3966, 4006,
4012, 4163, 4398, 4407, 4411, 4412, 4430,
4450, 4453, 4455, 4462, 4477, 4484, 4485,
4518, 4698, 4737

Triangulation Data Check, 1498

Triangulation File Utilities, 1514, 1517, 1645, 1658,
2634

Trim, 165, 1438, 1502, 1627, 1629, 1638, 2043, 2650,
2651, 3405, 3409, 3837, 4438, 4442, 4446

Trim 3D Polyline, 190

Trimble, 95, 103, 381, 461, 472, 497, 498, 544, 547,
550, 564, 601, 602, 609, 610, 667, 745,
1683, 1720, 1721, 2519, 2549, 2551, 2890,
2896, 2897, 2899, 2913, 2967, 2975–2977

Trim by Point Symbol, 426, 427

Trim Contour-Plines by Pline, 1551

Trim Isopach Within Fault Zone, 3383

Trim Linework Through Text, 250

Trim Solid, 3833

Troubleshooting and Customizing Setup, 19

Tunnel Network, 3786–3788

Tunnel Section Comparison Report, 3889

Tunnel Series Editor, 3888

Turn OFF ALL Layers, 1367

Turn Off By Pick, 216

Turn Off By Selection, 209, 217

Turn ON ALL Layers, 1367

Turn On Layers, 217

Tutorials, 4235

Twist, 149, 151, 154, 163, 209, 307, 425, 697, 725,
775, 803, 915, 1091, 1714, 1836, 4062,
4328–4330, 4675, 4749

Twist Point Attributes, 425, 4328

Twist Screen: Line Pline or Text, 213

Twist Screen: Standard, 212

Twist Screen: Surveyor, 213

Twist To 3D View, 213, 214

Two Grid Surface Volumes, 1593, 1619, 1621
Two Surface Volumes, 1593–1595, 1597, 3525–3527,
3529, 4467
Two Triangulation Surface Volumes, 1587
Two Way Control, 885
Typical Alignment Scenarios, 2938
Typical Template, 1973

Underground Menu, 3693
Underground Mine Reserves, 3427, 3429, 3470, 3586,
3588
Underground Mining Module, 3417, 3621, 3648,
3690, 3917, 3920, 3932, 3993, 4001, 4011
Underground Timing, 46, 3207, 3685, 3694–3696,
3708, 3714, 3716, 3721, 3726, 3731, 3739,
3740, 3743, 3745, 3760, 3761, 3917, 3928,
3996, 4005, 4021, 4068

Undo Raster Edit, 2851
Unequal Radius, 1034
Universal Soil Loss, 2140, 2141
Universal Transverse Mercator, 389, 412, 890, 3050
Unix, 339
Unlock Attributes, 131
Unlock Points, 404
Untag Area Descriptions, 932
Untag Hard Breakline Polylines, 1496
UnTag Limit Polylines, 3395
Untag Linework, 1015
Untag No-Building Area, 1025
Untag Non-Surface Points, 1492
Untag Predefined Boundaries, 1498
Untag Strata Polylines, 3391
Untag Sub-Area, 1007
Update Block Data, 2578, 2579
Update Colors For Set Elevations, 2666
Update Cut/Fill, 3180
Update Drawing Coordinates, 2583
Update Drawing from CRD File, 385, 392
Update Edited Panel, 3724, 3729, 3737
Update Grid File, 4162, 4163
Update Pad Elevation Labels, 1478
Update Point Table, 1088
Update Sections from Polylines, 1949
Update Sections from Slope Stake, 1942
USDA Soil Area Download, 2166
Utility Network, 2469–2471, 2473–2475, 2493

Valley Fill, 4171
Valley Pond, 1644, 1645, 2257, 4428, 4446, 4447

Valley Pond Design, 1644, 2261
Variable Offset, 138
Vector To Text, 2854
Vehicle Path Tracking, 2022, 2023, 2028
Vehicle Path Tracking Profile, 2017
Ventilation, 36, 3648, 3653–3655, 3657
Vertical Adjustment, 650, 1712
Vertical Angle, 483, 504, 528, 535, 573, 624, 639,
642, 647, 860, 863, 866, 867, 1197, 1356,
1381, 2879, 3625, 3934
Vertical Angle Mode, 309, 371
Vertical Curve, 313, 1240
Vertical Curve Design, 1257
Vertical Datum Utilities, 434
Vertical Pit Quantities, 322, 4161, 4162
View, 17, 26, 30, 36, 41, 42, 45, 47, 48, 53, 67, 87,
93, 109, 121, 124, 161, 162, 194–196, 198,
199, 201, 202, 204–214, 216–218, 226, 294,
307–309, 325, 341, 356, 432, 452, 457, 459,
512, 514, 719, 725, 729, 797, 803, 810, 819,
839, 840, 843, 955, 984, 997, 1000, 1008,
1032, 1060–1062, 1066, 1079, 1141, 1150,
1151, 1164, 1296, 1325, 1357, 1371, 1375,
1380, 1386–1388, 1401, 1405, 1425, 1427,
1471, 1502, 1551, 1581, 1594, 1624, 1629,
1632, 1647, 1648, 1654, 1660, 1662–1667,
1674, 1687, 1750, 1751, 1776, 1820, 1851,
1859, 1891, 1941, 1958, 1981, 2045, 2113,
2114, 2123, 2226, 2302, 2324, 2395, 2414,
2415, 2432, 2471, 2473, 2474, 2477, 2479,
2490, 2497, 2525, 2560, 2603–2605, 2607,
2608, 2612, 2618, 2620, 2625, 2626, 2643,
2663, 2679, 2691, 2695–2697, 2718, 2727,
2834, 2836, 2857, 2858, 2860, 2863, 2864,
2867, 2913, 2968, 2985–2987, 2993, 3002,
3003, 3007–3009, 3013, 3016–3018, 3021,
3022, 3028, 3031, 3032, 3038, 3041, 3058,
3077, 3114, 3123, 3129, 3135, 3169, 3181,
3182, 3185–3188, 3194, 3198, 3411, 3412,
3415, 3442, 3450, 3474, 3479, 3480, 3482,
3513, 3514, 3526, 3548, 3607, 3608, 3614,
3765, 3768, 3777, 3778, 3780, 3785, 3790,
3792, 3799, 3800, 3814, 3817, 3820, 3825,
3833, 3838, 3859, 3989, 3990, 4008, 4047,
4048, 4077, 4081, 4084, 4087, 4094, 4106,
4114, 4173, 4223, 4241, 4249, 4251, 4256,
4262–4264, 4268, 4274–4276, 4280, 4293–
4296, 4298, 4299, 4301, 4316, 4317, 4325,

4328, 4333, 4348, 4357, 4379, 4397, 4402,
 4405, 4406, 4409, 4411, 4422, 4424, 4431,
 4454, 4462, 4473, 4474, 4479, 4493, 4511,
 4516, 4530, 4532, 4533, 4535, 4537, 4541,
 4544, 4546, 4557, 4558, 4579, 4582, 4592,
 4596, 4598, 4603, 4606, 4607, 4613, 4614,
 4618–4620, 4622, 4637, 4639, 4669, 4675,
 4708
 View 3D Surface History, 4081, 4198
 View 3D Timing, 3816, 4083
 View Longitudinal Profile, 3193, 3194
 View Menu, 194, 602, 1374, 1403, 3016, 4046
 View Overexcavate Surface, 2679
 View Solid, 3790
 View Top Surface, 2683
 Visual COGO, 862, 864, 865, 4736
 Visual Map Check, 1225
 Volume, 27, 446, 1445, 1497, 1498, 1585, 1587–
 1589, 1591, 1593, 1594, 1596, 1597, 1599,
 1613, 1617–1623, 1627, 1628, 1633, 1639,
 1824, 1834, 1870, 1908, 1909, 1917, 1924,
 1926, 1928, 1942, 1951, 1952, 1954, 2032,
 2037, 2039, 2168, 2172, 2205, 2257, 2261,
 2272, 2273, 2277, 2279, 2486, 2666, 2675,
 2687, 2701, 2706–2710, 2713–2715, 3076,
 3123, 3153, 3188, 3189, 3364, 3365, 3367,
 3421, 3425, 3490, 3491, 3493, 3523, 3525,
 3526, 3528, 3560, 3575, 3751, 3805, 3812,
 3854, 3876, 3877, 3885, 3970, 3980, 3984,
 3988, 3990, 3991, 3993, 4015, 4019, 4040,
 4049, 4074, 4130, 4137, 4170, 4176, 4189,
 4235, 4425, 4427, 4428, 4436, 4440, 4463,
 4464, 4468, 4470, 4488, 4526, 4545, 4547,
 4548, 4625, 4632, 4666, 4718
 Volumes, 4740
 Volumes By Layer, 1597, 4464, 4469, 4470, 4488
 Volumes By Triangulation, 2706, 4464, 4476, 4480,
 4488
 Voronoi Diagram, 3448

 Wall Polyline Profile Editor, 1426
 Watershed Above Points, 2160
 Watershed Analysis, 2152, 2154, 2155, 2157–2160,
 2325, 2502, 4235, 4560, 4561, 4563, 4586,
 4588, 4589
 Watershed Area Grid, 2162
 Watershed Menu, 2143
 Watershed Settings (Save and Load), 2194
 Web Feature Service (WFS), 2558, 2561, 2605

Web Site, 9, 23
What is New, 25
White Solid Behind Text, 250
Width Analysis, 1929
Wild, 116, 374, 380, 383, 480, 512, 541, 840, 1190,
1194, 1195, 1842, 1861, 1863, 1885, 2604,
3160, 4301
Wildsoft, 393, 479, 480, 496
Windows Calculator, 1371
Works Menu, 3641
Write Block by Polyline, 2596
Write Polyline File, 114, 163

X Intersections, 1260

Y Intersections, 1261

Zeiss, 381, 382, 497, 2890
Zero Thickness, 321
Zone Polylines, 105, 115
Zoom Extents on All Layouts, 195
Zoom Most, 195
Zoom Points, 194
Zoom Selection, 194
Zoom to Point ID, 1370

